Ansys 2021 R2
Engineering What's Ahead.

©2021 ANSYS, Inc.
All Rights Reserved.
Unauthorized use, distribution or duplication is prohibited.

Ansys Fluent User's Guide

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

Release 2021 R2
July 2021

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

Copyright and Trademark Information

© 2021 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

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

Disclaimer Notice

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

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

U.S. Government Rights

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

Third-Party Software

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

Published in the U.S.A.

Table of Contents

I. Getting Started 127
1. Introduction to Ansys Fluent 129
1.1. The Ansys Product Improvement Program 131
1.2. Program Capabilities 135
1.3. Known Limitations in Ansys Fluent 2021 R2 136
2. Basic Steps for CFD Analysis using Ansys Fluent 155
2.1. Steps in Solving Your CFD Problem 155
2.2. Planning Your CFD Analysis 155
3. Guide to a Successful Simulation Using Ansys Fluent 161
4. Starting and Executing Ansys Fluent 163
4.1. Starting Ansys Fluent 163
4.1.1. Selecting the Licensing Level 163
4.1.2. Starting Ansys Fluent Using Fluent Launcher 164
4.1.2.1. Setting General Options in Fluent Launcher 166
4.1.2.2. Single-Precision and Double-Precision Solvers 168
4.1.2.3. Setting Parallel Options in Fluent Launcher 169
4.1.2.4. Setting Remote Options in Fluent Launcher 170
4.1.2.5. Setting Scheduler Options in Fluent Launcher 170
4.1.2.6. Setting Environment Options in Fluent Launcher 171
4.1.3. Starting Ansys Fluent on a Windows System 172
4.1.4. Starting Ansys Fluent on a Linux System 174
4.1.5. Command Line Startup Options 174
4.1.5.1. ACT Option 177
4.1.5.2. Application Option 177
4.1.5.3. Application Script Option 178
4.1.5.4. Graphics Options 178
4.1.5.5. Meshing Mode Option 179
4.1.5.6. Performance Options 179
4.1.5.7. Parallel Options 179
4.1.5.8. Postprocessing Option 181
4.1.5.9. Remote Visualization Options 181
4.1.5.10. Scheduler Options 181
4.1.5.11. Text Command Option 182
4.1.5.12. Version, Release Options, and Environment Variables 183
4.1.5.13. System Coupling Options 183
4.1.5.14. Other Startup Options 184
4.2. Running Ansys Fluent in Batch Mode 184
4.2.1. Background Execution on Linux Systems 184
4.2.2. Background Execution on Windows Systems 186
4.2.3. Batch Execution Options 186
4.3. Switching Between Meshing and Solution Modes 188
4.4. Checkpointing an Ansys Fluent Simulation 188
4.5. Cleaning Up Processes From an Ansys Fluent Simulation 190
4.6. Exiting Ansys Fluent 191
Glossary of Terms 193
II. Meshing Mode 197
1. Introduction to Meshing Mode in Fluent 199
1.1. Meshing Approach 199
1.2. Meshing Mode Capabilities 199

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Starting Fluent in Meshing Mode 201
2.1. Starting the Dual Process Build 201
2.2. Dynamically Spawning Processes Between Fluent Meshing and Fluent Solution Modes 202
3. Graphical User Interface 205
3.1. User Interface Components 206
3.1.1. The Ribbon 206
3.1.2. The Workflow Tab 212
3.1.3. The Outline View Tab 213
3.1.4. The Graphics Window 220
3.1.5. Quick Search 220
3.1.6. The Console 221
3.1.7. The Toolbars 221
3.1.7.1. Pointer Tools 222
3.1.7.2. View Tools 223
3.1.7.3. Graphics Effects Tools 224
3.1.7.4. Mesh Display Tools 225
3.1.7.5. Visibility Tools 225
3.1.7.6. Copy Tools 226
3.1.7.7. Object Selection/Display Tools 226
3.1.7.8. Filter Toolbar 227
3.1.7.9. CAD Tools 227
3.1.7.10. Tools 228
3.1.7.11. Context Toolbar 228
3.1.8. ACT Start Page 228
3.2. Customizing the User Interface 228
3.3. Setting User Preferences/Options 229
3.4. Using the Help System 231
3.4.1. Help for Text Interface Commands 231
3.4.2. Obtaining a Listing of Other License Users 231
4. Text User Interface 233
5. Reading and Writing Files 235
5.1. Shortcuts for Reading and Writing Files 235
5.1.1. Binary Files 235
5.1.2. Reading and Writing Compressed Files 236
5.1.2.1. Reading Compressed Files 236
5.1.2.2. Writing Compressed Files 236
5.1.3. Tilde Expansion (LINUX Systems Only) 237
5.1.4. Disabling the Overwrite Confirmation Prompt 237
5.2. Mesh Files 237
5.2.1. Reading Mesh Files 238
5.2.1.1. Reading Multiple Mesh Files 239
5.2.1.2. Reading 2D Mesh Files in the 3D Version of Fluent 239
5.2.2. Reading Boundary Mesh Files 239
5.2.3. Reading Faceted Geometry Files from Ansys Workbench in Fluent 240
5.2.4. Appending Mesh Files 240
5.2.5. Writing Mesh Files 240
5.2.6. Writing Boundary Mesh Files 241
5.3. Case Files 241
5.3.1. Reading Case Files 242
5.3.1.1. Reading Files Using the Legacy Format 243
5.3.2. Writing Case Files 243

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.3.2.1. Writing Files Using the Legacy Format 244
5.4. Reading and Writing Size-Field Files 244
5.5. Reading Scheme Source Files 245
5.6. Creating and Reading Journal Files 245
5.7. Creating Transcript Files 247
5.8. Reading and Writing Domain Files 248
5.9. Importing Files 248
5.9.1. Importing CAD Files 249
5.10. Saving Picture Files 256
5.10.1. Using the Save Picture Dialog Box 256
6. Working With Fluent Guided Workflows 261
6.1. Getting Started with the Fluent Guided Workflows 261
6.1.1. Prerequisites for the Fluent Guided Workflows 262
6.1.2. Limitations of the Fluent Guided Workflows 263
6.1.3. Customizing Workflows 268
6.1.4. Understanding Task States 269
6.1.5. Operating on Tasks 270
6.1.6. Grouping Tasks 270
6.1.7. Editing Tasks 270
6.1.8. Monitoring Task Updates 271
6.1.9. Accessing Advanced Options 272
6.1.10. Filtering Lists and Using Wildcards 272
6.1.11. Saving and Loading Workflows 273
6.1.12. Setting Preferences for Workflows 273
6.1.13. Getting Help for Workflow Tasks 275
6.2. Using the Watertight Geometry Workflow 275
6.2.1. Importing Geometries 276
6.2.2. Importing Body of Influence Geometries 278
6.2.3. Adding Local Sizing 279
6.2.4. Generating the Surface Mesh 282
6.2.5. Setting Up Periodic Boundaries 288
6.2.6. Describing the Geometry 292
6.2.7. Applying Share Topology 292
6.2.7.1. Troubleshooting Gap Marking 294
6.2.8. Enclosing Fluid Regions 296
6.2.9. Creating Regions 299
6.2.10. Updating Regions 300
6.2.11. Adding Boundary Layers 301
6.2.12. Generating the Volume Mesh 304
6.2.13. Updating Boundaries 309
6.2.14. Improving the Surface Mesh 310
6.2.15. Adding Boundary Types 311
6.2.16. Improving the Volume Mesh 312
6.2.17. Transforming the Volume Mesh 312
6.2.18. Extruding the Volume Mesh 314
6.2.19. Adding Linear Mesh Patterns 318
6.2.19.1. Creating Custom Patterns Using Scripts 321
6.2.19.1.1. Examples of Creating Custom Patterns Using Scripts 324
6.2.19.1.1.1. Pattern Example Using Explicit Name Rule 325
6.2.19.1.1.2. Pattern Example of Using Both Explicit and Regular Name Rule 326
6.2.20. Managing Zones 329

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.2.21. Modifying Mesh Refinement 331
6.2.22. Creating Local Refinement Regions 332
6.2.23. Running Custom Journal Commands 336
6.3. Using the Fault-tolerant Meshing Workflow 337
6.3.1. Importing CAD Geometries and Managing CAD Parts 337
6.3.1.1. Appending CAD Files 341
6.3.1.2. Working with the CAD Model Tree 342
6.3.1.3. Working with the Meshing Model Tree 344
6.3.1.4. Setting Properties for Meshing Model Objects 346
6.3.1.4.1. Creating Automatic Meshing Objects on a Per Part Basis 346
6.3.1.4.2. Creating Customized Meshing Objects 349
6.3.1.5. Performing Operations on Meshing Model Objects 352
6.3.1.5.1. Performing Transformation Operations on Meshing Model Objects 352
6.3.1.5.2. Performing Refaceting Operations on Meshing Model Objects 354
6.3.1.5.3. Listing Meshing Operations 356
6.3.1.6. Faceting Considerations 357
6.3.1.7. Setting Display Options for CAD Model and Meshing Objects 358
6.3.1.8. Using Hot Key Shortcuts in the Model Trees and the Graphics Window 359
6.3.2. Describing the Geometry and the Flow 359
6.3.3. Enclosing Fluid Regions 361
6.3.4. Creating External Flow Boundaries 364
6.3.5. Creating Local Refinement Regions 366
6.3.6. Identifying Construction Surfaces 371
6.3.7. Extracting Edge Features 373
6.3.8. Adding Thickness to Your Geometry 377
6.3.9. Creating Porous Regions 380
6.3.10. Identifying Regions 382
6.3.11. Defining Leakage Thresholds 385
6.3.12. Updating Your Region Settings 386
6.3.13. Choosing Mesh Control Options 388
6.3.14. Adding Local Size Controls 391
6.3.15. Generating the Surface Mesh 392
6.3.16. Updating Boundaries 395
6.3.17. Describing Overset Features 396
6.3.17.1. Creating Collar Meshes 396
6.3.17.2. Creating Component Meshes 398
6.3.18. Adding Boundary Layers 400
6.3.19. Identifying Deviated Faces 402
6.3.20. Generating the Volume Mesh 403
6.3.21. Creating Overset Mesh Interfaces 405
6.3.22. Identifying Orphans 406
6.3.23. Transforming the Volume Mesh 407
6.3.24. Extruding the Volume Mesh 408
6.3.25. Separating Contacts 412
7. CAD Assemblies 413
7.1. CAD Assemblies Tree 413
7.1.1. FMDB File 414
7.1.2. CAD Entity Path 415
7.1.3. CAD Assemblies Tree Options 415
7.2. Visualizing CAD Entities 416
7.3. Updating CAD Entities 416

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4. Manipulating CAD Entities 417
7.4.1. Creating and Modifying Geometry/Mesh Objects 417
7.4.2. Managing Labels 418
7.4.3. Setting CAD Entity States 418
7.4.4. Modifying CAD Entities 419
7.5. CAD Association 419
8. Size Functions and Scoped Sizing 421
8.1. Types of Size Functions or Scoped Sizing Controls 422
8.1.1. Curvature 422
8.1.2. Proximity 423
8.1.3. Meshed 427
8.1.4. Hard 427
8.1.5. Soft 428
8.1.6. Body of Influence 429
8.2. Defining Size Functions 430
8.2.1. Creating Default Size Functions 430
8.3. Defining Scoped Sizing Controls 431
8.3.1. Size Control Files 431
8.4. Computing the Size Field 432
8.4.1. Size Field Files 432
8.4.2. Using Size Field Filters 432
8.4.3. Visualizing Sizes 433
8.5. Using the Size Field 435
9. Objects and Material Points 437
9.1. Objects 437
9.1.1. Object Attributes 438
9.1.1.1. Creating Objects 440
9.1.2. Object Entities 441
9.1.2.1. Using Face Zone Labels 442
9.1.3. Managing Objects 443
9.1.3.1. Using hotkeys and onscreen tools 443
9.1.3.1.1. Creating Objects for CAD Entities 443
9.1.3.1.2. Creating Objects for Unreferenced Zones 444
9.1.3.1.3. Creating Multiple Objects 444
9.1.3.1.4. Easy Object Creation and Modification 445
9.1.3.1.5. Changing Object Properties 445
9.1.3.1.6. Automatic Alignment of Objects 445
9.1.3.1.7. Remeshing Geometry Objects 446
9.1.3.1.8. Creating Edge Zones 446
9.1.3.2. Using the Manage Objects Dialog Box 446
9.1.3.2.1. Defining Objects 447
9.1.3.2.2. Object Manipulation Operations 448
9.1.3.2.3. Object Transformation Operations 449
9.2. Material Points 450
9.2.1. Creating Material Points 452
10. Object-Based Surface Meshing 455
10.1. Surface Mesh Processes 455
10.2. Preparing the Geometry 457
10.2.1. Using a Bounding Box 458
10.2.2. Closing Annular Gaps in the Geometry 458
10.2.3. Patching Tools 458

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.2.3.1. Using the Patch Options Dialog Box 459
10.2.3.2. Using the Loop Selection Tool 462
10.2.4. Using User-Defined Groups 463
10.3. Diagnostic Tools 463
10.3.1. Geometry Issues 464
10.3.2. Face Connectivity Issues 465
10.3.3. Quality Checking 466
10.3.4. Summary 467
10.4. Connecting Objects 467
10.4.1. Using the **Join/Intersect** Dialog Box 470
10.4.2. Using the **Join** Dialog Box 471
10.4.3. Using the **Intersect** Dialog Box 472
10.5. Advanced Options 472
10.5.1. Object Management 472
10.5.2. Removing Gaps Between Mesh Objects 473
10.5.3. Removing Thickness in Mesh Objects 475
10.5.4. Sewing Objects 477
10.5.4.1. Resolving Thin Regions 479
10.5.4.2. Processing Slits 479
10.5.4.3. Removing Voids 479
11. Object-Based Volume Meshing 481
11.1. Volume Mesh Process 481
11.2. Volumetric Region Management 483
11.2.1. Computing and Verifying Regions 483
11.2.2. Volumetric Region Operations 485
11.3. Generating the Volume Mesh 486
11.3.1. Meshing All Regions Collectively Using Auto Mesh 487
11.3.2. Meshing Regions Selectively Using Auto Fill Volume 489
11.4. Cell Zone Options 490
12. Manipulating the Boundary Mesh 493
12.1. Manipulating Boundary Nodes 493
12.1.1. Free and Isolated Nodes 493
12.2. Intersecting Boundary Zones 495
12.2.1. Intersecting Zones 495
12.2.2. Joining Zones 496
12.2.3. Stitching Zones 498
12.2.4. Using the Intersect Boundary Zones Dialog Box 499
12.2.5. Using Shortcut Keys/Icons 500
12.3. Modifying the Boundary Mesh 501
12.3.1. Using the Modify Boundary Dialog Box 501
12.3.2. Operations Performed: Modify Boundary Dialog Box 501
12.3.3. Locally Remeshing a Boundary Zone or Faces 508
12.3.4. Moving Nodes 508
12.4. Improving Boundary Surfaces 509
12.4.1. Improving the Boundary Surface Quality 509
12.4.2. Smoothing the Boundary Surface 509
12.4.3. Swapping Face Edges 510
12.5. Refining the Boundary Mesh 510
12.5.1. Procedure for Refining Boundary Zones 510
12.6. Creating and Modifying Features 512
12.6.1. Creating Edge Zones 513

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.6.2. Modifying Edge Zones 516
12.6.3. Using the Feature Modify Dialog Box 517
12.7. Remeshing Boundary Zones 519
12.7.1. Creating Edge Zones 519
12.7.2. Modifying Edge Zones 520
12.7.3. Remeshing Boundary Face Zones 520
12.7.4. Using the Surface Retriangulation Dialog Box 521
12.8. Faceted Stitching of Boundary Zones 522
12.9. Triangulating Boundary Zones 523
12.10. Separating Boundary Zones 524
12.10.1. Separating Face Zones using Hotkeys 524
12.10.2. Using the Separate Face Zones dialog box 525
12.11. Projecting Boundary Zones 528
12.12. Creating Groups 528
12.13. Manipulating Boundary Zones 529
12.14. Manipulating Boundary Conditions 530
12.15. Creating Surfaces 531
12.15.1. Creating a Bounding Box 531
12.15.1.1. Using the Bounding Box Dialog Box 531
12.15.1.2. Using the Construct Geometry Tool 532
12.15.2. Creating a Planar Surface Mesh 533
12.15.2.1. Using the Plane Surface Dialog Box 534
12.15.3. Creating a Cylinder/Frustum 535
12.15.3.1. Using the Cylinder Dialog Box 537
12.15.3.2. Using the Construct Geometry Tool 538
12.15.4. Creating a Swept Surface 539
12.15.4.1. Using the Swept Surface Dialog Box 539
12.15.5. Creating a Revolved Surface 540
12.15.5.1. Using the Revolved Surface Dialog Box 540
12.15.6. Creating Periodic Boundaries 541
12.16. Removing Gaps Between Boundary Zones 544
12.17. Using the Loop Selection Tool 544
13. Wrapping Objects 547
13.1. The Wrapping Process 548
13.1.1. Extract Edge Zones 550
13.1.2. Create Intersection Loops 551
13.1.2.1. Individually 551
13.1.2.2. Collectively 552
13.1.3. Setting Geometry Recovery Options 553
13.1.4. Fixing Holes in Objects 553
13.1.5. Shrink Wrapping the Objects 558
13.1.6. Improving the Mesh Objects 561
13.1.7. Object Wrapping Options 562
13.1.7.1. Resolving Thin Regions During Object Wrapping 562
13.1.7.2. Detecting Holes in the Object 563
13.1.7.3. Improving Feature Capture For Mesh Objects 563
14. Creating a Mesh 565
14.1. Choosing the Meshing Strategy 565
14.1.1. Boundary Mesh Containing Only Triangular Faces 566
14.1.2. Mixed Boundary Mesh 567
14.1.3. Hexcore Mesh 568

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.1.4. CutCell Mesh 569
14.1.5. Rapid Octree Mesh 570
14.1.6. Additional Meshing Tasks 571
14.1.7. Inserting Isolated Nodes into a Tet Mesh 572
14.2. Using the Auto Mesh Dialog Box 575
14.3. Generating a Thin Volume Mesh 578
14.4. Generating Pyramids 579
14.4.1. Creating Pyramids 579
14.4.2. Zones Created During Pyramid Generation 581
14.4.3. Pyramid Meshing Problems 581
14.5. Creating a Non-Conformal Interface 583
14.5.1. Separating the Non-Conformal Interface Between Cell Zones 583
14.6. Creating a Heat Exchanger Zone 584
14.7. Parallel Meshing 585
14.7.1. Auto Partitioning 585
14.7.1.1. Availability of Graphical User Interface Options After Parallel Meshing 586
14.7.1.2. Availability of Text Interface Options After Parallel Meshing 588
14.7.2. Computing Partitions 590
14.7.3. Controlling the Threads 591
15. Generating Prisms 593
15.1. The Prism Generation Process 594
15.1.1. Zones Created During Prism Generation 595
15.2. Procedure for Creating Zone-based Prisms 596
15.3. Prism Meshing Options for Zone-Specific Prisms 599
15.3.1. Growth Options for Zone-Specific Prisms 600
15.3.1.1. Growing Prisms Simultaneously from Multiple Zones 600
15.3.1.2. Growing Prisms on a Two-Sided Wall 603
15.3.1.3. Ignoring Invalid Normals 603
15.3.1.4. Detecting Proximity and Collision 604
15.3.1.5. Splitting Prism Layers 607
15.3.1.6. Preserving Orthogonality 607
15.3.2. Offset Distances 608
15.3.3. Direction Vectors 611
15.3.4. Using Adjacent Zones as the Sides of Prisms 613
15.3.5. Post Prism Mesh Quality Improvement 616
15.3.5.1. Improving the Prism Cell Quality 616
15.3.5.2. Removing Poor Quality Cells 617
15.3.5.3. Improving Warp 618
15.4. Prism Meshing Options for Scoped Prisms 618
15.5. Prism Meshing Problems 621
16. Generating Tetrahedral Meshes 625
16.1. Automatically Creating a Tetrahedral Mesh 625
16.1.1. Automatic Meshing Procedure for Tetrahedral Meshes 625
16.1.2. Using the Auto Mesh Tool 627
16.1.3. Automatic Meshing of Multiple Cell Zones 627
16.1.4. Automatic Meshing for Hybrid Meshes 628
16.1.5. Further Mesh Improvements 629
16.2. Manually Creating a Tetrahedral Mesh 629
16.2.1. Manual Meshing Procedure for Tetrahedral Meshes 629
16.3. Initializing the Tetrahedral Mesh 632
16.3.1. Initializing Using the Tet Dialog Box 633

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.4. Refining the Tetrahedral Mesh 634
16.4.1. Using Local Refinement Regions 635
16.4.2. Refinement Using the Tet Dialog Box 636
16.5. Common Tetrahedral Meshing Problems 637
17. Generating the Hexcore Mesh 641
17.1. Hexcore Meshing Procedure 641
17.2. Using the Hexcore Dialog Box 643
17.3. Controlling Hexcore Parameters 644
17.3.1. Maximum or Minimum Cell Length 644
17.3.2. Buffer Layers 644
17.3.3. Peel Layers 645
17.3.4. Defining Hexcore Extents 646
17.3.4.1. Hexcore to Selected Boundaries 647
17.3.5. Local Refinement Regions 649
18. Generating Polyhedral Meshes 651
18.1. Meshing Process for Polyhedral Meshes 651
18.2. Steps for Creating the Polyhedral Mesh 652
18.2.1. Further Mesh Improvements 655
18.2.2. Transferring the Poly Mesh to Solution Mode 655
19. Generating Poly-Hexcore Meshes 657
19.1. Steps for Creating the Poly-Hexcore Mesh 657
20. Generating the CutCell Mesh 661
20.1. The CutCell Meshing Process 661
20.2. Using the CutCell Dialog Box 666
20.2.1. Handling Zero-Thickness Walls 667
20.2.2. Handling Overlapping Surfaces 668
20.2.3. Resolving Thin Regions 669
20.3. Improving the CutCell Mesh 670
20.4. Post CutCell Mesh Generation Cleanup 671
20.5. Generating Prisms for the CutCell Mesh 671
20.6. The Cut-Tet Workflow 676
21. Generating Rapid Octree Meshes 679
21.1. Using the Rapid Octree Mesher 679
21.1.1. Geometry 681
21.1.1.1. Specifying the Input Object 681
21.1.1.2. Specifying the Volume 682
21.1.1.3. Defining the Bounding Box 682
21.1.1.4. Reporting the Base Length 684
21.1.2. Boundary Treatment 684
21.1.3. Mesh Parameters 686
21.1.3.1. Custom Boundary Sizes 688
21.1.3.1.1. Creating Size Functions 689
21.1.3.1.2. Draw, Change, and Delete Functions 689
21.1.3.2. Refinement Regions 690
21.2. Limitations of the Rapid Octree Mesher 690
22. Improving the Mesh 693
22.1. Smoothing Nodes 693
22.1.1. Laplace Smoothing 693
22.1.2. Variational Smoothing of Tetrahedral Meshes 694
22.1.3. Skewness-Based Smoothing of Tetrahedral Meshes 694
22.2. Swapping 695

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.3. Improving the Mesh 696
22.4. Removing Slivers from a Tetrahedral Mesh 696
22.4.1. Automatic Sliver Removal 697
22.4.2. Removing Slivers Manually 697
22.5. Modifying Cells 699
22.5.1. Using the Modify Cells Dialog Box 699
22.6. Moving Nodes 701
22.6.1. Automatic Correction 701
22.6.2. Semi-Automatic Correction 702
22.6.3. Repairing Negative Volume Cells 703
22.7. Cavity Remeshing 703
22.7.1. Tetrahedral Cavity Remeshing 704
22.7.2. Hexcore Cavity Remeshing 706
22.8. Manipulating Cell Zones 709
22.8.1. Active Zones and Cell Types 709
22.8.2. Copying and Moving Cell Zones 710
22.9. Manipulating Cell Zone Conditions 711
22.10. Using Domains to Group and Mesh Boundary Faces 711
22.10.1. Using Domains 711
22.10.2. Defining Domains 711
22.11. Checking the Mesh 712
22.12. Selectively Checking the Volume Mesh 713
22.13. Checking the Mesh Quality 716
22.14. Clearing the Mesh 716
23. Examining the Mesh 719
23.1. Displaying the Mesh 719
23.1.1. Generating the Mesh Display using Onscreen Tools 719
23.1.2. Generating the Mesh Display Using the Display Grid Dialog Box 721
23.1.2.1. Mesh Display Attributes 721
23.2. Controlling Display Options 724
23.3. Modifying and Saving the View 727
23.3.1. Mirroring a Non-symmetric Domain 727
23.3.2. Controlling Perspective and Camera Parameters 727
23.4. Composing a Scene 728
23.4.1. Changing the Display Properties 729
23.4.2. Transforming Geometric Entities in a Scene 729
23.4.3. Adding a Bounding Frame 729
23.4.4. Using the Scene Description Dialog Box 730
23.5. Controlling the Mouse Buttons 732
23.6. Controlling the Mouse Probe Function 735
23.7. Annotating the Display 737
23.8. Setting Default Controls 737
24. Determining Mesh Statistics and Quality 739
24.1. Determining Mesh Statistics 739
24.2. Determining Mesh Quality 740
24.2.1. Determining Surface Mesh Quality 740
24.2.2. Determining Volume Mesh Quality 741
24.2.3. Determining Boundary Cell Quality 742
24.2.4. Quality Measure 742
24.3. Reporting Mesh Information 751
A. Importing Boundary and Volume Meshes 753

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A.1. GAMBIT Meshes 753
A.2. TetraMesher Volume Mesh 753
A.3. Meshes from Third-Party CAD Packages 753
A.3.1. I-deas Universal Files 755
A.3.1.1. Recognized I-deas Datasets 755
A.3.1.2. Grouping Elements to Create Zones for a Surface Mesh 755
A.3.1.3. Grouping Nodes to Create Zones for a Volume Mesh 756
A.3.1.4. Periodic Boundaries 756
A.3.1.5. Deleting Duplicate Nodes 756
A.3.2. PATRAN Neutral Files 756
A.3.2.1. Recognized PATRAN Datasets 756
A.3.2.2. Grouping Elements to Create Zones 757
A.3.2.3. Periodic Boundaries 757
A.3.3. Ansys Files 757
A.3.3.1. Recognized Datasets 757
A.3.3.2. Periodic Boundaries 758
A.3.4. ARIES Files 758
A.3.5. NASTRAN Files 758
A.3.5.1. Recognized NASTRAN Bulk Data Entries 758
A.3.5.2. Periodic Boundaries 759
A.3.5.3. Deleting Duplicate Nodes 759
B. Mesh File Format 761
B.1. Guidelines 761
B.2. Formatting Conventions in Binary Files and Formatted Files 761
B.3. Grid Sections 762
B.3.1. Comment 762
B.3.2. Header 763
B.3.3. Dimensions 763
B.3.4. Nodes 763
B.3.5. Periodic Shadow Faces 764
B.3.6. Cells 765
B.3.7. Faces 766
B.3.8. Edges 768
B.3.9. Face Tree 769
B.3.10. Cell Tree 770
B.3.11. Interface Face Parents 770
B.4. Non-Grid Sections 771
B.4.1. Zone 771
B.5. Example Files 773
C. Shortcut Keys 777
C.1. Shortcut Key Actions 777
C.1.1. Entity Information 788
Bibliography 791
III. Solution Mode 793
Using This Manual ..... dccxcvii
1. Typographical Conventions ..... dccxcvii
2. Mathematical Conventions ..... dccxcix
1. Graphical User Interface (GUI) 801
1.1. GUI Components 801
1.1.1. The Ribbon 802
1.1.2. The Outline View 803

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.1.3. Graphics Windows 806
1.1.4. Quick Search 810
1.1.5. Toolbars 810
1.1.5.1. The Standard Toolbar 811
1.1.5.2. The Graphics Toolbars 811
1.1.5.2.1. Mesh Display 812
1.1.5.2.2. Pointer Tools 812
1.1.5.2.3. View Tools 813
1.1.5.2.4. Visibility Tools 814
1.1.5.2.5. Copy Tools 815
1.1.5.2.6. Object Selection/Display Tools 815
1.1.5.2.7. Graphics Effects Tools 816
1.1.5.2.8. Additional Display Options 816
1.1.6. Task Pages 816
1.1.7. The Console 817
1.1.8. Dialog Boxes 818
1.1.8.1. Input Controls 821
1.1.8.1.1. Tabs 821
1.1.8.1.2. Buttons 821
1.1.8.1.3. Check Boxes 821
1.1.8.1.4. Radio Buttons 821
1.1.8.1.5. Text Entry Boxes 821
1.1.8.1.6. Integer Number Entry Boxes 821
1.1.8.1.7. Real Number Entry Boxes 822
1.1.8.1.8. Filter Text Entry Boxes 822
1.1.8.1.9. Single-Selection Lists 822
1.1.8.1.10. Multiple-Selection Lists 823
1.1.8.1.11. Drop-Down Lists 824
1.1.8.1.12. Scales 824
1.1.8.2. Types of Dialog Boxes 825
1.1.8.2.1. Information Dialog Boxes 825
1.1.8.2.2. Warning Dialog Boxes 825
1.1.8.2.3. Error Dialog Boxes 826
1.1.8.2.4. The Working Dialog Box 826
1.1.8.2.5. Question Dialog Box 827
1.1.8.2.6. The Select File Dialog Box 827
1.1.9. Quick Property Editor for Boundaries 829
1.2. Customizing the Graphical User Interface 830
1.3. Setting User Preferences/Options 832
1.4. Fluent Graphical User Interface Other Languages 834
1.5. Having the Session Close After Sitting Idle 837
1.5.1. Timeout Using the Set Idle Timeout Dialog Box 837
1.5.2. Timeout Using FLUENT_MAX_IDLE_TIMEOUT 838
1.5.3. Idle Timeout Limitations 838
1.6. Using the Help System 838
1.6.1. Task Page and Dialog Box Help 839
1.6.2. Obtaining License Use Information 839
1.6.3. Version and Release Information 839
2. Text User Interface (TUI) 841
3. Reading and Writing Files 843
3.1. Shortcuts for Reading and Writing Files 844

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.1.1. Default File Suffixes 844
3.1.2. Binary Files 845
3.1.3. Detecting File Format 846
3.1.4. Recent File List 846
3.1.5. Reading and Writing Compressed Files 846
3.1.5.1. Reading Compressed Files 846
3.1.5.2. Writing Compressed Files 847
3.1.6. Tilde Expansion (Linux Systems Only) 848
3.1.7. Automatic Numbering of Files 848
3.1.8. Disabling the Overwrite Confirmation Prompt 849
3.2. Reading Mesh Files 849
3.3. Reading and Writing Case and Data Files 850
3.3.1. Reading and Writing Case Files 852
3.3.2. Reading and Writing Data Files 852
3.3.3. Reading and Writing Case and Data Files Together 853
3.3.4. Reading and Writing Files in the Legacy Format 853
3.3.5. Automatic Saving of Case and Data Files 854
3.4. Reading Fluent/UNS and RAMPANT Case and Data Files 857
3.5. Reading and Writing Profile Files 858
3.5.1. Reading Profile Files 858
3.5.2. Writing Profile Files 858
3.6. Reading and Writing Boundary Conditions 860
3.7. Writing a Boundary Mesh 861
3.8. Reading Scheme Source Files 861
3.9. Creating and Reading Journal Files 861
3.9.1. Procedure 863
3.9.2. Multiple Journal Files 864
3.10. Creating Transcript Files 865
3.11. Importing Files 865
3.11.1. ABAQUS Files 868
3.11.2. CFX Files 869
3.11.3. Meshes and Data in CGNS Format 869
3.11.4. EnSight Files 870
3.11.5. Ansys FIDAP Neutral Files 871
3.11.6. GAMBIT and GeoMesh Mesh Files 871
3.11.7. HYPERMESH ASCII Files 871
3.11.8. I-deas Universal Files 871
3.11.9. LSTC Files 872
3.11.10. Marc POST Files 872
3.11.11. Mechanical APDL Files 872
3.11.12. NASTRAN Files 873
3.11.13. PATRAN Neutral Files 873
3.11.14. PLOT3D Files 873
3.11.15. PTC Mechanica Design Files 874
3.11.16. Tecplot Files 874
3.11.17. Fluent 4 Case Files 874
3.11.18. PreBFC Files 875
3.11.19. Partition Files 875
3.11.20. CHEMKIN Mechanism 875
3.12. Exporting Solution Data 875
3.12.1. Exporting Limitations 876

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.13. Exporting Solution Data after a Calculation 877
3.13.1. ABAQUS Files 879
3.13.2. Mechanical APDL Input Files 879
3.13.3. ASCII Files 880
3.13.4. AVS Files 880
3.13.5. CDAT for CFD-Post and EnSight 880
3.13.6. CGNS Files 882
3.13.7. Common Fluids Format - Post Files 883
3.13.8. Data Explorer Files 884
3.13.9. EnSight Case Gold Files 884
3.13.10. FAST Files 887
3.13.11. FAST Solution Files 888
3.13.12. FieldView Unstructured Files 888
3.13.13. I-deas Universal Files 889
3.13.14. NASTRAN Files 890
3.13.15. PATRAN Files 891
3.13.16. TAITherm Files 891
3.13.17. Tecplot Files 892
3.14. Exporting Steady-State Particle History Data 892
3.15. Exporting Data During a Transient Calculation 894
3.15.1. Creating Automatic Export Definitions for Solution Data 896
3.15.2. Creating Automatic Export Definitions for Transient Particle History Data 898
3.16. Exporting to Ansys CFD-Post 900
3.17. Parallel Exporting to Ansys EnSight 901
3.18. Managing Solution Files 902
3.19. Mesh-to-Mesh Solution Interpolation 904
3.19.1. Performing Mesh-to-Mesh Solution Interpolation 904
3.19.2. Format of the Interpolation File 906
3.20. Mapping Data for Fluid-Structure Interaction (FSI) Applications 907
3.20.1. FEA File Formats 908
3.20.2. Using the FSI Mapping Dialog Boxes 908
3.21. Saving Picture Files 912
3.21.1. Using the Save Picture Dialog Box 913
3.21.1.1. Choosing the Picture File Format 915
3.21.1.2. Specifying the Color Mode 917
3.21.1.3. Choosing the File Type 917
3.21.1.4. Defining the Resolution 918
3.21.1.5. Picture Options 918
3.21.2. Picture Options for PostScript Files 918
3.21.2.1. Window Dumps (Linux Systems Only) 919
3.21.2.2. Previewing the Picture Image 919
3.22. Setting Data File Quantities 920
3.23. The .fluent File 922
3.24. Coupled Simulations in Ansys Fluent with Functional Mock-up Unit (FMU) Files 922
4. Unit Systems 927
4.1. Restrictions on Units 927
4.2. Units in Mesh Files 928
4.3. Built-In Unit Systems in Ansys Fluent 928
4.4. Customizing Units 929
4.4.1. Listing Current Units 929
4.4.2. Changing the Units for a Quantity 929

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.4.3. Defining a New Unit 929
4.4.3.1. Determining the Conversion Factor 930
5. Fluent Expressions Language 931
5.1. Introduction to Expressions 931
5.1.1. Expression Syntax 931
5.1.1.1. Expression Data Types 931
5.1.1.2. Expression Values 931
5.1.1.3. Expression Operations and Functions 932
5.1.2. Units Validation 938
5.2. Expression Sources 938
5.2.1. Field Variables 939
5.2.2. Solution Variables 939
5.2.3. Scientific Constants 940
5.2.4. Aliases 940
5.2.5. Profiles 941
5.3. Creating and Using Expressions 942
5.3.1. Directly Applied Expressions 943
5.3.1.1. Expressions for Cell Zones and Boundary Conditions 943
5.3.1.2. Expressions for Material Properties 946
5.3.1.3. Expressions in the Console 947
5.3.2. Named Expressions 947
5.3.3. Context Specification 951
5.3.4. Plotting Expressions 951
5.3.5. Postprocessing Expressions 953
5.3.6. Expression Manager 954
5.4. Expression Examples 955
5.4.1. Parabolic Inflow Profile 955
5.4.2. Time-Varied Parabolic Inflow 959
5.4.3. Controlled Outlet Temperature 962
5.4.4. Computing Forces with Parameterized Angle of Attack 966
5.5. Appendix: Supported Field Variables 969
6. Reading and Manipulating Meshes 1015
6.1. Mesh Topologies 1015
6.1.1. Examples of Acceptable Mesh Topologies 1017
6.1.2. Face-Node Connectivity in Ansys Fluent 1021
6.1.2.1. Face-Node Connectivity for Triangular Cells 1022
6.1.2.2. Face-Node Connectivity for Quadrilateral Cells 1023
6.1.2.3. Face-Node Connectivity for Tetrahedral Cells 1024
6.1.2.4. Face-Node Connectivity for Wedge Cells 1025
6.1.2.5. Face-Node Connectivity for Pyramidal Cells 1026
6.1.2.6. Face-Node Connectivity for Hex Cells 1027
6.1.2.7. Face-Node Connectivity for Polyhedral Cells 1028
6.1.3. Choosing the Appropriate Mesh Type 1028
6.1.3.1. Setup Time 1028
6.1.3.2. Computational Expense 1029
6.1.3.3. Numerical Diffusion 1030
6.2. Mesh Requirements and Considerations 1030
6.2.1. Geometry/Mesh Requirements 1030
6.2.2. Mesh Quality 1031
6.2.2.1. Mesh Element Distribution 1033
6.2.2.2. Cell Quality 1034

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.2.2.3. Smoothness 1035
6.2.2.4. Flow-Field Dependency 1035
6.3. Mesh Sources 1035
6.3.1. Ansys Meshing Mesh Files 1036
6.3.2. Fluent Meshing Mode Mesh Files 1036
6.3.3. Fluent Meshing Mesh Files 1036
6.3.4. GAMBIT Mesh Files 1036
6.3.5. GeoMesh Mesh Files 1037
6.3.6. PreBFC Mesh Files 1037
6.3.6.1. Structured Mesh Files 1037
6.3.6.2. Unstructured Triangular and Tetrahedral Mesh Files 1037
6.3.7. ICEM CFD Mesh Files 1037
6.3.8. I-deas Universal Files 1038
6.3.8.1. Recognized I-deas Datasets 1038
6.3.8.2. Grouping Nodes to Create Face Zones 1039
6.3.8.3. Grouping Elements to Create Cell Zones 1039
6.3.8.4. Deleting Duplicate Nodes 1039
6.3.9. NASTRAN Files 1039
6.3.9.1. Recognized NASTRAN Bulk Data Entries 1039
6.3.9.2. Deleting Duplicate Nodes 1040
6.3.10. PATRAN Neutral Files 1040
6.3.10.1. Recognized PATRAN Datasets 1041
6.3.10.2. Grouping Elements to Create Cell Zones 1041
6.3.11. Mechanical APDL Files 1041
6.3.11.1. Recognized Ansys 5.4 and 5.5 Datasets 1042
6.3.12. CFX Files 1042
6.3.13. Using the fe2ram Filter to Convert Files 1043
6.3.14. Removing Hanging Nodes/Edges 1044
6.3.14.1. Limitations 1044
6.3.15. Fluent/UNS and RAMPANT Case Files 1045
6.3.16. FLUENT 4 Case Files 1045
6.3.17. Ansys FIDAP Neutral Files 1046
6.3.18. Reading Multiple Mesh/Case/Data Files 1046
6.3.18.1. Reading Multiple Mesh Files via the Solution Mode of Fluent 1046
6.3.18.2. Reading Multiple Mesh Files via the Meshing Mode of Fluent 1047
6.3.18.3. Reading Multiple Mesh Files via tmerge 1048
6.3.19. Reading Surface Mesh Files 1050
6.4. Reference Frames 1050
6.4.1. Creating and Using Reference Frames 1051
6.5. Non-Conformal Meshes 1053
6.5.1. Non-Conformal Mesh Calculations 1054
6.5.1.1. The Periodic Boundary Condition Option 1056
6.5.1.2. The Periodic Repeats Option 1058
6.5.1.3. The Coupled Wall Option 1060
6.5.1.4. Matching Option 1061
6.5.1.5. The Mapped Option 1062
6.5.1.6. The Static Option 1064
6.5.1.7. Interface Zones Automatic Naming Conventions 1064
6.5.1.7.1. Default (No Options Enabled) 1065
6.5.1.7.2. Periodic Boundary Condition 1065
6.5.1.7.3. Periodic Repeats 1065

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.5.1.7.4. Coupled Wall 1065
6.5.1.7.5. Matching 1066
6.5.1.7.6. Mapped 1066
6.5.1.7.7. Static 1066
6.5.2. Non-Conformal Interface Algorithm 1067
6.5.3. Requirements and Limitations of Non-Conformal Meshes 1067
6.5.4. Using a Non-Conformal Mesh in Ansys Fluent 1069
6.5.4.1. Manually Creating Mesh Interfaces 1082
6.5.4.2. Transferring Motion Across a Mesh Interface 1084
6.6. Overset Meshes 1085
6.6.1. Introduction 1086
6.6.2. Overset Topologies 1087
6.6.3. Overset Domain Connectivity 1091
6.6.3.1. Hole Cutting 1091
6.6.3.1.1. Hole Cutting Control 1092
6.6.3.2. Overlap Minimization 1093
6.6.3.3. Donor Search 1095
6.6.4. Diagnosing Overset Interface Issues 1095
6.6.4.1. Flood Filling Fails During Hole Cutting 1096
6.6.4.1.1. Incorrect Seed Cells 1096
6.6.4.1.2. Leakage Between Overlapping Boundaries 1096
6.6.4.2. Donor Search Fails Due to Orphan Cells 1097
6.6.5. Overset Mesh Adaption 1098
6.6.5.1. Donor Size Adaption 1098
6.6.5.2. Orphan Adaption 1098
6.6.5.3. Using Manual Overset Adaption 1098
6.6.5.4. Using Automatic Overset Adaption 1099
6.6.5.5. Overset Adaption Controls 1099
6.6.6. Overset Meshing Best Practices 1100
6.6.7. Overset Meshing Limitations and Compatibilities 1102
6.6.7.1. Limitations 1102
6.6.7.2. Compatibilities 1102
6.6.8. Setting up an Overset Interface 1104
6.6.9. Postprocessing Overset Meshes 1106
6.6.9.1. Overset Mesh Display 1106
6.6.9.2. Overset Field Functions 1107
6.6.9.3. Overset Cell Marks 1110
6.6.9.4. Overset Interface Listing 1110
6.6.9.5. Overset Postprocessing Limitations 1110
6.6.10. Writing and Reading Overset Files 1111
6.7. Controlling Flow in Narrow Gaps and Valves 1111
6.7.1. The Gap Model Approach 1111
6.7.2. Limitations of the Gap Model 1111
6.7.3. Recommendations for the Setup of a Simulation with Gaps 1113
6.7.4. Using the Gap Model 1113
6.8. Checking the Mesh 1119
6.8.1. Mesh Check Report 1120
6.8.2. Repairing Meshes 1121
6.9. Reporting Mesh Statistics 1124
6.9.1. Mesh Size 1124
6.9.2. Memory Usage 1125

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.9.2.1. Linux Systems 1125
6.9.2.2. Windows Systems 1125
6.9.3. Mesh Zone Information 1126
6.9.4. Partition Statistics 1126
6.10. Converting the Mesh to a Polyhedral Mesh 1126
6.10.1. Converting the Domain to a Polyhedra 1127
6.10.1.1. Limitations 1131
6.10.2. Converting Skewed Cells to Polyhedra 1132
6.10.2.1. Limitations 1132
6.10.3. Converting Cells with Hanging Nodes / Edges to Polyhedra 1133
6.10.3.1. Limitations 1133
6.11. Modifying the Mesh 1134
6.11.1. Merging Zones 1135
6.11.1.1. When to Merge Zones 1135
6.11.1.2. Using the Merge Zones Dialog Box 1136
6.11.2. Separating Zones 1136
6.11.2.1. Separating Face Zones 1137
6.11.2.1.1. Methods for Separating Face Zones 1137
6.11.2.1.2. Inputs for Separating Face Zones 1137
6.11.2.2. Separating Cell Zones 1139
6.11.2.2.1. Methods for Separating Cell Zones 1139
6.11.2.2.2. Inputs for Separating Cell Zones 1139
6.11.3. Fusing Face Zones 1141
6.11.3.1. Inputs for Fusing Face Zones 1142
6.11.3.1.1. Fusing Zones on Branch Cuts 1143
6.11.4. Creating Periodic Zones and Interfaces 1143
6.11.5. Decoupling Periodic Zones 1146
6.11.6. Slitting Face Zones 1147
6.11.6.1. Inputs for Slitting Face Zones 1148
6.11.7. Orienting Face Zones 1148
6.11.8. Extruding Face Zones 1148
6.11.8.1. Specifying Extrusion by Displacement Distances 1149
6.11.8.2. Specifying Extrusion by Parametric Coordinates 1149
6.11.9. Replacing, Deleting, Deactivating, and Activating Zones 1149
6.11.9.1. Replacing Zones 1149
6.11.9.2. Deleting Zones 1151
6.11.9.3. Deactivating Zones 1151
6.11.9.4. Activating Zones 1152
6.11.10. Copying Cell Zones 1153
6.11.11. Replacing the Mesh 1153
6.11.11.1. Inputs for Replacing the Mesh 1154
6.11.11.2. Limitations 1155
6.11.12. Managing Adjacent Zones 1155
6.11.12.1. Renaming Zones Using the Adjacency Dialog Box 1156
6.11.13. Reordering the Domain 1159
6.11.14. Scaling the Mesh 1159
6.11.14.1. Scaling the Entire Mesh 1160
6.11.14.1.1. Changing the Unit of Length 1160
6.11.14.1.2. Unscaling the Mesh 1161
6.11.14.1.3. Changing the Physical Size of the Mesh 1161
6.11.14.2. Scaling Individual Cell Zones 1161

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.11.15. Translating the Mesh 1161
6.11.15.1. Translating the Entire Mesh 1162
6.11.15.2. Translating Individual Cell Zones 1162
6.11.16. Rotating the Mesh 1163
6.11.16.1. Rotating the Entire Mesh 1163
6.11.16.2. Rotating Individual Cell Zones 1164
6.11.17. Improving the Mesh by Smoothing and Swapping 1164
6.11.17.1. Smoothing 1164
6.11.17.1.1. Quality-Based Smoothing 1165
6.11.17.1.2. Laplacian Smoothing 1166
6.11.17.1.3. Skewness-Based Smoothing 1168
6.11.17.2. Face Swapping 1169
6.11.17.2.1. Triangular Meshes 1169
6.11.17.2.2. Tetrahedral Meshes 1170
6.11.17.3. Combining Skewness-Based Smoothing and Face Swapping 1171
7. Cell Zone and Boundary Conditions 1173
7.1. Overview 1173
7.1.1. Available Cell Zone and Boundary Types 1174
7.1.2. The Cell Zone and Boundary Conditions Task Pages 1174
7.1.3. Changing Cell and Boundary Zone Types 1175
7.1.4. Setting Cell Zone and Boundary Conditions 1177
7.1.5. Copying Cell Zone and Boundary Conditions 1178
7.1.6. Changing Cell or Boundary Zone Names 1179
7.1.7. Defining Non-Uniform Cell Zone and Boundary Conditions 1180
7.1.8. Defining and Viewing Parameters 1180
7.1.8.1. Creating a New Parameter 1183
7.1.8.2. Working With Advanced Parameter Options 1185
7.1.8.2.1. Defining Scheme Procedures With Input Parameters 1185
7.1.8.2.2. Defining UDFs With Input Parameters 1187
7.1.8.2.3. Using the Text User Interface to Define UDFs and Scheme Procedures With Input Parameters 1188
7.1.9. Selecting Cell or Boundary Zones in the Graphics Display 1189
7.1.10. Operating and Periodic Conditions 1191
7.1.11. Highlighting Selected Boundary Zones 1192
7.1.12. Saving and Reusing Cell Zone and Boundary Conditions 1192
7.2. Cell Zone Conditions 1192
7.2.1. Fluid Conditions 1193
7.2.1.1. Inputs for Fluid Zones 1193
7.2.1.1.1. Defining the Fluid Material 1194
7.2.1.1.2. Defining Sources 1194
7.2.1.1.3. Defining Fixed Values 1195
7.2.1.1.4. Specifying a Laminar Zone 1195
7.2.1.1.5. Specifying a Reaction Mechanism 1195
7.2.1.1.6. Specifying the Rotation Axis 1195
7.2.1.1.7. Defining Zone Motion 1196
7.2.1.1.8. Defining Radiation Parameters 1199
7.2.2. Solid Conditions 1199
7.2.2.1. Inputs for Solid Zones 1199
7.2.2.1.1. Defining the Solid Material 1200
7.2.2.1.2. Defining a Heat Source 1200
7.2.2.1.3. Defining a Fixed Temperature 1200

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.2.1.4. Specifying the Rotation Axis for Boundary Zones 1200
7.2.2.1.5. Defining Zone Motion 1201
7.2.2.1.6. Defining Radiation Parameters 1206
7.2.3. Porous Media Conditions 1206
7.2.3.1. Limitations and Assumptions of the Porous Media Model 1207
7.2.3.2. Momentum Equations for Porous Media 1208
7.2.3.2.1. Darcy's Law in Porous Media 1208
7.2.3.2.2. Inertial Losses in Porous Media 1209
7.2.3.3. Relative Viscosity in Porous Media 1210
7.2.3.4. Treatment of the Energy Equation in Porous Media 1210
7.2.3.4.1. Equilibrium Thermal Model Equations 1210
7.2.3.4.2. Non-Equilibrium Thermal Model Equations 1211
7.2.3.5. Treatment of Turbulence in Porous Media 1212
7.2.3.6. Effect of Porosity on Transient Scalar Equations 1212
7.2.3.7. Modeling Porous Media Based on Physical Velocity 1212
7.2.3.7.1. Single Phase Porous Media 1213
7.2.3.7.2. Multiphase Porous Media 1213
7.2.3.7.2.1. The Continuity Equation 1214
7.2.3.7.2.2. The Momentum Equation 1214
7.2.3.7.2.3. The Energy Equation 1214
7.2.3.8. User Inputs for Porous Media 1215
7.2.3.8.1. Defining the Porous Zone 1217
7.2.3.8.2. Defining the Porous Velocity Formulation 1217
7.2.3.8.3. Defining the Fluid Passing Through the Porous Medium 1217
7.2.3.8.4. Enabling Reactions in a Porous Zone 1217
7.2.3.8.5. Including the Relative Velocity Resistance Formulation 1218
7.2.3.8.6. Defining the Viscous and Inertial Resistance Coefficients 1218
7.2.3.8.7. Deriving Porous Media Inputs Based on Superficial Velocity, Using a Known Pressure Loss 1222
7.2.3.8.8. Using the Ergun Equation to Derive Porous Media Inputs for a Packed Bed 1223
7.2.3.8.9. Using an Empirical Equation to Derive Porous Media Inputs for Turbulent Flow Through a Perforated Plate 1223
7.2.3.8.10. Using Tabulated Data to Derive Porous Media Inputs for Laminar Flow Through a Fibrous Mat 1224
7.2.3.8.11. Deriving the Porous Coefficients Based on Experimental Pressure and Velocity Data 1224
7.2.3.8.12. Using the Power-Law Model 1226
7.2.3.8.13. Defining Porosity 1226
7.2.3.8.14. Specifying the Heat Transfer Settings 1226
7.2.3.8.14.1. Equilibrium Thermal Model 1226
7.2.3.8.14.2. Non-Equilibrium Thermal Model 1227
7.2.3.8.15. Specifying the Relative Viscosity 1229
7.2.3.8.16. Specifying the Relative Permeability 1229
7.2.3.8.17. Specifying the Capillary Pressure 1233
7.2.3.8.17.1. Brooks-Corey Model 1234
7.2.3.8.17.2. Van-Genuchten Model 1234
7.2.3.8.17.3. Leverett J-Function 1235
7.2.3.8.17.4. Skjaeveland Model 1235
7.2.3.8.17.5. Capillary Pressure Data in a Tabular Format 1236
7.2.3.8.17.5.1. Specifying Variables in a Tabular Format 1237
7.2.3.8.17.6. Capillary Pressure Usage 1238

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.3.8.18. Defining Sources 1241
7.2.3.8.19. Defining Fixed Values 1241
7.2.3.8.20. Suppressing the Turbulent Viscosity in the Porous Region 1241
7.2.3.8.21. Specifying the Rotation Axis and Defining Zone Motion 1241
7.2.3.9. Solution Strategies for Porous Media 1241
7.2.3.10. Postprocessing for Porous Media 1242
7.2.4. 3D Fan Zones 1243
7.2.4.1. Momentum Equations for 3D Fan Zones 1243
7.2.4.2. User Inputs for 3D Fan Zones 1245
7.2.4.2.1. Defining the Geometry of a 3D Fan Zone 1246
7.2.4.2.2. Defining the Properties of a 3D Fan Zone 1247
7.2.4.3. 3D Fan Zone Limitations 1248
7.2.5. Fixing the Values of Variables 1248
7.2.5.1. Overview of Fixing the Value of a Variable 1249
7.2.5.1.1. Variables That Can Be Fixed 1250
7.2.5.2. Procedure for Fixing Values of Variables in a Zone 1251
7.2.5.2.1. Fixing Velocity Components 1251
7.2.5.2.2. Fixing Temperature and Enthalpy 1252
7.2.5.2.3. Fixing Species Mass Fractions 1252
7.2.5.2.4. Fixing Turbulence Quantities 1252
7.2.5.2.5. Fixing User-Defined Scalars 1253
7.2.6. Locking the Temperature for Solid and Shell Zones 1253
7.2.7. Defining Mass, Momentum, Energy, and Other Sources 1253
7.2.7.1. Sign Conventions and Units 1254
7.2.7.2. Procedure for Defining Sources 1254
7.2.7.2.1. Mass Sources 1255
7.2.7.2.2. Momentum Sources 1256
7.2.7.2.3. Energy Sources 1256
7.2.7.2.4. Turbulence Sources 1256
7.2.7.2.4.1. Turbulence Sources for the k- ε Model 1256
7.2.7.2.4.2. Turbulence Sources for the Spalart-Allmaras Model 1256
7.2.7.2.4.3. Turbulence Sources for the k- ω Model 1256
7.2.7.2.4.4. Turbulence Sources for the Reynolds Stress Model 1256
7.2.7.2.5. Mean Mixture Fraction and Variance Sources 1257
7.2.7.2.6. P-1 Radiation Sources 1257
7.2.7.2.7. Progress Variable Sources 1257
7.2.7.2.8. NO, HCN, and NH3 Sources for the NOx Model 1257
7.2.7.2.9. User-Defined Scalar (UDS) Sources 1257
7.3. Operating Conditions 1258
7.3.1. Buoyancy-Driven Flows and Natural Convection 1258
7.3.1.1. Modeling Natural Convection in a Closed Domain 1258
7.3.1.2. The Boussinesq Model 1258
7.3.1.3. Limitations of the Boussinesq Model 1259
7.3.1.4. Steps in Solving Buoyancy-Driven Flow Problems 1259
7.3.1.5. If you are using the incompressible ideal gas law, check that the Operating Density 1261
7.3.1.5.1. Setting the Operating Density for a Single Phase Flow 1261
7.3.1.5.2. Setting the Operating Density for a Multiphase Flow 1262
7.3.1.6. Solution Strategies for Buoyancy-Driven Flows 1262
7.3.1.6.1. Guidelines for Solving High-Rayleigh-Number Flows 1262
7.4. Boundary Conditions 1263

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.1. Flow Inlet and Exit Boundary Conditions 1264
7.4.2. Using Flow Boundary Conditions 1264
7.4.2.1. Determining Turbulence Parameters 1265
7.4.2.1.1. Specification of Turbulence Quantities Using Profiles 1265
7.4.2.1.2. Uniform Specification of Turbulence Quantities 1266
7.4.2.1.3. Turbulence Intensity 1266
7.4.2.1.4. Turbulence Length Scale and Hydraulic Diameter 1267
7.4.2.1.5. Turbulent Viscosity Ratio 1268
7.4.2.1.6. Relationships for Deriving Turbulence Quantities 1268
7.4.2.1.7. Estimating Modified Turbulent Viscosity from Turbulence Intensity and Length Scale 1268
7.4.2.1.8. Estimating Turbulent Kinetic Energy from Turbulence Intensity 1268
7.4.2.1.9. Estimating Turbulent Dissipation Rate from a Length Scale 1269
7.4.2.1.10. Estimating Turbulent Dissipation Rate from Turbulent Viscosity Ratio 1269
7.4.2.1.11. Estimating Turbulent Dissipation Rate for Decaying Turbulence 1269
7.4.2.1.12. Estimating Specific Dissipation Rate from a Length Scale 1270
7.4.2.1.13. Estimating Specific Dissipation Rate from Turbulent Viscosity Ratio 1270
7.4.2.1.14. Estimating Reynolds Stress Components from Turbulent Kinetic Energy 1270
7.4.2.1.15. Specifying Inlet Turbulence for LES 1270
7.4.3. Pressure Inlet Boundary Conditions 1271
7.4.3.1. Inputs at Pressure Inlet Boundaries 1271
7.4.3.1.1. Summary 1271
7.4.3.1.1.1. Pressure Inputs and Hydrostatic Head 1272
7.4.3.1.1.2. Defining Total Pressure and Temperature 1273
7.4.3.1.1.3. Defining the Flow Direction 1274
7.4.3.1.1.4. Defining Static Pressure 1277
7.4.3.1.1.5. Prevent Reverse Flow 1277
7.4.3.1.1.6. Defining Turbulence Parameters 1278
7.4.3.1.1.7. Defining Radiation Parameters 1278
7.4.3.1.1.8. Defining Species Mass or Mole Fractions 1278
7.4.3.1.1.9. Defining Non-Premixed Combustion Parameters 1278
7.4.3.1.1.10. Defining Premixed Combustion Boundary Conditions 1278
7.4.3.1.1.11. Defining Discrete Phase Boundary Conditions 1278
7.4.3.1.1.12. Defining Multiphase Boundary Conditions 1278
7.4.3.1.1.13. Defining Open Channel Boundary Conditions 1278
7.4.3.2. Default Settings at Pressure Inlet Boundaries 1278
7.4.3.3. Calculation Procedure at Pressure Inlet Boundaries 1279
7.4.3.3.1. Incompressible Flow Calculations at Pressure Inlet Boundaries 1279
7.4.3.3.2. Compressible Flow Calculations at Pressure Inlet Boundaries 1279
7.4.4. Velocity Inlet Boundary Conditions 1280
7.4.4.1. Inputs at Velocity Inlet Boundaries 1280
7.4.4.1.1. Summary 1280
7.4.4.1.2. Defining the Velocity 1282
7.4.4.1.3. Setting the Velocity Magnitude and Direction 1282
7.4.4.1.4. Setting the Velocity Magnitude Normal to the Boundary 1282
7.4.4.1.5. Setting the Velocity Components 1283
7.4.4.1.6. Setting the Angular Velocity 1283
7.4.4.1.7. Defining Static Pressure 1283
7.4.4.1.8. Defining the Temperature 1284
7.4.4.1.9. Defining Outflow Gauge Pressure 1284
7.4.4.1.10. Defining Turbulence Parameters 1284

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.4.1.11. Defining Radiation Parameters 1284
7.4.4.1.12. Defining Species Mass or Mole Fractions 1284
7.4.4.1.13. Defining Non-Premixed Combustion Parameters 1284
7.4.4.1.14. Defining Premixed Combustion Boundary Conditions 1284
7.4.4.1.15. Defining Discrete Phase Boundary Conditions 1284
7.4.4.1.16. Defining Multiphase Boundary Conditions 1285
7.4.4.2. Default Settings at Velocity Inlet Boundaries 1285
7.4.4.3. Calculation Procedure at Velocity Inlet Boundaries 1285
7.4.4.3.1. Treatment of Velocity Inlet Conditions at Flow Inlets 1285
7.4.4.3.2. Treatment of Velocity Inlet Conditions at Flow Exits 1286
7.4.4.3.3. Density Calculation 1286
7.4.5. Mass-Flow Inlet Boundary Conditions 1286
7.4.5.1. Limitations and Special Considerations 1287
7.4.5.2. Inputs at Mass-Flow Inlet Boundaries 1287
7.4.5.2.1. Summary 1287
7.4.5.2.2. Selecting the Reference Frame 1288
7.4.5.2.3. Defining the Mass Flow Rate or Mass Flux 1288
7.4.5.2.4. More About Mass Flux and Average Mass Flux 1289
7.4.5.2.5. Defining the Total Temperature 1290
7.4.5.2.6. Defining Static Pressure 1290
7.4.5.2.7. Defining the Flow Direction 1291
7.4.5.2.8. Defining Turbulence Parameters 1292
7.4.5.2.9. Defining Radiation Parameters 1292
7.4.5.2.10. Defining Species Mass or Mole Fractions 1292
7.4.5.2.11. Defining Non-Premixed Combustion Parameters 1292
7.4.5.2.12. Defining Premixed Combustion Boundary Conditions 1292
7.4.5.2.13. Defining Discrete Phase Boundary Conditions 1292
7.4.5.2.14. Defining Open Channel Boundary Conditions 1292
7.4.5.3. Default Settings at Mass-Flow Inlet Boundaries 1293
7.4.5.4. Calculation Procedure at Mass-Flow Inlet Boundaries 1293
7.4.5.4.1. Flow Calculations at Mass Flow Boundaries for Ideal Gases 1293
7.4.5.4.2. Flow Calculations at Mass Flow Boundaries for Incompressible Flows 1294
7.4.5.4.3. Flux Calculations at Mass Flow Boundaries 1294
7.4.6. Mass-Flow Outlet Boundary Conditions 1294
7.4.6.1. Limitations 1294
7.4.6.2. Inputs at Mass-Flow Outlet Boundaries 1294
7.4.6.2.1. Summary 1294
7.4.6.2.2. Selecting the Reference Frame 1295
7.4.6.2.3. Defining the Mass Flow Rate or Mass Flux 1295
7.4.6.2.4. Defining Radiation Parameters 1297
7.4.6.2.5. Defining Discrete Phase Boundary Conditions 1297
7.4.6.3. Default Settings at Mass-Flow Outlet Boundaries 1297
7.4.6.4. Calculation Procedure at Mass-Flow Outlet Boundaries 1297
7.4.6.4.1. Exit Corrected Mass Flow Rate 1298
7.4.7. Inlet Vent Boundary Conditions 1298
7.4.7.1. Inputs at Inlet Vent Boundaries 1298
7.4.7.1.1. Specifying the Loss Coefficient 1300
7.4.8. Intake Fan Boundary Conditions 1301
7.4.8.1. Inputs at Intake Fan Boundaries 1301
7.4.8.1.1. Specifying the Pressure Jump 1303
7.4.9. Pressure Outlet Boundary Conditions 1303

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.9.1. Inputs at Pressure Outlet Boundaries 1303
7.4.9.1.1. Summary 1303
7.4.9.1.2. Defining Static Pressure 1305
7.4.9.1.3. Defining Backflow Conditions 1306
7.4.9.1.3.1. Prevent Reverse Flow 1308
7.4.9.1.4. Defining Radiation Parameters 1308
7.4.9.1.5. Defining Discrete Phase Boundary Conditions 1308
7.4.9.1.6. Defining Open Channel Boundary Conditions 1308
7.4.9.2. Default Settings at Pressure Outlet Boundaries 1308
7.4.9.3. Calculation Procedure at Pressure Outlet Boundaries 1309
7.4.9.3.1. Average Pressure Specification 1309
7.4.9.3.1.1. Strong Averaging 1309
7.4.9.3.1.2. Weak Averaging 1310
7.4.9.4. Other Optional Inputs at Pressure Outlet Boundaries 1311
7.4.9.4.1. Non-Reflecting Boundary Conditions Option 1311
7.4.9.4.2. Target Mass Flow Rate Option 1311
7.4.10. Pressure Far-Field Boundary Conditions 1313
7.4.10.1. Limitations 1313
7.4.10.2. Inputs at Pressure Far-Field Boundaries 1314
7.4.10.2.1. Summary 1314
7.4.10.2.2. Defining Static Pressure, Mach Number, and Static Temperature 1315
7.4.10.2.3. Defining the Flow Direction 1315
7.4.10.2.4. Defining Turbulence Parameters 1316
7.4.10.2.5. Defining Radiation Parameters 1316
7.4.10.2.6. Defining Species Transport Parameters 1316
7.4.10.3. Defining Discrete Phase Boundary Conditions 1316
7.4.10.4. Default Settings at Pressure Far-Field Boundaries 1316
7.4.10.5. Calculation Procedure at Pressure Far-Field Boundaries 1316
7.4.10.6. Tangency Correction 1317
7.4.11. Outflow Boundary Conditions 1317
7.4.11.1. Ansys Fluent's Treatment at Outflow Boundaries 1318
7.4.11.2. Using Outflow Boundaries 1318
7.4.11.3. Mass Flow Split Boundary Conditions 1320
7.4.11.4. Other Inputs at Outflow Boundaries 1320
7.4.11.4.1. Radiation Inputs at Outflow Boundaries 1320
7.4.11.4.2. Defining Discrete Phase Boundary Conditions 1321
7.4.12. Outlet Vent Boundary Conditions 1321
7.4.12.1. Inputs at Outlet Vent Boundaries 1321
7.4.12.1.1. Specifying the Loss Coefficient 1322
7.4.13. Exhaust Fan Boundary Conditions 1323
7.4.13.1. Inputs at Exhaust Fan Boundaries 1323
7.4.13.1.1. Specifying the Pressure Jump 1324
7.4.14. Degassing Boundary Conditions 1325
7.4.14.1. Limitations 1326
7.4.14.2. Inputs at Degassing Boundaries 1326
7.4.15. Wall Boundary Conditions 1326
7.4.15.1. Inputs at Wall Boundaries 1327
7.4.15.1.1. Summary 1327
7.4.15.2. Wall Motion 1327
7.4.15.2.1. Defining a Stationary Wall 1328
7.4.15.2.2. Velocity Conditions for Moving Walls 1328

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.15.2.3. Shear Conditions at Walls 1330
7.4.15.2.4. No-Slip Walls 1330
7.4.15.2.5. Specified Shear 1330
7.4.15.2.6. Specularity Coefficient 1331
7.4.15.2.7. Marangoni Stress 1332
7.4.15.2.8. Partial Slip for Rarefied Gases 1333
7.4.15.2.9. Wall Roughness Effects in Turbulent Wall-Bounded Flows 1335
7.4.15.2.9.1. Standard Law-of-the-Wall Modified for Roughness 1335
7.4.15.2.9.1.1. Setting the Roughness Parameters 1339
7.4.15.2.9.2. Additional Roughness Models for Icing Simulations 1340
7.4.15.2.9.2.1. Specified Roughness 1341
7.4.15.2.9.2.2. NASA Correlation 1341
7.4.15.2.9.2.3. Shin-et-al 1341
7.4.15.2.9.2.4. ICE3D Roughness File 1341
7.4.15.3. Thermal Boundary Conditions at Walls 1342
7.4.15.3.1. Heat Flux Boundary Conditions 1343
7.4.15.3.2. Temperature Boundary Conditions 1343
7.4.15.3.3. Convective Heat Transfer Boundary Conditions 1343
7.4.15.3.4. External Radiation Boundary Conditions 1344
7.4.15.3.5. Combined Convection and External Radiation Boundary Conditions 1344
7.4.15.3.6. Augmented Heat Transfer 1344
7.4.15.3.7. Thin-Wall Thermal Resistance Parameters 1344
7.4.15.3.8. Thermal Conditions for Two-Sided Walls 1346
7.4.15.3.8.1. Orthogonality-Based Secondary Gradient Limiting at Coupled Two-Sided Walls 1347
7.4.15.3.9. Shell Conduction 1348
7.4.15.3.10. Heat Transfer Boundary Conditions Through System Coupling 1349
7.4.15.3.11. Heat Transfer Boundary Conditions Across a Mapped Interface 1349
7.4.15.3.12. Temperature Jump for Rarefied Gases 1350
7.4.15.4. Species Boundary Conditions for Walls 1352
7.4.15.4.1. Reaction Boundary Conditions for Walls 1353
7.4.15.5. Radiation Boundary Conditions for Walls 1353
7.4.15.6. Discrete Phase Model (DPM) Boundary Conditions for Walls 1354
7.4.15.6.1. Wall Adhesion Contact Angle for VOF Model 1354
7.4.15.7. User-Defined Scalar (UDS) Boundary Conditions for Walls 1354
7.4.15.8. Wall Film Conditions for Walls 1354
7.4.15.9. Structural Model Conditions for Walls 1354
7.4.15.10. Default Settings at Wall Boundaries 1354
7.4.15.11. Shear-Stress Calculation Procedure at Wall Boundaries 1355
7.4.15.11.1. Shear-Stress Calculation in Laminar Flow 1355
7.4.15.11.2. Shear-Stress Calculation in Turbulent Flows 1355
7.4.15.12. Heat Transfer Calculations at Wall Boundaries 1355
7.4.15.12.1. Temperature Boundary Conditions 1355
7.4.15.12.2. Heat Flux Boundary Conditions 1356
7.4.15.12.3. Convective Heat Transfer Boundary Conditions 1356
7.4.15.12.4. External Radiation Boundary Conditions 1356
7.4.15.12.5. Combined External Convection and Radiation Boundary Conditions 1357
7.4.15.12.6. Calculation of the Fluid-Side Heat Transfer Coefficient 1357
7.4.15.13. The Ablation Condition at Wall Boundaries 1357
7.4.16. Perforated Wall Boundary Conditions 1360
7.4.16.1. Overview and Limitations 1360

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.16.2. Modeling Concept 1360
7.4.16.3. Setting Perforated Walls 1362
7.4.16.4. Procedure for Manual Setup of Perforated Walls 1364
7.4.16.5. Perforated Wall File Format 1370
7.4.16.6. Postprocessing for Perforated Walls 1373
7.4.17. Symmetry Boundary Conditions 1373
7.4.17.1. Examples of Symmetry Boundaries 1374
7.4.17.2. Calculation Procedure at Symmetry Boundaries 1375
7.4.18. Periodic Boundary Conditions 1376
7.4.18.1. Examples of Periodic Boundaries 1376
7.4.18.2. Inputs for Periodic Boundaries 1376
7.4.18.3. Default Settings at Periodic Boundaries 1378
7.4.18.4. Calculation Procedure at Periodic Boundaries 1378
7.4.19. Axis Boundary Conditions 1378
7.4.19.1. Calculation Procedure at Axis Boundaries 1379
7.4.20. Fan Boundary Conditions 1379
7.4.20.1. Limitations of Fan Boundary Conditions 1379
7.4.20.2. Fan Equations 1379
7.4.20.2.1. Modeling the Pressure Rise Across the Fan 1379
7.4.20.2.2. Modeling the Fan Swirl Velocity 1380
7.4.20.3. User Inputs for Fans 1380
7.4.20.3.1. Identifying the Fan Zone 1381
7.4.20.3.2. Defining the Pressure Jump 1381
7.4.20.3.2.1. Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function 1382
7.4.20.3.2.2. Constant Value 1383
7.4.20.3.2.3. User-Defined Function or Profile 1383
7.4.20.3.2.4. Example: Determining the Pressure Jump Function 1383
7.4.20.3.3. Defining Discrete Phase Boundary Conditions for the Fan 1384
7.4.20.3.4. Defining the Fan Swirl Velocity 1385
7.4.20.3.4.1. Polynomial Function 1385
7.4.20.3.4.2. Constant Value 1385
7.4.20.3.4.3. User-Defined Function or Profile 1386
7.4.20.4. Postprocessing for Fans 1386
7.4.20.4.1. Reporting the Pressure Rise Through the Fan 1386
7.4.20.4.2. Graphical Plots 1386
7.4.21. Radiator Boundary Conditions 1386
7.4.21.1. Radiator Equations 1387
7.4.21.1.1. Modeling the Pressure Loss Through a Radiator 1387
7.4.21.1.2. Modeling the Heat Transfer Through a Radiator 1387
7.4.21.1.2.1. Calculating the Heat Transfer Coefficient 1388
7.4.21.2. User Inputs for Radiators 1388
7.4.21.2.1. Identifying the Radiator Zone 1389
7.4.21.2.2. Defining the Pressure Loss Coefficient Function 1389
7.4.21.2.2.1. Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function 1390
7.4.21.2.2.2. Constant Value 1390
7.4.21.2.2.3. Example: Calculating the Loss Coefficient 1390
7.4.21.2.3. Defining the Heat Flux Parameters 1392
7.4.21.2.3.1. Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function 1392
7.4.21.2.3.2. Constant Value 1392
7.4.21.2.3.3. Example: Determining the Heat Transfer Coefficient Function 1392
7.4.21.2.4. Defining Discrete Phase Boundary Conditions for the Radiator 1393

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.21.3. Postprocessing for Radiators 1393
7.4.21.3.1. Reporting the Radiator Pressure Drop 1393
7.4.21.3.2. Reporting Heat Transfer in the Radiator 1393
7.4.21.3.3. Graphical Plots 1393
7.4.22. Porous Jump Boundary Conditions 1394
7.4.22.1. User Inputs for the Porous Jump Model 1394
7.4.22.1.1. Identifying the Porous Jump Zone 1395
7.4.22.1.2. Defining Discrete Phase Boundary Conditions for the Porous Jump 1396
7.4.22.2. Postprocessing for the Porous Jump 1396
7.5. Editing Multiple Boundary Conditions at Once 1396
7.6. Boundary Acoustic Wave Models 1398
7.6.1. Turbo-Specific Non-Reflecting Boundary Conditions 1399
7.6.1.1. Overview 1399
7.6.1.2. Limitations 1399
7.6.1.3. Theory 1402
7.6.1.3.1. Equations in Characteristic Variable Form 1402
7.6.1.3.2. Inlet Boundary 1404
7.6.1.3.3. Outlet Boundary 1406
7.6.1.3.4. Updated Flow Variables 1407
7.6.1.4. Using Turbo-Specific Non-Reflecting Boundary Conditions 1407
7.6.1.4.1. Using the NRBCs with the Mixing-Plane Model 1409
7.6.1.4.2. Using the NRBCs in Parallel Ansys Fluent 1409
7.6.2. General Non-Reflecting Boundary Conditions 1409
7.6.2.1. Overview 1409
7.6.2.2. Restrictions and Limitations 1409
7.6.2.3. Theory 1410
7.6.2.4. Using the General Non-Reflecting Boundary Condition 1414
7.6.3. Impedance Boundary Conditions 1416
7.6.3.1. Restrictions and Limitations 1416
7.6.3.2. Theory 1416
7.6.3.3. Using the Impedance Boundary Condition 1417
7.6.3.4. Calculating Impedance Parameters 1419
7.6.4. Transparent Flow Forcing Boundary Conditions 1424
7.6.4.1. Restrictions and Limitations 1425
7.6.4.2. Theory 1425
7.6.4.3. Using the Transparent Flow Forcing Boundary Condition 1425
7.7. User-Defined Fan Model 1427
7.7.1. Steps for Using the User-Defined Fan Model 1427
7.7.2. Example of a User-Defined Fan 1428
7.7.2.1. Setting the User-Defined Fan Parameters 1428
7.7.2.2. Sample User-Defined Fan Program 1430
7.7.2.3. Initializing the Flow Field and Profile Files 1431
7.7.2.4. Selecting the Profiles 1431
7.7.2.5. Performing the Calculation 1432
7.7.2.6. Results 1433
7.8. Profiles 1434
7.8.1. Profile Specification Types 1434
7.8.2. Profile File Formats 1435
7.8.2.1. Standard Profiles 1436
7.8.2.1.1. Example 1437
7.8.2.2. CSV Profiles 1437

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.8.3. Using Profiles 1439
7.8.3.1. Checking and Deleting Profiles 1441
7.8.3.2. Viewing Profile Data 1442
7.8.3.3. Example 1442
7.8.4. Reorienting Profiles 1443
7.8.4.1. Steps for Changing the Profile Orientation 1443
7.8.4.2. Profile Orienting Example 1446
7.8.5. Replicating Profiles 1448
7.8.5.1. Steps for Replicating a Profile 1448
7.8.6. Defining Transient Cell Zone and Boundary Conditions 1451
7.8.6.1. Standard Transient Profiles 1451
7.8.6.2. Tabular Transient Profiles 1452
7.8.6.3. Profiles for Moving and Deforming Meshes 1453
7.9. Coupling Boundary Conditions with GT-POWER 1454
7.9.1. Requirements and Restrictions 1454
7.9.2. User Inputs 1455
7.9.3. Torque-Speed Coupling with GT-POWER 1458
7.10. Coupling Boundary Conditions with WAVE 1459
7.10.1. Requirements and Restrictions 1459
7.10.2. User Inputs 1460
8. Physical Properties 1463
8.1. Defining Materials 1463
8.1.1. Physical Properties for Solid Materials 1464
8.1.2. Material Types and Databases 1464
8.1.3. Using the **Create/Edit Materials** Dialog Box 1465
8.1.3.1. Modifying Properties of an Existing Material 1467
8.1.3.2. Renaming an Existing Material 1468
8.1.3.3. Copying Materials from the Ansys Fluent Database 1468
8.1.3.4. 2.2 Copying Materials from the Ansys GRANTA MDS Database 1470
8.1.3.5. Creating a New Material 1472
8.1.3.6. Saving Materials and Properties 1472
8.1.3.7. Deleting a Material 1473
8.1.3.8. Changing the Order of the Materials List 1473
8.1.4. Using a User-Defined Materials Database 1473
8.1.4.1. Opening a User-Defined Database 1474
8.1.4.2. Viewing Materials in a User-Defined Database 1475
8.1.4.3. Copying Materials from a User-Defined Database 1475
8.1.4.4. Copying Materials from the Case to a User-Defined Database 1476
8.1.4.5. Modifying Properties of an Existing Material 1477
8.1.4.6. Creating a New Materials Database and Materials 1478
8.1.4.7. Deleting Materials from a Database 1481
8.2. Defining Properties Using Temperature-Dependent Functions 1482
8.2.1. Inputs for Polynomial Functions 1483
8.2.2. Inputs for Piecewise-Linear Functions 1484
8.2.3. Inputs for Piecewise-Polynomial Functions 1486
8.2.4. Inputs for NASA-9-Piecewise-Polynomial Functions 1488
8.2.5. Checking and Modifying Existing Profiles 1490
8.3. Density 1490
8.3.1. Defining Density for Various Flow Regimes 1491
8.3.1.1. Mixing Density Relationships in Multiple-Zone Models 1491
8.3.2. Input of Constant Density 1492

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.3.3. Inputs for the Boussinesq Approximation 1492
8.3.4. Compressible Liquid Density Method 1492
8.3.4.1. Compressible Liquid Inputs 1493
8.3.4.2. Compressible Liquid Density Method Availability 1496
8.3.5. Density as a Profile Function of Temperature 1496
8.3.6. Incompressible Ideal Gas Law 1496
8.3.6.1. Density Inputs for the Incompressible Ideal Gas Law 1497
8.3.7. Ideal Gas Law for Compressible Flows 1497
8.3.7.1. Density Inputs for the Ideal Gas Law for Compressible Flows 1498
8.3.8. Composition-Dependent Density for Multicomponent Mixtures 1498
8.4. Viscosity 1500
8.4.1. Input of Constant Viscosity 1501
8.4.2. Viscosity as a Function of Temperature 1501
8.4.2.1. Sutherland Viscosity Law 1502
8.4.2.1.1. Inputs for Sutherland's Law 1502
8.4.2.2. Power-Law Viscosity Law 1503
8.4.2.2.1. Inputs for the Power Law 1503
8.4.3. Defining the Viscosity Using Kinetic Theory 1504
8.4.4. Composition-Dependent Viscosity for Multicomponent Mixtures 1504
8.4.5. Viscosity for Non-Newtonian Fluids 1505
8.4.5.1. Temperature Dependent Viscosity 1506
8.4.5.2. Power Law for Non-Newtonian Viscosity 1507
8.4.5.2.1. Inputs for the Non-Newtonian Power Law 1507
8.4.5.3. The Carreau Model for Pseudo-Plastics 1507
8.4.5.3.1. Inputs for the Carreau Model 1508
8.4.5.4. Cross Model 1508
8.4.5.4.1. Inputs for the Cross Model 1509
8.4.5.5. Herschel-Bulkley Model for Bingham Plastics 1509
8.4.5.5.1. Inputs for the Herschel-Bulkley Model 1510
8.5. Thermal Conductivity 1510
8.5.1. Constant Thermal Conductivity 1511
8.5.2. Thermal Conductivity as a Function of Temperature 1512
8.5.3. Thermal Conductivity Using Kinetic Theory 1512
8.5.4. Composition-Dependent Thermal Conductivity for Multicomponent Mixtures 1512
8.5.5. Anisotropic Thermal Conductivity for Solids 1513
8.5.5.1. Anisotropic Thermal Conductivity 1514
8.5.5.2. Biaxial Thermal Conductivity 1515
8.5.5.3. Orthotropic Thermal Conductivity 1516
8.5.5.4. Cylindrical Orthotropic Thermal Conductivity 1518
8.5.5.5. Principal Axes and Principal Values 1519
8.5.5.6. User-Defined Anisotropic Thermal Conductivity 1521
8.6. User-Defined Scalar (UDS) Diffusivity 1522
8.6.1. Isotropic Diffusion 1522
8.6.2. Anisotropic Diffusion 1523
8.6.2.1. Anisotropic Diffusivity 1524
8.6.2.2. Orthotropic Diffusivity 1525
8.6.2.3. Cylindrical Orthotropic Diffusivity 1527
8.6.3. User-Defined Anisotropic Diffusivity 1528
8.7. Specific Heat Capacity 1529
8.7.1. Input of Constant Specific Heat Capacity 1529
8.7.2. Specific Heat Capacity as a Function of Temperature 1529

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.7.3. Defining Specific Heat Capacity Using Kinetic Theory 1530
8.7.4. Specific Heat Capacity as a Function of Composition 1530
8.8. Radiation Properties 1531
8.8.1. Absorption Coefficient 1531
8.8.1.1. Inputs for a Constant Absorption Coefficient 1532
8.8.1.2. Inputs for a Composition-Dependent Absorption Coefficient 1532
8.8.1.2.1. Path Length Inputs 1532
8.8.1.2.1.1. Inputs for a Non-Gray Radiation Absorption Coefficient 1532
8.8.1.2.1.2. Effect of Particles and Soot on the Absorption Coefficient 1533
8.8.2. Scattering Coefficient 1533
8.8.2.1. Inputs for a Constant Scattering Coefficient 1533
8.8.2.2. Inputs for the Scattering Phase Function 1533
8.8.2.2.1. Isotropic Phase Function 1533
8.8.2.2.2. Linear-Anisotropic Phase Function 1533
8.8.2.2.3. Delta-Eddington Phase Function 1534
8.8.2.2.4. User-Defined Phase Function 1534
8.8.3. Refractive Index 1534
8.8.4. Reporting the Radiation Properties 1534
8.9. Mass Diffusion Coefficients 1534
8.9.1. Fickian Diffusion 1535
8.9.2. Full Multicomponent Diffusion 1536
8.9.2.1. General Theory 1536
8.9.2.2. Maxwell-Stefan Equations 1536
8.9.3. Anisotropic Species Diffusion 1537
8.9.4. Thermal Diffusion Coefficients 1538
8.9.4.1. Thermal Diffusion Coefficient Inputs 1538
8.9.5. Mass Diffusion Coefficient Inputs 1539
8.9.5.1. Constant Dilute Approximation Inputs 1540
8.9.5.2. Dilute Approximation Inputs 1541
8.9.5.3. Multicomponent Method Inputs 1541
8.9.5.4. Unity Lewis Number 1543
8.9.6. Mass Diffusion Coefficient Inputs for Turbulent Flow 1544
8.10. Standard State Enthalpies 1544
8.11. Standard State Entropies 1544
8.12. Unburnt Thermal Diffusivity 1545
8.13. Kinetic Theory Parameters 1545
8.13.1. Inputs for Kinetic Theory 1546
8.14. Operating Pressure 1546
8.14.1. The Significance of Operating Pressure 1546
8.14.2. Operating Pressure, Gauge Pressure, and Absolute Pressure 1547
8.14.3. Setting the Operating Pressure 1547
8.15. Using a Reference Pressure to Adjust the Gauge Pressure Field 1548
8.16. Real Gas Models 1550
8.16.1. Introduction 1550
8.16.2. Choosing a Real Gas Model 1552
8.16.3. Cubic Equation of State Models 1553
8.16.3.1. Overview and Limitations 1553
8.16.3.2. Equation of State 1555
8.16.3.3. Enthalpy, Entropy, and Specific Heat Calculations 1556
8.16.3.4. Critical Constants for Pure Components 1558
8.16.3.5. Calculations for Mixtures 1559

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.16.3.5.1. Using the Cubic Equation of State Real Gas Models 1561
8.16.3.5.2. Solution Strategies and Considerations for Cubic Equations of State Real Gas Models 1565
8.16.3.5.3. Using the Cubic Equation of State Models with the Lagrangian Dispersed Phase Models 1567
8.16.3.5.4. Postprocessing the Cubic Equations of State Real Gas Model 1569
8.16.4. The NIST Real Gas Models 1570
8.16.4.1. Limitations of the NIST Real Gas Models 1570
8.16.4.2. The REFPROP v9.1 Database 1571
8.16.4.3. Using the NIST Real Gas Models 1573
8.16.4.3.1. Activating the NIST Real Gas Model 1573
8.16.4.3.2. Creating Full NIST Look-up Tables 1575
8.16.4.3.3. Creating Binary Mixture Saturation Tables for Binary Mixtures 1578
8.16.4.3.4. Changing the REFPROP Library and Fluid Files 1580
8.16.4.4. Solution Strategies and Considerations for NIST Real Gas Model Simulation 1580
8.16.4.4.1. Writing Your Case File 1581
8.16.4.4.2. Postprocessing 1581
8.16.5. The User-Defined Real Gas Model 1581
8.16.5.1. Limitations of the User-Defined Real Gas Model 1581
8.16.5.2. Writing the UDRGM C Function Library 1584
8.16.5.3. Compiling Your UDRGM C Functions and Building a Shared Library File 1588
8.16.5.3.1. Compiling the UDRGM Using the Graphical Interface 1588
8.16.5.3.2. Compiling the UDRGM Using the Text Interface 1589
8.16.5.3.3. Loading the UDRGM Shared Library File 1590
8.16.5.4. UDRGM Example: Ideal Gas Equation of State 1590
8.16.5.4.1. Ideal Gas UDRGM Code Listing 1591
8.16.5.5. Additional UDRGM Examples 1594
8.16.6. Using Real Gas Property (RGP) Table Files 1594
8.16.6.1. Overview 1594
8.16.6.2. Defining Material Properties Using RGP Tables 1594
8.16.6.3. Defining Saturation Properties via RGP Tables 1595
9. Modeling Basic Fluid Flow 1597
9.1. User-Defined Scalar (UDS) Transport Equations 1597
9.1.1. Introduction 1597
9.1.2. UDS Theory 1597
9.1.2.1. Single Phase Flow 1598
9.1.2.2. Multiphase Flow 1598
9.1.3. Setting Up UDS Equations in Ansys Fluent 1599
9.1.3.1. Single Phase Flow 1600
9.1.3.2. Multiphase Flow 1605
9.2. Periodic Flows 1607
9.2.1. Overview and Limitations 1607
9.2.1.1. Overview 1607
9.2.1.2. Limitations for Modeling Streamwise-Periodic Flow 1608
9.2.2. User Inputs for the Pressure-Based Solver 1609
9.2.2.1. Setting Parameters for the Calculation of β 1611
9.2.3. User Inputs for the Density-Based Solvers 1611
9.2.4. Monitoring the Value of the Pressure Gradient 1612
9.2.5. Postprocessing for Streamwise-Periodic Flows 1612
9.3. Swirling and Rotating Flows 1613
9.3.1. Overview of Swirling and Rotating Flows 1614

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.3.1.1. Axisymmetric Flows with Swirl or Rotation 1614
9.3.1.1.1. Momentum Conservation Equation for Swirl Velocity 1614
9.3.1.2. Three-Dimensional Swirling Flows 1614
9.3.1.3. Flows Requiring a Moving Reference Frame 1615
9.3.2. Turbulence Modeling in Swirling Flows 1615
9.3.3. Mesh Setup for Swirling and Rotating Flows 1616
9.3.3.1. Coordinate System Restrictions 1616
9.3.3.2. Mesh Sensitivity in Swirling and Rotating Flows 1616
9.3.4. Modeling Axisymmetric Flows with Swirl or Rotation 1616
9.3.4.1. Problem Setup for Axisymmetric Swirling Flows 1617
9.3.4.2. Solution Strategies for Axisymmetric Swirling Flows 1617
9.3.4.2.1. Step-By-Step Solution Procedures for Axisymmetric Swirling Flows 1618
9.3.4.2.2. Improving Solution Stability by Gradually Increasing the Rotational or Swirl Speed 1619
9.3.4.2.2.1. Postprocessing for Axisymmetric Swirling Flows 1619
9.4. Compressible Flows 1619
9.4.1. When to Use the Compressible Flow Model 1620
9.4.2. Physics of Compressible Flows 1621
9.4.2.1. Basic Equations for Compressible Flows 1621
9.4.2.2. The Compressible Form of the Gas Law 1621
9.4.3. Modeling Inputs for Compressible Flows 1622
9.4.3.1. Boundary Conditions for Compressible Flows 1623
9.4.4. Floating Operating Pressure 1623
9.4.4.1. Limitations 1624
9.4.4.2. Theory 1624
9.4.4.3. Enabling Floating Operating Pressure 1624
9.4.4.4. Setting the Initial Value for the Floating Operating Pressure 1624
9.4.4.5. Storage and Reporting of the Floating Operating Pressure 1625
9.4.4.6. Monitoring Absolute Pressure 1625
9.4.5. Solution Strategies for Compressible Flows 1625
9.4.6. Reporting of Results for Compressible Flows 1626
9.5. Inviscid Flows 1626
9.5.1. Setting Up an Inviscid Flow Model 1626
9.5.2. Solution Strategies for Inviscid Flows 1627
9.5.3. Postprocessing for Inviscid Flows 1627
10. Modeling Flows with Moving Reference Frames 1629
10.1. Introduction 1629
10.2. Flow in Single Moving Reference Frames (SRF) 1631
10.2.1. Mesh Setup for a Single Moving Reference Frame 1632
10.2.2. Setting Up a Single Moving Reference Frame Problem 1632
10.2.2.1. Choosing the Relative or Absolute Velocity Formulation 1635
10.2.2.1.1. Example 1635
10.2.3. Solution Strategies for a Single Moving Reference Frame 1636
10.2.3.1. Gradual Increase of the Rotational Speed to Improve Solution Stability 1637
10.2.4. Postprocessing for a Single Moving Reference Frame 1637
10.3. Flow in Multiple Moving Reference Frames 1639
10.3.1. The Multiple Reference Frame Model 1639
10.3.1.1. Overview 1639
10.3.1.2. Limitations 1640
10.3.2. Mesh Setup for a Multiple Moving Reference Frame 1641
10.3.3. Setting Up a Multiple Moving Reference Frame Problem 1641

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.3.3.1. Setting Up Multiple Reference Frames 1642
10.3.4. Solution Strategies for MRF and Problems 1645
10.3.5. Postprocessing for MRF Problems 1646
11. Modeling Flows Using Sliding and Dynamic Meshes 1647
11.1. Introduction 1647
11.2. Sliding Mesh Examples 1648
11.3. The Sliding Mesh Technique 1650
11.4. Sliding Mesh Interface Shapes 1651
11.5. Using Sliding Meshes 1654
11.5.1. Requirements, Constraints, and Considerations 1654
11.5.2. Setting Up the Sliding Mesh Problem 1655
11.5.3. Solution Strategies for Sliding Meshes 1658
11.5.3.1. Saving Case and Data Files 1659
11.5.3.2. Time-Periodic Solutions 1659
11.5.4. Postprocessing for Sliding Meshes 1660
11.6. Using Dynamic Meshes 1662
11.6.1. Setting Dynamic Mesh Modeling Parameters 1664
11.6.2. Dynamic Mesh Update Methods 1666
11.6.2.1. Smoothing Methods 1666
11.6.2.1.1. Diffusion-Based Smoothing 1667
11.6.2.1.1.1. Diffusivity Based on Boundary Distance 1671
11.6.2.1.1.2. Diffusivity Based on Cell Volume 1673
11.6.2.1.1.3. Applicability of the Diffusion-Based Smoothing Method 1674
11.6.2.1.2. Spring-Based Smoothing 1674
11.6.2.1.2.1. Applicability of the Spring-Based Smoothing Method 1678
11.6.2.1.3. Linearly Elastic Solid Based Smoothing Method 1678
11.6.2.1.3.1. Applicability of the Linearly Elastic Solid Based Smoothing Method 1680
11.6.2.1.4. Smoothing from a Reference Position 1680
11.6.2.1.5. Laplacian Smoothing Method 1681
11.6.2.1.6. Boundary Layer Smoothing Method 1681
11.6.2.2. Dynamic Layering 1684
11.6.2.2.1. Applicability of the Dynamic Layering Method 1687
11.6.2.3. Remeshing 1688
11.6.2.3.1. Methods-Based Remeshing 1691
11.6.2.3.1.1. Local Remeshing Method 1691
11.6.2.3.1.1.1. Local Cell Remeshing Method 1692
11.6.2.3.1.1.2. Local Face Remeshing Method 1692
11.6.2.3.1.1.3. Local Remeshing Based on Sizing Function 1693
11.6.2.3.1.2. Cell Zone Remeshing Method 1699
11.6.2.3.1.2.1. Limitations of the Cell Zone Remeshing Method 1699
11.6.2.3.1.3. Face Region Remeshing Method 1699
11.6.2.3.1.3.1. Face Region Remeshing with Wedge Cells in Prism Layers 1701
11.6.2.3.1.3.2. Applicability of the Face Region Remeshing Method 1703
11.6.2.3.1.4. 2.5D Surface Remeshing Method 1704
11.6.2.3.1.4.1. Applicability of the 2.5D Surface Remeshing Method 1705
11.6.2.3.1.4.2. Using the 2.5D Model 1706
11.6.2.3.2. Unified Remeshing 1708
11.6.2.4. Volume Mesh Update Procedure 1709
11.6.2.5. Transient Considerations for Remeshing and Layering 1709
11.6.3. Feature Detection 1710
11.6.3.1. Applicability of Feature Detection 1710

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.6.4. In-Cylinder Settings 1711
11.6.4.1. Using the In-Cylinder Option 1716
11.6.4.1.1. Overview 1716
11.6.4.1.2. Defining the Mesh Topology 1717
11.6.4.1.3. Defining Motion/Geometry Attributes of Mesh Zones 1719
11.6.4.1.4. Defining Valve Opening and Closure 1725
11.6.5. Six DOF Solver Settings 1725
11.6.5.1. Setting Rigid Body Motion Attributes for the Six DOF Solver 1727
11.6.6. Implicit Update Settings 1729
11.6.7. Contact Detection Settings 1731
11.6.7.1. Flow Control Using Contact Zones 1733
11.6.7.2. Flow Control Using Contact Marks 1733
11.6.7.2.1. Selecting Parameters for Flow Control 1734
11.6.7.2.2. Modifying and Displaying Contact Cell Marks 1735
11.6.8. Defining Dynamic Mesh Events 1735
11.6.8.1. Procedure for Defining Events 1736
11.6.8.2. Defining Events for In-Cylinder Applications 1738
11.6.8.2.1. Events 1739
11.6.8.2.2. Changing the Zone Type 1739
11.6.8.2.3. Copying Zone Boundary Conditions 1739
11.6.8.2.4. Activating a Cell Zone 1739
11.6.8.2.5. Deactivating a Cell Zone 1739
11.6.8.2.6. Creating a Sliding Interface 1739
11.6.8.2.7. Deleting a Sliding Interface 1741
11.6.8.2.8. Changing the Motion Attribute of a Dynamic Zone 1741
11.6.8.2.9. Changing the Time Step Size 1741
11.6.8.2.10. Changing the Under-Relaxation Factor 1741
11.6.8.2.11. Inserting a Boundary Zone Layer 1741
11.6.8.2.12. Removing a Boundary Zone Layer 1742
11.6.8.2.13. Inserting an Interior Zone Layer 1742
11.6.8.2.14. Removing an Interior Zone Layer 1743
11.6.8.2.15. Inserting a Cell Layer 1744
11.6.8.2.16. Removing a Cell Layer 1744
11.6.8.2.17. Executing a Command 1744
11.6.8.2.18. Replacing the Mesh 1744
11.6.8.2.19. Resetting Inert EGR 1744
11.6.8.2.20. Diesel Unsteady Flamelet Reset 1744
11.6.8.3. Exporting and Importing Events 1745
11.6.9. Specifying the Motion of Dynamic Zones 1745
11.6.9.1. General Procedure 1745
11.6.9.1.1. Creating a Dynamic Zone 1745
11.6.9.1.2. Modifying a Dynamic Zone 1745
11.6.9.1.3. Checking the Center of Gravity 1745
11.6.9.1.4. Deleting a Dynamic Zone 1745
11.6.9.2. Stationary Zones 1746
11.6.9.3. Rigid Body Motion 1747
11.6.9.4. Deforming Motion 1751
11.6.9.5. User-Defined Motion 1754
11.6.9.5.1. Specifying Boundary Layer Deformation Smoothing 1755
11.6.9.6. System Coupling Motion 1756
11.6.9.7. Intrinsic FSI Motion 1757

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.6.9.8. Solution Stabilization for Dynamic Mesh Boundary Zones 1759
11.6.9.9. Solid-Body Kinematics 1760
11.6.10. Previewing the Dynamic Mesh 1763
11.6.10.1. Previewing Zone Motion 1763
11.6.10.2. Previewing Mesh Motion 1764
11.6.11. Steady-State Dynamic Mesh Applications 1765
11.6.11.1. An Example of Steady-State Dynamic Mesh Usage 1767
12. Modeling Turbomachinery Flows 1771
12.1. Frozen Gust / Inlet Disturbance Flow Modeling 1771
12.2. Blade Row Interaction Modeling 1772
12.2.1. Pitch-Change Models 1774
12.2.1.1. Pitch-Scale interface 1775
12.2.1.2. No Pitch-Scale interface 1776
12.2.1.3. Mixing-Plane interface 1776
12.2.2. Creating and Editing General Turbo Interfaces 1778
12.2.3. Legacy Mixing Plane Model 1781
12.2.3.1. Limitations 1782
12.2.3.2. Setting Up the Legacy Mixing Plane Model 1782
12.2.3.2.1. Modeling Options 1785
12.2.3.2.1.1. Fixing the Pressure Level for an Incompressible Flow 1785
12.2.3.2.1.2. Conserving Swirl Across the Mixing Plane 1786
12.2.3.2.1.3. Conserving Total Enthalpy Across the Mixing Plane 1786
12.2.3.3. Solution Strategies for Mixing Plane Problems 1787
12.3. Aerodynamic Damping (Blade Flutter Analysis) 1787
12.3.1. Traveling Wave Mode and Energy Method 1787
12.3.2. Setting up a Blade Flutter Case 1789
12.3.3. Reading the Mode Shapes 1790
12.3.4. Using Dynamic Mesh Zones in a Blade Flutter Simulation 1792
12.3.4.1. Turning on Dynamic Mesh 1792
12.3.4.2. Defining the Periodic Displacement of the Blades 1793
12.3.4.3. Creating and Applying Dynamic Mesh Zones 1799
12.3.5. Visualizing and Exporting Blade Flutter Harmonics 1803
12.3.6. Configuring Run Calculation Settings 1803
12.3.7. Postprocessing a Blade Flutter Simulation 1805
12.3.7.1. Postprocessing Harmonic Variables 1806
12.4. Turbomachinery Postprocessing 1807
12.4.1. Defining the Turbomachinery Topology 1807
12.4.1.1. Boundary Types 1810
12.4.2. Generating Reports of Turbomachinery Data 1811
12.4.2.1. Computing Turbomachinery Quantities 1812
12.4.2.1.1. Mass Flow 1812
12.4.2.1.2. Swirl Number 1813
12.4.2.1.3. Average Total Pressure 1813
12.4.2.1.4. Average Total Temperature 1813
12.4.2.1.5. Average Flow Angles 1814
12.4.2.1.6. Passage Loss Coefficient 1815
12.4.2.1.7. Axial Force 1815
12.4.2.1.8. Torque 1815
12.4.2.1.9. Efficiencies for Pumps and Compressors 1816
12.4.2.1.9.1. Incompressible Flows 1816
12.4.2.1.9.2. Compressible Flows 1817

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12.4.2.1.10. Efficiencies for Turbines 1818
12.4.2.1.10.1. Incompressible Flows 1818
12.4.2.1.10.2. Compressible Flows 1818
12.4.3. Displaying Turbomachinery Averaged Contours 1819
12.4.3.1. Steps for Generating Turbomachinery Averaged Contour Plots 1819
12.4.3.2. Contour Plot Options 1821
12.4.4. Displaying Turbomachinery 2D Contours 1821
12.4.4.1. Steps for Generating Turbo 2D Contour Plots 1822
12.4.4.2. Contour Plot Options 1823
12.4.5. Generating Averaged XY Plots of Turbomachinery Solution Data 1823
12.4.5.1. Steps for Generating Turbo Averaged XY Plots 1823
12.4.6. Globally Setting the Turbomachinery Topology 1825
12.4.7. Turbomachinery-Specific Variables 1826
13. Modeling Turbulence 1827
13.1. Introduction 1827
13.2. Choosing a Turbulence Model 1830
13.2.1. Reynolds Averaged Navier-Stokes (RANS) Turbulence Models 1830
13.2.1.1. Spalart-Allmaras One-Equation Model 1831
13.2.1.2. k-ε Models 1831
13.2.1.3. k-ω Models 1831
13.2.1.4. Generalized k-ω (GEKO) Model 1832
13.2.1.5. Reynold Stress Models 1834
13.2.1.6. Laminar-Turbulent Transition Models 1835
13.2.1.7. Curvature Correction for the Spalart-Allmaras and Two-Equation Models 1837
13.2.1.8. Corner Flow Correction 1837
13.2.1.9. Production Limiters for Two-Equation Models 1837
13.2.1.10. Model Enhancements 1837
13.2.1.11. Wall Treatment for RANS Models 1838
13.2.1.12. Grid Resolution for RANS Models 1838
13.2.2. Scale-Resolving Simulation (SRS) Models 1839
13.2.2.1. Large Eddy Simulation (LES) 1839
13.2.2.2. Hybrid RANS-LES Models 1839
13.2.2.2.1. Scale-Adaptive Simulation (SAS) 1840
13.2.2.2.2. Detached Eddy Simulation (DES) 1841
13.2.2.2.3. Shielded Detached Eddy Simulation (SDES) and Stress-Blended Eddy Simulation (SBES) 1842
13.2.2.3. Zonal Modeling and Embedded LES (ELES) 1842
13.2.3. Grid Resolution SRS Models 1842
13.2.3.1. Wall Boundary Layers 1842
13.2.3.2. Free Shear Flows 1843
13.2.4. Numerics Settings for SRS Models 1843
13.2.4.1. Time Discretization 1843
13.2.4.2. Spatial Discretization 1844
13.2.4.3. Iterative Scheme 1845
13.2.4.3.1. Convergence Control 1845
13.2.5. Model Hierarchy 1846
13.3. Steps in Using a Turbulence Model 1846
13.4. Setting Up the Spalart-Allmaras Model 1850
13.5. Setting Up the k-ε Model 1851
13.5.1. Setting Up the Standard or Realizable k-ε Model 1851
13.5.2. Setting Up the RNG k-ε Model 1853

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.6. Setting Up the k-ω Model 1855
13.6.1. Setting Up the Standard k-ω Model 1855
13.6.2. Setting Up the Baseline (BSL) k-ω Model 1856
13.6.3. Setting Up the Shear-Stress Transport k-ω Model 1858
13.6.4. Setting up the Generalized k-ω (GEKO) Model 1860
13.7. Setting Up the Transition k-kl-ω Model 1862
13.8. Setting Up the Transition SST Model 1863
13.9. Setting Up the Algebraic or Intermittency Transition Model 1864
13.10. Setting Up the Reynolds Stress Model 1867
13.11. Setting Up Scale-Adaptive Simulation (SAS) Modeling 1871
13.12. Setting Up the Detached Eddy Simulation Model 1873
13.12.1. Setting Up DES with the Spalart-Allmaras Model 1873
13.12.2. Setting Up DES with the Realizable k-ε Model 1875
13.12.3. Setting Up DES with the SST k-ω Model 1877
13.12.4. Setting Up DES with the BSL k-ω Model 1878
13.12.5. Setting Up DES with the Transition SST Model 1880
13.13. Setting Up the Large Eddy Simulation Model 1882
13.14. Model Constants 1883
13.15. Setting Up the Embedded Large Eddy Simulation (ELES) Model 1883
13.16. Setup Options for All Turbulence Modeling 1887
13.16.1. Including the Viscous Heating Effects 1888
13.16.2. Including Buoyancy Effects on Turbulence 1888
13.16.3. Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models 1890
13.16.4. Including Corner Flow Correction 1891
13.16.5. Including the Compressibility Effects Option 1893
13.16.6. Including Production Limiters for Two-Equation Models 1893
13.16.7. Vorticity- and Strain/Vorticity-Based Production 1894
13.16.8. Delayed Detached Eddy Simulation (DDES) 1894
13.16.9. Differential Viscosity Modification 1894
13.16.10. Swirl Modification 1894
13.16.11. Low-Re Corrections 1895
13.16.12. Shear Flow Corrections 1895
13.16.13. Turbulence Damping 1895
13.16.14. Including Pressure Gradient Effects 1895
13.16.15. Including Thermal Effects 1896
13.16.16. Including the Wall Reflection Term 1896
13.16.17. Solving the k Equation to Obtain Wall Boundary Conditions 1896
13.16.18. Quadratic Pressure-Strain Model 1896
13.16.19. Stress-Omega and Stress-BSL Models 1897
13.16.20. Subgrid-Scale Model 1897
13.16.21. Customizing the Turbulent Viscosity 1897
13.16.22. Customizing the Turbulent Prandtl and Schmidt Numbers 1898
13.16.23. Modeling Turbulence with Non-Newtonian Fluids 1898
13.16.24. Including Scale-Adaptive Simulation with ω-Based URANS Models 1898
13.16.25. Including Detached Eddy Simulation with the Transition SST Model 1898
13.16.26. Including the SDES or SBES Model with RANS Models 1899
13.16.27. Shielding Functions for the BSL / SST / Transition SST Detached Eddy Simulation Model 1902
13.17. Defining Turbulence Boundary Conditions 1902
13.17.1. Wall Roughness Effects 1902

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.17.2. The Spalart-Allmaras Model 1903
13.17.3. k-ε Models and k-ω Models 1903
13.17.4. Reynolds Stress Model 1903
13.17.5. Large Eddy Simulation Model 1905
13.18. Providing an Initial Guess for k and ε (or k and ω) 1906
13.19. Solution Strategies for Turbulent Flow Simulations 1906
13.19.1. Mesh Generation 1907
13.19.2. Accuracy 1907
13.19.3. Convergence 1907
13.19.4. RSM-Specific Solution Strategies 1908
13.19.4.1. Under-Relaxation of the Reynolds Stresses 1908
13.19.4.2. Disabling Calculation Updates of the Reynolds Stresses 1909
13.19.4.3. Residual Reporting for the RSM 1909
13.19.5. LES-Specific Solution Strategies 1909
13.19.5.1. Temporal Discretization 1910
13.19.5.2. Spatial Discretization 1910
13.20. Postprocessing for Turbulent Flows 1910
13.20.1. Custom Field Functions for Turbulence 1919
13.20.2. Postprocessing Turbulent Flow Statistics 1919
13.20.3. Troubleshooting 1921
14. Modeling Thermal Energy 1923
14.1. Introduction 1923
14.2. Modeling Conductive and Convective Heat Transfer 1923
14.2.1. Solving Heat Transfer Problems 1924
14.2.1.1. Limiting the Predicted Temperature Range 1926
14.2.1.2. Modeling Heat Transfer in Two Separated Fluid Regions 1927
14.2.2. Solution Strategies for Heat Transfer Modeling 1927
14.2.2.1. Under-Relaxation of the Energy Equation 1927
14.2.2.2. Under-Relaxation of Temperature When the Enthalpy Equation is Solved 1928
14.2.2.3. Disabling the Species Diffusion Term 1928
14.2.2.4. Step-by-Step Solutions 1928
14.2.2.4.1. Decoupled Flow and Heat Transfer Calculations 1928
14.2.2.4.2. Coupled Flow and Heat Transfer Calculations 1929
14.2.2.5. Transient Conjugate Heat Transfer 1929
14.2.2.5.1. Specifying the Solid Time Step Size 1929
14.2.2.5.1.1. Automatic Time Step Size Calculation 1931
14.2.2.5.2. Loosely Coupled Conjugate Heat Transfer 1931
14.2.2.5.3. Time Averaged Explicit Thermal Coupling 1934
14.2.3. Postprocessing Heat Transfer Quantities 1935
14.2.3.1. Available Variables for Postprocessing 1936
14.2.3.2. Definition of Enthalpy and Energy in Reports and Displays 1936
14.2.3.3. Reporting Heat Transfer Through Boundaries 1936
14.2.3.4. Reporting Heat Transfer Through a Surface 1936
14.2.3.5. Reporting Averaged Heat Transfer Coefficients 1937
14.2.3.6. Exporting Heat Flux Data 1937
14.2.4. Natural Convection 1937
14.2.5. Shell Conduction 1937
14.2.5.1. Introduction 1938
14.2.5.2. Physical Treatment 1938
14.2.5.3. Limitations of Shell Conduction Walls 1939
14.2.5.4. Managing Conduction Walls 1940

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.2.5.5. Initializing Shells 1943
14.2.5.6. Locking the Temperature for Shells 1943
14.2.5.7. Postprocessing Shells 1943
14.3. Modeling Radiation 1945
14.3.1. Using the Radiation Models 1946
14.3.2. Setting Up the P-1 Model with Non-Gray Radiation 1948
14.3.3. Setting Up the DTRM 1949
14.3.3.1. Defining the Rays 1949
14.3.3.2. Controlling the Clusters 1951
14.3.3.3. Controlling the Rays 1951
14.3.3.4. Writing and Reading the DTRM Ray File 1951
14.3.3.5. Displaying the Clusters 1952
14.3.4. Setting Up the S2S Model 1952
14.3.4.1. View Factors and Clustering Settings 1954
14.3.4.1.1. Forming Surface Clusters 1955
14.3.4.1.1.1. Setting the Split Angle for Clusters 1958
14.3.4.1.2. Setting Up the View Factor Calculation 1958
14.3.4.1.2.1. Selecting the Basis for Computing View Factors 1958
14.3.4.1.2.2. Selecting the Method for Computing View Factors 1959
14.3.4.1.2.3. Accounting for Blocking Surfaces 1960
14.3.4.1.2.4. Specifying Boundary Zone Participation 1960
14.3.4.2. Computing View Factors 1963
14.3.4.3. Reading View Factors into Ansys Fluent 1965
14.3.5. Setting Up the DO Model 1965
14.3.5.1. Angular Discretization 1966
14.3.5.2. Defining Non-Gray Radiation for the DO Model 1966
14.3.5.3. Enabling DO/Energy Coupling 1968
14.3.6. Setting Up the MC Model 1968
14.3.7. Defining Material Properties for Radiation 1970
14.3.7.1. Absorption Coefficient for a Non-Gray Model 1971
14.3.7.2. Refractive Index for a Non-Gray Model 1971
14.3.8. Defining Boundary Conditions for Radiation 1971
14.3.8.1. Inlet and Exit Boundary Conditions 1971
14.3.8.1.1. Emissivity 1971
14.3.8.1.2. Black Body Temperature 1972
14.3.8.2. Wall Boundary Conditions for the DTRM, P-1, S2S, and Rosseland Models 1972
14.3.8.2.1. Boundary Conditions for the S2S Model 1973
14.3.8.3. Wall Boundary Conditions for the DO Model 1973
14.3.8.3.1. Opaque Walls 1973
14.3.8.3.2. Semi-Transparent Walls 1975
14.3.8.4. Wall Boundary Conditions for the MC Model 1979
14.3.8.4.1. Opaque Walls 1979
14.3.8.4.2. Semi-Transparent Walls 1985
14.3.8.5. Solid Cell Zones Conditions for the DO or MC Models 1987
14.3.8.6. Thermal Boundary Conditions 1988
14.3.9. Solution Strategies for Radiation Modeling 1988
14.3.9.1. P-1 Model Solution Parameters 1988
14.3.9.2. DTRM Solution Parameters 1989
14.3.9.3. S2S Solution Parameters 1991
14.3.9.4. DO Solution Parameters 1991
14.3.9.5. MC Solution Parameters 1991

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.3.9.6. Running the Calculation 1992
14.3.9.6.1. Residual Reporting for the P-1 Model 1992
14.3.9.6.2. Residual Reporting for the DO Model 1992
14.3.9.6.3. Residual Reporting for the DTRM 1992
14.3.9.6.4. Residual Reporting for the S2S Model 1993
14.3.9.6.5. Disabling the Update of the Radiation Fluxes 1993
14.3.10. Postprocessing Radiation Quantities 1993
14.3.10.1. Available Variables for Postprocessing 1994
14.3.10.2. Reporting Radiative Heat Transfer Through Boundaries 1995
14.3.10.3. Overall Heat Balances When Using the DTRM 1995
14.3.10.4. Displaying Rays and Clusters for the DTRM 1995
14.3.10.4.1. Displaying Clusters 1996
14.3.10.4.2. Displaying Rays 1996
14.3.10.4.3. Including the Mesh in the Display 1997
14.3.10.5. Reporting Radiation in the S2S Model 1997
14.3.11. Solar Load Model 1998
14.3.11.1. Introduction 1999
14.3.11.2. Solar Ray Tracing 1999
14.3.11.2.1. Shading Algorithm 2000
14.3.11.2.2. Glazing Materials 2001
14.3.11.2.3. Inputs 2001
14.3.11.3. Solar Irradiation 2003
14.3.11.4. Solar Calculator 2003
14.3.11.4.1. Inputs/Outputs 2003
14.3.11.4.2. Theory 2004
14.3.11.4.3. Computation of Load Distribution 2005
14.3.11.5. Using the Solar Load Model 2006
14.3.11.5.1. User-Defined Functions (UDFs) for Solar Load 2006
14.3.11.5.2. Setting Up the Solar Load Model 2007
14.3.11.5.3. Setting Boundary Conditions for Solar Loading 2012
14.3.11.5.4. Solar Ray Tracing 2012
14.3.11.5.5. Solar Irradiation 2017
14.3.11.5.6. Text Interface-Only Commands 2019
14.3.11.5.6.1. Automatically Saving Solar Ray Tracing Data 2019
14.3.11.5.6.2. Automatically Reading Solar Data 2019
14.3.11.5.6.3. Aligning the Camera Direction With the Position of the Sun 2020
14.3.11.5.6.4. Specifying the Scattering Fraction 2020
14.3.11.5.6.5. Applying the Solar Load on Adjacent Fluid Cells 2020
14.3.11.5.6.6. Specifying Quad Tree Refinement Factor 2021
14.3.11.5.6.7. Specifying Ground Reflectivity 2021
14.3.11.5.6.8. Reverting to Single Band Implementation of DO Model 2021
14.3.11.5.6.9. Additional Text Interface Commands 2021
14.3.11.6. Postprocessing Solar Load Quantities 2022
14.3.11.6.1. Solar Load Animation at Different Sun Positions 2023
14.3.11.6.2. Reporting and Displaying Solar Load Quantities 2024
14.4. Modeling Periodic Heat Transfer 2025
14.4.1. Overview and Limitations 2025
14.4.1.1. Overview 2025
14.4.1.2. Constraints for Periodic Heat Transfer Predictions 2025
14.4.2. Theory 2026

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.4.2.1. Definition of the Periodic Temperature for Constant- Temperature Wall Conditions 2026
14.4.2.2. Definition of the Periodic Temperature Change σ for Specified Heat Flux Conditions 2027
14.4.3. Using Periodic Heat Transfer 2027
14.4.4. Solution Strategies for Periodic Heat Transfer 2029
14.4.5. Monitoring Convergence 2029
14.4.6. Postprocessing for Periodic Heat Transfer 2030
14.5. Modeling Heat Exchangers 2030
14.5.1. Choosing a Heat Exchanger Model 2031
14.5.2. The Dual Cell Model 2033
14.5.2.1. Restrictions 2033
14.5.2.2. Using the Dual Cell Heat Exchanger Model 2034
14.5.3. The Macro Heat Exchanger Models 2043
14.5.3.1. Restrictions 2044
14.5.3.2. Using the Ungrouped Macro Heat Exchanger Model 2045
14.5.3.2.1. Selecting the Zone for the Heat Exchanger 2050
14.5.3.2.2. Specifying Heat Exchanger Performance Data 2050
14.5.3.2.3. Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions 2051
14.5.3.2.4. Defining the Macros 2051
14.5.3.2.4.1. Viewing the Macros 2052
14.5.3.2.5. Specifying the Auxiliary Fluid Properties and Conditions 2053
14.5.3.2.6. Setting the Pressure-Drop Parameters and Effectiveness 2054
14.5.3.2.6.1. Using the Default Core Porosity Model 2054
14.5.3.2.6.2. Defining a New Core Porosity Model 2055
14.5.3.2.6.3. Reading Heat Exchanger Parameters from an External File 2056
14.5.3.2.6.4. Viewing the Parameters for an Existing Core Model 2056
14.5.3.3. Using the Grouped Macro Heat Exchanger Model 2056
14.5.3.3.1. Selecting the Fluid Zones for the Heat Exchanger Group 2062
14.5.3.3.2. Selecting the Upstream Heat Exchanger Group 2062
14.5.3.3.3. Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions 2063
14.5.3.3.4. Specifying the Auxiliary Fluid Properties 2063
14.5.3.3.5. Specifying Supplementary Auxiliary Fluid Streams 2063
14.5.3.3.6. Initializing the Auxiliary Fluid Temperature 2063
14.5.4. Postprocessing for the Heat Exchanger Model 2064
14.5.4.1. Heat Exchanger Reporting 2064
14.5.4.1.1. Computed Heat Rejection 2064
14.5.4.1.2. **Inlet/Outlet Temperature** 2065
14.5.4.1.3. **Mass Flow Rate** 2067
14.5.4.1.4. **Specific Heat** 2068
14.5.4.2. Total Heat Rejection Rate 2068
14.5.5. Useful Reporting TUI Commands 2069
14.6. The Two-Temperature Model 2070
14.6.1. Using the Two-Temperature Model 2070
15. Modelling with Finite-Rate Chemistry 2083
15.1. Modeling Species Transport and Finite-Rate Chemistry 2083
15.1.1. Volumetric Reactions 2084
15.1.1.1. Overview of User Inputs for Modeling Species Transport and Reactions 2084
15.1.1.1.1. Mixture Materials 2085
15.1.1.2. Enabling Species Transport and Reactions and Choosing the Mixture Material 2087
15.1.1.3. Importing a Volumetric Kinetic Mechanism in CHEMKIN Format 2095

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.1.1.3.1. Using Ansys Encrypted Mechanisms 2096
15.1.1.3.2. Procedure for Importing Volumetric CHEMKIN Mechanisms 2096
15.1.1.3.3. CHEMKIN Mechanisms Included with Ansys Fluent 2100
15.1.1.4. Defining Properties for the Mixture and Its Constituent Species 2101
15.1.1.4.1. Defining the Species in the Mixture 2101
15.1.1.4.1.1. Overview of the Species Dialog Box 2103
15.1.1.4.1.2. Adding Species to the Mixture 2104
15.1.1.4.1.3. Removing Species from the Mixture 2105
15.1.1.4.1.4. Assigning the Last Species 2105
15.1.1.4.1.5. The Naming and Ordering of Species 2105
15.1.1.4.2. Defining Reactions 2106
15.1.1.4.2.1. Inputs for Reaction Definition 2106
15.1.1.4.2.2. Defining Species and Reactions for Fuel Mixtures 2114
15.1.1.4.3. Defining Zone-Based Reaction Mechanisms 2114
15.1.1.4.3.1. Inputs for Reaction Mechanism Definition 2114
15.1.1.4.4. Defining Physical Properties for the Mixture 2117
15.1.1.4.5. Defining Physical Properties for the Species in the Mixture 2117
15.1.1.5. Setting up Coal Simulations with the Coal Calculator Dialog Box 2118
15.1.1.6. Defining Cell Zone and Boundary Conditions for Species 2121
15.1.1.6.1. Diffusion at Inlets with the Pressure-Based Solver 2122
15.1.1.7. Defining Other Sources of Chemical Species 2122
15.1.1.8. Solution Procedures for Chemical Mixing and Finite-Rate Chemistry 2122
15.1.1.8.1. Stability and Convergence in Reacting Flows 2122
15.1.1.8.2. Two-Step Solution Procedure (Steady-state Only) 2123
15.1.1.8.3. Density Under-Relaxation 2123
15.1.1.8.4. Ignition in Steady-State Combustion Simulations 2123
15.1.1.8.5. Solution of Stiff Chemistry Systems 2124
15.1.1.8.6. Eddy-Dissipation Concept Model Solution Procedure 2125
15.1.1.9. Postprocessing for Species Calculations 2125
15.1.1.9.1. Averaged Species Concentrations 2126
15.1.2. Wall Surface Reactions and Chemical Vapor Deposition 2127
15.1.2.1. Overview of Surface Species and Wall Surface Reactions 2127
15.1.2.2. Importing a Surface Kinetic Mechanism in CHEMKIN Format 2127
15.1.2.2.1. Compatibility and Limitations for Gas Phase Reactions 2130
15.1.2.2.2. Compatibility and Limitations for Surface Reactions 2130
15.1.2.3. Manual Inputs for Wall Surface Reactions 2130
15.1.2.4. Including Mass Transfer To Surfaces in the Continuity Equation 2132
15.1.2.5. Wall Surface Mass Transfer Effects in the Energy Equation 2132
15.1.2.6. Modeling the Heat Release Due to Wall Surface Reactions 2133
15.1.2.7. Solution Procedures for Wall Surface Reactions 2133
15.1.2.8. Postprocessing for Surface Reactions 2133
15.1.3. Particle Surface Reactions 2133
15.1.3.1. User Inputs for Particle Surface Reactions 2134
15.1.3.2. Modeling Gaseous Solid Catalyzed Reactions 2135
15.1.3.3. Using the Multiple Surface Reactions Model for Discrete-Phase Particle Combustion 2135
15.1.4. Electrochemical Reactions 2136
15.1.4.1. Overview of Electrochemical Reactions 2136
15.1.4.2. User Inputs for Electrochemical Reactions 2136
15.1.4.3. Electrochemical Reaction Effects in the Energy Equation 2144
15.1.4.4. Electrochemical Reaction Effects in the Species Transport Equation 2144

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.1.4.5. Including Mass Transfer in Continuity 2144
15.1.4.6. Solution Procedures for Electrochemical Reactions 2144
15.1.5. Species Transport Without Reactions 2145
15.1.6. Reacting Channel Model 2146
15.1.6.1. Overview and Limitations of the Reacting Channel Model 2146
15.1.6.2. Enabling the Reacting Channel Model 2147
15.1.6.3. Boundary Conditions for Channel Walls 2152
15.1.6.4. Postprocessing for Reacting Channel Model Calculations 2152
15.1.7. Reactor Network Model 2154
15.1.7.1. Overview and Limitations of the Reactor Network Model 2155
15.1.7.2. Solving Reactor Networks 2155
15.1.7.3. Postprocessing Reactor Network Calculations 2159
15.2. Modeling a Composition PDF Transport Problem 2159
15.2.1. Limitation 2159
15.2.2. Steps for Using the Composition PDF Transport Model 2159
15.2.3. Enabling the Lagrangian Composition PDF Transport Model 2161
15.2.4. Enabling the Eulerian Composition PDF Transport Model 2164
15.2.4.1. Defining Species Boundary Conditions 2166
15.2.4.1.1. Equilibrating Inlet Streams 2167
15.2.5. Initializing the Solution 2167
15.2.6. Monitoring the Solution 2168
15.2.6.1. Running Unsteady Composition PDF Transport Simulations 2170
15.2.6.2. Running Compressible Lagrangian PDF Transport Simulations 2170
15.2.6.3. Running Lagrangian PDF Transport Simulations with Conjugate Heat Transfer 2170
15.2.7. Postprocessing for Lagrangian PDF Transport Calculations 2170
15.2.7.1. Reporting Options 2170
15.2.7.2. Particle Tracking Options 2171
15.2.8. Postprocessing for Eulerian PDF Transport Calculations 2172
15.2.8.1. Reporting Options 2172
15.3. Using Chemistry Acceleration 2173
15.3.1. Using ISAT 2174
15.3.1.1. ISAT Parameters 2175
15.3.1.2. Monitoring ISAT 2175
15.3.1.3. Using ISAT Efficiently 2176
15.3.1.4. Reading and Writing ISAT Tables 2177
15.3.2. Using Dynamic Mechanism Reduction 2178
15.3.2.1. Mechanism Reduction Parameters 2179
15.3.2.2. Monitoring and Postprocessing Dynamic Mechanism Reduction 2181
15.3.2.3. Using Dynamic Mechanism Reduction Effectively 2182
15.3.3. Using Chemistry Agglomeration 2182
15.3.4. Dimension Reduction 2183
15.3.5. Using Dynamic Cell Clustering 2184
15.3.6. Using Dynamic Adaptive Chemistry with Ansys Fluent CHEMKIN-CFD Solver 2185
16. Modelling of Turbulent Combustion With Reduced Order 2187
16.1. Modeling Non-Premixed Combustion 2187
16.1.1. Steps in Using the Non-Premixed Model 2187
16.1.1.1. Preliminaries 2188
16.1.1.2. Defining the Problem Type 2188
16.1.1.3. Overview of the Problem Setup Procedure 2188
16.1.2. Setting Up the Equilibrium Chemistry Model 2192
16.1.2.1. Choosing Adiabatic or Non-Adiabatic Options 2193

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.1.2.2. Specifying the Operating Pressure for the System 2194
16.1.2.3. Enabling a Secondary Inlet Stream 2194
16.1.2.4. Choosing to Define the Fuel Stream(s) Empirically 2195
16.1.2.5. Enabling the Rich Flammability Limit (RFL) Option 2196
16.1.3. Setting Up the Steady and Unsteady Diffusion Flamelet Models 2197
16.1.3.1. Choosing Adiabatic or Non-Adiabatic Options 2198
16.1.3.2. Specifying the Operating Pressure for the System 2198
16.1.3.3. Specifying a Chemical Mechanism File for Flamelet Generation 2198
16.1.3.4. Importing a Flamelet 2198
16.1.3.5. Using the Unsteady Diffusion Flamelet Model 2198
16.1.3.6. Using the Diesel Unsteady Laminar Flamelet Model 2200
16.1.3.6.1. Recommended Settings for Internal Combustion Engines 2202
16.1.3.7. Resetting Diesel Unsteady Flamelets 2203
16.1.4. Defining the Stream Compositions 2203
16.1.4.1. Setting Boundary Stream Species 2206
16.1.4.1.1. Including Condensed Species 2206
16.1.4.2. Modifying the Database 2206
16.1.4.3. Composition Inputs for Empirically-Defined Fuel Streams 2206
16.1.4.4. Modeling Liquid Fuel Combustion Using the Non-Premixed Model 2207
16.1.4.5. Modeling Coal Combustion Using the Non-Premixed Model 2207
16.1.4.5.1. Defining the Coal Composition: Single-Mixture-Fraction Models 2208
16.1.4.5.2. Defining the Coal Composition: Two-Mixture-Fraction Models 2209
16.1.4.5.3. Additional Coal Modeling Inputs in Ansys Fluent 2211
16.1.4.5.4. Postprocessing Non-Premixed Models of Coal Combustion 2212
16.1.4.5.5. The Coal Calculator 2212
16.1.5. Setting Up Control Parameters 2214
16.1.5.1. Forcing the Exclusion and Inclusion of Equilibrium Species 2215
16.1.5.2. Defining the Flamelet Controls 2216
16.1.5.3. Zeroing Species in the Initial Unsteady Flamelet 2217
16.1.6. Calculating the Flamelets 2217
16.1.6.1. Steady Diffusion Flamelet 2218
16.1.6.2. Unsteady Diffusion Flamelet 2220
16.1.6.3. Saving the Flamelet Data 2222
16.1.6.4. Postprocessing the Flamelet Data 2222
16.1.7. Calculating the Look-Up Tables 2225
16.1.7.1. Full Tabulation of the Two-Mixture-Fraction Model 2230
16.1.7.2. Stability Issues in Calculating Chemical Equilibrium Look-Up Tables 2230
16.1.7.3. Saving the Look-Up Tables 2230
16.1.7.4. Postprocessing the Look-Up Table Data 2231
16.1.8. Standard Files for Diffusion Flamelet Modeling 2235
16.1.8.1. Sample Standard Diffusion Flamelet File 2236
16.1.8.2. Missing Species 2237
16.1.9. Setting Up the Inert Model 2237
16.1.9.1. Setting Boundary Conditions for Inert Transport 2239
16.1.9.2. Initializing the Inert Stream 2240
16.1.9.2.1. Inert Fraction 2240
16.1.9.2.2. Inert Composition 2240
16.1.9.3. Resetting Inert EGR 2241
16.1.10. Defining Non-Premixed Boundary Conditions 2242
16.1.10.1. Input of Mixture Fraction Boundary Conditions 2242
16.1.10.2. Diffusion at Inlets 2243

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.1.10.3. Input of Thermal Boundary Conditions and Fuel Inlet Velocities 2243
16.1.11. Defining Non-Premixed Physical Properties 2243
16.1.12. Solution Strategies for Non-Premixed Modeling 2244
16.1.12.1. Single-Mixture-Fraction Approach 2244
16.1.12.2. Two-Mixture-Fraction Approach 2245
16.1.12.3. Starting a Non-Premixed Calculation From a Previous Case File 2245
16.1.12.3.1. Retrieving the PDF File During Case File Reads 2246
16.1.12.4. Solving the Flow Problem 2246
16.1.12.4.1. Under-Relaxation Factors for PDF Equations 2247
16.1.12.4.2. Density Under-Relaxation 2247
16.1.12.4.3. Tuning the PDF Parameters for Two-Mixture-Fraction Calculations 2247
16.1.13. Postprocessing the Non-Premixed Model Results 2248
16.1.13.1. Postprocessing for Inert Calculations 2250
16.2. Modeling Premixed Combustion 2251
16.2.1. Limitations of the Premixed Combustion Model 2251
16.2.2. Using the Premixed Combustion Model 2252
16.2.2.1. Enabling the Premixed Combustion Model 2253
16.2.2.2. Choosing an Adiabatic or Non-Adiabatic Model 2254
16.2.3. Setting Up the C-Equation and G-Equation Models 2254
16.2.3.1. Modifying the Constants for the Zimont Flame Speed Model 2255
16.2.3.2. Modifying the Constants for the Peters Flame Speed Model 2255
16.2.3.3. Additional Options for the G-Equation Model 2255
16.2.3.4. Defining Physical Properties for the Unburnt Mixture 2255
16.2.3.5. Setting Boundary Conditions for the Progress Variable 2256
16.2.3.6. Initializing the Progress Variable 2256
16.2.4. Postprocessing for Premixed Combustion Calculations 2257
16.2.4.1. Computing Species Concentrations 2257
16.3. Modeling Partially Premixed Combustion 2258
16.3.1. Limitations 2258
16.3.2. Using the Partially Premixed Combustion Model 2258
16.3.2.1. Setup and Solution Procedure 2259
16.3.2.2. Importing a Flamelet 2261
16.3.2.3. Flamelet Generated Manifold 2261
16.3.2.3.1. Premixed Flamelet Generated Manifolds 2263
16.3.2.3.1.1. Editing the Flamelet Grid Distribution 2265
16.3.2.3.2. Diffusion Flamelet Generated Manifolds 2267
16.3.2.3.3. Using the Heat Loss Modeling Capability for Nonadiabatic FGM 2268
16.3.2.4. Calculating the Look-Up Tables 2270
16.3.2.4.1. Postprocessing the Look-Up Tables with Flamelet Generated Manifolds 2274
16.3.2.5. Standard Files for Flamelet Generated Manifold Modeling 2277
16.3.2.5.1. Sample Standard FGM File 2277
16.3.2.6. Setting Premix Flame Propagation Parameters 2278
16.3.2.7. Modifying the Unburnt Mixture Property Polynomials 2280
16.3.2.8. Modeling Strained Laminar Flame Speed 2283
16.3.2.9. Modeling In Cylinder Combustion 2284
16.3.2.10. Postprocessing for FGM Scalar Transport Calculations 2285
17. Modeling Engine Ignition 2287
17.1. Using the Spark Model 2287
17.2. Using the Autoignition Models 2290
17.3. Using the Crevice Model 2293
17.3.1. Setup Procedure 2293

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17.3.2. Crevice Model Solution Details 2296
17.3.3. Postprocessing for the Crevice Model 2296
17.3.3.1. Using the Crevice Output File 2297
18. Modeling Pollutant Formation 2301
18.1. NOx Formation 2301
18.1.1. Using the NOx Model 2301
18.1.1.1. Decoupled Analysis: Overview 2301
18.1.1.2. Enabling the NOx Models 2302
18.1.1.3. Defining the Fuel Streams 2304
18.1.1.4. Specifying a User-Defined Function for the NOx Rate 2306
18.1.1.5. Setting Thermal NOx Parameters 2307
18.1.1.6. Setting Prompt NOx Parameters 2307
18.1.1.7. Setting Fuel NOx Parameters 2308
18.1.1.7.1. Setting Gaseous and Liquid Fuel NOx Parameters 2308
18.1.1.7.2. Setting Solid (Coal) Fuel NOx Parameters 2309
18.1.1.8. Setting N2O Pathway Parameters 2310
18.1.1.9. Setting Parameters for NOx Reburn 2311
18.1.1.10. Setting SNCR Parameters 2312
18.1.1.11. Setting Turbulence Parameters 2314
18.1.1.12. Defining Boundary Conditions for the NOx Model 2316
18.1.2. Solution Strategies 2316
18.1.3. Postprocessing 2317
18.2. Soot Formation 2318
18.2.1. Using the Soot Models 2318
18.2.1.1. Setting Up the One-Step Model 2319
18.2.1.2. Setting Up the Two-Step Model 2321
18.2.1.3. Setting Up the Moss-Brookes Model and the Hall Extension 2324
18.2.1.3.1. Specifying a User-Defined Function for the Soot Oxidation Rate 2328
18.2.1.3.2. Specifying a User-Defined Function for the Soot Precursor Concentration 2329
18.2.1.3.3. Species Definition for the Moss-Brookes Model with a User-Defined Precursor Correlation 2329
18.2.1.4. Setting Up the Method of Moments Soot Model 2332
18.2.1.5. Defining Boundary Conditions for the Soot Model 2337
18.2.1.6. Reporting Soot Quantities 2337
18.3. Using the Decoupled Detailed Chemistry Model 2338
19. Predicting Aerodynamically Generated Noise 2341
19.1. Overview 2341
19.1.1. Direct Method 2341
19.1.2. Integral Method by Ffowcs Williams and Hawkings 2342
19.1.3. Method Based on Wave Equation 2343
19.1.4. Broadband Noise Source Models 2343
19.2. Using the Ffowcs Williams and Hawkings Acoustics Model 2344
19.2.1. Enabling the FW-H Acoustics Model 2345
19.2.1.1. Setting Model Constants 2346
19.2.1.2. Computing Sound “on the Fly” 2347
19.2.1.3. Writing Source Data Files 2348
19.2.1.3.1. Exporting Source Data Without Enabling the FW-H Model: Using the Ansys Fluent ASD Format 2349
19.2.1.3.2. Exporting Source Data Without Enabling the FW-H Model: Using the CGNS Format 2349
19.2.2. Specifying Source Surfaces 2351

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.2.2.1. Saving Source Data 2353
19.2.3. Specifying Acoustic Receivers 2354
19.2.4. Specifying the Time Step Size 2355
19.2.5. Postprocessing the FW-H Acoustics Model Data 2357
19.2.5.1. Writing Acoustic Signals 2357
19.2.5.2. Reading Unsteady Acoustic Source Data 2357
19.2.5.2.1. Pruning the Signal Data Automatically 2359
19.2.5.3. Reporting the Static Pressure Time Derivative 2360
19.2.5.4. Using the FFT Capabilities for Sound Pressure Signals 2360
19.2.6. FFT of Acoustic Sources: Band Analysis and Export of Surface Pressure Spectra 2360
19.2.6.1. Using the FFT of Acoustic Sources 2361
19.3. Using the Acoustics Wave Equation Model 2369
19.3.1. Specifying Source Mask and Sponge Regions 2370
19.3.2. Solution Controls for the Acoustics Wave Equation 2372
19.3.3. Solution Initialization 2374
19.3.4. Postprocessing 2375
19.3.5. Using the Kirchhoff Integral Model 2376
19.4. Using the Broadband Noise Source Models 2379
19.4.1. Enabling the Broadband Noise Source Models 2380
19.4.1.1. Setting Model Constants 2380
19.4.2. Postprocessing the Broadband Noise Source Model Data 2381
20. Modeling Discrete Phase 2383
20.1. Introduction 2383
20.1.1. Concepts 2384
20.1.1.1. Uncoupled vs. Coupled DPM 2384
20.1.1.2. Steady vs. Unsteady Tracking 2385
20.1.1.3. Parcels 2385
20.1.2. Limitations 2386
20.1.2.1. Limitation on the Particle Volume Fraction 2386
20.1.2.2. Limitation on Modeling Continuous Suspensions of Particles 2387
20.1.2.3. Limitations on Modeling Particle Rotation 2387
20.1.2.4. Limitations on Using the Cloud Model 2387
20.1.2.5. Limitations on Using the Discrete Phase Model with Other Ansys Fluent Models .. 2387
20.1.2.6. Limitations on Using the Hybrid Parallel Method 2388
20.1.2.7. Limitations on Using the Lagrangian Wall Film Model 2389
20.2. Steps for Using the Discrete Phase Models 2391
20.2.1. Options for Interaction with the Continuous Phase 2392
20.2.2. Steady/Transient Treatment of Particles 2393
20.2.3. Tracking Settings for the Discrete Phase Model 2397
20.2.4. Drag Laws 2406
20.2.5. Physical Models for the Discrete Phase Model 2407
20.2.5.1. Including Radiation Heat Transfer Effects on the Particles 2408
20.2.5.2. Including Thermophoretic Force Effects on the Particles 2408
20.2.5.3. Including Saffman Lift Force Effects on the Particles 2408
20.2.5.4. Including the Virtual Mass Force and Pressure Gradient Effects on Particles 2408
20.2.5.5. Monitoring Erosion/Accretion of Particles at Walls 2408
20.2.5.6. Pressure Options for Vaporization Models 2409
20.2.5.7. Enabling Pressure Dependent Boiling 2409
20.2.5.8. Including the Effect of Droplet Temperature on Latent Heat 2410
20.2.5.9. Including the Effect of Particles on Turbulent Quantities 2410
20.2.5.10. Including Collision and Droplet Coalescence 2411

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.2.5.11. Including the DEM Collision Model 2411
20.2.5.12. Including Droplet Breakup 2411
20.2.5.13. Modeling Collision Using the DEM Model 2411
20.2.5.13.1. Limitations 2417
20.2.5.13.2. Numeric Recommendations 2417
20.2.6. User-Defined Functions 2417
20.2.7. Numerics of the Discrete Phase Model 2419
20.2.7.1. Numerics for Tracking of the Particles 2421
20.2.7.2. Including Coupled Heat-Mass Solution Effects on the Particles 2422
20.2.7.3. Tracking in a Reference Frame 2423
20.2.7.4. Node Based Averaging of Particle Data 2423
20.2.7.5. Linearized Source Terms 2424
20.2.7.6. Staggering of Particles in Space and Time 2425
20.2.7.7. Under-Relaxing Lagrangian Wall Film Height 2426
20.3. Setting Initial Conditions for the Discrete Phase 2426
20.3.1. Injection Types 2428
20.3.2. Particle Types 2430
20.3.3. Point Properties for Single Injections 2432
20.3.4. Point Properties for Group Injections 2433
20.3.5. Point Properties for Cone Injections 2434
20.3.6. Point Properties for Surface Injections 2437
20.3.6.1. Using the Rosin-Rammler Diameter Distribution Method for Surface Injections 2438
20.3.7. Point Properties for Volume Injections 2439
20.3.7.1. Using the Rosin-Rammler Diameter Distribution Method for Volume Injections 2439
20.3.8. Point Properties for Plain-Orifice Atomizer Injections 2440
20.3.9. Point Properties for Pressure-Swirl Atomizer Injections 2441
20.3.10. Point Properties for Air-Blast/Air-Assist Atomizer Injections 2442
20.3.11. Point Properties for Flat-Fan Atomizer Injections 2443
20.3.12. Point Properties for Effervescent Atomizer Injections 2445
20.3.13. Point Properties for File Injections 2445
20.3.13.1. Steady File Format 2446
20.3.13.2. Unsteady File Format 2446
20.3.13.3. User Input for File Injections 2447
20.3.14. Point Properties for Condensate Injections 2448
20.3.15. Using the Rosin-Rammler Diameter Distribution Method 2448
20.3.15.1. The Stochastic Rosin-Rammler Diameter Distribution Method 2451
20.3.16. Creating and Modifying Injections 2452
20.3.16.1. Creating Injections 2453
20.3.16.2. Modifying Injections 2453
20.3.16.3. Copying Injections 2453
20.3.16.4. Deleting Injections 2453
20.3.16.5. Listing Injections 2453
20.3.16.6. Reading and Writing Injections 2454
20.3.17. Defining Injection Properties 2454
20.3.18. Specifying Injection-Specific Physical Models 2462
20.3.18.1. Drag Laws 2462
20.3.18.2. Particle Rotation 2463
20.3.18.3. Rough Wall Model 2464
20.3.18.4. Brownian Motion Effects 2465
20.3.18.5. Breakup 2465
20.3.19. Specifying Turbulent Dispersion of Particles 2468

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.3.19.1. Stochastic Tracking 2468
20.3.19.2. Cloud Tracking 2470
20.3.20. Custom Particle Laws 2470
20.3.21. Defining Properties Common to More than One Injection 2472
20.3.21.1. Modifying Properties 2473
20.3.21.2. Modifying Properties Common to a Subset of Selected Injections 2474
20.3.22. Point Properties for Transient Injections 2474
20.4. Setting Boundary Conditions for the Discrete Phase 2475
20.4.1. Discrete Phase Boundary Condition Types 2476
20.4.1.1. The **reflect** Boundary Condition 2478
20.4.1.2. The **trap** Boundary Condition 2478
20.4.1.3. The **escape** Boundary Condition 2479
20.4.1.4. The **wall-jet** Boundary Condition 2479
20.4.1.5. The **wall-film** Boundary Condition 2479
20.4.1.6. The **interior** Boundary Condition 2482
20.4.1.7. The **user-defined** Boundary Condition 2482
20.4.2. Default Discrete Phase Boundary Conditions 2482
20.4.3. Coefficients of Restitution 2482
20.4.4. Friction Coefficient 2483
20.4.5. Particle-Wall Impingement Heat Transfer 2483
20.4.6. Film Condensation Model 2484
20.4.7. Wall Boundary Layer Model 2487
20.4.8. Setting Particle Erosion and Accretion Parameters 2487
20.5. Particle Erosion Coupled with Dynamic Meshes 2492
20.5.1. Preliminaries 2493
20.5.2. Limitations 2493
20.5.3. Procedure for the Erosion Coupled with Dynamic Mesh Setup and Solution 2494
20.5.4. Postprocessing for Erosion Dynamic Mesh Calculations 2499
20.6. Setting Material Properties for the Discrete Phase 2499
20.6.1. Summary of Property Inputs 2499
20.6.2. Setting Discrete-Phase Physical Properties 2503
20.6.2.1. The Concept of Discrete-Phase Materials 2503
20.6.2.1.1. Defining Additional Discrete-Phase Materials 2504
20.6.2.2. Description of the Properties 2505
20.7. Solution Strategies for the Discrete Phase 2515
20.7.1. Performing Trajectory Calculations 2515
20.7.1.1. Uncoupled Calculations 2516
20.7.1.2. Coupled Calculations 2516
20.7.1.2.1. Procedures for a Coupled Two-Phase Flow 2517
20.7.1.2.2. Stochastic Tracking in Coupled Calculations 2518
20.7.1.2.3. Under-Relaxation of the Interphase Exchange Terms 2518
20.7.2. Resetting the Interphase Exchange Terms 2520
20.7.3. Patching the Wall Film 2520
20.8. Postprocessing for the Discrete Phase 2521
20.8.1. Displaying of Trajectories 2522
20.8.1.1. Options for Particle Trajectory Plots 2525
20.8.1.2. Controlling the Particle Tracking Style 2526
20.8.1.3. Controlling the Vector Style of Particle Tracks 2528
20.8.1.4. Importing Particle Data 2532
20.8.1.5. Particle Filtering 2533
20.8.1.6. Graphical Display for Axisymmetric Geometries 2534

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.8.2. Reporting of Trajectory Fates 2534
20.8.2.1. Trajectory Fates 2534
20.8.2.2. Summary Reports 2535
20.8.2.2.1. Elapsed Time 2537
20.8.2.2.2. Mass Transfer Summary 2537
20.8.2.2.3. Energy Transfer Summary 2538
20.8.2.2.4. Heat Rate and Energy Reporting 2538
20.8.2.2.4.1. Change of Heat and Change of Energy Reporting 2540
20.8.2.2.5. Combusting Particles 2541
20.8.2.2.6. Combusting Particles with the Multiple Surface Reaction Model 2541
20.8.2.2.7. Multicomponent Particles 2541
20.8.3. Step-by-Step Reporting of Trajectories 2542
20.8.4. Reporting of Current Positions for Unsteady Tracking 2544
20.8.5. Reporting of Interphase Exchange Terms (Discrete Phase Sources) 2546
20.8.6. Reporting of Discrete Phase Variables 2546
20.8.7. Reporting of Unsteady DPM Statistics 2549
20.8.8. Sampling of Trajectories 2551
20.8.9. Histogram Reporting of Samples 2553
20.8.9.1. Analysis, Investigation, and Reporting of Samples 2554
20.8.9.2. Data Reduction of Samples 2556
20.8.10. Summary Reporting of Current Particles 2558
20.8.11. Postprocessing of Erosion/Accretion Rates 2560
20.8.12. Assessing the Risk for Solids Deposit Formation During Selective Catalytic Reduction Process 2560
20.9. Parallel Processing for the Discrete Phase Model 2563
21. Modeling Macroscopic Particles 2569
21.1. Overview and Limitations 2569
21.2. Loading the MPM add-on Module 2570
21.3. Setting up Macroscopic Particle Model Simulations 2570
21.4. Modeling Macroscopic Particles 2571
21.4.1. Specifying Particle Tracking Parameters 2573
21.4.2. Specifying the Drag Law 2574
21.4.3. Defining Parameters for Particle-Particle and Particle-Wall Collisions 2576
21.4.4. Specifying Deposition Parameters 2577
21.4.5. Specifying Injection Parameters 2579
21.4.5.1. Defining MPM Injection Properties 2580
21.4.5.2. Inputs for `point` Injections 2582
21.4.5.3. Inputs for `plane` Injections 2583
21.4.5.4. Inputs for `packing` Injections 2585
21.4.5.5. Inputs for `from-file` Injections 2586
21.4.6. Defining Field Forces 2587
21.4.7. Initializing the MPM 2587
22. Modeling Multiphase Flows 2591
22.1. Introduction 2591
22.2. Steps for Using a Multiphase Model 2591
22.2.1. Enabling the Multiphase Model 2594
22.2.1.1. Inputs for the VOF Model 2595
22.2.1.2. Inputs for the Mixture Multiphase Model 2596
22.2.1.3. Inputs for the Eulerian Multiphase Model 2597
22.2.2. Choosing Volume Fraction Formulation 2598
22.2.2.1. Interface Modeling Type 2599

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.2.2.2. Spatial Discretization Schemes for Volume Fraction 2601
22.2.2.3. Volume Fraction Limits 2602
22.2.2.4. Expert Options 2603
22.2.3. Solving a Homogeneous Multiphase Flow 2605
22.2.4. Modeling Buoyancy-Driven Multiphase Flow 2606
22.2.4.1. Setting the Operating Density for a Buoyancy-Driven Multiphase Flow 2606
22.2.4.2. The Boussinesq Approximation in a Multiphase Flow 2607
22.2.5. Modeling Compressible Flows 2607
22.2.6. Defining the Phases 2608
22.2.7. Including Body Forces 2609
22.2.8. Modeling Multiphase Species Transport 2610
22.2.9. Specifying Heterogeneous Reactions 2613
22.2.10. Including Mass Transfer Effects 2616
22.2.10.1. Alternative Modeling of Energy Sources 2619
22.2.10.2. Mass Transfer Mechanisms 2621
22.2.10.2.1. Constant-Rate Option 2621
22.2.10.2.2. User-Defined Option 2621
22.2.10.2.3. Population-Balance Mechanism 2622
22.2.10.2.4. Cavitation Mechanism 2622
22.2.10.2.5. Evaporation-Condensation Mechanism 2626
22.2.10.2.6. Species-Mass-Transfer Mechanism 2628
22.2.10.2.7. Boiling Mechanism 2631
22.2.11. Defining Multiphase Cell Zone and Boundary Conditions 2632
22.2.11.1. Steps for Setting Boundary Conditions 2632
22.2.11.2. Steps for Setting Cell Zone Conditions 2638
22.2.11.3. Boundary and Cell Zone Conditions for the Mixture and the Individual Phases ... 2640
22.2.11.3.1. VOF Model 2640
22.2.11.3.2. Mixture Model 2642
22.2.11.3.3. Eulerian Model 2643
22.2.11.4. Steps for Copying Cell Zone and Boundary Conditions 2649
22.2.12. Setting Initial Conditions 2649
22.2.12.1. Setting Initial Volume Fractions 2649
22.2.12.1.1. Options for Patching Volume Fraction 2650
22.2.12.2. Setting the Initial Turbulence Field 2652
22.3. Setting Up the VOF Model 2652
22.3.1. Solving Steady-State VOF Problems 2653
22.3.2. Guidelines for Using the Multiphase Pseudo Transient Solver 2653
22.3.3. Including Coupled Level Set with the VOF Model 2653
22.3.4. Mesh Adaption with the VOF Model 2654
22.3.5. Modeling Open Channel Flows 2654
22.3.5.1. Defining Inlet Groups 2656
22.3.5.2. Defining Outlet Groups 2656
22.3.5.3. Setting the Inlet Group 2656
22.3.5.4. Setting the Outlet Group 2657
22.3.5.5. Determining the Free Surface Level 2657
22.3.5.6. Determining the Bottom Level 2658
22.3.5.7. Specifying the Total Height 2659
22.3.5.8. Determining the Velocity Magnitude 2659
22.3.5.9. Determining the Secondary Phase for the Inlet 2659
22.3.5.10. Determining the Secondary Phase for the Outlet 2660
22.3.5.11. Choosing the Pressure Specification Method 2661

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.3.5.12. Choosing the Density Interpolation Method 2661
22.3.5.13. Open Channel Flow Compatibility with Velocity Inlet 2662
22.3.5.13.1. Velocity Inlet, Open Channel Flow, Steady-State 2662
22.3.5.13.2. Velocity Inlet, Open Channel Flow, Transient 2662
22.3.5.14. Limitations 2663
22.3.5.15. Recommendations for Setting Up an Open Channel Flow Problem 2663
22.3.6. Modeling Open Channel Wave Boundary Conditions 2664
22.3.6.1. Summary Report and Regime Check 2672
22.3.6.2. Transient Profile Support for Wave Inputs 2674
22.3.6.3. Alternative Stokes Wave Theory Variant 2674
22.3.7. Recommendations for Open Channel Initialization 2675
22.3.7.1. Reporting Parameters for Open Channel Wave BC Option 2678
22.3.8. Numerical Beach Treatment for Open Channels 2679
22.3.8.1. Solution Strategies 2682
22.3.9. Defining the Phases for the VOF Model 2683
22.3.9.1. Defining the Primary Phase 2683
22.3.9.2. Defining a Secondary Phase 2684
22.3.10. Defining Phase Interaction Terms 2685
22.3.10.1. Including Surface Tension and Adhesion Effects 2685
22.3.10.2. Discretizing Using the Phase Localized Compressive Scheme 2689
22.3.11. Setting Time-Dependent Parameters for the Explicit Volume Fraction Formulation 2691
22.3.12. Modeling Solidification/Melting 2693
22.3.13. Using the VOF-to-DPM Model Transition for Dispersion of Liquid in Gas 2693
22.3.13.1. Limitations on Using the VOF-to-DPM Model Transition 2695
22.3.13.2. Setting up the VOF-to-DPM Model Transition 2695
22.3.13.3. Best Practice Guidelines for Considering Diffuse Lumps 2701
22.3.13.4. Postprocessing for VOF-to-DPM Model Transition Calculations 2703
22.3.14. Using the DPM-to-VOF Model Transition 2703
22.3.14.1. Setting up the DPM-to-VOF Model Transition 2704
22.3.14.2. Limitations 2710
22.4. Setting Up the Mixture Model 2711
22.4.1. Defining the Phases for the Mixture Model 2711
22.4.1.1. Defining the Primary Phase 2711
22.4.1.2. Defining a Non-Granular Secondary Phase 2711
22.4.1.3. Defining a Granular Secondary Phase 2712
22.4.1.4. Defining the Interfacial Area Concentration via the Transport Equation 2715
22.4.1.5. Defining the Algebraic Interfacial Area Concentration 2718
22.4.1.6. Defining Drag Between Phases 2719
22.4.1.7. Defining the Slip Velocity 2720
22.4.1.8. Including Surface Tension and Wall Adhesion Effects 2721
22.4.2. Including Mixture Drift Force 2721
22.4.3. Including Cavitation Effects 2721
22.4.4. Including Semi-Mechanistic Boiling 2722
22.4.4.1. Overview and Limitations for the Semi-Mechanistic Boiling Model 2722
22.4.4.2. Using the Semi-Mechanistic Boiling Model 2722
22.4.4.3. Cell Zone Specific Boiling 2726
22.4.4.4. Expert Options for the Semi-Mechanistic Boiling Model 2726
22.4.4.5. Solution Strategies for the Semi-Mechanistic Boiling Model 2729
22.5. Setting Up the Eulerian Model 2730
22.5.1. Additional Guidelines for Eulerian Multiphase Simulations 2730
22.5.2. Defining the Phases for the Eulerian Model 2731

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.5.2.1. Defining the Primary Phase 2731
22.5.2.2. Defining a Non-Granular Secondary Phase 2731
22.5.2.3. Defining a Granular Secondary Phase 2732
22.5.2.4. Defining the Interfacial Area Concentration 2737
22.5.2.5. Defining the Interaction Between Phases 2739
22.5.2.5.1. Specifying the Drag Function 2740
22.5.2.5.1.1. Drag Modification 2744
22.5.2.5.2. Specifying the Restitution Coefficients (Granular Flow Only) 2744
22.5.2.5.3. Including the Lift Force 2745
22.5.2.5.4. Including the Lift Correlation 2746
22.5.2.5.5. Including the Wall Lubrication Force 2746
22.5.2.5.6. Including the Turbulent Dispersion Force 2750
22.5.2.5.7. Including Surface Tension and Wall Adhesion Effects 2753
22.5.2.5.8. Including the Virtual Mass Force 2753
22.5.3. Modeling Turbulence 2754
22.5.3.1. Including Turbulence Interaction Source Terms 2756
22.5.3.2. Customizing the k-ε Multiphase Turbulent Viscosity 2758
22.5.4. Including Heat Transfer Effects 2758
22.5.5. Using an Algebraic Interfacial Area Model 2760
22.5.6. Using the Algebraic Interfacial Area Density (AIAD) Model 2761
22.5.6.1. Limitations 2761
22.5.6.2. Procedure for Setting the AIAD Model 2761
22.5.6.3. Solution Strategies 2768
22.5.7. Using the Generalized Two Phase Flow (GENTOP) Model 2768
22.5.7.1. Limitations 2768
22.5.7.2. Steps for Using the GENTOP Model 2769
22.5.7.3. Solution Strategies 2771
22.5.8. Including the Dense Discrete Phase Model 2771
22.5.8.1. Defining a Granular Discrete Phase 2776
22.5.9. Including the Boiling Model 2777
22.5.10. Setting Up Polydisperse Boiling 2786
22.5.11. Including the Multi-Fluid VOF Model 2787
22.6. Population Balance Model 2790
22.6.1. Population Balance Model Setup 2790
22.6.1.1. Enabling the Population Balance Model 2790
22.6.1.1.1. Generated DQMOM Values 2803
22.6.1.2. Defining Population Balance Boundary Conditions 2807
22.6.1.2.1. Initializing Bin Fractions With a Log-Normal Distribution 2809
22.6.1.3. Specifying Population Balance Solution Controls 2810
22.6.1.4. Coupling With Fluid Dynamics 2810
22.6.1.5. Specifying Interphase Mass Transfer Due to Nucleation and Growth 2811
22.6.1.6. Size Calculator 2815
22.6.2. Postprocessing for the Population Balance Model 2817
22.6.2.1. Population Balance Solution Variables 2817
22.6.2.2. Reporting Derived Population Balance Variables 2818
22.6.2.2.1. Computing Moments 2818
22.6.2.2.2. Displaying a Number Density Function 2819
22.6.3. UDFs for Population Balance Modeling 2821
22.6.3.1. Population Balance Variables 2821
22.6.3.2. Population Balance `DEFINE` Macros 2822
22.6.3.2.1. `DEFINE_PB_BREAK_UP_RATE_FREQ` 2822

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.6.3.2.1.1. Usage 2822
22.6.3.2.1.2. Example 2823
22.6.3.2.2. DEFINE_PB_BREAK_UP_RATE_PDF 2823
22.6.3.2.2.1. Usage 2823
22.6.3.2.2.2. Example 2824
22.6.3.2.3. DEFINE_PB_COALESCENCE_RATE 2824
22.6.3.2.3.1. Usage 2824
22.6.3.2.3.2. Example 2825
22.6.3.2.4. DEFINE_PB_NUCLEATION_RATE 2825
22.6.3.2.4.1. Usage 2825
22.6.3.2.4.2. Example 2826
22.6.3.2.5. DEFINE_PB_GROWTH_RATE 2827
22.6.3.2.5.1. Usage 2827
22.6.3.2.5.2. Example 2827
22.6.3.3. Hooking a Population Balance UDF to Ansys Fluent 2828
22.6.4. DEFINE_HET_RXN_RATE Macro 2828
22.6.4.1. Description 2829
22.6.4.2. Usage 2829
22.6.4.3. Example 2830
22.6.4.4. Hooking a Heterogeneous Reaction Rate UDF to Ansys Fluent 2830
22.7. Setting Up the Wet Steam Model 2831
22.7.1. Using User-Defined Thermodynamic Wet Steam Properties 2832
22.7.2. Writing the User-Defined Wet Steam Property Functions (UDWSPF) 2833
22.7.3. Compiling Your UDWSPF and Building a Shared Library File 2835
22.7.4. Loading the UDWSPF Shared Library File 2836
22.7.5. UDWSPF Example 2837
22.8. Solution Strategies for Multiphase Modeling 2841
22.8.1. General Solution Strategies 2841
22.8.1.1. Coupled Solution for Eulerian Multiphase Flows 2842
22.8.1.2. Coupled Solution for VOF and Mixture Multiphase Flows 2843
22.8.1.3. Selecting the Pressure-Velocity Coupling Method 2844
22.8.1.3.1. Limitations and Recommendations of the Coupled with Volume Fraction Options for the VOF and Mixture Models 2846
22.8.1.3.2. Solving N-Phase Volume Fraction Equations 2847
22.8.1.4. Controlling the Volume Fraction Coupled Solution 2847
22.8.1.5. Default and Stability Controls 2850
22.8.1.5.1. Default Controls 2850
22.8.1.5.2. VOF Solution Stability Controls 2851
22.8.1.5.3. Text User Interface for VOF Stability Controls 2853
22.8.1.6. Steady-State Solution Strategies 2855
22.8.2. Model-Specific Solution Strategies 2855
22.8.2.1. VOF Model 2856
22.8.2.1.1. Setting the Reference Pressure Location 2856
22.8.2.1.2. Pressure Interpolation Scheme 2856
22.8.2.1.3. Discretization Scheme Selection 2857
22.8.2.1.4. High-Order Rhie-Chow Face Flux Interpolation 2857
22.8.2.1.5. Treatment of Unsteady Terms in Rhie-Chow Face Flux Interpolation 2858
22.8.2.1.6. Pressure-Velocity Coupling and Under-Relaxation for the Time-dependent Formulations 2858
22.8.2.1.7. Under-Relaxation for the Steady-State Formulation 2859
22.8.2.2. Mixture Model 2859

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.8.2.2.1. Setting the Under-Relaxation Factor for the Slip Velocity 2859
22.8.2.2.2. Calculating an Initial Solution 2859
22.8.2.3. Eulerian Model 2859
22.8.2.3.1. Calculating an Initial Solution 2859
22.8.2.3.2. Temporarily Ignoring Lift and Virtual Mass Forces 2860
22.8.2.3.3. Using W-Cycle Multigrid 2860
22.8.2.3.4. Including the Anisotropic Drag Law 2860
22.8.2.3.5. Controlling NITA Solution Options via the Text Interface 2861
22.8.2.4. Wet Steam Model 2862
22.8.2.4.1. Boundary Conditions, Initialization, and Patching 2862
22.8.2.4.2. Solution Limits for the Wet Steam Model 2862
22.8.2.4.3. Solution Strategies for the Wet Steam Model 2862
22.9. Multiphase Case Check 2863
22.10. Postprocessing for Multiphase Modeling 2864
22.10.1. Model-Specific Variables 2864
22.10.1.1. VOF Model 2865
22.10.1.2. Mixture Model 2865
22.10.1.3. Eulerian Model 2865
22.10.1.4. Multiphase Species Transport 2866
22.10.1.5. Wet Steam Model 2868
22.10.1.6. Dense Discrete Phase Model 2868
22.10.2. Displaying Velocity Vectors 2869
22.10.3. Reporting Fluxes 2869
22.10.4. Reporting Forces on Walls 2870
22.10.5. Reporting Flow Rates 2870
23. Modeling Solidification and Melting 2873
23.1. Setup Procedure 2873
23.2. Procedures for Modeling Continuous Casting 2876
23.3. Modeling Thermal and Solutal Buoyancy 2877
23.4. Solution Procedure 2878
23.5. Postprocessing 2879
24. Modeling Fluid-Structure Interaction (FSI) Within Fluent 2881
24.1. Overview and Limitations 2881
24.2. Setting Up an Intrinsic Fluid-Structure Interaction (FSI) Simulation 2883
24.2.1. Using Intrinsic FSI With Non-Conformal Interfaces 2890
25. Modeling Eulerian Wall Films 2893
25.1. Limitations 2893
25.2. Overview of Using the Eulerian Wall Film Model 2894
25.3. Setting Eulerian Wall Film Model Options 2895
25.4. Setting Eulerian Wall Film Solution Controls 2902
25.5. Setting Eulerian Wall Film Boundary, Initial, and Source Term Conditions 2905
25.5.1. Specifying the Boundary Type 2905
25.5.2. Setting the Source Terms 2907
25.5.3. Setting the Phase Change 2908
25.5.4. Setting the Surface Contact 2909
25.5.5. Setting the DPM interaction 2910
25.5.6. Setting the VOF interaction 2911
25.6. Coupling of Eulerian Wall Film with the VOF Multiphase Model 2913
25.7. Postprocessing the Eulerian Wall Film 2913
26. Modeling Electric Potential Field and Lithium-ion Battery 2917
26.1. Using the Electric Potential Model 2917

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.1.1. Limitation of the Electric Potential Model 2917
26.1.2. Setting Up the Electric Potential Model 2918
26.2. Using the Lithium-ion Battery Model 2919
26.2.1. Limitations of the Detailed Lithium-ion Battery Model 2920
26.2.2. Setting Up the Lithium-ion Battery Model 2920
26.3. Postprocessing Electric Potential Field and Li-ion Battery Quantities 2926
27. Modeling Batteries 2929
27.1. Introduction 2929
27.1.1. Overview 2929
27.1.2. General Procedure 2929
27.2. Using the MSMD-Based Battery Models 2930
27.2.1. Limitations 2930
27.2.2. Geometry Definition 2931
27.2.3. Setting up the Battery Model 2931
27.2.3.1. Specifying Battery Model Options 2933
27.2.3.2. Specifying Conductive Zones 2943
27.2.3.3. Specifying Electric Contacts 2944
27.2.3.4. Specifying Battery Model Parameters 2947
27.2.3.4.1. Inputs for the CHT Coupling Method 2948
27.2.3.4.2. Inputs for the FMU-CHT Coupling Method 2949
27.2.3.4.3. Inputs for the NTGK Empirical Model 2950
27.2.3.4.4. Inputs for the Equivalent Circuit Model 2956
27.2.3.4.4.1. The HPPC Library 2962
27.2.3.4.5. Inputs for the Newman's P2D Model 2964
27.2.3.4.6. Input for the User-Defined E-Model 2969
27.2.3.5. Hooking User-Defined Functions 2969
27.2.3.6. Specifying Advanced Options 2970
27.2.3.7. Specifying External and Internal Short-Circuit Resistances 2977
27.2.4. Using Parameter Estimation Tools 2977
27.2.4.1. Using Parameter Estimation Tools in the GUI 2978
27.2.4.2. Using Parameter Estimation Tools in the TUI 2978
27.2.4.2.1. Using the Parameter Estimation Tool for the NTGK Model in the TUI 2978
27.2.4.2.2. Using the Parameter Estimation Tool for the ECM Model in the TUI 2980
27.2.4.2.3. Using the Parameter Estimation Tool for the Thermal Abuse Model in the TUI 2982
27.2.5. Initializing the Battery Model 2983
27.2.6. Modifying Material Properties 2983
27.2.7. Solution Controls for the MSMD Battery Model 2983
27.2.8. Postprocessing the MSMD Battery Model 2983
28. Modeling Fuel Cells 2987
28.1. Using the PEMFC Model 2987
28.1.1. Overview and Limitations 2988
28.1.2. Geometry Definition for the PEMFC Model 2988
28.1.3. Installing the PEMFC Model 2988
28.1.4. Loading the PEMFC Module 2989
28.1.5. Workflow for Using the PEMFC Module 2989
28.1.6. Setting Up the PEMFC Module 2990
28.1.6.1. Specifying Model Options (**Model** Tab) 2992
28.1.6.2. Specifying Model Parameters (**Parameters** Tab) 2996
28.1.6.3. Specifying Anode Properties (**Anode** Tab) 2999
28.1.6.3.1. Specifying Current Collector Properties for the Anode 3000

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.1.6.3.2. Specifying Flow Channel Properties for the Anode 3001
28.1.6.3.3. Specifying Porous Electrode Properties for the Anode 3002
28.1.6.3.4. Specifying Catalyst Layer Properties for the Anode 3004
28.1.6.3.5. Specifying Micro Porous Layer (Optional) Properties for the Anode 3006
28.1.6.3.6. Specifying Cell Zone Conditions for the Anode 3007
28.1.6.4. Specifying Electrolyte/Membrane Properties (**Electrolyte** Tab) 3007
28.1.6.4.1. Specifying Cell Zone Conditions for the Membrane 3009
28.1.6.5. Specifying Cathode Properties (**Cathode** Tab) 3009
28.1.6.5.1. Specifying Current Collector Properties for the Cathode 3009
28.1.6.5.2. Specifying Flow Channel Properties for the Cathode 3009
28.1.6.5.3. Specifying Porous Electrode Properties for the Cathode 3009
28.1.6.5.4. Specifying Catalyst Layer Properties for the Cathode 3010
28.1.6.5.5. Specifying Micro Porous Layer (Optional) Properties for the Cathode 3012
28.1.6.5.6. Specifying Cell Zone Conditions for the Cathode 3012
28.1.6.6. Setting the External Electrical Tabs (**Electrical Tabs** Tab) 3012
28.1.6.7. Setting Advanced Properties (**Advanced** Tab) 3013
28.1.6.7.1. Setting Contact Resistivities for the PEMFC Model 3014
28.1.6.7.2. Setting Coolant Channel Properties for the PEMFC Model (Optional) 3015
28.1.6.7.3. Managing Stacks for the PEMFC Model 3016
28.1.6.8. Reporting on the Solution (**Reports** Tab) 3017
28.1.7. PEMFC Model Boundary Conditions 3020
28.1.8. Solution Guidelines for the PEMFC Model 3021
28.1.9. Postprocessing the PEMFC Model 3021
28.1.10. User-Accessible Functions 3023
28.1.10.1. Compiling the Customized PEMFC Source Code 3027
28.1.10.1.1. Compiling the Customized Source Code Under Linux 3027
28.1.10.1.2. Compiling the Customized Source Code under Windows 3028
28.2. Using the Fuel Cell and Electrolysis Model 3028
28.2.1. Overview and Limitations 3029
28.2.2. Geometry Definition for the Fuel Cell and Electrolysis Model 3029
28.2.3. Installing the Fuel Cell and Electrolysis Model 3030
28.2.4. Loading the Fuel Cell and Electrolysis Module 3030
28.2.5. Workflow for Using the Fuel Cell and Electrolysis Module 3030
28.2.6. Setting Up the Fuel Cell and Electrolysis Module 3031
28.2.6.1. Specifying Model Options (**Model** Tab) 3033
28.2.6.2. Specifying Model Parameters (**Parameters** Tab) 3035
28.2.6.3. Specifying Anode Properties (**Anode** Tab) 3037
28.2.6.3.1. Specifying Current Collector Properties for the Anode 3037
28.2.6.3.2. Specifying Flow Channel Properties for the Anode 3038
28.2.6.3.3. Specifying Porous Electrode Properties for the Anode 3039
28.2.6.3.4. Specifying Catalyst Layer Properties for the Anode 3040
28.2.6.3.5. Specifying Cell Zone Conditions for the Anode 3040
28.2.6.4. Specifying Electrolyte/Membrane Properties (**Electrolyte** Tab) 3041
28.2.6.4.1. Specifying Cell Zone Conditions for the Membrane 3041
28.2.6.5. Specifying Cathode Properties (**Cathode** Tab) 3042
28.2.6.5.1. Specifying Current Collector Properties for the Cathode 3042
28.2.6.5.2. Specifying Flow Channel Properties for the Cathode 3043
28.2.6.5.3. Specifying Porous Electrode Properties for the Cathode 3044
28.2.6.5.4. Specifying Catalyst Layer Properties for the Cathode 3045
28.2.6.5.5. Specifying Cell Zone Conditions for the Cathode 3045
28.2.6.6. Setting Advanced Properties (**Advanced** Tab) 3046

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.2.6.6.1. Setting Contact Resistivities for the Fuel Cell and Electrolysis Model 3046
28.2.6.6.2. Setting Coolant Channel Properties for the Fuel Cell and Electrolysis Model 3047
28.2.6.6.3. Managing Stacks for the Fuel Cell and Electrolysis Model 3048
28.2.6.7. Reporting on the Solution (**Reports** Tab) 3050
28.2.7. Modeling Current Collectors 3051
28.2.8. Fuel Cell and Electrolysis Model Boundary Conditions 3052
28.2.9. Solution Guidelines for the Fuel Cell and Electrolysis Model 3053
28.2.10. Postprocessing the Fuel Cell and Electrolysis Model 3053
28.2.11. User-Accessible Functions 3055
28.2.11.1. Compiling the Customized Fuel Cell and Electrolysis Source Code 3058
28.2.11.1.1. Compiling the Customized Source Code Under Linux 3059
28.2.11.1.2. Compiling the Customized Source Code Under Windows 3059
28.3. Using the Solid Oxide Fuel Cell With Unresolved Electrolyte Model 3060
28.3.1. Limitation on Modeling Solid Oxide Fuel Cells 3060
28.3.2. Installing the Solid Oxide Fuel Cell With Unresolved Electrolyte Model 3060
28.3.3. Loading the Solid Oxide Fuel Cell With Unresolved Electrolyte Module 3060
28.3.4. Solid Oxide Fuel Cell With Unresolved Electrolyte Module Set Up Procedure 3062
28.3.5. Setting the SOFC Model 3065
28.3.5.1. Setting the Parameters for the SOFC With Unresolved Electrolyte Model 3066
28.3.5.2. Setting Up the Electrochemistry Parameters 3068
28.3.5.3. Setting Up the Electrode-Electrolyte Interfaces 3070
28.3.5.4. Setting Up the Electric Field Model Parameters 3072
28.3.6. User-Accessible Functions for the Solid Oxide Fuel Cell With Unresolved Electrolyte Model 3074
28.3.6.1. Compiling the Customized Solid Oxide Fuel Cell With Unresolved Electrolyte Source Code 3075
28.3.6.1.1. Compiling the Customized Source Code Under Linux 3075
28.3.6.1.2. Compiling the Customized Source Code Under Windows 3076
29. Modeling Magnetohydrodynamics 3079
29.1. Introduction 3079
29.2. Implementation 3079
29.2.1. Solving Magnetic Induction and Electric Potential Equations 3080
29.2.2. Calculation of MHD Variables 3080
29.2.3. MHD Interaction with Fluid Flows 3081
29.2.4. MHD Interaction with Discrete Phase Model 3081
29.2.5. General User-Defined Functions 3081
29.3. Using the Ansys Fluent MHD Module 3081
29.3.1. MHD Module Installation 3081
29.3.2. Loading the MHD Module 3082
29.3.3. MHD Model Setup 3083
29.3.3.1. Enabling the MHD Model 3083
29.3.3.2. Selecting an MHD Method 3084
29.3.3.3. Applying an External Magnetic Field 3085
29.3.3.4. Setting Up Boundary Conditions 3089
29.3.3.5. Solution Controls 3093
29.3.4. MHD Solution and Postprocessing 3094
29.3.4.1. MHD Model Initialization 3095
29.3.4.2. Iteration 3095
29.3.4.3. Postprocessing 3095
29.3.5. Limitations 3096

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.4. Guidelines For Using the Ansys Fluent MHD Model 3097
29.4.1. Installing the MHD Module 3097
29.4.2. An Overview of Using the MHD Module 3097
29.5. Definitions of the Magnetic Field 3100
29.6. External Magnetic Field Data Format 3100
30. Modeling Continuous Fibers 3103
30.1. Installing the Continuous Fiber Module 3104
30.2. Loading the Continuous Fiber Module 3104
30.3. Getting Started With the Continuous Fiber Module 3105
30.3.1. User-Defined Memory and the Adjust Function Setup 3106
30.3.2. Source Term UDF Setup 3106
30.4. Fiber Models and Options 3107
30.4.1. Choosing a Fiber Model 3108
30.4.2. Including Interaction With Surrounding Flow 3109
30.4.3. Including Lateral Drag on Surrounding Flow 3109
30.4.4. Including Fiber Radiation Interaction 3109
30.4.5. Viscous Heating of Fibers 3109
30.4.6. Drag, Heat and Mass Transfer Correlations 3109
30.5. Fiber Material Properties 3110
30.5.1. The Concept of Fiber Materials 3110
30.5.2. Description of Fiber Properties 3110
30.6. Defining Fibers 3113
30.6.1. Overview 3113
30.6.2. Fiber Injection Types 3114
30.6.3. Working with Fiber Injections 3115
30.6.3.1. Creating Fiber Injections 3115
30.6.3.2. Modifying Fiber Injections 3115
30.6.3.3. Copying Fiber Injections 3115
30.6.3.4. Deleting Fiber Injections 3116
30.6.3.5. Initializing Fiber Injections 3116
30.6.3.6. Computing Fiber Injections 3116
30.6.3.7. Print Fiber Injections 3116
30.6.3.8. Read Data of Fiber Injections 3117
30.6.3.9. Write Data of Fiber Injections 3117
30.6.3.10. Write Binary Data of Fiber Injections 3117
30.6.3.11. List Fiber Injections 3117
30.6.4. Defining Fiber Injection Properties 3118
30.6.5. Point Properties Specific to Single Fiber Injections 3122
30.6.6. Point Properties Specific to Line Fiber Injections 3122
30.6.7. Point Properties Specific to Matrix Fiber Injections 3122
30.6.8. Define Fiber Grids 3123
30.6.8.1. Equidistant Fiber Grids 3123
30.6.8.2. One-Sided Fiber Grids 3124
30.6.8.3. Two-Sided Fiber Grids 3124
30.6.8.4. Three-Sided Fiber Grids 3124
30.7. User-Defined Functions (UDFs) for the Continuous Fiber Model 3125
30.7.1. UDF Setup 3126
30.7.1.1. Linux Systems 3126
30.7.1.2. Windows Systems 3126
30.7.2. Customizing the fiber_fluent_interface.c File for Your Fiber Model Application 3126
30.7.2.1. Example: Heat Transfer Coefficient UDF 3127

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

30.7.2.2. Example: Fiber Specific Heat Capacity UDF 3128
30.7.3. Compile Fiber Model UDFs 3129
30.7.3.1. Linux Systems 3129
30.7.3.2. NT/Windows Systems 3130
30.7.4. Hook UDFs to the Continuous Fiber Model 3131
30.8. Fiber Model Solution Controls 3132
30.9. Postprocessing for the Continuous Fibers 3134
30.9.1. Display of Fiber Locations and Grid Points 3135
30.9.2. Exchange Terms of Fibers 3137
30.9.3. Analyzing Fiber Variables 3138
30.9.3.1. XY Plots 3138
30.9.3.2. Fiber Display 3139
30.9.4. Running the Fiber Module in Parallel 3140
31. Creating Reduced Order Models (ROMs) 3143
31.1. Defining a ROM 3143
31.2. Reduced Order Model (ROM) Evaluation in Fluent 3145
31.3. Exporting Reduced Order Model (ROM) Results from Fluent 3150
31.4. ROM Limitations 3151
32. Using the Solver 3153
32.1. Overview of Using the Solver 3153
32.1.1. Choosing the Solver 3155
32.2. Choosing the Spatial Discretization Scheme 3157
32.2.1. First-Order Accuracy vs. Second-Order Accuracy 3158
32.2.1.1. First- to Higher-Order Blending 3158
32.2.2. Other Discretization Schemes 3159
32.2.3. Choosing the Pressure Interpolation Scheme 3159
32.2.4. Choosing the Density Interpolation Scheme 3160
32.2.5. High Order Term Relaxation (HOTR) 3160
32.2.5.1. Limitations 3162
32.2.6. User Inputs 3163
32.3. Pressure-Based Solver Settings 3165
32.3.1. Choosing the Pressure-Velocity Coupling Method 3166
32.3.1.1. SIMPLE vs. SIMPLEC 3166
32.3.1.2. PISO 3167
32.3.1.3. Fractional Step Method 3167
32.3.1.4. Coupled 3168
32.3.1.5. User Inputs 3168
32.3.2. Mass Flux Types 3169
32.3.3. Setting Under-Relaxation Factors 3169
32.3.3.1. User Inputs 3170
32.3.4. Setting Solution Controls for the Non-Iterative Solver 3172
32.3.4.1. User Inputs 3173
32.3.4.2. Hybrid NITA for the VOF Model 3175
32.3.4.3. NITA Expert Options 3178
32.3.4.4. Compatibility of the NITA Scheme with Other Ansys Fluent Models 3178
32.3.5. Equation Order 3180
32.3.6. Using the Correction Form of Momentum Discretization 3180
32.4. Density-Based Solver Settings 3181
32.4.1. Changing the Courant Number 3182
32.4.1.1. Courant Numbers for the Density-Based Explicit Formulation 3182
32.4.1.2. Courant Numbers for the Density-Based Implicit Formulation 3182

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.4.1.3. User Inputs 3183
32.4.2. Convective Flux Types 3184
32.4.3. Convergence Acceleration for Stretched Meshes (CASM) 3184
32.4.4. Enabling High-Speed Numerics 3187
32.4.5. Preventing Divergence Using Local Under-Relaxation 3188
32.4.6. Specifying the Explicit Relaxation 3188
32.4.7. Turning On FAS Multigrid 3189
32.4.7.1. Setting Coarse Grid Levels 3189
32.4.7.2. Using Residual Smoothing to Increase the Courant Number 3190
32.5. Setting Algebraic Multigrid Parameters 3190
32.5.1. Specifying the Multigrid Cycle Type 3193
32.5.2. Setting the Termination and Residual Reduction Parameters 3193
32.5.3. Setting the Stabilization Method 3193
32.5.4. Additional Algebraic Multigrid Parameters 3194
32.5.4.1. Fixed Cycle Parameters 3195
32.5.4.2. Coarsening Parameters 3195
32.5.4.3. Smoother Types 3196
32.5.4.4. Flexible Cycle Parameters 3197
32.5.4.5. Setting the Verbosity 3197
32.5.4.6. Returning to the Default Multigrid Parameters 3198
32.5.5. Setting FAS Multigrid Parameters 3198
32.5.5.1. Combating Convergence Trouble 3198
32.5.5.2. "Industrial-Strength" FAS Multigrid 3198
32.6. Setting Solution Limits 3201
32.6.1. Limiting the Values of Solution Variables 3203
32.6.2. Adjusting the Positivity Rate Limit 3203
32.6.3. Resetting Solution Limits 3204
32.7. Setting Multi-Stage Time-Stepping Parameters 3204
32.7.1. Changing the Multi-Stage Scheme 3204
32.7.1.1. Changing the Coefficients and Number of Stages 3205
32.7.1.2. Controlling Updates to Dissipation and Viscous Stresses 3205
32.7.1.3. Resetting the Multi-Stage Parameters 3206
32.8. Selecting Gradient Limiters 3206
32.9. Initializing the Solution 3207
32.9.1. Initializing the Entire Flow Field Using Standard Initialization 3208
32.9.1.1. Saving and Resetting Initial Values 3210
32.9.2. Patching Values in Selected Cells 3210
32.9.2.1. Using Registers 3212
32.9.2.2. Using Field Functions 3212
32.9.2.3. Using Patching Later in the Solution Process 3212
32.10. Full Multigrid (FMG) Initialization 3213
32.10.1. Steps in Using FMG Initialization 3213
32.10.2. Convergence Strategies for FMG Initialization 3214
32.11. Hybrid Initialization 3214
32.11.1. Steps in Using Hybrid Initialization 3215
32.11.2. Solution Strategies for Hybrid Initialization 3217
32.12. Performing Steady-State Calculations 3218
32.12.1. Updating UDF Profiles 3220
32.12.2. Resetting Data 3220
32.12.3. Data Sampling for Steady Statistics 3220
32.13. Performing Pseudo Transient Calculations 3222

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.13.1. Setting Pseudo Transient Explicit Relaxation Factors 3224
32.13.1.1. User Inputs 3225
32.13.2. Setting Solution Controls for the Pseudo Transient Method 3226
32.13.3. Solving Pseudo-Transient Flow 3227
32.14. Performing Time-Dependent Calculations 3231
32.14.1. User Inputs for Time-Dependent Problems 3232
32.14.1.1. Additional Inputs 3247
32.14.2. CFL-Based Time Stepping 3247
32.14.2.1. The CFL-Based Time Stepping Algorithm 3247
32.14.2.2. Specifying Parameters for CFL-Based Time Stepping 3248
32.14.3. Error-Based Time Stepping 3249
32.14.3.1. The Error-Based Time Stepping Algorithm 3249
32.14.3.2. Specifying Parameters for Error-Based Time Stepping 3250
32.14.4. Multiphase-Specific Time Stepping 3251
32.14.4.1. The Multiphase-Specific Time Stepping Algorithm 3251
32.14.4.2. Specifying Parameters for Multiphase-Specific Time Stepping 3253
32.14.5. Postprocessing for Time-Dependent Problems 3255
32.15. Monitoring Solution Convergence 3256
32.15.1. Monitoring Residuals 3256
32.15.1.1. Definition of Residuals for the Pressure-Based Solver 3256
32.15.1.2. Definition of Residuals for the Density-Based Solver 3258
32.15.1.3. Overview of Using the Residual Monitors Dialog Box 3259
32.15.1.4. Printing and Plotting Residuals 3261
32.15.1.5. Storing Residual History Points 3262
32.15.1.6. Controlling Normalization and Scaling 3262
32.15.1.7. Choosing a Convergence Criterion 3263
32.15.1.8. Modifying Convergence Criteria 3265
32.15.1.9. Disabling Monitoring 3266
32.15.1.10. Plot Parameters 3266
32.15.1.11. Postprocessing Residual Values 3266
32.15.2. Monitoring Statistics 3267
32.15.3. Monitoring Solution Quantities 3268
32.16. Convergence Conditions 3268
32.16.1. Setting Up the Convergence Conditions Dialog Box 3270
32.17. Executing Commands During the Calculation 3272
32.17.1. Defining Macros 3274
32.17.2. Saving Files During the Calculation 3276
32.18. Automatic Initialization of the Solution and Case Modification 3276
32.18.1. Altering the Solution Initialization and Case Modification after Calculating 3281
32.19. Animating the Solution 3282
32.19.1. Creating an Animation Definition 3282
32.19.1.1. Guidelines for Creating an Animation Definition 3284
32.19.2. Playing an Animation Sequence 3285
32.19.2.1. Modifying the View 3286
32.19.2.2. Modifying the Playback Speed 3286
32.19.2.3. Playing Back an Excerpt 3286
32.19.2.4. “Fast-Forwarding” the Animation 3286
32.19.2.5. Continuous Animation 3287
32.19.2.6. Stopping the Animation 3287
32.19.2.7. Advancing the Animation Frame by Frame 3287
32.19.2.8. Deleting an Animation Sequence 3287

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.19.3. Saving an Animation Sequence 3287
32.19.3.1. Solution Animation File 3288
32.19.3.2. Picture File 3288
32.19.3.3. Video File 3289
32.19.4. Reading an Animation Sequence 3291
32.20. Checking Your Case Setup 3292
32.20.1. Automatic Implementation 3293
32.20.2. Manual Implementation 3293
32.20.2.1. Checking the Mesh 3294
32.20.2.2. Checking Model Selections 3296
32.20.2.3. Checking Boundary and Cell Zone Conditions 3298
32.20.2.4. Checking Material Properties 3301
32.20.2.5. Checking the Solver Settings 3302
32.21. Convergence and Stability 3305
32.21.1. Judging Convergence 3305
32.21.2. Step-by-Step Solution Processes 3306
32.21.2.1. Selecting a Subset of the Solution Equations 3307
32.21.2.2. Turning Reactions On and Off 3308
32.21.3. Modifying Algebraic Multigrid Parameters 3308
32.21.4. Modifying the Multi-Stage Parameters 3308
32.21.5. Robustness with Meshes of Poor Quality 3308
32.21.6. Warped-Face Gradient Correction 3311
32.21.7. Numerical Noise Filter for the Energy Equation 3312
32.22. Solution Steering 3312
32.22.1. Overview of Solution Steering 3313
32.22.2. Solution Steering Strategy 3313
32.22.2.1. Initialization 3313
32.22.3. Using Solution Steering 3314
33. Adapting the Mesh 3319
33.1. Using Adaption 3319
33.1.1. Adaption Example 3320
33.1.2. Adaption Guidelines 3321
33.2. Refining and Coarsening 3323
33.2.1. Predefined Criteria for Adaption 3328
33.2.1.1. Aerodynamics Adaption 3329
33.2.1.2. Combustion Adaption 3330
33.2.1.3. VOF Adaption 3334
33.3. Adaption Examples 3337
33.3.1. Boundary Cell Register 3337
33.3.2. Region Cell Register 3340
33.3.3. Field Variable Cell Registers (gradients, scaling, and so on) 3343
33.3.4. Expression Adaption Refinement 3345
33.4. Legacy Anisotropic Adaption 3347
33.4.1. Limitations of Legacy Anisotropic Adaption 3348
33.4.2. Performing Legacy Anisotropic Adaption 3348
33.4.3. Boundary Layer Redistribution 3350
33.5. Geometry-Based Adaption 3350
33.5.1. Performing Geometry-Based Adaption 3350
34. Creating Surfaces and Cell Registers for Displaying and Reporting Data 3353
34.1. Using Surfaces 3353
34.1.1. Zone Surfaces 3355

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.1.2. Partition Surfaces 3356
34.1.3. Imprint Surfaces 3358
34.1.4. Point Surfaces 3360
34.1.4.1. Using the Point Tool 3362
34.1.5. Structural Point Surfaces 3363
34.1.6. Line and Rake Surfaces 3365
34.1.6.1. Using the Line Tool 3367
34.1.6.1.1. Initializing the Line Tool 3367
34.1.6.1.2. Translating the Line Tool 3368
34.1.6.1.3. Rotating the Line Tool 3368
34.1.6.1.4. Resizing the Line Tool 3369
34.1.6.1.5. Resetting the Line Tool 3369
34.1.7. Plane Surfaces 3369
34.1.7.1. Using the Plane Tool 3372
34.1.8. Quadric Surfaces 3375
34.1.9. Iso-surfaces 3377
34.1.10. Clipping Surfaces 3379
34.1.11. Transforming Surfaces 3382
34.1.12. Grouping, Editing, Renaming, and Deleting Surfaces 3384
34.1.12.1. Grouping Surfaces 3386
34.1.12.2. Editing and Renaming Surfaces 3386
34.1.12.3. Deleting Surfaces 3387
34.1.12.4. Surface Statistics 3387
34.2. Using Cell Registers 3387
34.2.1. Region 3388
34.2.1.1. Defining a Region 3388
34.2.1.2. Setting Up a Region Cell Register 3388
34.2.2. Boundary 3390
34.2.3. Variable Limiter 3391
34.2.4. Field Variable 3392
34.2.4.1. Approaches For Deriving Field Values 3392
34.2.4.2. Setting Up a Field Variable Cell Register 3394
34.2.5. Residuals 3395
34.2.6. Volume 3396
34.2.6.1. Volume Cell Register Approach 3396
34.2.6.2. Setting Up a Volume Cell Register 3397
34.2.7. Yplus/Ystar 3398
34.2.7.1. Yplus/Ystar Approach 3398
34.2.7.2. Setting up a Yplus/Ystar Cell Register 3399
34.2.8. Manage Cell Registers 3400
34.2.9. Cell Register Operations 3401
34.2.10. Copying and Renaming Cell Registers 3403
35. Displaying Graphics 3405
35.1. Basic Graphics Generation 3405
35.1.1. Graphics Performance 3406
35.1.2. Displaying the Mesh 3408
35.1.2.1. Generating Mesh or Outline Plots 3411
35.1.2.2. Mesh and Outline Display Options 3413
35.1.2.2.1. Modifying the Mesh Colors 3413
35.1.2.2.2. Realistic Rendering of Materials 3416
35.1.2.2.3. Adding Features to an Outline Display 3419

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.1.2.2.4. Drawing Partition Boundaries 3420
35.1.2.2.5. Shrinking Faces and Cells in the Display 3420
35.1.2.3. Creating and Using Mesh Plot Definitions 3422
35.1.3. Displaying Contours and Profiles 3422
35.1.3.1. Quickly Coloring Surfaces by Field Variable Value 3423
35.1.3.2. Generating Contour and Profile Plots 3424
35.1.3.3. Contour and Profile Plot Options 3427
35.1.3.3.1. Drawing Filled Contours or Profiles 3427
35.1.3.3.2. Specifying the Range of Magnitudes Displayed 3428
35.1.3.3.3. Including the Mesh in the Contour Plot 3430
35.1.3.3.4. Choosing Node or Cell Values and Node or Boundary Values 3431
35.1.3.3.5. Storing Contour Plot Settings 3431
35.1.3.4. Creating and Using Contour Plot Definitions 3432
35.1.4. Displaying Vectors 3433
35.1.4.1. Generating Vector Plots 3434
35.1.4.2. Displaying Relative Velocity Vectors 3436
35.1.4.3. Vector Plot Options 3436
35.1.4.3.1. Scaling the Vectors 3437
35.1.4.3.2. Skipping Vectors 3437
35.1.4.3.3. Drawing Vectors in the Plane of the Surface 3438
35.1.4.3.4. Displaying Fixed-Length Vectors 3438
35.1.4.3.5. Displaying Vector Components 3438
35.1.4.3.6. Specifying the Range of Magnitudes Displayed 3439
35.1.4.3.7. Changing the Scalar Field Used for Coloring the Vectors 3439
35.1.4.3.8. Displaying Vectors Using a Single Color 3439
35.1.4.3.9. Including the Mesh in the Vector Plot 3440
35.1.4.3.10. Changing the Arrow Characteristics 3440
35.1.4.4. Creating and Managing Custom Vectors 3440
35.1.4.4.1. Creating Custom Vectors 3440
35.1.4.4.2. Manipulating, Saving, and Loading Custom Vectors 3441
35.1.4.5. Creating and Using Vector Plot Definitions 3443
35.1.5. Displaying Pathlines 3443
35.1.5.1. Steps for Generating Pathlines 3444
35.1.5.2. Options for Pathline Plots 3445
35.1.5.2.1. Including the Mesh in the Pathline Display 3445
35.1.5.2.2. Controlling the Pathline Style 3446
35.1.5.2.3. Controlling Pathline Colors 3447
35.1.5.2.4. "Thinning" Pathlines 3447
35.1.5.2.5. Coarsening Pathlines 3447
35.1.5.2.6. Reversing the Pathlines 3447
35.1.5.2.7. Plotting Oil-Flow Pathlines 3448
35.1.5.2.8. Controlling the Pulse Mode 3448
35.1.5.2.9. Controlling the Accuracy 3448
35.1.5.2.10. Plotting Relative Pathlines 3448
35.1.5.2.11. Generating an XY Plot Along Pathline Trajectories 3448
35.1.5.2.12. Saving Pathline Data 3449
35.1.5.2.12.1. Standard Type 3449
35.1.5.2.12.2. Geometry Type 3450
35.1.5.2.12.3. EnSight Type 3451
35.1.5.2.13. Choosing Node or Cell Values 3452
35.1.5.3. Creating and Using Pathline Definitions 3452

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.1.6. Displaying a Scene 3453
35.1.6.1. Generating a Scene 3453
35.1.7. Displaying Results on a Sweep Surface 3454
35.1.7.1. Steps for Generating a Plot Using a Sweep Surface 3454
35.1.7.2. Animating a Sweep Surface Display 3456
35.1.8. Hiding the Graphics Window Display 3456
35.2. Customizing the Graphics Display 3457
35.2.1. Embedded Graphics Window Dashboards 3458
35.2.1.1. Manually Embedding Windows 3460
35.2.1.2. Automatically Embedding Windows 3462
35.2.2. Advanced Graphics Overlays 3463
35.2.3. Managing Multiple Graphics Windows 3464
35.2.3.1. Setting the Active Window 3466
35.2.4. Showing Boundary Markers 3466
35.2.5. Changing the Legend Display 3467
35.2.5.1. Controlling the Titles, Axes, Ruler, Logo, and Colormap 3467
35.2.5.2. Editing the Legend 3468
35.2.5.3. Adding a Title to the Caption 3468
35.2.5.4. Enabling/Disabling the Axes 3468
35.2.5.5. Enabling/Disabling the Ruler 3468
35.2.5.6. Modifying and Displaying/Hiding the Logo 3469
35.2.5.7. Colormap Alignment 3469
35.2.6. Adding Text to the Graphics Window 3469
35.2.6.1. Adding Text Using the Annotate Dialog Box 3470
35.2.6.2. Editing Existing Annotation Text 3470
35.2.6.3. Clearing Annotation Text 3471
35.2.7. Changing the Colormap 3471
35.2.7.1. Colormap Nomenclature 3473
35.2.7.2. Predefined Colormaps 3473
35.2.7.3. Selecting a Colormap 3486
35.2.7.3.1. Specifying the Colormap Size and Scale 3487
35.2.7.3.2. Changing the Number Format 3487
35.2.7.4. Displaying Colormap Labels 3487
35.2.7.5. Creating a Customized Colormap 3488
35.2.7.6. Colormap References 3491
35.2.8. Adding Lights 3491
35.2.8.1. Controlling Lighting Effects with the Display Options Dialog Box 3492
35.2.8.2. Controlling Lighting Effects with the **Lights** Dialog Box 3492
35.2.8.3. Defining Light Sources 3492
35.2.8.3.1. Removing a Light 3494
35.2.8.3.2. Resetting the Light Definitions 3494
35.2.9. Modifying the Rendering Options 3494
35.2.9.1. Graphics Device Information 3496
35.3. Enhanced Graphics Visual Effects 3496
35.3.1. Predefined Graphics Effects Grouped for Optimization 3497
35.3.2. Graphics Effects Options 3498
35.4. Controlling the Mouse Button Functions 3503
35.5. Viewing the Application Window 3506
35.6. Controlling the Display State and Modifying the View 3506
35.6.1. Specifying a Display State 3507
35.6.2. Selecting a View 3508

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.6.3. Manipulating the Display 3510
35.6.3.1. Scaling and Centering 3511
35.6.3.2. Rotating the Display 3511
35.6.3.2.1. Spinning the Display with the Mouse 3512
35.6.3.3. Translating the Display 3512
35.6.3.4. Zooming the Display 3513
35.6.4. Controlling Perspective and Camera Parameters 3514
35.6.4.1. Perspective and Orthographic Views 3514
35.6.4.2. Modifying Camera Parameters 3514
35.6.5. Saving and Restoring Views 3515
35.6.5.1. Restoring the Default View 3516
35.6.5.2. Returning to Previous Views 3516
35.6.5.3. Saving Views 3516
35.6.5.4. Reading View Files 3517
35.6.5.5. Deleting Views 3517
35.6.6. Mirroring and Periodic Repeats 3517
35.6.6.1. Periodic Repeats for Graphics 3520
35.6.6.2. Mirroring for Graphics 3521
35.7. Advanced Scene Composition 3522
35.7.1. Selecting the Object(s) to be Manipulated 3523
35.7.2. Changing an Object's Display Properties 3523
35.7.2.1. Controlling Visibility 3524
35.7.2.2. Controlling Object Color and Transparency 3525
35.7.3. Transforming Geometric Objects in a Scene 3526
35.7.3.1. Translating Objects 3527
35.7.3.2. Rotating Objects 3527
35.7.3.3. Scaling Objects 3527
35.7.3.4. Displaying the Meridional View 3528
35.7.4. Modifying Iso-Values 3528
35.7.4.1. Steps for Modifying Iso-Values 3528
35.7.5. Modifying Pathline Attributes 3528
35.7.6. Deleting an Object from the Scene 3529
35.7.7. Adding a Bounding Frame 3529
35.8. Animating Graphics 3531
35.8.1. Creating an Animation 3533
35.8.1.1. Deleting Key Frames 3533
35.8.2. Playing an Animation 3533
35.8.2.1. Playing Back an Excerpt 3534
35.8.2.2. "Fast-Forwarding" the Animation 3534
35.8.2.3. Continuous Animation 3534
35.8.2.4. Stopping the Animation 3534
35.8.2.5. Advancing the Animation Frame by Frame 3535
35.8.3. Saving an Animation 3535
35.8.3.1. Animation File 3535
35.8.3.2. Picture File 3535
35.8.3.3. MPEG File 3536
35.8.4. Reading an Animation File 3536
35.8.5. Notes on Animation 3536
35.9. Histogram and XY Plots 3537
35.9.1. Plot Types 3537
35.9.1.1. XY Plots 3537

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.9.1.2. Histograms 3538
35.9.1.3. Enhanced Interactive Plots 3539
35.9.2. XY Plots of Solution Data 3541
35.9.2.1. Steps for Generating Solution XY Plots 3541
35.9.2.2. Options for Solution XY Plots 3545
35.9.2.2.1. Including External Data in the Solution XY Plot 3545
35.9.2.2.2. Choosing Node or Cell Values 3545
35.9.2.2.3. Saving the Plot Data to a File 3546
35.9.3. Creating an XY Plot From Multiple Data Sources (Including Files) 3546
35.9.3.1. Steps for Generating XY Plots of Data from Multiple Sources 3546
35.9.4. XY Plots of Profiles 3548
35.9.4.1. Steps for Generating Plots of Profile Data 3548
35.9.4.2. Steps for Generating Plots of Interpolated Profile Data 3549
35.9.5. XY Plots of Circumferential Averages 3550
35.9.5.1. Steps for Generating an XY Plot of Circumferential Averages 3550
35.9.5.2. Customizing the Appearance of the Plot 3552
35.9.6. XY Plot File Format 3552
35.9.7. Residual Plots 3553
35.9.8. Histograms 3553
35.9.8.1. Steps for Generating Histogram Plots 3554
35.9.8.2. Options for Histogram Plots 3555
35.9.8.2.1. Specifying the Range of Values Plotted 3555
35.9.9. Modifying Axis Attributes 3555
35.9.9.1. Using the Axes Dialog Box 3556
35.9.9.1.1. Changing the Axis Label 3556
35.9.9.1.2. Changing the Format of the Data Labels 3556
35.9.9.1.3. Choosing Logarithmic or Decimal Scaling 3557
35.9.9.1.4. Resetting the Range of the Axis 3557
35.9.9.1.5. Controlling the Major and Minor Rules 3557
35.9.10. Modifying Curve Attributes 3557
35.9.10.1. Using the Curves Dialog Box 3558
35.9.10.1.1. Changing the Line Style 3558
35.9.10.1.2. Changing the Marker Style 3559
35.9.10.1.3. Previewing the Curve Style 3559
35.10. Fast Fourier Transform (FFT) Postprocessing 3559
35.10.1. Limitations of the FFT Algorithm 3560
35.10.2. Windowing 3560
35.10.3. Fast Fourier Transform (FFT) 3561
35.10.4. Using the FFT Utility 3562
35.10.4.1. Loading Data for Spectral Analysis 3563
35.10.4.2. Customizing the Input and Defining the Spectrum Smoothing 3564
35.10.4.2.1. Customizing the Input Signal Data Set 3564
35.10.4.2.2. Spectrum Smoothing Through Signal Segmentation 3565
35.10.4.2.3. Viewing Data Statistics 3565
35.10.4.2.4. Customizing Titles and Labels 3565
35.10.4.2.5. Applying the Changes in the Input Signal Data 3566
35.10.4.3. Customizing the Output 3566
35.10.4.3.1. Specifying a Function for the Y Axis 3566
35.10.4.3.2. Specifying a Function for the X Axis 3568
35.10.4.3.3. Specifying Output Options 3570
35.10.4.3.4. Specifying a Windowing Technique 3570

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.10.4.3.5. Specifying Labels and Titles 3570
35.10.4.4. Performing a VRXPERIENCE Sound Analysis 3570
35.11. Cumulative Force, Moment, and Coefficients Plots 3573
35.11.1. Steps for Generating Cumulative Plots 3574
36. Reporting Alphanumeric Data 3577
36.1. Reporting Conventions 3577
36.2. Monitoring and Reporting Solution Data 3578
36.2.1. Creating Report Definitions 3578
36.2.1.1. Surface Report Definitions 3582
36.2.1.2. Volume Report Definitions 3583
36.2.1.3. Force and Moment Report Definitions 3585
36.2.1.4. Flux Report Definition 3590
36.2.1.5. Aerodamping Report Definition 3592
36.2.1.6. DPM Report Definition 3594
36.2.1.7. User Defined Report Definition 3596
36.2.1.7.1. User Defined Report Definition Function 3596
36.2.1.7.2. User Defined Report Definition Function Hooking 3597
36.2.1.8. Expression Report Definition 3597
36.2.2. Report Files and Report Plots 3599
36.2.2.1. Creating Report Files 3599
36.2.2.2. Creating Report Plots 3602
36.2.2.3. Moving Average Monitors 3605
36.2.2.4. Clearing File and Plot Histories 3608
36.3. Creating Output Parameters 3609
36.4. Fluxes Through Boundaries 3611
36.4.1. Generating a Flux Report 3611
36.4.2. Flux Reporting for Reacting Flows 3614
36.4.3. Flux Reporting with Particles 3615
36.4.4. Flux Reporting with Multiphase 3616
36.4.5. Flux Reporting with Other Volumetric Sources 3617
36.5. Forces on Boundaries 3617
36.5.1. Generating a Force, Moment, or Center of Pressure Report 3617
36.5.1.1. Example 3620
36.6. Projected Surface Area Calculations 3621
36.7. Surface Integration 3622
36.7.1. Generating a Surface Integral Report 3623
36.8. Volume Integration 3625
36.8.1. Generating a Volume Integral Report 3625
36.9. Histogram Reports 3626
36.10. Discrete Phase 3627
36.11. S2S Information 3627
36.12. Reference Values 3627
36.12.1. Setting Reference Values 3627
36.12.2. Setting the Reference Zone 3629
36.13. Summary Reports of Case Settings 3629
36.13.1. Modified Settings Summary 3629
36.13.2. Generating a Summary Report 3630
36.14. System Resource Usage 3631
36.14.1. Processor Information 3631
36.14.2. Memory Information 3632
36.14.3. Process and Model Timers 3633

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37. Field Function Definitions 3635
37.1. Node, Cell, and Facet Values 3635
37.1.1. Cell Values 3635
37.1.2. Node Values 3636
37.1.2.1. Vertex Values for Points That Are Not Mesh Nodes 3636
37.1.3. Facet Values 3636
37.1.3.1. Facet Values on Zone Surfaces 3637
37.1.3.2. Facet Values on Postprocessing Surfaces 3637
37.2. Velocity Reporting Options 3637
37.3. Field Variables Listed by Category 3639
37.4. Alphabetical Listing of Field Variables and Their Definitions 3669
37.5. Custom Field Functions 3738
37.5.1. Creating a Custom Field Function 3738
37.5.1.1. Using the Calculator Buttons 3740
37.5.1.2. Using the Field Functions List 3740
37.5.2. Manipulating, Saving, and Loading Custom Field Functions 3741
37.5.3. Sample Custom Field Functions 3742
38. Parallel Processing 3745
38.1. Introduction to Parallel Processing 3745
38.1.1. Recommended Usage of Parallel Ansys Fluent 3747
38.2. Starting Parallel Ansys Fluent Using Fluent Launcher 3748
38.2.1. Setting Parallel Scheduler Options in Fluent Launcher 3750
38.2.2. Setting Additional Options When Running on Remote Linux Machines 3753
38.2.2.1. Setting Job Scheduler Options When Running on Remote Linux Machines 3755
38.3. Starting Parallel Ansys Fluent on a Windows System 3758
38.3.1. Starting Parallel Ansys Fluent on a Windows System Using Command Line Options 3758
38.3.1.1. Starting Parallel Ansys Fluent with the Microsoft Job Scheduler 3760
38.4. Starting Parallel Ansys Fluent on a Linux System 3763
38.4.1. Starting Parallel Ansys Fluent on a Linux System Using Command Line Options 3763
38.4.2. Setting Up Your Secure Shell Clients 3766
38.4.2.1. Configuring the `ssh` Client 3766
38.5. Mesh Partitioning and Load Balancing 3767
38.5.1. Overview of Mesh Partitioning 3767
38.5.2. Partitioning the Mesh Automatically 3768
38.5.2.1. Reporting During Auto Partitioning 3770
38.5.3. Partitioning the Mesh Manually and Balancing the Load 3770
38.5.3.1. Guidelines for Partitioning the Mesh 3770
38.5.4. Using the Partitioning and Load Balancing Dialog Box 3771
38.5.4.1. Partitioning 3771
38.5.4.1.1. Example of Setting Selected Cell Registers to Specified Partition IDs 3777
38.5.4.1.2. Partitioning Within Zones or Registers 3779
38.5.4.1.3. Reporting During Partitioning 3779
38.5.4.1.4. Resetting the Partition Parameters 3780
38.5.4.2. Load Balancing 3780
38.5.5. Mesh Partitioning Methods 3782
38.5.5.1. Partition Methods 3783
38.5.5.2. Optimizations 3787
38.5.5.3. Pretesting 3788
38.5.5.4. Using the Partition Filter 3789
38.5.6. Checking the Partitions 3789
38.5.6.1. Interpreting Partition Statistics 3790

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.5.6.2. Examining Partitions Graphically 3793
38.5.7. Load Distribution 3793
38.5.8. Troubleshooting 3793
38.6. Using General Purpose Graphics Processing Units (GPGPUs) With the Algebraic Multigrid (AMG) Solver 3794
38.6.1. Requirements 3794
38.6.2. Limitations 3794
38.6.3. Using and Managing GPGPUs 3795
38.7. Controlling the Threads 3795
38.8. Checking Network Connectivity 3796
38.9. Checking and Improving Parallel Performance 3797
38.9.1. Parallel Check 3797
38.9.2. Checking Parallel Performance 3798
38.9.2.1. Checking Latency and Bandwidth 3800
38.9.3. Optimizing the Parallel Solver 3802
38.9.3.1. Increasing the Report Interval 3802
38.9.3.2. Accelerating View Factor Calculations for General Purpose Computing on Graphics Processing Units (GPGPUs) 3802
38.9.3.3. Accelerating Discrete Ordinates (DO) Radiation Calculations 3803
38.9.4. Clearing the Linux File Cache Buffers 3803
39. Using Simulation Reports 3805
39.1. Overview of Simulation Reports 3805
39.1.1. Limitations for Simulations Reports 3808
39.2. Preparing Simulation Reports 3808
39.2.1. Setting General Report Properties 3809
39.2.2. Organizing Your Simulation Report 3809
39.3. Generating Simulation Reports 3811
39.4. Viewing Simulation Reports 3812
39.4.1. Viewing System Information 3813
39.4.2. Viewing Geometry and Mesh Information 3813
39.4.3. Viewing Simulation Setup Information 3815
39.4.4. Viewing Run Information 3819
39.4.5. Viewing Solution Status 3820
39.4.6. Viewing Named Expression Information 3821
39.4.7. Viewing Report Definition Information 3821
39.4.8. Viewing Plot Information 3821
39.4.9. Viewing Contours, Vectors, Pathlines, XY Plots, Scenes, and Animations 3823
39.4.9.1. Changing the Layout of Your Results 3827
39.5. Saving Simulation Reports 3828
39.6. Customizing Simulation Reports 3828
39.6.1. Adding Additional Graphics to Your Report 3828
39.6.2. Hiding and Showing Report Sections 3829
40. Design Analysis and Optimization 3831
40.1. The Adjoint Solver 3831
40.1.1. General Observables 3833
40.1.2. General Operations 3837
40.1.3. Discrete Versus Continuous Adjoint Solver 3839
40.1.4. Discrete Adjoint Solver Overview 3839
40.1.5. Adjoint Solver Stabilization 3843
40.1.6. Solution-Based Adaption 3844
40.1.7. Using The Data To Improve A Design 3845

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.1.7.1. Smoothing and Mesh Morphing 3846
40.1.7.1.1. Polynomials-Based Approach 3846
40.1.7.1.2. Direct Interpolation Method 3847
40.1.7.1.3. Radial Basis Function (RBF) 3848
40.2. Using the Adjoint Solver 3848
40.2.1. Model Considerations for Using Adjoint Solver 3849
40.2.1.1. Basic Assumptions and Consistency Checks 3849
40.2.1.2. User-Defined Sources 3852
40.2.2. Defining Observables 3853
40.2.2.1. Creating New Observables 3854
40.2.2.2. Editing Observable Definitions 3855
40.2.2.3. Selecting an Observable for Sensitivity Calculation 3859
40.2.3. Solving the Adjoint 3860
40.2.3.1. Using the Adjoint Solution Methods Dialog Box 3860
40.2.3.2. Using the Adjoint Solution Controls Dialog Box 3862
40.2.3.2.1. Stabilization Strategies, Schemes, and Settings 3866
40.2.3.2.1.1. Dissipation Scheme 3867
40.2.3.2.1.2. Residual Minimization Scheme 3869
40.2.3.2.1.3. Spatial Stabilization Scheme 3870
40.2.3.2.1.4. Modal Stabilization Scheme 3872
40.2.3.3. Working with Adjoint Residual Monitors 3874
40.2.3.4. Printing and Postprocessing the Adjoint Equation Residuals 3875
40.2.3.5. Running the Adjoint Calculation 3876
40.2.3.5.1. Automatic Saving of Case and Data Files During an Adjoint Calculation 3878
40.2.4. Postprocessing of Adjoint Solutions 3879
40.2.4.1. Field Data 3879
40.2.4.2. Scalar Data 3885
40.2.5. Modifying the Geometry Using the Design Tool 3886
40.2.5.1. Defining the Region for the Design Change 3888
40.2.5.2. Defining Region Conditions 3891
40.2.5.3. Exporting Sensitivity Data 3892
40.2.5.4. Defining Observable Objectives 3892
40.2.5.5. Defining Conditions for the Deformation 3893
40.2.5.6. Design Tool Numerics 3900
40.2.5.7. Shape Modification 3904
40.2.6. Using the Gradient-Based Optimizer 3910
40.3. The Mesh Morpher/Optimizer 3921
40.3.1. Limitations 3921
40.3.2. The Optimization Process 3921
40.3.3. Optimizers 3922
40.3.3.1. The Compass Optimizer 3922
40.3.3.2. The NEWUOA Optimizer 3923
40.3.3.3. The Simplex Optimizer 3923
40.3.3.4. The Torczon Optimizer 3923
40.3.3.5. The Powell Optimizer 3924
40.3.3.6. The Rosenbrock Optimizer 3924
40.4. Using the Mesh Morpher/Optimizer 3924
41. Performing System Coupling Simulations Using Fluent 3951
41.1. Performing System Coupling in Ansys Workbench 3951
41.2. Performing System Coupling in the GUI or CLI 3953
41.2.1. Generating a System Coupling File 3953

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

41.3. Supported Capabilities and Limitations 3954
41.4. Variables Available for System Coupling 3957
41.4.1. Force transferred to System Coupling from a Wall Boundary 3957
41.4.2. Force transferred to System Coupling from a Porous Jump Boundary 3958
41.4.3. Displacement transferred from System Coupling 3959
41.4.4. Displacement transferred from System Coupling to a Sliding Mesh Zone 3960
41.4.5. Absolute Pressure Example 3960
41.5. System Coupling Related Settings in Fluent 3960
41.6. FSI Setup Recommendations for Fluent-Mechanical Couplings 3962
41.6.1. Using Contact Detection for Fluent-Mechanical FSI Problems 3962
41.6.2. Recommendations for Dynamic Mesh Settings for Fluent-Mechanical FSI 3964
41.6.3. Pathologies & Candidate Resolutions for Fluent-Mechanical FSI 3966
41.6.3.1. Mesh Folds within the First Coupling Steps 3967
41.6.3.2. Deformed Prism Layers 3967
41.6.3.2.1. Using Boundary Layer Smoothing and Region Face Remeshing 3968
41.6.3.2.2. Overset Meshes 3972
41.6.3.3. Interior Elements have High Skewness or Are Too Large/small 3974
41.6.3.4. Divergence if Flow Block-Off is Established at the Beginning of a Run 3976
41.7. How Fluent's Execution is Affected by System Couplings 3977
41.8. Restarting Fluent Analyses as Part of System Couplings 3977
41.8.1. Generating Fluent Restart Files 3977
41.8.2. Specify a Restart Point in Fluent 3977
41.8.3. Making Changes in Fluent Before Restarting 3978
41.8.4. Recovering the Fluent Restart Point after a Workbench Crash 3978
41.9. System Coupling case with Fluent using Patched Data 3979
41.10. Running Fluent as a Participant from System Coupling's GUI or CLI 3980
41.11. Troubleshooting Two-Way Coupled Analysis Problems 3980
41.12. Product Licensing Considerations when using System Coupling 3981
42. Customizing Fluent 3983
43. Task Page Reference Guide 3985
43.1. Meshing Task Page 3985
43.2. Setup Task Page 3985
43.3. General Task Page 3986
43.3.1. Scale Mesh Dialog Box 3988
43.3.2. Mesh Display Dialog Box 3990
43.3.3. Set Units Dialog Box 3994
43.3.4. Define Unit Dialog Box 3995
43.3.5. Mesh Colors Dialog Box 3996
43.4. Models Task Page 3997
43.4.1. Multiphase Model Dialog Box 4001
43.4.2. Energy Dialog Box 4016
43.4.3. Viscous Model Dialog Box 4017
43.4.4. Radiation Model Dialog Box 4037
43.4.5. View Factors and Clustering Dialog Box 4042
43.4.6. Participating Boundary Zones Dialog Box 4046
43.4.7. Solar Calculator Dialog Box 4048
43.4.8. Heat Exchanger Model Dialog Box 4049
43.4.9. Dual Cell Heat Exchanger Dialog Box 4050
43.4.10. Set Dual Cell Heat Exchanger Dialog Box 4051
43.4.11. Heat Transfer Data Table Dialog Box 4054
43.4.12. NTU Table Dialog Box 4056

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

43.4.13. Copy From Dialog Box 4057
43.4.14. Ungrouped Macro Heat Exchanger Dialog Box 4057
43.4.15. Velocity Effectiveness Curve Dialog Box 4062
43.4.16. Core Porosity Model Dialog Box 4062
43.4.17. Macro Heat Exchanger Group Dialog Box 4064
43.4.18. Species Model Dialog Box 4069
43.4.19. Coal Calculator Dialog Box 4091
43.4.20. Integration Parameters Dialog Box 4095
43.4.21. Flamelet 3D Surfaces Dialog Box 4097
43.4.22. Flamelet 2D Curves Dialog Box 4100
43.4.23. Unsteady Flamelet Parameters Dialog Box 4101
43.4.24. Flamelet Fluid Zones Dialog Box 4102
43.4.25. Select Transported Scalars Dialog Box 4103
43.4.26. Distribution of Points Dialog Box 4103
43.4.27. PDF Table Dialog Box 4105
43.4.28. Spark Ignition Dialog Box 4109
43.4.29. Set Spark Ignition Dialog Box 4110
43.4.30. Autoignition Model Dialog Box 4111
43.4.31. Inert Dialog Box 4114
43.4.32. NOx Model Dialog Box 4116
43.4.33. Soot Model Dialog Box 4125
43.4.34. Sticking Coefficients Dialog Box 4135
43.4.35. Mechanism Dialog Box 4135
43.4.36. Reactor Network Dialog Box 4137
43.4.37. Decoupled Detailed Chemistry Dialog Box 4140
43.4.38. Reacting Channel Model Dialog Box 4141
43.4.39. Reacting Channel 2D Curves Dialog Box 4144
43.4.40. Discrete Phase Model Dialog Box 4145
43.4.41. DEM Collisions Dialog Box 4155
43.4.42. Create Collision Partner Dialog Box 4156
43.4.43. Copy Collision Partner Dialog Box 4156
43.4.44. Rename Collision Partner Dialog Box 4157
43.4.45. DEM Collision Settings Dialog Box 4157
43.4.46. Solidification and Melting Dialog Box 4158
43.4.47. Acoustics Model Dialog Box 4160
43.4.48. Acoustic Sources Dialog Box 4164
43.4.49. Acoustic Receivers Dialog Box 4165
43.4.50. Basic Shapes Dialog Box 4166
43.4.51. Integration Surface Dialog Box 4167
43.4.52. Interior Cell Zone Selection Dialog Box 4168
43.4.53. Structural Model Dialog Box 4168
43.4.54. Eulerian Wall Film Dialog Box 4169
43.4.55. Potential/Li-ion Battery Dialog Box 4176
43.5. Materials Task Page 4177
43.5.1. Create/Edit Materials Dialog Box 4180
43.5.2. Fluent Database Materials Dialog Box 4191
43.5.3. GRANTA MDS Materials Dialog Box 4193
43.5.4. Open Database Dialog Box 4195
43.5.5. User-Defined Database Materials Dialog Box 4196
43.5.6. Copy Case Material Dialog Box 4197
43.5.7. Material Properties Dialog Box 4198

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

43.5.8. Edit Property Methods Dialog Box 4199
43.5.9. New Material Name Dialog Box 4200
43.5.10. Polynomial Profile Dialog Box 4200
43.5.11. Piecewise-Linear Profile Dialog Box 4201
43.5.12. Piecewise-Polynomial Profile Dialog Box 4203
43.5.13. NASA-9-Coefficient Piecewise-Polynomial Profile Dialog Box 4205
43.5.14. Model Options Dialog Box 4207
43.5.15. Compressible Liquid Dialog Box 4208
43.5.16. User-Defined Functions Dialog Box 4209
43.5.17. Sutherland Law Dialog Box 4210
43.5.18. Power Law Dialog Box 4211
43.5.19. Non-Newtonian Power Law Dialog Box 4213
43.5.20. Carreau Model Dialog Box 4214
43.5.21. Cross Model Dialog Box 4215
43.5.22. Herschel-Bulkley Dialog Box 4216
43.5.23. Biaxial Conductivity Dialog Box 4218
43.5.24. Cylindrical Orthotropic Conductivity Dialog Box 4218
43.5.25. Orthotropic Conductivity Dialog Box 4220
43.5.26. Anisotropic Conduction - Principal Components Dialog Box 4221
43.5.27. Anisotropic Conductivity Dialog Box 4222
43.5.28. Species Dialog Box 4223
43.5.29. Reactions Dialog Box 4224
43.5.30. Backward Reaction Parameters Dialog Box 4229
43.5.31. Third-Body Efficiency Dialog Box 4230
43.5.32. Pressure-Dependent Reaction Dialog Box 4231
43.5.33. Coverage-Dependent Reaction Dialog Box 4233
43.5.34. Reference Mass Fractions Dialog Box 4234
43.5.35. Reaction Mechanisms Dialog Box 4235
43.5.36. Site Parameters Dialog Box 4236
43.5.37. Mass Diffusion Coefficients Dialog Box 4237
43.5.38. Thermal Diffusion Coefficients Dialog Box 4239
43.5.39. UDS Diffusion Coefficients Dialog Box 4240
43.5.40. WSGGM User Specified Dialog Box 4241
43.5.41. Gray-Band Absorption Coefficient Dialog Box 4242
43.5.42. Delta-Eddington Scattering Function Dialog Box 4243
43.5.43. Gray-Band Refractive Index Dialog Box 4243
43.5.44. Single Rate Model Dialog Box 4244
43.5.45. Secondary Rate Model Dialog Box 4245
43.5.46. Two Competing Rates Model Dialog Box 4246
43.5.47. CPD Model Dialog Box 4248
43.5.48. Kinetics/Diffusion-Limited Combustion Model Dialog Box 4249
43.5.49. Intrinsic Combustion Model Dialog Box 4249
43.5.50. Multiple Surface Reactions Dialog Box 4251
43.5.51. Edit Material Dialog Box 4251
43.6. Cell Zone Conditions Task Page 4252
43.6.1. Fluid Dialog Box 4255
43.6.2. Solid Dialog Box 4266
43.6.3. Copy Conditions Dialog Box 4271
43.6.4. Operating Conditions Dialog Box 4271
43.6.5. Select Input Parameter Dialog Box 4274
43.6.6. Profiles Dialog Box 4275

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

43.6.7. Replicate Profile Dialog Box 4277
43.6.8. Orient Profile Dialog Box 4279
43.6.9. Write Profile Dialog Box 4280
43.7. Boundary Conditions Task Page 4282
43.7.1. Axis Dialog Box 4284
43.7.2. Degassing Dialog Box 4285
43.7.3. Exhaust Fan Dialog Box 4286
43.7.4. Fan Dialog Box 4292
43.7.5. Inlet Vent Dialog Box 4295
43.7.6. Intake Fan Dialog Box 4302
43.7.7. Interface Dialog Box 4309
43.7.8. Interior Dialog Box 4309
43.7.9. Mass-Flow Inlet Dialog Box 4310
43.7.10. Mass-Flow Outlet Dialog Box 4318
43.7.11. Outflow Dialog Box 4322
43.7.12. Outlet Vent Dialog Box 4324
43.7.13. Overset Dialog Box 4331
43.7.14. Periodic Dialog Box 4331
43.7.15. Porous Jump Dialog Box 4332
43.7.16. Pressure Far-Field Dialog Box 4334
43.7.17. Pressure Inlet Dialog Box 4340
43.7.18. Pressure Outlet Dialog Box 4347
43.7.19. Radiator Dialog Box 4356
43.7.20. RANS/LES Interface Dialog Box 4358
43.7.21. Symmetry Dialog Box 4359
43.7.22. Velocity Inlet Dialog Box 4360
43.7.23. Wall Dialog Box 4370
43.7.24. Periodic Conditions Dialog Box 4389
43.7.25. **Perforated Walls** Dialog Box 4390
43.8. Overset Interfaces Task Page 4393
43.8.1. Create/Edit Overset Interfaces Dialog Box 4395
43.9. Dynamic Mesh Task Page 4396
43.9.1. Mesh Method Settings Dialog Box 4399
43.9.2. Mesh Smoothing Parameters Dialog Box 4402
43.9.3. Advanced Remeshing Settings Dialog Box 4405
43.9.4. Mesh Scale Info Dialog Box 4405
43.9.5. Options Dialog Box 4406
43.9.6. In-Cylinder Output Controls Dialog Box 4411
43.9.7. Six DOF Properties Dialog Box 4412
43.9.8. Flow Control Settings Dialog Box 4415
43.9.9. Dynamic Mesh Events Dialog Box 4417
43.9.10. Define Event Dialog Box 4418
43.9.11. Events Preview Dialog Box 4420
43.9.12. Dynamic Mesh Zones Dialog Box 4421
43.9.13. Orientation Calculator Dialog Box 4429
43.9.14. Zone Scale Info Dialog Box 4431
43.9.15. Zone Motion Dialog Box 4431
43.9.16. Mesh Motion Dialog Box 4432
43.9.17. Autosave Case During Mesh Motion Preview Dialog Box 4434
43.10. Reference Values Task Page 4435
43.11. Solution Task Page 4437

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

43.12. Solution Methods Task Page 4438
43.12.1. Relaxation Options Dialog Box 4442
43.13. Solution Controls Task Page 4443
43.13.1. Equations Dialog Box 4446
43.13.2. Solution Limits Dialog Box 4447
43.13.3. Advanced Solution Controls Dialog Box 4448
43.14. Solution Initialization Task Page 4459
43.14.1. Acoustics Initialization Dialog Box 4462
43.14.2. Patch Dialog Box 4463
43.14.3. Hybrid Initialization Dialog Box 4464
43.15. Calculation Activities Task Page 4466
43.15.1. Autosave Dialog Box 4468
43.15.2. Data File Quantities Dialog Box 4470
43.15.3. Automatic Export Dialog Box 4471
43.15.4. Automatic Particle History Data Export Dialog Box 4476
43.15.5. Execute Commands Dialog Box 4478
43.15.6. Define Macro Dialog Box 4479
43.15.7. Automatic Solution Initialization and Case Modification Dialog Box 4480
43.16. Run Calculation Task Page 4481
43.16.1. Case Check Dialog Box 4491
43.16.2. Adaptive Time Stepping Dialog Box 4492
43.16.3. Simulation Status Dialog Box 4493
43.16.4. Solution Steering Dialog Box 4494
43.16.5. Acoustic Sources FFT Dialog Box 4496
43.16.6. Acoustic Signals Dialog Box 4502
43.16.7. Sampling Options Dialog Box 4504
43.16.8. Zone-Specific Sampling Options Dialog Box 4505
43.17. Results Task Page 4506
43.18. Graphics and Animations Task Page 4507
43.18.1. Profile Options Dialog Box 4510
43.18.2. Vector Options Dialog Box 4511
43.18.3. Custom Vectors Dialog Box 4512
43.18.4. Vector Definitions Dialog Box 4512
43.18.5. Path Style Attributes Dialog Box 4514
43.18.6. Ribbon Attributes Dialog Box 4514
43.18.7. Particle Filter Attributes Dialog Box 4515
43.18.8. Reporting Variables Dialog Box 4516
43.18.9. Track Style Attributes Dialog Box 4517
43.18.10. Particle Sphere Style Attributes Dialog Box 4518
43.18.11. Particle Vector Style Attributes Dialog Box 4519
43.18.12. Sweep Surface Dialog Box 4521
43.18.13. Create Surface Dialog Box 4522
43.18.14. Animate Dialog Box 4522
43.18.15. Save Picture Dialog Box 4525
43.18.16. Playback Dialog Box 4528
43.18.17. Video Options Dialog Box 4531
43.18.18. Advanced Video Quality Options Dialog Box 4532
43.18.19. Display Options Dialog Box 4533
43.18.20. Scene Description Dialog Box 4536
43.18.21. Display Properties Dialog Box 4538
43.18.22. Transformations Dialog Box 4540

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

43.18.23. Iso-Value Dialog Box 4541
43.18.24. Pathline Attributes Dialog Box 4542
43.18.25. Bounding Frame Dialog Box 4542
43.18.26. Views Dialog Box 4543
43.18.27. Write Views Dialog Box 4545
43.18.28. Mirror Planes Dialog Box 4545
43.18.29. Graphics Periodicity Dialog Box 4546
43.18.30. Camera Parameters Dialog Box 4548
43.18.31. Lights Dialog Box 4549
43.18.32. Colormap Dialog Box 4551
43.18.33. Colormap Editor Dialog Box 4553
43.18.34. Annotate Dialog Box 4555
43.19. Plots Task Page 4557
43.19.1. Solution XY Plot Dialog Box 4559
43.19.2. Histogram Dialog Box 4563
43.19.3. Plot Data Sources Dialog Box 4565
43.19.4. Plot Profile Data Dialog Box 4567
43.19.5. Plot Interpolated Data Dialog Box 4568
43.19.6. Fourier Transform Dialog Box 4570
43.19.7. VRXperience Sound Analysis Dialog Box 4572
43.19.8. Cumulative Plot Dialog Box 4574
43.19.9. Plot/Modify Input Signal Dialog Box 4576
43.19.10. Axes Dialog Box 4580
43.19.11. Curves Dialog Box 4583
43.20. Reports Task Page 4585
43.20.1. Flux Reports Dialog Box 4587
43.20.2. Force Reports Dialog Box 4588
43.20.3. Projected Surface Areas Dialog Box 4590
43.20.4. Surface Integrals Dialog Box 4591
43.20.5. Volume Integrals Dialog Box 4596
43.20.6. Sample Trajectories Dialog Box 4598
43.20.7. Trajectory Sample Histograms Dialog Box 4600
43.20.8. Particle Summary Dialog Box 4603
43.20.9. Heat Exchanger Report Dialog Box 4604
43.20.10. Parameters Dialog Box 4605
43.20.11. Use Input Parameter in Scheme Procedure Dialog Box 4608
43.20.12. Use Input Parameter for UDF Dialog Box 4609
43.20.13. Rename Dialog Box 4610
43.20.14. Parameter Expression Dialog Box 4610
43.20.15. Save Output Parameter Dialog Box 4612
43.21. Parameters and Customization Task Page 4613
44. Ribbon Reference Guide 4615
44.1. File Ribbon Tab 4615
44.1.1. File/Read/Mesh... 4617
44.1.1.1. Read Mesh Options Dialog Box 4617
44.1.2. File/Read/Case... 4618
44.1.3. File/Read/Data... 4618
44.1.4. File/Read/Case & Data... 4618
44.1.5. File/Read/PDF... 4619
44.1.6. File/Read/ISAT Table... 4619
44.1.7. File/Read/DTRM Rays... 4619

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

44.1.8. File/Read/View Factors... 4619
44.1.9. File/Read/Profile... 4619
44.1.10. File/Read/Scheme... 4619
44.1.11. File/Read/Journal... 4619
44.1.12. File/Write/Case... 4619
44.1.13. File/Write/Data... 4620
44.1.14. File/Write/Case & Data... 4620
44.1.15. File/Write/PDF... 4620
44.1.16. File/Write/ISAT Table... 4621
44.1.17. File/Write/Flamelet... 4621
44.1.18. File/Write/Profile... 4621
44.1.19. File/Write/Autosave... 4621
44.1.20. File/Write/Boundary Mesh... 4621
44.1.21. File/Write/Start Journal... 4621
44.1.22. File/Write/Stop Journal 4621
44.1.23. File/Write/Start Transcript... 4621
44.1.24. File/Write/Stop Transcript 4622
44.1.25. File/Import/ABAQUS/Input File... 4622
44.1.26. File/Import/ABAQUS/Filbin File... 4622
44.1.27. File/Import/ABAQUS/ODB File... 4622
44.1.28. File/Import/CFX/Definition File... 4622
44.1.29. File/Import/CFX/Result File... 4622
44.1.30. File/Import/CGNS/Mesh... 4622
44.1.31. File/Import/CGNS/Data... 4622
44.1.32. File/Import/CGNS/Mesh & Data... 4622
44.1.33. File/Import/EnSight... 4623
44.1.34. File/Import/FIDAP... 4623
44.1.35. File/Import/GAMBIT... 4623
44.1.36. File/Import/HYPERMESH ASCII... 4623
44.1.37. File/Import/I-deas Universal... 4623
44.1.38. File/Import/LSTC/Input File... 4623
44.1.39. File/Import/LSTC/State File... 4623
44.1.40. File/Import/Marc POST... 4623
44.1.41. File/Import/Mechanical APDL/Input File... 4623
44.1.42. File/Import/Mechanical APDL/Result File... 4624
44.1.43. File/Import/NASTRAN/Bulkdata File... 4624
44.1.44. File/Import/NASTRAN/Op2 File... 4624
44.1.45. File/Import/PATRAN/Neutral File... 4624
44.1.46. File/Import/PLOT3D/Grid File... 4624
44.1.47. File/Import/PLOT3D/Result File... 4624
44.1.48. File/Import/PTC Mechanica Design... 4624
44.1.49. File/Import/Tecplot... 4624
44.1.50. File/Import/Fluent 4 Case File... 4625
44.1.51. File/Import/PreBFC File... 4625
44.1.52. File/Import/Partition/Metis... 4625
44.1.53. File/Import/Partition/Metis Zone... 4625
44.1.54. File/Import/CHEMKIN Mechanism... 4625
44.1.54.1. Import CHEMKIN Format Mechanism Dialog Box 4625
44.1.55. File/Import/FMU... 4627
44.1.56. File/Export/Solution Data... 4627
44.1.56.1. Export Dialog Box 4628

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

44.1.57. File/Export/Particle History Data... 4633
44.1.57.1. Export Particle History Data Dialog Box 4633
44.1.58. File/Export/During Calculation/Solution Data... 4635
44.1.59. File/Export/During Calculation/Particle History Data... 4635
44.1.60. File/Export to CFD-Post... 4635
44.1.60.1. Export to CFD-Post Dialog Box 4635
44.1.61. File/Table File Manager... 4636
44.1.62. File/Solution Files... 4636
44.1.62.1. Solution Files Dialog Box 4636
44.1.63. File/Interpolate... 4637
44.1.63.1. Interpolate Data Dialog Box 4638
44.1.64. File/FSI Mapping/Volume... 4639
44.1.64.1. Volume FSI Mapping Dialog Box 4639
44.1.65. File/FSI Mapping/Surface... 4642
44.1.65.1. Surface FSI Mapping Dialog Box 4642
44.1.66. File/Save Picture... 4646
44.1.67. File/Data File Quantities... 4646
44.1.68. File/Batch Options... 4646
44.1.68.1. Batch Options Dialog Box 4646
44.1.69. File/Idle Timeout... 4647
44.1.69.1. Set Idle Timeout Dialog Box 4647
44.1.70. File/Exit 4648
44.2. Dialog Boxes Available from the Ribbon 4648
44.2.1. 1D Simulation Library Dialog Box 4651
44.2.2. Activate Cell Zones Dialog Box 4652
44.2.3. Adaption Criteria Settings Dialog Box 4653
44.2.4. Adjacency Dialog Box 4655
44.2.5. Advanced Options Dialog Box 4657
44.2.6. Aerodamping Report Definition Dialog Box 4658
44.2.7. Animation Definition Dialog Box 4660
44.2.8. Anisotropic Adaption Dialog Box 4663
44.2.9. Application About to Exit Dialog Box 4664
44.2.10. Auto Partition Mesh Dialog Box 4665
44.2.11. Automatic Mesh Adaption Dialog Box 4666
44.2.12. Cell Register Display Options Dialog Box 4667
44.2.13. Compiled UDFs Dialog Box 4669
44.2.14. Conduction Layers Dialog Box 4670
44.2.15. Conduction Manager Dialog Box 4671
44.2.16. Contours Dialog Box 4673
44.2.17. Convergence Conditions Dialog Box 4677
44.2.18. Create/Edit Mesh Interfaces Dialog Box 4678
44.2.19. Create/Edit Turbo Interfaces Dialog Box 4682
44.2.20. Custom Field Function Calculator Dialog Box 4684
44.2.21. Custom Laws Dialog Box 4686
44.2.22. Deactivate Cell Zones Dialog Box 4687
44.2.23. Define Control Points Dialog Box 4687
44.2.24. Delete Cell Zones Dialog Box 4689
44.2.25. Display Options - Adaption Dialog Box 4690
44.2.26. Display States Dialog Box 4692
44.2.27. DPM Report Definition Dialog Box 4695
44.2.28. DPM Source Report Definition Dialog Box 4699

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

44.2.29. Drag Report Definition Dialog Box 4700
44.2.30. DTRM Graphics Dialog Box 4703
44.2.31. DTRM Rays Dialog Box 4705
44.2.32. Edit Gap Region Dialog Box 4706
44.2.33. Edit Mesh Interfaces Dialog Box 4707
44.2.34. Edit Report File Dialog Box 4710
44.2.35. Edit Report Plot Dialog Box 4711
44.2.36. Execute on Demand Dialog Box 4713
44.2.37. Expression Dialog Box 4714
44.2.38. Expression Editor Dialog Box 4718
44.2.39. Expression Manager Dialog Box 4720
44.2.40. Expression Report Definition Dialog Box 4721
44.2.41. Field Function Definitions Dialog Box 4724
44.2.42. Flux Report Definition Dialog Box 4725
44.2.43. Force Report Definition Dialog Box 4728
44.2.44. Fuse Face Zones Dialog Box 4731
44.2.45. Gap Model Dialog Box 4732
44.2.46. General Adaption Controls Dialog Box 4734
44.2.47. Geometry Based Adaption Controls Dialog Box 4735
44.2.48. Geometry Based Adaption Dialog Box 4736
44.2.49. Import Particle Data Dialog Box 4737
44.2.50. Imprint Surface Dialog Box 4739
44.2.51. Improve Mesh Dialog Box 4740
44.2.52. Injections Dialog Box 4741
44.2.53. Input Summary Dialog Box 4742
44.2.54. Interface Creation Options Dialog Box 4743
44.2.55. Interpreted UDFs Dialog Box 4744
44.2.56. Iso-Clip Dialog Box 4745
44.2.57. Iso-Surface Dialog Box 4746
44.2.58. Lift Report Definition Dialog Box 4748
44.2.59. Line/Rake Surface Dialog Box 4751
44.2.60. Manual Mesh Adaption Dialog Box 4752
44.2.61. Manage Adaption Criteria Dialog Box 4754
44.2.62. Mapped Interface Options Dialog Box 4755
44.2.63. Merge Zones Dialog Box 4756
44.2.64. Mesh Interfaces Dialog Box 4757
44.2.65. Mesh Morpher/Optimizer Dialog Box 4760
44.2.66. Mixing Planes Dialog Box 4770
44.2.67. Moment Report Definition Dialog Box 4773
44.2.68. Motion Settings Dialog Box 4775
44.2.69. Multi Edit Dialog Box 4781
44.2.70. New Report File Dialog Box 4782
44.2.71. New Report Plot Dialog Box 4784
44.2.72. Objective Function Definition Dialog Box 4786
44.2.73. Optimization History Monitor Dialog Box 4788
44.2.74. Parallel Connectivity Dialog Box 4790
44.2.75. Parameter Bounds Dialog Box 4790
44.2.76. Particle Tracks Dialog Box 4791
44.2.77. Partition Surface Dialog Box 4798
44.2.78. Partitioning and Load Balancing Dialog Box 4799
44.2.79. Pathlines Dialog Box 4804

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

44.2.80. Plane Surface Dialog Box 4810
44.2.81. Point Surface Dialog Box 4812
44.2.82. Quadric Surface Dialog Box 4813
44.2.83. Reduced Order Model Dialog Box 4815
44.2.84. Reference Frame Dialog Box 4817
44.2.85. Replace Cell Zone Dialog Box 4818
44.2.86. Report Definitions Dialog Box 4819
44.2.87. Report File Definitions Dialog Box 4821
44.2.88. Report Plot Definitions Dialog Box 4822
44.2.89. Residual Monitors Dialog Box 4824
44.2.90. Rotate Mesh Dialog Box 4827
44.2.91. S2S Information Dialog Box 4828
44.2.92. Select Window Dialog Box 4829
44.2.93. Separate Cell Zones Dialog Box 4830
44.2.94. Separate Face Zones Dialog Box 4831
44.2.95. Set Injection Properties Dialog Box 4832
44.2.96. Set Multiple Injection Properties Dialog Box 4845
44.2.97. Structural Point Surface Dialog Box 4846
44.2.98. Surface Meshes Dialog Box 4847
44.2.99. Surface Report Definition Dialog Box 4848
44.2.100. Surfaces Dialog Box 4851
44.2.101. Thread Control Dialog Box 4853
44.2.102. Transform Surface Dialog Box 4854
44.2.103. Translate Mesh Dialog Box 4855
44.2.104. Turbo 2D Contours Dialog Box 4856
44.2.105. Turbo Averaged Contours Dialog Box 4858
44.2.106. Turbo Averaged XY Plot Dialog Box 4860
44.2.107. Turbo Options Dialog Box 4862
44.2.108. Turbo Report Dialog Box 4862
44.2.109. Turbo Topology Dialog Box 4865
44.2.110. UDF Library Manager Dialog Box 4867
44.2.111. User-Defined Fan Model Dialog Box 4868
44.2.112. User-Defined Function Hooks Dialog Box 4869
44.2.113. User-Defined Memory Dialog Box 4873
44.2.114. User Defined Report Definition Dialog Box 4873
44.2.115. User-Defined Scalars Dialog Box 4875
44.2.116. Vectors Dialog Box 4876
44.2.117. Volume Report Definition Dialog Box 4880
44.2.118. Warning Dialog Box 4883
44.2.119. Zone Surface Dialog Box 4884
A. Ansys Fluent Model Compatibility 4887
B. Ansys Fluent File Formats 4891
B.1. CFF File Format 4891
B.1.1. CFF Case File Layout 4891
B.1.2. CFF Solution Data File Layout 4893
B.1.3. Variable Sized Data on Collected Element / Node Sets 4894
B.2. Legacy Case and Data File Formats 4895
B.2.1. Guidelines 4895
B.2.2. Formatting Conventions in Binary and Formatted Files 4896
B.2.3. Grid Sections 4896
B.2.3.1. Comment 4896

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B.2.3.2. Header 4897
B.2.3.3. Dimensions 4897
B.2.3.4. Nodes 4897
B.2.3.5. Periodic Shadow Faces 4898
B.2.3.6. Cells 4899
B.2.3.7. Faces 4901
B.2.3.8. Face Tree 4903
B.2.3.9. Cell Tree 4903
B.2.3.10. Interface Face Parents 4904
B.2.3.11. Example Files 4904
B.2.3.11.1. Example 1 4904
B.2.3.11.2. Example 2 4905
B.2.3.11.3. Example 3 4906
B.2.4. Other (Non-Grid) Legacy Case Sections 4908
B.2.4.1. Zone 4908
B.2.4.2. Partitions 4910
B.2.5. Data Sections 4911
B.2.5.1. Grid Size 4911
B.2.5.2. Data Field 4911
B.2.5.3. Residuals 4912
B.3. Mesh Morpher/Optimizer File Formats 4913
B.4. Conduction Settings File Format 4914
B.5. 3D Fan Curve File Format 4914
C. Controlling CHEMKIN-CFD Solver Parameters Using Text Commands 4917
C.1. Advanced Parameters Used in the Steady-State Solution Algorithm 4921
C.2. Setting Up Monitor Cells for the Ansys CHEMKIN-CFD Chemistry Solver 4922
C.3. Diagnostic Files and Error Messages 4922
C.4. Error Messages Printed in the Ansys Fluent Graphical User Interface 4924
C.5. Diagnostic Messages in the `KINetics-log.txt` File 4926
D. Nomenclature 4929
Bibliography 4933
IV. Workspaces 4943
1. Remote Visualization and Accessing Fluent Remotely 4945
1.1. Starting Remote Visualization 4945
1.1.1. Steps for Starting the Server 4945
1.1.1.1. Port Management 4946
1.1.2. Steps For Starting the Remote Visualization Client 4947
1.2. Using a Job Scheduler with Remote Visualization 4948
1.3. Operating in the Fluent Remote Visualization Environment 4948
1.3.1. Adding New Remote Client Connections 4948
1.3.2. Setting Preferences 4949
1.3.3. Initializing, Starting, Pausing, and Interrupting the Calculation 4950
1.3.4. Modifying Solution Settings 4952
1.3.5. Graphics Window Interactions and Context Menus 4953
1.3.6. Surfaces 4954
1.3.6.1. Point Surfaces 4955
1.3.6.2. Line Surfaces 4956
1.3.6.3. Rake Surfaces 4957
1.3.6.4. Plane Surfaces 4958
1.3.6.5. Iso-Surfaces 4962
1.3.7. Graphics Objects 4966

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.3.7.1. Creating and Displaying Graphics objects 4967
1.3.7.2. Creating and Displaying Plot objects 4968
1.3.7.3. Creating and Displaying Scenes 4968
1.3.7.4. Saving Pictures of the Graphics Window 4969
1.3.7.5. Modifying the Views 4970
1.3.7.6. Synchronizing with the Server 4970
1.3.8. Messaging and Text Commands 4971
1.3.9. Saving Case and Data Files 4972
1.3.10. Disconnecting the Server and Client 4973
1.3.10.1. Disconnecting from Within the Remote Client Session 4973
1.3.10.2. Disconnecting from Within the Remote Server Session 4974
1.3.11. Modifying Preferences 4974
1.4. Python, Scripting and Transcripts in the Remote Client 4974
1.4.1. Python Scripting 4975
1.4.2. Starting and Stopping a Transcript 4977
1.5. Remote Visualization Best Practices 4977
1.6. Remote Visualization Client Environment Variables 4978
1.7. Limitations 4979
2. Fluent Icing 4981
2.1. Overview of Fluent Icing 4981
2.2. Known Limitations in Fluent Icing 2021 R2 4982
2.3. Quick Start 4983
2.4. Starting Fluent Icing 4985
2.5. Fluent Icing Graphical User Interface Layout 4987
2.6. Creating or Opening a Fluent Icing Project 4992
2.6.1. Creating a Fluent Icing Project 4993
2.6.2. Opening a Fluent Icing Project 4994
2.6.3. Project Library 4995
2.6.4. Project Close 4996
2.7. Creating or Loading a Fluent Icing Simulation 4996
2.7.1. Case File Requirements 4997
2.7.2. Fluent Solver and License Requirements 4997
2.7.3. Creating a New Simulation by Importing Loading a Case File 4997
2.7.4. Loading a Simulation 4999
2.7.5. Use Custom Solver Launch Settings to Load in Solver 5000
2.7.6. Disconnecting from a Simulation 5002
2.7.7. Duplicating a Simulation 5003
2.7.8. Loading Multiple Simulations 5004
2.8. Setting-Up a Fluent Icing Simulation 5006
2.8.1. Setup 5007
2.8.1.1. Airflow 5009
2.8.1.2. Particles 5014
2.8.1.3. Ice 5021
2.8.2. Boundary Conditions 5023
2.8.2.1. Inlets 5024
2.8.2.2. Walls 5033
2.8.2.3. Outlets 5035
2.8.3. Solution 5036
2.8.3.1. Airflow 5037
2.8.3.2. Particles 5042
2.8.3.3. Ice 5048

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.8.3.4. Multi-Shot 5050
2.8.4. Results 5053
2.8.4.1. Quick-View 5058
2.9. Using the Project View to Interact with Fluent Icing Simulations 5061
2.9.1. Simulation Folder Commands 5061
2.9.2. Run Folder Commands 5064
2.9.3. Case File Commands 5065
2.9.4. Solution File Commands 5067
2.9.5. The Use of Bold in Project View 5068
2.9.6. Project View Organization Options 5068
2.10. Post Processing with Viewmerical and CFD-Post from Fluent Icing 5073
2.10.1. Viewmerical 5074
2.10.1.1. Accessing Viewmerical from Quick-View 5074
2.10.1.2. Accessing Viewmerical from Project View 5074
2.10.1.3. Comparing Multiple Solutions with Viewmerical 5075
2.10.2. CFD-Post 5075
2.10.2.1. Accessing CFD-Post from Quick-View 5076
2.10.2.2. Accessing CFD-Post from Project View 5076
2.10.3. EnSight 5076
2.11. Preferences 5077
2.12. Advanced Settings 5078
2.13. File Types 5079
2.14. Appendix 5081
2.14.1. Python Console 5081
2.14.2. Data Structures 5083
2.14.3. Project API 5084
2.14.4. Fluent Journal Commands 5085

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

List of Figures

1.1. Ansys Fluent Architecture 130
4.1. The General Options Tab of Fluent Launcher 167
4.2. The Parallel Settings Tab of Fluent Launcher 169
4.3. The Remote Tab of Fluent Launcher 170
4.4. The Scheduler Tab of Fluent Launcher (Windows 64 Version) 171
4.5. The Environment Tab of Fluent Launcher 172
4.6. The Batch Options Dialog Box 187
8. Cell Types 194
3.1. The User Interface Components 205
3.2. Clipping Plane and Clipping Plane Tool 208
3.3. The Watertight Geometry Workflow 213
3.4. The Outline View Tree 213
3.5. Model Level Menu 214
3.6. CAD Assemblies Tree 214
3.7. CAD Assemblies Menu 215
3.8. CAD Component/Body Level Menu 215
3.9. CAD Label Level Menu 216
3.10. Global Object Level Menu 216
3.11. Individual Object Level Menu 217
3.12. Face Zone Labels Level Menu 218
3.13. Individual Label Menu 219
3.14. Unreferenced Zones Menu 219
3.15. The Graphics Effects Tools 224
3.16. The Mesh Display Tools 225
3.17. The Copy Tools 226
3.18. The Object Selection/Display Tools 226
3.19. Preferences Dialog Box 230
5.1. Splitting the Face of a Coiled Geometry 255
5.2. Imported Coiled Geometry 256
6.1. The Dockable Workflow Editor 271
6.2. Setting the Default Workflow Editor View 271
6.3. Messages and Progress Bar During Meshing Tasks 272
6.4. Example of a Self-Intersection: Double Faces Appear When Share Topology is Not Enabled 286
6.5. Example of a Self-Intersection: Local Mesh Size is Significantly Larger Than the Pipe Thickness 286
6.6. Showing Marked Gaps 294
6.7. Applying Share Topology to Marked Gaps 295
6.8. Incomplete Gap Marking (Max Gap Distance = 0.12) 295
6.9. Complete Gap Marking (Max Gap Distance = 0.15) 295
6.10. Excessive Gap Marking Around a Washer (Max Gap Distance = 1.2) 296
6.11. Proper Gap Marking Around a Washer (Max Gap Distance = 0.8) 296
6.12. Example of a Single Surface Cap with Multiple Faces 297
6.13. Example of a Single Surface Cap with Multiple Faces 297
6.14. Example of an Annular Cap Type 298
6.15. Example of a Problematic Tilted Annular Opening 298
6.16. Example of a Self-Intersection: Additional Cap Intersects With Other Surfaces 299
6.17. Example of a Fluid and a Solid Volume Mesh 309
6.18. Example of a Copying and Rotating a Volume Mesh (Periodic) 313
6.19. Example of Copying and Translating a Volume Mesh (Periodic) 313
6.20. Example of a Geometry With Part Selected for Adding a Mesh Pattern 318

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.21. Example of a Geometry With a Preview of the Mesh Pattern 319
6.22. Example of a Geometry With a Mesh Pattern 320
6.23. An Example of a Custom Singular Pattern 324
6.24. An Example of a Custom Dual Pattern 325
6.25. An Example of a Refinement Region Around a Car 332
6.26. An Example of Multiple Refinement Regions Around a Car 335
6.27. An Example of Multiple Refinement Regions Around a Vehicle 336
6.28. A CAD File Loaded into the CAD Model Tree 342
6.29. Selected Portions of a CAD File Loaded into the Meshing Model Tree 344
6.30. CAD Properties of a Selected Per-Part Meshing Model Object 347
6.31. Properties of a Selected Custom Meshing Model Object 350
6.32. Translation Example 354
6.33. Rotation Example 354
6.34. Faceting Example 357
6.35. Example of a Single Surface Cap with Multiple Faces 362
6.36. Example of a Single Surface Cap with Multiple Faces 362
6.37. Example of an Annular Cap Type 363
6.38. Example of a Problematic Tilted Annular Opening 363
6.39. Example of a Self-Intersection: Additional Cap Intersects With Other Surfaces 364
6.40. An Example of an External Flow Boundary 365
6.41. An External Flow Bounding Box 366
6.42. An External Flow Boundary Around a Car 366
6.43. An Example of a Refinement Region Around a Car 367
6.44. An Example of Multiple Refinement Regions Around a Car 370
6.45. An Example of Multiple Refinement Regions Around a Vehicle 370
6.46. An Example of Addressing Sharp Angles - CAD Geometry 376
6.47. An Example of Addressing Sharp Angles - Final Mesh 376
6.48. Sharp Angles With and Without the Zones Separated By Face 377
6.49. An Example of a Porous Region 380
6.50. An Example of a Porous Region: Fins and Tubes in a Heat Exchanger 380
6.51. The Points of a Porous Region 381
6.52. The Buffer Size for a Porous Region 382
6.53. Identifying a Fluid Region in the Wake Behind a Car 383
6.54. Identifying Potential Leakages Within a Car 386
6.55. Collar and Component Meshes for a Propeller and Hub Geometry 396
6.56. A Collar Mesh for a Propeller and Hub Geometry 397
6.57. A Component Mesh for a Propeller and Hub Geometry 398
6.58. Example of Surface Mesh Quality Contours 403
6.59. Example of Orphan Cells in an Offset Mesh 406
8.1. Use of Curvature Sizing 423
8.2. Use of Proximity Sizing 424
8.3. Use of the Face Boundary Option for Face Proximity 425
8.4. Use of the Ignore Orientation Option for Face Proximity 426
8.5. Use of Meshed Sizing 427
8.6. Use of Soft Sizing 428
8.7. Use of Body of Influence Sizing 429
8.8. Contours of Size 434
8.9. Display of Mesh Size Based on Size Field 434
9.1. Mesh With Different Cell Zone Types 439
9.2. Use of the Object Priority for Overlapping Objects 440
9.3. Creating Objects—Example 440

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.4. Objects Defined Using the Subtract Method 441
9.5. Using Material Points—Example 450
9.6. Example—CutCell Mesh, Only Objects Defined 451
9.7. Example—Fluid Surface Extracted From Geometry Objects and Material Point 452
10.1. Closing a Radial Gap 458
10.2. Creating a Surface Using an Edge 460
10.3. Creating a Surface Using Nodes 461
10.4. Overlapping Surfaces 468
10.5. Connected Surfaces After Join 468
10.6. Intersecting Surfaces 469
10.7. Connected Surfaces After Intersect 469
10.8. Orientation of Normals in Gap 474
10.9. Removing Gaps Between Objects—Face-Face Option 475
10.10. Removing Gaps Between Objects—Face-Edge Option 475
10.11. Gap and Thickness Configurations 476
10.12. Removing Thickness in Objects 477
10.13. Mesh Objects to be Connected 478
10.14. Mesh Object Created by Sewing 478
12.1. Free Nodes 494
12.2. Example of a Thin Wall 494
12.3. Intersection of Boundary Zones 495
12.4. Intersection (A) Without and (B) With the Refine Option 496
12.5. Partially Overlapping Faces 496
12.6. Joining of Overlapping Faces 497
12.7. Remeshing of Joined Faces 497
12.8. Nearest Point Projection for Stitching 498
12.9. Surfaces Before Stitch 499
12.10. Surfaces After Stitch 499
12.11. Refining a Triangular Boundary Face 511
12.12. Boundary Mesh (A) Before and (B) After Refining Based on Proximity 511
12.13. Surface Mesh - Feature Angle = 60 514
12.14. Edge Zone for Face Zone Approach and Fixed Angle = 65 514
12.15. Edge Zones for Face Zone Approach and Fixed Angle = 55 (or Adaptive Angle) 515
12.16. Edge Zone for Face Seed Approach and Fixed Angle = 65 515
12.17. Edge Zones for Face Seed Approach and Fixed Angle = 55 (or Adaptive Angle) 516
12.18. Mesh (A) Before and (B) After Using the Faceted Stitch Option 523
12.19. Triangulating a Boundary Zone 524
12.20. Face Separation Based on Region 526
12.21. Face Separation Based on Cell Neighbor 526
12.22. Planar Points Method 534
12.23. Cylinder Defined by 3 Arc Nodes, Radial Gap, and Axial Delta 535
12.24. Cylinder Defined by 3 Arc Nodes and a Height Node 536
12.25. Cylinder Defined by Axial Points and Radii 537
12.26. Loop Selection Toolbar 544
13.1. Schematic Representation of Wrapping Process 549
13.2. Individual Object Loop 552
13.3. Collective Object Loops 552
13.4. Overlaid Geometry Clipped with the Pan Plane 555
13.5. Leak Detection Using the Pan Regions Dialog Box 556
13.6. Wrapping Individual Objects 558
13.7. Multiple Solids 559

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.8. Single Solid Surface 559
13.9. Extracting the Flow Volume 560
14.1. Possible Mesh Cell Shapes 566
14.2. Mesh with Prisms in a Boundary Layer Region 567
14.3. Surface Mesh Containing Only Tetrahedra 567
14.4. Surface Mesh 568
14.5. Hexcore Mesh 569
14.6. CutCell Mesh 570
14.7. Rapid Octree Mesh with Projection Boundary Treatment 570
14.8. Extending an Existing Tetrahedral Mesh Using Prisms 571
14.9. Example of a Non-Conformal Interface 572
14.10. Mesh Generated Using Isolated Nodes to Concentrate Cells 574
14.11. Mesh Generated Without Using Isolated Nodes 574
14.12. Pyramid Cell—Transition from a Hexahedron to a Tetrahedron 579
14.13. Pyramid Cells Intersecting Each Other and Boundary 582
14.14. Fixed Intersecting Pyramid Cells Using Triangular Faces 583
14.15. Creating the Heat Exchanger Mesh 585
14.16. The Thread Control Dialog Box 591
15.1. Prism Shapes 593
15.2. Layer Heights Computed Using the Four Growth Methods 597
15.3. Different Growth Parameters on Adjacent Zones 601
15.4. Different Growth Parameters on Nonadjacent Zones—Using the Auto Mesh Option 602
15.5. Prism Growth on a Dangling Wall 603
15.6. Ignoring Invalid Normals 604
15.7. Collision of Prism Layers 605
15.8. Prism Layers Shrunk to Avoid Collision 605
15.9. Ignoring Areas of Proximity 606
15.10. Uniform Offset Distance Method 608
15.11. Minimum-Height Offset Distance Method 609
15.12. Last Ratio Method 610
15.13. Effect of Offset Smoothing 611
15.14. Uniform Direction Vector for a Straight-Sided Prism Region 611
15.15. Normal Direction Vectors for a Curved Prism Region 612
15.16. Normal Direction Vectors Before Smoothing 612
15.17. Normal Direction Vectors After Smoothing 613
15.18. Effect of Adjacent Zone Angle 614
15.19. Symmetry Zone and Car Wall Before Prism Generation 615
15.20. Symmetry Zone and Car Wall After Prism Generation Without Retriangulation 615
15.21. Symmetry Zone and Car Wall After Prism Generation and Retriangulation 616
15.22. Use of Multiple Scoped Prism Controls 620
15.23. Stair Stepped Prism Layers in Sharp Corner 621
16.1. Local Refinement Region for the Tetrahedral Mesh 636
17.1. Hexcore Mesh Using (A) Buffer Layers = 1 (B) Buffer Layers = 2 645
17.2. Hexcore Mesh Using (A) Peel Layers = 0 (B) Peel Layers = 2 645
17.3. Hexcore to the Far-Field Boundary 647
17.4. Hexcore to Boundaries 648
17.5. Local Refinement Region for the Hexcore Mesh 649
20.1. Schematic Representation of the Cartesian Grid Refinement Using Size Functions 662
20.2. Mesh After Refinement 663
20.3. Mesh After Projection 663
20.4. Cells Separated After Decomposition 664

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.5. CutCell Mesh After Boundary Recovery 665
20.6. Mesh Generated for Geometry Having Zero-Thickness Baffles 668
20.7. Recovering Overlapping Surfaces 669
20.8. Resolving Thin Regions 670
20.9. Rezoning Multiply Connected Faces 671
20.10. Generating Prisms for the CutCell Mesh 674
20.11. Prism Growth Limitations—Volumes Sharing an Edge 674
20.12. Prism Growth Limitations—Volumes Sharing an Edge 675
20.13. Prism Growth Limitations—Volumes Sharing the Prism Base 675
21.1. The Rapid Octree Dialog Box 680
21.2. The Geometry Group Box 681
21.3. Bounding Box Dialog Box 683
21.4. Bounding Box Surrounding Geometry 684
21.5. The Boundary Treatment List 685
21.6. Mesh Generated by Cartesian Snapping 685
21.7. Mesh Generated by Boundary Projection 686
21.8. The Mesh Parameters Group Box 687
21.9. Surface Mesh with Two Levels of Feature Angle Refinement and 10° Threshold. 688
21.10. The Surface Sizing Dialog Box 689
21.11. The Rapid Octree Refinement Region Dialog Box 690
21.12. Defeaturing Considerations 691
22.1. 2–3 and 3–2 Swap Configurations 695
22.2. 4–4 Swap Configuration 696
22.3. Sliver Formation 697
22.4. Movement of Boundary Nodes 703
22.5. Cavity Around a Mirror Remeshed With Tetrahedra 706
22.6. Cavity Around a Mirror Remeshed With Hexcore Mesh 709
22.7. Copying and Translating a Cell Zone 710
22.8. The Selective Mesh Check Dialog 714
23.1. Mesh Display (A) With Shrink Factor = 0 (B) With Shrink Factor = 0.01 722
23.2. Camera Definition 728
23.3. Graphics Display with Bounding Frame 729
23.4. The Navigation Branch of Preferences 733
24.1. Ideal and Skewed Triangles and Quadrilaterals 743
24.2. Vectors Used to Compute Ortho Skew/Inverse Orthogonal Quality for a Cell 747
24.3. Vectors Used to Compute Ortho Skew Quality for a Face 749
24.4. Calculating the Fluent Aspect Ratio for a Unit Cube 750
1. Quadrilateral Mesh 773
2. Quadrilateral Mesh with Periodic Boundaries 774
3. Quadrilateral Mesh with Hanging Nodes 775
1.1. The GUI Components 802
1.2. The Fluent Ribbon 803
1.3. The Fluent Outline View 804
1.4. Hover-Over Highlight 806
1.5. Graphics Window Context Menu: Single-Selection 807
1.6. Graphics Window Context Menu: Multiple-Selection 808
1.7. Displaying Two Graphics Windows 809
1.8. The View Ribbon Tab 811
1.9. The Standard Toolbar 811
1.10. The Graphics Toolbars 811
1.11. Mesh Display 812

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.12. The Pointer Tools 812
1.13. The View Tools 813
1.14. The Visibility Tools 814
1.15. The Copy Tools 815
1.16. The Object Selection/Display Tools 815
1.17. The Graphics Effects Tools 816
1.18. Additional Display Options 816
1.19. The Select File Dialog Box for Windows 827
1.20. The Select File Dialog Box for Linux Platforms 828
1.21. Another Version of the Select File Dialog Box for Linux Platforms 828
1.22. Quick Editor for a Velocity Inlet 830
1.23. Fluent in Dark Theme 831
1.24. Component Dock 832
1.25. Preferences Dialog Box 833
1.26. Fluent GUI in Japanese 835
1.27. Fluent GUI in Korean 835
1.28. Fluent GUI in Chinese 836
1.29. Set Idle Timeout Dialog Box 837
1.30. Application About to Exit Dialog Box 838
3.1. The Select File Dialog Box 845
3.2. The Autosave Dialog Box 855
3.3. The Write Profile Dialog Box 859
3.4. Multiple Selection of Journal Files 864
3.5. The Import Menu 867
3.6. The Export Dialog Box 878
3.7. The Export Particle History Data Dialog Box 893
3.8. The Calculation Activities Task Page 895
3.9. The Automatic Export Dialog Box 896
3.10. The Automatic Particle History Data Export Dialog Box 898
3.11. The Export to CFD-Post Dialog Box 901
3.12. The Solution Files Dialog Box 903
3.13. The Interpolate Data Dialog Box 904
3.14. The Volume FSI Mapping Dialog Box for Cell Zone Data 909
3.15. The Surface FSI Mapping Dialog Box for Face Zone Data 909
3.16. The Save Picture Dialog Box 914
3.17. The Data File Quantities Dialog Box 921
3.18. **Import FMU File** Dialog Box 924
3.19. **Import FMU File** Dialog Box 925
4.1. The Set Units Dialog Box 928
4.2. The Define Unit Dialog Box 930
5.1. Example Profile Expression 942
5.2. The Expression Editor Dialog Box 945
5.3. Example Expression for Water Density 947
5.4. The Expression Dialog Box 948
5.5. Plotting an Expression 952
5.6. Expressions... Postprocessing Field 953
5.7. Expression Manager Dialog Box 955
5.8. Contours of Velocity - Parabolic Inflow 956
5.9. Parabolic Inflow Velocity Over Time 960
5.10. Pipe Geometry Colored by ID (Heated Wall is Green) 963
5.11. Contours of Temperature (outlet is closest) 963

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.12. Plots of Inlet Temperature, Average Outlet Temperature, and Maximum Outlet Temperature 964
6.1. Cell Types 1016
6.2. Structured Quadrilateral Mesh for an Airfoil 1017
6.3. Unstructured Quadrilateral Mesh 1017
6.4. Multiblock Structured Quadrilateral Mesh 1018
6.5. O-Type Structured Quadrilateral Mesh 1018
6.6. Parachute Modeled With Zero-Thickness Wall 1018
6.7. C-Type Structured Quadrilateral Mesh 1019
6.8. 3D Multiblock Structured Mesh 1019
6.9. Unstructured Triangular Mesh for an Airfoil 1019
6.10. Unstructured Tetrahedral Mesh 1020
6.11. Hybrid Triangular/Quadrilateral Mesh with Hanging Nodes 1020
6.12. Non-Conformal Hybrid Mesh for a Rotor-Stator Geometry 1021
6.13. Polyhedral Mesh 1021
6.14. Face and Node Numbering for Triangular Cells 1022
6.15. Face and Node Numbering for Quadrilateral Cells 1023
6.16. Face and Node Numbering for Tetrahedral Cells 1024
6.17. Face and Node Numbering for Wedge Cells 1025
6.18. Face and Node Numbering for Pyramidal Cells 1026
6.19. Face and Node Numbering for Hex Cells 1027
6.20. An Example of a Polyhedral Cell 1028
6.21. Setup of Axisymmetric Geometries with the x Axis as the Centerline 1031
6.22. The Vectors Used to Compute Orthogonality 1032
6.23. Calculating the Aspect Ratio for a Unit Cube 1033
6.24. The Surface Meshes Dialog Box 1050
6.25. The Reference Frame Dialog Box 1051
6.26. The Motion Tab of the Reference Frame Dialog Box 1052
6.27. Point Surface Creation on Local Reference Frame 1053
6.28. Profile Definition on Local Reference Frame 1053
6.29. Completely Overlapping Mesh Interface Intersection 1054
6.30. Partially Overlapping Mesh Interface Intersection 1054
6.31. Two-Dimensional Non-Conformal Mesh Interface 1055
6.32. Non-Conformal Periodic Boundary Condition (Translational) 1057
6.33. Non-Conformal Periodic Boundary Condition (Rotational) 1058
6.34. Translational Non-Conformal Interface with the Periodic Repeats Option 1059
6.35. Rotational Non-Conformal Interface with the Periodic Repeats Option 1060
6.36. Non-Conformal Coupled Wall Interfaces 1061
6.37. Matching Non-Conformal Wall Interfaces 1062
6.38. Non-Conformal Mapped Interface with a Gap and Penetration 1063
6.39. A Circular Non-Conformal Interface 1068
6.40. The Mesh Interfaces Dialog Box for the One-to-One Method 1071
6.41. The Mesh Interfaces Dialog Box for the Many-to-Many Method 1072
6.42. The Interface Creation Options Dialog Box 1073
6.43. Managing One-to-One Interfaces from the Outline View Tree 1074
6.44. Managing One-to-One Interfaces from the Mesh Interfaces Dialog Box 1075
6.45. The Edit Mesh Interface Dialog Box 1075
6.46. Creating a Coupled Wall at a One-to-One Mesh Interface 1076
6.47. The Edit Mesh Interfaces Dialog Box 1077
6.48. Contours of Interface Overlap Fraction 1079
6.49. Displaying the Intersected Zone 1080
6.50. The Create/Edit Mesh Interfaces Dialog Box 1082

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.51. Transferring Displacements 1085
6.52. Projecting Nodes 1085
6.53. Overset Component and Background Mesh 1086
6.54. Solve Cells After Initialization 1087
6.55. Valid Overset Meshes with Components in Close Proximity 1088
6.56. Second Component Modifying Existing Body 1089
6.57. Existing Body Modification After Initialization 1089
6.58. Multiple Components Bridged by Collars Meshes 1090
6.59. Multiple Components with Collar Meshes Initialized 1090
6.60. Adding Fluid to a Region Using Cut Control 1091
6.61. Cut Controlled Region After Initialization 1091
6.62. Overset Component and Background Meshes Before Hole Cutting 1092
6.63. Overset Component and Background Meshes After Hole Cutting 1092
6.64. Overset Component and Background Meshes After Overlap Minimization 1093
6.65. Overset Mesh Before Hole Cutting 1094
6.66. Overset Mesh After Minimization Based on Boundary Distance 1094
6.67. Valid Overlap 1095
6.68. Invalid Overlap Creating Orphans 1097
6.69. Create/Edit Overset Interfaces Dialog Box 1105
6.70. Contours of Overset Cell Type: Background Mesh 1109
6.71. Contours of Overset Cell Type: Component Mesh 1109
6.72. The Gap Model Dialog Box 1114
6.73. The Edit Gap Region Dialog Box 1115
6.74. Controlling Gap Shape by Excluding Zones 1116
6.75. The Advanced Options Dialog Box 1117
6.76. The Mesh Display Dialog Box for a Gap Model Simulation 1118
6.77. The Solution Methods Task Page 1122
6.78. Connection of Edge Centroids with Face Centroids 1128
6.79. A Polyhedral Cell 1129
6.80. A Converted Polyhedral Cell with Preserved Hexahedral Cell Shape 1129
6.81. Treatment of Wedge Boundary Layers 1130
6.82. The Original Tetrahedral Mesh 1130
6.83. The Converted Polyhedral Mesh 1131
6.84. The Merge Zones Dialog Box 1135
6.85. The Separate Face Zones Dialog Box 1138
6.86. Cell Zone Separation Based on Region 1139
6.87. The Separate Cell Zones Dialog Box 1140
6.88. The Fuse Face Zones Dialog Box 1141
6.89. The Create Periodic Dialog Box 1145
6.90. The Replace Cell Zone Dialog Box 1150
6.91. The Delete Cell Zones Dialog Box 1151
6.92. The Deactivate Cell Zones Dialog Box 1152
6.93. The Activate Cell Zones Dialog Box 1153
6.94. The Select File Dialog Box 1154
6.95. The Adjacency Dialog Box 1156
6.96. The Scale Mesh Dialog Box 1160
6.97. The Translate Mesh Dialog Box 1162
6.98. The Rotate Mesh Dialog Box 1163
6.99. The Improve Mesh Dialog Box 1165
6.100. Result of Smoothing Operator on Node Position 1166
6.101. Initial Mesh Before Smoothing Operation 1167

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.102. Mesh Smoothing Causing Mesh-Line Crossing 1168
6.103. Examples of Cell Configurations in the Circle Test 1170
6.104. Swapped Faces to Satisfy the Delaunay Circle Test 1170
6.105. 3D Face Swapping 1171
7.1. The Boundary Conditions Task Page 1175
7.2. The Copy Conditions Dialog Box 1179
7.3. The Parameters Dialog Box 1181
7.4. The New Input Parameter... Selection 1183
7.5. The Parameter Expression Dialog Box 1184
7.6. Use Input Parameter in Scheme Procedure Dialog Box 1186
7.7. Use Input Parameter for UDF Dialog Box 1187
7.8. Selecting Multiple Boundaries for Display in the Graphics Window 1190
7.9. Example Operations for Multiple Selected Surfaces in the Graphics Window 1191
7.10. The Fluid Dialog Box 1194
7.11. Rotation Specified in the Absolute Reference Frame 1197
7.12. Rotation Specified Relative to a Moving Zone 1198
7.13. The Solid Dialog Box 1200
7.14. Single Rotating Solid Zone 1203
7.15. Rotating solid zone separated from another fluid or solid zone separated by a surface of revolution. 1203
7.16. Multiple rotating solid zones having the same material and motion specifications, separated by mesh interfaces or coupled walls. 1204
7.17. Two solids in contact where one is stationary and the other is moving with translational motion. The translational motion of the moving solid should be described in the Solid Motion tab. 1204
7.18. Two solids in contact where one is stationary and the other is rotating. The rotational motion of the moving solid should be described in the Solid Motion tab. Both solids may also have rotation about the y-axis described in the Frame Motion tab. 1205
7.19. Rotating solid with boundaries which are not tangential to the motion. 1205
7.20. Two solids in contact with some squish. At the contact, the rotational motion has some normal component, so the solver will not achieve global energy conservation. However, the temperature field might still be acceptable for engineering purposes. 1206
7.21. The Fluid Dialog Box for a Porous Zone 1216
7.22. Cone Half Angle 1220
7.23. The Heat Transfer Settings Group Box of the Fluid Dialog Box 1228
7.24. The Fluid Dialog Box: Relative Permeability 1231
7.25. The Table Input Dialog Box for Relative Permeability 1232
7.26. Skjaeveland Correlation Behavior [131] 1236
7.27. The Table File Manager Dialog Box 1238
7.28. The Table Input Dialog Box for Capillary Pressure 1239
7.29. The Fluid Dialog Box for a 3D Fan Zone 1246
7.30. The Inflection Point Ratio of a Pitched Blade Turbine 1247
7.31. Fixing Values for the Flow in a Stirred Tank 1250
7.32. Defining a Source for a Tiny Inlet 1254
7.33. The Operating Conditions Dialog Box 1259
7.34. The Pressure Inlet Dialog Box 1272
7.35. Cylindrical Velocity Components in 3D, 2D, and Axisymmetric Domains 1276
7.36. The Velocity Inlet Dialog Box 1281
7.37. The Mass-Flow Inlet Dialog Box 1288
7.38. The Mass-Flow Outlet Dialog Box 1295
7.39. The Inlet Vent Dialog Box 1300
7.40. The Intake Fan Dialog Box 1302

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.41. The Pressure Outlet Dialog Box 1305
7.42. Pressures at the Face of a Pressure Outlet Boundary 1310
7.43. The Pressure Outlet Dialog Box with the Target Mass Flow Rate Option Enabled 1312
7.44. The Pressure Far-Field Dialog Box 1315
7.45. Choice of the Outflow Boundary Condition Location 1319
7.46. The Outflow Dialog Box 1320
7.47. The Outlet Vent Dialog Box 1322
7.48. The Exhaust Fan Dialog Box 1325
7.49. The Wall Dialog Box for a Moving Wall 1328
7.50. The Wall Dialog Box for Specified Shear 1331
7.51. The Wall Dialog Box for the Specularity Coefficient 1332
7.52. The Wall Dialog Box for Marangoni Stress 1333
7.53. The Wall Dialog Box for Partial Slip Shear Condition 1335
7.54. Downward Shift of the Logarithmic Velocity Profile 1337
7.55. Illustration of Equivalent Sand-Grain Roughness 1338
7.56. The Wall Dialog Box for **High Roughness (Icing)** Models 1340
7.57. The Wall Dialog Box (Thermal Tab) 1343
7.58. A Thin Wall 1345
7.59. Uncoupled Thin Walls 1347
7.60. 2D Interface with Penetration and Gaps 1350
7.61. The Wall Dialog Box for Temperature Jump Thermal Condition 1352
7.62. The Wall Dialog Box for Species Boundary Condition Input 1353
7.63. The Perforated Walls Dialog Box 1358
7.64. Modeling Perforated Walls 1360
7.65. Uniform and Discrete Approaches 1361
7.66. The Perforated Walls Dialog Box 1363
7.67. The Injection Holes Dialog Box 1365
7.68. The Static Setup Dialog Box (Uniform Injection) 1367
7.69. The Dynamic Setup Dialog Box (Uniform Injection) 1368
7.70. Injection Surface and Tangential Angles 1369
7.71. Use of Symmetry to Model One Quarter of a 3D Duct 1374
7.72. Use of Symmetry to Model One Quarter of a Circular Cross-Section 1374
7.73. Inappropriate Use of Symmetry 1375
7.74. Use of Periodic Boundaries to Define Swirling Flow in a Cylindrical Vessel 1376
7.75. Example of Translational Periodicity - Physical Domain 1377
7.76. Example of Translational Periodicity - Modeled Domain 1377
7.77. The Periodic Dialog Box 1377
7.78. Use of an Axis Boundary as the Centerline in an Axisymmetric Geometry 1379
7.79. The Fan Dialog Box 1381
7.80. Polynomial Profile Dialog Box for Pressure Jump Definition 1382
7.81. A Fan Located In a 2D Duct 1384
7.82. The Radiator Dialog Box 1389
7.83. Polynomial Profile Dialog Box for Loss Coefficient Definition 1390
7.84. A Simple Duct with a Radiator 1391
7.85. The Porous Jump Dialog Box 1395
7.86. The Multi Edit Wall Settings Dialog Box 1397
7.87. Mesh and Prescribed Boundary Conditions in a 3D Axial Flow Problem 1401
7.88. Mesh and Prescribed Boundary Conditions in a 3D Radial Flow Problem 1401
7.89. Mesh and Prescribed Boundary Conditions in a 2D Case 1402
7.90. Prescribed Inlet Angles 1405

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.91. The Local Orthogonal Coordinate System onto which Euler Equations are Recasted for the General NRBC Method 1411
7.92. Waves Leaving and Entering a Boundary Face on Inflow and Outflow Boundaries. The Wave Amplitudes are Shown with the Associated Eigenvalues for a Subsonic Flow Condition 1412
7.93. The Pressure Outlet Dialog Box With the Non-Reflecting Boundary Enabled 1414
7.94. The Impedance Data Fitting Dialog Box 1420
7.95. The Inlet, Fan, and Pressure Outlet Zones for a Circular Fan Operating in a Cylindrical Domain 1428
7.96. The User-Defined Fan Model Dialog Box 1429
7.97. The Fan Dialog Box 1432
7.98. Transverse Velocities at the Site of the Fan 1433
7.99. Static Pressure Jump Across the Fan 1434
7.100. The Profiles Dialog Box 1441
7.101. Example of Using Profiles as Boundary Conditions 1443
7.102. The Orient Profile Dialog Box 1444
7.103. Scalar Profile at the Outlet 1446
7.104. Problem Specification 1448
7.105. The Replicate Profile Dialog Box 1449
7.106. The 1D Simulation Library Dialog Box 1456
7.107. Using GT-POWER Data for Boundary Conditions 1457
7.108. Cell Zone Conditions for Torque-Speed Coupling with GT-POWER 1458
7.109. The 1D Simulation Library Dialog Box with WAVE Selected 1460
7.110. Using WAVE Data for Boundary Conditions 1461
8.1. The Materials Task Page 1466
8.2. The Materials Branch in the **Outline View** 1467
8.3. Fluent Database Materials Dialog Box 1469
8.4. GRANTA MDS Materials Dialog Box 1471
8.5. Open Database Dialog Box 1474
8.6. User-Defined Database Materials Dialog Box 1475
8.7. New Material Name Dialog Box 1476
8.8. Copy Case Material Dialog Box 1477
8.9. User-Defined Database Materials Dialog Box: Blank 1479
8.10. Material Properties Dialog Box: Blank 1480
8.11. Edit Property Methods Dialog Box 1481
8.12. The Polynomial Profile Dialog Box 1483
8.13. The Piecewise-Linear Profile Dialog Box 1485
8.14. Piecewise-Linear Definition of Viscosity as a Function of Temperature 1486
8.15. The Piecewise-Polynomial Profile Dialog Box 1487
8.16. The NASA-9-Coefficient Piecewise-Polynomial Profile Dialog Box 1489
8.17. Compressible Liquid Materials Setting 1494
8.18. Compressible Liquid Density Settings Panel 1495
8.19. Variation of Viscosity with Shear Rate According to the Carreau Model 1508
8.20. The Carreau Model Dialog Box 1508
8.21. Variation of Shear Stress with Shear Rate According to the Herschel-Bulkley Model 1510
8.22. The Create/Edit Materials Dialog Box 1511
8.23. The Anisotropic Conductivity Dialog Box 1515
8.24. The Biaxial Conductivity Dialog Box 1516
8.25. The Orthotropic Conductivity Dialog Box 1517
8.26. The Cylindrical Orthotropic Conductivity Dialog Box 1518
8.27. Unaligned Principal Axes 1520
8.28. The Anisotropic Conductivity - Principal Components Dialog Box 1521
8.29. The UDS Diffusion Coefficients Dialog Box 1523

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.30. The Anisotropic UDS Diffusivity Dialog Box 1525
8.31. The Orthotropic UDS Diffusivity Dialog Box 1526
8.32. The Cylindrical Orthotropic UDS Diffusivity Dialog Box 1527
8.33. The UDS Diffusion Coefficients Dialog Box 1528
8.34. Anisotropic Species Diffusion Matrix 1538
8.35. The Thermal Diffusion Coefficients Dialog Box 1539
8.36. The Mass Diffusion Coefficients Dialog Box for Dilute Approximation 1541
8.37. The Mass Diffusion Coefficients Dialog Box for the Multicomponent Method 1542
8.38. Connected and Disconnected Cell Zones 1549
8.39. Typical PT Diagram of a Pure Material 1551
8.40. Typical PV Diagram of a Pure Material 1552
8.41. The Cubic Equation of State Model for a Real-Gas Fluid 1562
8.42. The Cubic Equation of State Model for a Real-Gas Mixture 1563
8.43. The Operating Conditions for a Real Gas State 1565
8.44. The PV Diagram for the Cubic Equation of State Real Gas Model 1566
8.45. The Table Settings Dialog Box 1595
8.46. The RGP Table Saturation Data Dialog Box 1596
9.1. The User-Defined Scalars Dialog Box 1601
9.2. The Fluid Dialog Box with Inputs for Source Terms for a User-Defined Scalar 1602
9.3. The User Scalar Sources Dialog Box 1603
9.4. The Materials Dialog Box with Input for Diffusivity for UDS Equations 1604
9.5. The User-Defined Scalars Dialog Box for a Multiphase Flow 1606
9.6. Example of Periodic Flow in a 2D Heat Exchanger Geometry 1608
9.7. The Periodic Conditions Dialog Box 1610
9.8. The Periodic Dialog Box 1612
9.9. Periodic Pressure Field Predicted for Flow in a 2D Heat Exchanger Geometry 1613
9.10. Rotating Flow in a Cavity 1616
9.11. Swirling Flow in a Gas Burner 1617
9.12. Flow in a Converging-Diverging Nozzle 1620
10.1. Single Component (Blower Wheel Blade Passage) 1630
10.2. Multiple Component (Blower Wheel and Casing) 1631
10.3. Single Blade Model with Rotationally Periodic Boundaries 1632
10.4. The Fluid Dialog Box Displaying Frame Motion Inputs 1634
10.5. Geometry with the Rotating Impeller 1635
10.6. Absolute Velocity Vectors 1638
10.7. Relative Velocity Vectors 1638
10.8. The Solution Initialization Task Page for Moving Reference Frames 1645
11.1. Two Passing Trains in a Tunnel 1649
11.2. Rotor-Stator Interaction (Stationary Guide Vanes with Rotating Blades) 1649
11.3. Blower 1650
11.4. Initial Position of the Meshes 1651
11.5. Rotor Mesh Slides with Respect to the Stator 1651
11.6. 2D Linear Mesh Interface 1652
11.7. 2D Circular-Arc Mesh Interface 1652
11.8. 3D Conical Mesh Interface 1653
11.9. 3D Planar-Sector Mesh Interface 1653
11.10. The Mesh Interfaces Dialog Box 1657
11.11. Lift Coefficient Plot for a Time-Periodic Solution 1660
11.12. Contours of Static Pressure for the Rotor-Stator Example 1661
11.13. The Dynamic Mesh Task Page 1665
11.14. The Smoothing Tab of the Mesh Method Settings Dialog Box 1666

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.15. The Mesh Smoothing Parameters Dialog Box 1667
11.16. The Initial Mesh 1669
11.17. Valid Mesh After 45 Degree Rotation Using Diffusion-Based Smoothing 1670
11.18. Degenerated Mesh After 40 Degree Rotation Using Spring-Based Smoothing 1670
11.19. Effect of Diffusion Parameter of 0 on Interior Node Motion 1672
11.20. Effect of Diffusion Parameter of 1 on Interior Node Motion 1673
11.21. Spring-Based Smoothing on Interior Nodes: Start 1675
11.22. Spring-Based Smoothing on Interior Nodes: End 1676
11.23. Interior Nodes Extend Beyond Boundary (Spring Constant Factor = 1) 1677
11.24. Interior Nodes Remain Within Boundary (Spring Constant Factor = 0) 1677
11.25. The Undeformed Mesh 1682
11.26. The Deformed Mesh 1682
11.27. Zooming into the Undeformed Compliant Strip 1683
11.28. Zooming into the Deformed Compliant Strip with Boundary Layer Smoothing Applied 1684
11.29. Dynamic Layering 1685
11.30. Results of Splitting Layer with the Height-Based Option 1685
11.31. Results of Splitting Layer with the Ratio-Based Option 1686
11.32. The Layering Tab in the Mesh Method Settings Dialog Box 1687
11.33. Use of Sliding Interfaces to Transition Between Adjacent Cell Zones and the Dynamic Layering Cell Zone 1688
11.34. The Remeshing Tab in the Mesh Method Settings Dialog Box for Methods-Based Remeshing 1690
11.35. The Remeshing Tab in the Mesh Method Settings Dialog Box for Unified Remeshing 1691
11.36. Mesh at the End of a Dynamic Mesh Simulation Without Sizing Functions 1694
11.37. Mesh at the End of a Dynamic Mesh Simulation With Sizing Functions 1694
11.38. Sizing Function Determination at Background Mesh Vertex I 1695
11.39. Interpolating the Value of the Sizing Function 1696
11.40. Determining the Normalized Distance 1697
11.41. Expanding Cylinder Before Region Face Remeshing 1700
11.42. Expanding Cylinder After Region Face Remeshing 1701
11.43. Volume Decomposition for Prism Layers 1702
11.44. Volume Decomposition for the Base of the Prism Layers 1703
11.45. Close-Up of 2.5D Extruded Flow Meter Pump Geometry Before Remeshing and Laplacian Smoothing 1704
11.46. Close-Up of 2.5D Extruded Flow Meter Pump Geometry After Remeshing and Laplacian Smoothing . 1705
11.47. The Remeshing Tab for the 2.5D Model 1706
11.48. 2.5D Extruded Gear Pump Geometry 1707
11.49. The Advanced Remeshing Settings Dialog Box 1708
11.50. Cross Section of a 3D Corner 1710
11.51. The In-Cylinder Tab of the Options Dialog Box 1711
11.52. Determining the Sign of the Piston Pin Offset 1712
11.53. The In-Cylinder Output Controls Dialog Box 1713
11.54. Sample Output File Showing Various Quantities 1716
11.55. A 2D In-Cylinder Geometry 1716
11.56. Mesh Topology Showing the Various Mesh Regions 1717
11.57. Mesh Associated With the Chosen Topology 1718
11.58. The Use of Sliding Interfaces to Connect the Exhaust Valve Layering Zone to the Remeshing Zone ... 1719
11.59. Mesh Sequence 1 1720
11.60. Mesh Sequence 2 1720
11.61. Mesh Sequence 3 1721
11.62. Mesh Sequence 4 1721
11.63. Mesh Sequence 5 1722

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11.64. Mesh Sequence 6 1722
11.65. Piston Position (m) as a Function of Crank Angle (deg) 1723
11.66. Intake and Exhaust Valve Lift (m) as a Function of Crank Angle (deg) 1724
11.67. Definition of Valve Zone Attributes (Intake Valve) 1725
11.68. The Six DOF Tab of the Options Dialog Box 1726
11.69. The Six DOF Properties Dialog Box 1728
11.70. A Check Valve with One DOF Translation 1729
11.71. The Implicit Update Tab of the Options Dialog Box 1730
11.72. The Contact Detection Tab of the Options Dialog Box 1732
11.73. The Flow Control Settings Dialog Box with Contact Zones 1733
11.74. The Flow Control Settings Dialog Box with Contact Marks 1734
11.75. The Dynamic Mesh Events Dialog Box 1736
11.76. The Define Event Dialog Box 1737
11.77. The Events Preview Dialog Box for In-Cylinder Flows 1738
11.78. The Define Event Dialog Box for the Creating Sliding Interface Option 1740
11.79. Boundary Zone Before Insertion 1742
11.80. Boundary Zone After Insertion 1742
11.81. Interior Zone Before Insertion 1743
11.82. Interior Zone After Insertion 1743
11.83. The Dynamic Mesh Zones Dialog Box for a Stationary Zone 1746
11.84. The Dynamic Mesh Zones Dialog Box for a Rigid Body Motion 1747
11.85. Orientation Calculator Dialog Box 1749
11.86. The Dynamic Mesh Zones Dialog Box for a Rigid Body Motion Using the Six DOF Solver 1750
11.87. The Dynamic Mesh Zones Dialog Box for a Deforming Motion with Cell Zone Options 1752
11.88. The Dynamic Mesh Zones Dialog Box for an Intrinsic FSI Zone 1758
11.89. Solid Body Rotation Coordinates 1762
11.90. The Zone Motion Dialog Box 1763
11.91. The Mesh Motion Dialog Box 1764
11.92. The Mesh Motion Dialog Box for Steady-State Dynamic Meshes 1767
11.93. Initial Object Position 1768
11.94. The Mesh Motion Dialog Box After 40 Updates 1769
11.95. Final Object Position After 40 Executions 1769
12.1. Frozen Gust variations 1771
12.2. Create/Edit Turbo Interfaces Dialog Box 1779
12.3. The Mixing Planes Dialog Box 1784
12.4. Passage Numbering from Profile Expansion 1791
12.5. Periodic Displacement Example for Real Only Mode Shape 1797
12.6. Periodic Displacement Example for Complex Mode Shape 1798
12.7. Aero Report Definition Dialog Box 1805
12.8. The Turbo Topology Dialog Box 1808
12.9. Turbomachinery Boundary Types 1811
12.10. The Turbo Report Dialog Box 1812
12.11. Pump or Compressor 1816
12.12. Turbine 1818
12.13. The Turbo Averaged Contours Dialog Box 1820
12.14. Turbo Averaged Filled Contours of Static Pressure 1821
12.15. The Turbo 2D Contours Dialog Box 1822
12.16. The Turbo Averaged XY Plot Dialog Box 1824
12.17. The Turbo Options Dialog Box 1825
13.1. Velocity Profiles for Axi-symmetric Diffuser Flow (Case CS0 – Driver). Impact of Variation of C_{SEP} 1833

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.2. Impact of Changes in C_{MIX} on Free Mixing Layer. Left: Velocity Profiles, Right: Turbulence Kinetic Energy Profiles 1833
13.3. Impact of Changes in C_{NW} on Backward Facing Jet with Heat Transfer. Left: Wall Shear Stress Coefficient, C_f, Right: Wall Heat Transfer Coefficient, S_t 1834
13.4. Impact of Changes in C_{JET} on Free Jet Flows. Left: Plane Jet, Right: Round Jet 1834
13.5. Illustration of SST-URANS vs. SST-SAS Models 1840
13.6. The Viscous Model Dialog Box 1848
13.7. The Viscous Model Dialog Box Displaying the Spalart-Allmaras Production 1850
13.8. The Viscous Model Dialog Box Displaying the Standard k-ε Model 1852
13.9. The Viscous Model Dialog Box Displaying the RNG k-ε Model 1854
13.10. The Viscous Model Dialog Box Displaying the Standard k-ω Model 1856
13.11. The Viscous Model Dialog Box Displaying the BSL k-ω Model 1858
13.12. The Viscous Model Dialog Box Displaying the SST k-ω Model 1860
13.13. The Viscous Model Dialog Box with GEKO Options for the Full Model 1862
13.14. The Viscous Model Dialog Box for the Transition SST Model 1864
13.15. Transition Option enabled in Combination with the SST k-ω Model 1866
13.16. The Viscous Model Dialog Box Displaying the Reynolds Stress Model Options 1868
13.17. The Viscous Model Dialog Box Displaying the Stress-Omega Model Options 1870
13.18. Scale-Adaptive Simulation (SAS) in Combination with the SST Turbulence Model 1871
13.19. Scale-Adaptive Simulation (SAS) in Combination with the Transition SST Model 1872
13.20. The Viscous Model Dialog Box Displaying Options for DES with the Spalart-Allmaras Model 1874
13.21. The Viscous Model Dialog Box Displaying Options for DES with the Realizable k-ε Model 1876
13.22. The Viscous Model Dialog Box Displaying Options for DES with the SST k-ω Model 1878
13.23. The Viscous Model Dialog Box Displaying Options for DES with the BSL k-ω Model 1880
13.24. The Viscous Model Dialog Box Displaying Options for DES with the Transition SST Model 1881
13.25. The Viscous Model Dialog Box Displaying the Large Eddy Simulation Model Options 1882
13.26. Specifying an ELES Zone in the Fluid Dialog Box 1885
13.27. Specifying the RANS/LES Interface 1886
13.28. The RANS/LES Interface Dialog Box 1887
13.29. SST Model with the **Buoyancy Effects: Only Turbulence Production** Option Enabled 1889
13.30. The Viscous Model Dialog Box with Corner Flow Correction option enabled 1892
13.31. The Viscous Model Dialog Box with the SBES Options for ω-based RANS models 1901
13.32. Specifying Inlet Boundary Conditions for the Reynolds Stresses 1905
13.33. The Sampling Options Dialog Box 1920
13.34. The Zone-Specific Sampling Options Dialog Box 1921
14.1. Enabling the Energy Equation 1924
14.2. The Conduction Manager Dialog Box 1926
14.3. Typical Counterflow Heat Exchanger Involving Heat Transfer Between Two Separated Fluid Streams .. 1927
14.4. The Run Calculation Task Page Showing Solid Time Stepping 1930
14.5. The Run Calculation Task Page with Loosely Coupled Conjugate Heat Transfer 1933
14.6. The Time Averaged Explicit Thermal Coupling Dialog Box 1935
14.7. A Boundary Wall with Shell Conduction 1939
14.8. A Two-Sided Wall with Shell Conduction 1939
14.9. The Conduction Manager Dialog Box 1941
14.10. Conduction Layers Dialog Box 1942
14.11. Shell Surface Names for a Boundary Wall 1944
14.12. Shell Surface Names for a Two-Sided Wall 1944
14.13. The Radiation Model Dialog Box (DO Model) 1947
14.14. The Radiation Model Dialog Box (Non-Gray P-1 Model) 1949
14.15. The DTRM Rays Dialog Box 1950

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.16. The Radiation Model Dialog Box (S2S Model) 1953
14.17. The View Factors and Clustering Dialog Box 1955
14.18. The Wall Dialog Box 1957
14.19. The Participating Boundary Zones Dialog Box 1961
14.20. The Thread Control Dialog Box 1965
14.21. The Radiation Model Dialog Box (Non-Gray DO Model) 1967
14.22. The Radiation Model Dialog Box with DO/Energy Coupling Enabled 1968
14.23. The Radiation Model Dialog Box (MC) 1970
14.24. The Wall Dialog Box Showing Radiation Conditions for an Opaque Wall 1974
14.25. The Wall Dialog Box Showing External Emissivity and External Radiation Temperature Thermal Conditions 1975
14.26. The Wall Dialog Box for a Semi-Transparent Wall Boundary 1976
14.27. The Wall Dialog Box for an Interior Semi-Transparent Wall 1978
14.28. The Wall Dialog Box for an Opaque Wall with MC Model (Gray) 1979
14.29. The Wall Dialog Box for an Opaque Wall with MC Model (Boundary Source) 1980
14.30. The Wall Dialog Box for an Opaque Wall with MC Model (Polar Distribution with Expression) 1982
14.31. The Wall Dialog Box for an Opaque Wall with MC Model (Polar Distribution with Table) 1983
14.32. The Wall Dialog Box for an Opaque Wall with MC Model (Non-gray) 1984
14.33. The Wall Dialog Box Showing External Emissivity and External Radiation Temperature Thermal Conditions 1985
14.34. The Wall Dialog Box for a Semi-transparent Wall with MC Model 1986
14.35. The Solid Dialog Box 1987
14.36. The Radiation Model Dialog Box (DTRM) 1990
14.37. The DTRM Graphics Dialog Box 1996
14.38. Ray Display 1997
14.39. The S2S Information Dialog Box 1998
14.40. The Radiation Model Dialog Box 2007
14.41. The Radiation Model Dialog Box (With Solar Load Model Solar Ray Tracing Option) 2008
14.42. The Radiation Model Dialog Box (with Solar Load Model Solar Irradiation Option) 2009
14.43. The Solar Calculator Dialog Box 2011
14.44. The Velocity Inlet Dialog Box 2013
14.45. The Wall Dialog Box 2014
14.46. The Wall Dialog Box 2015
14.47. The Porous Jump Dialog Box 2016
14.48. The Wall Dialog Box Radiation tab with Solar Irradiation 2018
14.49. The Contours Dialog Box 2023
14.50. The Execute Commands Dialog Box 2024
14.51. Temperature Field in a 2D Heat Exchanger Geometry With Fixed Temperature Boundary Conditions 2030
14.52. An Example of a Four-Pass Heat Exchanger 2031
14.53. Heat Exchanger Modeling Options 2033
14.54. The Heat Exchanger Model Dialog Box 2034
14.55. The Dual Cell Heat Exchanger Dialog Box 2035
14.56. The Set Dual Cell Heat Exchanger Dialog Box 2036
14.57. The Heat Rejection Tab 2037
14.58. The Performance Data Tab 2038
14.59. The Heat Transfer Data Table Dialog Box 2039
14.60. The Frontal Area Tab 2040
14.61. The Coupling Tab 2041
14.62. An Example of a Four-Pass Heat Exchanger 2041
14.63. The Heat Exchanger Model Dialog Box 2045

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.64. The Ungrouped Macro Heat Exchanger Dialog Box Displaying the Model Data Tab 2046
14.65. The Heat Transfer Data Table Dialog Box for the NTU Model 2047
14.66. The Ungrouped Macro Heat Exchanger Dialog Box Displaying the Geometry Tab 2048
14.67. The Ungrouped Macro Heat Exchanger Dialog Box Displaying the Auxiliary Fluid Tab 2049
14.68. 1x4x3 Macros 2052
14.69. Mesh Display With Macros 2053
14.70. The Core Porosity Model Dialog Box 2055
14.71. The Macro Heat Exchanger Group Dialog Box 2057
14.72. The Heat Transfer Data Table Dialog Box for the NTU Model 2058
14.73. The Macro Heat Exchanger Group Dialog Box - Geometry Tab 2059
14.74. The Macro Heat Exchanger Group Dialog Box - Auxiliary Fluid Tab 2060
14.75. The Macro Heat Exchanger Group Dialog Box - Supplementary Auxiliary Fluid Stream Tab 2061
14.76. The Heat Exchanger Report Dialog Box for Reporting Computed Heat Rejection 2065
14.77. The Heat Exchanger Report Dialog Box for Reporting the Inlet Temperature 2066
14.78. The Heat Exchanger Report Dialog Box for Reporting Mass Flow Rate 2067
14.79. The Heat Exchanger Report Dialog Box for Reporting Specific Heat 2068
14.80. The Volume Report Definition Dialog Box 2069
14.81. The Energy Dialog Box 2070
14.82. The Create/Edit Materials Dialog Box 2071
14.83. The Blottner Curve Fit Dialog Box 2072
14.84. Species Model Dialog Box with Settings for Two-Temperature Model 2073
14.85. Reactions Dialog Box with Settings for Two-Temperature Model 2074
14.86. The Fluid Dialog Box 2075
14.87. Pressure Far-Field Dialog Box 2076
14.88. Pressure Far-Field Dialog Box with Non-Equilibrium Boundary Option 2077
14.89. Two-sided Wall Dialog Box with Coupled Option 2078
14.90. The Solution Methods Task Page 2079
14.91. The Solution Controls Task Page 2080
14.92. The Equations Dialog Box 2080
14.93. The Residual Monitors Dialog Box 2081
15.1. The Species Model Dialog Box 2087
15.2. The Species Model Dialog Box Displaying the Thickened Flame Model 2093
15.3. The Select Boundary Species Dialog Box 2094
15.4. The Select Residual Monitored Species 2095
15.5. The Import CHEMKIN Format Mechanism Dialog Box for Volumetric Kinetics 2097
15.6. The Material Dialog Box When Importing CHEMKIN Transport Properties 2099
15.7. The Create/Edit Materials Dialog Box (Showing a Mixture Material) 2102
15.8. The Species Dialog Box 2103
15.9. The Reactions Dialog Box 2107
15.10. The Third-Body Efficiency Dialog Box 2110
15.11. The Pressure-Dependent Reaction Dialog Box 2111
15.12. The Coverage Dependent Reaction Dialog Box 2112
15.13. Backward Reaction Parameters Dialog Box 2113
15.14. The Reaction Mechanisms Dialog Box 2115
15.15. The Site Parameters Dialog Box 2116
15.16. The Coal Calculator Dialog Box 2119
15.17. The Import CHEMKIN Format Mechanism Dialog Box for Surface Kinetics 2128
15.18. The Species Model Dialog Box with Electrochemical Reactions Enabled 2137
15.19. The Reactions Dialog Box 2139
15.20. The Reaction Mechanisms Dialog Box 2141
15.21. Wall Potential Boundary Condition 2142

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.22. Optimal Surface Mesh on the Reacting Channel Wall 2147
15.23. The Reacting Channel Model Dialog Box 2148
15.24. The Reacting Channel Model Dialog Box (Group Inlet Conditions Tab) 2150
15.25. The Wall Boundary Condition Dialog Box for the Reacting Channel Model 2152
15.26. Reacting Channel 2D Curves Dialog Box (Plot) 2153
15.27. Reacting Channel 2D Curves Dialog Box (Report) 2154
15.28. Reactor Network Dialog Box (Steady-State Flow) 2156
15.29. Reactor Network Dialog Box - Expert Options 2157
15.30. The Species Model Dialog Box for Lagrangian Composition PDF Transport 2162
15.31. The Integration Parameters Dialog Box 2163
15.32. The Species Model Dialog Box for Eulerian Composition PDF Transport 2165
15.33. The Velocity Inlet Dialog Box for Eulerian Composition PDF Transport 2167
15.34. The Solution Initialization Task Page for Eulerian Composition PDF Transport 2168
15.35. The Run Calculation Task Page for Composition PDF Transport 2169
15.36. The Particle Tracks Dialog Box for Tracking PDF Particles 2172
15.37. The Integration Parameters Dialog Box 2174
15.38. The **Select DAC Target Species** Dialog Box 2186
16.1. Defining Equilibrium Chemistry 2189
16.2. Defining Steady Diffusion Flamelet Chemistry 2189
16.3. Defining Chemical Boundary Species 2190
16.4. Calculating Steady Diffusion Flamelets 2191
16.5. Calculating the Chemistry Look-Up Table 2191
16.6. The Species Model Dialog Box (Chemistry Tab) 2193
16.7. The Chemistry Tab for the Unsteady Diffusion Flamelet Model 2197
16.8. The Enabled Diesel Unsteady Flamelet Model 2200
16.9. The Unsteady Flamelet Parameters Dialog Box 2201
16.10. The Flamelet Fluid Zones Dialog Box 2202
16.11. The Species Model Dialog Box (Boundary Tab) 2204
16.12. The Coal Calculator Dialog Box 2213
16.13. The Species Model Dialog Box (Control Tab) 2215
16.14. The Species Model Dialog Box (Control Tab) for the Steady Diffusion Flamelet Model 2216
16.15. Method to Zero Out the Slow Chemistry Species 2217
16.16. The Species Model Dialog Box (Flamelet Tab) 2218
16.17. The Flamelet Tab for the Unsteady Diffusion Flamelet Model 2221
16.18. The Flamelet 2D curves Dialog Box 2222
16.19. The Flamelet 3D Surfaces Dialog Box 2223
16.20. Example 2D Plot of Flamelet Data 2224
16.21. Example 3D Plot of Flamelet Data 2225
16.22. The Species Model Dialog Box (Table) Tab Displaying Automated Grid Refinement 2226
16.23. The Species Model Dialog Box (Table) Tab Excluding Automated Grid Refinement 2227
16.24. The PDF Table Dialog Box (Non-Adiabatic Case With Flamelets) 2231
16.25. Mean Species Mole Fraction Derived From an Equilibrium Chemistry Calculation 2234
16.26. Mean Temperature Derived From an Equilibrium Chemistry Calculation 2234
16.27. 3D Plot of Look-Up Table for Temperature Generated for a Simple Hydrocarbon System 2235
16.28. The Inert Model Dialog Box 2238
16.29. The Inert Model Dialog Box 2239
16.30. The Velocity Inlet Dialog Box Showing Mixture Fraction Boundary Conditions 2242
16.31. The Species Model Dialog Box for a Two-Mixture-Fraction Calculation 2248
16.32. Predicted Contours of Mixture Fraction in a Methane Diffusion Flame 2250
16.33. Predicted Contours of CO2 Mass Fraction Using the Non-Premixed Combustion Model 2250
16.34. The Species Model Dialog Box for Premixed Combustion 2253

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.35. The Species Model Dialog Box for the G-Equation Model 2254
16.36. The Progress Variable Definition Dialog Box 2262
16.37. Premixed Flamelet Generated Manifolds (Flamelet Tab) 2263
16.38. The Distribution of Points Dialog Box 2266
16.39. Diffusion Flamelet Generated Manifolds (Flamelet Tab) 2267
16.40. Non-adiabatic Premixed Flamelet Generated Manifolds (**Flamelet** Tab) 2269
16.41. The Species Model Dialog Box: Table Tab with no Automated Grid Refinement 2270
16.42. The Species Model Dialog Box: Table Tab Displaying Automated Grid Refinement 2272
16.43. The Select Transported Scalars Dialog Box 2273
16.44. The PDF Table Dialog Box (Adiabatic Case With FGM) 2275
16.45. The PDF Table Dialog Box (Non-Adiabatic Case With FGM) 2276
16.46. The Species Model Dialog Box(Premix Tab) 2279
16.47. The Species Model Dialog Box (Properties Tab) 2281
16.48. The Quadratic of Mixture Fraction Dialog Box 2281
16.49. The Piecewise Linear Dialog Box 2282
16.50. The Species Model Dialog Box: Strained Laminar Flame Speed 2283
16.51. The Flamelet 2D Curves Dialog Box 2284
17.1. The Spark Ignition Dialog Box 2288
17.2. The Set Spark Ignition Dialog Box 2289
17.3. The Ignition Delay Model in the Autoignition Model Dialog Box 2291
17.4. The Knock Model in the Autoignition Model Dialog Box 2291
17.5. The Ignition Delay Model for the Partially Premixed Combustion Model 2292
17.6. The Knock Model with the Partially Premixed Combustion Model Enabled 2293
17.7. Experimental Engine Mesh 2294
17.8. Cylinder Mass vs. Crank Angle 2296
17.9. Cylinder Pressure vs. Crank Angle 2297
17.10. Crevice Pressures 2299
18.1. The NOx Model Dialog Box 2303
18.2. The NOx Model Dialog Box Displaying the Fuel Streams 2306
18.3. The NOx Dialog Box Displaying the Reburn Reduction Method 2312
18.4. The NOx Dialog Box Displaying the SNCR Reduction Method 2312
18.5. The NOx Model Dialog Box and the Turbulence Interaction Mode Tab 2314
18.6. The Soot Model Dialog Box for the One-Step Model 2320
18.7. The Soot Model Dialog Box for the Two-Step Model 2322
18.8. The Soot Model Dialog Box for the Moss-Brookes Model 2325
18.9. The Soot Model Dialog Box for the Moss-Brookes Model with a User-Defined Precursor Correlation ... 2330
18.10. The Piecewise-Polynomial Profile Dialog Box 2331
18.11. The Soot Model Dialog Box for the Method of Moments Model 2333
18.12. Sticking Coefficients for Soot Precursors 2334
18.13. Settings for the Nucleation Mechanism 2335
18.14. The Decoupled Detailed Chemistry Dialog Box 2339
19.1. The Acoustics Model Dialog Box 2346
19.2. The Acoustics Model Dialog Box for a 3D Steady-State Case with a Single Moving Reference Frame 2348
19.3. The Acoustics Model Dialog Box for Exporting in CGNS Format 2350
19.4. The Acoustics Model Dialog Box 2351
19.5. The Interior Cell Zone Selection Dialog Box 2352
19.6. An Interior Source Surface 2353
19.7. The Acoustic Receivers Dialog Box 2354
19.8. The Run Calculation Task Page 2356
19.9. The Acoustic Signals Dialog Box 2358
19.10. The Read ASD Files Tab of the Acoustic Source FFT Dialog Box 2362

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.11. The Compute FFT Fields Tab of the Acoustic Source FFT Dialog Box 2363
19.12. The FFT Surface Variables Tab of the Acoustic Source FFT Dialog Box for the Octave Bands 2364
19.13. Bar Chart of Surface Pressure Level for Octave Bands 2366
19.14. The FFT Surface Variables Tab of the Acoustic Source FFT Dialog Box for a Set of Individual Modes ... 2366
19.15. The Write CGNS Files Tab of the Acoustic Source FFT Dialog Box 2368
19.16. The Acoustics Model Dialog Box 2370
19.17. The Basic Shapes Dialog Box 2372
19.18. The Acoustics Wave Equation Solver Controls Task Page 2373
19.19. The Acoustics Initialization Dialog Box 2374
19.20. Acoustics Model Dialog Box with Kirchhoff Integral Options 2377
19.21. Integration Surface Dialog Box 2378
19.22. The Acoustics Model Dialog Box for Broadband Noise 2380
20.1. Valid Configuration for the Lagrangian Wall film Model with Overset Mesh 2390
20.2. Invalid Configuration for the Lagrangian Wall film Model with Overset Mesh 2390
20.3. The Discrete Phase Model Dialog Box - Tracking Tab 2398
20.4. A Subtet Formed From Decomposing a Hexagonal Cell 2400
20.5. Degenerate Subtet in a Polyhedral Mesh 2402
20.6. Degenerate Subtet in a Hex Mesh 2402
20.7. Lagrangian Wall Film Tracking 2403
20.8. The Discrete Phase Model Dialog Box - Physical Models Tab 2407
20.9. Discrete Phase Model Dialog Box with DEM Collision Model 2412
20.10. Wall Boundary Condition for the DEM Model 2414
20.11. Collision Dialog Box 2415
20.12. DEM Collision Settings Dialog Box 2415
20.13. The Discrete Phase Model Dialog Box - UDF Tab 2418
20.14. The Discrete Phase Model Dialog Box - Numerics Tab 2420
20.15. Particle Injection Defining a Single Particle Stream 2430
20.16. Particle Injection Defining an Initial Spatial Distribution of the Particle Streams 2430
20.17. Particle Injection Defining an Initial Spray Distribution of the Particle Velocity 2430
20.18. Cone Injector Geometry 2435
20.19. Flat Fan Viewed from Above and from the Side 2444
20.20. Example of Cumulative Size Distribution of Particles 2449
20.21. Rosin-Rammler Curve Fit for the Example Particle Size Data 2450
20.22. The Injections Branch in the **Outline View** 2452
20.23. The Injections Dialog Box 2452
20.24. The Set Injection Properties Dialog Box 2455
20.25. Setting Surface Injection Properties 2457
20.26. Mean Trajectory in a Turbulent Flow 2469
20.27. Stochastic Trajectories in a Turbulent Flow 2470
20.28. The Custom Laws Dialog Box 2471
20.29. The Set Multiple Injection Properties Dialog Box 2472
20.30. Discrete Phase Boundary Conditions in the Wall Dialog Box 2477
20.31. "Trap" Boundary Condition for the Discrete Phase 2478
20.32. "Escape" Boundary Condition for the Discrete Phase 2479
20.33. The Wall Dialog Box: the Particle-Wall Heat Exchange Option 2484
20.34. The Set Injection Properties Dialog Box: Condensate Injection 2486
20.35. The **Generic Erosion Model Parameters** Dialog Box 2488
20.36. The **Finnie Model Parameters** Dialog Box 2488
20.37. The **McLaury Model Parameters** Dialog Box 2489
20.38. The **Oka Model Parameters** Dialog Box 2490
20.39. The **DNV Model Parameters** Dialog Box 2491

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.40. The **Shear Stress Model Parameters** Dialog Box 2491
20.41. The **Erosion Dynamic Mesh Coupling Setup** Dialog Box 2494
20.42. The **Run Erosion-Dynamic Mesh Simulation** Dialog Box 2496
20.43. The **Graphics Objects** Dialog Box 2498
20.44. The Components Tab 2504
20.45. Uncoupled Discrete Phase Calculations 2516
20.46. Coupled Discrete Phase Calculations 2517
20.47. Effect of Number of Source Term Updates on Source Term Applied to Flow Equations 2519
20.48. The Particle Tracks Dialog Box 2523
20.49. The Track Style Attributes Dialog Box 2526
20.50. The Particle Sphere Style Attributes Dialog Box 2527
20.51. Particles with the Vector Style 2529
20.52. Particles with the Centered Vector Style 2530
20.53. Particles with the Centered Cylinder Style 2531
20.54. The Particle Vector Style Attributes Dialog Box 2532
20.55. The Import Particle Data Dialog Box 2532
20.56. The Particle Filter Attributes Dialog Box 2533
20.57. The Reporting Variables Dialog Box 2542
20.58. The Sample Trajectories Dialog Box 2552
20.59. The Trajectory Sample Histograms Dialog Box 2554
20.60. The Trajectory Sample Histograms Dialog Box: Correlation 2556
20.61. The Trajectory Sample Histograms Dialog Box: Data File Reduction 2557
20.62. The Particle Summary Dialog Box 2559
20.63. The Shared Memory Option with Workpile Algorithm Enabled 2565
21.1. Macroscopic Particle Model Dialog Box (Particle Tracking Tab) 2572
21.2. Macroscopic Particle Model Dialog Box (Drag Tab) 2574
21.3. Macroscopic Particle Model Dialog Box (Collision Tab) 2576
21.4. Macroscopic Particle Model Dialog Box (Deposition Tab) 2578
21.5. Macroscopic Particle Model Dialog Box (Injection Tab) 2579
21.6. Macroscopic Particle Model Dialog Box (Attraction Forces Tab) 2587
21.7. Macroscopic Particle Model Dialog Box (Initialize MPM Tab) 2588
22.1. Multiphase Model Dialog Box for the VOF Model 2595
22.2. Multiphase Model Dialog Box for the Mixture Model 2596
22.3. Multiphase Model Dialog Box for the Eulerian Model 2597
22.4. Numerical Flotsams in the Volume Fraction Field 2604
22.5. The Volume Fraction Field for the Cell Based Flotsam Detection 2604
22.6. The Volume Fraction Field for the Node Based Flotsam Detection 2605
22.7. The Volume Fraction Field for the Node-Averaged Filtering 2605
22.8. The Operating Conditions Dialog Box for Multiphase Flows 2606
22.9. The **Multiphase Model** dialog box - **Phases** tab 2609
22.10. The Species Model Dialog Box with a Multiphase Model Enabled 2611
22.11. The Phase Properties Dialog Box 2612
22.12. The Reactions Tab for Heterogeneous Reactions 2614
22.13. The Mass Tab for Mass Transfer 2617
22.14. The Cavitation Model Dialog Box 2622
22.15. Table Input for Vaporization Pressure 2623
22.16. The Evaporation-Condensation Model Dialog Box (Eulerian Multiphase Model) 2626
22.17. The Species Mass Transfer Model Dialog Box 2629
22.18. The Pressure Inlet Dialog Box for a Mixture 2633
22.19. The Wall Dialog Box for a Mixture in a Multiphase Calculation with Wall Adhesion 2634
22.20. Measuring the Contact Angle 2635

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.21. The Porous Jump Dialog Box Displaying Jump Adhesion 2636
22.22. The Wall Dialog Box for a Phase 2637
22.23. The Pressure Outlet Dialog Box for a Phase 2638
22.24. The Cell Zone Conditions Task Page 2639
22.25. Mass-Flow Inlet Boundary Condition Dialog Box 2645
22.26. Determining the Free Surface Level and the Bottom Level 2659
22.27. Pressure Inlet for Open Channel Flow 2660
22.28. Density Interpolation Method for Open Channel Flow 2662
22.29. The Velocity Inlet for Open Channel Wave BC 2665
22.30. Segregated Velocity Inputs for Open Channel Wave BC 2666
22.31. The Velocity Inlet for Open Channel Wave BC (Explicit Formulation) 2668
22.32. The Solution Initialization Task Page 2676
22.33. The Fluid Dialog Box to Enable Numerical Beach 2679
22.34. Numerical Beach Sketch 2682
22.35. Defining the Primary Phase in the Phases Tab 2684
22.36. Defining the Secondary Phase in the Phases Tab 2685
22.37. The Multiphase Model Dialog Box (Forces Tab) 2686
22.38. The Multiphase Model Dialog Box for the VOF Model (Discretization Tab) 2690
22.39. The VOF-to-DPM Transition Parameters Dialog Box 2698
22.40. The DPM-to-VOF Transition Parameters Dialog Box 2705
22.41. DPM Particles at the Gas-Liquid Interface 2708
22.42. Surface Mesh Before the Model Transition is Triggered 2708
22.43. Gas-Liquid Interface after the Model Transition is Triggered 2709
22.44. Surface Mesh After the Mesh Adaption and Phase Transition 2710
22.45. Defining the Secondary Phase for the Mixture Model 2712
22.46. Defining a Granular Phase in the Mixture Model 2713
22.47. The Multiphase Model Dialog Box Displaying the Interfacial Area Concentration Settings 2716
22.48. The Multiphase Model Dialog Box for the Mixture Model (Forces Tab) 2720
22.49. The Evaporation-Condensation Model Dialog Box 2723
22.50. Boiling Model Expert Options 2727
22.51. Transition Function vs. Volume Fraction of Liquid 2729
22.52. The Multiphase Model Dialog Box for a Non-Granular Phase (**Phases** Tab) 2732
22.53. The Multiphase Model Dialog Box for a Granular Phase (**Phases** Tab) 2733
22.54. **Syamlal Obrien Model** Dialog Box 2742
22.55. Antal et al. Model Dialog Box 2747
22.56. Tomiyama Model Dialog Box 2747
22.57. Frank Model Dialog Box 2748
22.58. Hosokawa Model Dialog Box 2749
22.59. Lopez de Bertodano Model Dialog Box 2750
22.60. Simonin Model Dialog Box 2751
22.61. Burns et al. Model Dialog Box 2751
22.62. Diffusion—in—vof Model Dialog Box 2752
22.63. The Viscous Model Dialog Box for an Eulerian Multiphase Calculation 2755
22.64. Troshko-Hassan Model Dialog Box 2756
22.65. Sato Model Dialog Box 2757
22.66. Simonin-et-al Model Dialog Box 2757
22.67. The Multiphase Model Dialog Box for Heat Transfer 2758
22.68. The Multiphase Model Dialog Box for Interfacial Area 2760
22.69. Defining a Secondary Phase with the AIAD Continuous Phase Treatment 2763
22.70. Example of Defining a Secondary Entrained Phase for Droplets 2764
22.71. The **AIAD Model Parameters** Dialog Box 2766

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.72. The **AIAD Entrainment Parameters** Dialog Box 2767
22.73. The Dense Discrete Phase Model 2772
22.74. The Multiphase Model Dialog Box for DDPM 2773
22.75. The Set Injection Properties Dialog Box 2775
22.76. The Multiphase Model Dialog Box for a Granular Phase 2776
22.77. The Boiling Model 2778
22.78. The Multiphase Model Dialog Box for the Boiling Model (Phases Tab) 2780
22.79. Defining Forces in the Forces Tab 2781
22.80. The Boiling Model Dialog Box 2783
22.81. The Population Balance Model Dialog Box 2792
22.82. The **Liao Aggregation Model Parameters** Dialog Box 2796
22.83. The Surface Tension for Population Balance Dialog Box 2797
22.84. The Hamaker Constant for Population Balance Dialog Box 2797
22.85. The **Prince and Blanch Model Parameters** Dialog Box 2798
22.86. **Liao Breakage Model Parameters** Dialog Box 2799
22.87. The Surface Tension and Weber Number Dialog Box 2800
22.88. The Ghadiri Breakage Constant for Population Balance Dialog Box 2800
22.89. The Shape Factor for Parabolic PDF Dialog Box 2801
22.90. The Generalized pdf for multiple breakage Dialog Box 2801
22.91. The Population Balance Model Tab for the DQMOM Model 2804
22.92. DQMOM Values Produced From a PDF File 2805
22.93. Specifying Inlet Boundary Conditions for the Population Balance Model 2808
22.94. The Equations Dialog Box 2810
22.95. Setting the Secondary Phase for Hydrodynamic Coupling 2811
22.96. The Phase Interaction Tab for Non-reacting Species 2813
22.97. The Reactions Tab for a Heterogeneous Reaction 2815
22.98. The Size Calculator Dialog Box 2816
22.99. The Population Balance Moments Dialog Box 2819
22.100. The Number Density Function Dialog Box 2820
22.101. The Multiphase Model Dialog Box with the Wet Steam Model Selected (Pressure-Based) 2831
22.102. The Multiphase Model Dialog Box with the Wet Steam Model Selected (Density-Based) 2831
22.103. The Solution Methods Task Page Displaying The Pressure-Velocity Coupling Options 2846
22.104. The Solution Controls Task Page Displaying the Coupled Volume Fraction Method for the VOF and Mixture Models 2848
22.105. The Solution Controls Task Page Displaying the Coupled Volume Fraction Method for the Eulerian Multiphase Model 2850
22.106. The Stabilization Methods Dialog Box 2852
22.107. The Velocity Limiting Treatment Dialog Box 2853
23.1. The Solidification and Melting Dialog Box 2873
23.2. The Create/Edit Materials Dialog Box for Melting and Solidification 2875
23.3. The Solidification and Melting Dialog Box 2878
23.4. Liquid Fraction Contours for Continuous Crystal Growth 2880
24.1. The Structural Model Dialog Box 2884
24.2. The Structure Tab of the Wall Dialog Box 2885
24.3. The Wall Dialog Box for a Two-Sided Wall 2886
25.1. The Eulerian Wall Film Dialog Box - Model Options and Setup tab (with DPM Interaction) 2896
25.2. The Eulerian Wall Film Dialog Box - Model Options and Setup tab (with Phase Interaction) 2900
25.3. Eulerian Wall Film Solution Controls (Steady Flow) 2902
25.4. Eulerian Wall Film Solution Controls (Unsteady Flow) 2903
25.5. Wall Dialog Box - Boundary Type Tab 2906
25.6. Wall Dialog Box - Sources Terms Tab 2907

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25.7. Wall Dialog Box - Phase Change Tab 2908
25.8. Wall Dialog Box - Surface Contact Tab 2909
25.9. Wall Dialog Box - DPM Interaction Tab 2910
25.10. Wall Dialog Box - VOF Interaction Tab 2912
26.1. The **Potential/Li-ion Battery** Dialog Box - Lithium-ion Battery Model 2921
26.2. The **Potential/Li-ion Battery** Dialog Box - Echem Rate Tab 2922
26.3. The **Potential/Li-ion Battery** Dialog Box - Material Properties Tab 2923
26.4. The **Potential/Li-ion Battery** Dialog Box - Report Tab 2926
27.1. The Battery Model Option in the **Outline View** 2932
27.2. The Battery Model Dialog Box (Model Options Tab) 2934
27.3. The Calendar Life Parameters Dialog Box 2938
27.4. The Capacity Fade Table Dialog Box - Cycle Life 2939
27.5. The Battery Model Dialog Box (Conductive Zones Tab) 2944
27.6. The Battery Model Dialog Box (Electric Contacts Tab) 2945
27.7. Model Parameters Tab—CHT Coupling Method 2948
27.8. Model Parameters Tab—FMU-CHT Coupling Method 2949
27.9. Model Parameters Tab—NTGK Model 2950
27.10. NTGK U-parameter Data Table Dialog Box 2952
27.11. The Import Raw Data Dialog Box 2953
27.12. The Parameter Estimation Dialog Box for the NTGK Model 2954
27.13. Model Parameters Tab—Equivalent Circuit Model 2956
27.14. The Parameter Estimation Dialog Box for the ECM 2958
27.15. The HPPC Data Library Dialog Box 2960
27.16. Overview of the Folder Structure in the HPPC Library 2963
27.17. Model Parameters Tab—Newman's P2D Model 2964
27.18. The Echem Material Database Dialog Box 2967
27.19. Model Parameters Tab—User-Defined E-Model 2969
27.20. The Battery Model Dialog Box (UDF Tab) 2970
27.21. The Battery Model Dialog Box (Advanced Option Tab) 2971
27.22. The Material Database for Abuse Kinetics Dialog Box 2976
28.1. The PEMFC Option in the **Outline View** 2991
28.2. The Model Options in the PEM Fuel Cell Model Dialog Box 2992
28.3. The Parameters Tab of the PEM Fuel Cell Model Dialog Box 2996
28.4. The Anode Tab of the PEM Fuel Cell Model Dialog Box with Current Collector Selected 3000
28.5. The Anode Tab of the PEM Fuel Cell Model Dialog Box with Flow Channel Selected 3001
28.6. The Anode Tab of the PEM Fuel Cell Model Dialog Box with Porous Electrode Selected 3002
28.7. The Anode Tab of the PEM Fuel Cell Model Dialog Box with TPB Layer (Catalyst) Selected 3004
28.8. The Anode Tab of the PEM Fuel Cell Model Dialog Box with Micro Porous Layer Selected 3006
28.9. The Electrolyte Tab of the PEM Fuel Cell Model Dialog Box 3008
28.10. The Cathode Tab of the PEM Fuel Cell Model Dialog Box with TPB Layer (Catalyst) Selected 3010
28.11. The Advanced Tab of the Fuel Cell and Electrolysis Models Dialog Box for Contact Resistivities 3013
28.12. The Advanced Tab of the PEM Fuel Cell Model Dialog Box for Contact Resistivities 3014
28.13. The Advanced Tab of the PEM Fuel Cell Model Dialog Box for the Coolant Channel 3015
28.14. The Advanced Tab of the PEM Fuel Cell Model Dialog Box for Stack Management 3016
28.15. The Reports Tab of the PEM Fuel Cell Model Dialog Box 3018
28.16. The Fuel Cell and Electrolysis Option in the Tree 3032
28.17. The Model Options in the Fuel Cell and Electrolysis Models Dialog Box—PEMFC Enabled 3033
28.18. The Parameters Tab of the Fuel Cell and Electrolysis Models Dialog Box 3036
28.19. The Anode Tab of the Fuel Cell and Electrolysis Models Dialog Box With Current Collector Selected .. 3037
28.20. The Anode Tab of the Fuel Cell and Electrolysis Models Dialog Box With Flow Channel Selected 3038
28.21. The Anode Tab of the Fuel Cell and Electrolysis Models Dialog Box With Porous Electrode Selected .. 3039

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.22. The Anode Tab of the Fuel Cell and Electrolysis Models Dialog Box With TPB Layer (Catalyst) Selected 3040
28.23. The Electrolyte Tab of the Fuel Cell and Electrolysis Models Dialog Box 3041
28.24. The Cathode Tab of the Fuel Cell and Electrolysis Models Dialog Box With Current Collector Selected 3042
28.25. The Cathode Tab of the Fuel Cell and Electrolysis Models Dialog Box With Flow Channel Selected 3043
28.26. The Cathode Tab of the Fuel Cell and Electrolysis Models Dialog Box With Porous Electrode Selected 3044
28.27. The Cathode Tab of the Fuel Cell and Electrolysis Models Dialog Box With TPB Layer (Catalyst) Selected 3045
28.28. The Advanced Tab of the Fuel Cell and Electrolysis Models Dialog Box for Contact Resistivities 3046
28.29. The Advanced Tab of the Fuel Cell and Electrolysis Models Dialog Box for the Coolant Channel 3047
28.30. The Advanced Tab of the Fuel Cell and Electrolysis Models Dialog Box for Stack Management 3048
28.31. The Reports Tab of the Fuel Cell and Electrolysis Models Dialog Box 3050
28.32. The Electric Conductivity Field in the Create/Edit Materials Dialog Box 3051
28.33. Opening the SOFC Model Dialog Box in the **Outline View** 3062
28.34. The Model Parameters Tab in the SOFC Model Dialog Box 3066
28.35. The Electrochemistry Tab in the SOFC Model Dialog Box 3069
28.36. The Electrolyte and Tortuosity Tab in the SOFC Model Dialog Box 3071
28.37. The Electric Field Tab in the SOFC Model Dialog Box 3073
29.1. Enabling the MHD Model Dialog Box 3084
29.2. The MHD Model Dialog Box 3084
29.3. The MHD Model Dialog Box for Patching an External Magnetic Field 3086
29.4. The MHD Model Dialog Box for Specifying a Moving Field 3087
29.5. The MHD Model Dialog Box for Importing an External Magnetic Field 3088
29.6. Apply External B0 Field Dialog Box 3089
29.7. Cell Boundary Condition Setup 3090
29.8. Editing Material Properties within Boundary Condition Setup 3091
29.9. Wall Boundary Condition Setup 3092
29.10. Conducting Wall Boundary Conditions in Electrical Potential Method 3093
29.11. Solution Control Tab in the MHD Model Dialog Box 3094
30.1. The Fiber Model Dialog Box 3108
30.2. The Fiber Injections Dialog Box 3115
30.3. The Set Fiber Injection Properties Dialog Box 3119
30.4. The Set Fiber Injection Properties Dialog Box With Take-Up Point Properties 3121
30.5. Line Injections 3122
30.6. Matrix Injections 3123
30.7. Equidistant Fiber Grid 3123
30.8. One-Sided Fiber Grid 3124
30.9. Two-Sided Fiber Grid 3124
30.10. Three-Sided Fiber Grid 3125
30.11. Defining a Three-Sided Fiber Grid Using the Set Fiber Injection Properties Dialog Box 3125
30.12. The Fiber Model Dialog Box 3132
30.13. Fiber Solution Controls Dialog Box 3133
30.14. Displaying Fiber Locations Using the Contours Dialog Box 3135
30.15. The Fiber Mesh Display Dialog Box 3136
30.16. The Fiber Style Attributes Dialog Box 3137
30.17. The Fiber Display Dialog Box 3140
31.1. The Reduced Order Model Dialog Box 3145
31.2. The Write Profile Dialog Box 3151
32.1. The General Task Page 3156

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.2. The Solution Methods Task Page for the HOTR Option 3161
32.3. The Relaxation Options Dialog Box 3162
32.4. The Solution Methods Task Page for the Pressure-Based Segregated Algorithm 3164
32.5. The Solution Controls Task Page for the Pressure-Based Solver 3171
32.6. The Advanced Solution Controls Dialog Box for the Pressure-Based Segregated Non-Iterative Solver . 3174
32.7. The NITA Options Dialog Box 3176
32.8. The Solution Controls Task Page for the Density-Based Explicit Formulation 3183
32.9. The Solution Methods Task Page for the Density-Based Implicit Formulation 3185
32.10. The Multigrid Tab 3192
32.11. The Advanced Solution Controls Dialog Box 3200
32.12. The Solution Limits Dialog Box 3202
32.13. The Multi-Stage Tab 3205
32.14. The Solution Initialization Task Page 3209
32.15. The Patch Dialog Box 3211
32.16. The Solution Initialization Task Page for Hybrid Initialization 3215
32.17. The Hybrid Initialization Dialog Box 3216
32.18. The Run Calculation Task Page 3219
32.19. The Solution Methods Task Page 3224
32.20. The Solution Controls Task Page for the Pseudo Transient Runs 3225
32.21. The Advanced Solution Controls Dialog Box for the Pseudo Transient Method 3227
32.22. The Run Calculation Task Page for the User-Specified Pseudo Transient Time Step Method 3228
32.23. The Run Calculation Task Page for the Automatic Pseudo Transient Option 3230
32.24. Time-Dependent Calculation of Vortex Shedding (t=36.6 sec) 3231
32.25. Time-Dependent Calculation of Vortex Shedding (t=41.6 sec) 3232
32.26. The General Task Page for a Transient Calculation 3233
32.27. The Solution Methods Task Page for a Transient Calculation 3234
32.28. The Run Calculation Task Page for Implicit Transient Calculations 3238
32.29. The Zone-Specific Sampling Options Dialog Box 3243
32.30. The Sampling Options Dialog Box 3245
32.31. Lift Coefficient Plot for a Time-Periodic Solution 3255
32.32. The Residual Monitors Dialog Box 3260
32.33. The Residual Monitors Dialog Box with Advanced Options Shown 3261
32.34. The Residual Monitors Dialog Box Displaying Relative or Absolute Convergence 3264
32.35. Report File for 'flow-time', 'delta-time', and 'iters-per-timestep' 3268
32.36. Fluctuating Simulation Example 3271
32.37. The Execute Commands Dialog Box 3273
32.38. The Define Macro Dialog Box 3275
32.39. The Automatically Initialize Solution and Modify Case Option 3277
32.40. The Automatic Solution Initialization and Case Modification Dialog Box 3278
32.41. The Case Modification Tab 3279
32.42. The Run Calculation Task Page 3281
32.43. The Edit Automatic Initialization and Case Modifications Dialog Box 3282
32.44. The Animation Definition Dialog Box 3283
32.45. The Playback Dialog Box 3285
32.46. The Video Options Dialog Box 3290
32.47. The Advanced Video Quality Options Dialog Box 3291
32.48. The Case Check Dialog Box 3292
32.49. The Information Dialog Box 3292
32.50. The Mesh Tab in the Case Check Dialog Box 3294
32.51. The Models Tab in the Case Check Dialog Box 3296
32.52. The Boundaries and Cell Zones Tab in the Case Check Dialog Box 3298

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.53. The Materials Tab in the Case Check Dialog Box 3301
32.54. The Solver Tab in the Case Check Dialog Box 3302
32.55. The Run Calculation Task Page with Solution Steering Enabled 3315
32.56. The Solution Steering Dialog Box 3317
32.57. The FMG Settings Tab in the Solution Steering Dialog Box 3318
33.1. Turbine Cascade Mesh Before Adaption 3320
33.2. Turbine Cascade Mesh after Adaption 3321
33.3. The Manual Mesh Adaption Dialog Box 3323
33.4. The Automatic Mesh Adaption Dialog Box 3324
33.5. The General Adaption Controls Dialog Box 3325
33.6. Additional Refinement Layers: 1, 2, 3 3327
33.7. The Manage Adaption Criteria Dialog Box 3328
33.8. The Predefined Criteria Drop-Down List 3329
33.9. The Field Variable Register Dialog Box for the Refinement Criterion 3330
33.10. The Predefined Criteria Drop-Down List 3331
33.11. The Adaption Criteria Settings Dialog Box for the Flame Indicator Criterion 3332
33.12. The Expression Dialog Box for the Refinement Criterion 3333
33.13. The Field Variable Register Dialog Box for the Refinement Expression 3334
33.14. The Predefined Criteria Drop-Down List 3335
33.15. The Adaption Criteria Settings Dialog Box for the VOF-to-DPM [Advanced] Criterion 3335
33.16. The Expression Dialog Box for the Refinement Criterion 3336
33.17. The Field Variable Register Dialog Box for the Refinement Criterion 3337
33.18. Marking Boundary Cells 3338
33.19. Mesh Before Adaption 3339
33.20. Mesh after Boundary Adaption 3340
33.21. Wing Mesh Before Adaption 3341
33.22. Marking Cells Based on Region 3342
33.23. Wing Mesh After Region Cell Register-Based Adaption 3343
33.24. Field Variable Refinement Register 3344
33.25. Field Variable Coarsening Register 3344
33.26. Controls for Refining and Coarsening the Mesh 3345
33.27. Simple Expression Refinement Setting 3345
33.28. Cells Marked for Simple Expression Refinement 3346
33.29. Advanced Expression Refinement Setting 3346
33.30. Cells Marked for Advanced Expression Refinement 3347
33.31. The Anisotropic Adaption Dialog Box 3349
33.32. The Geometry Based Adaption Dialog Box 3351
33.33. The Geometry Based Adaption Controls Dialog Box 3351
34.1. The Zone Surface Dialog Box 3355
34.2. Contours of Cell Partitions on Partition Surface Overlaid on Mesh 3357
34.3. The Partition Surface Dialog Box 3357
34.4. The Imprint Surface Dialog Box 3359
34.5. Imprinted Surface (pink) Superimposed Over Imported Surface (white) 3360
34.6. The Point Surface Dialog Box 3361
34.7. The Point Tool 3362
34.8. Using the Point Tool with Front Faces Transparent 3363
34.9. The Structural Point Surface Dialog Box 3364
34.10. The Line/Rake Surface Dialog Box 3366
34.11. The Line Tool 3368
34.12. The Plane Surface Dialog Box 3370
34.13. The Plane Tool for the YZ Plane Method 3373

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.14. The Plane Tool for the Point and Normal Method 3374
34.15. The Plane Tool for the Three Points Method 3374
34.16. The Plane Tool for Creating Multiple Planes 3375
34.17. The Quadric Surface Dialog Box 3376
34.18. The Iso-Surface Dialog Box 3378
34.19. External Wall Surface Isoclipped to Values of x Coordinate 3380
34.20. The Iso-Clip Dialog Box 3381
34.21. The Transform Surface Dialog Box 3383
34.22. The Surfaces Dialog Box 3385
34.23. Region Register Dialog Box 3389
34.24. Boundary Register Dialog Box 3391
34.25. Limit Register Dialog Box 3392
34.26. Field Variable Register Dialog Box 3394
34.27. Residual Register Dialog Box 3396
34.28. Volume Change—Ratio of the Volumes of the Cells 3397
34.29. Volume Register Dialog Box 3397
34.30. Airfoil Wall Cells Marked by Y+ Values 3399
34.31. Yplus/Ystar Register Dialog Box 3400
34.32. Manage Cell Registers Dialog Box 3401
34.33. Report Register Dialog Box 3402
34.34. Manage Register Operations Dialog Box 3403
35.1. Performance Preferences 3407
35.2. Outline Display 3408
35.3. Mesh Edge Display 3409
35.4. Mesh Face (Filled Mesh) Display 3410
35.5. Node Display 3411
35.6. The Mesh Display Dialog Box 3412
35.7. The Mesh Colors Dialog Box (Classic Color Scheme) 3414
35.8. The Mesh Colors Dialog Box (Pastel Color Scheme) 3414
35.9. Mesh Display with Pastel Color Scheme 3415
35.10. Jeep with Multiple Materials Assigned 3416
35.11. Standard Outline of Complex Duct 3419
35.12. Feature Outline of Complex Duct 3420
35.13. Mesh Display with Shrink Factor = 0 3421
35.14. Mesh Display with Shrink Factor = 0.01 3421
35.15. Contours of Static Pressure 3422
35.16. Profile Plot of y Velocity 3423
35.17. Coloring Surfaces by Static Pressure 3424
35.18. The Contours Dialog Box 3425
35.19. The Profile Options Dialog Box 3426
35.20. Filled Contours of Static Pressure 3428
35.21. Filled Contours with Clip to Range On 3429
35.22. Filled Contours with Clip to Range Off 3430
35.23. Velocity Vector Plot 3434
35.24. The Vectors Dialog Box 3435
35.25. The Vector Options Dialog Box 3437
35.26. Velocity Vectors Generated Using the In Plane Option 3438
35.27. The Custom Vectors Dialog Box 3441
35.28. The Vector Definitions Dialog Box 3442
35.29. Pathline Plot 3443
35.30. The Pathlines Dialog Box 3444

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.31. The Scene Dialog Box 3453
35.32. The Sweep Surface Dialog Box 3455
35.33. The Create Surface Dialog Box 3456
35.34. Example of Embedded Windows 3458
35.35. Embedding a Window 3461
35.36. Exploded Scene Display of Temperature and Velocity 3464
35.37. Outline View 'Display in' Example 3465
35.38. The Select Window Dialog Box 3465
35.39. Boundary Markers on an Inlet and Outlet 3466
35.40. Graphics Window with Text Annotation 3469
35.41. The Annotate Dialog Box 3470
35.42. The Colormap Dialog Box 3472
35.43. The Colormap Quick-Edit Panel 3472
35.44. The Colormap with Skipped Labels 3488
35.45. The Colormap Editor Dialog Box 3489
35.46. Double-Click to Add Color Stops 3490
35.47. The Lights Dialog Box 3493
35.48. Model with Reflections Enabled 3499
35.49. Model with Static Shadows Enabled 3500
35.50. Model with Dynamic Shadows Enabled 3501
35.51. Model with Ground Plane Grid Displayed 3502
35.52. Model with All Graphics Effects Enabled 3503
35.53. The Navigation Branch of Preferences 3504
35.54. The Display States Dialog Box 3508
35.55. Using the Triad to Change the Orientation of the Object 3509
35.56. The Views Dialog Box 3510
35.57. The Camera Parameters Dialog Box 3511
35.58. Zooming In (Magnifying the Display) 3513
35.59. Zooming Out (Shrinking the Display) 3514
35.60. Camera Definition 3515
35.61. The Write Views Dialog Box 3517
35.62. Mirroring Across a Symmetry Boundary 3518
35.63. The Views Dialog Box 3518
35.64. Before Applying Periodicity 3519
35.65. After Applying Periodicity 3519
35.66. The Graphics Periodicity Dialog Box 3520
35.67. The Mirror Planes Dialog Box 3521
35.68. The Scene Description Dialog Box 3522
35.69. The Display Properties Dialog Box 3524
35.70. Velocity Vectors Translated Outside the Domain for Better Viewing 3526
35.71. The Transformations Dialog Box 3527
35.72. The Iso-Value Dialog Box 3528
35.73. The Pathline Attributes Dialog Box 3529
35.74. Graphics Display with Bounding Frame 3530
35.75. The Bounding Frame Dialog Box 3530
35.76. The Animate Dialog Box 3532
35.77. Sample XY Plot 3538
35.78. Sample Histogram 3539
35.79. Enabling Enhanced Plots 3540
35.80. The Solution XY Plot Dialog Box 3542
35.81. Geometry Used for XY Plot 3544

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

35.82. Data Plotted at Outlet Using a Plot Direction of (1,0,0) 3544
35.83. Data Plotted at Outlet Using a Plot Direction of (0,1,0) 3545
35.84. The Plot Data Sources Dialog Box 3547
35.85. The Plot Profile Data Dialog Box 3549
35.86. The Plot Interpolated Data Dialog Box 3550
35.87. Iso-Clips Created For Circumferential Averaging 3551
35.88. XY Plot of Circumferential Averages 3552
35.89. The Histogram Dialog Box 3554
35.90. The Axes Dialog Box 3556
35.91. The Curves Dialog Box 3558
35.92. The Fourier Transform Dialog Box 3563
35.93. The Plot/Modify Input Signal Dialog Box 3564
35.94. A-, B-, and C-Weighting Functions 3568
35.95. The VRXPERIENCE Sound Analysis Dialog Box 3571
35.96. The Cumulative Plot Dialog Box 3574
36.1. Report Definitions Dialog Box 3581
36.2. Surface Report Definition Dialog Box 3583
36.3. Volume Report Definition Dialog Box 3584
36.4. Force Report Definition Dialog Box 3586
36.5. Drag Report Definition Dialog Box 3587
36.6. Lift Report Definition Dialog Box 3588
36.7. Moment Report Definition Dialog Box 3589
36.8. Flux Report Definition Dialog Box 3591
36.9. DPM Source Report Definition Dialog Box 3592
36.10. Aerodamping Report Dialog Box 3593
36.11. DPM Report Definition Dialog Box 3595
36.12. User Defined Report Definition Dialog Box 3597
36.13. Expression Report Definition Dialog Box 3598
36.14. New Report File Dialog Box 3600
36.15. Report File Definitions Dialog Box 3601
36.16. Edit Report File Dialog Box 3602
36.17. New Report Plot Dialog Box 3602
36.18. Report Plot Definitions Dialog Box 3604
36.19. Edit Report Plot Dialog Box 3604
36.20. Plot with Instantaneous and Averaged Values 3606
36.21. The Flux Reports Dialog Box 3612
36.22. The Save Output Parameter Dialog Box 3613
36.23. The Flux Reports Dialog Box 3615
36.24. The Flux Reports Dialog Box with DPM 3616
36.25. The Force Reports Dialog Box 3618
36.26. An Airfoil with its Computed Center of Pressure 3620
36.27. The Force Reports Dialog Box for a Center of Pressure Report 3621
36.28. The Projected Surface Areas Dialog Box 3622
36.29. The Surface Integrals Dialog Box 3623
36.30. The Volume Integrals Dialog Box 3625
36.31. The Reference Values Task Page 3628
36.32. The Modified Settings Summary Table 3630
36.33. The Input Summary Dialog Box 3631
37.1. Computing Node Values 3636
37.2. Cylindrical Velocity Components in 3D, 2D, and Axisymmetric Domains 3638
37.3. The Custom Field Function Calculator Dialog Box 3739

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.4. The Field Function Definitions Dialog Box 3741
38.1. Ansys Fluent Architecture 3746
38.2. The Parallel Settings Tab of Fluent Launcher 3749
38.3. The Scheduler Tab of Fluent Launcher (Windows 64 Version) 3751
38.4. The Remote Tab of Fluent Launcher 3754
38.5. Partitioning the Mesh 3768
38.6. The Auto Partition Mesh Dialog Box 3769
38.7. The Partitioning and Load Balancing Dialog Box 3771
38.8. The Weighting Tab in the Partitioning and Load Balancing Dialog Box 3774
38.9. The Partitioned Mesh 3777
38.10. The Partitioned ID Set to Zero 3778
38.11. The Partitioned ID Set to 1 3779
38.12. The Dynamic Load Balancing Tab 3781
38.13. Partitions Created with the Cartesian Axes Method 3785
38.14. Partitions Created with the Cartesian Strip or Cartesian X-Coordinate Method 3786
38.15. Partitions Created with the Principal Axes Method 3786
38.16. Partitions Created with the Principal Strip or Principal X-Coordinate Method 3787
38.17. Partitions Created with the Polar Axes or Polar Theta-Coordinate Method 3787
38.18. The Smooth Optimization Scheme 3788
38.19. The Merge Optimization Scheme 3788
38.20. The Thread Control Dialog Box 3796
38.21. The Parallel Connectivity Dialog Box 3797
39.1. Accessing the Simulation Report Outline Task Page 3806
39.2. The Simulation Report Outline Task Page 3807
39.3. The Report Sections Outline View 3808
39.4. Drag-and-Drop Rearrangement of Nodes 3810
39.5. Drag-and-Drop Onto a Node 3810
39.6. Drag and Drop Indicators 3810
39.7. Positioning Between Two Nodes 3810
39.8. A Generated Report 3812
39.9. An Example of the System Information Report Section 3813
39.10. An Example of the Geometry and Mesh Report Section (Mesh Quality) 3814
39.11. An Example of the Geometry and Mesh Report Section (Orthogonal Quality) 3815
39.12. An Example of the Physics / Models Settings 3816
39.13. An Example of the Physics / Material Properties Settings 3816
39.14. An Example of the Physics / Cell Zone Conditions Section 3817
39.15. An Example of the Physics / Boundary Conditions Section 3818
39.16. An Example of the Physics / Reference Values Section 3819
39.17. An Example of the Solver Settings Section 3819
39.18. An Example of the Run Information Section 3820
39.19. An Example of the Solution Status Section 3820
39.20. An Example of the Named Expressions Section 3821
39.21. An Example of the Report Definitions Section 3821
39.22. An Example of the Plots Section (Residuals) 3822
39.23. Hovering and Showing Data in a Plot 3823
39.24. Hovering and Comparing Data in a Plot 3823
39.25. Example of Contours 3824
39.26. Example of Vectors 3824
39.27. Example of Pathlines 3825
39.28. Example of XY Plots 3826
39.29. Example of a Scene 3826

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

39.30. Example of an Animation 3827
39.31. Layout Settings for Result Plots 3827
39.32. Example of Contours (Columns) 3828
40.1. Adjoint Observables Dialog Box 3853
40.2. Create New Observable Dialog Box (Observable Types) 3854
40.3. Create New Observable Dialog Box (Operation Types) 3855
40.4. Manage Adjoint Observables Dialog Box 3856
40.5. Adjoint Observables Dialog Box 3859
40.6. Adjoint Solution Methods Dialog Box 3860
40.7. The Stabilized Strategy and Scheme Settings Dialog Box 3866
40.8. The Dissipation Scheme Settings 3868
40.9. The Residual Minimization Scheme Settings 3869
40.10. Adjoint Residual Monitors Dialog Box 3875
40.11. Run Adjoint Calculation Dialog Box 3877
40.12. Adjoint Autosave Dialog Box 3878
40.13. Adjoint Reporting Dialog Box 3886
40.14. Design Tool Dialog Box 3887
40.15. A Cylindrical Region 3890
40.16. The Design Condition Display Options Dialog Box 3894
40.17. Specifying a Bounding Plane for Design Changes 3895
40.18. The Bounding Orientation Dialog Box 3897
40.19. The Strict Conditions Dialog Box 3905
40.20. The Design Export Dialog Box 3907
40.21. The Preview Morphing Dialog Box 3908
40.22. The Mesh History Dialog Box 3909
40.23. The Export STL Dialog Box 3910
40.24. Gradient-Based Optimizer Dialog Box 3912
40.25. Adjoint Optimizer Observables Dialog Box 3913
40.26. Adjoint Optimizer Conditions Dialog Box 3913
40.27. Adjoint Optimization History Monitor Dialog Box 3916
40.28. Adjoint Optimizer Autosave Dialog Box 3918
40.29. The Regions Tab of the Mesh Morpher/Optimizer Dialog Box 3926
40.30. The Regions Tab of the Mesh Morpher/Optimizer Dialog Box for an Unstructured Distribution 3928
40.31. Displaying the Control Points for a Regular Distribution 3930
40.32. The Define Control Points Dialog Box 3932
40.33. Displaying the Control Points for an Unstructured Distribution 3934
40.34. The Constraints Tab of the Mesh Morpher/Optimizer Dialog Box 3936
40.35. The Deformation Tab of the Mesh Morpher/Optimizer Dialog Box 3937
40.36. The Parameter Bounds Dialog Box 3938
40.37. The Motion Settings Dialog Box for a Regular Distribution 3939
40.38. The Motion Settings Dialog Box for an Unstructured Distribution 3942
40.39. The Optimizer Tab of the Mesh Morpher/Optimizer Dialog Box 3945
40.40. The Objective Function Definition Dialog Box 3946
40.41. The Optimization History Monitor Dialog Box 3949
41.1. Exporting System Coupling Files from Workbench 3952
41.2. Force transferred to System Coupling when Porous Jump Thickness is Non-Zero 3959
41.3. Undeformed mesh 3965
41.4. Deformed mesh with diffusion smoothing 3965
41.5. Deformed mesh with linearly elastic smoothing and non-FSI boundary set to Unspecified 3966
41.6. Skewed prism cells due to translational motion between two bodies in close proximity 3968
41.7. Example geometry 3969

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

41.8. Cell Zone containing the inner boundary layer mesh 3970
41.9. Cell Zone containing the outer boundary layer mesh 3970
41.10. Cell Zone containing the interior mesh 3971
41.11. Prism layer mesh quality maintained for large deformations 3972
41.12. Overset mesh generated using 3 separate meshes 3973
41.13. Prism layer mesh quality is maintained 3973
41.14. Original Mesh (left) and deformed mesh (right) with smoothing and remeshing enabled 3975
41.15. Original Mesh (left) and deformed mesh (right) using smoothing and Region Face Remeshing 3976
43.1. The Basic Shapes Dialog Box 4166
43.2. Integration Surface Dialog Box 4167
43.3. Orientation Calculator Dialog Box 4430
43.4. The Acoustics Initialization Dialog Box 4462
1. The CFF Case File Layout 4892
2. The CFF Data File Layout 4893
3. Variable Sized Data Appended to the Case File 4894
4. Variable Sized Data Appended to the Data File 4895
5. Quadrilateral Mesh 4905
6. Quadrilateral Mesh with Periodic Boundaries 4906
7. Quadrilateral Mesh with Hanging Nodes 4907
1.1. Progress Bar with Start Server Option 4946
1.2. Progress Bar with Start Client Option 4947
1.3. Preferences Dialog Box 4950
1.4. Actions Ribbon Tab 4951
1.5. Run Calculation Properties 4951
1.6. Example Solution Methods 4952
1.7. Example Solution Controls 4952
1.8. Example Residuals Properties 4953
1.9. Creating a Contour Via the Context Menu 4954
1.10. Properties of a Point Surface 4955
1.11. Properties of a Line Surface 4956
1.12. Properties of a Rake Surface 4957
1.13. Properties of a Plane Surface 4959
1.14. Create Multiple Planes Dialog Box 4961
1.15. Properties of an Iso-Surface 4962
1.16. Create Multiple Iso-Surfaces Dialog Box 4964
1.17. Example Graphics Object Properties 4967
1.18. Example Plot Object Properties 4968
1.19. Scene Properties 4969
1.20. Scene Dialog Box 4969
1.21. Viewing Ribbon Tab 4970
1.22. Example of Sending a Command: Changing the Velocity Units to cm/s 4972
1.23. Writing Case and/or Data from the Client 4973
2.1. The Fluent Icing Graphical User Interface 4987
2.2. Hierarchical Structure of a Project Folder 4993
2.3. Pressure Far Field and Velocity Inlet Properties 5025
2.4. Mass Flow Inlet and Pressure Inlet Properties 5026

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

List of Tables

1.1. Supported Versions of Third-Party Software 151
4.1. Available Command Line Options for Linux and Windows Platforms 175
23.1. Default Style Attributes 724
24.1. Skewness Ranges and Cell Quality 743
5. Mini Flow Chart Symbol Descriptions ..... dccxcvii
3.1. CGNS Variables Supported by Ansys Fluent 869
3.2. FEA File Extensions for FSI Mapping 908
3.3. Units Associated with the Temperature Units Drop-Down List Selections 912
5.1. Operations and Functions 932
5.2. Solution Variables 939
5.3. Scientific Constants 940
5.4. Variable Aliases 940
7.1. Zone Types by Category 1174
7.2. Air-side Radiator Data 1391
7.3. Reduced Radiator Data 1391
7.4. **CSV Profile Section Identifiers** 1438
7.5. Profile Types and the Corresponding Required Field Labels 1438
8.1. Recommended Settings for Operating Pressure 1547
8.2. Temperature Limits for Droplet Materials in Ansys Fluent Database prodb.scm 1568
8.3. Fluids Supported by REFPROP v9.1 1571
14.1. NTU Model Vs. Simple Effectiveness Model 2032
15.1. Modified Specific Heat Capacity (Cp) Polynomial Coefficients 2118
19.1. Source Data Saved in Source Data Files 2348
20.1. Property Inputs for Inert Particles 2499
20.2. Property Inputs for Droplet Particles 2500
20.3. Property Inputs for Combusting Particles (Laws 1–4) 2501
20.4. Property Inputs for Combusting Particles (Law 5) 2502
20.5. Property Inputs for Multicomponent Particles (Law 7) 2502
20.6. Common Mean Diameters and Their Fields of Application 2559
22.1. Spatial Discretization Schemes for the VOF and Eulerian with Multi-Fluid VOF Models 2601
22.2. Spatial Discretization Schemes for the Eulerian Model without Multi-Fluid VOF 2601
22.3. Spatial Discretization Schemes for the Mixture Model 2601
22.4. Phase-Specific and Mixture Conditions for the VOF Model 2641
22.5. Phase-Specific and Mixture Conditions for the Mixture Model 2643
22.6. Phase-Specific and Mixture Conditions for the Eulerian Model (for Laminar Flow) 2646
22.7. Phase-Specific and Mixture Conditions for the Eulerian Model (with the Mixture Turbulence Model) .. 2647
22.8. Phase-Specific and Mixture Conditions for the Eulerian Model (with the Dispersed Turbulence Model) 2647
22.9. Phase-Specific and Mixture Conditions for the Eulerian Model (with the Per-Phase Turbulence Model) 2648
22.10. Open Channel Boundary Parameters for the VOF Model 2655
22.11. Slope Limiter Discretization Scheme 2691
22.12. Parameters for the Coalescence and Breakage Kernels 2717
22.13. Parameters for the Coalescence and Breakage Kernels 2738
22.14. Macros for Population Balance Variables Defined in `sg_pb.h` 2821
28.1. User-Defined Scalar Allocations 3021
28.2. User-Defined Memory Allocations 3021
28.3. User-Defined Scalar Allocations 3053
28.4. User-Defined Memory Allocations 3054

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.5. User-Defined Memory Allocations 3064
28.6. User-Defined Scalar Allocations 3065
29.1. User-Defined Scalars in MHD Model 3085
29.2. MHD Vectors 3096
30.1. Source Terms and Corresponding UDFs 3107
35.1. Standard Views 3509
35.2. Numbers of Data Points Supported by the Prime-Factor FFT Algorithm 3562
35.3. Octave Band Frequencies and Weightings 3569
37.1. **Expressions** Category 3643
37.2. **Pressure** and **Density** Categories 3643
37.3. **Velocity** Category 3644
37.4. **Temperature**, **Radiation**, **Solidification/Melting**, and **Two-Temperature Model** Categories 3645
37.5. **Turbulence** Category 3647
37.6. **Species**, **Reactions**, **Pdf**, and **Premixed Combustion** Categories 3649
37.7. **NOx**, **Soot**, and **Steady|Unsteady Statistics** Categories 3652
37.8. **Phases**, **Discrete Phase Model**, **Granular Pressure**, **Granular Temperature**, and **Wall Film** Categories 3654
37.9. **Properties** Category 3657
37.10. **Eulerian Wall Film** Category 3658
37.11. **Sensitivities** Category 3659
37.12. **Wall Fluxes**, **User Defined Scalars**, and **User Defined Memory** Categories 3661
37.13. **Cell Info** and **Mesh** Categories 3662
37.14. **Perforated Walls** Category 3665
37.15. **Mesh** Category (Turbomachinery-Specific Variables) 3665
37.16. **Residuals** Category 3665
37.17. **Derivatives** Category 3666
37.18. **Potential** Category 3667
37.19. **Lithium** Category 3667
37.20. **Acoustics** Category 3667
37.21. **Structure** Category 3668
38.1. Examples for GPGPUs per Machine 3748
38.2. Supported Interconnects for the Windows Platform 3760
38.3. Available MPIs for Windows Platforms 3760
38.4. Supported MPIs for Windows Architectures (Per Interconnect) 3760
38.5. Supported Interconnects for Linux Platforms (Per Platform) 3765
38.6. Available MPIs for Linux Platforms 3765
38.7. Supported MPIs for Linux Architectures (Per Interconnect) 3766
41.1. Variables On Boundary Wall Regions 3957
41.2. Variables On Cell Zone Regions 3957
41.3. Variables On Porous Jump Boundary 3957
1. Moving Domain Models vs. Multiphase Models 4887
2. Multiphase Models vs. Turbulence Models 4887
3. Combustion Models vs. Multiphase Models 4887
4. Moving Domain Models vs. Turbulence Models 4888
5. Combustion Models vs. Moving Domain Models 4888
6. Combustion Models vs. Turbulence Models 4888
1. Summary of Basic CHEMKIN-CFD Parameters 4918
2. Summary of Advanced CHEMKIN-CFD Parameters 4919
3. Diagnostic Output Files Created During a CHEMKIN-CFD Run 4922
4. Error Messages that May Be Printed to the Fluent GUI 4924
5. Other Error Messages in `KINetics-log.txt` 4927

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.1. Remote Visualization Client Environment Variables 4978
2.1. Pressure Far-Field, Mapping of Airflow Fluent Icing Boundary Condition into Fluent & FENSAP Boundary Conditions 5026
2.2. Velocity Inlet, Mapping of Airflow Fluent Icing Boundary Condition into Fluent & FENSAP Boundary Conditions 5027
2.3. Mass Flow Inlet, Mapping of Airflow Fluent Icing Boundary Condition into Fluent & FENSAP Boundary Conditions 5028
2.4. Pressure Inlet, Mapping of Airflow Fluent Icing Boundary Condition Into Fluent Boundary Conditions .. 5028

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Part I: Getting Started

The section describes getting started with Ansys Fluent.

- Introduction to Ansys Fluent (p. 129), introduction to Ansys Fluent.
- Basic Steps for CFD Analysis using Ansys Fluent (p. 155), basic steps for CFD analysis using Ansys Fluent.
- Guide to a Successful Simulation Using Ansys Fluent (p. 161), guide to a successful simulation using Ansys Fluent.
- Starting and Executing Ansys Fluent (p. 163), provides instructions for starting and executing Ansys Fluent.
- Glossary of Terms (p. 193), this glossary contains a listing of terms commonly used throughout the Ansys Fluent documentation.

Chapter 1: Introduction to Ansys Fluent

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

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

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

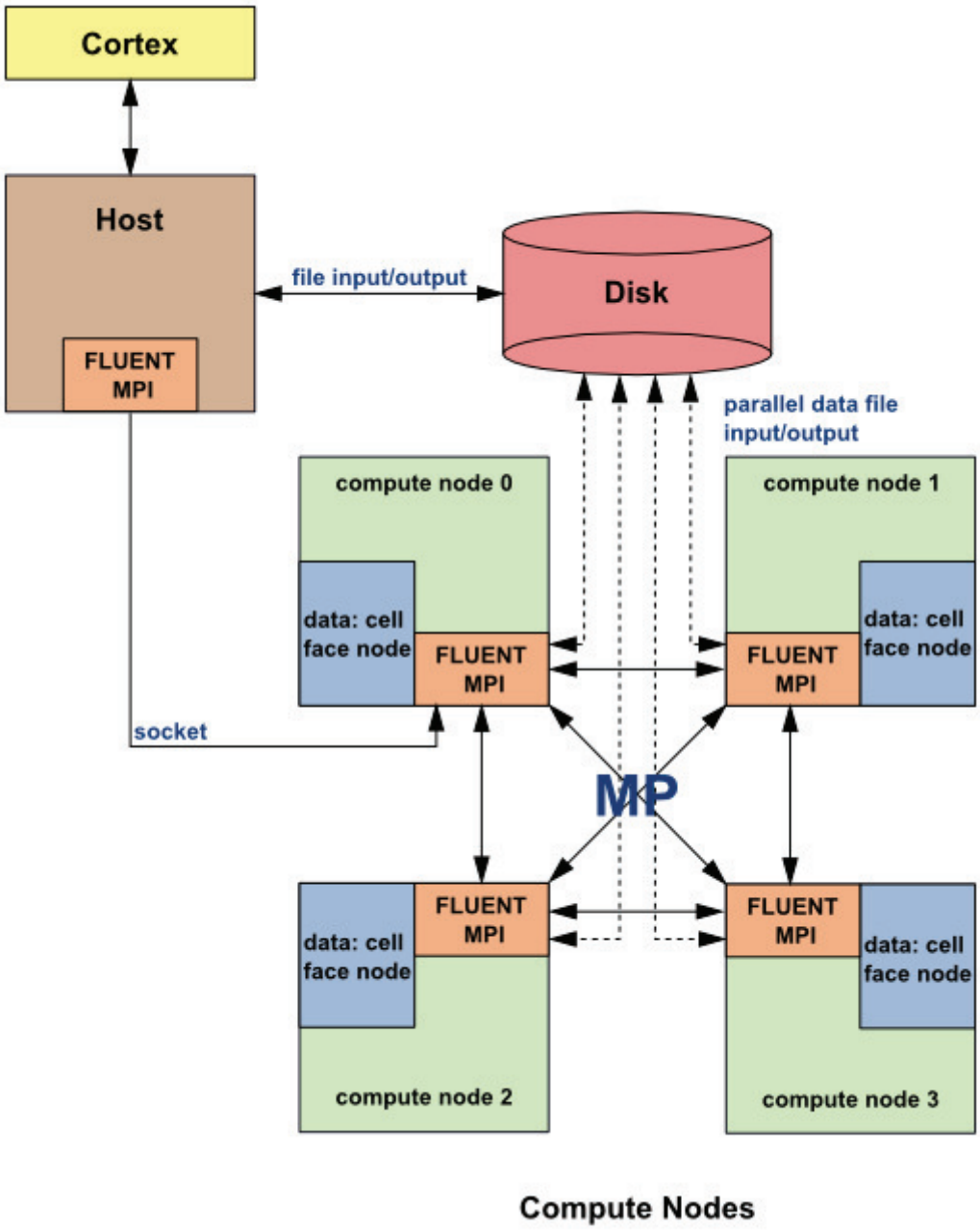
You can read your mesh into Ansys Fluent, or, for 3D geometries, create your mesh using the meshing mode of Fluent (see the Fluent User's Guide (p. 1) for further details). All remaining operations are performed within the solution mode of Fluent, including setting boundary conditions, defining fluid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

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

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

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.1: Ansys Fluent Architecture



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

For more information about Ansys Fluent's parallel processing capabilities, message passing interfaces (MPI), and so on, refer to Parallel Processing (p. 3745) in the User's Guide (p. 1).

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

For more information, see the following sections:

1.1. The Ansys Product Improvement Program

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.2. Program Capabilities
1.3. Known Limitations in Ansys Fluent 2021 R2

1.1. The Ansys Product Improvement Program

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

How to Participate

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

How the Program Works

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

Data We Collect

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

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

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

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

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

Session: Characteristics of the session, such as the:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

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

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

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

Analysis: Characteristics of the analysis, such as the:

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

Solution: Characteristics of the solution performed, including:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

Specialty: Special options or features used, such as:

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

Data We Do Not Collect

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

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

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

Opting Out of the Program

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

The Ansys, Inc., Privacy Policy

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

Frequently Asked Questions

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

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

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

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

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

    Yes. Perform the following steps on the file server:

    a. Navigate to the installation directory: `[Drive:]\v212\commonfiles\globalsettings`

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

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

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.2. Program Capabilities

When in meshing mode, Ansys Fluent functions as a robust unstructured-volume-mesh generator (see Meshing Mode Capabilities (p. 199) in the Fluent User's Guide (p. 1) for further details). When in solution mode, Fluent allows you to simulate the following:

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

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

Ansys Fluent is available at three different licensing levels, which control the availability of the above-mentioned features and functionalities:

- **Enterprise**—full access to all Fluent Meshing and Fluent solver capabilities, as documented in the Fluent User's Guide. Additionally, this licensing level provides access to Ansys EnSight, Polyflow, Ansys FENSAP-ICE, the Model Fuel Library, and the Fluent workspaces.
- **Premium**—full access to all Fluent Meshing and Fluent solver capabilities, as documented in the Fluent User's Guide.
- **Pro**—access to the Water-Tight Meshing workflow in Fluent Meshing and access to a reduced set of Fluent solver capabilities allowing the solution of incompressible and compressible steady-state, single-phase, turbulent, non-reacting flows, and heat transfer. See Ansys Capability Chart 2021 R2 for a comprehensive list of supported functionalities at the CFD-Pro licensing level. *Note that only supported options and functionalities are displayed in the interface.*

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

1.3. Known Limitations in Ansys Fluent 2021 R2

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

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Graphical User Interface

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

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

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

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

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

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

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

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

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

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

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

  – When exporting EnSight Case Gold files for transient simulations, the solver cannot be switched between serial and parallel, and the number of compute nodes cannot be changed for a given parallel run. Otherwise, the exported EnSight Case Gold files for each time step will not be compatible.

  – EnSight export with topology changes is not supported.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

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

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

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

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

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

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

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

- Mesh

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

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

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

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

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

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

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

- Models

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

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

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

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

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

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

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

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

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

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

  (dpm-parameters-changed)
  ```

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

– The shell conduction model is not applicable on moving walls.

– The heat exchanger model is not compatible with mesh adaption.

– The Fluent/Ansys Reaction Design KINetics coupling is not available on the win64 platform.

– DO-Energy coupling is recommended for large optical thickness cases (> 10) only.

– FMG initialization is not available with the shell conduction model.

– FMG initialization is not compatible with the unsteady solver.

– The MHD module is not compatible with Eulerian multiphase models.

– Bounded 2nd order discretization in time is not compatible with moving and deforming meshes.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - When simulating porous media, the value of the **Porosity** (defined in the **Fluid** dialog box) cannot be `0` or `1` (that is, it must be in between these values) if the non-equilibrium thermal model is enabled.

  - When simulating porous media, the non-equilibrium thermal model is not supported with radiation and/or multiphase models.

  - For porous media simulations, the relative velocity resistance formulation is not supported with axisymmetric-swirl when there are non-zero swirl resistances.

  - The junction of a wall with shell conduction enabled and a non-conformal coupled wall is not supported. Such a junction will not be thermally connected, that is, there will be no heat transfer between the shell and the mesh interface wall.

  - After you enable the Eulerian Wall Film model, Fluent will not allow you to save the mesh modifications, such as separating cells, extruding face zones, and changing the cell zones type. If you want to modify the mesh in Fluent, be sure to complete all mesh operations prior to enabling the Eulerian Wall Film model.

  - The Transition SST model (also known as the γ-Re_θ model) is not Galilean invariant and should therefore not be applied to surfaces that move relative to the coordinate system for which the velocity field is computed; for such cases, the Intermittency Transition model (also known as the γ model) should be used instead.

  - In simulations that use the discrete phase model, particle mass may be lost when simulation of transient particles released with constant parcel size is combined with auto-save of case files.

  - User-defined wall functions are not compatible with the Eulerian multiphase formulation and cannot be used.

  - The view factor files generated as part of a surface-to-surface radiation model calculation for version 16.0 or 16.1 of Fluent may not be compatible with newer versions if the **Matching** option was enabled for a mapped interface. For any case file with such a setup, you must recompute the view factors in the newer version to ensure correct results.

- Parallel processing

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

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

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

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

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

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

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

- Message Passing Interfaces (MPIs)
  - For Intel MPI + IB on Mellanox OFED 5.1, users will need to specify `-pib.ofed` instead of just `-pib`. (338029)
  - The IBM MPI has been discontinued by the vendor, and is no longer available in the standard installation of Fluent. The Intel MPI is now the default MPI, and is known to have the following limitations:

    → The Intel MPI version upgrade in Fluent release 2019 R1 and later breaks compatibility with older releases on Windows when running on multiple processes. **Workaround:** To switch between running older releases and the latest release on a cluster, you must make sure that the appropriate services have been configured to run by the administrator as follows:

    - To switch from old to new (for example, 19.2 to 2021 R2), run the following command on all nodes from the command prompt with administrator privileges:

      ```
      "C:\Program Files (x86)\IntelSWTools\compilers_and_libraries_2018.3.210\windows\mpi\intel64\bin\
      hydra_service.exe" -install
      ```

    - To switch from new to old (for example, 2021 R2 to 19.2), run the following command on all nodes from the command prompt with administrator privileges:

      ```
      "C:\Program Files (x86)\IntelSWTools\compilers_and_libraries_2017.4.210\windows\mpi\intel64\bin\
      hydra_service.exe" -install
      ```

    After installing both versions of Intel MPI, the `I_MPI_ROOT` environment variable should be deleted from all nodes.

    (180338 / 292942)

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

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

    ```
    I_MPI_PLATFORM zero
    ```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you encounter issues that cannot be remedied using the supported MPI options, you can add the discontinued IBM MPI to your Fluent installation and use it as a last resort. It is provided as-is, and, going forward, Ansys, Inc. may be unable to address any issues reported about IBM MPI. To set it up, locate your Fluent installation and perform the following steps:

→ For Linux:

1. Download the `IBMMPI.gz` package from the following website: https://release212.s3.amazonaws.com/IBMMPI.TGZ?AWSAccessKeyId=AKIAJ6ZOVHCFGGNWYUKA&Expires=1639840801&Signature=nAHWdM6Fhr9JZ594N9Et%2FDb3Naw%3D
2. Move the package to the following directory where Fluent is installed: `ansys_inc/`.
3. Extract the package using the following command in the `ansys_inc/` directory:

```
gunzip < IBMMPI.gz | tar -xvf -
```

→ For Windows:

1. Download the `IBMMPI.zip` package from the following website: https://release212.s3.amazonaws.com/IBMMPI.zip?AWSAccessKeyId=AKIAJ6ZOVHCFGGNWYUKA&Expires=1639840801&Signature=JBuLZJ26WUB%2B28qItrwMoQ9jEUw%3D
2. Extract the package.
3. Copy the `v212` folder from within the extracted package and paste it in the following folder where Fluent is installed: `ANSYS Inc\` (for example, `C:\Program Files\ANSYS Inc\`). This will overlay files in the existing `v212` folder.

– The following error message may be encountered while postprocessing when using the Microsoft MPI (`msmpi`) on Windows:

```
[0] fatal error
Fatal error in MPI_Recv: Other MPI error, error stack:
MPI_Recv(...) failed
Out of memory
```

Workaround: Use the default Intel MPI (`intel`). (267120)

- Platform support and drivers

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

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

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

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

  – The minimum OS requirements for Linux are SLES 11 SP2 or RHEL 6.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

  - If you are installing Ansys Fluent 2021 R2 on a Windows machine that already has one or more previous versions of Ansys Fluent, then after installing the Intel MPI library from the prerequisites, make sure to delete the environment variable I_MPI_ROOT. Otherwise there will be a conflict while running previous Ansys Fluent versions in parallel mode.

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

  - Itanium platform (lnia64) is no longer supported.

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

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

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

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

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

- Remote display

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

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

  - (Exceed onDemand and VNC Viewer) The **Num Lock** number keys and arrows may not function in the graphical user interface or console. As a workaround, you can define the `XKB_DEFAULT_RULES`

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

environment variable set equal to `base`, which resolves the arrow keys' functionality. Number entries must still be completed using the number keys at the top of the standard keyboard. (172528)

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

    → xcb-util-wm-0.4.1-5.el6.x86_64

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

    → xcb-util-0.4.0-2.2.el6.x86_64

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

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

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

- Cell Zones and Boundary Conditions

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

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

- Reference Frames

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

- Solver

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

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

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

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

  - The following models are not available for the density-based solvers:

    → Volume-of-fluid (VOF) model

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ Multiphase mixture model

→ Eulerian multiphase model

→ Non-premixed combustion model

→ Premixed combustion model

→ Partially premixed combustion model

→ Composition PDF transport model

→ Soot model

→ Rosseland radiation model

→ Melting/solidification model

→ Enhanced Coherent Flamelet model

→ Inert model: transport of inert species (EGR in IC engines)

→ Dense discrete phase model

→ Shell conduction model

→ Floating operating pressure

→ Spark ignition and auto-ignition models

→ Physical velocity formulation for porous media

→ Selective multigrid (SAMG)

– The pressure-based coupled solver is not available with the following features:

→ Fixed velocity

– On some Linux platforms, pressing **Ctrl**+**C** will not interrupt the solution. A suggested workaround is to use the checkpoint mechanism in Fluent to save files and/or exit Fluent. (Checkpointing an Ansys Fluent Simulation in the *Fluent User's Guide* (p. 188))

– In certain cases with tetrahedral or hybrid meshes, the use of the Least-Squares Cell Based gradient method in combination with the cell-to-cell limiter may cause divergence. If this is observed, it is recommended that you either change the gradient method to Green-Gauss Node Based or change the limiter type to the cell-to-face limiter.

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

– For transient and time-dependent cases, the solution advances to the next time step based on various inputs including time step size and convergence criteria, but it is not directly dependent

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

on the flow time interval specified when 'flow-time' is selected for monitoring (report definitions, autosaving, transient export and so on). For these flow time dependent operations, the action will occur when the flow time meets or exceeds the specified flow time interval.

– Note the following issues affecting residuals:

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

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

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

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

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

→ if you newly enable one or both of the following so that both are enabled:

- a reacting flow model and/or the species transport model
- the pseudo transient solution method

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

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

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

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

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

- User-Defined

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

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

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

- User-defined functions (UDFs)

  - Interpreted UDFs cannot be used with an Infiniband interconnect or, when running in parallel, on the Cray platform. The compiled UDF approach must be used instead.

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

- Graphics, Reporting, and Postprocessing

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

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

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

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

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

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

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

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

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

- Mean and root-mean-squared-error (RMSE) quantities of custom field functions are only available for mixtures. In previous releases it was possible to specify these quantities for phases, which was an incorrect behavior. This behavior is no longer allowed in R16.0 or later releases. If you are running a pre-R16.0 case set to output such quantities in R2021 R2, you may get a segmentation error. To avoid the error, redefine the previously defined monitors reporting mean or RMSE quantities of phases.

- The mouse-annotate feature is no longer available. Annotations can still be created using the Annotate dialog box (see Annotate Dialog Box in the *Fluent User's Guide* (p. 4555) for additional information).

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

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

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

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

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

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

– When meshes contain a large number of cells (for example, ~150 million cells or higher), the meshing mode of Fluent may report an incorrect number of skewed cells, based on an incorrect inverse orthoskew (IOS) value. As a workaround, you can use the `(tgapi-util-set-number-of-parallel-compute-threads 1)` command. (168453)

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

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

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

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

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

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

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

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

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

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

– Right-clicking point surfaces to display them from the Outline View tree is not working due to a 3rd party issue. As a workaround, you can display point surfaces using the **Mesh Display** dialog box. (181843)

• Fluent in Workbench

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

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

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

– Fluent cases with either of:

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

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

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

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

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

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Fluent as a Server
  - When running Fluent with the `-aaS` option, if you have a mesh with a very low cell count and have set a large number of iterations to be stored using the **Residual Monitors** dialog box (see Storing Residual History Points in the *Fluent User's Guide* (p. 3262)), you will see a relative degradation of performance. Reducing the number of stored iterations will reduce this degradation.
  - When launching Fluent with the `-gu` or `-g` command line options and Fluent as a server enabled, Fluent will run with the graphic user interface minimized.
- Third-party software
  - Fluent-Platform LSF integration is not supported on the MS Windows platform.
  - Fluent-SGE integration is supported only on Linux platforms.
  - Wave and GT-POWER coupling are available only with stand-alone Ansys Fluent and not in the Workbench environment.
  - Supported versions of third-party software are listed below:

Table 1.1: Supported Versions of Third-Party Software

Third-Party Software	Supported Version
Abaqus	6.14 ODB Library: 6.14.5
Altair HYPERMESH	5.1
AVS	5.0
CGNS	3.3.1
Cray MPI (MPT)	7.0
Data Explorer	4.2
EnSight 6 (TUI only)	Ensight 6
EnSight Case Gold	10.1.6
FAST	1.3
Fieldview	16.0
GT-POWER	Version 2021 Build 1
Hierarchical Data Format version 5 (HDF5)	1.10.5
HOOPS	23.00-1
I-deas	I-deas NX Series 11
Intel MPI	Linux: 2018.3.222 Windows with distributed memory on a cluster: 2018.3.210 Windows with shared memory on a local machine: 2019.8.254
libpng	1.6.18
LSTC-DYNA	970.0

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Third-Party Software	Supported Version
Microsoft MPI	10.0
MPCCI	3.0.5
NASTRAN	Bulk data input file - MSC.NASTRAN 2010 OUTPUT2 data file - NX/NASTRAN 10
NIST	9.1
Open MPI	4.0.5
PATRAN	3.0
PTC MECHANICA	PTC/Mechanica Wildfire 4.0
Sundials	2.5.0
TECPLOT	Tecplot file format, version 11.2
VKI	4.4.6.r1
WAVE	2020.2
zlib	1.2.8

- Other
  - Images of embedded graphics windows captured using the Save Picture dialog box are saved without borders on the embedded windows, regardless of the setting specified in Preferences. (336851)
  - (Operating under the **Pro** license) While the **user-defined** specification method is available for some fields, such as material properties, you cannot load user-defined functions, as they are not supported at the Pro capability level. (321331)
  - Exports of Common Fluids Format - Post files from Fluent may incorrectly specify the pressure gradient as zero. Contact support for a workaround. (239127, 243318)
  - If the network connection is lost during a serial or parallel calculation, the Fluent session may terminate abnormally.
  - The IRIS Image and HPGL hard copy formats are no longer supported in Ansys Fluent.
  - When using Ansys Fluent with the Remote Solve Manager (RSM):

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

    → Maxwell coupling is not supported.

    → UDFs are supported with limitations as detailed in Submitting Fluent Jobs to Remote Solve Manager in Workbench User's Guide.
  - The turbo-averaged contour plot in turbomachinery postprocessing may give an unexpected contour region in a selected topology.
  - The **Inverse Distance** and **Least Squares** profile interpolation methods are not applicable when a profile is attached to cell zones.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - When opening Ansys Help from Fluent in Linux, you may receive an error message in the Linux console. This can result when another user has created the installation and run Fluent, thus creating a registry file; if you then run this same installation, there will be a permissions conflict. As a workaround, remove the registry file:

    path`/ansys_inc/v212/Tools/mono/Linux64/etc/mono/registry`

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

    path`/ansys_inc/v212/Tools/mono/Linux64/etc/mono`

  - On Windows, mesh reading into a serial Ansys Fluent session may fail if you use more than 20 million cells per core.

  - Fluent does not support non-ASCII characters in the names of files, zones, and boundaries.

  - When exiting a Fluent session on Linux that was started with the Load ACT option, Fluent may become unresponsive. If this occurs, the Fluent process must be manually terminated. In the console, the error message specifies that a corrupted double-linked list is responsible for the error. Because this unexpected shutdown occurs after the project has been saved, no data is lost.

  - When Fluent is run on Linux with the Load ACT option, Fluent may repeatedly issue the following warning during solution iteration:

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

    This warning is harmless and does not impact Fluent or ACT usage.

  - When running a stand-alone instance of Fluent in a mixed Windows / Linux configuration or from a remote Windows installation, ACT does not open. To correct the problem, you must set the `AWP_ROOT212` environment variable to point to the Ansys installation directory.

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

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

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

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

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

- Remote Visualization Client

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

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

- Fluent Meshing

  - Limitations related to the Fluent guided meshing workflows are documented separately. See Limitations of the Fluent Guided Workflows (p. 263) for details.

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

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

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

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

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

    → For RHEL, xorg-x11-apps

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

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

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 2: Basic Steps for CFD Analysis using Ansys Fluent

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

For more information, see the following sections:

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

2.1. Steps in Solving Your CFD Problem

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

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

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

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

2.2. Planning Your CFD Analysis

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

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

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ What are your modeling options?

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

→ What simplifying assumptions do you have to make?

→ What simplifying assumptions can you make?

→ Do you require a unique modeling capability?

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

– What degree of accuracy is required?

– How quickly do you need the results?

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

– Where will the computational domain begin and end?

→ Do you have boundary condition information at these boundaries?

→ Can the boundary condition types accommodate that information?

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

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

- **Creating Your Model Geometry and Mesh**

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

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

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

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

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

  → How complex is the geometry and flow?

  → Will you need a non-conformal interface?

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

    → Is the resolution sufficient for the geometry?

    → Can you predict regions with high gradients?

    → Will you use adaption to add resolution?

  - Do you have sufficient computer memory?

    → How many cells are required?

    → How many models will be used?

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

  For a given problem, you will need to:

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

    → Turbulence, combustion, multiphase, and so on.

  - Define material properties.

    → Fluid

    → Solid

    → Mixture

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

- **Computing and Monitoring Your Solution**

  - The discretized conservation equations are solved iteratively.

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

  - Convergence is reached when:

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

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - Residuals provide a mechanism to help monitor this trend.

  → Overall property conservation is achieved.

– The accuracy of a converged solution is dependent upon:

  → Appropriateness and accuracy of physical models.

  → Mesh resolution and independence.

  → Problem setup.

- **Examining and Saving Your Results**

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

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

    → What is the overall flow pattern?

    → Is there separation?

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

    → Are key flow features being resolved?

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

    → Forces and moments

    → Average heat transfer coefficients

    → Surface and volume integrated quantities

    → Flux balances

- **Revising Your Model**

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

  – Are physical models appropriate?

    → Is flow turbulent?

    → Is flow unsteady?

    → Are there compressibility effects?

    → Are there 3D effects?

  – Are boundary conditions correct?

    → Is the computational domain large enough?

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ Are boundary conditions appropriate?

→ Are boundary values reasonable?

– Is the mesh adequate?

→ Can the mesh be adapted to improve results?

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

→ Does boundary resolution need to be improved?

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 3: Guide to a Successful Simulation Using Ansys Fluent

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

1. Examine the quality of the mesh in Fluent.

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

   - Perform a mesh check to avoid problems due to incorrect mesh connectivity, and so on. In particular, you should be sure that the minimum reported cell volume is not negative.
   - Look at maximum cell skewness (for example, using the **Compute** button in the **Contours** dialog box after initializing the model). As a rule of thumb, the skewness should be below 0.98. You can also use the **Report Quality** function to calculate the minimum cell orthogonality. You can find more details about mesh quality considerations in Mesh Quality in the *Fluent User's Guide* (p. 1031).

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

2. Scale the mesh and check length units.

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

3. Employ the appropriate physical models.

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

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

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

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

6. Monitor convergence with residuals history.

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

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

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

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

9. Verify that property conservation is satisfied.

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

10. Check for mesh dependence.

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

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

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

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

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 4: Starting and Executing Ansys Fluent

This chapter provides instructions for starting and executing Ansys Fluent.

4.1. Starting Ansys Fluent

4.2. Running Ansys Fluent in Batch Mode

4.3. Switching Between Meshing and Solution Modes

4.4. Checkpointing an Ansys Fluent Simulation

4.5. Cleaning Up Processes From an Ansys Fluent Simulation

4.6. Exiting Ansys Fluent

4.1. Starting Ansys Fluent

The following sections describe how start Ansys Fluent:

Important:

- Ensure your graphics driver is up-to-date to avoid issues with graphics displays (such as the display coming upside-down).
- If your %HOMEDRIVE% environment variable is set to a network drive and you experience issues such as delays in the Fluent Launcher appearing, add a copy of the preferences file to the %HOMEDRIVE% network location (`%HOMEDRIVE%%HOMEPATH%\.fluent-conf\21.2.0\preferences`). Refer to Setting User Preferences/Options (p. 832) for additional information on the preferences file location.

4.1.1. Selecting the Licensing Level

4.1.2. Starting Ansys Fluent Using Fluent Launcher

4.1.3. Starting Ansys Fluent on a Windows System

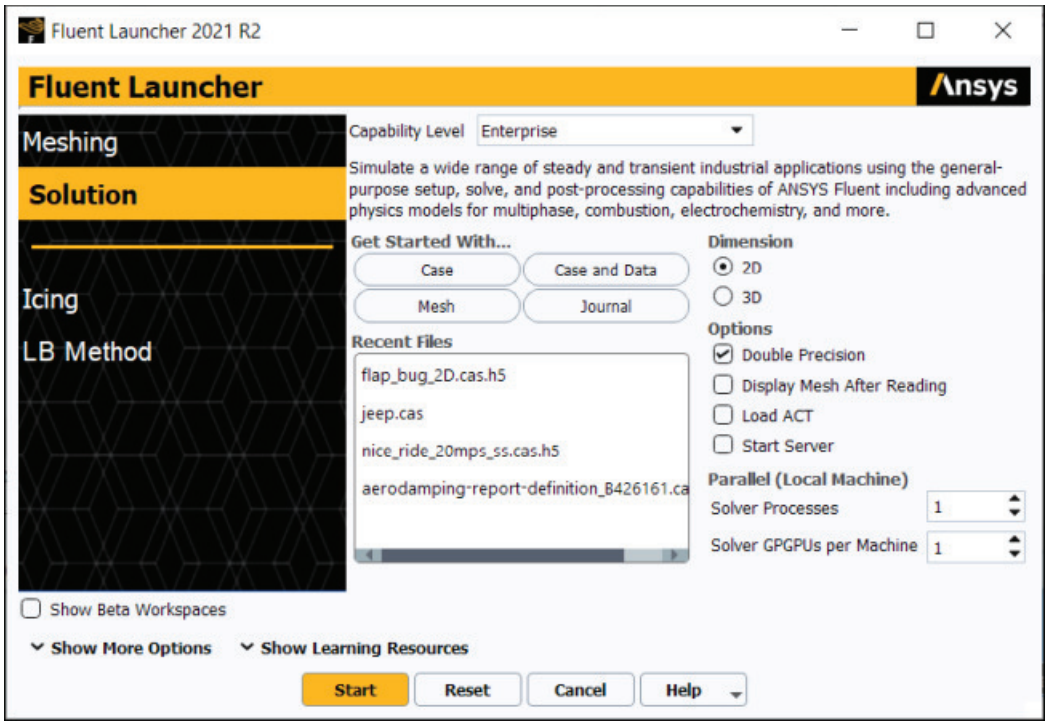
4.1.4. Starting Ansys Fluent on a Linux System

4.1.5. Command Line Startup Options

4.1.1. Selecting the Licensing Level

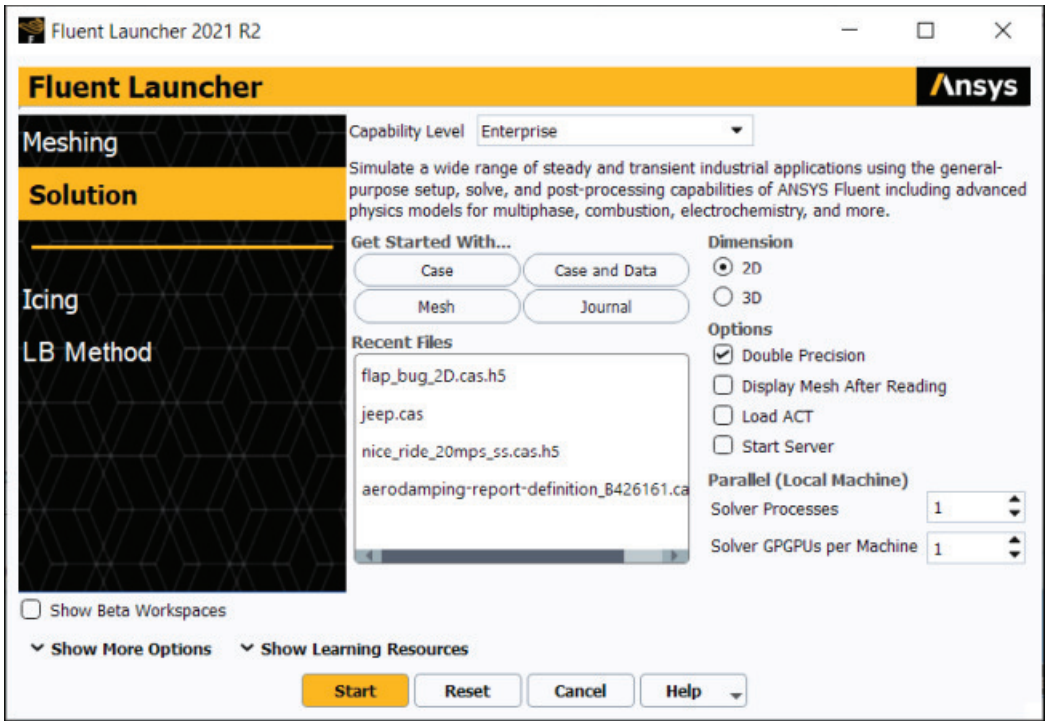
You specify the licensing level using the **Capability Level** drop-down list in the Fluent Launcher. The feature availability at the different licensing levels is discussed in Program Capabilities (p. 135).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4.1.2. Starting Ansys Fluent Using Fluent Launcher

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



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

- Start Ansys Fluent from the Linux or Windows command line with no arguments.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Start Ansys Fluent from the Windows Start menu.
- Start Ansys Fluent from the Windows desktop or Quick Launch bar.

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

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

- **Meshing**—See Introduction to Meshing Mode in Fluent (p. 199) and subsequent chapters for further details about using the Fluent Meshing workspace.
- **Solution**—See Graphical User Interface (GUI) (p. 801) and subsequent chapters for further details about using the Fluent Solution workspace.
- **Icing**—See Fluent Icing (p. 4981) for further details about using the Fluent Icing workspace.

*You can enable **Show Beta Workspaces** to expose additional workspaces that are not yet considered "full" features. These workspaces are documented in the Fluent User's Guide (p. 1).*

Important:

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

Select a case| case and data | mesh | journal file to start with or specify the **Dimension** of the simulation you intend to perform. *Beginning with a journal file allows you to automatically load the case, compile any user-defined functions, iterate until the solution converges, and write results to an output file.*

Note:

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

Select your required **Options**.

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

  Note:

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

- Enable the **Load ACT** option to load Ansys ACT. For additional information on ACT, see Customizing Fluent in the *Fluent User's Guide* (p. 3983).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Choose to perform solution calculations in **Double Precision** mode, if desired. (Default is single-precision mode) See Single-Precision and Double-Precision Solvers (p. 168) to help with your decision.

  Note:

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

- Enable **Start Server** to launch the Fluent workspace as a server so that you can launch the Fluent Remote Visualization Client and connect to this Fluent session. Refer to Remote Visualization and Accessing Fluent Remotely (p. 4945) to learn more about this option.

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

- Set **Meshing Processes** to **1** to restrict the meshing calculations to a single processor core.
- Set **Solver Processes** to **1** to restrict the solution calculations to a single processor core.
- Set **Meshing Processes** more than 1 to allow multiple simultaneous processes. *Note that increasing the number of meshing processes also increases the solver processes to the same processor count.* See Setting Parallel Options in Fluent Launcher (p. 169) for additional information.
- Set **Solver Processes** more than 1 to allow multiple simultaneous processes. See Setting Parallel Options in Fluent Launcher (p. 169) for additional information.
- Set **Solver GPGPUs per Machine** to more than 0 to utilize your graphics card for solver computations.

Select **Show More Options** to expand the Fluent Launcher window to reveal more options. (Figure 4.1: The General Options Tab of Fluent Launcher (p. 167)). Note that once Fluent Launcher expands, the **Show More Options** button becomes the **Show Fewer Options** button, allowing you to hide the additional options.

Select **Show Learning Resources** to expand the Fluent Launcher window to reveal links to: Documentation, News, Tutorials, Online Resources, and Videos. Note that once Fluent Launcher expands, the **Show Learning Resources** button becomes the **Hide Learning Resources** button, allowing you to hide the links.

Important:

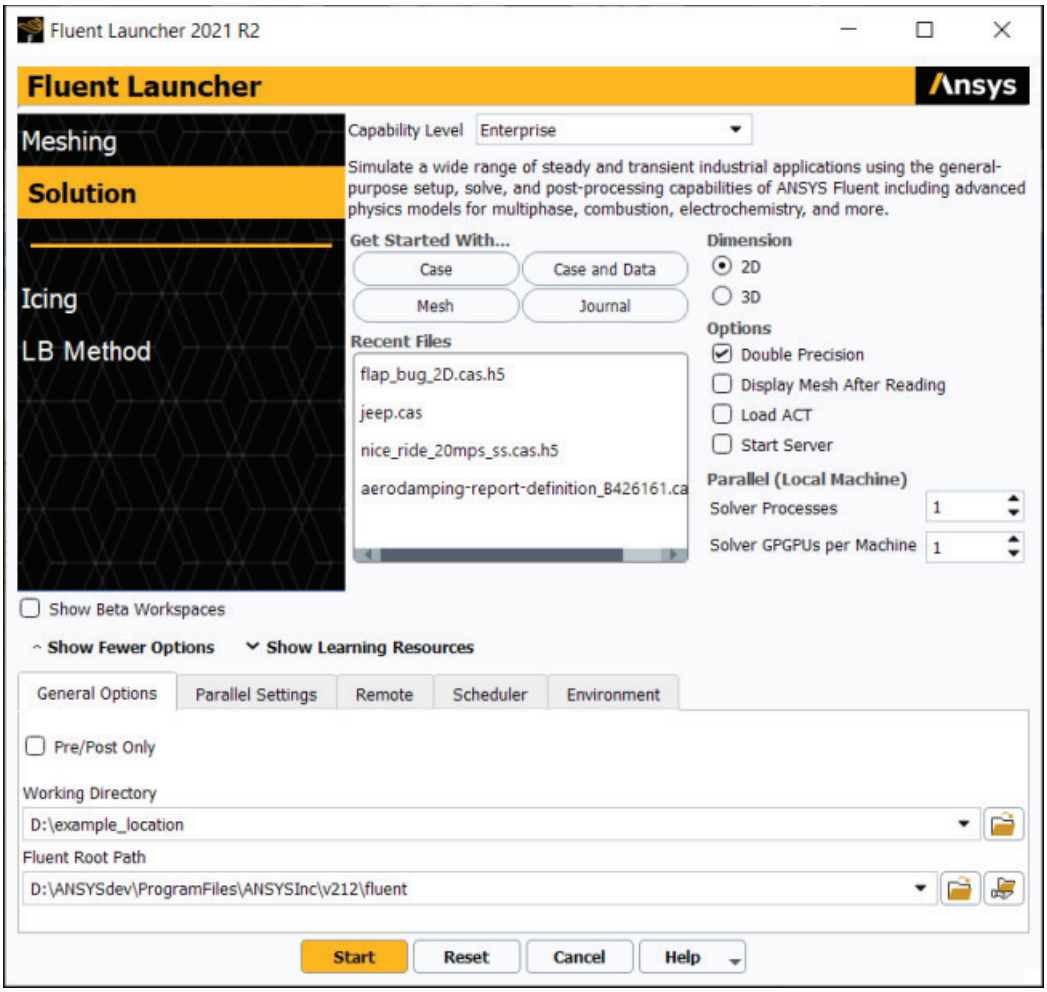
Fluent Launcher also appears when you start Ansys Fluent within Ansys Workbench. For more information, see the separate Ansys Fluent in Workbench User's Guide.

4.1.2.1. Setting General Options in Fluent Launcher

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

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.1: The General Options Tab of Fluent Launcher



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

2. Specify the path of your current working directory using the **Working Directory** field or click to browse through your directory structure.

Note:

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

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

   Note:

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

4.1.2.2. Single-Precision and Double-Precision Solvers

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

- If your geometry has features of very disparate length scales (for example, a very long, thin pipe), single-precision calculations may not be adequate. Note that nodal coordinates are always stored in double precision (even for the single-precision version of Ansys Fluent), so they are not a concern in this regard.
- If your geometry involves multiple enclosures connected via small-diameter pipes (for example, automotive manifolds), mean pressure levels in all but one of the zones can be quite large (since you can set only one global reference pressure location). Double-precision calculations may therefore be necessary to resolve the pressure differences that drive the flow, since these will typically be much smaller than the pressure levels.
- For conjugate problems involving high thermal-conductivity ratios and/or high-aspect-ratio meshes, convergence and/or accuracy may be impaired with the single-precision solver, due to inefficient transfer of boundary information.
- For multiphase problems where the population balance model is used to resolve particle size distributions, which could have statistical moments whose values span many orders of magnitude.

  Note:

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

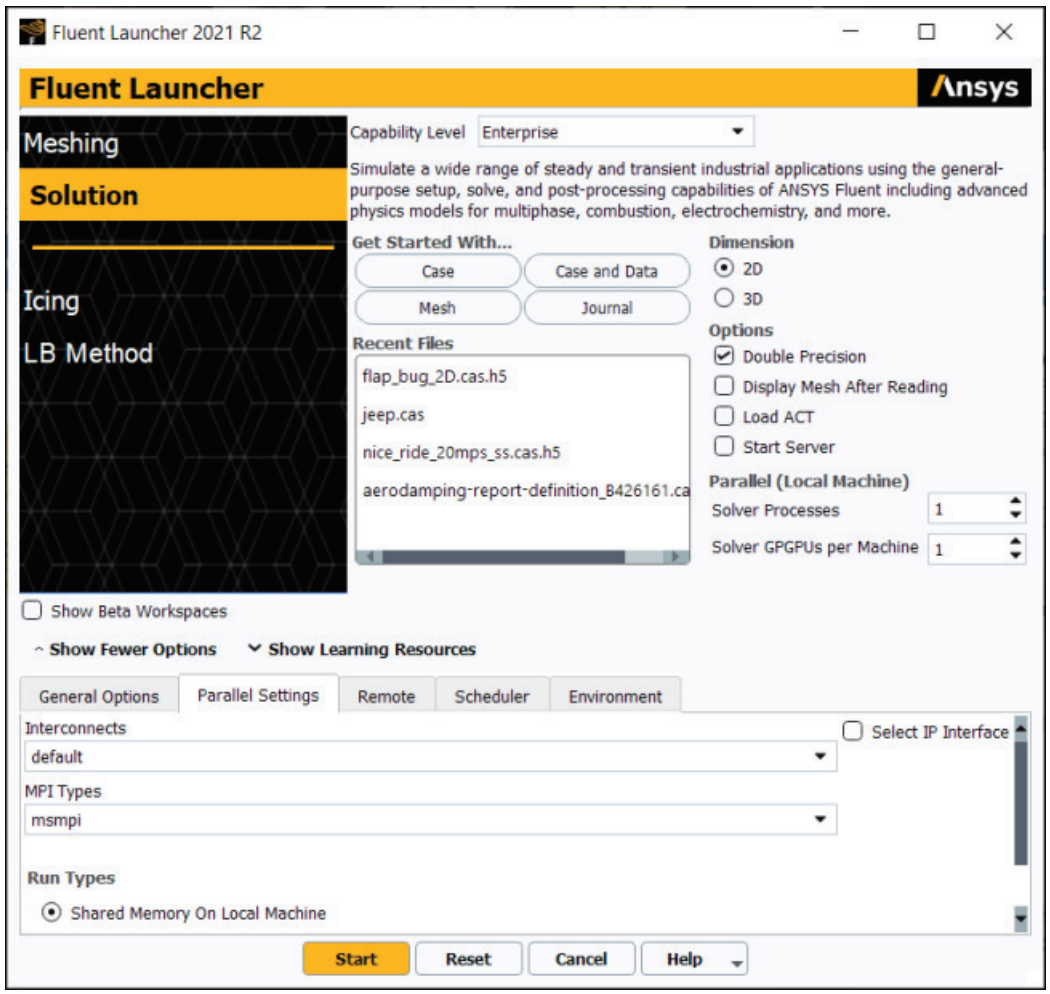
  Ansys Workbench accepts commas as decimal delimiters. These are translated into periods when data is passed to Ansys Fluent.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.1.2.3. Setting Parallel Options in Fluent Launcher

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

Figure 4.2: The Parallel Settings Tab of Fluent Launcher



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

2. (**Meshing** and **Solution** workspaces only) Enter the number of processes to be used for solution under **Solver Processes**. On Linux with the Intel MPI, additional processes will be spawned as necessary when you change to solution mode, in order to bring the total number of processes to this value; this must be set to a value greater than or equal to **Meshing Processes**. For details on this dynamic spawning, see Dynamically Spawning Processes Between Fluent Meshing and Fluent Solution Modes (p. 202).

3. (**Meshing** and **Solution** workspaces only) If your machine is equipped with General Purpose Graphics Processing Units you can also specify **Solver GPGPUs per Machine**.

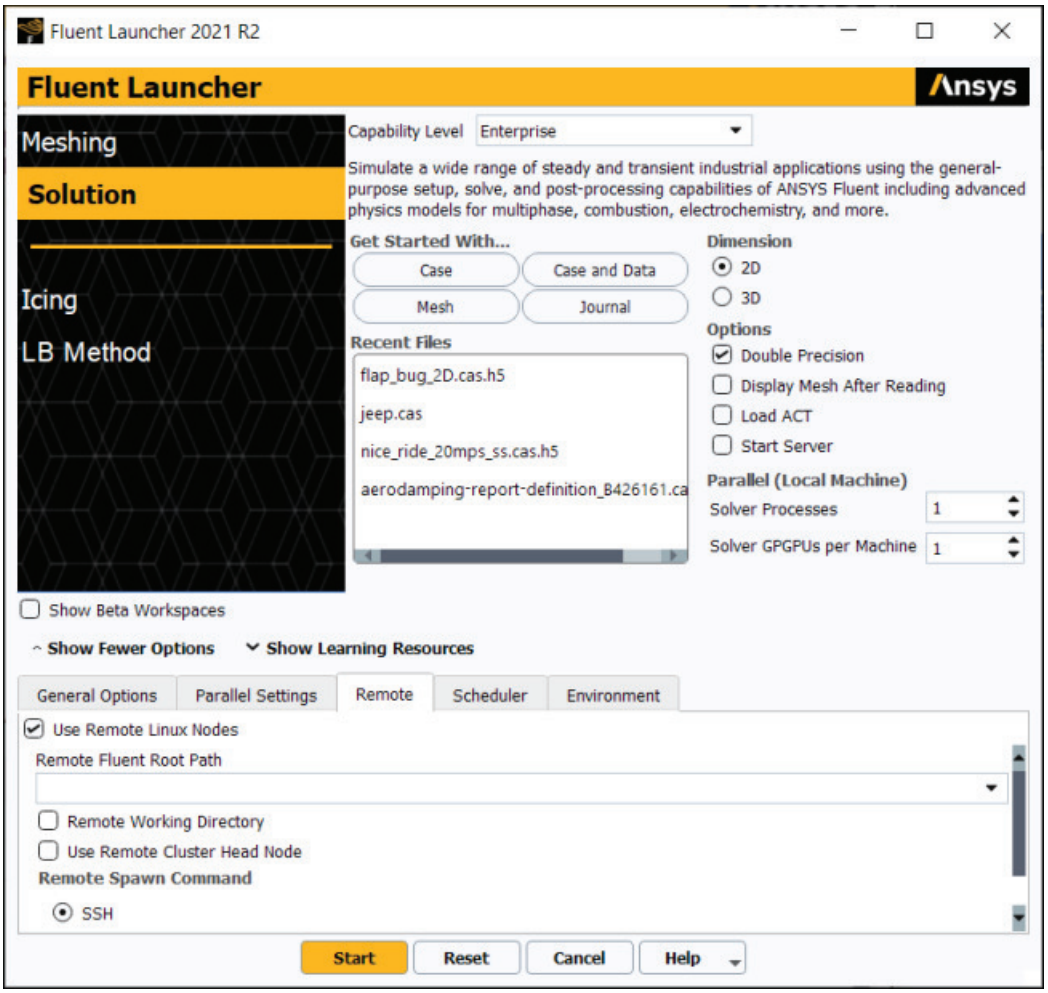
Refer to Parallel Processing in the *Fluent User's Guide* (p. 3745) for details on parallel processing using Fluent and Starting Parallel Ansys Fluent Using Fluent Launcher in the *Fluent User's Guide* (p. 3748) for additional details parallel process configuration options on this tab.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.1.2.4. Setting Remote Options in Fluent Launcher

The **Remote** tab (Figure 4.3: The Remote Tab of Fluent Launcher (p. 170)) allows you to specify settings for running Ansys Fluent parallel simulations on Linux clusters, via the Windows interface.

Figure 4.3: The Remote Tab of Fluent Launcher



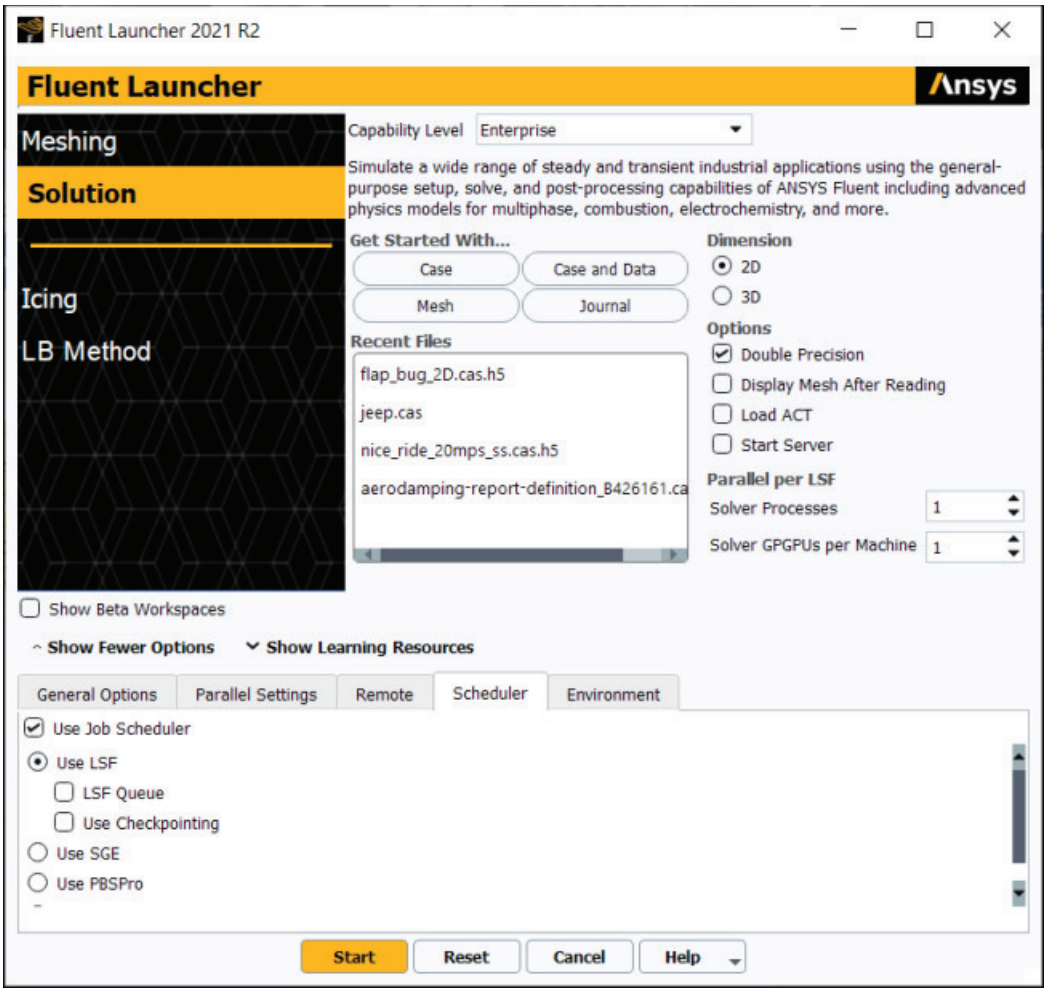
For additional information about this tab, see Setting Additional Options When Running on Remote Linux Machines (p. 3753).

4.1.2.5. Setting Scheduler Options in Fluent Launcher

Enable **Use Job Scheduler** to specify settings in the **Scheduler** tab (Figure 4.4: The Scheduler Tab of Fluent Launcher (Windows 64 Version) (p. 171)) for running Ansys Fluent with various job schedulers (for example, the Microsoft Job Scheduler for Windows, or LSF, SGE, PBS Pro, and Slurm on Linux).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.4: The Scheduler Tab of Fluent Launcher (Windows 64 Version)



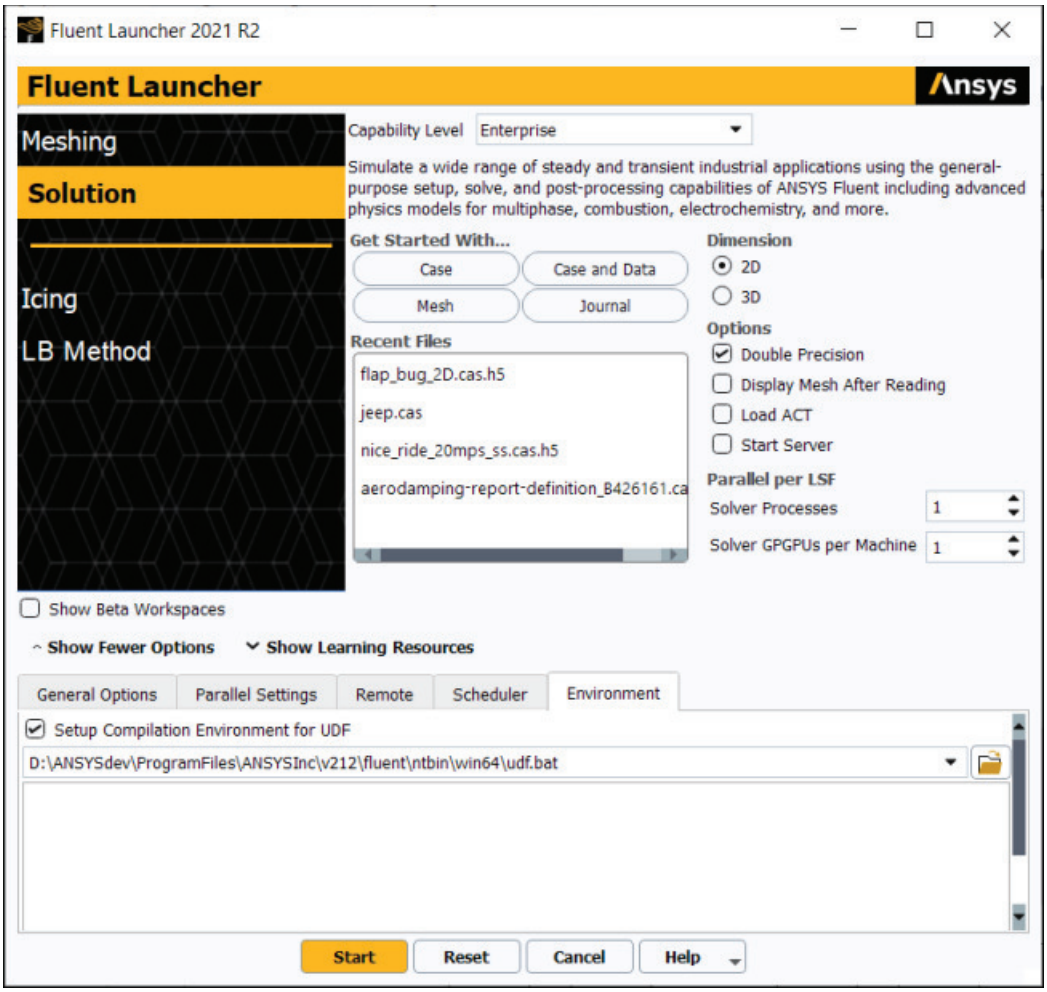
For additional information about this tab, see Setting Parallel Scheduler Options in Fluent Launcher (p. 3750).

4.1.2.6. Setting Environment Options in Fluent Launcher

The **Environment** tab (Figure 4.5: The Environment Tab of Fluent Launcher (p. 172)) allows you to specify compiler settings for compiling user-defined functions (UDFs) with Ansys Fluent (Windows only). The **Environment** tab also allows you to specify environment variable settings for running Ansys Fluent.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.5: The Environment Tab of Fluent Launcher



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

Under **Other Environment Variables**, enter or edit license file or environment variable information in the text field. For example, `FLUENT_AFFINITY=<x>` specifies the process binding (affinity) setting, in the same manner as the `-affinity=<x>` command line option (see Parallel Options (p. 179) for details). Using the **Default** button resets the default value(s).

4.1.3. Starting Ansys Fluent on a Windows System

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

- From the Windows **Start** menu, click **Start > ANSYS 2021 R2 > Fluid Dynamics > Fluent 2021 R2**

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This option starts Fluent Launcher (see Starting Ansys Fluent Using Fluent Launcher (p. 164)). The Fluent Launcher may also be accessed via an icon on your desktop or in the Quick Launch bar.

Note:

If the default "Ansys 2021 R2" program group name was changed when Ansys Fluent was installed, you will find the **Fluent** menu item in the program group with the new name that was assigned, rather than in the **Ansys 2021 R2** program group.

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

  For additional information on starting Fluent from the command prompt, see Command Line Startup Options (p. 174).

Important:

To be able to start Ansys Fluent from the command prompt, be sure the path to your Ansys Fluent home directory is in your command search path environment variable by executing the `setenv.exe` program located in the Ansys Fluent directory (for example, `C:\Program Files\ANSYS Inc\v212\fluent\ntbin\win64`).

Tip:

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

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

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.1.4. Starting Ansys Fluent on a Linux System

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

- Start Fluent from the command line without specifying a version, and then use Fluent Launcher to choose the appropriate version along with other options. See Starting Ansys Fluent Using Fluent Launcher (p. 164) for details.
- Start the appropriate version from the command line by typing `fluent version`, where `version` is replaced with one of the four options specifying the dimension and precision of the solver.
  - `2d` for the 2D, single-precision solver.
  - `3d` for the 3D, single-precision solver.
  - `2ddp` for the 2D, double-precision solver.
  - `3ddp` for the 3D double-precision solver.

Tip:

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

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

Note:

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

4.1.5. Command Line Startup Options

Table 4.1: Available Command Line Options for Linux and Windows Platforms (p. 175) lists the available command line arguments for Linux and Windows. More detailed descriptions of these options can be found in the following sections.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

Option	Platform	Description
`-aas`	all	Start Fluent in server mode.
`-act`	all	Load ACT on Fluent startup.
`-affinity=<x>`	all	Specifies the process binding (affinity) setting, as described in Parallel Options (p. 179).
`-app=flremote`	all	Launches the Remote Visualization Client.
`-appscript=<scriptfile>`	all	Runs the specified script in the specified application (must be used with the `-app=<x>` startup option.
`-command="<TUI command>"`	all	Executes the specified text command (`<TUI command>`) at Fluent startup.
`-ccp <x>`	Windows only	Uses the Microsoft Job Scheduler, where `<x>` is the head node name.
`-cflush`	Linux only	Ensures that the file cache buffers are flushed.
`-cnf=<x>`	all	Specifies that `<x>` is the hosts file or (for Linux) machine list.
`-driver <name>`	all	Sets the graphics driver (available drivers vary by platform, and include `opengl`, `opengl2`, `x11`, and `null` for Linux and `opengl`, `opengl2`, `dx11`, `msw`, and `null` for Windows).
`-env`	all	Show environment variables.
`-g`	all	Run without the GUI or graphics.
`-gpgpu=<n>`	Linux and Win64 only	Specifies the number of GPGPUs per machine that should be used for AMG acceleration. Only available in parallel.
`-gr`	all	Run without graphics.
`-gu`	all	Run without the GUI but with graphics. You cannot interact with the displayed graphics objects.
`-gui_machine=<hostname>`	Linux only	Specifies that `<hostname>` is used for running Cortex (the process that manages the GUI and graphics).
`-h<heap size>`	all	Specifies the heap space for Cortex (the process that manages the GUI and graphics)
`-help`	all	Display command line options.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Option	Platform	Description
-hidden	Windows only	Run in minimized mode.
-host_ip=<host:ip>	all	Specifies that the IP interface <host:ip> is to be used by the host.
-i <journal>	all	Reads the specified journal file(s). Read multiple journals at once as follows: -i example1.jou -i example2.jou -i example3.jou ... *AAS Mode does not support multiple journals from the command line.*
-meshing	all	Start Fluent in meshing mode (you must specify Fluent as either 3d or 3ddp).
-mpi=<mpi>	all	Specifies that the MPI implementation is <mpi> (for example, intel).
-mpitest	all	Launches an MPI program to collect network performance data and prints to console (Linux) or to the working directory (Windows).
-nm	all	Do not display mesh after reading.
-p<ic>	all	Specify interconnect; <ic>={default | eth | ib}
-pcheck	Linux only	Check the network connections before spawning compute nodes.
-platform=<x>	Linux only	Loads a binary that is specially ported for a particular platform, as described in Performance Options (p. 179).
-post	all	Run the Ansys Fluent postprocessing-only executable.
-r	all	List all releases installed in the current directory.
-r<x>	all	Run release <x> of Ansys Fluent.
-remote_node=<hostname>	Linux only	Specify the machine to be used for executing mpirun to launch the node processes; if =<hostname> is omitted, the first node in the hosts file will be used.
-scheduler=<scheduler>	Linux only	Run Ansys Fluent under a scheduler; <scheduler> can be set to lsf (LSF), pbs (PBS Professional), sge (Univa Grid Engine—formerly Sun Grid Engine), or slurm (Slurm).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Option	Platform	Description
-scheduler_account=<account>	Linux only	Specifies that the account is set to <account> when running under Slurm.
-scheduler_custom_script	Linux only	Ensures the use of environment variables when using custom scheduler scripts.
-scheduler_headnode=<head-node>	Linux only	Specifies the scheduler job submission machine name.
-scheduler_opt=<opt>	Linux only	Enables an additional option <opt> that is relevant for the selected scheduler; this command line option can be included multiple times.
-scheduler_pe=<pe>	Linux only	Sets the parallel environment to <pe> when running under SGE.
-scheduler_queue=<queue>	Linux only	Sets the scheduler queue or partition to <queue>.
-scheduler_stderr=<err-file>	Linux only	Sets the scheduler standard error file to <err-file>.
-scheduler_stdout=<out-file>	Linux only	Sets the scheduler standard output file to <out-file>.
-scheduler_tight_coupling	Linux only	Enables a job-scheduler-supported native remote node access mechanism.
-setenv="<var>=<value>"	all	Sets the environment variable <var> to <value>.
-sifile=<name>.txt	all	Run Ansys Fluent and start the remote visualization server. *You can provide a path before the server info filename to specify where the file is created.*
-stream	Linux only	Prints the memory bandwidth.
-t<x>	all	Specifies that the number of processors is <x>.
-tm<x>	all	Specifies that the number of processors for meshing is <x>.

4.1.5.1. ACT Option

`fluent -act` loads Ansys ACT at Fluent startup. For additional information about ACT in Fluent, see Customizing Fluent in the *Fluent User's Guide* (p. 3983).

4.1.5.2. Application Option

`fluent -app=flremote` launches either the Fluent Launcher or the specified dimension of Fluent (for example, 3ddp), along with the Fluent Remote Visualization Client. For additional information about the Fluent Remote Visualization Client, refer to Remote Visualization and Accessing Fluent Remotely (p. 4945).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.1.5.3. Application Script Option

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

4.1.5.4. Graphics Options

Note:

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

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

`fluent -driver` allows you to specify the graphics driver to be used in the session. When enabling graphics display, you have various options: on Linux, the available drivers include `fluent -driver opengl2`, `fluent -driver opengl`, and `fluent -driver x11`; on Windows, the available drivers include `fluent -driver opengl2`, `fluent -driver opengl`, `fluent -driver dx11`, and `fluent -driver msw` (the latter instructs Ansys Fluent to use the Operating Systems Windows driver). For both Linux and Windows, you can disable graphics display using `fluent -driver null`. For a comprehensive list of the drivers available to you, open a Fluent session, enter the `display/set/rendering-options/driver` text command, and then press the **Enter** key at the `driver>` prompt. For more details about using the driver options, see Hiding the Graphics Window Display (p. 3456) in the User's Guide (p. 1).

Note:

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

`fluent -gui_machine=<hostname>` will run Cortex on a specified machine (`<hostname>`). This option is only available when running on Linux, and may be needed to ensure optimal graphics performance when running Fluent under a scheduler / load manager (using the `-scheduler=<scheduler>` option, as described in Scheduler Options (p. 181)).

Important:

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

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

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

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

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

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

`fluent -h<heap size>` will update the heap space for Cortex processes to the specified size. For example, `fluent 3ddp -h50000000`. The default heap size is `6000000`.

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

Important:

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

4.1.5.5. Meshing Mode Option

`fluent -meshing` specifies that Fluent opens in meshing mode rather than the default solution mode. See the Fluent User's Guide (p. 1) for further details about the meshing mode.

4.1.5.6. Performance Options

`-cflush` specifies that memory is allocated in such a way as to ensure that all of the associated file cache buffers are flushed. This may resolve processing performance issues. For more details, see Clearing the Linux File Cache Buffers (p. 3803) in the User's Guide (p. 1).

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

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

4.1.5.7. Parallel Options

These options are used in association with the parallel solver.

`-affinity=<x>` specifies the process binding (affinity) settings. The default behavior depends upon the platform on which you are running:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When running on Linux, there are two options available:
  - If <x> = core, each process is assigned to an individual core in an optimized manner. This is the default for when running in exclusive mode, that is, when the machine is not already loaded.
  - If <x> = sock, processes are assigned to all cores in sockets rather than the individual cores. This is the default for when running in non-exclusive mode.
- When running on Windows, the default behavior depends upon the selected message passing interface (MPI).
  - For the Intel MPI (which is the default), Fluent by default allows Intel MPI to manage process binding. You can set -affinity=1 to use Fluent-managed affinity, which assigns each process to an individual core.
  - For the Microsoft MPI (which is the default when running under the Microsoft Job Scheduler), the scheduler itself manages process binding by default. Whereas, when the Microsoft MPI is selected for shared memory runs, Fluent by default assigns each process to an individual core.
- If <x> = off, the Fluent-managed affinity settings are disabled for both platforms (Windows and Linux).

-ccp <x> (where <x> is the name of the head node) runs the parallel job through the Microsoft Job Scheduler as described in Starting Parallel Ansys Fluent with the Microsoft Job Scheduler (p. 3760) in the User's Guide (p. 1).

-cnf=<x> (where <x> is the name of a hosts file or a list of Linux machines) spawns a compute node on each of the specified machines. For details, see Starting Parallel Ansys Fluent on a Windows System Using Command Line Options (p. 3758) or Starting Parallel Ansys Fluent on a Linux System Using Command Line Options (p. 3763) in the User's Guide (p. 1).

-gpgpu=<n> specifies the number of general purpose graphics processing units (GPGPUs) per machine to be used for AMG acceleration. For more information, see Using General Purpose Graphics Processing Units (GPGPUs) With the Algebraic Multigrid (AMG) Solver in the *Fluent User's Guide* (p. 3794).

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

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

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

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

-p<ic> specifies the use of parallel interconnect <ic>, where <ic> can be any of the interconnects listed in Starting Parallel Ansys Fluent on a Windows System Using Command Line Options (p. 3758) or Starting Parallel Ansys Fluent on a Linux System Using Command Line Options (p. 3763) in the User's Guide (p. 1).

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

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

`-t<x>` specifies that `<x>` processors are to be used. For more information about starting the parallel version of Ansys Fluent, see Starting Parallel Ansys Fluent on a Windows System (p. 3758) or Starting Parallel Ansys Fluent on a Linux System (p. 3763) in the User's Guide (p. 1).

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

4.1.5.8. Postprocessing Option

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

```
fluent 3d -post
```

To use this same feature from the graphical interface on Windows or Linux, select the **Pre/Post** option in the **General** tab of Fluent Launcher, as described in Starting Ansys Fluent Using Fluent Launcher (p. 164).

4.1.5.9. Remote Visualization Options

The `-sifile=<name>.txt` option starts Ansys Fluent and the server that is necessary for running the remote visualization client. For additional information on remote visualization, refer to Remote Visualization and Accessing Fluent Remotely (p. 4945).

Note:

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

4.1.5.10. Scheduler Options

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

- `lsf`: this allows you to run Ansys Fluent under IBM Spectrum LSF software, and thereby take advantage of the checkpointing features of that load management tool. For further details, see Part 1: Running Fluent Under LSF.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `pbs`: this runs Ansys Fluent under Altair PBS Professional, and allows you to use the features of this software to manage your distributed computing resources. For further details, see Part 2: Running Fluent Under PBS Professional.
- `sge`: this runs Ansys Fluent under Univa Grid Engine (previously known as Sun Grid Engine, or SGE) software, and allows you to use the features of this software to manage your distributed computing resources. For further details, see Part 3: Running Fluent Under SGE.
- `slurm`: this runs Ansys Fluent under Slurm, and allows you to use the features of this software to manage your distributed computing resources. For further details, see Part 4: Running Fluent Under Slurm.

Note:

You can use the `-scheduler=<scheduler>` option along with the `-gui_machine=<hostname>` (described in Graphics Options (p. 178)), in order to ensure optimal graphics performance. When running under Slurm, the `-gui_machine=<hostname>` is also needed to allow dynamic spawning (which is described in Dynamically Spawning Processes Between Fluent Meshing and Fluent Solution Modes (p. 202)), as well as the combination of Slurm + Open MPI + distributed memory on a cluster.

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

Other options are available when you use a scheduler: you can specify the scheduler job submission machine name using `-scheduler_headnode=<head-node>` (by default it is `localhost`); you can specify a queue / partition using `-scheduler_queue=<queue>`; you can enable an additional option for the scheduler using `-scheduler_opt=<opt>` (note that you can include multiple instances of this option when you want to use more than one scheduler option); you can specify the name and directory of the scheduler standard error file using `-scheduler_stderr=<err-file>` (by default it is saved as `fluent.<PID>.e` in the working directory, where `<PID>` is the process ID of the top-level Fluent startup script); you can specify the name and directory of the scheduler standard output file using `-scheduler_stdout=<out-file>` (by default it is saved as `fluent.<PID>.o` in the working directory); when running under Univa Grid Engine software, you can set the parallel environment using `-scheduler_pe=<pe>`; and when running under Slurm you can set the account using `-scheduler_account=<account>`.

It is also possible to enable a job-scheduler-supported native remote node access mechanism using `-scheduler_tight_coupling` in Linux. For details about the MPI / job scheduler combinations that are supported for this tight coupling, see Running Fluent Using a Load Manager.

4.1.5.11. Text Command Option

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

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

4.1.5.12. Version, Release Options, and Environment Variables

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

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

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

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

4.1.5.13. System Coupling Options

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

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

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

`-scname="<z>"` (where `<z>` is the name of the participant, in quotes) specifies the unique name used by the co-simulation participant to identify itself to the coupling service (see Server File (`scServer.scs`) in the *System Coupling User's Guide* for more information).

The general syntax for invoking Ansys Fluent for system coupling is:

`fluent 3d –schost=`*host name in quotes* `–scport=`*port number* `–scname=`*name of the solver in quotes*

For instance:

```
fluent 3d –schost="machine1.domain.com" –scport=1234 –scname="Solution1"
```

Once Ansys Fluent loads the case, initialize the solution using the following command:

```
s i i
```

Once your case is initialized, start the system coupling by typing the following command in the Ansys Fluent text user interface (TUI):

```
(sc-solve)
```

For more information, see Performing System Coupling Simulations Using Fluent (p. 3951) in the Fluent User's Guide, as well as the System Coupling User's Guide.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.1.5.14. Other Startup Options

There are other startup options that are not listed when you type the `fluent -help` command. These options can be used to customize your graphical user interface. For example, to change the Ansys Fluent window size and position you can either modify the .Xdefaults file described in Customizing the Graphical User Interface (p. 830) in the User's Guide (p. 1), or you can simply type the following command at startup:

```
fluent <version> -geometry <XX>x<YY>+<xx>-<yy>
```

where `<XX>` and `<YY>` are the width and height in pixels, respectively, and `+<xx>-<yy>` is the position of the window.

Therefore, typing `fluent 3d -geometry 700x500+20-400` will start the 3D version of Ansys Fluent, sizing the Ansys Fluent console to 700x500 pixels and positioning it on your monitor screen at +20-400.

There are additional Qt command line startup options for modifying the graphical stylesheet and more, which can be found in Qt documentation.

4.2. Running Ansys Fluent in Batch Mode

Ansys Fluent can be used interactively, with input from and display to your computer screen, or it can be used in a batch or background mode in which inputs are obtained from and outputs are stored in files. Generally you will perform problem setup, initial calculations, and postprocessing of results in an interactive mode. However, when you are ready to perform a large number of iterative calculations, you may want to run Ansys Fluent in batch or background mode. This allows the computer resources to be prioritized, enables you to control the process from a file (eliminating the need for you to be present during the calculation), and also provides a record of the calculation history (residuals) in an output file. While the procedures for running Ansys Fluent in a batch mode differ depending on your computer operating system, Background Execution on Linux Systems (p. 184) provides guidance for running in batch/background on Linux systems, and Background Execution on Windows Systems (p. 186) provides guidance for running in batch/background on Windows systems.

For additional information, see the following sections:

4.2.1. Background Execution on Linux Systems

4.2.2. Background Execution on Windows Systems

4.2.3. Batch Execution Options

4.2.1. Background Execution on Linux Systems

To run Ansys Fluent in the background in a C-shell (csh) on a Linux system, type a command of the following form at the system-level prompt:

```
fluent 2d -g < inputfile > & outputfile &
```

or in a Bourne/Korn-shell, type:

```
fluent 2d -g < inputfile > outputfile 2>&1 &
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In these examples,

- `fluent` is the command you type to run Ansys Fluent interactively.
- `-g` indicates that the program is to be run without the GUI or graphics (see Starting Ansys Fluent (p. 163)).
- `inputfile` is a file of Ansys Fluent commands that are identical to those that you would type interactively.
- `outputfile` is a file that the background job will create, which will contain the output that Ansys Fluent would normally print to the screen (for example, the menu prompts and residual reports).
- `&` tells the Linux system to perform this task in background and to send all standard system errors (if any) to `outputfile`.

The file `inputfile` can be a journal file created in an earlier Ansys Fluent session, or it can be a file that you have created using a text editor. In either case, the file must consist only of text interface commands (since the GUI is disabled during batch execution). A typical `inputfile` is shown below:

```
; Read case file
rc example.cas
; Initialize the solution
/solve/initialize/initialize-flow
; Calculate 50 iterations
it 50
; Write data file
wd example50.dat
; Calculate another 50 iterations
it 50
; Write another data file
wd example100.dat
; Exit Fluent
exit
yes
```

This example file reads a case file `example.cas`, initializes the solution, and performs 100 iterations in two groups of 50, saving a new data file after each 50 iterations. The final line of the file terminates the session. Note that the example input file makes use of the standard aliases for reading and writing case and data files and for iterating. (`it` is the alias for `/solve/iterate`, `rc` is the alias for `/file/read-case`, `wd` is the alias for `/file/write-data`, etc.) These predefined aliases allow you to execute commonly used commands without entering the text menu in which they are found. In general, Ansys Fluent assumes that input beginning with a `/` starts in the top-level text menu, so if you use any text commands for which aliases do not exist, you must be sure to type in the complete name of the command (for example, `/solve/initialize/initialize-flow`). Note also that you can include comments in the file. As in the example above, comment lines must begin with a `;` (semicolon).

An alternate strategy for submitting your batch run, as follows, has the advantage that the `outputfile` will contain a record of the commands in the `inputfile`. In this approach, you would submit the batch job in a C-shell using:

```
fluent 2d -g -i inputfile  >& outputfile &
```

or in a Bourne/Korn-shell using:

```
fluent 2d -g -i inputfile > outputfile 2>&1 &
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.2.2. Background Execution on Windows Systems

To run Ansys Fluent in the background on a Windows system, the following commands can be used:

```
fluent 3d -g -i journal

fluent 3d -g -wait -i journal

fluent 3d -hidden -i journal
```

In these examples,

- `fluent` is the command you type to run Ansys Fluent interactively.
- `-g` indicates that the program is to be run minimized in the task bar.
- `-i journal` reads the specified journal file.
- `-wait` is the command you type in a DOS batch file or some other script in a situation where the script must wait until Ansys Fluent has completed its run.
- `-hidden` is similar to the `-wait` command, but also runs Ansys Fluent completely hidden and non-interactively.

To get an output (or transcript) file while running Ansys Fluent in the background on a Windows system, the journal file must contain the following command to write a transcript file:

```
; start transcript file
 /file/start-transcript outputfile.trn
```

where the `outputfile` is a file that the background job will create, which will contain the output that Ansys Fluent would normally print to the screen (for example, the menu prompts and residual reports).

See Creating and Reading Journal Files (p. 861) in the User's Guide (p. 1) for details about journal files. See Creating Transcript Files (p. 865) in the User's Guide (p. 1) for details about transcript files.

4.2.3. Batch Execution Options

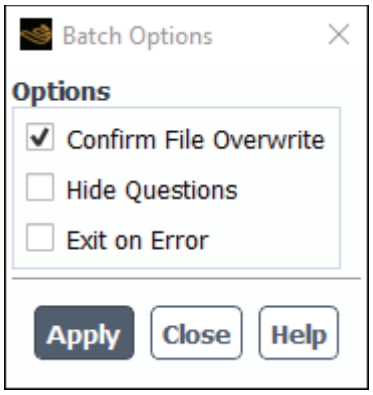
During a typical session, Ansys Fluent may require feedback from you in the event of a problem it encounters. Ansys Fluent usually communicates problems or questions through the use of **Error** dialog boxes, **Warning** dialog boxes, or **Question** dialog boxes. While executing Ansys Fluent in batch mode, you may want to suppress this type of interaction in order to, for example, create journal files more easily.

There are three common batch configuration options available to you when running Ansys Fluent in batch mode. You can access these options using the **Batch Options** dialog box (Figure 4.6: The Batch Options Dialog Box (p. 187)).

File → Batch Options...

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.6: The Batch Options Dialog Box



The **Batch Options** dialog box contains the following items:

Confirm File Overwrite

determines whether Ansys Fluent confirms a file overwrite. This option is turned on by default.

Hide Questions

allows you to hide **Question** dialog boxes. This option is turned off by default.

Exit on Error

allows you to automatically exit from batch mode when an error occurs. This option is disabled by default.

When run in batch mode through the command prompt or a journal file with **Exit on Error** enabled, Fluent will exit under the following circumstances:

- Normal run termination upon reaching the end of a journal (return value `0`)
- Error returned during scripted text command execution (return value `1`)
- Unexpected input (wrong type) to text command (return value `1`)
- Licensing error (return value `2`)

If an invalid text command is entered, Fluent will not exit, but proceed to the next text input.

Note that in Windows you must start Fluent with the `-wait` command line option.

`file` → `set-batch-options`

Any combination of these options can be turned on or off at any given time prior to running in batch mode.

Important:

Batch option settings are *not* saved with case files. They are meant to apply for the duration of the current Ansys Fluent session only. If you read in additional mesh or case files during

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

this session, the batch option settings will not be altered. As batch options are not saved with case files, journal files developed for use in batch mode should begin by enabling the desired batch option settings (if different from the default settings).

4.3. Switching Between Meshing and Solution Modes

You can switch from the meshing mode of Fluent to the solution mode by clicking the **Switch to Solution** button, located by default in the top left corner of the application window. The mesh from your meshing mode session will be transferred and read in the new solution mode session.

You can switch from the solution mode of Fluent to the meshing mode by using the `switch-to-meshing-mode` text command. Note that this text command is only available for 3D sessions, before you have read a mesh or case file.

Note:

When you read a non-conformal interface case file into meshing mode and later switched to the solution mode, note the following limitations:

- Zone ids may match, however, the corresponding zone names may be inconsistent.
- Boundary conditions on intersected threads are not preserved.
- Unassociated (dangling) non-conformal interface (NCI) surfaces remain present.

4.4. Checkpointing an Ansys Fluent Simulation

The checkpointing feature of Ansys Fluent allows you to save case and data files while your simulation is running. While similar to the autosave feature of Ansys Fluent (Automatic Saving of Case and Data Files (p. 854) in the User's Guide (p. 1)), which allows you to save files throughout a simulation, checkpointing allows you slightly more control in that you can save an Ansys Fluent job even after you have started the job and did not set the autosave option. Checkpointing also allows you to save case and data files and then exit out of Ansys Fluent. This feature is especially useful when you need to stop an Ansys Fluent job abruptly and save its data.

There are two different ways to checkpoint an Ansys Fluent simulation, depending upon how the simulation has been started.

1. Ansys Fluent running under LSF or SGE

   Ansys Fluent is integrated with load management tools like LSF and SGE. These two tools allow you to checkpoint any job running under them. You can use the standard method provided by these tools to checkpoint the Ansys Fluent job.

   For more information on using Ansys Fluent and SGE or LSF, see Part 3: Running Fluent Under SGE or Part 1: Running Fluent Under LSF, respectively.

2. Independently running Ansys Fluent

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When not using tools such as LSF or SGE, a different checkpointing mechanism can be used when running an Ansys Fluent simulation. You can checkpoint an Ansys Fluent simulation while iterating/time-stepping, so that Ansys Fluent saves the case and data files and then continues the calculation, or so that Ansys Fluent saves the case and data files and then exits.

- Saving case and data files and continuing the calculation:

  On Linux, create a file called `check-fluent`, that is,

  ```
  /tmp/check-fluent
  ```

  On Windows, create a file called `check-fluent.txt`, that is,

  ```
  C:\temp\check-fluent.txt
  ```

- Saving case and data files and exiting Ansys Fluent:

  On Linux, create a file called `exit-fluent`, that is,

  ```
  /tmp/exit-fluent
  ```

  On Windows, create a file called `exit-fluent.txt`, that is,

  ```
  C:\temp\exit-fluent.txt
  ```

The saved case and data files will have the current iteration number appended to their file names.

Ansys Fluent offers an alternate way to checkpoint an unsteady simulation. While the default behavior is to checkpoint the simulation at the end of the current iteration, for unsteady simulations you have the option of completing all of the iterations in the current time-step before checkpointing. This can be set by entering the following Scheme command prior to running the unsteady simulation:

```
(ckpt/time-step? #t)
```

Now when you save the checkpoint file (as described previously), the case and data file will be saved at the end of the current time step and named accordingly. To switch back to the default checkpointing mechanism at the end of the current iteration, use the following Scheme command:

```
(ckpt/time-step? #f)
```

Important:

Note that the `(ckpt/time-step? #t)` command will have the effect only in the case of an unsteady simulation.

Note:

It is recommended that you do *not* use checkpointing when using Ansys Fluent in Workbench. However, if checkpointing is necessary, the `exit-fluent/exit-fluent.txt` file can be used and the file will be checked in its default location (the `FFF/FLU` system directory

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

containing the `*.set` file). If Ansys Fluent is calculating, then the existence of the file is equivalent to an **interrupt** command. Similarly, the `check-fluent/check-fluent.txt` file can be used to save the project on demand when Ansys Fluent is calculating.

4.5. Cleaning Up Processes From an Ansys Fluent Simulation

Ansys Fluent lets you easily remove extraneous processes in the event that an Ansys Fluent simulation must be stopped.

When a session is started, Ansys Fluent creates a `cleanup-fluent` script file. The script can be used to clean up all Ansys Fluent-related processes. Ansys Fluent creates the cleanup-script file in the current working directory with a filename that includes the machine name and the process identification number (PID) (for example, `cleanup-fluent-mymachine-1234`).

If the current directory does not possess the proper write permissions, then Ansys Fluent will write the cleanup-script file to your home directory.

If, for example, Ansys Fluent is started on a machine called `thor` and the process identification number is `32895`, Ansys Fluent will create a cleanup-script called `cleanup-fluent-thor-32895` in the current directory. To run the cleanup-script, and clean up all Ansys Fluent processes related to your session, on Linux platforms, type the following command in the console window:

```
sh cleanup-fluent-thor-32895
```

Or, if the shell script already has executable permissions, simply type:

```
cleanup-fluent-thor-32895
```

To clean up extraneous Ansys Fluent processes on Windows (serial or parallel), double-click the corresponding batch file (for example, `cleanup-fluent-thor-32895.bat`) that Ansys Fluent generates at the beginning of each session.

Important:

During a normal run, this file will be deleted automatically after exiting Ansys Fluent. In abnormal situations, you may use this batch file to clean up the Ansys Fluent processes. Once an Ansys Fluent session has been closed, you can safely delete any left over cleanup scripts from your working directory.

Important:

If an Ansys Fluent session hangs or freezes on Windows, and you want to view the complete contents of the Ansys Fluent console output in a transcript file, you should use the `taskkill` command through the DOS command prompt, rather than terminating the Ansys Fluent application through the **Windows Task Manager**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.6. Exiting Ansys Fluent

You can exit Ansys Fluent by selecting **Exit** in the **File** ribbon tab. If the present state of the program has not been written to a file, a **Question** dialog box will open to confirm if you want to proceed. You can cancel the exit and write the appropriate file(s) or you can continue to exit without saving the case or data.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Glossary of Terms

This glossary contains a listing of terms commonly used throughout the documentation.

- adaption (p. 193)
- case files (p. 193)
- cell types (p. 194)
- computational fluid dynamics (CFD) (p. 194)
- console (p. 195)
- convergence (p. 195)
- cortex (p. 195)
- data files (p. 195)
- dialog boxes (p. 195)
- discretization (p. 195)
- GUI (p. 195)
- mesh (p. 195)
- models (p. 195)
- node (p. 195)
- postprocessing (p. 195)
- residuals (p. 195)
- skewness (p. 196)
- solvers (p. 196)
- terminal emulator (p. 196)
- TUI (p. 196)

adaption

A technique useful in improving overall mesh quality. The solution-adaptive mesh refinement feature of Ansys Fluent allows you to refine and/or coarsen your mesh based on geometric and numerical solution data. In addition, Ansys Fluent provides tools for creating and viewing adaption fields customized to particular applications.

case files

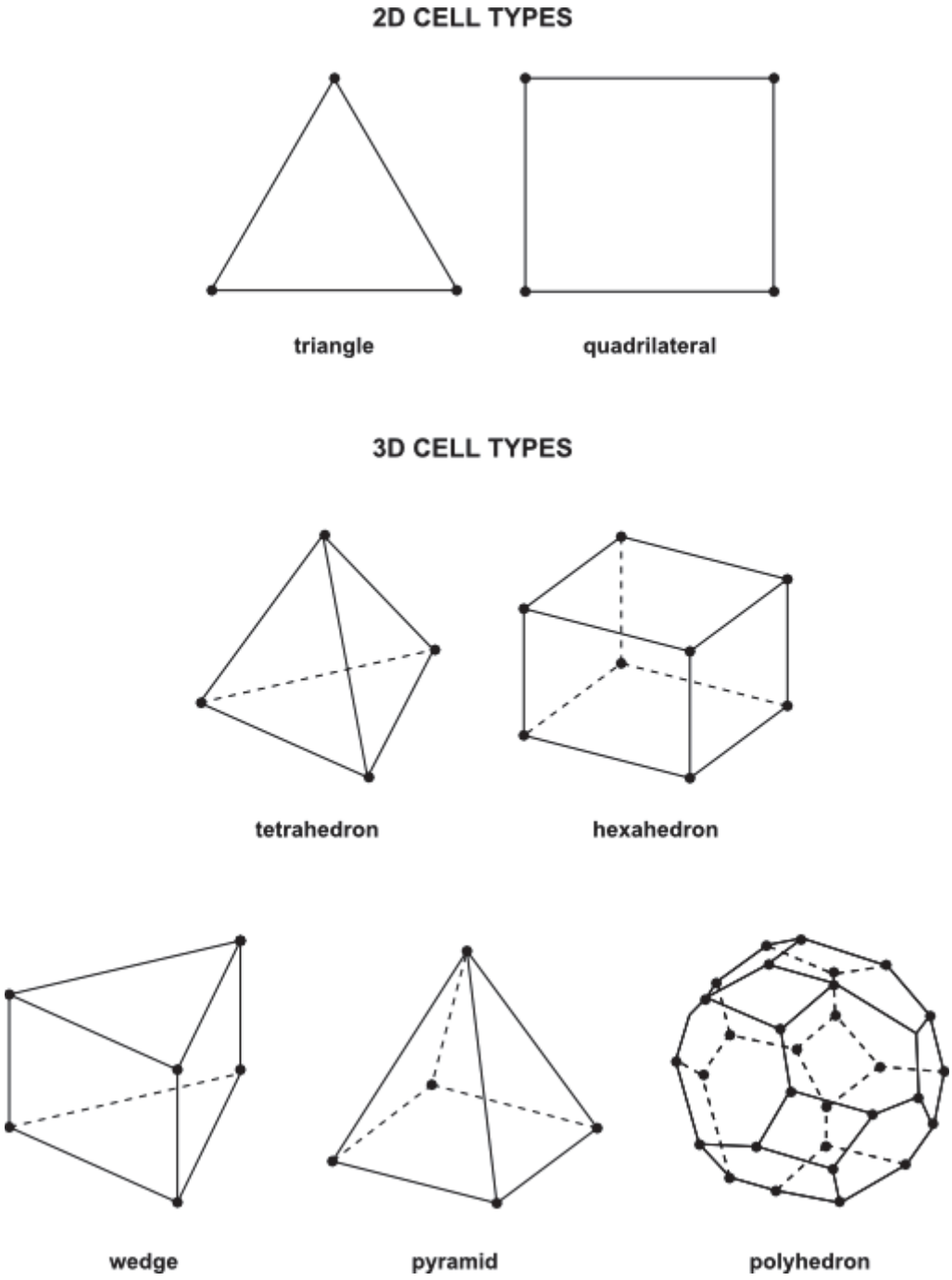
Files that contain the mesh, boundary conditions, and solution parameters for a problem. A case file also contains the information about the user interface and graphics environment.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

cell types

The various shapes or units that constitute the base elements of a mesh. Ansys Fluent can use meshes composed of tetrahedral, hexahedral, pyramid, wedge, or polyhedral cells (or a combination of these).

Figure 8: Cell Types



computational fluid dynamics (CFD)

The science of predicting fluid flow, heat transfer, mass transfer (as in perspiration or dissolution), phase change (as in freezing or boiling), chemical reaction (for example, combustion), mechanical movement (for example, fan rotation), stress or deformation of re-

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	lated solid structures (such as a mast bending in the wind), and related phenomena by solving the mathematical equations that govern these processes using a numerical algorithm on a computer.
console	The console is part of the Ansys Fluent application window that allows for text command input and the display of information.
convergence	The point at which the solution is no longer changing with each successive iteration. Convergence criteria, along with a reduction in residuals, also help in determining when a solution is complete. Convergence criteria are pre-set conditions on the residuals that indicate that a certain level of convergence has been achieved. If the residuals for all problem variables fall below the convergence criteria but are still in decline, the solution is still changing to a greater or lesser degree. A better indicator occurs when the residuals flatten in a traditional residual plot (of residual value vs. iteration). This point, sometimes referred to as convergence at the level of machine accuracy, takes time to reach, however, and may be beyond your needs. For this reason, alternative tools such as reports of forces, heat balances, or mass balances can be used instead.
cortex	A utility that manages Ansys Fluent's user interface and basic graphical functions.
data files	Files that contain the values of the flow field in each grid element and the convergence history (residuals) for that flow field.
dialog boxes	The separate windows that are used like forms to perform input tasks. Each dialog box is unique and employs various types of input controls that make up the form.
discretization	The act of replacing the differential equations that govern fluid flow with a set of algebraic equations that are solved at distinct points.
GUI	The graphical user interface, which consists of the main Ansys Fluent application window, dialog boxes, graphics windows, etc.
mesh	A collection of points representing the flow field, where the equations of fluid motion (and temperature, if relevant) are calculated.
models	Numerical algorithms that approximate physical phenomenon (for example, turbulence).
node	The distinct points of a mesh (p. 195) at which the equations of fluid motion are solved.
postprocessing	The act of analyzing the numerical results of your CFD simulation using reports, integrals, and graphical analysis tools such as contour plots, animations, etc.
residuals	The small imbalance that is created during the course of the iterative solution algorithm. This imbalance in each cell is a small, non-zero

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	value that, under normal circumstances, decreases as the solution progresses.
skewness	The difference between the shape of the cell and the shape of an equilateral cell of equivalent volume. Highly skewed cells can decrease accuracy and destabilize the solution.
solvers	Ansys Fluent has two distinct solvers, based on numerical precision (single-precision vs. double-precision). Within each of these categories, there are solver formulations: pressure based; density based explicit; and density based implicit.
terminal emulator	See console (p. 195).
TUI	The text user interface, which consists of textual commands that can be entered into the terminal emulator.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.