

ANSYS Fluent Meshing User's Guide



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

Release 15.0
November 2013

ANSYS, Inc. is
certified to ISO
9001:2008.

Copyright and Trademark Information

© 2013 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, EKM, Engineering Knowledge Manager, CFX, FLUENT, HFSS 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 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.

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. is certified to ISO 9001:2008.

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, please contact ANSYS, Inc.

Published in the U.S.A.

Table of Contents

1. Using This Manual	1
1.1.The Contents of This Manual	1
1.2.Typographical Conventions Used In This Manual	2
1.3.Contacting Technical Support	3
2. Introduction to Meshing Mode in ANSYS Fluent	7
2.1.Mesher Approach	7
2.2.Program Capabilities	7
2.3.ANSYS Fluent Documentation	8
2.3.1.Accessing the ANSYS Fluent Documentation	8
2.3.1.1.Accessing the Documentation Files Using the ANSYS Help Viewer	9
2.3.1.2.Downloading and Installing the PDF Documentation Files	9
3. Starting and Executing ANSYS Fluent in Meshing Mode	11
3.1.Starting ANSYS Fluent in Meshing Mode	11
3.1.1.Using Fluent Launcher	11
3.1.2.Starting on a Windows System	13
3.1.3.Starting on a Linux System	13
3.1.4.Starting the Dual Process Build	13
3.1.5.Startup Options	14
3.2.Parallel Processing in Meshing Mode	15
3.2.1.Dynamically Spawning Processes Between Fluent Meshing and Fluent Solution Modes	16
3.3.Exiting the Program	17
4. Graphical User Interface	19
4.1.GUI Components	19
4.1.1.Menu Bar	20
4.1.2.Toolbars	21
4.1.2.1.The Mode Toolbar	21
4.1.2.2.The Standard Toolbar	22
4.1.2.3.The Graphics Toolbars	23
4.1.2.4.The Objects Toolbar	24
4.1.3.The Navigation Pane	24
4.1.4.Task Pages	25
4.1.5.The Console	25
4.1.6.Dialog Boxes	26
4.1.6.1.Controls of a Dialog Box	27
4.1.6.2.Special Dialog Boxes	31
4.1.7.Graphics Windows	37
4.1.7.1.Printing the Contents of the Graphics Window (Windows Systems Only)	38
4.1.7.2.The Page Setup Dialog Box	38
4.2.Customizing the GUI (Linux Systems)	40
4.3.Using the GUI Help System	41
4.3.1.Task Page and Dialog Box Help	42
4.3.2.Context-Sensitive Help (Linux Only)	42
4.3.3.Opening the User's Guide Table of Contents	42
4.3.4.Opening the Reference Guide	43
4.3.5.Accessing Printable Manuals	44
4.3.6.Help for Text Interface Commands	44
4.3.7.Using Help	44
4.3.8.Accessing Online Technical Resources	44
4.3.9.Obtaining a Listing of Other License Users	45
4.3.10.Version and Release Information	45

5. Text User Interface	47
5.1.Text Menu System	47
5.1.1.Command Abbreviation	48
5.1.2.Scheme Evaluation	49
5.1.3.Aliases	49
5.2.Text Prompt System	50
5.2.1.Numbers	50
5.2.2BOOLEANS	51
5.2.3.Strings	51
5.2.4.Symbols	51
5.2.5.Filenames	52
5.2.6.Lists	52
5.2.7.Evaluation	53
5.2.8.Default Value Binding	53
5.3.Interrupts	53
5.4.System Commands	53
5.4.1.System Commands for LINUX-based Operating Systems	53
5.4.2.System Commands for Windows-based Operating Systems	54
5.5.Text Menu Input from Character Strings	55
5.6.Using the Text Interface Help System	55
6. Reading and Writing Files	57
6.1.Shortcuts for Reading and Writing Files	57
6.1.1.Default File Suffixes	57
6.1.2.Binary Files	58
6.1.3.Detecting File Format	58
6.1.4.Recent File List	58
6.1.5.Reading and Writing Compressed Files	58
6.1.5.1.Reading Compressed Files	58
6.1.5.2.Writing Compressed Files	59
6.1.6.Tilde Expansion (LINUX Systems Only)	59
6.1.7.Disabling the Overwrite Confirmation Prompt	59
6.1.8.Toolbar Buttons	59
6.2.Mesh Files	60
6.2.1.Reading Boundary Mesh Files	61
6.2.1.1.Reading Multiple Boundary Mesh Files	61
6.2.2.Reading Mesh Files	61
6.2.2.1.Reading Multiple Mesh Files	61
6.2.2.2.Reading 2D Mesh Files in the 3D Version of ANSYS Fluent	62
6.2.3.Reading Faceted Geometry Files from ANSYS Workbench in ANSYS Fluent	62
6.2.4.Appending Mesh Files	62
6.2.5.Writing Mesh Files	63
6.2.6.Writing Boundary Mesh Files	63
6.3.Case Files	63
6.3.1.Reading Case Files	64
6.3.1.1.Reading Multiple Case Files	64
6.3.2.Writing Case Files	64
6.4.Size-Field Files	65
6.4.1.Reading Size-Field Files	65
6.4.2.Writing Size-Field Files	66
6.5.Reading Scheme Source Files	66
6.6.Creating and Reading Journal Files	66
6.7.Creating Transcript Files	68

6.8. Domain Files	69
6.9. Importing Files	69
6.9.1. ANSYS Prep7 Files	70
6.9.2. CGNS Files	70
6.9.3. Importing FIDAP Neutral Mesh Files	70
6.9.4. Importing GAMBIT Neutral Mesh Files	71
6.9.5. HYPERMESH ASCII Files	71
6.9.6. I-deas Universal Files	71
6.9.7. NASTRAN Files	72
6.9.8. PATRAN Neutral Files	72
6.9.9. Importing CAD Files	72
6.9.9.1. Text Commands for Importing CAD Files	76
6.9.10. Grid Import Filter Options	78
6.9.10.1. Text Commands for Setting Filter Options	78
6.10. Exporting STL Files	78
6.11. Saving Picture Files	78
6.11.1. Using the Save Picture Dialog Box	78
6.11.2. Text Interface for Saving Picture Files	82
6.12. The .tgrid File	83
6.13. Exiting the Meshing Mode	84
7. Size Functions	85
7.1. Types of Size Functions	85
7.1.1. Curvature Size Function	86
7.1.2. Proximity Size Function	87
7.1.3. Meshed Size Function	90
7.1.4. Hard Size Function	91
7.1.5. Soft Size Function	91
7.1.6. Body of Influence Size Function	92
7.2. Defining Size Functions	93
7.2.1. Creating Default Size Functions	94
7.2.2. Computing the Size Field	95
7.2.2.1. Size Field Files	95
7.2.2.2. Using Size Field Filters	95
7.2.2.3. Visualizing Sizes	96
7.3. Using Size Functions/Size Field	97
7.4. Text Commands for Size Functions	98
8. Meshing Objects and Material Points	103
8.1. Objects	103
8.1.1. Object Attributes	103
8.1.1.1. Approaches For Creating Objects	105
8.1.2. Object Based Meshing	106
8.1.3. Using the Manage Objects Dialog Box	107
8.1.3.1. Defining Objects	107
8.1.3.2. Creating Multiple Objects	108
8.1.3.3. Object Manipulation Operations	108
8.1.3.4. Object Transformation Operations	109
8.1.3.5. Automatic Alignment of Objects	110
8.1.4. Text Commands for Objects	110
8.2. Material Points	114
8.2.1. Using the Material Point Dialog Box	117
8.2.2. Using Text Commands	117
9. Manipulating the Boundary Mesh	119

9.1. Manipulating Boundary Nodes	119
9.1.1. Free and Isolated Nodes	119
9.1.2. Text Commands for Manipulating Boundary Nodes	120
9.2. Intersecting Boundary Zones	121
9.2.1. Intersecting Zones	121
9.2.2. Joining Zones	122
9.2.3. Stitching Zones	123
9.2.4. Using the Intersect Boundary Zones Dialog Box	125
9.2.5. Text Commands for Boundary Intersection	126
9.3. Modifying the Boundary Mesh	128
9.3.1. Using the Modify Boundary Dialog Box	128
9.3.2. Operations Performed: Modify Boundary Dialog Box	128
9.3.3. Locally Remeshing a Boundary Zone or Faces	135
9.3.4. Text Commands for Boundary Modification	135
9.4. Improving Boundary Surfaces	138
9.4.1. Improving the Boundary Surface Quality	138
9.4.2. Smoothing the Boundary Surface	138
9.4.3. Swapping Face Edges	138
9.4.4. Text Commands for Improving Boundary Surfaces	139
9.5. Refining the Boundary Mesh	139
9.5.1. Procedure for Refining Boundary Zone(s)	139
9.5.2. Text Commands for Boundary Zone Refinement	141
9.6. Creating and Modifying Features	142
9.6.1. Creating Edge Loops	142
9.6.2. Modifying Edge Loops	145
9.6.3. Using the Feature Modify Dialog Box	146
9.6.4. Text Commands for Creating and Modifying Features	148
9.7. Remeshing Boundary Zones	149
9.7.1. Creating Edge Loops	150
9.7.2. Modifying Edge Loops	150
9.7.3. Remeshing Boundary Face Zones	150
9.7.4. Using the Surface Retriangulation Dialog Box	151
9.7.5. Text Commands for Remeshing	152
9.8. Faceted Stitching of Boundary Zones	154
9.9. Triangulating Boundary Zones	155
9.10. Separating Boundary Zones	155
9.10.1. Methods for Separating Face Zones	155
9.10.2. Text Commands for Separating Face Zones	158
9.11. Projecting Boundary Zones	159
9.11.1. Text Commands for Projecting Boundary Zones	159
9.12. Creating Groups	159
9.12.1. Text Commands for User-Defined Groups	160
9.13. Manipulating Boundary Zones	161
9.13.1. Text Commands for Manipulating Boundary Zones	161
9.14. Manipulating Boundary Conditions	162
9.14.1. Text Commands for Manipulating Boundary Conditions	163
9.15. Creating Surfaces	163
9.15.1. Creating a Bounding Box	163
9.15.1.1. Using the Bounding Box Dialog Box	163
9.15.2. Creating a Planar Surface Mesh	164
9.15.2.1. Using the Plane Surface Dialog Box	165
9.15.3. Creating a Cylinder/Frustum	166

9.15.3.1. Using the Cylinder Dialog Box	168
9.15.4. Creating a Swept Surface	169
9.15.4.1. Using the Swept Surface Dialog Box	169
9.15.5. Creating a Revolved Surface	170
9.15.5.1. Using the Revolved Surface Dialog Box	170
9.15.6. Creating Periodic Boundaries	171
9.15.7. Text Commands for Creating Surfaces	172
9.16. Additional Boundary Mesh Text Commands	172
10. Wrapping Boundaries	177
10.1. The Wrapping Process	178
10.2. Examining and Repairing the Input Geometry	179
10.3. Initializing the Cartesian Grid	182
10.4. Examining the Cartesian Grid for Leaks	186
10.4.1. Automatic Leak Detection	187
10.4.2. Manual Leak Detection	188
10.5. Extracting the Wrapper Surface	191
10.6. Checking the Quality of the Wrapper Surface	194
10.7. Post Wrapping Improvement Operations	195
10.7.1. Coarsening Options	195
10.7.2. Post Wrap Options	196
10.7.3. Zone Options	202
10.7.4. Expert Options	204
10.8. Text Commands for the Wrapper	206
11. Creating a Mesh	211
11.1. Choosing the Meshing Strategy	211
11.1.1. Boundary Mesh Containing Only Triangular Faces	212
11.1.2. Mixed Boundary Mesh	213
11.1.3. Hexcore Mesh	214
11.1.4. CutCell Mesh	214
11.1.5. Additional Meshing Tasks	215
11.1.6. Inserting Isolated Nodes into a Tet Mesh	217
11.2. Using the Auto Mesh Dialog Box	219
11.2.1. Text Commands for Auto Mesh	220
11.3. Generating Pyramids	220
11.3.1. Creating Pyramids	221
11.3.2. Zones Created During Pyramid Generation	222
11.3.3. Text Commands for Generating Pyramids	223
11.3.4. Pyramid Meshing Problems	223
11.4. Creating a Non-Conformal Interface	225
11.4.1. Separating the Non-Conformal Interface Between Cell Zones	225
11.4.2. Text Commands for Creating a Non-Conformal Interface	226
11.5. Creating a Heat Exchanger Zone	226
11.6. Parallel Meshing	227
11.6.1. Text Commands	228
12. Generating Prisms	229
12.1. Overview	229
12.2. Procedure for Generating Prisms	230
12.3. Prism Meshing Options	232
12.3.1. Growing Prisms Simultaneously from Multiple Zones	232
12.3.2. Growing Prisms on a Two-Sided Wall	234
12.3.3. Detecting Proximity and Collision	235
12.3.4. Ignoring Invalid Normals	238

12.3.5. Splitting Prism Layers	238
12.3.6. Preserving Orthogonality	239
12.4. Zones Created During Prism Generation	239
12.5. The Prism Generation Process	240
12.6. Using Adjacent Zones as the Sides of Prisms	241
12.7. Direction Vectors	244
12.8. Offset Distances	247
12.9. Improving Prism Quality	249
12.9.1. Edge Swapping and Smoothing	249
12.9.2. Node Smoothing	250
12.9.3. Improving the Prism Quality	250
12.9.4. Removing Poor Quality Cells	251
12.9.5. Improving Warp	252
12.10. Text Commands for Generating Prisms	252
12.11. Prism Meshing Problems	261
13. Generating Thin Volume Mesh	265
13.1. Overview	265
13.2. Procedure for Generating a Thin Volume Mesh	265
13.3. Text Command for Generating Thin Volume Mesh	266
14. Generating Tetrahedral Meshes	267
14.1. Automatically Creating a Tetrahedral Mesh	267
14.1.1. Automatic Meshing Procedure for Tetrahedral Meshes	267
14.1.2. Using the Auto Mesh Tool	269
14.1.3. Automatic Meshing of Multiple Cell Zones	269
14.1.4. Automatic Meshing for Hybrid Meshes	270
14.1.5. Further Mesh Improvements	270
14.2. Manually Creating a Tetrahedral Mesh	271
14.2.1. Manual Meshing Procedure for Tetrahedral Meshes	271
14.3. Initializing the Tetrahedral Mesh	274
14.3.1. Using the Tet Dialog Box	274
14.3.2. Text Commands for Initializing the Mesh	275
14.4. Refining the Tetrahedral Mesh	276
14.4.1. Using Local Refinement Regions	277
14.4.2. Using the Tet Dialog Box	278
14.4.3. Text Commands for Setting Refinement Controls	279
14.5. Additional Text Commands for Tetrahedral Mesh Generation	281
14.6. Common Tetrahedral Meshing Problems	282
15. Generating the Hexcore Mesh	285
15.1. Hexcore Meshing Procedure	285
15.2. Using the Hexcore Dialog Box	286
15.3. Controlling Hexcore Parameters	287
15.3.1. Defining Hexcore Extents	287
15.3.2. Hexcore to Selected Boundaries	288
15.3.3. Only Hexcore	289
15.3.4. Maximum Cell Length	290
15.3.5. Buffer Layers	291
15.3.6. Peel Layers	291
15.3.7. Local Refinement Regions	292
15.4. Text Commands for Hexcore Meshing	292
16. Generating the CutCell Mesh	295
16.1. The CutCell Meshing Process	295
16.2. Using the CutCell Dialog Box	300

16.2.1. Handling Zero-Thickness Walls	301
16.2.2. Handling Overlapping Surfaces	302
16.2.3. Resolving Thin Regions	303
16.3. Improving the CutCell Mesh	304
16.4. Post CutCell Mesh Generation Cleanup	305
16.5. Generating Prisms for the CutCell Mesh	305
16.6. The Cut-Tet Workflow	309
16.7. Text Commands for CutCell Meshing	311
17. Object-Based Meshing	317
17.1. Object-Based Meshing Workflow	317
17.1.1. Using the Mesh Generation Task Page	318
17.2. Preparing the Geometry	319
17.2.1. Using a Bounding Box	320
17.2.2. Closing Annular Gaps in the Geometry	320
17.2.3. Creating Capping Surfaces	320
17.2.4. Defining Material Points	322
17.2.5. Defining Size Functions	323
17.2.6. Using User-Defined Groups	323
17.3. Defining Objects	323
17.4. Diagnostics	323
17.4.1. Geometry Issues	324
17.4.2. Face Connectivity Issues	324
17.4.3. Quality Checking	325
17.5. Fixing Holes in Objects	325
17.6. Wrapping Objects	327
17.6.1. Object Wrapping Options	329
17.6.1.1. Wrapping Selected Objects	329
17.6.1.2. Creating a New Wrap Object	329
17.6.1.3. Resolving Thin Regions During Object Wrapping	331
17.6.2. Repairing Wrap Objects	331
17.6.2.1. Detecting Holes in the Wrap Object	331
17.6.2.2. Removing Gaps Between Wrap Objects	331
17.6.2.3. Removing Thickness in Wrap Objects	333
17.6.2.4. Improving Feature Capture For Wrap Objects	335
17.7. Sewing Objects	335
17.7.1. Improving Wrap Objects	337
17.7.2. Resolving Thin Regions	337
17.7.3. Processing Slits	337
17.8. Improving the Mesh Objects	338
17.8.1. Removing Voids	338
17.8.2. Improving Feature Capture For Mesh Objects	338
17.9. Build Topology	338
17.9.1. Text commands for Build Topology	341
17.10. Generating the Volume Mesh Based on Mesh Objects	342
17.11. Text Commands for Object Based Meshing	343
18. Improving the Mesh	347
18.1. Smoothing Nodes	347
18.1.1. Laplace Smoothing	347
18.1.2. Variational Smoothing	348
18.1.3. Skewness-Based Smoothing	348
18.1.4. Text Commands for Smoothing	348
18.2. Swapping	349

18.2.1. Text Commands for Swapping	350
18.3. Improving the Mesh	350
18.4. Removing Slivers from a Tetrahedral Mesh	350
18.4.1. Automatic Sliver Removal	351
18.4.2. Removing Slivers Manually	351
18.4.3. Text Commands for Sliver Removal	352
18.5. Modifying Cells	353
18.5.1. Using the Modify Cells Dialog Box	353
18.5.2. Text Commands for Modifying Cell Zones	355
18.6. Moving Nodes	356
18.6.1. Automatic Correction	356
18.6.2. Semi-Automatic Correction	357
18.6.3. Repairing Negative Volume Cells	358
18.6.4. Text Commands for Moving Nodes	358
18.7. Cavity Remeshing	358
18.7.1. Tetrahedral Cavity Remeshing	359
18.7.2. Hexcore Cavity Remeshing	361
18.7.3. Text Commands for Cavity Remeshing	363
18.8. Manipulating Cell Zones	364
18.8.1. Active Zones and Cell Types	364
18.8.2. Copying and Moving Cell Zones	364
18.8.3. Text Commands for Manipulating Cell Zones	365
18.9. Manipulating Cell Zone Conditions	368
18.9.1. Text Commands for Manipulating Cell Zone Conditions	368
18.10. Using Domains to Group and Mesh Boundary Faces	368
18.10.1. Using Domains	368
18.10.2. Defining Domains	369
18.10.3. Text Commands for Domains	369
18.11. Checking the Mesh	370
18.12. Checking the Mesh Quality	371
18.13. Clearing the Mesh	371
19. Examining the Mesh	373
19.1. Displaying the Grid	373
19.1.1. Generating the Grid Display Using the Display Grid Dialog Box	374
19.1.2. Grid Display Options	374
19.1.3. Text Commands for Displaying the Grid	375
19.1.4. Text Commands for Grid Colors	378
19.1.5. Text Commands for Style Attributes	378
19.1.6. Displaying Objects	379
19.1.6.1. Text Commands for Displaying Objects	379
19.2. Checking Face Distribution	380
19.2.1. Text Commands for Face Distribution	380
19.3. Checking Cell Distribution	380
19.3.1. Text Commands for Cell Distribution	380
19.4. Modifying the Attributes of the Plot Axes	381
19.4.1. Text Commands for Modifying Axes Attributes	381
19.5. Controlling Display Options	381
19.5.1. Modifying the Display Options	382
19.5.2. Text Commands for Controlling Display Options	383
19.6. Modifying the View	386
19.6.1. Text Commands for Modifying the View	386
19.7. Adding Lights	387

19.7.1. Enabling Lighting Effects	387
19.7.2. Text Commands for Adding Lights	388
19.8. Composing a Scene	389
19.8.1. Changing the Display Properties	389
19.8.2. Transforming Geometric Entities in a Scene	389
19.8.3. Using the Scene Description Dialog Box	389
19.8.4. Text Commands for Scene Description	391
19.9. Controlling the Mouse Buttons	391
19.9.1. Button Functions	392
19.9.2. Text Commands for Selecting Mouse Buttons	392
19.10. Controlling the Mouse Probe Functions	393
19.10.1. Text Commands for Mouse Probe Selection	393
19.11. Annotating the Display	393
19.11.1. Text Commands for Text Annotation	393
19.12. Shortcuts for Selecting Zones	394
19.13. Setting Default tgvars	395
20. Reporting Mesh Statistics	397
20.1. Reporting the Mesh Size	397
20.1.1. Text Commands for Reporting Mesh Size	397
20.2. Reporting Face Limits	397
20.2.1. Text Commands for Reporting Face Limits	398
20.3. Reporting Cell Limits	398
20.3.1. Text Commands for Reporting Cell Limits	398
20.4. Reporting Boundary Cell Limits	398
20.4.1. Text Commands for Reporting Boundary Cell Limits	399
20.5. Mesh Quality	399
20.5.1. Quality Measures	399
20.5.2. Text Commands for Selecting the Quality Measure	406
20.6. Printing Grid Information	406
20.6.1. Boundary Node	407
20.6.2. Node	407
20.6.3. Boundary Face	407
20.6.4. Cell	408
20.6.5. Face	408
20.7. Additional Text Commands for Reporting	408
21. Task Page, Menu, and Dialog Box Reference Guide	411
21.1. Mesh Generation Task Page	411
21.1.1. Bounding Box Dialog Box	416
21.1.1.1. Zone Type Dialog Box	417
21.1.2. Cylinder Dialog Box	417
21.1.2.1. Object/Zone Prefix Dialog Box	419
21.1.3. Capping Surface Dialog Box	420
21.1.4. Size Functions Dialog Box	421
21.1.5. Material Points Dialog Box	426
21.1.5.1. Create Material Point Dialog Box	427
21.1.6. User Defined Groups Dialog Box	428
21.1.6.1. The Group Name Dialog Box	429
21.1.7. Manage Objects Dialog Box	430
21.1.8. Diagnostic Tools Dialog Box	437
21.1.9. Fix Holes Dialog Box	448
21.1.10. Remove Gaps Dialog Box	452
21.1.11. Wrap Dialog Box	454

21.1.12. Sew Dialog Box	457
21.1.13. Improve Dialog Box	459
21.1.14. Build Topology Dialog Box	460
21.1.15. Auto Mesh Dialog Box	464
21.1.16. CutCell Dialog Box	466
21.2. File Menu	468
21.2.1. File/Read/Mesh...	468
21.2.2. File/Read/Case...	468
21.2.3. File/Read/Boundary Mesh...	468
21.2.4. File/Read/Size Field...	468
21.2.5. File/Read/Scheme...	468
21.2.6. File/Read/Journal...	468
21.2.7. File/Read/Domains...	468
21.2.8. File/Write/Mesh...	468
21.2.9. File/Write/Case...	468
21.2.10. File/Write/Boundaries...	468
21.2.10.1. The Write Boundaries Dialog Box	469
21.2.11. File/Write/Domains...	469
21.2.12. File/Write/Size Field...	469
21.2.13. File/Write/Start Journal...	469
21.2.14. File/Write/Stop Journal	469
21.2.15. File/Write/Start Transcript...	470
21.2.16. File/Write/Stop Transcript	470
21.2.17. File/Import/ANSYS prep7/cdb/Surface...	470
21.2.18. File/Import/ANSYS prep7/cdb/Volume...	470
21.2.19. File/Import/CGNS/Surface...	470
21.2.20. File/Import/CGNS/Volume...	470
21.2.21. File/Import/FIDAP neutral/Surface...	470
21.2.22. File/Import/FIDAP neutral/Volume...	470
21.2.23. File/Import/GAMBIT neutral/Surface...	470
21.2.24. File/Import/GAMBIT neutral/Volume...	470
21.2.25. File/Import/HYPERMESH Ascii/Surface...	470
21.2.26. File/Import/HYPERMESH Ascii/Volume...	471
21.2.27. File/Import/IDEAS universal/Surface...	471
21.2.28. File/Import/IDEAS universal/Volume...	471
21.2.29. File/Import/NASTRAN/Surface...	471
21.2.30. File/Import/NASTRAN/Volume...	471
21.2.31. File/Import/PATRAN neutral/Surface...	471
21.2.32. File/Import/PATRAN neutral/Volume...	471
21.2.33. File/Import/CAD...	471
21.2.33.1. Import CAD Geometry Dialog Box	471
21.2.33.2. CAD Options Dialog Box	473
21.2.34. File/Import/Fluent 2D Mesh...	479
21.2.35. File/Import/Options...	479
21.2.35.1. Filter Options Dialog Box	480
21.2.36. File/Save Picture...	481
21.2.36.1. Save Picture Dialog Box	481
21.3. Boundary Menu	482
21.3.1. Boundary/Merge Nodes...	482
21.3.1.1. Merge Boundary Nodes Dialog Box	482
21.3.2. Boundary/Intersect...	484
21.3.2.1. Intersect Boundary Zones Dialog Box	484

21.3.3. Boundary/Modify...	486
21.3.3.1. Modify Boundary Dialog Box	486
21.3.3.2. Create Boundary Zone Dialog Box	490
21.3.3.3. Move Nodes Dialog Box	491
21.3.4. Boundary/Wrap...	492
21.3.4.1. Boundary Wrapper Dialog Box	492
21.3.4.2. Wrapper Refinement Region Dialog Box	506
21.3.4.3. Pan Regions Dialog Box	508
21.3.4.4. Trace Path Dialog Box	509
21.3.4.5. Filter Features Dialog Box	509
21.3.5. Boundary/Mesh/Improve...	510
21.3.5.1. Boundary Improve Dialog Box	510
21.3.6. Boundary/Mesh/Refine...	513
21.3.6.1. Refine Boundary Zones Dialog Box	513
21.3.6.2. Boundary Refinement Region Dialog Box	515
21.3.7. Boundary/Mesh/Feature...	516
21.3.7.1. Feature Modify Dialog Box	516
21.3.8. Boundary/Mesh/Remesh...	520
21.3.8.1. Surface Retriangulation Dialog Box	520
21.3.9. Boundary/Mesh/Faceted Stitch...	523
21.3.9.1. Faceted Stitch Dialog Box	523
21.3.10. Boundary/Mesh/Triangulate...	523
21.3.10.1. Triangulate Zones Dialog Box	524
21.3.11. Boundary/Zone/Separate...	524
21.3.11.1. Separate Face Zones Dialog Box	524
21.3.12. Boundary/Zone/Project...	526
21.3.12.1. Project Face Zone Dialog Box	526
21.3.13. Boundary/Zone/Groups...	528
21.3.14. Boundary/Manage...	528
21.3.14.1. Manage Face Zones Dialog Box	528
21.3.15. Boundary/Boundary Conditions...	532
21.3.15.1. Boundary Conditions Dialog Box	532
21.3.16. Boundary/Create/Bounding Box...	533
21.3.17. Boundary/Create/Plane Surface...	533
21.3.17.1. Plane Surface Dialog Box	533
21.3.18. Boundary/Create/Cylinder...	534
21.3.19. Boundary/Create/Swept Surface...	535
21.3.19.1. Swept Surface Dialog Box	535
21.3.20. Boundary/Create/Revolved Surface...	536
21.3.20.1. Revolved Surface Dialog Box	536
21.3.21. Boundary/Create/Periodic...	537
21.3.21.1. Make Periodic Boundaries Dialog Box	537
21.4. Mesh Menu	539
21.4.1. Mesh/Auto Mesh...	539
21.4.2. Mesh/Prisms...	539
21.4.2.1. Prisms Dialog Box	539
21.4.2.2. Prisms Growth Options Dialog Box	549
21.4.2.3. Orient Normals Dialog Box	551
21.4.3. Mesh/Thin Volume Mesh...	551
21.4.3.1. Thin Volume Mesh Dialog Box	551
21.4.4. Mesh/Pyramids...	552
21.4.4.1. Pyramids Dialog Box	553

21.4.5. Mesh/Non Conforms...	555
21.4.5.1. Non Conforms Dialog Box	555
21.4.6. Mesh/Tet...	556
21.4.6.1. Tet Dialog Box	556
21.4.6.2. Tet Init Controls Dialog Box	559
21.4.6.3. Tet Refine Controls Dialog Box	559
21.4.6.4. Tet Refinement Region Dialog Box	561
21.4.7. Mesh/Hexcore...	563
21.4.7.1. Hexcore Dialog Box	563
21.4.7.2. Outer Domain Parameters Dialog Box	566
21.4.7.3. Hexcore Refinement Region Dialog Box	568
21.4.8. Mesh/CutCell...	569
21.4.9. Mesh/Create/Heat Exchanger...	569
21.4.9.1. Heat Exchanger Mesh Dialog Box	569
21.4.9.2. Zone Prefix Dialog Box	571
21.4.10. Mesh/Tools/Tet Improve...	571
21.4.10.1. Tet Improve Dialog Box	571
21.4.11. Mesh/Tools/Cell Modify...	574
21.4.11.1. Modify Cells Dialog Box	574
21.4.12. Mesh/Tools/Auto Node Move...	577
21.4.12.1. Auto Node Move Dialog Box	577
21.4.13. Mesh/Tools/Cavity Remesh...	579
21.4.13.1. Cavity Remesh Dialog Box	579
21.4.14. Mesh/Tools/Prism/Improve...	581
21.4.14.1. Prism Improve Dialog Box	581
21.4.15. Mesh/Tools/Prism/Split...	583
21.4.15.1. Prism Split Dialog Box	583
21.4.16. Mesh/Tools/Prism/Post Ignore...	584
21.4.16.1. Prism Post Ignore Dialog Box	585
21.4.17. Mesh/Tools/Prism/Tet Improve Cavity	588
21.4.17.1. Prism Tet Improve Cavity Dialog Box	588
21.4.18. Mesh/Manage...	590
21.4.18.1. Manage Cell Zones Dialog Box	590
21.4.19. Mesh/Cell Zone Conditions...	593
21.4.19.1. Cell Zone Conditions Dialog Box	593
21.4.20. Mesh/Domains...	593
21.4.20.1. Domains Dialog Box	593
21.4.21. Mesh/Clear	595
21.4.22. Mesh/Check	595
21.4.23. Mesh/Check Quality	596
21.5. Display Menu	596
21.5.1. Display/Grid...	596
21.5.1.1. Display Grid Dialog Box	596
21.5.1.2. Grid Colors Dialog Box	604
21.5.1.3. Style Attributes Dialog Box	605
21.5.2. Display/Plot/Face Distribution...	607
21.5.2.1. Face Distribution Dialog Box	607
21.5.2.2. Axes - Histogram Dialog Box	610
21.5.3. Display/Plot/Cell Distribution...	612
21.5.3.1. Cell Distribution Dialog Box	612
21.5.4. Display/Options...	613
21.5.4.1. Display Options Dialog Box	614

21.5.5. Display/Views...	617
21.5.5.1. Views Dialog Box	617
21.5.5.2. Write Views Dialog Box	618
21.5.5.3. Mirror Planes Dialog Box	619
21.5.5.4. Camera Parameters Dialog Box	619
21.5.6. Display/Lights...	621
21.5.6.1. Lights Dialog Box	621
21.5.7. Display/Scene...	623
21.5.7.1. Scene Description Dialog Box	623
21.5.7.2. Display Properties Dialog Box	624
21.5.7.3. Transformations Dialog Box	626
21.5.7.4. Bounding Frame Dialog Box	627
21.5.8. Display/Mouse Buttons...	628
21.5.8.1. Mouse Buttons Dialog Box	628
21.5.9. Display/Mouse Probe...	629
21.5.9.1. Mouse Probe Dialog Box	629
21.5.10. Display/Annotate...	630
21.5.10.1. Annotate Dialog Box	630
21.5.11. Display/Selection Helper...	632
21.5.11.1. Zone Selection Helper Dialog Box	632
21.5.12. Display/Controls...	633
21.5.12.1. Controls Dialog Box	633
21.6. Report Menu	633
21.6.1. Report/Mesh Size...	633
21.6.1.1. Report Mesh Size Dialog Box	634
21.6.2. Report/Face Limits...	635
21.6.2.1. Report Face Limits Dialog Box	635
21.6.3. Report/Cell Limits...	636
21.6.3.1. Report Cell Limits Dialog Box	636
21.6.4. Report/Boundary Cell Limits...	637
21.6.4.1. Report Boundary Cell Limits Dialog Box	637
21.6.5. Report/Quality Measure...	638
21.6.5.1. Quality Measure Dialog Box	638
21.7. Parallel Menu	639
21.7.1. Parallel Dialog Box	639
21.8. View Menu	640
21.8.1. View/Toolbars	640
21.8.2. View/Navigation Pane	640
21.8.3. View/Task Page	641
21.8.4. View/Graphics Window	641
21.8.5. View/Embed Graphics Window	641
21.8.6. View>Show All	641
21.8.7. View>Show Only Console	641
21.8.8. View/Graphics Window Layout	641
21.8.9. View/Save Layout	641
21.9. Help Menu	641
21.9.1. Help/User's Guide Contents...	641
21.9.2. Help/PDF	641
21.9.3. Help/Context Sensitive Help...	642
21.9.4. Help/Using Help...	642
21.9.5. Help/Online Technical Resources...	642
21.9.6. Help/License Usage...	642

21.9.7. Help/Version.....	642
21.10. Hot Key Activated Dialog Boxes	642
21.10.1. Local Remesh Dialog Box	642
21.10.2. Remove Boundary Gaps Dialog Box	643
21.10.3. Rename Objects/Zones Dialog Box	644
A. Importing Boundary and Volume Meshes	647
A.1. GAMBIT Meshes	647
A.2. TetraMesher Volume Mesh	647
A.3. Meshes from Third-Party CAD Packages	647
A.3.1. Using the fe2ram Filter to Convert Files	647
A.3.2. I-deas Universal Files	648
A.3.2.1. Recognized I-deas Datasets	648
A.3.2.2. Grouping Elements to Create Zones for a Surface Mesh	649
A.3.2.3. Grouping Nodes to Create Zones for a Volume Mesh	649
A.3.2.4. Periodic Boundaries	649
A.3.2.5. Deleting Duplicate Nodes	649
A.3.3. PATRAN Neutral Files	650
A.3.3.1. Recognized PATRAN Datasets	650
A.3.3.2. Grouping Elements to Create Zones	650
A.3.3.3. Periodic Boundaries	650
A.3.4. ANSYS Files	650
A.3.4.1. Recognized Datasets	651
A.3.4.2. Periodic Boundaries	651
A.3.5. ARIES Files	651
A.3.6. NASTRAN Files	651
A.3.6.1. Recognized NASTRAN Bulk Data Entries	652
A.3.6.2. Periodic Boundaries	652
A.3.6.3. Deleting Duplicate Nodes	652
B. Mesh File Format	653
B.1. Guidelines	653
B.2. Formatting Conventions in Binary Files and Formatted Files	653
B.3. Grid Sections	654
B.3.1. Comment	654
B.3.2. Header	654
B.3.3. Dimensions	655
B.3.4. Nodes	655
B.3.5. Periodic Shadow Faces	656
B.3.6. Cells	657
B.3.7. Faces	658
B.3.8. Edges	660
B.3.9. Face Tree	661
B.3.10. Cell Tree	662
B.3.11. Interface Face Parents	662
B.4. Non-Grid Sections	663
B.4.1. Zone	663
B.5. Example Files	665
C. Shortcut Keys	669
C.1. Arrow Keys	669
C.2. Help Keys	669
C.3. Hot Keys	670
D. Query Functions	677
D.1. Using Boolean Operations with Query Functions	679

D.2. Examples	679
Bibliography	681

Chapter 1: Using This Manual

This preface is divided into the following sections:

- 1.1. The Contents of This Manual
- 1.2. Typographical Conventions Used In This Manual
- 1.3. Contacting Technical Support

1.1. The Contents of This Manual

The ANSYS Fluent Meshing User's Guide tells you what you need to know to use ANSYS Fluent Meshing. At the end in the User's Guide, you will find the bibliography and the index.

Important

Under U. S. and international copyright law, ANSYS, Inc. is unable to distribute copies of the papers listed in the bibliography, other than those published internally by ANSYS, Inc. Use your library or a document delivery service to obtain copies of copyrighted papers.

A brief description of what is in each chapter follows:

- [Introduction to Meshing Mode in ANSYS Fluent \(p. 7\)](#) gives you an introduction to the meshing mode in ANSYS Fluent and an overview of its capabilities.
- [Starting and Executing ANSYS Fluent in Meshing Mode \(p. 11\)](#) provides instructions for starting and executing ANSYS Fluent in meshing mode.
- [Graphical User Interface \(p. 19\)](#) describes the use of the graphical user interface. It also explains how to use the on-line help system.
- [Text User Interface \(p. 47\)](#) describes the use of the text interface and the associated options. See the separate Text Command List for information about specific text interface commands.
- [Reading and Writing Files \(p. 57\)](#) describes the files that can be read and written (including picture files).
- [Size Functions \(p. 85\)](#) describes the size functions available.
- [Meshing Objects and Material Points \(p. 103\)](#) describes the use of meshing objects and material points.
- [Manipulating the Boundary Mesh \(p. 119\)](#) explains the need for a high-quality boundary mesh and describes the various options available for creating such meshes.
- [Wrapping Boundaries \(p. 177\)](#) contains information about creating a high-quality boundary mesh starting from bad surface mesh using the boundary wrapper tool.
- [Creating a Mesh \(p. 211\)](#) describes the meshing strategy and creation of pyramids, non-conformals, and heat exchanger mesh.

- [Generating Prisms \(p. 229\)](#) describes the procedure to create prism mesh. It also explains how to deal with common problems that can be faced while creating prisms.
- [Generating Tetrahedral Meshes \(p. 267\)](#) describes the meshing procedures for tetrahedral meshes.
- [Generating the Hexcore Mesh \(p. 285\)](#) describes the procedure and options for creating hexcore meshes.
- [Generating the CutCell Mesh \(p. 295\)](#) describes the CutCell meshing procedure and options available for CutCell meshing.
- [Object-Based Meshing \(p. 317\)](#) describes the object based meshing workflow.
- [Improving the Mesh \(p. 347\)](#) describes the options available for improving the quality of a volume mesh.
- [Examining the Mesh \(p. 373\)](#) describes the methods available for examining the mesh graphically.
- [Reporting Mesh Statistics \(p. 397\)](#) describes methods for checking the mesh diagnostically.
- [Task Page, Menu, and Dialog Box Reference Guide \(p. 411\)](#) contains the task page, menu, and dialog box reference guide.
- [Appendix A \(p. 647\)](#) describes filters that you can use to convert data from various software packages to a form that can be read.
- [Appendix B \(p. 653\)](#) describes the format of the mesh file.
- [Appendix C \(p. 669\)](#) lists all the hot-keys (shortcut keys) available.
- [Appendix D \(p. 677\)](#) lists the query functions available.

In addition to this User's Guide, you can refer to the following to help you use the meshing mode in ANSYS Fluent:

- Tutorials for release 15.0 are available on the ANSYS Customer Portal. To access tutorials and their input files on the ANSYS Customer Portal, go to <http://support.ansys.com/training>.
- The [Meshing Text Command List](#) provides a brief description of the commands in the text interface.

1.2. Typographical Conventions Used In This Manual

Several typographical conventions are used in the text of this manual to facilitate your learning process.

- Different type styles are used to indicate graphical user interface menu items and text interface menu items (e.g., **Display Grid** dialog box, display/grid command).
- The text interface type style is also used when illustrating exactly what appears on the screen or exactly what you need to type into a field in a dialog box. The information displayed on the screen is enclosed in a large box to distinguish it from the narrative text, and user inputs are often enclosed in smaller boxes.
- A mini flow chart is used to indicate the menu selections that lead you to a specific dialog box. For example,

Display → Grid...

indicates that the **Grid...** menu item can be selected from the **Display** pull-down menu.

In this manual, mini flow charts usually precede a description of a dialog box or a screen illustration showing how to use the dialog box. They allow you to look up information about a dialog box and quickly determine how to access it without having to search the preceding material.

- The menu selections that will lead you to a particular dialog box are also indicated (usually within a paragraph) using a " / ". For example, **Display/Grid...** tells you to choose the **Grid...** menu item from the **Display** menu.

1.3. Contacting Technical Support

If you encounter difficulties while using ANSYS Fluent Meshing, first refer to the section(s) of the manual containing information on the commands you are trying to use or the type of problem you are trying to solve. The product documentation is available from the online help, or from the ANSYS Customer Portal (www.ansys.com/customerportal).

If you encounter an error, write down the exact error message that appeared and note as much information as you can about what you were doing in ANSYS Fluent Meshing.

Technical Support for ANSYS, Inc. products is provided either by ANSYS, Inc. directly or by one of our certified ANSYS Support Providers. Please check with the ANSYS Support Coordinator (ASC) at your company to determine who provides support for your company, or go to www.ansys.com and select **Contact ANSYS > Contacts and Locations**.

If your support is provided by ANSYS, Inc. directly, Technical Support can be accessed quickly and efficiently from the ANSYS Customer Portal, which is available from the ANSYS Website (www.ansys.com) under **Support > Customer Portal**. The direct URL is: support.ansys.com.

One of the many useful features of the Customer Portal is the Knowledge Resources Search, which can be found on the Home page of the Customer Portal.

Systems and installation Knowledge Resources are easily accessible via the Customer Portal by using the following keywords in the search box: Systems/Installation. These Knowledge Resources provide solutions and guidance on how to resolve installation and licensing issues quickly.

NORTH AMERICA

All ANSYS, Inc. Products

Web: Go to the ANSYS Customer Portal (<http://support.ansys.com>) and select the appropriate option.

Toll-Free Telephone: 1.800.711.7199

Fax: 1.724.514.5096

Support for University customers is provided only through the ANSYS Customer Portal.

GERMANY

ANSYS Mechanical Products

Telephone: +49 (0) 8092 7005-55 (CADFEM)

Email: support@cadfem.de

All ANSYS Products

Web: Go to the ANSYS Customer Portal (<http://support.ansys.com>) and select the appropriate option.

National Toll-Free Telephone:

German language: 0800 181 8499

English language: 0800 181 1565

Austria: 0800 297 835

Switzerland: 0800 546 318

International Telephone:

German language: +49 6151 152 9981

English language: +49 6151 152 9982

Email: support-germany@ansys.com

UNITED KINGDOM

All ANSYS, Inc. Products

Web: Go to the ANSYS Customer Portal (<http://support.ansys.com>) and select the appropriate option.

Telephone: Please have your Customer or Contact ID ready.

UK: 0800 048 0462

Republic of Ireland: 1800 065 6642

Outside UK: +44 1235 420130

Email: support-uk@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

JAPAN

CFX , ICEM CFD and Mechanical Products

Telephone: +81-3-5324-8333

Fax: +81-3-5324-7308

Email:

CFX: japan-cfx-support@ansys.com;

Mechanical: japan-ansys-support@ansys.com

Fluent Products

Telephone: +81-3-5324-7305

Email:

Fluent: japan-fluent-support@ansys.com;

Polyflow: japan-polyflow-support@ansys.com;

FfC: japan-ffc-support@ansys.com;

FloWizard: japan-flowizard-support@ansys.com

Icepak

Telephone: +81-3-5324-7444

Email: japan-icepak-support@ansys.com

Licensing and Installation

Email: japan-license-support@ansys.com

INDIA

All ANSYS, Inc. Products

Web: Go to the ANSYS Customer Portal (<http://support.ansys.com>) and select the appropriate option.

Telephone: +91 1 800 209 3475 (toll free) or +91 20 6654 3000 (toll)

Fax: +91 80 6772 2600

Email:

FEA products: feasup-india@ansys.com;

CFD products: cfdsup-india@ansys.com;

Ansoft products: ansoftsup-india@ansys.com;

Installation: installation-india@ansys.com

FRANCE

All ANSYS, Inc. Products

Web: Go to the ANSYS Customer Portal (<http://support.ansys.com>) and select the appropriate option.

Toll-Free Telephone: +33 (0) 800 919 225 **Toll Number:** +33 (0) 170 489 087

Email: support-france@ansys.com

BELGIUM

All ANSYS Products

Web: Go to the ANSYS Customer Portal (<http://support.ansys.com>) and select the appropriate option.

Telephone: +32 (0) 10 45 28 61

Email: support-belgium@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

SWEDEN

All ANSYS Products

Web: Go to the ANSYS Customer Portal (<http://support.ansys.com>) and select the appropriate option.

Telephone: +44 (0) 870 142 0300

Email: support-sweden@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

SPAIN and PORTUGAL

All ANSYS Products

Web: Go to the ANSYS Customer Portal (<http://support.ansys.com>) and select the appropriate option.

Telephone: +34 900 933 407 (Spain), +351 800 880 513 (Portugal)

Email: support-spain@ansys.com, support-portugal@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

ITALY

All ANSYS Products

Web: Go to the ANSYS Customer Portal (<http://support.ansys.com>) and select the appropriate option.

Telephone: +39 02 89013378

Email: support-italy@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

Chapter 2: Introduction to Meshing Mode in ANSYS Fluent

When in meshing mode, ANSYS Fluent functions as a robust, unstructured grid generation program that can handle grids of virtually unlimited size and complexity, consisting of tetrahedral, hexahedral, prismatic, or pyramidal cells. Unstructured grid generation techniques couple basic geometric building blocks with extensive geometric data to automate the grid generation process.

A number of tools are available for checking and repairing the boundary mesh to ensure a good starting point for generating the volume mesh. The volume mesh can be generated from the boundary mesh using one of the approaches described.

The user interface is written in the Scheme language which is a dialect of LISP. Most features are accessible through the graphical interface or the interactive menu interface. The advanced user can customize and enhance the interface by adding or changing the Scheme functions.

- 2.1. Meshing Approach
- 2.2. Program Capabilities
- 2.3. ANSYS Fluent Documentation

2.1. Meshing Approach

There are two principal approaches to creating meshes in ANSYS Fluent Meshing:

- Generate a tetrahedral, hexcore, or hybrid volume mesh from an existing boundary mesh. In this case, you can import a boundary mesh from ANSYS Meshing or a third-party mesh generation package. Boundary meshes created in CAD/CAE packages can be imported using the appropriate menu item in the **File/Import** pull-down submenu (or the associated text commands) or converted using the appropriate stand-alone grid filter.
- Generate a tetrahedral, hexcore, or hybrid volume mesh based on meshing objects from a faceted geometry (from CAD or the .tgc format from ANSYS Meshing). In this case, you need to create a conformally connected surface mesh using the object wrapping and sewing operations before generating the volume mesh. You can alternatively use the CutCell mesher to directly create a hex-dominant volume mesh for the geometry (imported from CAD or the .tgc format from ANSYS Meshing) based on meshing objects.

When the mesh generation is complete, you can transfer the mesh to solution mode using the Mode toolbar or the command switch-to-solution-mode. The remaining operations like setting boundary conditions, defining fluid properties, executing the solution, and viewing and postprocessing the results are performed in solution mode (see the [User's Guide](#) for details).

2.2. Program Capabilities

When in meshing mode, ANSYS Fluent functions as a robust, unstructured volume mesh generator with the following meshing capabilities:

- Generates volume mesh that can be transferred to solution mode in ANSYS Fluent.
- Uses the Delaunay triangulation method for tetrahedra.

- Uses the advancing layer method for prisms.
- Generates hexcore mesh.
- Has a robust surface wrapper tool.
- Includes size functions which can produce ideal size distribution for many CFD calculations.
- Can directly create a hex-dominant mesh on faceted geometry (using the CutCell mesher).
- Can export polyhedral cells.
- Has tools for checking, repairing, and improving boundary mesh to ensure a good starting point for the mesh.
- Is capable of manipulating face/cell zones.
- Is flexible—it allows the most appropriate cell type to be used to generate the volume mesh:
 - Tet meshes are suitable for complex geometries.
 - Hexcore meshes can combine the flexibility of tet, hex, and prism meshes with a smaller cell count and higher hex to tet ratio.
 - CutCell (hex-dominant) meshes can be directly created from faceted geometry and can also be combined with prism layers.
- Hybrid meshes:
 - Prism layers near walls allow proper boundary layer resolution.
 - Allows flow alignments with grid lines.
 - Generates smaller volume mesh with highly stretched prismatic elements.
- Non-conformal meshes:
 - Suitable for studies involving selective replacement of parts.
 - Meshes generated separately can be glued together.

2.3. ANSYS Fluent Documentation

ANSYS Fluent documentation is available through the online help system. The online help system provides access to the ANSYS Fluent documentation, using the ANSYS help viewer, whether you are working in ANSYS Fluent or not.

2.3.1. Accessing the ANSYS Fluent Documentation

You can access the ANSYS Fluent documentation through the ANSYS help viewer or download PDF versions of the documentation files.

2.3.1.1. Accessing the Documentation Files Using the ANSYS Help Viewer

2.3.1.2. Downloading and Installing the PDF Documentation Files

2.3.1.1. Accessing the Documentation Files Using the ANSYS Help Viewer

To start the ANSYS help viewer, go to the following location from the Windows **Start** menu:

Start > Program Files > ANSYS 15.0 > Help > ANSYS Help

The ANSYS help viewer provides access to documentation for most ANSYS products.

To navigate to the ANSYS Fluent documentation, do the following:

1. Scroll down to **ANSYS** in the left-hand panel.
2. Expand the ANSYS Fluent documentation set by clicking the icon next to **ANSYS**.
3. Click a document title to display the table of contents for the selected document.
4. To find specific information, you can do any of the following:
 - a. In the Table of Contents, click the icon () next to a title to expand the tree hierarchy, or click an item in the tree hierarchy to display the corresponding information.
 - b. In the Search option on the toolbar, select either **Basic** or **Advanced Search**, enter a keyword or phrase and click **Search**.

For additional information on the ANSYS Help Viewer, see [Using Help](#).

2.3.1.2. Downloading and Installing the PDF Documentation Files

You can download the PDF documentation from the ANSYS Customer Portal (support.ansys.com/doc-downloads).

If you do not have Acrobat Reader, you can download it for free from Adobe (www.adobe.com).

Chapter 3: Starting and Executing ANSYS Fluent in Meshing Mode

This chapter provides instructions for starting and executing ANSYS Fluent in meshing mode.

- 3.1. Starting ANSYS Fluent in Meshing Mode
- 3.2. Parallel Processing in Meshing Mode
- 3.3. Exiting the Program

3.1. Starting ANSYS Fluent in Meshing Mode

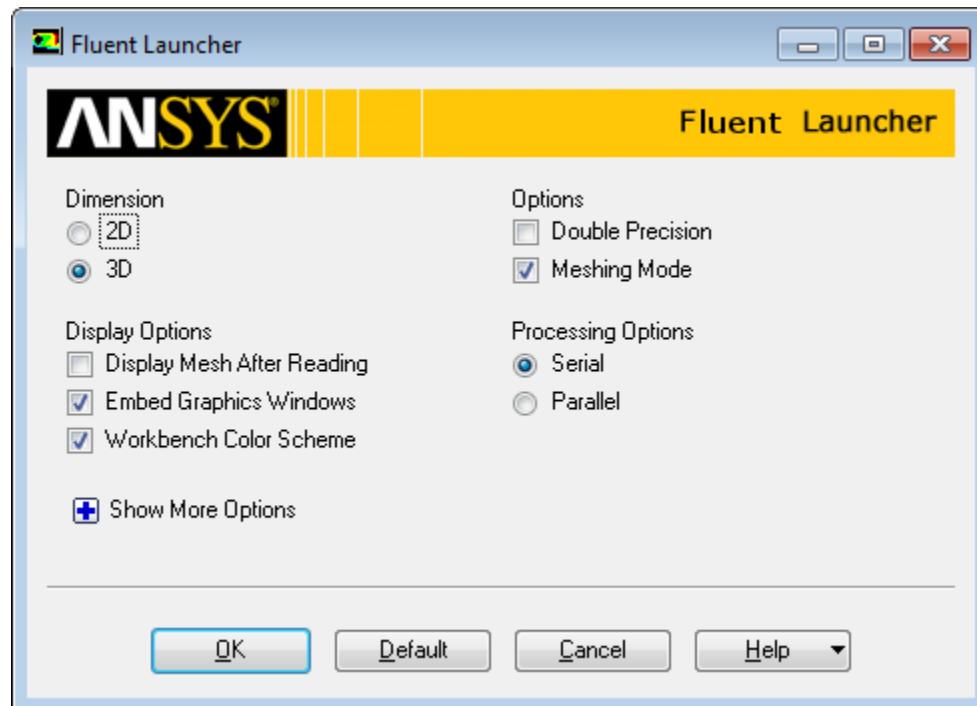
The following sections describe how to start ANSYS Fluent in meshing mode:

- 3.1.1. Using Fluent Launcher
- 3.1.2. Starting on a Windows System
- 3.1.3. Starting on a Linux System
- 3.1.4. Starting the Dual Process Build
- 3.1.5. Startup Options

3.1.1. Using Fluent Launcher

When you start ANSYS Fluent from the Linux or Windows command line with no arguments, from the Windows Programs menu, or from the Windows desktop, Fluent Launcher will appear. You can specify additional start up options in Fluent Launcher. See [Starting ANSYS Fluent Using Fluent Launcher](#) in the [Getting Started Guide](#) for details.

Figure 3.1: Fluent Launcher



1. Ensure that **3D** is selected under **Dimension**.

Important

The meshing mode is available only for 3D.

2. Enable **Meshing Mode** to start ANSYS Fluent in meshing mode.

Important

The meshing mode is always run in double precision mode. The option for **Double Precision** applies to the precision for solution mode only.

You can also choose the following options:

- You can choose to use various job schedulers using the **Use Job Scheduler** option (for example, the Microsoft Job Scheduler for Windows, or LSF, SGE, and PBS Pro on Linux). For more information about using Fluent Launcher with job schedulers, see [Starting ANSYS Fluent Using Fluent Launcher](#), as well as [Setting Parallel Scheduler Options in Fluent Launcher](#) in the [Fluent User's Guide](#).
- You can choose to run parallel simulations on Linux clusters, via the Windows interface using the **Use Remote Linux Nodes** option (see [Setting Remote Options in Fluent Launcher](#) for details).

3. Set the display options.

- The **Display Mesh After Reading** option allows you to have ANSYS Fluent automatically display the mesh immediately after reading the mesh file(s). This option is disabled by default.

This option is applicable only to volume meshes and not surface meshes. All of the boundary zones except for the interior zones will be displayed.

- The **Embed Graphics Windows** option allows you to have the graphics windows embedded within the ANSYS Fluent application window (enabled by default), rather than having floating graphics windows.
- You can use the default **Workbench Color Scheme** for the graphics windows (i.e., a blue background), rather than the classic black background.

4. Select **Serial** or **Parallel** for **Processing Options**, as appropriate.

Refer to [Parallel Processing in Meshing Mode \(p. 15\)](#) for details about using meshing mode in parallel.

If you select the **Show More Options** button, Fluent Launcher expands to reveal more options. Note that once Fluent Launcher expands, the **Show More Options** button becomes the **Show Fewer Options** button, allowing you to hide the additional options.

See [Starting ANSYS Fluent Using Fluent Launcher](#) in the [Getting Started Guide](#) for details on the Fluent Launcher options available.

3.1.2. Starting on a Windows System

There are two ways to start ANSYS Fluent in the meshing mode on a Windows system:

- Click the **Start** button, select the **All Programs** menu, select the **ANSYS 15.0** menu, select the **Fluid Dynamics** menu, and then select the **Fluent** program item.

Note

If the default ANSYS 15.0 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 15.0** program group. This starts Fluent Launcher (see [Using Fluent Launcher \(p. 11\)](#)).

- Start from a Command Prompt window by typing `fluent 3d -meshing` at the prompt.

Before doing so, ensure that the user environment is modified appropriately so that the command utility will find `fluent`. You can do this by executing the `setenv.exe` program located in the ANSYS Fluent home directory (e.g., `C:\Program Files\ANSYS Inc\v150\fluent\nt-bin\win64`). This program will add the ANSYS Fluent folder to your command search path.

You can also start the parallel version of ANSYS Fluent in meshing mode from the Command Prompt (see [Parallel Processing in Meshing Mode \(p. 15\)](#) for details).

3.1.3. Starting 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 [Using Fluent Launcher \(p. 11\)](#) for details.
- Start the meshing mode from the command line by typing `fluent 3d -meshing` at the prompt.

You can also start the parallel version of ANSYS Fluent in meshing mode from the Command Prompt (see [Parallel Processing in Meshing Mode \(p. 15\)](#) for details).

3.1.4. Starting the Dual Process Build

The dual process build allows you to run Cortex on your local machine (host) and ANSYS Fluent on a remote machine. The advantage of using the dual process build is faster response to graphic actions performed when you use ANSYS Fluent from a remote machine. For example, if you are handling a big mesh (e.g., underhood mesh), you can run ANSYS Fluent on a remote machine with only the display set to your local machine. In this case, the graphics actions (e.g., zoom-in, zoom-out, opening a dialog box, etc.) can be slow if the remote machine is located very far away or if the network connectivity is slow. Using the dual process build of ANSYS Fluent allows you to avoid the slow response of the graphics actions.

To start the dual process build of ANSYS Fluent, do the following:

- Start ANSYS Fluent on your local machine using the command `fluent -serv`.

The ANSYS Fluent window will appear with the version prompt in the console.

2. Type **listen** and press **Enter** on your keyboard.

You will be prompted for a timeout (the period of time to wait for a connection from remote ANSYS Fluent). The default value is 300 seconds. You can also specify the timeout value based on your requirement. Utilize this time to log into the remote machine and to start ANSYS Fluent.

3. Press **Enter** on your keyboard again.

A message will prompt you to start ANSYS Fluent on the remote machine with the following arguments:

`-cx host:p1:p2`

where,

`host` is the name of the host (local) machine on which Cortex is running.

`p1` and `p2` are the two integers indicating the connecting port numbers that are used to communicate information between Cortex on the host machine and ANSYS Fluent on the remote machine.

4. Login to the remote machine and set the display to the host machine.

5. Start ANSYS Fluent from the remote machine using the following command:`fluent 3d -cx host:p1:p2`

The host and port numbers are displayed in the message window.

Note

The GUI commands related to the **File** menu (e.g., reading files, importing files) and other **Select File** dialog boxes do not work for the dual process build. You need to use the TUI commands instead (e.g., `/file/read-mesh`).

Important

- The host cannot be detached and reattached, once the connection is broken the data is lost. You need to save the data if the machine needs to be shut down in between.
 - All graphics information will be sent over the network, so initially it could take a long time to assemble graphical information (especially if the host and remote server are across continents) but after that the graphics manipulation is fast.
-

3.1.5. Startup Options

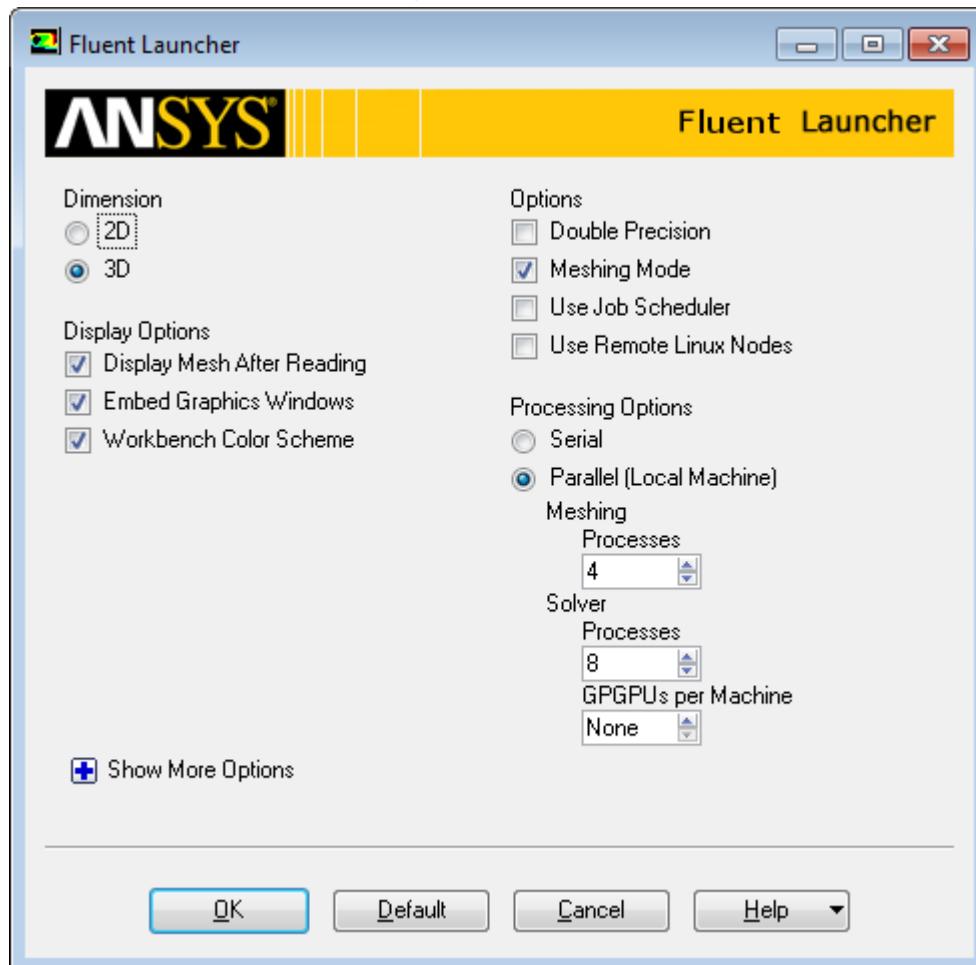
To obtain information about the available startup options, type `fluent -help` in a terminal window (on LINUX systems) or the Command Prompt window (on Windows systems).

See [Command Line Startup Options](#) in the [Getting Started Guide](#) for details about the options available.

3.2. Parallel Processing in Meshing Mode

When starting ANSYS Fluent in meshing mode, you can specify the use of parallel processing for meshing and solution. In order to specify the use of parallel processing, select **Parallel for Processing Options** in the Fluent Launcher. If parallel processing is enabled, the **Parallel** menu item will be visible in the **Menu Bar**.

1. Select **Parallel** under **Processing Options**



2. Enter the number of processes to be used for meshing under **Meshing Processes**.
3. Enter the number of processes to be used for solution under **Solver Processes**. When you change to solution mode, additional processes will be spawned as necessary to bring the total number of processes to this value. This must be set to a value greater than or equal to **Meshing Processes**.

Tip

You can also start the parallel version of ANSYS Fluent in meshing mode from the Command Prompt. To start the parallel version using x processes for meshing and y processes for solution, type `fluent 3d -meshing -tmx -ty` at the prompt, replacing x with the number of meshing processes and y with the number of total solution processes (e.g., `fluent -tm4 -t8` to run using 4 processes for meshing and 8 pro-

cesses for solution).The number of total solution processes must be greater than or equal to the number of meshing processes.

Note

With Parallel processing enabled, some global file read and write operations are affected. Unavailable options will have their menu entry greyed out.

Refer to [Starting Parallel ANSYS Fluent Using Fluent Launcher](#) in the *Fluent User's Guide* for additional details about options for parallel process configuration and [Parallel Processing](#) in the *Fluent User's Guide* for details on parallel processing using ANSYS Fluent.

3.2.1. Dynamically Spawning Processes Between Fluent Meshing and Fluent Solution Modes

For parallel simulations started in meshing mode with more than one process, the Fluent session will be started by using the number of processes requested for meshing. When switching to solution mode, Fluent will automatically spawn the remaining parallel node processes needed to achieve the requested number of total solution processes. Even though automatic spawning is used by default at start up for runs using more than one process, you can always change the number of additional processes to be spawned before switching to solution mode using the `/parallel/spawn-solver-process` text user interface (TUI) command.

This text command prompts you for:

- Total number of desired processes (must be greater than or equal to the number of meshing processes). Fluent will spawn additional processes as necessary.
 - (Linux and mixed Windows/Linux) Interconnect type to be used for the distributed parallel simulation. You can choose from infiniband, myrinet, ethernet, shared memory, or you can retain the default value.
 - Machine list or host file. If you decide to run in parallel using more than one machine, then you should provide the machine list or host file, otherwise, you can skip this option by pressing **Enter**.
 - (Linux and mixed Windows/Linux) Option to use `ssh` for distributed simulations, otherwise, you can skip this option by pressing **Enter** to retain the default option.
-

Important

Note the following:

- The Microsoft MPI does not support dynamic process spawning, therefore this feature is not available with the MSMPI option on the Windows platform.
- This feature is not supported for the Intel MPI option on both Linux and Windows platforms.
- While running under the Fluent-supported load managers (e.g., SGE/LSF/PBS Pro), the total number of required parallel node processes should be requested at the start of Fluent session, and Fluent will initially start with the specified number of processes for meshing mode and

will automatically spawn the remaining node processes while switching to solution mode. If instead, you want to specify the number of solver processes when switching from meshing to solution mode, then you should omit the `-t<n>` option from the Fluent command line at startup. You can then use the `spawn-solver-process` text command before switching to solution mode.

3.3. Exiting the Program

You can exit ANSYS Fluent by selecting **Exit** in the **File** pull-down menu. If the present state of the program has not been written to a file, a **Question** dialog box will appear, to confirm if you want to proceed. You can cancel the exit and save the appropriate file or continue to exit without saving the file.

Chapter 4: Graphical User Interface

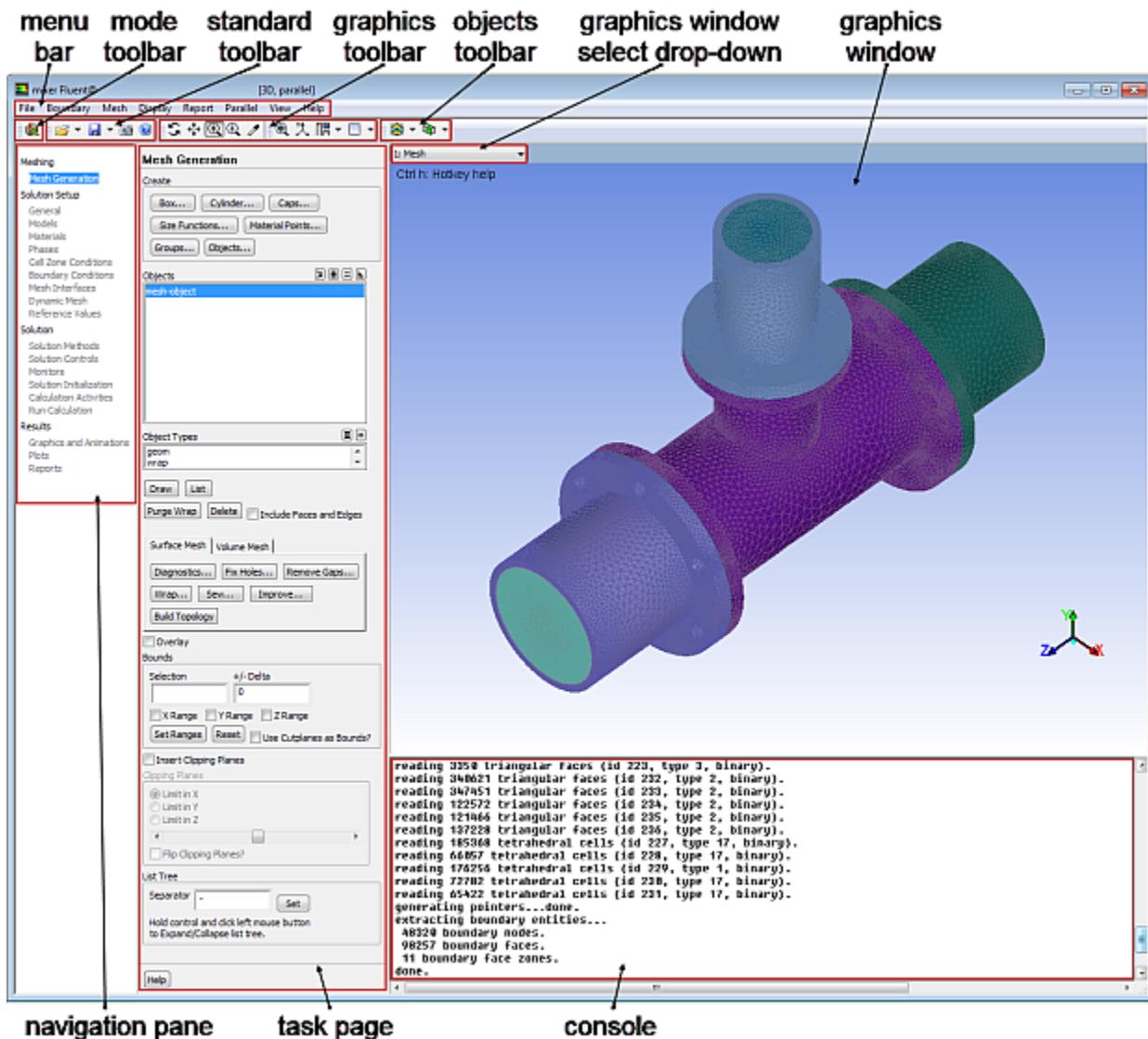
The graphical user interface (GUI) consists of a menu bar to access menus, a toolbar, a task page, a graphics toolbar, graphics window(s), and the console, which is a textual command line interface (described in [Text User Interface \(p. 47\)](#)). You will have access to the dialog boxes via the task page or the menus.

- [4.1. GUI Components](#)
- [4.2. Customizing the GUI \(Linux Systems\)](#)
- [4.3. Using the GUI Help System](#)

4.1. GUI Components

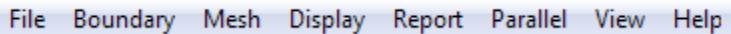
The graphical user interface (GUI) is made up of the following main components: the menu bar, toolbars, a navigation pane, task pages, a console, dialog boxes, and graphics windows. When you use the GUI, you will be interacting with one of these components at all times. [Figure 4.1: The GUI Components \(p. 20\)](#) is a sample screen shot showing all of the GUI components. The GUI components are described in detail in the subsequent sections.

On Linux systems, the attributes of the GUI (including colors and text fonts) can be customized to better match your platform environment. This is described in [Customizing the GUI \(Linux Systems\) \(p. 40\)](#).

Figure 4.1: The GUI Components

4.1.1. Menu Bar

The menu bar organizes the GUI menu hierarchy using a set of pull-down menus. A pull-down menu contain items that perform commonly executed actions. [Figure 4.2: The Meshing Mode Menu Bar\(p. 20\)](#) shows the menu bar for the meshing mode in ANSYS Fluent (for details on the solution mode menu bar, see the [Fluent User's Guide](#)). Menu items are arranged in a logical order to correspond to the sequence of actions that you perform in ANSYS Fluent, (i.e., from left to right and from top to bottom).

Figure 4.2: The Meshing Mode Menu Bar

To select an item from the pull-down menu, follow the procedure outlined:

1. Move the pointer to the name of the pull-down menu.
2. Click the left mouse button to display the pull-down menu.
3. Move the pointer to the item you wish to select and click it.

In addition to using the mouse, you can also select a pull-down menu item using the keyboard. If you press the **Alt** key, each pull-down menu label or menu item will display one underlined character, known as the mnemonic. If you then press the mnemonic character of a pull-down menu, the associated menu will be displayed (note that the mnemonic character is not case sensitive). After the pull-down menu is selected and displayed, you can type a mnemonic character associated with an item to select that item. For example, to display the **Help** menu and select the **Using Help...** option, press **Alt**, then **h**, and then **h** again. If at any time you wish to cancel a menu selection while a pull-down menu is displayed, you can press the **Esc** key.

4.1.2. Toolbars

Figure 4.3: The Full Set of Toolbars



The GUI includes toolbars located within the application window. These toolbars provide shortcuts to performing common tasks. By default, the toolbars are docked to the interface but can also be detached and moved to a new location. You can detach a toolbar by clicking the left mouse button on the outer portion, holding down the mouse, and dragging the toolbar to a new location. To move the detached toolbar, select the title bar and drag the toolbar to a new position in the application window. Once detached, the toolbars can be restored to their location in the interface by double-clicking the title region of the toolbar.

Important

Toolbars that are detached or moved to a new location will return to their original positions each time ANSYS Fluent is launched.

The small arrow button in some toolbars can be used to access additional functionality. For instance, there are additional selections available when you click the small arrow in the standard toolbar.

The graphical user interface includes the Mode toolbar, the Standard toolbar, the Graphics toolbars, and the Objects toolbar.

4.1.2.1. The Mode Toolbar

The **Mode** toolbar (Figure 4.4: The Mode Toolbar (p. 21)) contains the **Switch to Solution** button, which allows you to switch from meshing mode to solution mode in ANSYS Fluent.

Figure 4.4: The Mode Toolbar



Switch to Solution allows you to transfer all mesh data from meshing mode to solution mode in ANSYS Fluent, when you are satisfied with the generated mesh. When you click the **Switch to Solution** button, you will be asked to confirm that you want to switch to solution mode.

It is recommended that you check that the mesh quality is adequate before transferring the mesh data to solution mode. See [Checking the Mesh \(p. 370\)](#) and [Checking the Mesh Quality \(p. 371\)](#) for details.

Note

- Only volume meshes can be transferred to solution mode, surface meshes cannot be transferred.
- Hanging-node meshes will be converted to polyhedra during mesh transfer.

Important

You cannot switch back from solution mode to meshing mode after the mesh data has been transferred. The mode toolbar is deactivated in solution mode.

When no file has been read in solution mode, you can use the command `switch-to-meshing-mode` to switch to meshing mode to generate a mesh to be transferred, if you desire.

The command `switch-to-solution-mode` corresponds to the **Switch to Solution** option; it allows you to transfer the mesh data from meshing mode to solution mode in ANSYS Fluent. When you use the `switch-to-solution-mode` command, you will be asked to confirm that you want to switch to solution mode.

Note

The meshing and solution modes in ANSYS Fluent have different options available for some user configuration settings. Thus, these configuration settings may be changed when switching from meshing to solution mode and may not be the same when returning to one mode after using the other.

4.1.2.2. The Standard Toolbar

The standard toolbar ([Figure 4.5: The Standard Toolbar \(p. 22\)](#)) contains options for working with mesh files, saving images, and accessing the documentation.

Figure 4.5: The Standard Toolbar



The following is a brief description of each of the standard toolbar options.

- **Read a file** allows you to read in a mesh file, or open other file types using a file selection dialog box. Here, you can browse through your collection of folders, and locate a file. For more information, see [Reading and Writing Files \(p. 57\)](#).
- **Write a file** saves the current mesh, or other file types. For more information, see [Reading and Writing Files \(p. 57\)](#).

- **Save Picture**  allows you to capture an image of the active graphics window. For more information, see [Saving Picture Files \(p. 78\)](#).
- **Help**  allows you to access the User's Guide for help topics. For more information, see [Using the GUI Help System \(p. 41\)](#).

4.1.2.3. The Graphics Toolbars

The graphics toolbars (Figure 4.6: The Graphics Toolbars (p. 23)s) contain options that allow you to modify the way in which you view your model or select objects in the graphics window.

Figure 4.6: The Graphics Toolbars

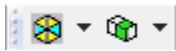


The following is a description of each of the graphics toolbar options.

- **Rotate View**  lets you rotate your model about a central point in the graphics window.
- **Pan**  allows you to pan horizontally or vertically across the view using the left mouse button.
- **Zoom In/Out**  allows you to zoom into and out of the model by holding the left mouse button down and moving the mouse down or up. You can also roll the view by holding the left mouse button down and moving the mouse left or right.
- **Zoom to Area**  allows you to focus on any part of your model. After selecting this option, position the mouse pointer at a corner of the area to be magnified, hold down the left mouse button and drag open a box to the desired size, and then release the mouse button. The enclosed area will then fill the graphics window. Note that you must drag the mouse to the right in order to zoom in. To zoom out, you must drag the mouse to the left.
- **Print information about selected item**  allows you to select items from the graphics windows and request information about displayed scenes. This behaves as a mouse probe button.
- **Fit to Window**  adjusts the overall size of your model to take maximum advantage of the graphics window's width and height.
- **Set view**  contains a drop-down of views, allowing you to display the model in isometric or one of six orthographic views.
- **Arrange the workspace**  provides you with several application window layout options. For example, you can choose to hide certain windows, or view multiple graphics windows. This is essentially the shortcut to the **View** menu.

4.1.2.4. The Objects Toolbar

Figure 4.7: The Objects Toolbar



The Objects toolbar contains options for the display of Faces and Objects in the graphics window.

- **Faces Options**

contains options for displaying face zones in the graphics window.

- **All**

draws all faces in the face zone(s) selected, colored by their zone type.

- **Free**

draws free faces (faces with no neighboring face on at least one edge) on the face zone(s).

- **Multi**

draws multiply-connected faces on the face zone(s) selected, along with their nodes. A multiply-connected face is a boundary face that shares an edge with more than one other face, while a multiply-connected node is a node that is on a multiply-connected edge (i.e., an edge that is shared by more than two boundary faces).

- **Edges**

draws the face edges in the face zone(s) selected.

- **Objects Options**

contains options for displaying objects in the graphics window.

- **Draw Face Zones**

displays the face zone(s) comprising the object(s) drawn in the graphics window.

- **Draw Edge Zones**

displays the edge zone(s) comprising the object(s) drawn in the graphics window.

4.1.3. The Navigation Pane

The navigation pane, located on the left side of the GUI, contains a list of task pages, as shown in [Figure 4.8: The Navigation Pane \(p. 25\)](#).

Figure 4.8: The Navigation Pane



The list consists of Meshing activities, Problem Setup task pages, Solution-related activities, and a Results section for postprocessing.

4.1.4. Task Pages

Task pages appear on the right side of the navigation pane when an item is highlighted in the navigation pane (see [Figure 4.1: The GUI Components \(p. 20\)](#)). The expected workflow is that you travel down the navigation pane, setting the controls provided in each task page until you are ready to run the calculation.

Each task page has a **Help** button. Clicking this button opens the related help topic in the Reference Guide. See [Using the GUI Help System \(p. 41\)](#) for more information.

4.1.5. The Console

The console is located below the graphics window, as shown in [Figure 4.1: The GUI Components \(p. 20\)](#). It is used to display various kinds of information (i.e., messages relating to meshing or solution procedures, etc.). ANSYS Fluent saves a certain amount of information that is written to the console into memory. You can review this information at any time by using the scroll bar on the right side of the console. The size of the console can be adjusted by raising or lowering the bottom frame of the graphics window.

The console behaves in a similar manner as the "xterm" or other Linux command shell tools, or to the MS-DOS Command Prompt window (on Windows systems). It allows you to interact with the TUI menu. For more information on the TUI, see [Text User Interface \(p. 47\)](#).

The console accepts a "break" command (pressing the **Ctrl** key and the **c** key at the same time) to let you interrupt the program while it is working. It also lets you perform text copy and paste operations

between the console and other X Window (or Windows) applications that support copy and paste. The following steps show you how to perform a copy and paste operation on a Windows system:

1. Move the pointer to the beginning of the text to be copied.
2. Press and hold down the left mouse button.
3. Move the pointer to the end of the text (text should be highlighted).
4. Release the left mouse button.
5. Press the **Ctrl** and <Insert> keys at the same time.
6. Move the pointer to the target window and click the left mouse button.
7. Press the **Ctrl** and v keys at the same time.

On a Linux system, follow the steps below to copy text to the clipboard:

1. Move the pointer to the beginning of the text to be copied.
2. Press and hold down the left mouse button.
3. Move the pointer to the end of the text (text should be highlighted).
4. Release the left mouse button.
5. Move the pointer to the target window.
6. Press the middle mouse button to “paste” the text.

4.1.6. Dialog Boxes

There are two types of dialog boxes in ANSYS Fluent. Some dialog boxes are used to perform simple input/output tasks, such as issuing warning and error messages, or asking a question requiring a yes or no answer. Other forms of dialog boxes allow you to perform more complicated input tasks.

A dialog box is a separate, temporary window that appears when ANSYS Fluent needs to communicate with you, or when various types of input controls are required to be set up. The types of controls you will see are described further in this section.

After entering the data using the controls of a dialog box, you can either apply the changes you have made, or abort the changes. Each dialog box falls into one of two behavioral categories, depending on how it was designed.

- The first category of dialog boxes is used in situations where you have to apply the changes and immediately close the dialog box. This type of dialog box includes two push button controls:

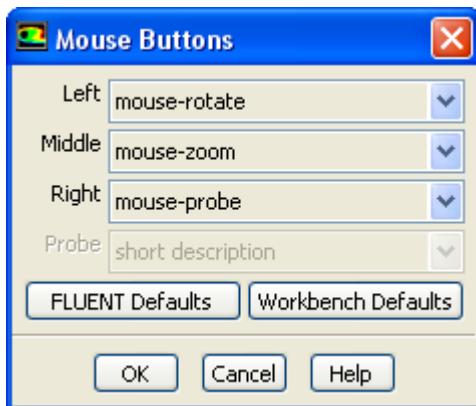
OK

applies any changes you have made to the dialog box and closes it.

Cancel

ignores the changes you have made and closes the dialog box.

An example of this type of dialog box is the **Mouse Buttons** dialog box:



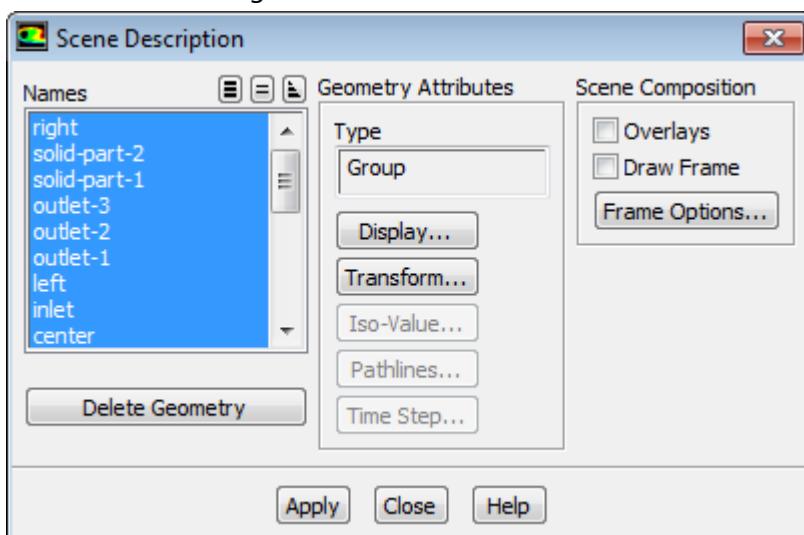
- The other category of dialog boxes is used in situations where you have to keep the dialog box displayed on the screen after changes have been applied. This makes it easy to quickly go back and make more changes. Dialog boxes used for mesh generation and display often fall into this category. This type of dialog box includes two push button controls:

Apply

applies any changes you have made, but does not close the dialog box. The name of this button is often changed to something more descriptive (for example, **Create** for the meshing dialog boxes).

Close

closes the dialog box.



All dialog boxes include the **Help** button used to access the ANSYS help.

Help

displays information about the controls in the dialog box. The help information will appear in the ANSYS Help Viewer.

4.1.6.1. Controls of a Dialog Box

Each type of input control utilized by the dialog boxes is described below. Note that the examples shown here are for a Windows system; if you are working on a Linux system, your dialog box controls may look slightly different, but they will work exactly as described here.

Tabs



Tabs in the dialog box are used to mark the different sections into which the dialog box is divided. The dialog box may be divided into different sections to reduce the amount of screen space it occupies. You can access each section of the dialog box by clicking the left mouse button on the corresponding tab.

Buttons



A button is used to perform a function indicated by the button label. Place the pointer over the button and click the left mouse button to perform the action associated with it.

Check Boxes



A check box, also referred to as check button is used to enable or disable an item or action indicated by the check box label. Click the left mouse button on the check box to toggle the state.

Radio Buttons



Radio buttons are a set of check buttons that allow only one to be set in the **on** position at a time. When you click the left mouse button on a radio button, it will turn on, and all others will be turned off. Radio buttons appear either as diamonds (in Linux systems) or as circles (as shown above).

Text Entry Boxes

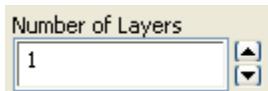


A text entry box lets you type the text input. It will often have a label associated with it to indicate what the entry is for.

Note

Zone/object names cannot begin with a number character.

Integer Number Entry Boxes

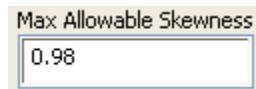


An integer number entry is similar to a text entry except it allows only integer numbers to be entered (e.g., 10, -10, 50000, and 5e4). You may find it easier to enter large integer numbers using scientific notation. For example, you could enter 350000 or 3.5E5.

The integer number entry also has arrow buttons that allow you to increase or decrease its value. For most integer number entry controls, the value will be increased (or decreased) by one when you click an arrow button. You can increase the size of the increment by holding down a keyboard key while clicking the arrow button. The keys used are:

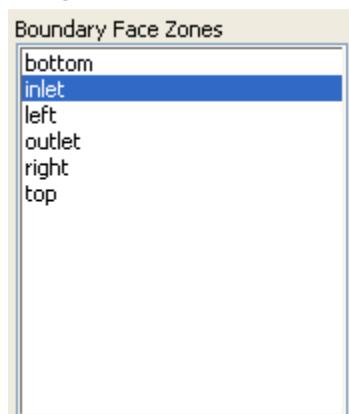
Key	Factor of Increase
Shift	10
Ctrl	100

Real Number Entry Boxes



A real number entry is similar to a text entry except it allows only real numbers to be entered (e.g., 10, -10.538, 50000.45, and 5.e-4). In most cases, the label will show the units associated with the real number entry.

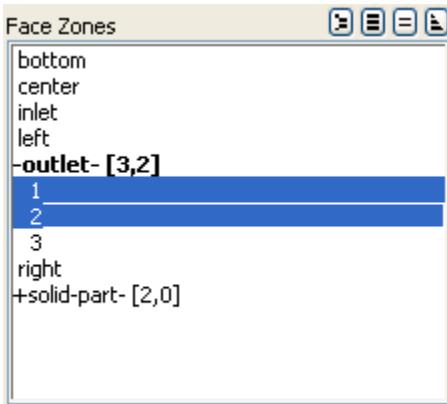
Single-Selection Lists



A single-selection list presents a list of items, with each item printed on a separate line. You can select an item by placing the pointer over the item line and clicking with the left mouse button. The selected item will be highlighted. Selecting another item will deselect the previously selected item in the list.

Some dialog boxes will also accept a double-click in order to invoke the dialog box action that is associated with the list selection.

Multiple-Selection Lists



A multiple-selection list is similar to a single-selection list, but allows you to select more than one item at a time. When you click the left mouse button on an item, its selection state will toggle. Clicking on an unselected item will select it. Clicking on a selected item will deselect it.

To select a range of items in a multiple-selection list, you can select the first desired item, and then select the last desired item while holding down the **Shift** key. The first and last items, and all the items between them, will be selected. You can also click and drag the left mouse button to select multiple items.

There are small push buttons in the upper right corner of the multiple selection list, which accelerate the task of selecting or deselecting items from the selection list.

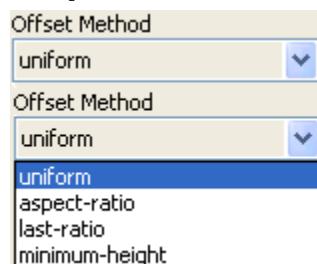
- Click to toggle the tree view for items listed in the selection list. This allows you to view list items with a common prefix in a tree view. The plus sign (+) before an item indicates the list can be expanded to view all items with the common prefix. The separator used to determine the common prefix can be specified in the **List Tree** group box in the [Mesh Generation Task Page \(p. 411\)](#).

Use **Ctrl** + left mouse button to expand the list to show items with the same prefix. The numbers listed next to the prefix indicate the number of list items having the common prefix and the number of such items selected, respectively. For example, **+outlet-[3, 2]** indicates that the selection list comprises 3 zones prefixed with **outlet-** of which 2 are currently selected.

This button is only available for face and edge zone lists, and object lists. Node and cell zone lists do not have this option.

- Click to select all the items in the selection list.
- Click to deselect all the items in the selection list.
- Click to invert the selection of items in the selection list.

Drop-Down Lists



A drop-down list is a hidden single-selection list that shows only the current selection to save space.

To change the selection, do the following:

1. Click the arrow button to display the list.
2. Place the pointer over the new list item.
3. Click the left mouse button on the item to make the selection and close the list.

To abort the selection operation while the list is displayed, move the pointer anywhere outside the list and click the left mouse button.

Scales



A scale is used to select a value from a predefined range by moving a slider. The number shows the current value. You can change the value by clicking the arrow buttons, by following one of the procedures:

1. Place the pointer over the slider.
2. Press and hold down the left mouse button.
3. Move the pointer along the slider bar to change the value.
4. Release the left mouse button.

OR

1. Place the pointer over the slider and click the left mouse button.
2. Using the arrow keys on the keyboard, move the slider bar left or right to change the value.

4.1.6.2. Special Dialog Boxes

The following special dialog boxes are available:

Information Dialog Box



The **Information** dialog box is used to report some information to you. After reading the information, click **OK** to close the dialog box.

Warning Dialog Box



The **Warning** dialog box is used to warn you of a potential problem or deliver an important message. You can proceed only when you acknowledge the warning by clicking the **OK** button.

Error Dialog Box



The **Error** dialog box is used to alert you of an error that has occurred. After reading the error information, click **OK** to close the dialog box.

Working Dialog Box



The **Working** dialog box is displayed when the application is busy performing a task. This is a special dialog box, because it requires no action by you. It is there to let you know that you must wait. When the application is finished, it will close the dialog box automatically. You can, however, abort the task that is being performed by clicking the **Cancel** button.

Question Dialog Box

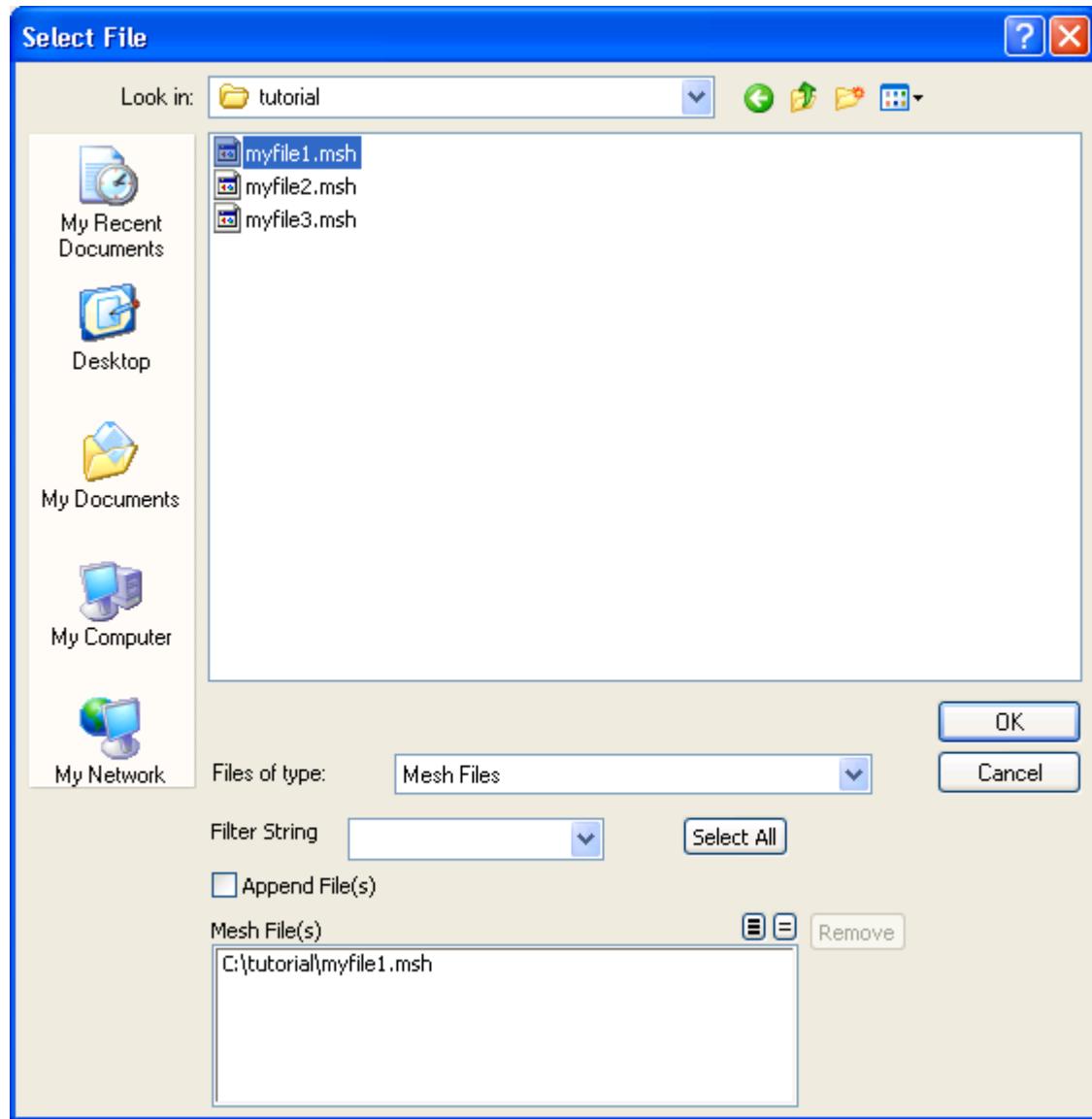


The **Question** dialog box is used to ask you a question. Sometimes the question will require a **Yes** or **No** answer, while other times it will require that you either allow an action to proceed (**OK**) or **Cancel** the action. You can click the appropriate button to answer the question.

Select File Dialog Box

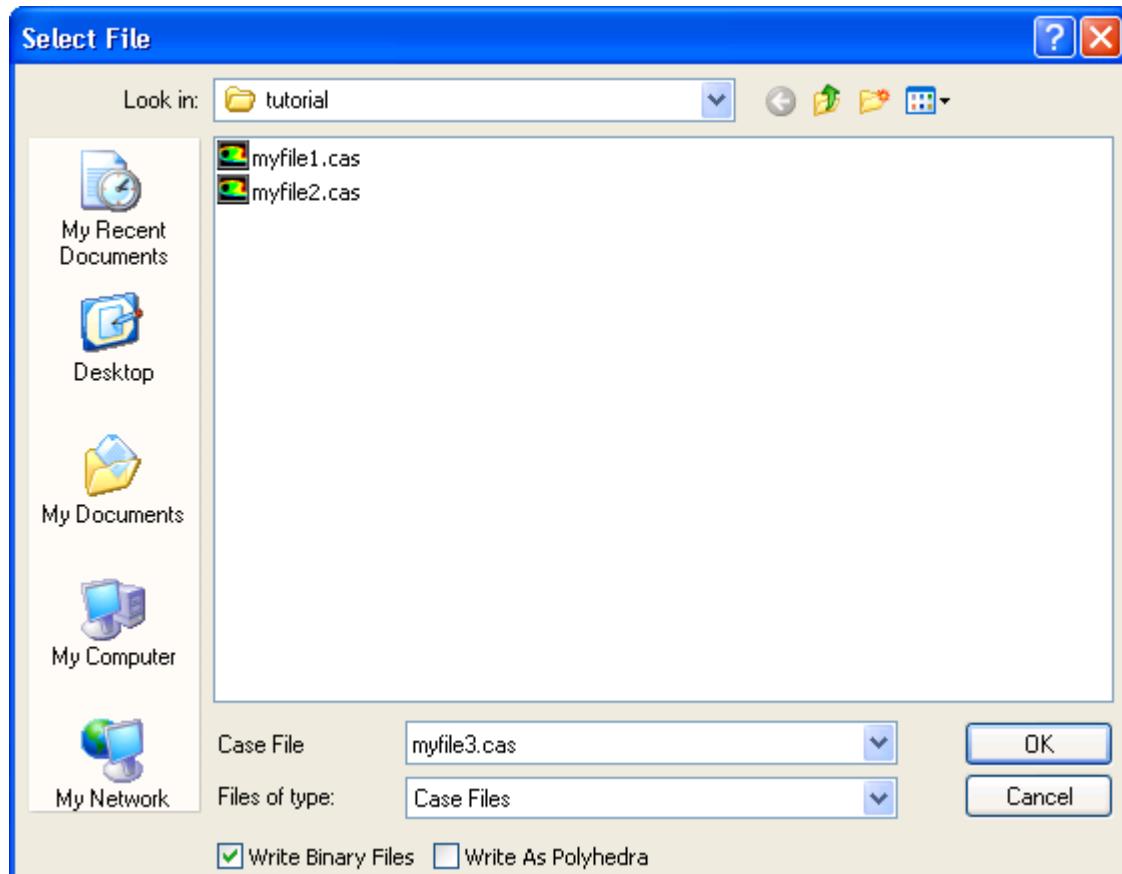
The **Select File** dialog box allows you to select a file (or multiple files) for reading or writing. You can use it to look at your system directories and to select a file. The appearance of the **Select File** dialog box will not always be the same.

- When you select the **File/Read/Mesh...**, **File/Read/Case...**, or **File/Read/Boundary Mesh...** menu item to read a mesh or case file, the **Select File** dialog box will look as shown in [Figure 4.9: The Select File Dialog Box—Read Form \(p. 34\)](#).

Figure 4.9: The Select File Dialog Box—Read Form

- If you are writing a case file, the dialog box will look as shown in [Figure 4.10: The Select File Dialog Box—Write Form \(p. 35\)](#).

Figure 4.10: The Select File Dialog Box—Write Form



- If you are reading or writing any other type of file, the dialog box will be similar to that in [Figure 4.10: The Select File Dialog Box—Write Form \(p. 35\)](#) except that the **Write Binary Files** and **Write As Polyhedra** buttons will not appear.

To select files on Linux systems, do the following:

1. Go to the appropriate directory. You can do this in two different ways:

- Enter the path to the desired directory in the **Filter** text entry box and press the <Enter> key or click the **Filter** button.

Note

Include the final "/" character in the pathname, before the optional search pattern.

- Double-click a directory, and then a subdirectory, etc. in the **Directories** list until you reach the directory you want. You can also click once on a directory and then click **Filter** instead of double-clicking.

Note

In the **Directories** list, the dot "." represents the current directory and the double dots ".." represents the parent directory.

2. Specify the file name by selecting it from the **Files** list or entering the file name in the **File** text entry box (if available) at the bottom of the dialog box. The name of this text entry box will change depending on the type of file you are selecting (**Mesh File**, **Case File**, **Journal File**, etc.).

If you are searching for an existing file with a non-standard extension, you may have to modify the search pattern at the end of the path in the **Filter** text entry box.

- If you are reading a mesh file, the default extension in the search path will be *.msh, MSH, cas, tgf*, and only those files that have one of these extensions will appear in the **Files** list.
- If you want files with a .grd extension to appear in the **Files** list, change the search pattern to *.grd*.
- If you want all the files in the directory to be listed in the **Files** list, enter * as the search pattern.

3. Reading multiple boundary mesh, volume mesh files, or case files.

- If you are reading multiple boundary mesh or volume mesh files, the selected file will be added to the list of **Mesh File(s)**. You can then select another file, which will also be added to this list.
- If you have put only the required files in the working directory and want to read all of them, click the **Select All** button. All the files available in the directory will be added to the list of **Mesh File(s)**.
- If you accidentally select the wrong file, select it in the **Mesh File(s)** list and click the **Remove** button to remove it from the list of files to be read. Repeat until only the required files are in the **Mesh File(s)** list.
- You can also read multiple boundary mesh or volume mesh files using the **Append File(s)** option. Open the **Select File** dialog box and read the first mesh file in ANSYS Fluent. Open the **Select File** dialog box again, enable the **Append File(s)** check button and read the remaining mesh files one after the other.

Important

The **Append File(s)** option will not be accessible while reading the first mesh file. It will be accessible only after reading the first mesh file.

4. To write a mesh file, enter the file name in the **Mesh File** text entry box and click **OK**.

Warning

The **Write Binary Files** check button is enabled by default. Binary files take up less space and can be read and written more quickly. The binary files are written in double-precision.

You can disable the **Write Binary Files** option to write the file in text format. You can read and edit the text file, but it will require more storage space than the corresponding binary file.

You can also use the following TUI command to toggle the **Write Binary Files** option:

/file/file-format

5. To write a hexcore or CutCell mesh, enable **Write As Polyhedra** check button in the **Select File** dialog box to allow handling of polyhedral cells. Polyhedral cells are created when hex and tet cells are merged with each other. Enabling this option allows the export of these cells instead of conformal meshes.

Important

This option is available only for hexcore and CutCell meshes.

6. Click **OK** to read or write the specified file(s). Shortcuts for this step are as follows:

- If the file appears in the **Files** list *and* you are *not* reading a mesh, double-click on it instead of just selecting it. This will automatically activate the **OK** button.
- If you are reading a mesh, you will always have to click **OK**. Clicking or double-clicking will just add the selected file to the **Mesh File(s)** list.
- If you entered the name of the file in the **File** text entry box, press the **Return** key instead of clicking **OK**.

File selection on Windows systems can be done using the standard Windows **Select File** dialog box. For further instructions, see documentation regarding your Windows system.

4.1.7. Graphics Windows

Graphics windows display the program's graphical output, and may be viewed within the ANSYS Fluent application window or in separate windows. The decision to embed the graphics window or to have floating graphics windows is made when you start ANSYS Fluent using the Fluent Launcher. For information about the Fluent Launcher, refer to Setting General Options in Fluent Launcher. When viewed within the application window, the graphics windows will be placed below the toolbar on the right, as shown in [Figure 4.1: The GUI Components \(p. 20\)](#).

The **Display Options** dialog box can be used to change the attributes of the graphics display or to open another display window. The **Mouse Buttons** dialog box can be used to set the action taken when a mouse button is pressed in the display window.

Important

To cancel a display operation, press **Ctrl-C** while the data is being processed in preparation for graphical display.

You cannot cancel the operation after the program begins to draw in the graphics window.

Tip

You can edit the text in the graphics window's caption block by clicking the left mouse button in the desired location. A cursor will appear, and you can then type new text or delete the text that was originally there. Text in the caption block will not be deleted when you clear annotations.

For **Windows** systems, there are special features for printing the contents of the graphics window directly. These features are not available on **Linux** systems.

4.1.7.1. Printing the Contents of the Graphics Window (Windows Systems Only)

If you are using the Windows version of ANSYS Fluent, the system menu of the graphics window displayed by clicking in the upper-left corner of the graphics window, contains the usual system commands (move, size, and close). Along with the system commands, ANSYS Fluent adds three more commands to the menu for printer and clipboard support.

Copy to Clipboard

places a copy of the current picture into the Windows clipboard. Some attributes of the copied picture can be changed using the **Page Setup** dialog box.

The size of your graphics window affects the size of the text fonts used in the picture. For best results, experiment with the graphics window size and examine the resulting clipboard picture using the Windows clipboard viewer.

Print...

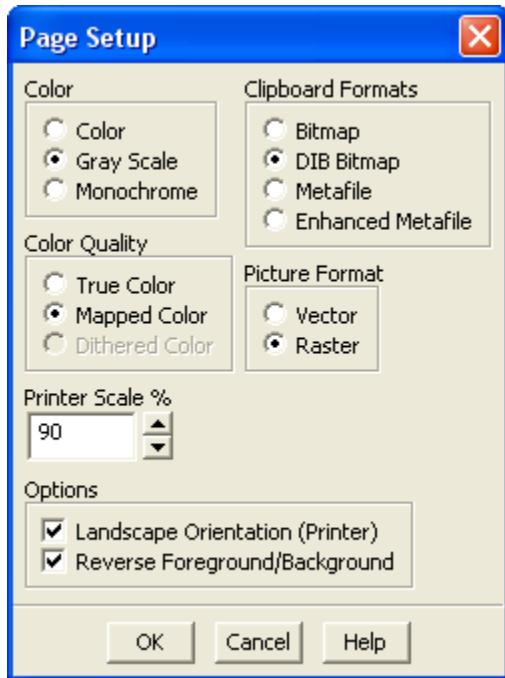
opens the Windows **Print** dialog box, which enables you to send a copy of the picture to a printer. Some attributes of the copied picture can be changed using the **Page Setup** dialog box. More attributes of the final print can be specified within the Microsoft Windows **Print** and **Print Setup** dialog boxes (see documentation for Windows and your printer for details).

Page Setup...

opens the **Page Setup** dialog box, which allows you to change attributes of the picture copied to the clipboard, or to a printer.

4.1.7.2. The Page Setup Dialog Box

To open the **Page Setup** dialog box, select the **Page Setup...** menu item in the system menu of the graphics display window.



Controls

Color

allows you to specify a color or non-color picture.

Color

selects a color picture.

Gray Scale

selects a gray-scale picture.

Monochrome

selects a black-and-white picture.

Color Quality

allows you to specify the color mode used for the picture.

True Color

creates a picture defined by RGB values. This assumes that your printer (or display) has at least 65536 colors, or unlimited colors.

Mapped Color

creates a picture that uses a colormap. This is the right choice for devices that have 256 colors.

Dithered Color

creates a dithered picture that uses 20 colors or less.

Clipboard Formats

allows you to choose the desired format copied to the clipboard. The size of your graphics window can affect the size of the clipboard picture. For best results, experiment with the graphics window size and examine the resulting clipboard picture using the Windows clipboard viewer.

Bitmap

is a bitmap copy of the graphics window.

DIB Bitmap

is a device-independent bitmap copy of the graphics window.

Metafile

is a Windows Metafile.

Enhanced Metafile

is a Windows Enhanced Metafile.

Picture Format

allows you to specify a raster or a vector picture.

Vector

creates a vector picture. This format will have a higher resolution when printed, but some large 3D pictures may take a long time to print.

Raster

creates a raster picture. This format will have a lower resolution when printed, but large 3D pictures may take much less time to print.

Printer Scale %

controls the amount of the page that the printed picture will cover. Decreasing the scaling will effectively increase the margin between the picture and the edge of the paper.

Options

contains options that control other attributes of the picture.

Landscape Orientation (Printer)

specifies the orientation of the picture. If selected, the picture is made in landscape mode. Otherwise, it is made in portrait mode. This option is applicable only when printing.

Reverse Foreground/Background

specifies that the foreground and background colors of the picture will be swapped. This feature allows you to make a copy of the picture with a white background and a black foreground, while the graphics window is displayed with a black background and white foreground respectively.

4.2. Customizing the GUI (Linux Systems)

On Linux systems, you can customize the GUI by changing the attributes such as text color, background color, and text fonts. The program will try to provide default text fonts that are satisfactory for the display size of your platform. In some cases, customization may be necessary if the default text fonts make the GUI too small or too large on your display, or if the default colors are undesirable.

The GUI in ANSYS Fluent is based on the X Window System Toolkit and OSF/Motif. The attributes of the GUI are represented by X Window "resources". If you are unfamiliar with the X Window System Resource Database, refer to any documentation you may have that describes how to use the X Window System or OSF/Motif applications. The default X Window resource values for a medium resolution display are as follows:

```
!
! General resources
!
Fluent*geometry:      +0-0
Fluent*fontList:      *-helvetica-bold-r-normal--12-
Fluent*MenuBar*fontList:  *-helvetica-bold-r-normal--12-
Fluent*XmText*fontList:  *-fixed-medium-r-normal--13-
Fluent*XmTextField*fontList:  *-fixed-medium-r-normal--13-
Fluent*foreground:     black
Fluent*background:     gray75
Fluent*activeForeground: black
Fluent*activeBackground: gray85
Fluent*disabledTextColor: gray55
Fluent*XmToggleButton.selectColor: green
Fluent*XmToggleButtonGadget.selectColor:green
Fluent*XmText.translations:\n    #overrideKeyDelete: delete-previous-character()
Fluent*XmTextField.translations:\n    #overrideKeyDelete: delete-previous-character()
!
! Console resources
!
Fluent*ConsoleText.rows:      24
Fluent*ConsoleText.columns:   80
Fluent*ConsoleText.background: linen
!
! Help Viewer resources
!
Fluent*Hyper.foreground:     black
Fluent*Hyper.background:     linen
Fluent*Hyper.hyperColor:     SlateBlue3
Fluent*Hyper*normalFont:\n    *-new century schoolbook-medium-r-normal--12-
Fluent*Hyper*hyperFont:\n    *
```

```

*-new century schoolbook-bold-r-normal--12-*
Fluent*Hyper*texLargeFont:\ 
*-new century schoolbook-bold-r-normal--14-*
Fluent*Hyper*texBoldFont:\ 
*-new century schoolbook-bold-r-normal--12-*
Fluent*Hyper*texFixedFont:\ 
*-courier-bold-r-normal--12-*
Fluent*Hyper*texItalicFont:\ 
*-new century schoolbook-medium-i-normal--12-*
Fluent*Hyper*texMathFont:\ 
*-symbol-medium-r-normal--14-*
Fluent*Hyper*texSansFont:\ 
*-helvetica-bold-r-normal--12-*

```

To customize one or more of the resources for a particular user, place appropriate resource specification lines in that user's file `$HOME/.Xdefaults` or whatever resource file is loaded by the X Window System on the user's platform.

To customize one or more of the resources for several users at a site, place the resource specification lines in an application defaults resource file called `Fluent`. This file should then be installed in a directory such as `/usr/lib/X11/app-defaults`, or on SUN workstations, the directory may be `/usr/openwin/lib/app-defaults`. See documentation regarding your platform for more information.

4.3. Using the GUI Help System

ANSYS Fluent includes an integrated help system that provides an easy access to the documentation. Using the graphical user interface, you can access the entire User's Guide and other documentation. The User's Guide and other manuals are displayed in the help viewer, which allows you to use the hypertext links and the browser's search and navigation tools to find the information you need.

There are many ways to access the information contained in the online help. You can get reference information from within a task page or dialog box, or (on Linux machines) request context-sensitive help for a particular menu item or dialog box. You can also go to the Meshing User's Guide contents page and use the hypertext links there to find the information you are looking for. In addition to the Meshing User's Guide, you can also access the other documentation (for example, the Tutorial Guide). The chapter [Task Page, Menu, and Dialog Box Reference Guide \(p. 411\)](#) contains a description of each task page, menu item, and dialog box available in the meshing mode in ANSYS Fluent.

The following sections provide information on how to get help for a task page or dialog box, and brief descriptions of the **Help** menu items in ANSYS Fluent. For more information, refer to the information available from the **Help** menu in the viewer itself. See also, [Using Help](#).

- [4.3.1. Task Page and Dialog Box Help](#)
- [4.3.2. Context-Sensitive Help \(Linux Only\)](#)
- [4.3.3. Opening the User's Guide Table of Contents](#)
- [4.3.4. Opening the Reference Guide](#)
- [4.3.5. Accessing Printable Manuals](#)
- [4.3.6. Help for Text Interface Commands](#)
- [4.3.7. Using Help](#)
- [4.3.8. Accessing Online Technical Resources](#)
- [4.3.9. Obtaining a Listing of Other License Users](#)
- [4.3.10. Version and Release Information](#)

4.3.1. Task Page and Dialog Box Help

To get help about the task page or dialog box that you are currently using, click the **Help** button in the task page or dialog box. The help viewer will open to the section of the User's Guide that explains the function of each item in the task page or dialog box. In this section, you will also find hypertext links to more specific section(s) of the User's Guide that discuss how to use the task page or dialog box and provide related information.

4.3.2. Context-Sensitive Help (Linux Only)

If you want to find out how or when a particular menu item or dialog box is used, you can use the context-sensitive help feature. Select the **Context-Sensitive Help** item in the **Help** pull-down menu.

Help → **Context-Sensitive Help**

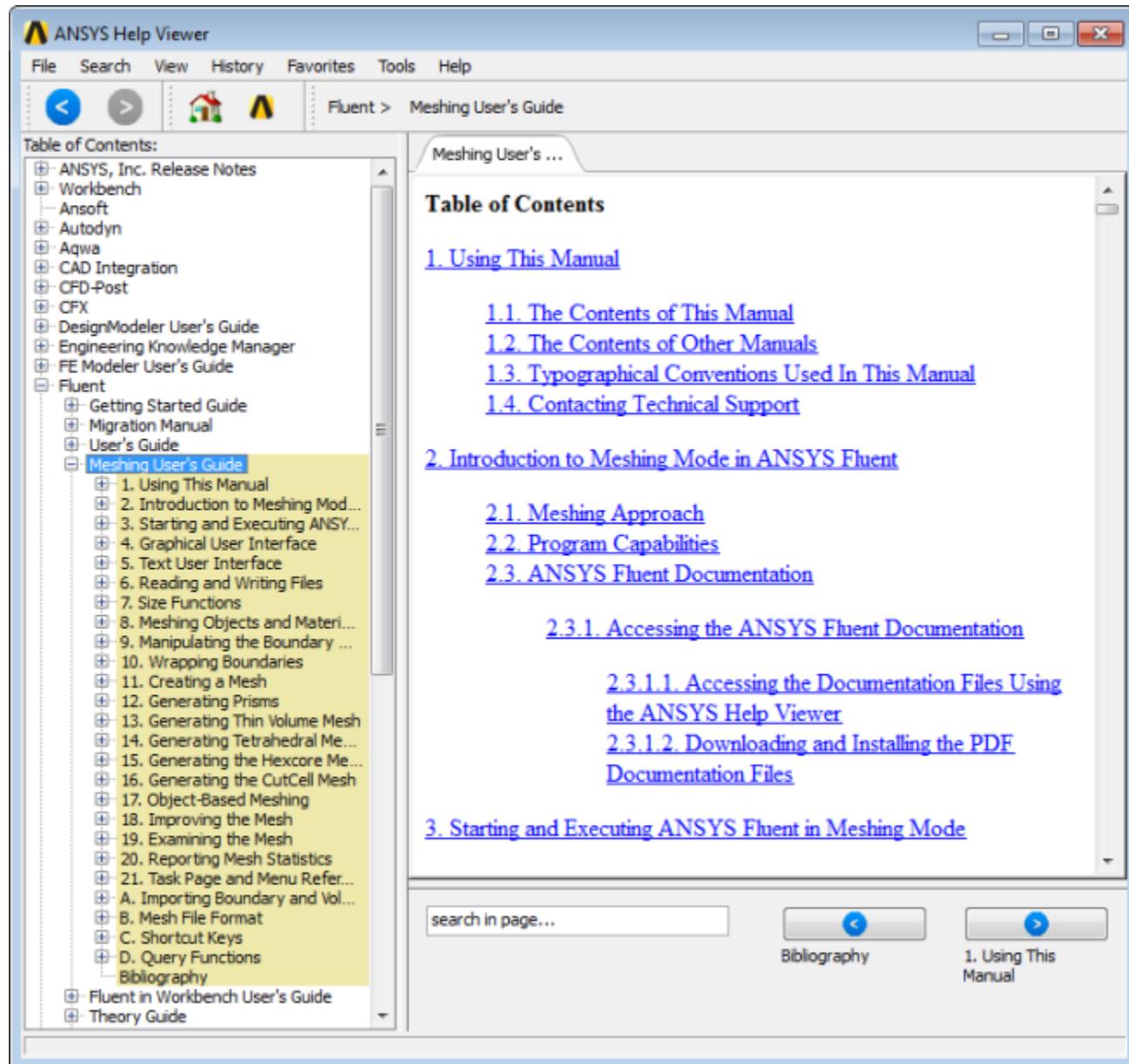
With the resulting question-mark cursor, select an item from a pull-down menu. The help viewer will open to the section of the User's Guide that discusses the selected item.

4.3.3. Opening the User's Guide Table of Contents

To see the User's Guide table of contents, select the **User's Guide Contents...** menu item in the **Help** pull-down menu.

Help → **User's Guide Contents...**

Selecting this item will open the help viewer to the contents page of the User's Guide ([Figure 4.11: The Meshing User's Guide Contents Page \(p. 43\)](#)). Each item in the table of contents is a hypertext link that you can click to view that chapter or section.

Figure 4.11: The Meshing User's Guide Contents Page

In addition, the other documents are listed in the **Table of Contents** of the help viewer. These listings can be expanded by clicking the icon to the left of the document title (⊕). The expanded list will contain hyperlinks to particular chapters or sections, and may also be expandable. For more information on navigation within the help viewer, refer to the information available from the **Help** menu in the viewer itself.

4.3.4. Opening the Reference Guide

The Reference Guide contains information about each task page, dialog box or menu item, arranged by pull-down menu. To open the help system to the first page of the Reference Guide, do the following in the help viewer:

1. Expand the **Meshing User's Guide** contents in the left pane of the viewer by clicking the icon to the left of the document title.

2. Scroll down and select the **Task Page and Menu Reference Guide** item in the left pane of the viewer.
3. Use the hyperlinks in the main viewer window to find the topic of interest or expand the item in the left pane of the viewer and scroll to the topic of interest.

4.3.5. Accessing Printable Manuals

In addition to accessing the manuals through the online help, you can access the ANSYS Fluent Meshing User's Guide and other manuals in printable format (PDF). To see the manuals available in PDF format, select the **PDF** menu item in the **Help** pull-down menu.

Help → PDF

When you select this item, a list of documents will be shown in a submenu. Click the appropriate item to open that document with Acrobat Reader (version 5.0 or higher).

Important

The first time you select a PDF document from the list, you will be directed to download the PDF files and install them in a specific location. Once installed in that location, PDF files can be opened through the **Help → PDF** menu.

If you do not have Acrobat Reader, you can download it for free from Adobe (www.adobe.com).

See the [Getting Started Guide](#) for more information.

4.3.6. Help for Text Interface Commands

To find information about text interface commands, you can either go to the **Meshing Text Command List** in the help viewer, or use the text interface help system described in [Using the Text Interface Help System \(p. 55\)](#).

4.3.7. Using Help

You can obtain information about using the online help by selecting the **Using Help...** menu item in the **Help** pull-down menu.

Help → Using Help...

When you select this item, the help viewer will open to the beginning of this section. See also, [Using Help](#).

4.3.8. Accessing Online Technical Resources

You can access the ANSYS Customer Portal web site by selecting the **Online Technical Resources...** menu item in the **Help** pull-down menu.

Help → Online Technical Resources...

ANSYS Fluent will direct your web browser to the appropriate web address.

4.3.9. Obtaining a Listing of Other License Users

If you are running with an existing ANSYS Fluent license (FluentLM) you can obtain a listing of current ANSYS Fluent users when you select the **License Usage...** menu item in the **Help** pull-down menu.

Help → License Usage...

A list of the current users of the ANSYS Fluent license feature will be displayed in the console.

If your installation of ANSYS Fluent is managed by the ANSYS License Manager (ANSLIC_ADMIN), you will see a message that will indicate that licensing is managed by ANSLIC_ADMIN. For additional information on licensing information, refer to the **Installation and Licensing Documentation** within the help viewer. This information can be found by doing the following in the help viewer:

1. Scroll down to the **Installation and Licensing Documentation** item in the left pane of the viewer.
2. Expand this document by clicking on the icon to the left of the document title.
3. Use the hyperlinks in the main viewer window to find the desired information, or, expand the items in the left pane of the viewer and scroll to the topic of interest.

4.3.10. Version and Release Information

You can obtain information about the version and release of ANSYS Fluent you are running by selecting the **Version...** menu item in the **Help** pull-down menu.

Chapter 5: Text User Interface

In addition to the graphical user interface, the ANSYS Fluent Meshing user interface also consists of a textual command line reference. The text user interface (TUI) is written in a dialect of Lisp called Scheme. Users familiar with Scheme will be able to use the interpretive capabilities of the interface to create customized commands. The TUI is described in the following sections:

- 5.1. Text Menu System
- 5.2. Text Prompt System
- 5.3. Interrupts
- 5.4. System Commands
- 5.5. Text Menu Input from Character Strings
- 5.6. Using the Text Interface Help System

5.1. Text Menu System

The text menu system provides a hierarchical interface to the program's underlying procedural interface.

- You can easily manipulate its operation with standard text-based tools: input can be saved in files, modified with text editors, and read back in to be executed, because it is text based.
- The text menu system is tightly integrated with the Scheme extension language, so it can easily be programmed to provide sophisticated control and customized functionality.

The menu system structure is similar to the directory tree structure of LINUX operating systems. When you first start ANSYS Fluent Meshing, you are in the "root" menu and the menu prompt is simply a caret/greater-than symbol:

>

To generate a listing of the submenus and commands in the current menu, press <Enter>.

```
> <Enter>
beta-feature-access file/    report/
boundary/   material-point/  size-functions/
display/   mesh/      switch-to-solution
exit      objects/
```

Use the command beta-feature-access to enable the features available in beta mode.

```
> beta-feature-access
Enable beta features? [no] yes
Enabling beta features...
It is recommended that you save your files before enabling beta features.
This will assist in reverting to released functionality if needed.
OK to proceed? [cancel] ok
```

To disable beta feature you can use the same command.

```
> beta-feature-access
Enable beta features? [yes] no
```

Disabling beta features...

Warning: Note that disabling beta features may in certain cases leave some currently used beta models active, even though further UI access to this functionality is disabled. Use your pre-saved files to fully revert to released functionality.

OK to proceed? [cancel] **ok**

By convention, submenu names end with a / to differentiate them from menu commands. To execute a command, type its name (or an abbreviation). Similarly, to move down into a submenu, enter its name or an abbreviation. When you move into the submenu, the prompt will change to reflect the current menu name.

```
> display  
/display > set  
/display/set >
```

To move back to the previously occupied menu, type **q** or **quit** at the prompt.

```
/display/set > q  
/display
```

You can move directly to a menu by giving its full pathname.

```
/display > /file  
/display//file >
```

In the above example, control was passed from `/display` to `/file` without stopping in the root menu. Therefore, when you quit from the `/file` menu, control will be passed directly back to `/display`.

```
/display//file > q  
/display >
```

If you execute a command without stopping in any of the menus along the way, control will again be returned to the menu from which you invoked the command.

```
/display /file start-journal jrn1  
/display >
```

The text menu system provides online help for menu commands. The text menu online help system is described in [Using the Text Interface Help System \(p. 55\)](#).

To edit the current command, you can position the cursor with the left and right arrow keys, delete with the `<Backspace>` key, and insert text simply by typing.

For additional information, refer to the following sections:

- [5.1.1. Command Abbreviation](#)
- [5.1.2. Scheme Evaluation](#)
- [5.1.3. Aliases](#)

5.1.1. Command Abbreviation

To select a menu command, you need not type the entire name; you can type an abbreviation that matches the command.

- A command name consists of "phrases" separated by hyphens.
- A command is matched by matching an initial sequence of its phrases.
- Matching of hyphens is optional.
- A phrase is matched by matching an initial sequence of its characters.
- A character is matched by typing that character.

The rules for "matching" a command are:

- If an abbreviation matches more than one command, then the command with the greatest number of matched phrases is chosen.
- If more than one command has the same number of matched phrases, then the first command to appear in the menu is chosen.

For example, each of the following will match the given command `set-ambientcolor: set-ambient-color, s-a-c, sac, and sa.`

- When abbreviating commands, sometimes your abbreviation will match more than one command. In such cases, the first command is selected.
- Occasionally, there is an anomaly such as `lint` not matching `lighting-interpolation` because the `li` gets absorbed in `lights-on?` and then the `nt` doesn't match `interpolation`.

This can be resolved by choosing a different abbreviation, such as `liin`, or `l-int`.

5.1.2. Scheme Evaluation

If you enter an open parenthesis "(" at the menu prompt, then that parenthesis and all characters up to and including the matching closing parenthesis are passed to Scheme to be evaluated. The result of evaluating the expression is then displayed.

```
> (define a 1)
a
> (+ a 2 3 4)
10
```

5.1.3. Aliases

Command aliases can be defined within the menu system. As with the LINUX `csh` shell, aliases take precedence over command execution. The following aliases are predefined in Cortex (`error`, `pwd`, `chdir`, `ls`, `.`, and `alias`):

error

displays the Scheme object that was the "irritant" in the most recent Scheme error interrupt.

pwd

prints the working directory in which all file operations will take place.

chdir

changes the working directory.

ls

lists the files in the working directory.

alias

displays the list of symbols currently aliased.

5.2. Text Prompt System

Commands require various arguments, including numbers, filenames, yes/no responses, character strings, and lists. A uniform interface to this input is provided by the text prompt system. A prompt consists of a prompt string, followed by an optional units string enclosed in parentheses, followed by a default value enclosed in square brackets

```
filled grids? [no] <Enter>  
shrink-factor [0.1] <Enter>  
line-weight [1] <Enter>  
title ["] <Enter>
```

- The default value for a prompt is accepted by pressing <Enter> or typing a comma (,).
-

Important

A comma is not a separator. It is a separate token that indicates a default value. The sequence "1 , 2" results in three values:

- the number 1 for the first prompt.
 - the default value for the second prompt.
 - the number 2 for the third prompt.
-

- A short help message can be displayed at any prompt by entering a ?. (See [Using the Text Interface Help System \(p. 55\)](#).)
- To abort a prompt sequence, use Control-C.

For additional information, refer to the following sections:

- [5.2.1. Numbers](#)
- [5.2.2. Booleans](#)
- [5.2.3. Strings](#)
- [5.2.4. Symbols](#)
- [5.2.5. Filenames](#)
- [5.2.6. Lists](#)
- [5.2.7. Evaluation](#)
- [5.2.8. Default Value Binding](#)

5.2.1. Numbers

The most common prompt type is a number. Numbers can be either integers or real numbers. Valid numbers are, for example, 16, -2 . 4, . 9e5, and +1e-5.

- Integers can also be specified in binary, octal, and hexadecimal form.
- The decimal integer 31 can be entered as 31, #b11111, #o37, or #x1f.
- In Scheme, integers are a subset of reals, so you do not need a decimal point to indicate that a number is real; 2 is just as much a real as 2 . 0.
- If you enter a real number at an integer prompt, any fractional part will be truncated, e.g., 1 . 9 will become 1.

5.2.2. Booleans

Some prompts require a yes-or-no response. A yes/no prompt will accept either `yes` or `y` for a positive response, or `no` or `n` for a negative response. These prompts are used for confirming potentially dangerous actions such as overwriting an existing file, exiting without saving the, mesh, case, etc.

Some prompts require actual Scheme boolean values (true or false). These are entered with the Scheme symbols for true `#t` and false `#f`.

5.2.3. Strings

Character strings are entered in double quotes, as in "`red`". Plot titles and plot legend titles are examples of character strings. Character strings can include any characters, including blank spaces and punctuation.

5.2.4. Symbols

Symbols are entered *without* quotes. Zone names, surface names, and material names are examples of symbols. Symbols must start with an alphabetical character (i.e., a letter), and cannot include any blank spaces or commas.

You can use wild cards to specify zone names when using the TUI. Some examples are:

- `*` will translate as "all zones".

For example,

- `/display/boundary-grid *` allows you to display all the boundary zones in the mesh.
- `/boundary/delete-island-faces wrap*` allows you to delete island faces on all zones prefixed by `wrap`.

- `>` will translate as "all zones visible in the graphics window".

For example, `/boundary/manage/delete >, yes` allows you to delete all visible zones.

- `^` will translate as "all zones selected in the graphics window".

For example, `/boundary/manage/delete ^, yes` allows you to delete all selected zones.

- `[object_name]` will translate as "all zones with the name `object_name`"

For example, `/boundary/manage/delete [box, yes` allows you to delete all zones of an object with the name `box`.

If you use a wild card for an operation that requires a single zone as input, you will be prompted to specify a single zone from the list of those that match the expression specified.

```
> /boundary/manage/name wall* <Enter>  
wall-1 wall-3 wall-5  
wall-2 wall-4 wall-6  
Zone Name [ ]
```

5.2.5. Filenames

Filenames are actually character strings. For convenience, filename prompts do not require the string to be surrounded with double quotes. If, for some exceptional reason, a filename contains an embedded space character, then the name must be surrounded with double quotes.

One consequence of this convenience is that filename prompts do not evaluate the response. For example, the following sequence will end up writing a hardcopy file with the name `fn`, not `valve.ps`.

```
> (define fn "valve.ps")  
fn  
> hc fn
```

Since the filename prompt did not evaluate the response, `fn` did not get a chance to evaluate `"valve.ps"` as it would for most other prompts.

5.2.6. Lists

Some functions in ANSYS Fluent Meshing require a "list" of objects such as numbers, strings, booleans, etc. A list is a Scheme object that is a sequence of objects terminated by the empty list, '`()`'. Lists are prompted for an element at a time, and the end of the list is signaled by entering an empty list. This terminating list forms the tail of the prompted list, and can either be empty or can contain values.

For convenience, the empty list can be entered as `()` as well as the standard form '`'()`'. Normally, list prompts save the previous argument list as the default. To modify the list, overwrite the desired elements and terminate the process with an empty list.

For example the following creates a list of three numbers: 1, 10, and 100.

```
element(1) [()] 1  
element(2) [()] 10  
element(3) [()] 100  
element(4) [()] <Enter>
```

Subsequently,

```
element(1) [1] <Enter>  
element(2) [10] <Enter>  
element(3) [100] <Enter>  
element(4) [()] <Enter>  
element(5) [()] <Enter>
```

adds a fourth element as shown below. Then

```
element(1) [1] <Enter>
element(2) [10] <Enter>
element(3) [100] ()
```

leaves only 1 and 10 in the list. Subsequently entering

```
element(1) [1] ,,(11 12 13)
```

creates a five element list: 1, 10, 11, 12, and 13. Finally, a single empty list removes all elements.

```
element(1) [1] ()
```

5.2.7. Evaluation

All responses to prompts (except filenames), are "evaluated" by the Scheme interpreter before they are used. Therefore you can enter any valid Scheme expression as the response to a prompt. For example, to enter a unit vector with one component equal to 1/3 (without using your calculator), do the following:

```
/foo > set-xy
x-component [1.0] (/ 1 3)

y-component [0.0] (sqrt (/ 8 9))
```

You can also first define a utility function to compute the second component of a unit vector as follows:

```
> (define (unit-y x) (sqrt (- 1.0 (* x x)))) 

unit-y
/foo > set-xy

x-component [1.0] (/ 1 3)

y-component [0.0] (unit-y (/ 1 3))
```

5.2.8. Default Value Binding

The default value at any prompt is bound to the Scheme symbol "_" (underscore) so that the default value can form part of a Scheme expression. For example, if you want to decrease a default value by one-third, you could enter

```
shrink-factor [0.8] (/ _ 3)
```

5.3. Interrupts

The execution of the code can be halted using Control-C, at which time the present operation stops at the next recoverable location.

5.4. System Commands

You can execute system commands with the ! (bang) shell escape character on both LINUX and Windows systems as follows:

[5.4.1. System Commands for LINUX-based Operating Systems](#)

[5.4.2. System Commands for Windows-based Operating Systems](#)

5.4.1. System Commands for LINUX-based Operating Systems

If you are running ANSYS Fluent Meshing on a LINUX operating system, all characters following the ! up to the next newline character will be executed in a subshell. Any further input related to these system

commands must be entered in the window in which you started the program, and any screen output will also appear in that window.

If you start ANSYS Fluent Meshing remotely, this input and output will be in the window in which you start Cortex.

```
> !rm junk.*  
> !vi script.rp
```

The `ls` and `pwd` aliases invoke the LINUX `ls` and `pwd` commands in the working directory. The `cd` alias changes the current working directory of the program. The `!ls` and `!pwd` commands will execute the LINUX commands in the directory in which Cortex was started. The screen output will appear in the window in which you started ANSYS Fluent Meshing, unless you started it remotely, in which case the output will appear in the window in which you started Cortex.

Note

The command `!cd` executes in a subshell, so it will not change the working directory either for ANSYS Fluent Meshing or for Cortex, and it is therefore not useful.

Typing `cd` with no arguments will move you to your home directory in the console. Examples of system commands entered in the console are as follows:

```
> !pwd  
> !ls valve.*
```

Example output (in the window in which ANSYS Fluent Meshing—or Cortex, if you started the program remotely—was started):

```
/home/cfd/run valve  
valve1.cas valve1.msh valve2.cas valve2.msh
```

5.4.2. System Commands for Windows-based Operating Systems

If you are running ANSYS Fluent Meshing on a Windows operating system, all characters following the `!` up to the next newline character will be executed. The results of a command will appear in the ANSYS Fluent Meshing console, or in a separate window if the command starts an external program, such as Notepad.

```
> !del junk.*  
> !notepad script.rp
```

`!cd` and `!dir` will execute the DOS commands and the screen output will appear in the ANSYS Fluent Meshing console. The `!cd` command with no argument will display the current working directory in the ANSYS Fluent Meshing console.

Example input and output (in the ANSYS Fluent Meshing console):

```
> !cd  
p:/cfd/run valve  
> !dir valve*.*/*  
  
Volume in drive P is users  
Volume Serial Number is 1234-5678  
Directory of p:/cfd/run valve  
valve1.cas      valve1.msh      valve2.cas      valve2.msh
```

```
4 File(s)          621,183 bytes
0 Dir(s)          1,830,088,704 bytes free
```

5.5. Text Menu Input from Character Strings

When writing a Scheme extension function for ANSYS Fluent Meshing, it is convenient to be able to include menu commands in the function. This can be done with `ti-menu-load-string`. For example, to open graphics window 2, use:

```
(ti-menu-load-string "di ow 2")
```

A Scheme loop that will open windows 1 and 2 and display the front view of the grid in window 1 and the back view in window 2 is given by

```
(for-each
  (lambda (window view)
    (ti-menu-load-string (format #f "di ow ~a gr view rv ~a"
      window view)))
  '(1 2)
  '(front back))
```

This loop makes use of the `format` function to construct the string used by `menu-load-string`. This simple loop can also be written without using menu commands, but you need to know the Scheme functions that get executed by the menu commands to do it:

```
(for-each
  (lambda (window view)
    (cx-open-window window)
    (display-grid)
    (cx-restore-view view)))
  '(1 2) '(front back))
```

String input can also provide an easy way to create aliases within ANSYS Fluent Meshing. For example, to create an alias that will display the grid, you can type the following:

```
(alias 'dg (lambda () (ti-menu-load-string "/di gr")))
```

Then any time you enter `dg` from anywhere in the menu hierarchy, the grid will be drawn in the active window.

Important

`ti-menu-load-string` evaluates the string argument in the top level menu. It ignores any menu you may be in when you invoke `ti-menu-loadstring`. Therefore, the command

```
(ti-menu-load-string "open-window 2 gr") ; incorrect usage
```

will not work even if you type it from within the `display/` menu. The string itself must cause control to enter the `display/` menu, as in

```
(ti-menu-load-string "display open-window 2 grid")
```

5.6. Using the Text Interface Help System

The text user interface provides context-sensitive online help. Within the text menu system, a brief description of each of the commands can be invoked by entering a `?` followed by the command in question.

Example:

```
> ?dis
display/: Enter the display menu.
```

You can also enter a lone ? to enter "help mode." In this mode, you need only enter the command or menu name to display the help message. To exit help mode type q or quit as for a normal menu.

Example:

```
> ?

[help-mode] > di

display/: Enter the display menu.

[help-mode] > pwd

pwd: #[alias]
(LAMBDA ()
(cx-send '(system "pwd")))

[help-mode] > q
```

Help can also be obtained when you are prompted for information by typing a ? at the prompt.

Example:

```
> display/annotate Annotation text ["] ?

Annotation text ["]
```

Chapter 6: Reading and Writing Files

During an ANSYS Fluent session, you may need to read and write several kinds of files. Files that can be read in the meshing mode include mesh, case, journal, Scheme, domain, and size-field files. Files that can be written in the meshing mode include mesh, case, journal, transcript, and domain files. You can also save pictures of graphics windows. These files and operations are described in the following sections.

- 6.1. Shortcuts for Reading and Writing Files
- 6.2. Mesh Files
- 6.3. Case Files
- 6.4. Size-Field Files
- 6.5. Reading Scheme Source Files
- 6.6. Creating and Reading Journal Files
- 6.7. Creating Transcript Files
- 6.8. Domain Files
- 6.9. Importing Files
- 6.10. Exporting STL Files
- 6.11. Saving Picture Files
- 6.12. The .tgrid File
- 6.13. Exiting the Meshing Mode

6.1. Shortcuts for Reading and Writing Files

The following features make reading and writing files convenient:

- 6.1.1. Default File Suffixes
- 6.1.2. Binary Files
- 6.1.3. Detecting File Format
- 6.1.4. Recent File List
- 6.1.5. Reading and Writing Compressed Files
- 6.1.6. Tilde Expansion (LINUX Systems Only)
- 6.1.7. Disabling the Overwrite Confirmation Prompt
- 6.1.8. Toolbar Buttons

6.1.1. Default File Suffixes

Each type of file read or written in ANSYS Fluent has a default file suffix associated with it. When you specify the first part of the file name (the prefix) for the commonly used files, the appropriate suffix is automatically appended or detected. For example, to write a mesh file named `myfile.msh`, just specify the prefix `myfile` in the **Select File** dialog box and `.msh` is automatically appended. Similarly, to read the mesh file named `myfile.msh`, you can just specify `myfile` and ANSYS Fluent automatically searches for a file of that name with the suffix `.msh`.

The default file suffix for mesh and case files, domain files, scheme files, journal files, etc., are automatically detected and appended. The appropriate default file suffix appears in the **Select File** dialog box for each type of file.

6.1.2. Binary Files

When you write a mesh, case, or size-field file, a binary file is saved by default. Binary files take up less memory than text files and can be read and written more quickly.

Note

You cannot read and edit a binary file, as you can do for a text file.

To save a text file, disable the **Write Binary Files** option in the **Select File** dialog box when you are writing the file. You can also use the /file/file-format command to disable the writing of binary files.

6.1.3. Detecting File Format

When you read a mesh, case, or size-field file, ANSYS Fluent automatically determines whether it is a text (formatted) or binary file.

6.1.4. Recent File List

At the bottom of the **File/Read** submenu there is a list of four ANSYS Fluent mesh files that you most recently read or wrote. To read one of these files into ANSYS Fluent, select it in the list. This allows you to read a recently used file without selecting it in the **Select File** dialog box.

6.1.5. Reading and Writing Compressed Files

ANSYS Fluent allows you to read and write compressed files. Use the **Select File** dialog box to read or write the files that have been compressed using `compress` or `gzip`.

6.1.5.1. Reading Compressed Files

6.1.5.2. Writing Compressed Files

6.1.5.1. Reading Compressed Files

If you select a compressed file with a `.Z` extension, ANSYS Fluent will automatically invoke `zcat` to import the file. If you select a compressed file with a `.gz` extension, ANSYS Fluent will invoke `gunzip` to import the file. For example, if you select a file named `flow.msh.gz`, the following message will be reported, indicating that the result of the `gunzip` is imported into ANSYS Fluent via an operating system pipe.

```
Reading "\ " | gunzip -c \"Z:\flow.msh.gz\""\..."
```

You can also read a compressed file using the text interface by entering the file name. First, ANSYS Fluent attempts to open a file with the input name. If it cannot find a file with that name, it attempts to locate files with default suffixes and extensions appended to the name. For example, if you enter the name `file-name`, it traverses the following list until it finds an existing file:

- `file-name`
- `file-name.gz`
- `file-name.Z`
- `file-name.suffix`

- file-name.suffix.gz
- file-name.suffix.Z

where `suffix` is a common extension to the file, such as `.cas` or `.msh`. ANSYS Fluent reports an error if it fails to find an existing file with one of these names.

Note

For Windows systems, only files that were compressed with `gzip` (i.e., files with a `.gz` extension) can be read. Files that were compressed using `compress` cannot be read into ANSYS Fluent on a Windows machine.

6.1.5.2. Writing Compressed Files

You can use the **Select File** dialog box to write a compressed file by appending a `.Z` or `.gz` extension onto the file name. For example, if you are prompted for a file name and you enter a file name with a `.gz` extension, a compressed file will be written. For example, if you enter `flow.gz` as the name for a mesh file, ANSYS Fluent reports the following message:

```
Writing " | gzip -cfv > Z:\flow.msh.gz"...
```

The status message indicates that the mesh file information is being piped into the `gzip` command, and that the output of the compression command is being redirected to the file with the specified name. In this particular example, the `.msh` extension was added automatically.

Note

For Windows systems, compression can be performed only with `gzip`. That is, you can write a compressed file by appending `.gz` to the name, but appending `.Z` does not compress the file.

6.1.6. Tilde Expansion (LINUX Systems Only)

On LINUX systems, if you specify `~/` as the first two characters of a file name, the `~` is expanded as your home directory. Similarly, you can start a file name with `~username/`, and the `~username` is expanded to the home directory of "username". If you specify `~/file` as the mesh file to be written, ANSYS Fluent saves the file `file.msh` in your home directory. You can specify a subdirectory of your home directory as well: if you enter `~/examples/file.msh`, ANSYS Fluent will save the file `file.msh` in the `examples` subdirectory.

6.1.7. Disabling the Overwrite Confirmation Prompt

By default, if you ask ANSYS Fluent to write a file with the same name as an existing file in that folder, it will ask you to confirm that it is "OK to overwrite" the existing file. If you do not want ANSYS Fluent to ask you for confirmation before it overwrites existing files, you can enter the `file/confirm-overwrite?` text command and answer `no`.

6.1.8. Toolbar Buttons

The standard toolbar provides buttons that make it easier to read and write files:

- The **Read a file** button () allows you to read existing files using a file selection dialog box. The files available for reading include all those available through the **File/Read** menu item, as described in this chapter.
- The **Write a file** button () allows you to write various types of files. The files available for writing include all those available through the **File/Write** menu item, as described in this chapter.

6.2. Mesh Files

Mesh files are created using the mesh generators (ANSYS Meshing, the meshing mode in ANSYS Fluent, GAMBIT, GeoMesh, and PreBFC), or by several third-party CAD packages. From the point of view of ANSYS Fluent, a mesh file is a subset of a case file (described in [Case Files \(p. 63\)](#)). The mesh file includes a list of the node coordinates, connectivity information that tells how the nodes are connected to one another to form faces and cells, and the zone types and numbers of all the faces (e.g., **wall-1, pressure-inlet-5, symmetry-2**). The mesh file does not contain any information on boundary conditions, flow parameters. For information about the format of the CAD package files, see [Appendix A \(p. 647\)](#), and for details on the mesh file format for ANSYS Fluent, see [Appendix B \(p. 653\)](#).

Note

You can also use the **File/Read/Case...** menu item to read a mesh file (described in [Case Files \(p. 63\)](#)) because a mesh file is a subset of a case file.

Important

If the mesh information is contained in two or more separate files generated by one of the CAD packages, you can read them one by one using the **Append File(s)** check button in the **Select File** dialog box. You can also read them together and assemble the complete mesh in the meshing mode.

By default, ANSYS Fluent saves the mesh files with the suffix `.msh`. You need not type the suffix while saving the mesh file, it will be added automatically. For example, if you enter the file name `myfile`, ANSYS Fluent will write to the file `myfile.msh`.

When ANSYS Fluent reads a mesh file, it first searches for a file with the exact name you typed. If a file with that name is not found, it will search for a file with `.msh` appended to the name.

The `/file/read-options` command allows you to set the following options for reading mesh files:

- Enforce mesh topology- This option is disabled by default. Enabling this option will orient the face zones consistently when the mesh file is read. If necessary, the zones being read will be separated, such that each boundary face zone has at most two cell zones as neighbors, one on either side. Also, internal face zones are inserted between neighboring cell zones that are connected by interior faces.
- Check read data- This option allows additional checks for the validity of the mesh. Enabling this option will check the mesh topology during file read. In case incorrect mesh topology is encountered, warning messages will be displayed and the erroneous entities will be deleted. Note that in case of mesh topology errors, no automatic mesh repair is done, and that parts of the mesh may be non-conformal, contain voids, or be erroneous in other ways. The purpose of the `check-read-data` option is to allow access to corrupt

files. This option is disabled by default with the assumption that correct data will be read, and to shorten file read times.

6.2.1. Reading Boundary Mesh Files

To read an ANSYS Fluent boundary mesh (contained in a mesh file created with GAMBIT or in an ANSYS Fluent case file) into ANSYS Fluent, you can do either of the following:

- Select the **File/Read/BoundaryMesh...** menu item to open the **Select File** dialog box and select the boundary mesh file to be read.
- Use the `file/read-boundary-mesh` text command and specify the name of the boundary mesh file to be read.

This option is convenient if you want to reuse the boundary mesh from a file containing a large volume mesh.

6.2.1.1. Reading Multiple Boundary Mesh Files

If the boundary mesh is contained in two or more separate files, you can read them in together and assemble the complete boundary mesh in ANSYS Fluent. Alternatively, you can use the `file/read-multi-bound-mesh` text command for reading multiple boundary mesh files.

6.2.2. Reading Mesh Files

To read a mesh, you can use either of the following:

- Select the **File/Read/Mesh...** menu item to open the **Select File** dialog box and select the mesh file to be read.
- Use the `file/read-mesh` text command and specify the name of the mesh file to be read.

You can also use either of these options to read an ANSYS Fluent mesh file created with GAMBIT, or to read the mesh contained in a case file. To do the latter, you can also use the **File/Read/Case...** menu item or the text command `file/read-case` (see [Reading Case Files \(p. 64\)](#)).

Note

Reading a case file as a mesh file will result in loss of boundary condition data as the mesh file does not contain any information on boundary conditions.

Important

You cannot read meshes from solvers that have been adapted using hanging nodes. To read one of these meshes in the meshing mode in ANSYS Fluent, coarsen the mesh within the solver until you have recovered the original unadapted grid.

6.2.2.1. Reading Multiple Mesh Files

If the mesh is contained in two or more separate files, you can read them together in ANSYS Fluent and assemble the complete mesh. For example, if you are creating a hybrid mesh by reading in a triangular boundary mesh and a volume mesh consisting of hexahedral cells, read both files at the same time

using the **File/Read/Mesh...** menu item or the `file/read-multiple-mesh` or `file/read-multiple-case` text commands.

You can also use the `file/read-meshes-by-tmerge` text command. This command uses the **tmerge** utility.

6.2.2.2. Reading 2D Mesh Files in the 3D Version of ANSYS Fluent

It is also possible to read 2D meshes from ANSYS Fluent into the 3D version of ANSYS Fluent. To read a 2D mesh into the 3D version of ANSYS Fluent, use the **File/Import/Fluent 2D Mesh...** menu item or the `file/import/fluent-2d-mesh` text command.

6.2.3. Reading Faceted Geometry Files from ANSYS Workbench in ANSYS Fluent

You can read faceted geometry files (*.tgf) exported from ANSYS Workbench in ANSYS Fluent. To read the faceted geometry file, use the **File/Read/Mesh...** or the **File/Read/Boundary Mesh...** menu item. Alternatively, you can use the `file/read-mesh` or `file/read-boundary-mesh` text command.

The naming of face zones can be controlled by Named Selections defined in ANSYS Workbench. For details on exporting faceted geometry from ANSYS Workbench, refer to the ANSYS Workbench Help.

6.2.4. Appending Mesh Files

You can also read multiple mesh files one by one instead of reading all of them at a time. This process is called as appending the mesh files. To append files, read in the first mesh file using the **Select File** dialog box. Reopen the dialog box and enable **Append File(s)** and read the remaining files one by one. For details, see [Select File Dialog Box \(p. 33\)](#).

Note

The **Append File(s)** check button is not accessible while reading the first mesh file.

You can also append files using the following commands:

- `file/append-meshes-by-tmerge` allows you to append the mesh files using **tmerge**. There is no GUI item for this TUI command.
- `file/append-mesh` allows you to append the mesh files. This command is same as that of the **Append File(s)** check button in the **Select File** dialog box.

Append Rules:

- If zone names and IDs are duplicated, they will be modified and the changes will be reported in the console.
- Domain information will be retained during the file append operation. If domain names are duplicated, they will be modified and the changes will be reported in the console.
- Refinement region information will be retained during the file append operation. If region names are duplicated, they will be modified and the changes will be reported in the console.
- You can append files comprising only edge zones (without face zones).

- Edge-face zone associations will be retained during the file append operation.
- Zone-specific prism parameter information will be retained during the file append operation.

6.2.5. Writing Mesh Files

To write a mesh file in the format that can be read by ANSYS Fluent, you can do either of the following:

- Select the **File/Write/Mesh...** menu item to open the **Select File** dialog box and specify the name of the mesh file to be written.
- Use the `/file/write-mesh` text command and specify the name of the mesh file to be written.

Note

It is recommended that you delete dead zones in the mesh before writing the mesh or case file for ANSYS Fluent.

Selecting the **File/Write/Mesh...** menu item will invoke the **Select File** dialog box, where you will specify the name of the mesh file to be written. By default, a binary file will be written when you write a mesh file. Binary files take up less memory than text files and can be read and written more quickly.

You can disable the **Write Binary Files** option in the **Select File** dialog to write the file in text format. The text file can be edited, but it will require more storage space than the corresponding binary file. You can also use the TUI command `/file/file-format` to toggle the writing of binary files.

The `/file/write-options` command allows you to set the enforce mesh topology option for writing mesh files. This option is disabled by default. Enabling this option will orient the face zones consistently when the mesh file is written. If necessary, the zones will be separated, such that each boundary face zone has at most two cell zones as neighbors, one on either side. Also, internal face zones will be inserted between neighboring cell zones that are connected by interior faces.

6.2.6. Writing Boundary Mesh Files

ANSYS Fluent allows you to write a mesh file comprising specific boundary zones. This is useful for large cases where you may want to mesh different parts of the mesh separately and then merge them together. This allows you to avoid frequent switching between domains for such cases. You can write out selected boundaries to a mesh file and then create the volume mesh for the part in a separate session. You can then read the saved mesh into the previous session using the **Append File(s)** option and merge the part with the rest of the mesh.

To write a mesh file comprising selected boundaries, do either of the following:

- Use the **File/Write/Boundaries...** menu item to invoke the **Write Boundaries** dialog box and specify the boundaries to be written.
- Use the `/file/write-boundaries` text command and specify the name of the file to be written and the boundaries to be written.

6.3. Case Files

Case files contain the mesh, boundary and cell zone conditions, and solution parameters for a problem. They also contain the information about the user interface and graphics environment. ANSYS Fluent can

read and write either text or binary case files. Binary files require less storage space and are faster to read and write. By default, ANSYS Fluent writes files in binary format. To write a text file, disable the **Write Binary Files** check button in the **Select File** dialog box. In addition, you can read and write either text or binary files in compressed formats (For details, see [Reading and Writing Compressed Files \(p. 58\)](#)). ANSYS Fluent automatically detects the file type when reading.

Important

Changing the ID of a thread in the meshing mode may affect the case set up. In such cases, you will be prompted to confirm that you want to proceed with the ID changing operation.

The commands used for reading case files can also be used to read native-format mesh files (as described in [Mesh Files \(p. 60\)](#)) because the mesh information is a subset of the case information. The commands for reading and writing case files are described in the following sections.

6.3.1. Reading Case Files

6.3.2. Writing Case Files

6.3.1. Reading Case Files

To read a case file, you can use either of the following:

- Select the **File/Read/Case...** menu item to open the **Select File** dialog box and select the case file to be read.
 - Use the `file/read-case` text command and specify the name of the case file to be read.
-

Note

Cell hierarchy in case files adapted in the solution mode will be lost when they are read in the meshing mode.

Case files containing polyhedral cells can also be read in the meshing mode of ANSYS Fluent. You can display the polyhedral mesh, perform certain mesh manipulation operations, check the mesh quality, etc.

6.3.1.1. Reading Multiple Case Files

To read in multiple case files, read both files at the same time using the **File/Read/Case...** menu item and selecting the respective files in the **Case File(s)** list in the **Select File** dialog box. You can also use the `file/read-multiple-case` text command.

6.3.2. Writing Case Files

To write a case file in the format that can be read by ANSYS Fluent, do either of the following:

- Select the **File/Write/Case...** menu item.

- Use the `file/write-case` text command and specify the name of the mesh file to write.

Note

It is recommended that you delete dead zones in the mesh before writing the mesh or case file for ANSYS Fluent.

Selecting the **File/Write/Case...** menu item will invoke the **Select File** dialog box, where you will specify the name of the case file to be written. By default, a binary file will be written when you write the case file. Binary files take up less memory than text files and can be read and written more quickly.

You can disable the **Write Binary Files** option in the **Select File** dialog to write the file in text format. The text file can be edited, but it will require more storage space than the corresponding binary file. You can also use the TUI command `/file/file-format` to toggle the writing of binary files.

If you are writing a hexcore or CutCell mesh, enable the **Write As Polyhedra** check button in the **Select File** dialog box. This allows hex cells which are either part of a hanging-node sub-division or are at the boundary of the hex-tet interface, to be converted to polyhedral cells. Enabling this option allows the export of these cells instead of non-conformal meshes.

Note

Further manipulation of the mesh is restricted after conversion to polyhedra. Only limited operations like displaying the polyhedral mesh, certain mesh manipulation operations, checking the mesh quality are available for polyhedral meshes.

Important

- Case files that have been read and re-written in the meshing mode are incompatible with previously saved data files. Do not read previously saved data files with the case file when such case files are transferred or read in the solution mode.
- If the zone topology changes due to operations performed in the meshing mode, it is recommended that you verify the case setup after transferring or reading the case in the solution mode.

6.4. Size-Field Files

Size-field files contain the size function definitions based on the parameters specified.

6.4.1. Reading Size-Field Files

6.4.2. Writing Size-Field Files

6.4.1. Reading Size-Field Files

Select the **File/Read/Size Field...** menu to read a size-field file. This will invoke the **Select File** dialog box, where you can specify the name of the size-field file to be read.

You can also use the `/file/read-size-field` command and specify the name of the file to be read.

Note

If you read a size-field file after scaling the model, ensure that the size-field file is appropriate for the scaled model (size-field vertices should match the scaled model).

6.4.2. Writing Size-Field Files

Select the **File/Write/Size Field...** menu to write a size-field file. This will invoke the **Select File** dialog box, where you can specify the name of the size-field file to be written.

The size-field file is written in binary format by default. You can use the `/file/file-format` command to disable the writing of binary files.

6.5. Reading Scheme Source Files

A Scheme source file can be loaded in three ways: through the menu system as a scheme file, through the menu system as a journal file, or through Scheme itself.

For large source files use the **Select File** dialog box invoked by selecting the **File/Read/Scheme...** menu item or the Scheme `load` function.

```
> (load "file.scm")
```

Shorter files can also be loaded with the **File/Read/Journal...** menu item or the `file/read-journal` command in the text interface (or its `.` or `source` alias).

```
> . file.scm  
> source file.scm
```

In this case, each character of the file is echoed to the console as it is read in the same way as if you were typing the contents of the file.

6.6. Creating and Reading Journal Files

A journal file contains a sequence of ANSYS Fluent commands, arranged as they would be typed interactively into the program or entered through the GUI or TUI. The GUI commands are recorded as Scheme code lines in journal files. You can also create journal files manually with a text editor. If you want to include comments in your file, be sure to put a semicolon (`;`) at the beginning of each comment line.

The purpose of a journal file is to automate a series of commands instead of entering them repeatedly on the command line. It can also be used to produce a record of the input to a program session for later reference, although transcript files are often more useful for this purpose (see [Creating Transcript Files \(p. 68\)](#)).

Command input is taken from the specified journal file until its end is reached, at which time control is returned to the standard input (usually the keyboard). Each line from the journal file is echoed to the standard output (usually the screen) as it is read and processed.

Important

A journal file by design is a simple record/playback facility. It contains no information about the state in which it was recorded or the state in which it is being played back.

- Be careful not to change the folder while recording a journal file. Also, try to recreate the state in which the journal was written before you read it into the program.

For example, if the journal file includes an instruction for ANSYS Fluent to save a new file with a specified name, check that no file with that name exists in your directory before you read in your journal file. If a file with that name exists and you read in your journal file, it will prompt for a confirmation to overwrite the old file when the program reaches the write instruction.

Since the journal file contains no response to the confirmation request, ANSYS Fluent will not be able to continue following the instructions of the journal file.

- Other conditions that may affect the program's ability to perform the instructions contained in a journal file can be created by modifications or manipulations that you make within the program.

For example, if your journal file displays certain surfaces, you must read in the appropriate mesh file before reading the journal file.

Important

At a given time, only one journal file can be open for recording. But you can read a journal file at any time. You can also write a journal and a transcript file simultaneously.

- Whether you choose to type the text command in full or use partial strings (as described in [Command Abbreviation \(p. 48\)](#)), complete commands are recorded in the journal files.

Important

- Only successfully completed commands are recorded. For example, if you stopped an execution of a command using <Ctrl> + c, it will not be recorded in the journal file.
- If a GUI event happens while a text command is in progress, the GUI event is recorded first.
- All default values are recorded (as in the example above).

Using the GUI

To start the journaling process, select the **File/Write/Start Journal...** menu item. Enter a name for the file in the **Select File** dialog box. The journal recording begins and the **Start Journal...** menu item becomes **Stop Journal** menu item. You can end journal recording by selecting **Stop Journal**, or by exiting the program.

You can read a journal file into the program using the **Select File** dialog box invoked by selecting the **File/Read/Journal...** menu item.

Journal files are always loaded in the main (i.e., top-level) text menu, regardless of where you are in the text menu hierarchy when you invoke the read command.

Using Text Commands

To start the journaling process use the `file/start-journal` command, and end it with the `file/stop-journal` command (or by exiting the program). To read a journal file into the program, use the `file/read-journal` command.

Note

The `read-journal` command always loads the file in the main (i.e., top-level) menu, regardless of where you are in the menu hierarchy when you invoke it.

The standard period (.) alias is the same as the `file/read-journal` definition and is defined by:

```
(alias '.' (lambda () (ti-read-journal)))
```

6.7. Creating Transcript Files

A transcript file contains a complete record of all standard input to and output from ANSYS Fluent (usually all keyboard and GUI input and all screen output).

The GUI commands are recorded as Scheme code lines in transcript files. ANSYS Fluent creates a transcript file by recording everything typed as input or entered through the GUI, and everything printed as output in the text window.

The purpose of a transcript file is to produce a record of the program session for later reference. The transcript file cannot be read back into the program because they contain messages and other output transcript files.

Important

At a time, only one transcript file can be open for recording. But you can write a transcript and a journal file simultaneously. You can also read a journal file while a transcript recording is in progress.

Using the GUI

To start the transcription process, select the **File/Write/Start Transcript...** menu item. Enter a name for the file in the **Select File** dialog box. The transcript recording begins and the **Start Transcript...** menu item becomes the **Stop Transcript** menu item. You can end transcript recording by selecting **Stop Transcript**, or by exiting the program.

Using Text Commands

In the text interface, start the transcription process with the `file/start-transcript` command, and end it with the `file/stop-transcript` command (or by exiting the program).

6.8. Domain Files

Each mesh file written by ANSYS Fluent has a domain section. A domain file is the domain section of the mesh file and is written as a separate file. It contains a list of node, face, and cell zone IDs that comprise each domain in the mesh.

By convention, domain file names are composed of a root with the suffix .dom. If you conform to this convention, you do not have to type the suffix when prompted for a filename; it will be added automatically. When ANSYS Fluent reads a domain file, it first searches for a file with the exact name you typed. If a file with that name is not found, it will search for a file with .dom appended to the name. When ANSYS Fluent writes a domain file, .dom will be added to the name you type unless the name already ends with .dom.

Reading Domain Files

To read the domain files into ANSYS Fluent, do either of the following:

- Select the **File/Read/Domains...** menu item to invoke the **Select File** dialog box and specify the name of the domain file to be read.
- Enter the `file/read-domains` text command and specify the name of the domain file to be read.

If a domain that is being read already exists in the mesh, a warning message is displayed. ANSYS Fluent verifies if the zones defining the domains exist in the mesh. If not, it will display a warning message.

Writing Domain Files

To write domain files in ANSYS Fluent, do either of the following:

- Select the **File/Write/Domains...** menu item to invoke the **Select File** dialog box and specify the name of the domain file to be written.
- Enter the `file/write-domains` text command and specify the name of the domain file to be read.

6.9. Importing Files

You can import the following file formats using the menu items in the **Import** submenu, or using the associated text commands:

- ANSYS Prep7/cdb files
- CGNS files
- FIDAP neutral files
- GAMBIT neutral files
- HYPERMESH ASCII files
- I-deas Universal files
- NASTRAN files
- PATRAN neutral files

For information about the format of these files and details about importing them (if the import commands are not available on your computer), see [Appendix A \(p. 647\)](#). For information about changing the options related to grid import see [Grid Import Filter Options \(p. 78\)](#).

Importing Multiple Files

You can also import multiple files using the **File/Import** menu. Select the file format (e.g., ANSYS prep7/cdb) and the mesh type (surface or volume) to open the **Select File** dialog box. Select the appropriate files from the **Files** selection list and click **OK**. Alternatively, use the appropriate TUI command (e.g., `file/import/ansys-surf-mesh`) and specify the names of the files to be imported.

Appending Multiple External Files

You can also add files of any external format to an existing mesh. This is known as appending files. To append external files, read or import the first file. Use the **File/Import** menu and select the appropriate file format (e.g., **ANSYS prep7/cdb**) and the mesh type (surface or volume). Enable **Append File(s)** in the **Select File** dialog box and import the necessary files.

6.9.1. ANSYS Prep7 Files

- Use the **File/Import/ANSYS prep7/cdb/Surface...** menu item to read a surface Prep7 file. Specify the name of the ANSYS Prep7 file to be read in the **Select File** dialog box.

OR

Use the `file/import/ansys-surf-mesh` command to read a surface Prep7 file.

- Use the **File/Import/ANSYS prep7/cdb/Volume...** menu item to read a volume Prep7 file. Specify the name of the ANSYS Prep7 file to be read in the **Select File** dialog box.

OR

Use the `file/import/ansys-vol-mesh` command to read a volume Prep7 file.

6.9.2. CGNS Files

- Use the **File/Import/CGNS/Surface...** menu item to read a surface mesh. Specify the name of the CGNS file to be read in the **Select File** dialog box.

OR

Use the `file/import/cgns-surf-mesh` command to read a CGNS surface mesh.

- Use the **File/Import/CGNS/Volume...** menu item to read a volume mesh. Specify the name of the CGNS file to be read in the **Select File** dialog box.

OR

Use the `file/import/cgns-vol-mesh` command to read a CGNS volume mesh.

6.9.3. Importing FIDAP Neutral Mesh Files

- Use the **File/Import/FIDAP neutral/Surface...** menu item to read a surface mesh. Specify the name of the surface file to be read in the **Select File** dialog box.

OR

Use the `file/import/fidap-surf-mesh` command to read a surface mesh.

- Use the **File/Import/FIDAP neutral/Volume...** menu item to read a volume mesh. Specify the name of the volume file to be read in the **Select File** dialog box.

OR

Use the `file/import/fidap-vol-mesh` command to read a volume mesh.

6.9.4. Importing GAMBIT Neutral Mesh Files

- Use the **File/Import/GAMBIT neutral/Surface...** menu item to read a surface mesh. Specify the name of the GAMBIT surface file to be read in the **Select File** dialog box.

OR

Use the `file/import/gambit-surf-mesh` command to read the surface mesh.

- Use the **File/Import/GAMBIT neutral/Volume...** menu item to read a volume mesh. Specify the name of the GAMBIT volume file to be read in the **Select File** dialog box.

OR

Use the `file/import/gambit-vol-mesh` command to read the volume mesh.

6.9.5. HYPERMESH ASCII Files

- Use the **File/Import/HYPERMESH Ascii/Surface...** menu item to read a surface mesh. Specify the name of the HYPERMESH file to be read in the **Select File** dialog box.

OR

Use the `file/import/hypermesh-surf-mesh` command to read a surface mesh.

- Use the **File/Import/HYPERMESH Ascii/Volume...** menu item to read a volume mesh. Specify the name of the HYPERMESH file to be read in the **Select File** dialog box.

OR

Use the `file/import/hypermesh-vol-mesh` command to read a volume mesh.

6.9.6. I-deas Universal Files

- Use the **File/Import/IDEAS universal/Surface...** menu item to read a surface I-deas Universal file. Specify the name of the I-deas Universal file to be read in the **Select File** dialog box.

OR

Use the `file/import/ideas-surf-mesh` command to read a surface I-deas Universal file.

- Use the **File/Import/IDEAS universal/Volume...** menu item to read a volume I-deas Universal file. Specify the name of the I-deas Universal file to be read in the **Select File** dialog box.

OR

Use the `file/import/ideas-vol-mesh` command to read a volume I-deas Universal file.

6.9.7. NASTRAN Files

- Use the **File/Import/NASTRAN/Surface...** menu item to read a surface NASTRAN file. Specify the name of the NASTRAN file to be read in the **Select File** dialog box.

OR

Use the `file/import/nastran-surf-mesh` command to read a surface NASTRAN file.

- Select the **File/Import/NASTRAN/Volume...** menu item to read a volume NASTRAN file. Specify the name of the NASTRAN file to be read in the **Select File** dialog box.

OR

Use the `file/import/nastran-vol-mesh` command to read a volume NASTRAN file.

6.9.8. PATRAN Neutral Files

- Use the **File/Import/PATRAN neutral/Surface...** menu item to read a surface PATRAN mesh. Specify the name of the PATRAN file to be read in the **Select File** dialog box.

OR

Use the `file/import/patran-surf-mesh` command to read a surface PATRAN file.

- Use the **File/Import/PATRAN neutral/Volume...** menu item to read a volume PATRAN mesh. Specify the name of the PATRAN file to be read in the **Select File** dialog box.

OR

Use the `file/import/patran-vol-mesh` command to read a volume PATRAN file.

6.9.9. Importing CAD Files

You can import CAD models using the CAD readers or associative geometry interfaces (via plug-ins). Refer to [CAD Integration](#) in the ANSYS Help for detailed CAD-related information.

Information about past, present, and future platform support is viewable via the ANSYS, Inc. Web site (Support > Platform Support), see [ANSYS Platform Support](#).

Use the **File/Import/CAD...** menu item to open the [Import CAD Geometry Dialog Box \(p. 471\)](#), where you can set the basic options for importing CAD files.

The [CAD Options Dialog Box \(p. 473\)](#) contains additional options that can be set for importing CAD files. The following options are available:

- You can choose to import the curvature data from the nodes of the CAD facets.

- You can select the length unit to scale the mesh to on import; models created in other units will be scaled accordingly.

Important

The imported CAD models are scaled based on the length unit selected for the meshing mode session only. When the model is transferred to solution mode, the model units are reverted to the original CAD units. Refer to [Scaling the Mesh](#) in the [Fluent User's Guide](#) for details on scaling the mesh in solution mode.

- You can choose to read all CAD files in the subdirectories of the selected directory.

Note

CAD files with only line bodies/wires (edge zones) cannot be imported.

- You can save an intermediate PMDB (*.pmdb) file in the directory containing the CAD files imported. You can use this file to import the same CAD file(s) again with different options set, for a quicker import than the full import. A PMDB file will be saved per CAD file selected.
- You can import part names and body names from the CAD files. You can also import enclosure and symmetry named selections from ANSYS DesignModeler (*.agdb) files.
- You can continue to import the CAD file(s), despite errors or problems creating the faceting on certain surfaces, or other issues.

Note

Surfaces with failed conformal faceting will be separated into distinct face zones and will have "**failed**" identifier in the face zone name.

- You can specify options for controlling the granularity of objects imported from the CAD files. You can create an object per body, part, or CAD file imported. When an object is created per body, you can optionally choose to have one zone per CAD face imported. The object type and granularity on import is reported in the **CAD Options** dialog box.

Important

A suffix **-sheet** will be added to the object name for surface bodies imported (except for surface bodies imported using the **One Object per part** option).

For single part cases with baffles, separate objects are created for the baffles on import. However, the corresponding feature edges are not included in the baffle object created. You will need to reassociate the edges with the baffle objects, if required.

- The available object type and granularity when the tessellation refinement option is selected are:
 - One geom object per body, comprising one zone per body.

- One geom object per body, comprising one zone per CAD face.
 - One geom object per part, comprising one zone per part.
 - One geom object per file, comprising one zone per file.
- The available object type and granularity when the conformal tessellation option is selected are:
- One geom object per body, comprising one zone per body.
 - One geom object per body, comprising one zone per CAD face.
 - One mesh object per part, comprising one zone per body.
 - One geom object per file, comprising one zone per file.
- You can choose to process **Named Selections** from the CAD file(s), including Named Selections from ANSYS DesignModeler, publications from CATIA, etc.

Note

Named Selections defined in ANSYS Meshing cannot be imported.

Important

- A prefix **zone** will be added to the object name and face zone name if the body, part, file, or Named Selection name begins with a digit or a special character other than “_”.: For example, importing a file .test.agdb with the option **One object per file**, will create an object named **zone.test** comprising the face zone **zone.test**.
 - A prefix **zone** will be added to the face zone name along with the **Zone Name Prefix** specified in the [Import CAD Geometry Dialog Box \(p. 471\)](#) if the specified zone name prefix begins with a digit or the special characters “.+-. For example, if the **Zone Name Prefix** specified is +dsdb and the body name is **crank**, the face zone name **zone+dsdb-crank** will be generated.
- If you select **Tessellation Refinement**, you need to specify the tolerance for the tessellation (faceting) refinement. The default value is 0, which implies no tessellation (faceting) refinement during import.
-

Note

It is recommended that you use the default value of 0 for an initial (diagnostic) import. You can then determine the minimum size you intend to use for the mesh and import the file(s) again using a **Tolerance** value 1/10th the intended minimum size.

You can also specify a maximum facet size for the imported model to avoid very large facets and optionally recreate missing face tessellations during the file import.

- If you select **Conformal Tessellation**, you need to specify the minimum and maximum facet sizes, and the curvature normal angle to be used for refining the conformal faceting based on the underlying curve and surface curvature. You can optionally use the edge proximity size function for creating the

conformal tessellation, based on the number of cells per gap specified. You can also choose to save a size-field file based on these defined parameters.

Alternatively, you can use a previously saved size-field file to create the conformal tessellation by enabling **Use Size Field File**.

Note

- If you select a size-field file during CAD import, ensure that size-field file selected is appropriate for the length units selected.
- Surfaces with failed conformal tessellation will be separated into distinct face zones and will have "**failed**" identifier in the face zone name.

Missing face tessellations are always recreated during the file import when **Conformal Tessellation** is selected.

The `/file/import/cad` command allows you to set the basic options for importing CAD files. The commands in the `/file/import/cad-options` menu allow you to set additional options for importing CAD files.

Supported file formats include:

- ANSYS Workbench formats: `*.agdb`, `*.meshdat`, `*.mechdat`
- ANSYS legacy formats: `*.cmdb`, `*.dsdb`, ICEM CFD (`*.tin`), GAMBIT (`*.dbs`)
- Standard/Free CAD formats: IGES (`*.igs`, `*.iges`), STEP (`*.stp`, `*.step`), ACIS (`*.sat`, `*.sab`), Parasolid (`*.x_t`, `*.xmt_txt`, `*.x_b`, `*.xmt_bin`)
- Licensed Readers: Autodesk Inventor (`*.ipt`, `*.iam`), CATIA V4 (`*.model`, `*.exp`, `*.session`, `*.dlv`), CATIA V5 (`*.CATPart`, `*.CATProduct`), CATIA V6 (`*.3dxml`), Creo Parametric (`*.prt`, `*.asm`), JTOpen (`*.jt`), NX (`*.prt`), SolidWorks (`*.sldprt`, `*.sldasm`)
- Plug-ins: AutoCAD (`*.dwg`, `*.dxr`), Autodesk Inventor (`*.ipt`, `*.iam`), CATIA V5 (`*.CATPart`, `*.CATProduct`), Creo Parametric (`*.prt`, `*.asm`), Creo Elements/Direct Modeling (`*.pkg`, `*.ndl`, `*.ses`, `*.sda`, `*.sdp`, `*.sdac`, `*.sdpc`), NX (`*.prt`), SolidWorks (`*.sldprt`, `*.sldasm`), SpaceClaim (`*.scdoc`), Solid Edge (`*.par`, `*.asm`, `*.psm`, `*.pwd`)
- Non-CAD formats: STL (`*.stl`)
- Native format: PMDB (Part Manager Database, `*.pmdb`)

Note

- Compressed CAD files (e.g., `*.stl.zip`, `*.stl.gz`, `*.stl.bz2`) cannot be imported.
- Filenames with DOS style 8.3 path (shorter path), cannot be imported. Ensure that you give the path name in full while importing the CAD files.
- Virtual topology, suppressed parts/bodies, renamed parts/bodies defined in ANSYS Mechanical/ANSYS Meshing will be ignored during CAD import.

- To import ANSYS DesignModeler files saved with blade geometry created using ANSYS BladeModeler (plug-in for ANSYS DesignModeler), ensure that the Geometry license preference is set to ANSYS BladeModeler as follows:
 1. Select **Tools > License Preferences** in the ANSYS Workbench menu.
 2. Click the **Geometry** tab in the **License Preferences** dialog box.
 3. If ANSYS BladeModeler is not the first license listed, then select it and click **Move up** as required to move it to the top of the list.
 - Conformal tessellation options are not supported for ANSYS ICEM CFD (*.tin) files.
 - Tables of platform-specific supported CAD packages can be found at [Linux](#) or [Windows](#).
-

6.9.9.1. Text Commands for Importing CAD Files

The following commands can be used for importing CAD files:

/file/import/cad

allows you to set the basic options for importing CAD files.

/file/import/cad-options/continue-on-error?

allows you to continue the import of the CAD file(s), despite errors or problems creating the faceting on certain surfaces, or other issues. This option is disabled by default.

/file/import/cad-options/enclosure-symm-processing?

allows processing of enclosure and symmetry named selections during import. This option is disabled by default. This option is applicable only to ANSYS DesignModeler (*.agdb) files.

/file/import/cad-options/import-body-names?

allows import of Body names from the CAD file(s). This option is enabled by default.

Note

Any renaming of Body names in ANSYS Mechanical/ANSYS Meshing prior to the export of the mechdat/meshdat files is ignored during import. Only original Body names will be imported.

/file/import/cad-options/import-part-names?

allows import of Part names from the CAD file(s). This option is enabled by default.

Note

Any renaming of Part names in ANSYS Mechanical/ANSYS Meshing prior to the export of the mechdat/meshdat files is ignored during import. Only original Part names will be imported.

/file/import/cad-options/named-selections

allows you to import **Named Selections** from the CAD file(s), including Named Selections from ANSYS DesignModeler, publications from CATIA, etc. You can additionally choose to ignore import of certain Named Selections based on the pattern specified (e.g., Layer* to ignore layer Named Selections from CATIA), or by specifying multiple wild cards (e.g., ^ (Color | Layer | Material) . * to remove color, layer, and material Named Selections from CATIA).

Note

Named Selections defined in ANSYS Meshing cannot be imported.

/file/import/cad-options/one-object-per

allows you to create one object per body/part/file to be imported. When you choose to import one object per body, you can optionally choose to create one zone per CAD face imported.

/file/import/cad-options/read-all-cad-in-subdirectories?

when enabled, all files in the specified directory as well as in its subdirectories will be imported. This option is disabled by default.

/file/import/cad-options/recover-missing-faces?

allows you to recreate missing face tessellations during the file import.

/file/import/cad-options/save-PMDB?

saves a PMDB (*.pmdb) file in the directory containing the CAD files imported. You can use this file to import the same CAD file(s) again with different options set, for a quicker import than the full import. This option is disabled by default.

Note

Some options will not be available any more once the model is imported from a PMDB file (e.g., enclosure-symm-processing?), since they are processed before the PMDB file is created.

/file/import/cad-options/tessellation

allows you to control the tessellation (faceting) during file import. You can select either tessellation refinement or conformal tessellation.

Tessellation refinement allows you to control the tessellation based on the tessellation refinement tolerance and maximum facet size specified.

Conformal tessellation allows you to use a size field file, (Use size field file?). If you enter yes, it allows you to mention the size field file to be read. If you do not want to use a size field file, you can obtain conformal faceting based on the underlying curve and surface curvature (using the minimum and maximum facet sizes, and the facet curvature normal angle specified) and edge proximity (using the cells per gap specified). You can also save a size field in a file (size field is computed based on the specified parameters i.e. **Min Size**, **Max Size**, **Curvature Normal Angle**, **Cells Per Gap**). In addition, you can merge nodes at the object level with a tolerance of 1e-10.

6.9.10. Grid Import Filter Options

The filter used to import mesh files from third-party packages can take different arguments (see [Appendix A \(p. 647\)](#)). You can control these arguments and the extensions of the files to be converted, using the **Filter Options** dialog box or the related text commands.

File → Import → Options...

6.9.10.1. Text Commands for Setting Filter Options

To import mesh files from third-party packages, use the following text commands:

/file/filter-list

lists the names of the converters that are used to change third-party mesh files to ANSYS Fluent format.

/file/filter-options

allows you to change the extension (e.g., .cas, .msh, .neu) and arguments used with a specified filter.

For example, if you have saved the PATRAN files with a .NEU extension instead of .neu, you can either substitute or add .NEU to the extension list. For some filters, one of the arguments will be the dimensions of the grid. When you use the filter-options command.

6.10. Exporting STL Files

Use the **/file/export/stl** command to save the mesh to a file that can be read by third-party packages. Specify the name for the STL file.

Note

This option is available only through the text interface.

6.11. Saving Picture Files

Graphic window displays can be saved in various formats such as TIFF, EPS, and PostScript. There may be slight differences between the saved picture and the displayed graphics windows. This is because the picture is generated using the internal software renderer, while the graphics windows may utilize specialized graphics hardware for optimum performance.

Many systems provide a utility to “dump” the contents of a graphics window into a raster file. This is generally the fastest method of generating a picture (since the scene is already rendered in the graphics window), and guarantees that the picture will be identical to the window.

For additional information, see the following sections:

[6.11.1. Using the Save Picture Dialog Box](#)

[6.11.2. Text Interface for Saving Picture Files](#)

6.11.1. Using the Save Picture Dialog Box

You can use the **Save Picture** dialog box to set the parameters and to save the picture files. The controls and the equivalent text interface commands are described in [Save Picture Dialog Box \(p. 481\)](#) and [Text Interface for Saving Picture Files \(p. 82\)](#).

For your convenience, the **Save Picture** dialog box may also be opened using the **Save Picture** button () in the standard toolbar. The procedure for saving a picture file is as follows:

1. Select the appropriate file format.
2. Set the coloring.
3. Specify the file type, if applicable (optional).
4. Define the resolution, if applicable (optional).
5. Set the appropriate options (landscape orientation, white background).
6. If you are generating a window dump, specify the command to be used for the dump.
7. Preview the result (optional).
8. Click the **Save...** button and enter the filename in the resulting **Select File** dialog box.

Tip

Click **Apply** instead of **Save...** to save the current settings, instead of saving a picture. The applied settings will become the defaults for subsequent pictures.

Choosing the File Format

To choose the file format, select one of the following options in the **Format** list:

EPS

(Encapsulated PostScript) output is the same as PostScript output, with the addition of Adobe Document Structuring Conventions (v2) statements. Currently, no preview bitmap is included in EPS output. Often, programs which import EPS files use the preview bitmap to display on-screen, although the actual vector PostScript information is used for printing (on a PostScript device). You can save EPS files in raster or vector format.

JPEG

is a common raster file format.

PPM

output is a common raster file format.

PostScript

is a common vector file format. You can also choose to save a PostScript file in raster format.

TIFF

is a common raster file format. The TIFF driver may not be available on all platforms.

PNG

is a common raster file format.

VRML

is a graphics interchange format that allows export of 3D geometrical entities that you can display in the graphics window. This format can commonly be used by VR systems and in particular the 3D geometry can be viewed and manipulated in a web-browser graphics window.

Important

Non-geometric entities such as text, titles, color bars, and orientation axis are not exported. In addition, most display or visibility characteristics set in ANSYS Fluent, such as lighting, shading method, transparency, face and edge visibility, outer face culling, and hidden line removal, are not explicitly exported but are controlled by the software used to view the VRML file.

Window Dump

(LINUX systems only) selects a window dump operation for generating the picture. With this format, you will need to specify the appropriate **Window Dump Command**.

Specifying the Color Mode

For all formats except VRML and the window dump you can specify which type of **Coloring** you want to use for the picture file.

- Select **Color** for a color-scale copy.
- Select **Gray Scale** for a gray-scale copy.
- Select **Monochrome** for a black-and-white copy.

Most monochrome PostScript devices will render **Color** images in shades of gray. Select **Gray Scale** to ensure that the color ramp is rendered as a linearly-increasing gray ramp.

Choosing the File Type

When you save a PostScript or EPS file, you can choose either of the following file types:

- **Raster** : A raster file defines the color of each individual pixel in the image. Raster files have a fixed resolution. Raster supports EPS, JPEG, PPM, PostScript, TIFF, and PNG formats.
- **Vector** : A vector file defines the graphics image as a combination of geometric primitives like lines, polygons, and text. Vector files are usually scalable to any resolution. Vector supports PostScript, EPS, and VRML formats.

Defining the Resolution

For raster files, you can control the resolution of the image by specifying the size in pixels. Set the desired **Width** and **Height** under **Resolution**. If the values of **Width** and **Height** are both zero, the picture is generated at the same resolution as the active graphics window.

Note

For PostScript and EPS files, specify the resolution in dots per inch (**DPI**) instead of setting the width and height.

Picture Options

For all picture formats except VRML and the window dump, you can control two additional settings.

- Specify the orientation of the picture using the **Landscape Orientation** option. If this option is enabled (default), the image is made in landscape mode; otherwise it is made in portrait mode.
- Control the background color using the **White Background** option.

This feature allows you to make pictures with a white background and a black foreground.

ANSYS Fluent also provides options that allow you to save PostScript files that can be printed more quickly. These options are available in the display/set/picture/driver/post-format text menu.

Window Dumps (LINUX Systems)

If you select the **Window Dump** format, the program will use the specified **Window Dump Command** to save the picture file. For example, if you want to use xwd to capture a window, set the **Window Dump Command** to

```
xwd -id %w >
```

ANSYS Fluent will automatically interpret %w to be the ID number of the active window when the dump occurs.

- When you click the **Save...** button, the **Select File** dialog box will appear. Enter the filename for the output from the window dump (e.g., myfile.xwd)
- To make an animation, save the window dumps into numbered files, using the %n variable. To do this, use the Window Dump Command xwd -id %w and type myfile%n.xwd as the filename in the **Select File** dialog box.

Note

Each time you create a new window dump, the value of %n will increase by one, so you need not track numbers to the filenames manually.

If you use the **ImageMagick** animate program, saving the files in MIFF format (the native **ImageMagick** format) is more efficient. In such cases, use the **ImageMagick** tool import. For the **Window Dump Command** enter the default command:

```
import -window %w
```

Specify the output format to be MIFF by using the .miff suffix at the end of the filename.

The window-dump feature is both, system and graphics-driver-specific. The commands available for dumping windows depends on your system configuration.

Important

The window dump will capture the window exactly as it is displayed, including the resolution, colors, transparency, etc. For this reason, all of the inputs that control these characteristics are disabled in the **Save Picture** dialog box when you enable the **Window Dump** format.

If you are using an 8-bit graphics display, you might want to use one of the built-in raster drivers (e.g., TIFF) to generate higher-quality 24-bit color output rather than dumping the 8-bit window. On the other hand, if your hardware supports transparency, a window dump is the only method to generate a picture with transparent surfaces. If you save any other type of picture file, the transparency effects will not be captured in the resulting image.

Previewing the Image

Before saving a picture file, you can preview the image to be saved. Click **Preview** to apply the current settings to the active graphics window so that you can see the effects of different options interactively before saving the image.

6.11.2. Text Interface for Saving Picture Files

Text commands for saving picture files and modifying the picture options are:

display/save-picture

saves a picture file of the active graphics window.

display/set/picture/color-mode/

contains the available color modes.

display/set/picture/color-mode/color

selects full color.

display/set/picture/color-mode/gray-scale

selects gray scale (i.e., various shades of gray).

display/set/picture/color-mode/mono-chrome

selects black and white.

display/set/picture/color-mode/list

displays the current color mode.

display/set/picture/driver/

contains the available formats.

display/set/picture/driver/dump-window

sets the command to dump a graphics window to a file.

display/set/picture/driver/eps

sets Encapsulated PostScript format.

display/set/picture/driver/jpeg

sets JPEG image format.

display/set/picture/driver/post-script

sets PostScript format.

display/set/picture/driver/ppm

sets PPM format.

display/set/picture/driver/tiff

sets TIFF format.

display/set/picture/driver/list

displays the current format.

display/set/picture/driver/options

allows you to set options, such as landscape orientation, and physical size. The options may be entered on one line if you separate them with commas.

display/set/picture/driver/post-format/

contains commands for setting the **PostScript** driver format.

display/set/picture/driver/post-format/fast-raster

enables a raster file that may be larger than the standard raster file, but will print much more quickly.

display/set/picture/driver/post-format/raster

enables the standard raster file.

display/set/picture/driver/post-format/rle-raster

enables a run-length encoded raster file that will be about the same size as the standard raster file, but will print slightly more quickly. This is the default file type.

display/set/picture/driver/post-format/vector

enables the standard vector file.

display/set/picture/invert-background?

toggles the exchange of foreground and/or background colors for picture files.

display/set/picture/landscape?

toggles between landscape or portrait orientation.

display/set/picture/preview

applies the settings of the color-mode, invert-background, and landscape options to the currently active graphics window to preview the appearance of printed pictures.

display/set/picture/x-resolution

sets the width of raster format images in pixels (0 implies that the picture should use the same resolution as the active graphics window).

display/set/picture/y-resolution

sets the height of raster format images in pixels (0 implies that the picture should use the same resolution as the active graphics window).

6.12. The .tgrid File

When starting up in meshing mode, ANSYS Fluent looks in your home directory for an optional file called **.tgrid**. This file is then loaded with the Scheme load function. The **.tgrid** file may contain Scheme functions that customize the operation of the code in meshing mode.

The **.tgrid** file can also contain TUI commands that are executed via the Scheme function **ti-menu-load-string**. For example, if the **.tgrid** file contains (**ti-menu-load-string "file read-**

case test.cas"), then the case file test.cas will be read in. For more details about the function ti-menu-load-string, see [Strings \(p. 51\)](#).

Important

Another optional file, .fluent, if present, is also loaded at start up. This file may contain Scheme functions that customize the operation of the code in solution mode. When both the .tgrid and .fluent files are present, the .fluent file will be loaded first, followed by the .tgrid file, when the meshing mode is launched. Hence, the functions in the .tgrid file will take precedence over those in the .fluent file for the meshing mode.

The .fluent file is not loaded again automatically when switching to solution mode from meshing mode. You will need to load the file separately using the Scheme Load function, if needed.

6.13. Exiting the Meshing Mode

To exit the application, do one of the following:

- Select **Exit** in the **File** pull-down menu.
File → Exit...
- Type /exit within any menu.

If the present state has not been written to a file, you will receive a warning message. You can either cancel the exit and write the mesh or continue to exit without saving the file.

Chapter 7: Size Functions

The size functions provide control over how the mesh size is distributed on a surface or within the volume. Enabling the advanced size functions provides more accurate sizing information for the mesh distribution and more precise refinement control.

You can remesh surfaces and edges based on the size functions defined. The CutCell mesher also uses the size functions to refine the initial Cartesian mesh.

Important

Size functions can be computed only for triangulated zones. For zones comprising non-triangular elements, you can triangulate the zones manually before computing the size functions. Alternatively, you can use the command `triangulate-quad-faces?` before computing the size functions. This command identifies the zones comprising non-triangular elements and uses a triangulated copy of these zones for computing the size functions.

When the advanced size functions are used, the mesh distribution is influenced by

- The minimum and maximum size values
- The growth rate
- The size source which can be any one of the following:
 - Edge and face curvature, based on the normal angle variation between adjacent edges or faces.
 - Edge and face proximity, based on the number of element layers created in a gap between edges or faces.
 - The body of influence defined.
 - Constant user-defined sizes through hard and soft behaviors. The curvature, proximity, body of influence, and soft size functions have soft behavior. The meshed and hard size functions have hard behavior.

This chapter contains the following sections:

- 7.1.Types of Size Functions
- 7.2.Defining Size Functions
- 7.3.Using Size Functions/Size Field
- 7.4.Text Commands for Size Functions

7.1.Types of Size Functions

The following advanced size functions are available:

- 7.1.1.Curvature Size Function
- 7.1.2.Proximity Size Function
- 7.1.3.Mesched Size Function

[7.1.4. Hard Size Function](#)[7.1.5. Soft Size Function](#)[7.1.6. Body of Influence Size Function](#)

7.1.1. Curvature Size Function

The curvature size function examines curvature on edges and faces and computes element sizes on these entities such that the size will not violate the size or the normal angle, which are either automatically computed or defined.

The curvature size function is defined by the following parameters:

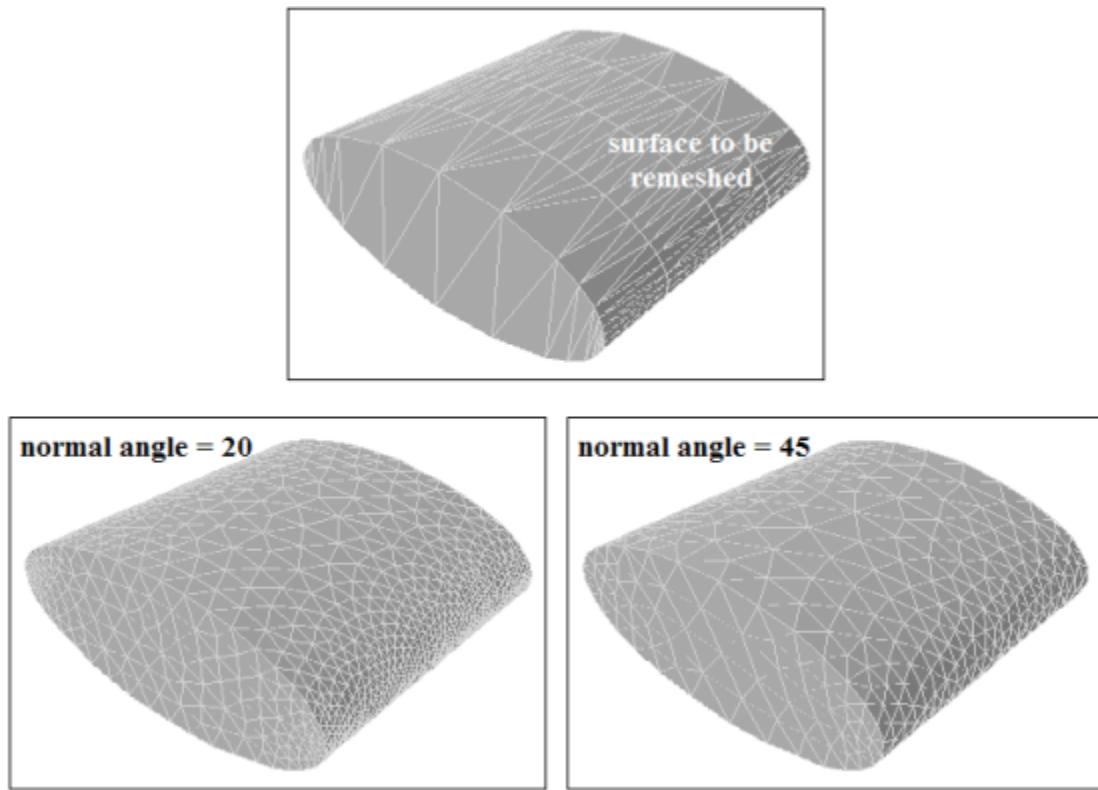
- Min, Max size
- Growth rate
- Normal angle

The curvature normal angle controls the size computation for the curvature size function. The normal angle is the maximum allowable angle that one element edge is allowed to span. For example a value of 5 implies that a division will be made when the angle change along the curve is 5 degree. Hence, a 90 degree arc will be divided into approximately 18 segments.

Note

As the curvature values are computed approximately using edges and face facets, there may be some numerical errors, especially when face facets are excessively stretched.

[Figure 7.1: Use of the Curvature Size Function \(p. 87\)](#) shows an example where the surface has been remeshed based on a curvature size function. The change in normal angle and growth rate controls the size distribution.

Figure 7.1: Use of the Curvature Size Function

7.1.2. Proximity Size Function

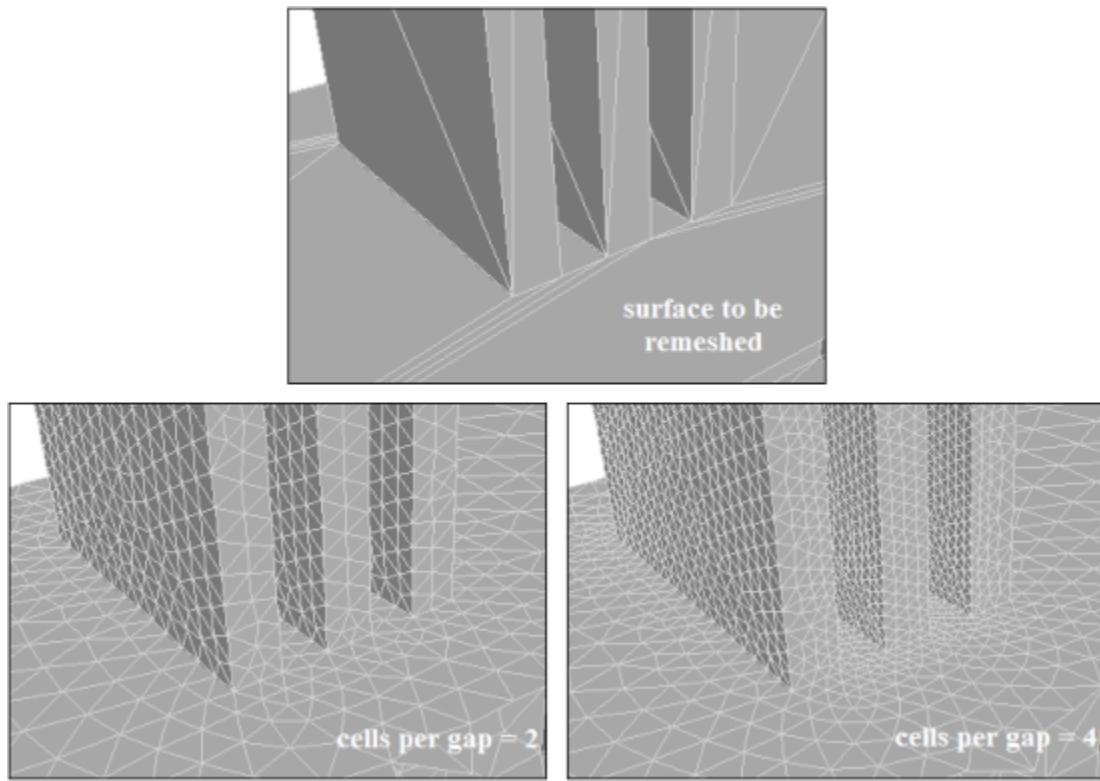
The proximity size function allows you to specify the minimum number of element layers created in regions that constitute 'gaps'. For the purposes of specifying a proximity size function, a 'gap' is defined in one of two ways:

- The area between two opposing boundary edges of a face
- The internal volumetric region between two faces

The proximity size function is defined by the following parameters:

- Min, Max size
- Growth rate
- Cells per gap

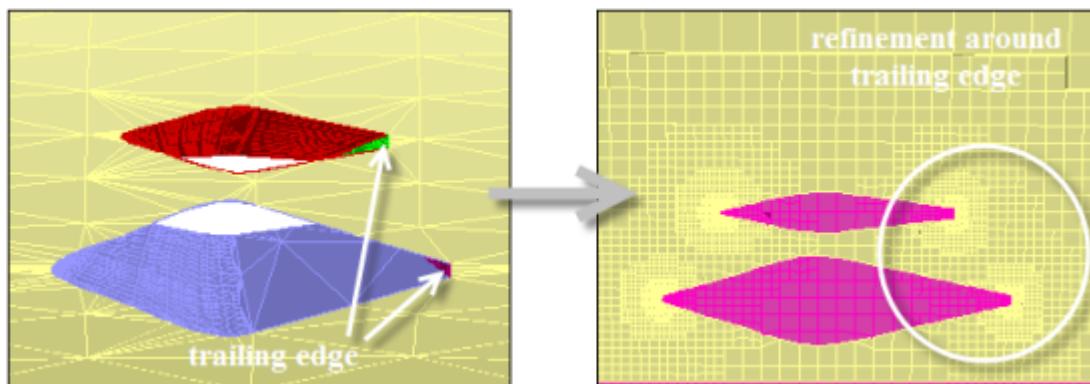
[Figure 7.2: Use of the Proximity Size Function \(p. 88\)](#) shows an example where the surface has been remeshed based on a proximity size function. The change in the cells per gap specified and the growth rate controls the size distribution.

Figure 7.2: Use of the Proximity Size Function

Additional options for defining the face proximity size function are as follows:

- The **Face Boundary** option allows you to compute the shell proximity (edge-edge proximity within each face). The proximity between feature edges on the face zone(s) selected is computed. This option is particularly useful for resolving trailing edges and thin plates without using the hard size function.

The example in [Figure 7.3: Use of the Face Boundary Option for Face Proximity \(p. 89\)](#) shows the use of this option for a blade configuration. Though the normals on the blade surface point outward, the cells across the trailing edges will be refined based on the proximity size function defined for the trailing surfaces when the **Face Boundary** option is enabled.

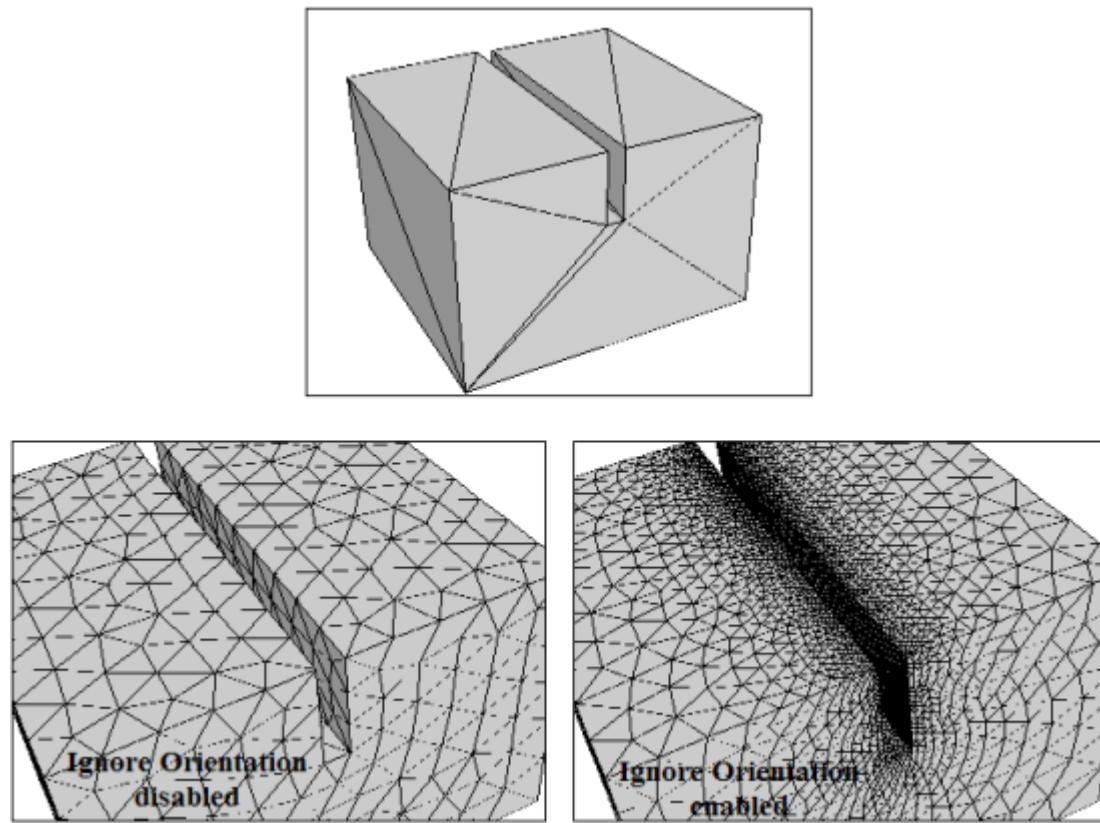
Figure 7.3: Use of the Face Boundary Option for Face Proximity**Note**

The **Face Boundary** option works on the internally extracted boundary edge zones of the face zones. Edge zones extracted in the meshing mode/CAD imported edges will not be considered for the proximity calculation.

- The **Face - Face** option allows you to compute the proximity between two faces in the face zone(s) selected. When the **Face - Face** option is enabled, additional options for ignoring self proximity (**Ignore Self**) and ignoring the face normal orientation (**Ignore Orientation**) are also available.

The **Ignore Self** option can be used with the **Face - Face** option in cases where self proximity (proximity between faces in the same face zone) is to be ignored. This option is disabled by default.

The **Ignore Orientation** option can be used to ignore the face normal orientation during the proximity calculation. This option is enabled by default. In general, the proximity depends on the direction of face normals. An example is shown in [Figure 7.4: Use of the Ignore Orientation Option for Face Proximity \(p. 90\)](#). The normals on the grooved box point inward. With only the **Face - Face** option, the proximity size function does not refine the surface along the entire groove length. When the **Ignore Orientation** option is enabled along with the **Face - Face** option, the surface will be refined along the groove length.

Figure 7.4: Use of the Ignore Orientation Option for Face Proximity

Note

You must select at least one of the **Face Boundary** and **Face - Face** options, otherwise an error will be reported.

The edge proximity size function depends only on the distance between the edges, irrespective of their association with a face zone or the orientation of the face zone(s) associated with the edge zone(s).

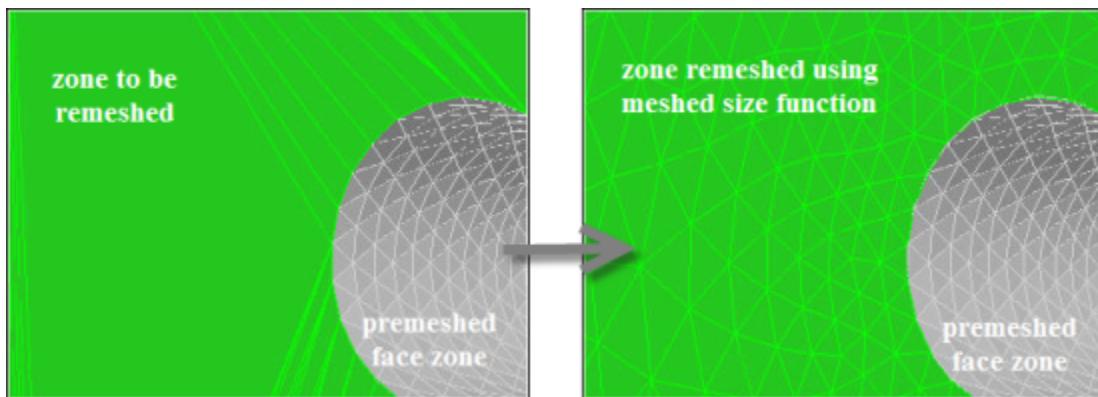
7.1.3. Meshed Size Function

The meshed size function allows you to set the size based on existing sizes. This size function will provide gradation between the minimum and maximum size based on the specified growth rate. The meshed size function will override other size functions (curvature/proximity/soft) defined within its range of influence.

The meshed size function is defined by the following parameters:

- Growth rate

In [Figure 7.5: Use of the Meshed Size Function \(p. 91\)](#), the face zone is remeshed based on the premeshed face zone indicated.

Figure 7.5: Use of the Meshed Size Function

7.1.4. Hard Size Function

The hard size function allows you to maintain a uniform size based on the size specified, while the growth rate from the defined size influences the size on adjacent zones. The hard size function will override any other size function specified.

The hard size function is defined by the following parameters:

- Min Size
- Growth rate

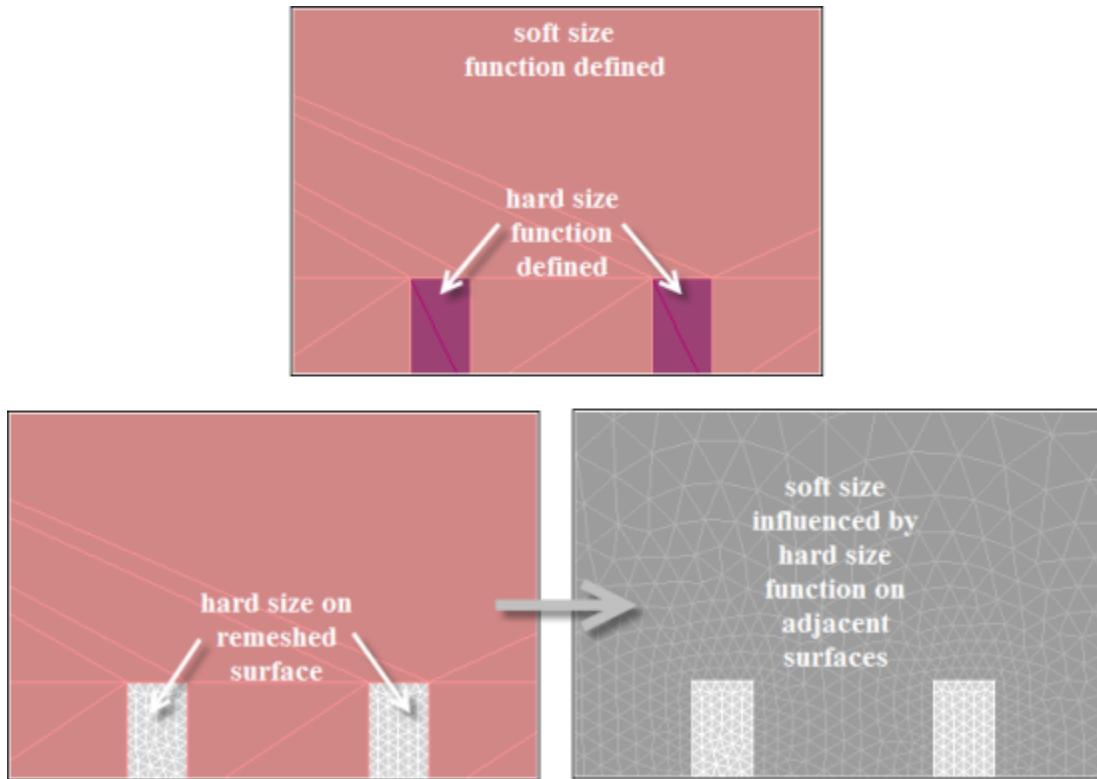
7.1.5. Soft Size Function

The soft size function allows you to set the maximum size on the selected zone, while the specified growth rate from the defined size influences the size on adjacent zones. When the **soft** size function is selected for edges and/or faces, the size control will be affected by other size functions. The minimum size on the zone will be determined based on the influence of other size functions, else a uniform size will be maintained. In other words, a soft size function is ignored in a region where other size functions specify smaller sizes.

The soft size function is defined by the following parameters:

- Max size
- Growth rate

In the example in [Figure 7.6: Use of the Soft Size Function \(p. 92\)](#), the minimum size is determined by the hard size function applied on the smaller face zones indicated, and maximum size is limited by the soft size function applied.

Figure 7.6: Use of the Soft Size Function

7.1.6. Body of Influence Size Function

The body of influence size function allows you to specify a body of influence i.e., a source for sizing control. The maximum mesh size will be equal to the specified size within the body of influence. The minimum size will be determined based on the influence of other size functions. An example is shown in [Figure 7.7: Use of the Body of Influence Size Function \(p. 93\)](#).

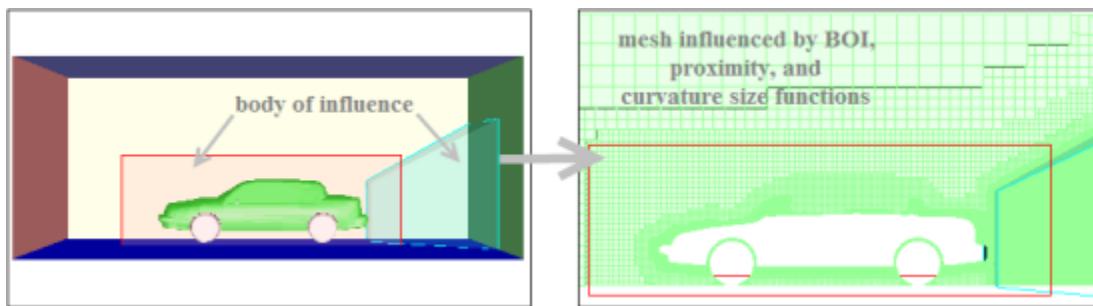
Note

The set of face zones selected to define the body of influence should constitute a geometrically closed body. If an open surface is used as a body of influence, the size function will be processed as a soft size function.

The body of influence size function is defined by the following parameters:

- Max Size
- Growth rate

In [Figure 7.7: Use of the Body of Influence Size Function \(p. 93\)](#), the mesh is generated based on the body of influence size functions defined. The finer mesh size is obtained due to other size functions (e.g., curvature, proximity) defined in addition to the body of influence size functions.

Figure 7.7: Use of the Body of Influence Size Function

7.2. Defining Size Functions

Size functions can be defined using the [Size Functions Dialog Box](#) or using the relevant TUI commands.

Using the Size Functions Dialog Box

The generic procedure to define size functions using the **Size Functions** dialog box is as follows:

1. Ensure that the global controls are set as required. The relevant size function parameter values (minimum and maximum size, growth rate) will be updated based on the global controls specified.
2. Enable **Face Zones** (or **Edge Zones**) as appropriate.
3. Select the boundary zones (or the edge zones) for which the size function is to be defined in the **Boundary Zones** (or **Edge Zones**) selection list.

Note

All boundary face zones and edge zones included in the global domain are available for defining size functions, even if a local domain has been activated.

4. Select the appropriate size function type in the **Size Function Type** drop-down list in the **Define Size Function** group box.
5. Enter an appropriate size function name in the **Name** field or leave the field blank if you want to have the name generated automatically. In this case, the **Name** will be assigned according to the zone type (face or edge) and the size function type (e.g., **face-curvature-sf-5** indicates that the curvature size function is defined for face zones. The size function ID is 5.).
6. Specify the size function parameters applicable for the selected size function as appropriate and click **Create**.

The defined size function will be available in the **Size Functions** list.

Using TUI Commands

Size functions can be defined using the commands in the `size-functions` menu. The command `size-functions/create` allows you to define size functions. You need to specify the size function type, whether face or edge zones are to be used, the boundary or edge zones for which the size function is to be defined, the size function name, and the parameters relevant to the size function type. You can

also have the name generated automatically by retaining the default entry for the size function name. The size function name will be assigned based on the zone type (face or edge) and the size function type (e.g., the size function **face-curvature-sf-5** indicates that the curvature size function is defined for face zones. The size function ID is 5.)

An example is

```
size-functions/create curvature face wall:x wall:y, curv-size-function 0.01 0.1 1.2 5
```

where, **curvature** is the size function type, **face** indicates that face zones are to be used, **wall:x**, **wall:y** indicate the boundary zones for which the size function is defined, and **curv-size-function** is the size function name. The remaining values correspond to the parameters for the curvature size function, i.e., minimum and maximum size, growth rate, and normal angle.

Note

All boundary face zones and edge zones included in the global domain are available for defining size functions, even if a local domain has been activated.

7.2.1. Creating Default Size Functions

You can create default size functions based on face and edge curvature and proximity using the **Create Defaults** option in the [Size Functions Dialog Box \(p. 421\)](#). Alternatively, you can use the command `/size-functions/create-defaults` to create the default size functions.

The following size functions will be defined:

- Curvature size function on all edge zones, with the global minimum and maximum sizes and growth rate, and a normal angle of **18**.
- Curvature size function on all face zones, with the global minimum and maximum sizes and growth rate, and a normal angle of **18**.
- Proximity size function on all edge zones, with the global minimum and maximum sizes and growth rate, and the cells per gap set to **3**.
- Proximity size function on all face zones, with the global minimum and maximum sizes and growth rate, and the cells per gap set to **3**.

When the **Create Defaults** option is used after the default size functions have been created, the previous definitions will be updated based on any changes to the global minimum and maximum sizes and growth rate.

7.2.2. Computing the Size Field

The size field can be computed based on the specified parameters by clicking **Compute** in the **Size Functions** group box in the [Size Functions Dialog Box \(p. 421\)](#). Alternatively, use the command `/size-functions/compute` to compute the size field.

Important

If the size field file has been computed in the current session, sizes will be based on the computed size field. You cannot define additional size functions or modify the current sizes without deleting the size field.

7.2.2.1. Size Field Files

Size field files contain the size function definitions based on the parameters specified.

Select the **File/Read/Size Field...** menu to read a size field file. This will invoke the **Select File** dialog box, where you can specify the name of the size field file to be read. Alternatively, you can use the `/file/read-size-field` command and specify the name of the file to be read.

Important

If a size field file has been read in the current session, sizes will be based on the size field read. You cannot define additional size functions or modify the current sizes without deleting the size field.

Select the **File/Write/Size Field...** menu to save a size field file based on the parameters set. This will invoke the **Select File** dialog box, where you can specify the name of the size field file to be written. Alternatively, you can use the `/file/write-size-field` command and specify the name of the file to be written.

7.2.2.2. Using Size Field Filters

Additional size field filtering options are available after the size field is computed/read.

- You can specify a scale factor to filter the size output from the size field, without deleting and recomputing the size field. The scaling filter can be applied as follows:
 1. Specify an appropriate value for **Factor**, **Min** and **Max** (for **Scale**) in the **Size Field Filters** group box in the [Size Functions Dialog Box \(p. 421\)](#).
 2. Click **Apply** for **Scale** in the **Size Field Filters** group box.

Alternatively, you can use the command `/size-functions/set-scaling-filter`, and specify the scale factor, minimum and maximum size values.

- You can specify periodicity for a specific zone as follows:
 1. Ensure that the periodic boundary is set up.

2. Select the periodic boundary zone in the **Boundary Zones** selection list in the [Size Functions Dialog Box \(p. 421\)](#).

Note

You can select a single source zone to specify periodicity.

Only rotational periodicity is supported, translational periodicity is not supported currently.

3. Click **Apply** for **Periodic** in the **Size Field Filters** group box.

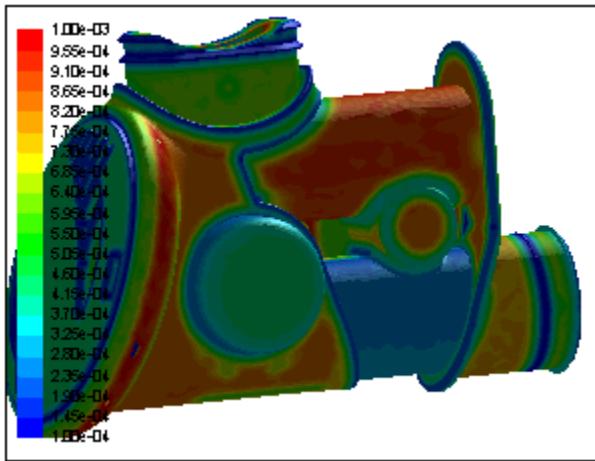
Alternatively, you can use the command `/size-functions/set-periodicity-filter` and specify the periodic source zone.

Click **List** in the **Size Field Filters** group box or use the command `/size-functions/list-periodicity-filter` to view the details of the source zone and rotational periodic parameters specified.

7.2.2.3. Visualizing Sizes

You can also display contours of size on the selected face zones after the size field has been computed or read (see [Figure 7.8: Contours of Size \(p. 96\)](#)).

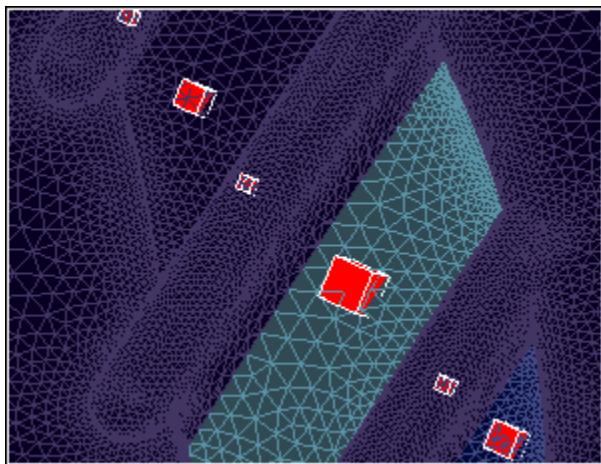
Figure 7.8: Contours of Size



You can display the contours of size as follows:

1. Define size functions and compute the size field. Alternatively, read in the size field file.
2. Select the face zones in the **Boundary Zones** selection list in the [Size Functions Dialog Box \(p. 421\)](#).
3. Specify appropriate values for **Min** and **Max** in the **Contours** group box.
4. Click **Draw** (in the **Contours** group box).

A visual indication of mesh size is also available using the mouse probe if the selection filter is set to **size** (hot-key **Ctrl+y**). Right-click at the required locations to see the size boxes indicating the mesh size (see [Figure 7.9: Display of Mesh Size Based on Size Field \(p. 97\)](#)).

Figure 7.9: Display of Mesh Size Based on Size Field

7.3. Using Size Functions/Size Field

The defined size functions or the size field can be used to remesh surfaces and edges. The CutCell mesher also uses the size functions/size field to refine the initial Cartesian mesh.

Remeshing Surfaces

The generic procedure for remeshing surfaces is as follows:

1. Select the surface to be remeshed in the **Face Zones** list in the **Surface Retriangulation** dialog box.
Boundary → Mesh → Remesh...
2. Enable **Size Function** in the **Face Remesh Options** group box and click the **Specify** button to open the **Size Functions** dialog box.
3. Make sure the size functions are defined as appropriate (see [Defining Size Functions \(p. 93\)](#)) in the **Size Functions** dialog box. Alternatively, ensure that the size field has been computed or read in.
4. Set the other options for face remeshing as appropriate.
5. Click **Remesh** in the **Surface Retriangulation** dialog box.

Important

Edge zone(s) associated with face zones are not remeshed implicitly. If you have feature edge zone(s) associated with the surface being remeshed, you need to remesh them before remeshing the face zone(s).

Remeshing Edges

The generic procedure for remeshing edges is as follows:

1. Ensure that the edge zones are extracted as required.
Boundary → Mesh → Feature...

2. Select the edges to be remeshed in the **Edge Zones** list in the **Feature Modify** dialog box.
3. Select **Remesh** in the **Options** list and select **Size Function** in the **Method** drop-down list.
4. Make sure the size functions are defined as appropriate in the **Size Functions** dialog box (see [Defining Size Functions \(p. 93\)](#)). Alternatively, ensure that the size field has been computed or read in.
5. Click **Remesh** in the **Surface Retriangulation** dialog box.

Refining the CutCell Mesh

The CutCell mesher uses the size functions/size field to refine the initial Cartesian mesh as described in [The CutCell Meshing Process \(p. 295\)](#).

7.4. Text Commands for Size Functions

The following text commands allow you to define and manipulate the size functions:

/size-functions/compute

computes the size functions based on the defined parameters.

/size-functions/contours/draw

displays contours in the graphics window. Compute the size field using **/size-functions/compute** or read in a size field file prior to displaying the contours of size.

/size-functions/contours/set/refine-facets?

allows you to specify smaller facets if the original are too large. Default is no.

/size-functions/create

defines the size function based on the specified parameters.

/size-functions/create-defaults

creates default size functions based on face and edge curvature and proximity.

/size-functions/delete

deletes the specified size function or the current size field.

/size-functions/delete-all

deletes all the defined size functions.

/size-functions/list

lists all the defined size functions and the corresponding parameter values defined.

/size-functions/list-periodicity-filter

lists the details of the source zone and rotational periodic parameters specified for the size field.

/size-functions/reset-global-controls

resets the global controls to their default values.

/size-functions/set-global-controls

sets the values for the global minimum and maximum size, and the growth rate.

Note

If you set the global minimum size to a value greater than the local minimum size defined for existing proximity, curvature, or hard size functions, a warning will appear, indicating that the global minimum size cannot be greater than the specified local minimum size.

/size-functions/set-periodicity-filter

allows you to apply periodicity to the size field by selecting one source face zone.

Note

Ensure that periodicity is previously defined in the **Make Periodic Boundaries** dialog box.

Only rotational periodicity is supported, translational periodicity is not supported currently.

/size-functions/set-prox-gap-tolerance

sets the tolerance relative to minimum size to take gaps into account. Gaps whose thickness is less than the global minimum size multiplied by this factor will not be regarded as a proximity gap.

Tip

False proximity computed can cause unnecessary over-refinement in areas of overlapping facets. You can increase the relative tolerance value in such cases.

/size-functions/set-scaling-filter

allows you specify the scale factor, and minimum and maximum size values to filter the size output from the size field.

/size-functions/triangulate-quad-faces?

identifies the zones comprising non-triangular elements and uses a triangulated copy of these zones for computing the size functions.

/size-functions/un-set-periodicity-filter

removes periodicity from the size field.

/boundary/remesh/size-functions/compute

computes the size function based on the defined parameters.

/boundary/remesh/size-functions/contours/draw

displays contours in the graphics window. Compute the size field using **/size-functions/compute** or read in a size field file prior to displaying the contours of size.

/boundary/remesh/size-functions/contours/set/refine-facets?

allows you to specify smaller facets if the original are too large. Default is no.

/boundary/remesh/size-functions/create

defines the size function based on the specified parameters.

/boundary/remesh/size-functions/create-defaults

creates default size functions based on face and edge curvature and proximity.

/boundary/remesh/size-functions/delete

deletes the specified size function or the current size field.

/boundary/remesh/size-functions/delete-all

deletes all the defined size functions.

/boundary/remesh/size-functions/list

lists all the defined size functions and the corresponding parameter values defined.

/boundary/remesh/size-functions/list-periodicity-filter

lists the details of the source zone and rotational periodic parameters specified for the size field.

/boundary/remesh/size-functions/reset-global-controls

resets the global controls to their default values.

/boundary/remesh/size-functions/set-global-controls

sets the values for the global minimum and maximum size, and the growth rate.

Note

If you set the global minimum size to a value greater than the local minimum size defined for existing proximity, curvature, or hard size functions, a warning will appear, indicating that the global minimum size cannot be greater than the specified local minimum size.

/boundary/remesh/size-functions/set-periodicity-filter

allows you to apply periodicity to the size field by selecting one source face zone.

Note

Ensure that periodicity is previously defined in the **Make Periodic Boundaries** dialog box.

Only rotational periodicity is supported, translational periodicity is not supported currently.

/boundary/remesh/size-functions/set-prox-gap-tolerance

sets the tolerance relative to minimum size to take gaps into account. Gaps whose thickness is less than the global minimum size multiplied by this factor will not be regarded as a proximity gap.

Tip

False proximity computed can cause unnecessary over-refinement in areas of overlapping facets. You can increase the relative tolerance value in such cases.

/boundary/remesh/size-functions/set-scaling-filter

allows you specify the scale factor, and minimum and maximum size values to filter the size output from the size field.

/boundary/remesh/size-functions/triangulate-quad-faces?

identifies the zones comprising non-triangular elements and uses a triangulated copy of these zones for computing the size functions.

/boundary/remesh/size-functions/un-set-periodicity-filter

removes periodicity from the size field.

/mesh/cutcell/size-functions/compute

computes the size function based on the defined parameters.

/mesh/cutcell/size-functions/contours/draw

displays contours in the graphics window. Compute the size field using /size-functions/compute or read in a size field file prior to displaying the contours of size.

/mesh/cutcell/size-functions/contours/set/refine-facets?

allows you to specify smaller facets if the original are too large. Default is no.

/mesh/cutcell/size-functions/create

defines the size function based on the specified parameters.

/mesh/cutcell/size-functions/create-defaults

creates default size functions based on face and edge curvature and proximity.

/mesh/cutcell/size-functions/delete

deletes the specified size function or the current size field.

/mesh/cutcell/size-functions/delete-all

deletes all the defined size functions.

/mesh/cutcell/size-functions/list

lists all the defined size functions and the corresponding parameter values defined.

/mesh/cutcell/size-functions/list-periodicity-filter

lists the details of the source zone and rotational periodic parameters specified for the size field.

/mesh/cutcell/size-functions/reset-global-controls

resets the global controls to their default values.

/mesh/cutcell/size-functions/set-global-controls

sets the values for the global minimum and maximum size, the growth rate, and the refine factor.

Note

If you set the global minimum size to a value greater than the local minimum size defined for existing proximity, curvature, or hard size functions, a warning will appear, indicating that the global minimum size cannot be greater than the specified local minimum size.

/mesh/cutcell/size-functions/set-periodicity-filter

allows you to apply periodicity to the size field by selecting one source face zone.

Note

Ensure that periodicity is previously defined in the **Make Periodic Boundaries** dialog box.

Only rotational periodicity is supported, translational periodicity is not supported currently.

/mesh/cutcell/size-functions/set-prox-gap-tolerance

sets the tolerance relative to minimum size to take gaps into account. Gaps whose thickness is less than the global minimum size multiplied by this factor will not be regarded as a proximity gap.

Tip

False proximity computed can cause unnecessary over-refinement in areas of overlapping facets. You can increase the relative tolerance value in such cases.

/mesh/cutcell/size-functions/set-scaling-filter

allows you specify the scale factor, and minimum and maximum size values to filter the size output from the size field.

/mesh/cutcell/size-functions/triangulate-quad-faces?

identifies the zones comprising non-triangular elements and uses a triangulated copy of these zones for computing the size functions.

/mesh/cutcell/size-functions/un-set-periodicity-filter

removes periodicity from the size field.

Chapter 8: Meshing Objects and Material Points

This chapter describes the use of meshing objects and material points.

- 8.1. Objects
- 8.2. Material Points

8.1. Objects

Objects are used to identify the domain to be meshed. An object is generally a set of face zones and edge zones. Including edge zones in the object definition improves the quality of mesh generated from the object. Objects are generally closed solid volumes or closed fluid (wetted) volumes, or capping surfaces or individual face zones which can be used for meshing. For example, using capping surfaces in conjunction with a material point and a closed solid volume allows you to extract the fluid volume.

Note

Objects (defined or imported) are independent of each other, i.e., objects do not share face and/or edge zones. In cases where objects are defined using a common face/edge zone (see [Figure 8.5: Objects Defined Using the Domain Method \(p. 106\)](#)), the common face/edge zone(s) will be duplicated to make the objects independent.

Objects can be defined and manipulated using the options in the **Manage Objects** dialog box or using the commands in the objects menu.

- 8.1.1. Object Attributes
- 8.1.2. Object Based Meshing
- 8.1.3. Using the Manage Objects Dialog Box
- 8.1.4. Text Commands for Objects

8.1.1. Object Attributes

Each object defined (or imported) has attributes such as object type, cell zone type and priority.

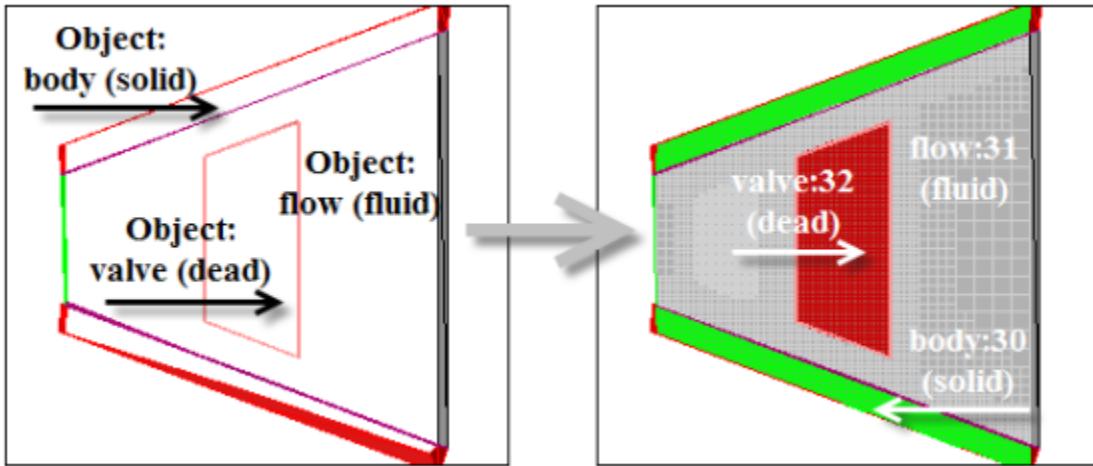
The following types of objects are available:

- Geometry: objects imported through CAD using the import options available (see [Importing CAD Files \(p. 72\)](#)) or created for a given geometry. The geometry objects may be non-conformal.
- Wrap: objects which are good quality, body-conformal representations of each topological body. Wrap objects may contain zero or more volumes, but no two volumes may be connected via a shared face. They may be created using the object wrapping options, or directly from capping surfaces, bounding box surfaces, or cylindrical/annular surfaces.
- Mesh: objects which are good quality, well-connected, conformal surface mesh representations of the geometry. Mesh objects may contain multiple volumes with shared faces, and they may be created using the Sew, Improve operations, or Build Topology operations.

The object cell zone type indicates the type of cell zone created when the mesh is generated based on objects.

[Figure 8.1: CutCell Mesh With Different Cell Zone Types \(p. 104\)](#) shows the CutCell mesh with different cell zone type assigned to respective objects.

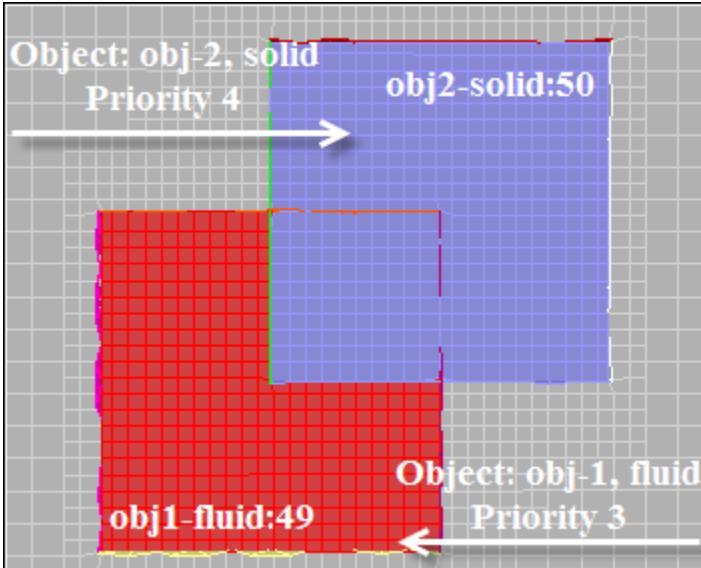
Figure 8.1: CutCell Mesh With Different Cell Zone Types



The object priority controls the inclusion of the mesh entities. In case of overlapping objects, the entities in the overlapped region will be included with the object having a higher priority value.

[Figure 8.2: Use of the Object Priority for Overlapping Objects \(p. 104\)](#) shows an example with overlapping objects. The overlapped region is included with the zone corresponding to the object having the higher priority value (in this case, 4).

Figure 8.2: Use of the Object Priority for Overlapping Objects



Note

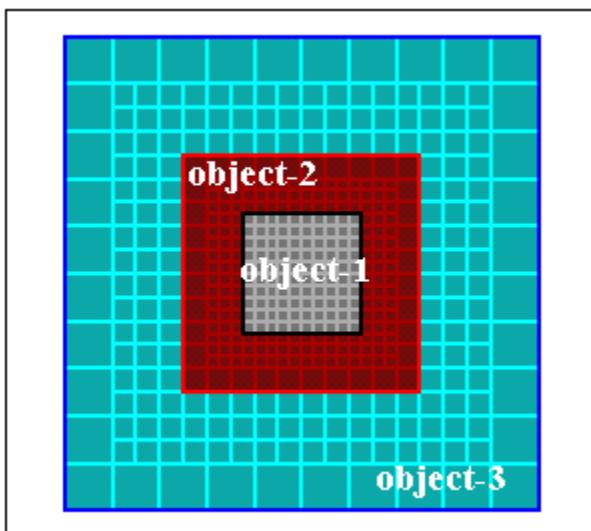
Multiple objects having the same priority assigned will be merged into a single cell zone, irrespective of cell zone type.

Priority is also important when objects are created for bounding boxes or wind tunnels, etc. In such cases, the object created for a bounding box or wind tunnel must be assigned the lowest priority.

8.1.1.1. Approaches For Creating Objects

Figure 8.3: Creating Objects—Example (p. 105) shows an example with three non-intersecting bodies.

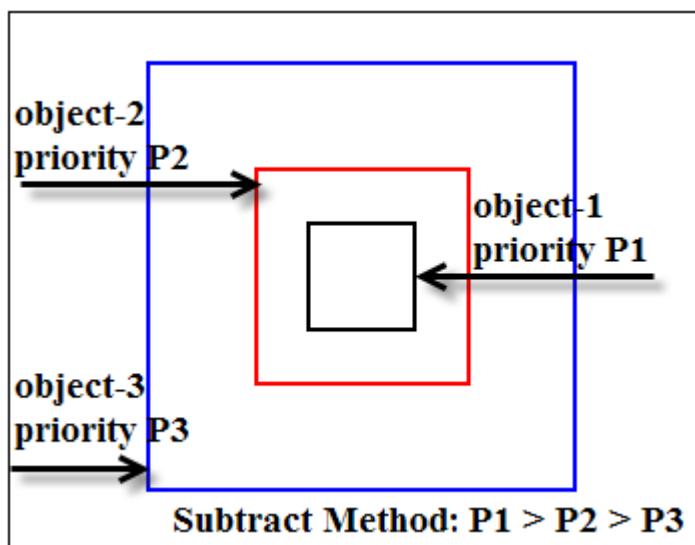
Figure 8.3: Creating Objects—Example



There are two possible methods to set up the objects in this case.

- **Subtract Method** The order of priority assigned to individual objects is important when using the Subtract method. In this case the order of priority is object-1 > object-2 > object-3.

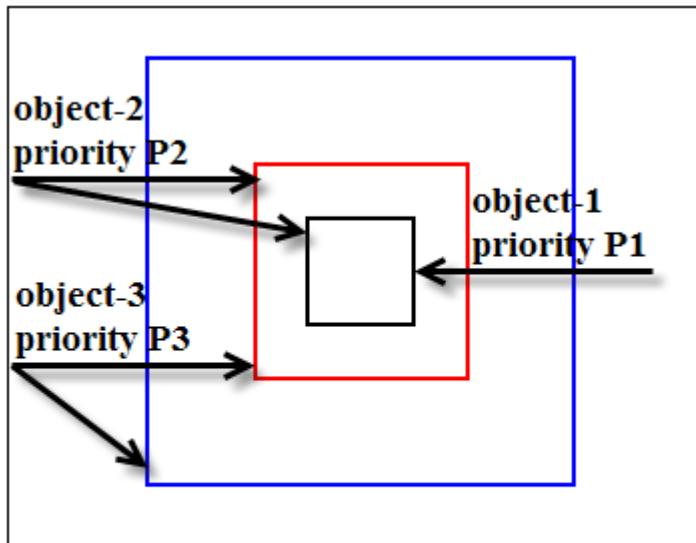
Figure 8.4: Objects Defined Using the Subtract Method



1. Select the face and edge zones comprising the innermost body, set the **Cell Zone** type, and set the highest **Priority** value.
2. Select the face and edge zones comprising the inner body, set the **Cell Zone** type, and set an intermediate **Priority** value.

3. Select the face and edge zones comprising the outer body, set the **Cell Zone** type, and set the lowest **Priority** value.
- **Domain Method** The Domain method requires you to select the boundary zones enclosing each domain for the object definition. This method only requires that the individual objects have different priorities assigned, the order of priority is not important.

Figure 8.5: Objects Defined Using the Domain Method



Domain Method: Zones Selected Define Domain to be Meshed, $P1 \neq P2 \neq P3$

1. Select the face and edge zones comprising the innermost body and set the **Cell Zone** type and **Priority**.
2. Select the face and edge zones comprising the innermost and the inner body, and set the **Cell Zone** type and **Priority**.
3. Select the face and edge zones comprising the inner and the outer body, and set the **Cell Zone** type and **Priority**.

Note

The common face and/or edge zone(s) will be duplicated to ensure that the objects defined are independent of each other (do not share face/edge zones).

8.1.2. Object Based Meshing

Objects can be used for mesh generation as described:

Auto Mesh Mesh objects can be used along with material points to identify the volume to be meshed when using the **Auto Mesh** dialog box. You can select the appropriate mesh object in the **Object** drop-

down list along with the necessary material point(s). See [Using the Auto Mesh Dialog Box \(p. 219\)](#) for details on using the **Auto Mesh** dialog box.

Note

The **Merge Cell Zones** and **Auto Identify Topology** options are not available when a mesh object is selected for volume meshing.

CutCell Meshing Objects can be used to test inclusion of Cartesian grid entities for CutCell meshing. The CutCell mesher uses the objects of **geometry** or **wrap** type as the input geometry, and not the **geometry** thread type. The Cartesian grid will be refined and snapped to any face zone included in the object(s) selected for CutCell meshing.

8.1.3. Using the Manage Objects Dialog Box

The **Manage Objects** dialog box contains options allowing you to define objects and perform certain object manipulation operations as described in the following sections:

- 8.1.3.1. Defining Objects
- 8.1.3.2. Creating Multiple Objects
- 8.1.3.3. Object Manipulation Operations
- 8.1.3.4. Object Transformation Operations
- 8.1.3.5. Automatic Alignment of Objects

8.1.3.1. Defining Objects

You can define the objects using the options in the **Define** tab of the **Manage Objects** dialog box.

1. Select the set of face zones comprising the object in the **Face Zones** selection list.

Note

Only tri face zones can be selected for defining objects.

2. Select the set of edge zones comprising the object in the **Edge Zones** selection list.

Ensure that edge zones have been created to include them in the object definition.

3. Enter an appropriate name in the **Object Name** field.

You can also have the object name generated automatically by leaving the **Object Name** field blank. In this case, the object name will be assigned based on the prefix, cell zone type, and priority specified (e.g., an object named **object-fluid:3-20** has prefix **object-**, cell zone type **fluid**, priority **3**, and object ID **20**).

4. Select the appropriate option from the **Cell Zone Type** drop-down list.
5. Set the priority.
6. Select the appropriate type from the **Object Type** drop-down list (default, **geom**).
7. Click **Create**.

You can modify the object definition using the **Change** option in the **Manage Objects** dialog box. Select the object to be modified, make the necessary changes in the **Definition** tab, and click **Change**.

Note

If conformal tessellation options are selected during import, you may convert geometry objects to wrap or mesh type directly using the **Change/Change Type** (available when multiple objects are selected) option in the [Manage Objects Dialog Box \(p. 430\)](#) or the `change-object-type` command. When you change the object type to wrap or mesh, it is assumed that all selected objects have conformal faceting and that the quality of the surface mesh triangles is similar to that in a CFD surface mesh.

Objects can be deleted using the **Delete** option in the **Definition** tab. You can also enable **Include Faces and Edges** to delete the faces and edges comprising the object, when the object is deleted.

Note

When an object is deleted along with the face and edge zones comprising the object, any corresponding face/edge groups will also be deleted.

8.1.3.2. Creating Multiple Objects

You can also create one object per selected zone using the command `/objects/create-multiple`. This is particularly useful for CAD exported models, as you can define an object per part. An object will be created for each selected zone and will be named automatically based on the specified prefix and priority. The name assigned is **prefix face zone name-priority:object ID** (e.g., an object named **object-wall-3:20** will be created for the face zone **wall**, with the specified prefix **object-** and priority **3**. The object ID is **20**).

Specify the first and last zone for which objects are to be created. Note that you need to specify valid zone names or IDs. You can also use wild-cards for specifying the face zones to be considered. Specify the prefix to be used for the object name and the cell zone type. Specify the priority for the first object (for the first zone selected) and the increment in priority. When the increment is set to a value greater than zero, the priority will be assigned in the order of face zone ID. If the increment is set to zero, all objects will have the same priority.

A geometry object will be created for the first and last zone (as specified) and for all valid face zones having IDs between the first and last zone.

8.1.3.3. Object Manipulation Operations

The following object manipulation operations can be performed in the **Operations** tab:

- Geometry objects can be merged using the **Merge Objects** option.
- Wall face zones comprising objects can be merged using the **Merge Walls** option.

- The edge zones comprising an object can be merged into a single edge zone using the **Merge Edges** option.

Note

If the object comprises edge zones of different types (boundary and interior), the edge zones of the same type (boundary or interior) will be merged into a single edge zone.

- Intersection loops can be created within an object or between objects using the options in the **Intersection Loops** group box.
- Edge zones can be extracted from the face zones included in the specified objects, based on the feature angle value specified using the options in the **Extract Edges** group box. You can specify whether only feature edges or all edges are to be extracted for the object(s) selected. Previously created edges will automatically be de-associated from the object and deleted.
- A face group and an edge group comprising the face zone(s) and edge zone(s) included in the specified object(s) can be created using the options in the **Zone Group** group box.
- The face zone(s) comprising the object can be separated based on the angle or seed specified using the options in the **Separate Faces** group box.
- When the face zones and/or edge zones comprising an object are deleted, you will be prompted to update the object definition. You can use the command /objects/update-objects to update the defined objects per the changes.
- You can rename the face and edge zones comprising the object based on the object name and also specify the separator to be used.

8.1.3.4. Object Transformation Operations

The following object transformation operations can be performed using the options available in the **Transformations** tab:

- Objects can be rotated using the **Rotate** option. Specify the angle of rotation and the pivot point and the axis of rotation by selecting 1-6 nodes in the graphics window.

The pivot point and the axis of rotation can be defined by selecting 1-6 nodes as follows:

- If only 1 node is selected, the pivot point is at the node location and the axis of rotation is the global z-axis.
- For 2 nodes, the pivot point is at the midpoint of the nodes selected and the axis of rotation is the global z-axis.
- For 3 nodes, the pivot point is at the first node selected. The axis of rotation is the local z-axis normal to the plane defined by the three points, the positive direction is determined by the right-hand rule.
- For 4, 5 or 6 nodes, the first 3 points define a circle. The pivot point is at the center of the circle. The axis of rotation is the local z-axis normal to the circular plane, the positive direction is determined by the right-hand rule.

- Objects can be scaled using the **Scale** option. Specify the scale factors (X, Y, Z) for the scaling operation.
- Objects can be translated using the **Translate** option. Specify the vector components to define the translation, or click **Define** and select two screen locations to determine the translation.

8.1.3.5. Automatic Alignment of Objects

You may also fit objects together in precise alignment, for example to position a flange on a body by aligning bolt holes.

The procedure is initiated via a hotkey and uses temporary local coordinate systems (LCS) to achieve the alignment.

Defining a local coordinate system by selecting 1-6 nodes works as follows:

- If only 1 node is selected, the LCS origin is at the node location and axes are aligned to global coordinate system.
- For 2 nodes, the LCS origin is at the midpoint of nodes and axes are aligned to global coordinate system.
- For 3 nodes, the origin is at the first point, the LCS x-axis is along a vector from the first to the second point, and the LCS y-axis is along a vector from the first point to the 3rd point.
- For 4, 5 or 6 nodes, the first 3 points define a circle. The LCS origin is at the center of the circle and the z-axis is normal to the circular plane (positive direction is determined by the right-hand rule).
 - For 4 nodes, the x-axis is defined by a vector from the center of the circle to the projection of the 4th point on the circular plane.
 - For 5 nodes, the x-axis is defined by a vector from the center of the circle to the projection of the mid-point of 4th and 5th points on the circular plane.
 - For 6 nodes, the x-axis is defined by a vector from the center of the circle to the projection of the circumcenter of 4th, 5th and 6th points on the circular plane.

8.1.4. Text Commands for Objects

The text commands related to defining and manipulating objects are as follows:

/objects/change-object-type

allows you to change the object type (geom, wrap, or mesh).

/objects/create

creates the object based on the priority, cell zone type, face zone(s), edge zone(s), and object type specified. You can specify the object name or retain the default blank entry to have the object name generated automatically.

/objects/create-and-activate-domain

creates and activates the domain comprising the face zone(s) from the object(s) specified.

/objects/create-groups

creates a face group and an edge group comprising the face zone(s) and edge zone(s) included in the specified object(s), respectively.

/objects/create-intersection-loops

allows you to create intersection loops for objects.

- The `collectively` option creates an interior edge loop at the intersection between two adjacent face zones included in the same object and between multiple objects.
- The `individually` option creates an interior edge loop at the intersection between two adjacent face zones included in the same object.

/objects/create-multiple

creates multiple objects by creating an object per face zone specified. The objects will be named automatically based on the prefix and priority specified.

/objects/delete

deletes the specified object(s).

/objects/delete-all

deletes all the defined objects.

/objects/delete-all-geom-and-wrap

deletes all the defined objects of type `geom` and `wrap`.

/objects/delete-unreferenced-faces-and-edges

deletes all the faces and edges which are not included in any defined objects.

/objects/extract-edges

extracts the edge zone(s) from the face zone(s) included in the specified object(s), based on the edge-feature-angle value specified (`/objects/set/set-edge-feature-angle`).

/objects/list

lists the defined objects, indicating the respective cell zone type, priority, face zone(s) and edge zone(s) comprising the object, object type, and object reference point in the console.

/objects/merge

merges the specified objects into a single geometry object. The resulting object inherits the name of the object selected first.

/objects/merge-edges

merges all the edge zones in an object into a single edge zone.

Note

If the object comprises edge zones of different types (boundary and interior), the edge zones of the same type (boundary or interior) will be merged into a single edge zone.

/objects/merge-nodes

merges the free nodes at the object level based on the specified tolerance or using a tolerance that is a specified percentage of shortest connected edge length.

/objects/merge-walls

merges all the face zones of type `wall` in an object into a single face zone.

/objects/purge-wrap

deletes the face zones associated with the wrap object(s) selected and reverts the object(s) to their geometry representations.

/objects/rename-object-zones

renames the face and edge zones comprising the object based on the object name. You can also specify the separator to be used.

/objects/rotate

rotates the object(s) based on the angle of rotation, pivot point, and axis of rotation specified.

/objects/scale

scales the object(s) based on the scale factors specified.

/objects/separate-faces-by-angle

separates the face zone(s) comprising the object based on the angle specified.

/objects/separate-faces-by-seed

separates the face zone(s) comprising the object based on the seed face specified.

/objects/set/set-edge-feature-angle

sets the edge feature angle to be used for extracting edge zone(s) from the face zone(s) included in the object(s).

/objects/set/show-edge-zones?

displays the edge zone(s) comprising the object(s) drawn in the graphics window.

/objects/set/show-face-zones?

displays the face zone(s) comprising the object(s) drawn in the graphics window.

/objects/translate

translates the object(s) based on the translation offsets specified.

/objects/update

allows you to update the objects defined when the face and/or edge zone(s) comprising the object have been deleted.

/mesh/cutcell/objects/change-object-type

allows you to change the object type (geom, wrap, or mesh).

/mesh/cutcell/objects/create

creates the object based on the priority, cell zone type, face zone(s), edge zone(s), and object type specified. You can specify the object name or retain the default blank entry to have the object name generated automatically.

/mesh/cutcell/objects/create-and-activate-domain

creates and activates the domain comprising the face zone(s) from the object(s) specified.

/mesh/cutcell/objects/create-groups

creates a face group and an edge group comprising the face zone(s) and edge zone(s) included in the specified object(s), respectively.

/mesh/cutcell/objects/create-intersections

allows you to create intersection loops for objects.

- The **collectively** option creates an interior edge loop at the intersection between two adjacent face zones included in the same object and between multiple objects.
- The **individually** option creates an interior edge loop at the intersection between two adjacent face zones included in the same object.

/mesh/cutcell/objects/create-multiple

creates multiple objects by creating an object per face zone specified. The objects will be named automatically based on the prefix and priority specified.

/mesh/cutcell/objects/delete

deletes the specified object(s).

/mesh/cutcell/objects/delete-all

deletes all the defined objects.

/mesh/cutcell/objects/delete-all-geom-and-wrap

deletes all the defined objects of type geom and wrap.

/mesh/cutcell/objects/delete-unreferenced-faces-and-edges

deletes all the faces and edges which are not included in any defined objects.

/mesh/cutcell/objects/extract-edges

extracts the edge zone(s) from the face zone(s) included in the specified object(s), based on the edge-feature-angle value specified (/mesh/cutcell/objects/set/set-edge-feature-angle).

/mesh/cutcell/objects/merge

merges the specified geometry objects into a single object.

/mesh/cutcell/objects/merge-edges

merges all the edge zones in an object into a single edge zone.

Note

If the object comprises edge zones of different types (boundary and interior), the edge zones of the same type (boundary or interior) will be merged into a single edge zone.

/mesh/cutcell/objects/merge-nodes

merges the free nodes at the object level based on the specified tolerance or using a tolerance that is a specified percentage of shortest connected edge length.

/mesh/cutcell/objects/merge-walls

merges all the face zones of type wall in an object into a single face zone.

/mesh/cutcell/objects/purge-wrap

deletes the face zones associated with the wrap object(s) selected and reverts the object(s) to their geometry representations.

/mesh/cutcell/objects/rename-object-zones

renames the face and edge zones comprising the object based on the object name. You can also specify the separator to be used.

/mesh/cutcell/objects/rotate

rotates the object(s) based on the angle of rotation, pivot point, and axis of rotation specified.

/mesh/cutcell/objects/scale

scales the object(s) based on the scale factors specified.

/mesh/cutcell/objects/separate-faces-by-angle

separates the face zone(s) comprising the object based on the angle specified.

/mesh/cutcell/objects/separate-faces-by-seed

separates the face zone(s) comprising the object based on the seed face specified.

/mesh/cutcell/objects/set/set-edge-feature-angle

sets the edge feature angle to be used for extracting edge zone(s) from the face zone(s) included in the object(s).

/mesh/cutcell/objects/set/show-edge-zones?

displays the edge zone(s) comprising the object(s) drawn in the graphics window.

/mesh/cutcell/objects/set/show-face-zones?

displays the face zone(s) comprising the object(s) drawn in the graphics window.

/mesh/cutcell/objects/translate

translates the object(s) based on the translation offsets specified.

/mesh/cutcell/objects/update

allows you to update the objects defined when the face and/or edge zone(s) comprising the object have been deleted.

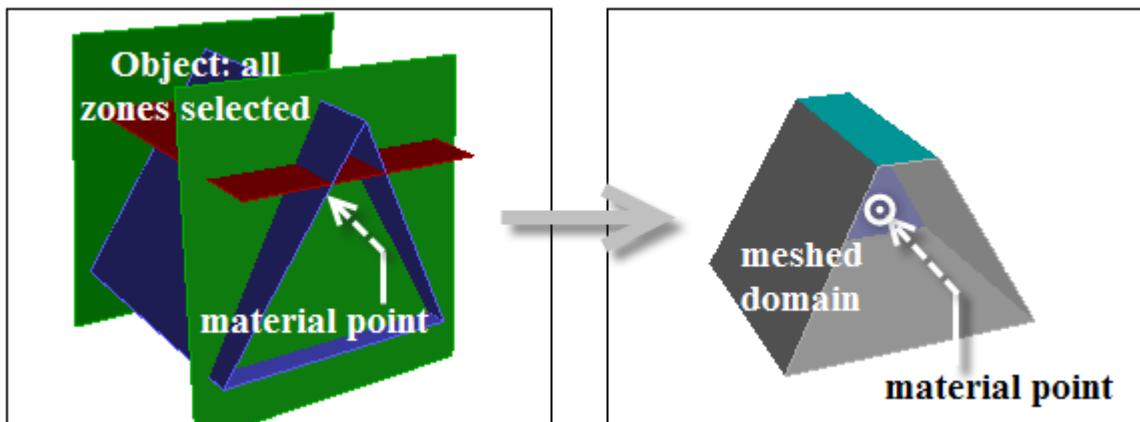
For additional commands related to the object based meshing workflow, refer to [Text Commands for Object Based Meshing \(p. 343\)](#).

8.2. Material Points

In addition to objects, material points can be defined to allow the mesher to separate the cell zone. Typically, a material point can be defined to retrieve a cell zone for which an object cannot be defined, or the object definition alone is not sufficient to retrieve the cell zone. Material points can also be used to retrieve region(s) from a non-contiguous cell zone. Contiguous regions will be separated based on the respective material points defined. The cell zone retrieved based on the defined material point will be of the type fluid and have the specified name.

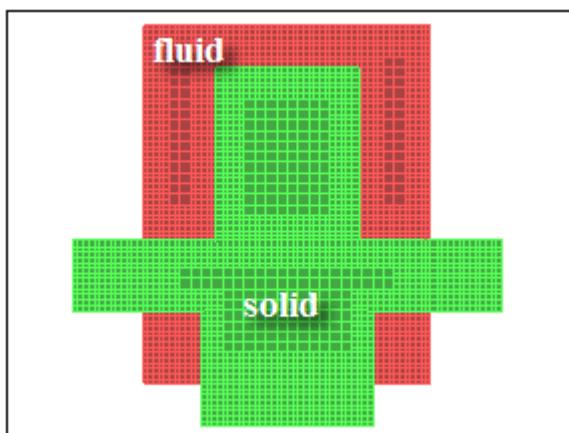
The following examples demonstrate the use of a material point in addition to objects:

- Some cases involving “dirty” geometry may result in multiple voids. In this case, the volume to be meshed can be recovered by defining an object comprising the zones enclosing the domain to be meshed, combined with a material point within the expected meshed domain (see [Figure 8.6: Using Material Points—Example \(p. 115\)](#)).

Figure 8.6: Using Material Points—Example**Note**

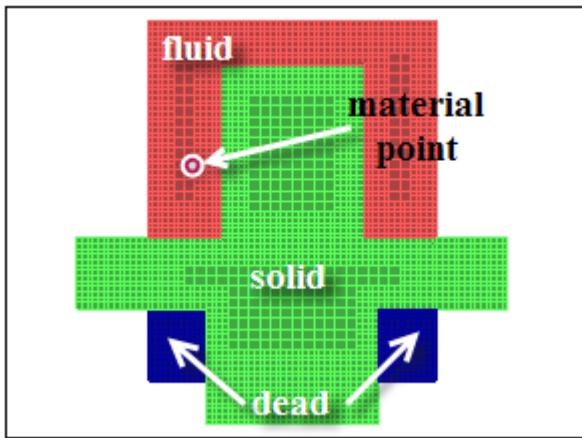
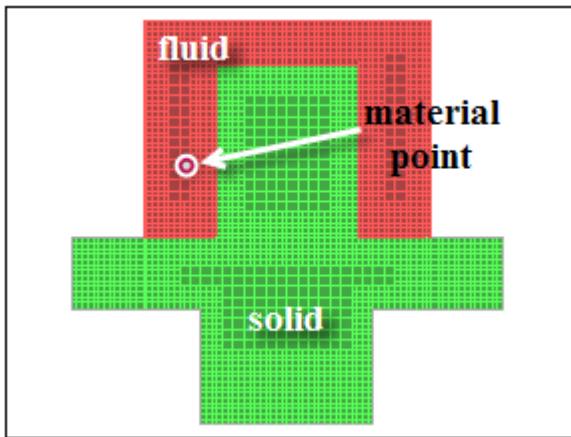
The **create-all-intrst-loops** command can be used to recover the intersecting features accurately.

- In [Figure 8.7: Example—CutCell Mesh, Only Objects Defined \(p. 115\)](#), the use of only objects to define the meshed domain results in a mesh with two cell zones, **solid** and **fluid**.

Figure 8.7: Example—CutCell Mesh, Only Objects Defined

By specifying a material point in addition to the object definition (fluid or dead), the **fluid** zone in [Figure 8.7: Example—CutCell Mesh, Only Objects Defined \(p. 115\)](#) will be further separated into a fluid zone (containing the material point) and a dead zone (see [Figure \(A\) \(p. 115\)](#)). If **auto-delete-dead-zones?** is enabled (default), the separated dead zone(s) will be deleted automatically ([Figure \(B\) \(p. 116\)](#)).

(A) CutCell Mesh—Material Point and Objects Defined, **auto-delete-dead-zones?** Disabled

(B) CutCell Mesh—Material Point and Objects Defined, `auto-delete-dead-zones?` Enabled

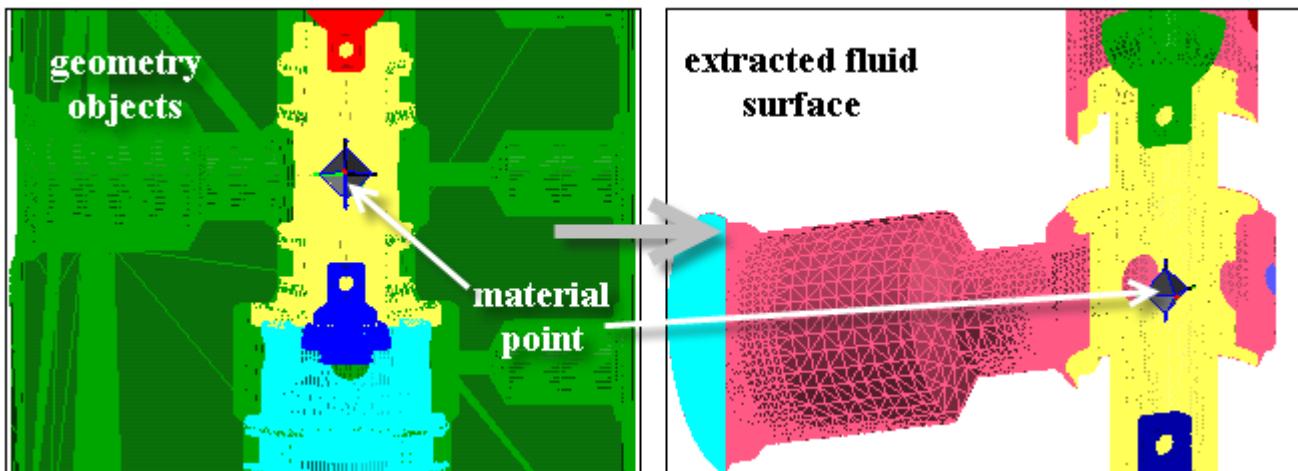
Note

For cases where a single region is separated by a double-sided surface (**fan**, **radiator**, or **porous-jump**), you need to define a material point for each of the regions to be recovered (i.e., both upstream and downstream of the double-sided surface). Separate cell zones will be recovered for each region on either side of the double-sided surface.

You can merge the cell zones manually after the mesh has been generated.

- [Figure 8.8: Example—Fluid Surface Extracted From Geometry Objects and Material Point \(p. 117\)](#) shows the use of a material point in addition to objects defined to extract the internal fluid surface, using the object wrapping operation.

Figure 8.8: Example—Fluid Surface Extracted From Geometry Objects and Material Point



8.2.1. Using the Material Point Dialog Box

The **Material Points** button opens the **Material Points** dialog box, containing options to **Create**, **List**, **Delete** or **Draw** material points.

The **Create** button opens the **Create Material Point** dialog box. Follow the process described.

1. Select the appropriate zone(s) in the graphics window. The zone(s) selected should be such that the material point created will lie at a central point in the fluid domain.
2. Click **Compute** to obtain the material point coordinates based on the zone(s) selected.
You can also specify the coordinates manually if the material point location is known.
3. Enable **Preview** to verify that the location is appropriate.
4. Enter an appropriate fluid zone name in the **Name** field.
5. Click **Create**.

8.2.2. Using Text Commands

The command `/material-point/create-material-point` allows you to define the material point. Specify the fluid zone name and the material point location.

Other commands related to material points are as follows:

`/material-point/create-material-point`

allows you to define a material point. Specify the fluid zone name and the location to define the material point.

`/material-point/delete-all-material-points`

allows you to delete all defined material points.

`/material-point/delete-material-point`

deletes the specified material point.

/material-point/list-material-points

lists all the defined material points.

/mesh/cutcell/set/create-material-point

allows you to define a material point. Specify the fluid zone name and the location to define the material point.

/mesh/cutcell/set/delete-all-material-points

allows you to delete all defined material points.

/mesh/cutcell/set/delete-material-point

deletes the specified material point.

/mesh/cutcell/set/list-material-points

lists all the defined material points.

Chapter 9: Manipulating the Boundary Mesh

The first step in producing an unstructured grid is to define the shape of the domain boundaries. You can create a boundary mesh in which the boundaries are defined by triangular or quadrilateral facets using a preprocessor (GAMBIT or a third-party CAD package) and then create a mesh in the meshing mode in ANSYS Fluent. You can also modify the boundary mesh to improve its quality and create surface meshes on certain primitive shapes. The following sections discuss mesh quality requirements and various techniques for generating an adequate boundary mesh for numerical analysis.

- 9.1. Manipulating Boundary Nodes
- 9.2. Intersecting Boundary Zones
- 9.3. Modifying the Boundary Mesh
- 9.4. Improving Boundary Surfaces
- 9.5. Refining the Boundary Mesh
- 9.6. Creating and Modifying Features
- 9.7. Remeshing Boundary Zones
- 9.8. Faceted Stitching of Boundary Zones
- 9.9. Triangulating Boundary Zones
- 9.10. Separating Boundary Zones
- 9.11. Projecting Boundary Zones
- 9.12. Creating Groups
- 9.13. Manipulating Boundary Zones
- 9.14. Manipulating Boundary Conditions
- 9.15. Creating Surfaces
- 9.16. Additional Boundary Mesh Text Commands

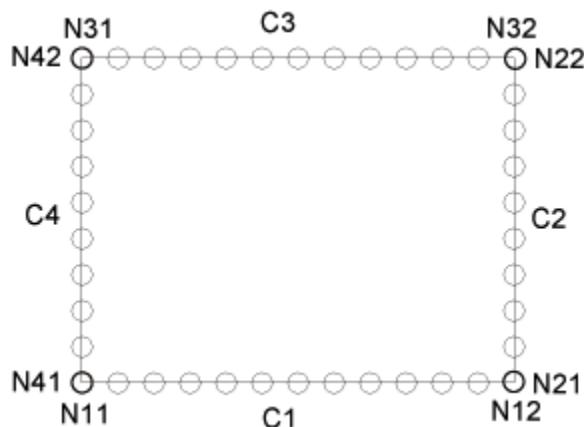
9.1. Manipulating Boundary Nodes

Manipulation of boundary nodes is an effective way to influence the boundary mesh quality. Operations for deleting unwanted boundary nodes can be performed in the **Merge Boundary Nodes** dialog box or with the associated text commands.

- 9.1.1. Free and Isolated Nodes
- 9.1.2. Text Commands for Manipulating Boundary Nodes

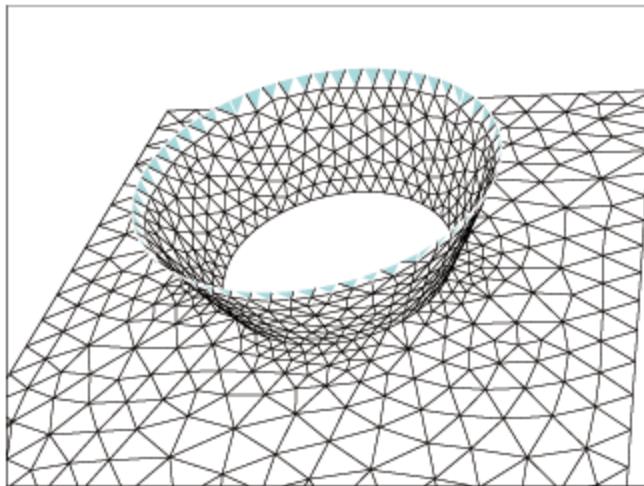
9.1.1. Free and Isolated Nodes

The mesh generation algorithm does not permit duplicate nodes, i.e., two nodes with the same Cartesian coordinates. Duplicate nodes may be created by grid generators that preserve the node locations at adjoining edges of adjacent surfaces, but give different labels to the two sets of nodes. The nodes and edges at which these surfaces meet are termed free nodes and free edges.

Figure 9.1: Free Nodes

[Figure 9.1: Free Nodes \(p. 120\)](#) shows a simple geometry in which the free nodes are marked. Although the node at the end of curve C1 (N12) is located in the same position as the node at the beginning of curve C2 (N21), each is a free node because it is not connected in any way to the adjoining curve.

Though the nodes have the same location, the mesher only knows that they have different names, and not that the curves meet at this location. Similarly, a free edge is a surface edge that is used by only one boundary face. To check the location of free nodes, see [Display Grid Dialog Box \(p. 596\)](#). Free edges are acceptable when modeling a zero-thickness wall ("thin wall") in the geometry (e.g., [Figure 9.2: Example of a Thin Wall \(p. 120\)](#)). Isolated nodes are nodes that are not used by any boundary faces. You can either retain these nodes to influence the generation of the interior mesh (see [Inserting Isolated Nodes into a Tet Mesh \(p. 217\)](#)), or delete them.

Figure 9.2: Example of a Thin Wall

9.1.2. Text Commands for Manipulating Boundary Nodes

The commands performing the same functions as the controls in the **Boundary Nodes** dialog box are:

/boundary/count-unused-bound-node
counts unused boundary nodes in the domain.

/boundary/count-unused-nodes

counts unused boundary and interior nodes in the domain.

/boundary/delete-unused-nodes

deletes boundary nodes that are not used by any boundary faces.

/boundary/count-free-nodes

reports the number of boundary nodes associated with edges having only one attached face.

/boundary/merge-nodes

merges duplicate nodes. If two nodes of a face are merged, the face is deleted.

Note

This operation will be carried out only when valid boundary zones are specified.

9.2. Intersecting Boundary Zones

You can connect triangular boundary zones in the geometry using the set of intersection commands available. These commands can be used to resolve intersections, overlaps, and for connecting zones along the free boundaries.

9.2.1. Intersecting Zones

9.2.2. Joining Zones

9.2.3. Stitching Zones

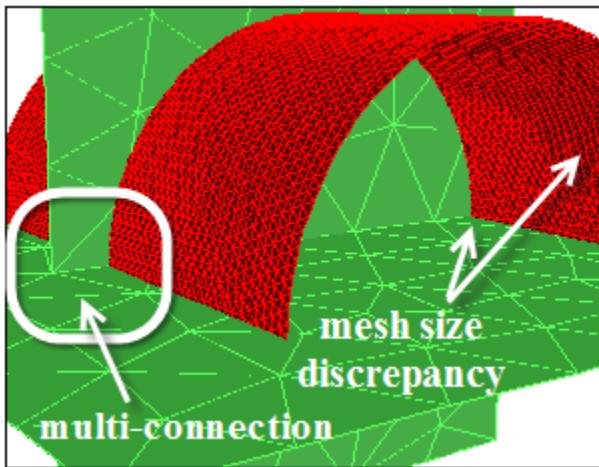
9.2.4. Using the Intersect Boundary Zones Dialog Box

9.2.5. Text Commands for Boundary Intersection

9.2.1. Intersecting Zones

The intersect option is used to connect intersecting tri boundary zones. [Figure 9.3: Intersection of Boundary Zones \(p. 121\)](#) shows an example where the intersect option can be used. The connection is made along the curve (or line) of intersection of the boundary zones. You can use the intersection operation on multi-connected faces as well as in regions of mesh size discrepancy.

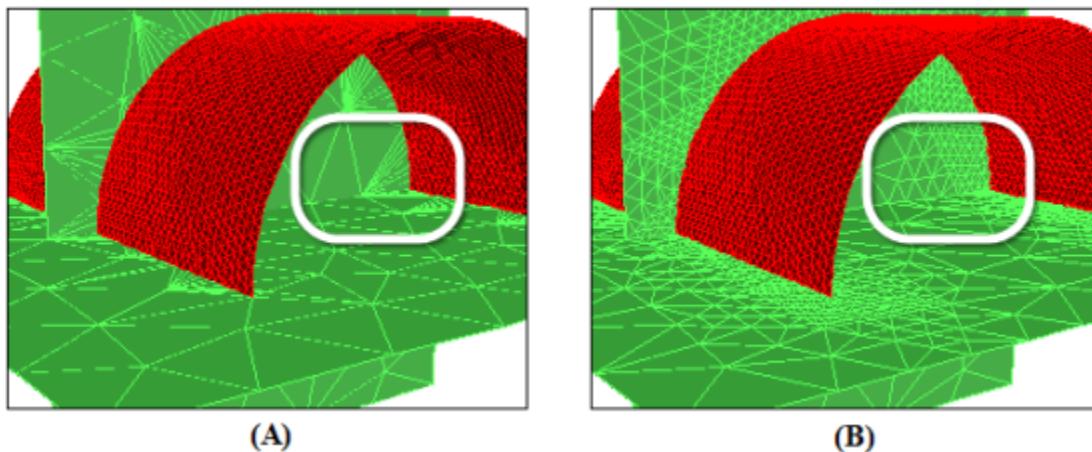
Figure 9.3: Intersection of Boundary Zones



You can intersect boundary zones having a gap between them by specifying an appropriate **Tolerance** value. All zones with the distance between them less the specified tolerance value will be intersected.

The tolerance can be either relative or absolute. When intersecting zones having different mesh size you can enable the **Refine** option to obtain a better graded mesh around the intersecting faces (see [Figure 9.4: Intersection \(A\) Without and \(B\) With the Refine Option \(p. 122\)](#)).

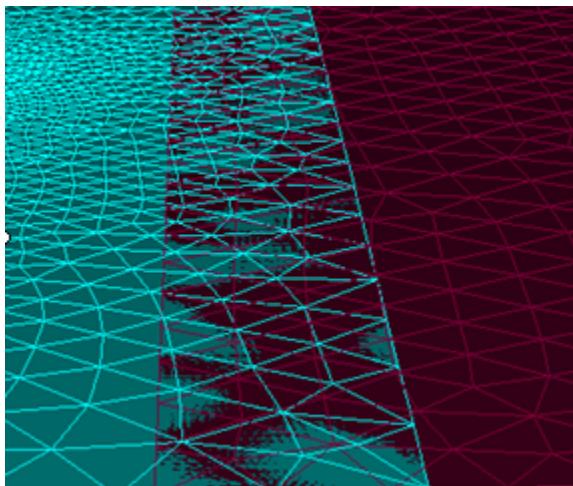
Figure 9.4: Intersection (A) Without and (B) With the Refine Option



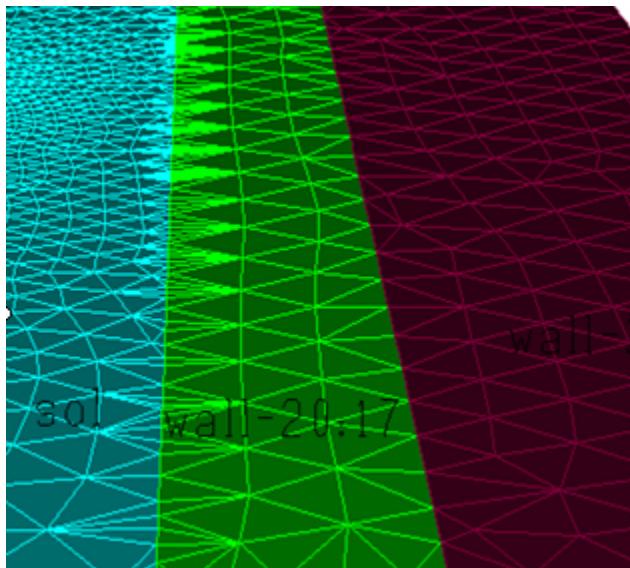
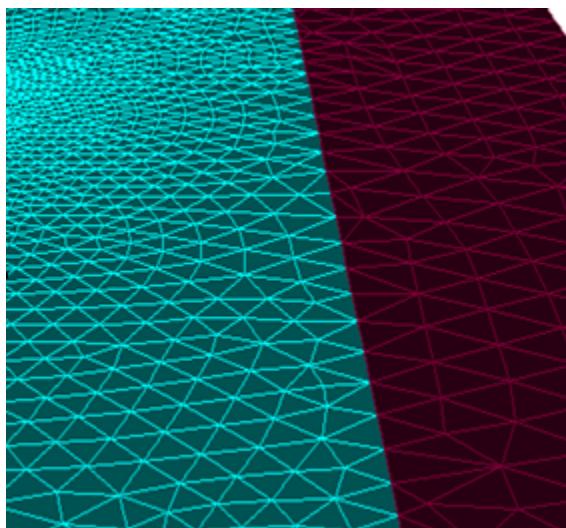
9.2.2. Joining Zones

The join option is used to connect two overlapping tri boundary zones ([Figure 9.5: Partially Overlapping Faces \(p. 122\)](#)). The overlapping areas of both the boundary zones are merged and the mesh at the boundary of the region of overlap is made conformal. To join surfaces that are on top of each other but not connected (with a small gap) specify an appropriate **Tolerance** value. The portion of the surfaces within the tolerance value will be joined. The boundary zone selected in the **Intersect Tri Zone** defines the shape of the combined surface in the overlap region. The shape in the **With Tri Zone** may be changed to perform the join operation.

Figure 9.5: Partially Overlapping Faces



[Figure 9.6: Joining of Overlapping Faces \(p. 123\)](#) and [Figure 9.7: Remeshing of Joined Faces \(p. 123\)](#) show the overlapped faces after joining and after remeshing the joined faces.

Figure 9.6: Joining of Overlapping Faces**Figure 9.7: Remeshing of Joined Faces**

9.2.3. Stitching Zones

The stitch option is used to connect two tri boundary zones along their free edges. You cannot use this option to connect the surfaces at a location other than the free edges in the mesh. Gaps within the given tolerance are closed using closest point projection.

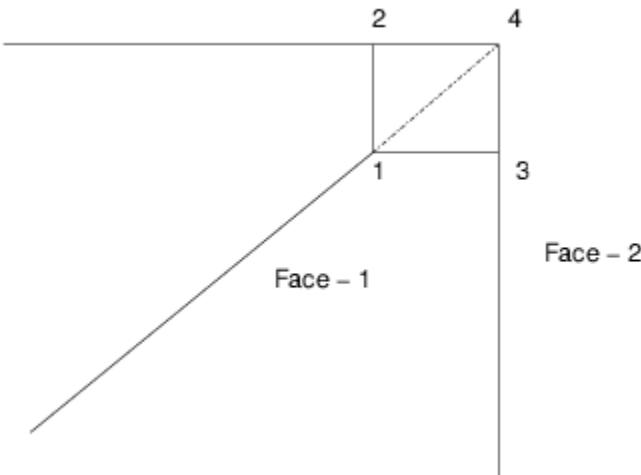
Figure 9.8: Nearest Point Projection for Stitching

Figure 9.8: Nearest Point Projection for Stitching (p. 124) shows a cut through the two surfaces Face-1 and Face-2 which are separated by a gap. The points of nearest projection will determine the location of the intersection curve. Therefore, point-1 will be connected to point-2 or point-3. All the three connect operations allow a small gap (within the tolerance specified) between the intersecting boundary zones. However, the gap should not distort the shape of the geometry.

Figure 9.9: Surfaces Before Stitch (p. 124) and Figure 9.10: Surfaces After Stitch (p. 125) show the surfaces before and after the stitch operation, respectively

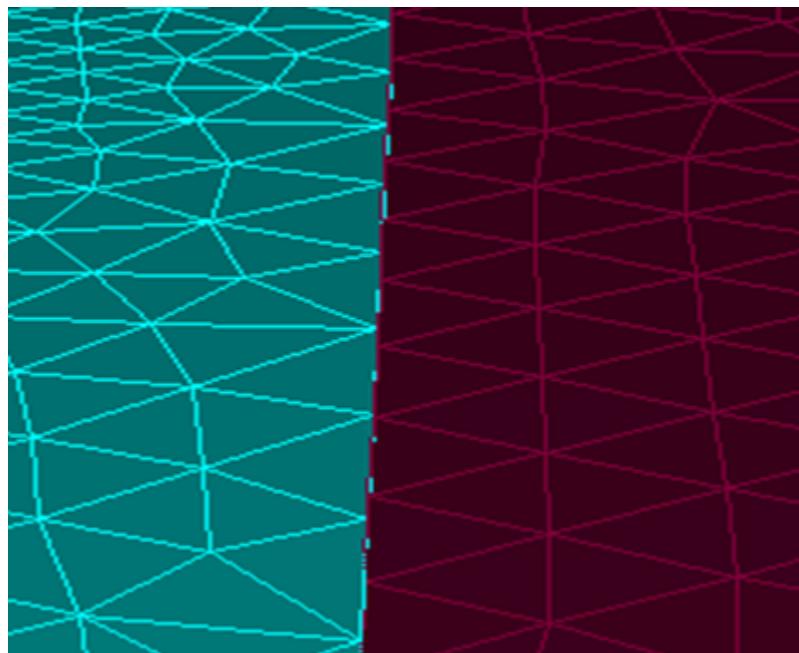
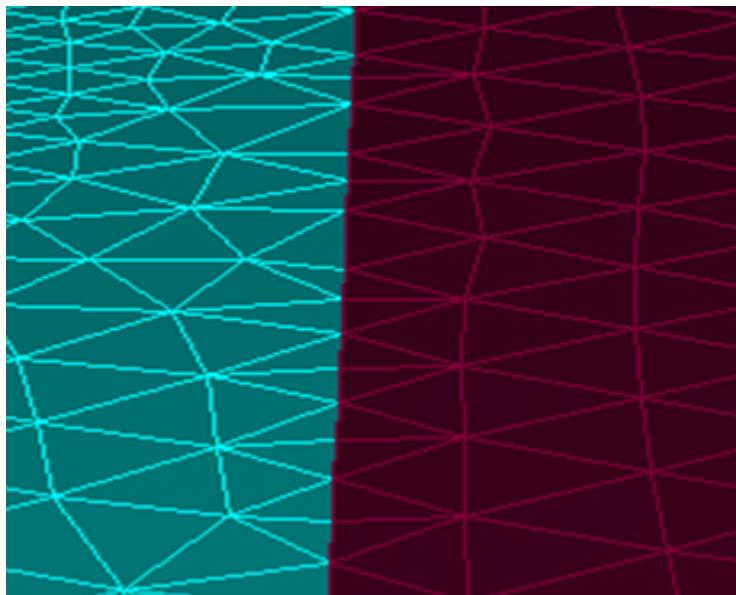
Figure 9.9: Surfaces Before Stitch

Figure 9.10: Surfaces After Stitch

9.2.4. Using the Intersect Boundary Zones Dialog Box

In general, all the three connect operations calculate the intersection curve (or line) between the two surfaces to be connected. The intersection curve is constructed as follows:

- **Intersect** constructs the curve as the intersection of two zones.
- **Join** constructs the curve as the outer boundary of the overlapping region within the specified tolerance of the two surfaces.
- **Stitch** constructs the curve along the free boundaries and within the specified tolerance.

The intersection curve is remeshed with a local spacing calculated from the intersecting surfaces. The intersection curve is inserted into the surfaces and will result in a retriangulation of the surfaces along the intersection curve.

The `/boundary/remesh/remesh-overlapping-zones` command extracts the boundary loops from the zone to imprint. The intersecting curve is inserted into the zones. During the insertion, the zones are retriangulated.

To perform any of the intersection operations, do the following:

1. Select the boundary zone(s) you want to intersect in the **Intersect Tri Zone** list.
2. Select the boundary zone(s) with which you want to intersect the selected boundary zone in the **With Tri Zone** list.
3. Select the appropriate operation from the **Operation** list.
4. Specify the appropriate **Tolerance** value (if the surfaces have a gap between them).
5. Enable **Absolute Tolerance**, **Refine**, or **Separate** as appropriate.
6. Click **Mark**.

The faces that will be affected by the intersection operation are highlighted. This also helps you decide whether the specified tolerance is appropriate.

7. Click **Apply**.

9.2.5. Text Commands for Boundary Intersection

The text interface commands for intersecting boundary zones are listed below:

/boundary/remesh/clear-marked-faces

clears the highlights for the marked faces.

/boundary/remesh/controls/intersect/absolute-tolerance?

allows you to switch between the use of absolute and relative tolerance. The relative tolerance is used by default.

/boundary/remesh/controls/intersect/delete-overlap?

enables the automatic deleting of the region of overlap of the two surfaces. This option is used by while remeshing overlapping zones and retriangulating prisms. This option is enabled by default.

/boundary/remesh/controls/intersect/feature-angle

allows you to specify the minimum feature angle to be considered while retriangulating the boundary zones.

/boundary/remesh/controls/intersect/join-match-angle

specifies the allowed maximum angle between the normals of the two overlapping surfaces to be joined. This parameter is used to control the size of the join region.

/boundary/remesh/controls/intersect/join-project-angle

specifies the allowed maximum angle between the face normal and the project direction for the overlapping surfaces to be joined. This parameter is used to control the size of the join region.

/boundary/remesh/controls/intersect/refine-region?

enables the refining of the intersecting regions after performing the intersection operation. This option is disabled by default.

/boundary/remesh/controls/intersect/retri-improve?

allows you to improve the mesh. After performing any intersection operation, the slivers are removed along the curve of intersection, Laplace smoothing is performed, and followed by the edge swapping. Laplace smoothing is also performed for insert-edge-zone, remesh-overlapped-zones, and prism-retriangulation options. Smoothing is performed again. The smooth-swap operations can be controlled by changing the various defaults such as swapping iterations, smoothing iterations, etc.

/boundary/remesh/controls/intersect/separate?

enables the automatic separation of intersected zones.

/boundary/remesh/controls/intersect/stitch-preserve?

indicates that the geometry and location of the intersect zone (the zone in the **Intersect Tri Zone** list) is to be preserved. This option is enabled by default.

/boundary/remesh/controls/intersect/tolerance

allows you to specify the tolerance value.

/boundary/remesh/controls/intersect/within-tolerance?

performs the intersection operation only within the specified tolerance value. This is relevant only for the **Intersect** option.

/boundary/remesh/create-all-intrst-loops

creates the edge loop of intersection for all boundary zones in the current domain.

/boundary/remesh/create-edge-loops

allows you to create edge loops for the selected face zone.

/boundary/remesh/create-intersect-loop

creates the edge loop along the line (or curve) of intersection.

/boundary/remesh/create-join-loop

creates edge loop on the boundary of the region of overlap of two surfaces.

/boundary/remesh/create-stitch-loop

creates edge loops for connecting two surfaces along their free edges.

/boundary/remesh/delete-overlapped-edges

deletes overlapped edges created during remeshing.

/boundary/remesh/insert-edge-zone

allows you to insert an edge zone into a triangulated boundary face zone.

/boundary/remesh/intersect-all-face-zones

allows you to intersect all the face zones.

/boundary/remesh/intersect-face-zones

allows you to intersect the specified face zones.

/boundary/remesh/join-all-face-zones

allows you to join all the face zones.

/boundary/remesh/join-face-zones

allows you to join the specified face zones.

/boundary/remesh/mark-intersecting-faces

highlights the tri faces in the neighborhood of the line of intersection.

/boundary/remesh/mark-join-faces

highlights the tri faces in the neighborhood of the join edge loop.

/boundary/remesh/mark-stitch-faces

highlights the tri faces in the neighborhood of the stitch edge loop.

/boundary/remesh/remesh-overlapping-zones

allows you to remesh overlapping face zones.

/boundary/remesh/stitch-all-face-zones

allows you to stitch all the face zones.

/boundary/remesh/stitch-face-zones

allows you to connect two surfaces along their free edges.

9.3. Modifying the Boundary Mesh

Tools are available for making boundary repairs, enabling you to perform primitive operations on the boundary mesh, such as creating and deleting nodes and faces, moving nodes, swapping edges, merging and smoothing nodes, collapsing nodes, edge(s), and face(s), splitting faces, and moving faces to another boundary zone.

- 9.3.1. Using the Modify Boundary Dialog Box
- 9.3.2. Operations Performed: Modify Boundary Dialog Box
- 9.3.3. Locally Remeshing a Boundary Zone or Faces
- 9.3.4. Text Commands for Boundary Modification

9.3.1. Using the Modify Boundary Dialog Box

This section describes the generic procedure for modifying the boundary mesh using the **Modify Boundary** dialog box. In addition to the **Modify Boundary** dialog box, you may also use the **Display Grid** dialog box during the modification process.

1. Display the boundary zone(s) that you want to modify, using the **Display Grid** dialog box. If you need to modify many zones, display them one at a time to make the graphics display less cluttered.
2. Select the type of entity you want to select with the mouse: **edge**, **node**, **position**, etc. in the **Filter** list in the **Modify Boundary** dialog box.
3. Select the entities you want to operate on using the mouse-probe button (the right-button, by default) in the graphics window.

You can select individual entities one at a time, or select a group of them by defining a selection region. See [Controlling the Mouse Probe Functions \(p. 393\)](#) for details. The selected entities will appear in the **Selections** list in the **Modify Boundary** dialog box.

4. Click the appropriate **Operation** button to perform the boundary modification.

The mesh is automatically re-displayed after the operation is performed, allowing you to immediately see the effect of your change.

5. Repeat the process to perform different operations on different entities.

Warning

Save the mesh periodically as it is not always possible to undo an operation.

9.3.2. Operations Performed: Modify Boundary Dialog Box

You can perform the following operations using the **Modify Boundary** dialog box:

Creating Nodes

To create nodes, do the following:

1. Select the required position(s) (or enter node coordinates explicitly in the **Enter Selection** box).
2. Select **node** in the **Filter** list or use the hot-key **Ctrl+N**.

3. Click the **Create** button or press **F5** (on the keyboard).

Creating Faces

To create a face, do the following:

1. Select 3 or 4 nodes and the optional zone.

Use the hot keys **Ctrl+N** and **Ctrl+F** to select node and face as **Filter**, respectively.

2. Click **Create** or press **F5**.

While creating a face:

- If you do not select a zone, the new face will be in the same zone as an existing face that uses one of the specified nodes.
- If the nodes you use to create a face are used by faces in different zones, make sure that the new face is in the right zone.
- If you create a face and it is in the wrong zone, use the rezoning feature.

Creating a Zone

To create a new zone, do the following:

1. Select **zone** in the **Filter** list (hot-key **Ctrl+Z**).
2. Click **Create** or press **F5**. The **Create Boundary Zone** dialog box will open, prompting you for the zone name and type.
3. Specify the name and zone type as appropriate in the **Create Boundary Zone** dialog box.
4. Click **OK**. The new zone will automatically be added to the **Selections** list in the **Modify Boundary** dialog box.

Deleting a Node/Face/Zone

To delete the node(s) or face(s), do the following:

1. Select the node(s) or face(s) or zone(s) to be deleted.
2. Click **Delete** or press **Ctrl+W** on the keyboard.

Merging Nodes

To merge nodes, do the following:

1. Select the two nodes to be merged.

You can merge multiple pairs of nodes by selecting more than two nodes before clicking **Merge** (or pressing **F9**). The first and second nodes will be merged, then the third and fourth, and so on.

Note

To merge multiple pairs, select an even number of nodes and ensure that you select them in the correct order.

2. Click **Merge** or press **F9**. The first node selected is retained, and the second node is merged onto the first node.

Warning

Save the boundary mesh before merging nodes because merging is not reversible (i.e., clicking **Undo** will not undo a merge operation).

Moving Nodes

To move the node to any position in the domain, do the following:

1. Select **node** in the filter list (hot key **Ctrl+N**).
2. Select the node you want to move.
3. Choose **position** in the filter list (hot key **Ctrl+X**).
4. Select the position coordinates or click on the position in the graphics window to which you want to move the selected node.
5. Click **Move To**.

To move the node by specifying the magnitude of the movement, do the following:

1. Select **node** in the filter list (hot key **Ctrl+N**).
2. Select the node you want to move.
3. Enter the magnitude by which you want to move the selected node.
4. Click **Move By**.

Rezoning Faces

To rezone one or more faces, do the following:

1. Select the face(s) you want to move.
2. Select the zone to which you want the selected faces to move.
3. Click **Rezone** (hot-key **Ctrl+O**). You can create a zone if you need to move faces to a new zone.

Collapsing Nodes/Edges/Faces

To collapse nodes, edges, or faces, do the following:

1. Select the appropriate **Filter**.
2. Select the two nodes (or edge(s)/face(s)) you want to collapse.
3. Click **Collapse** (hot-key **Ctrl+^**).

While collapsing:

- If a pair of nodes is selected, both the nodes are moved towards each other (at the midpoint) and collapsed into a single node.
- If an edge is selected, the two nodes of the edge collapse onto the midpoint of the edge and surrounding nodes are connected to the newly created node.
- If a triangular face is selected, a new node is created at the centroid of the triangle and the selected triangular face gets deleted.

Note

You can also collapse multiple pairs of entities by selecting multiple entities before clicking **Collapse**. Ensure that an even number of entities is selected. The first and the second entity will be collapsed, then the third and the fourth, and so on.

Important

Save the boundary mesh before performing the collapse operation because collapsing is not reversible (i.e., the **Undo** button will not undo a collapse operation).

Smoothing Nodes

To smooth nodes, do the following:

1. Select the node(s) you want to smooth.
2. Click **Smooth** or press the **F6** key on the keyboard.

The node will be placed at a position computed from the average of the surrounding nodes.

Splitting Edges

To split edges, do the following:

1. Select the edge(s) you want to split.
2. Click **Split** or press the **F7** key on the keyboard.

All faces sharing the edge will be split into two faces.

If you select multiple edges and they share a face, the split operation may not be completed. If the face referenced by the split operation for the second edge has already been split by the operation on the first edge, the second split operation will not be possible because the referenced face that no longer exists. If this happens, redisplay the mesh and reselect the edge that was not split. In such cases it may also be easier to split the face rather than the edge.

Splitting Faces

To split faces, do the following:

1. Select the face(s) you want to split.
2. Click **Split** or press the **F7** key.

Each triangular face will be split into three faces by adding a node at the centroid. Each quadrilateral face will be split into two triangular faces.

Perform edge swapping after this step to improve the quality of the local refinement.

Swapping Edges

To swap an edge of a triangular face, do the following:

1. Select the edge(s) as appropriate.
2. Click **Swap** or press the **F8** key on the keyboard. If the triangular boundary face on which you perform edge swapping is the cap face of a prism layer, the swapping will automatically propagate through the prism layers, as described in [Edge Swapping and Smoothing \(p. 249\)](#).

Note

Edge swapping is not available for quadrilateral faces.

Finding Coordinates of the Centroid

To find the location of the centroid of a face or cell, do the following:

1. Set **Filter** to **face** or **cell** as appropriate.
2. Select the face or the cell using the mouse probe button.
3. Click the **Centroid** button (hot-key **Ctrl+L**).

The face or cell centroid location will be printed in the console window.

Calculating Distance Between Entities

To compute the distance between two entities, do the following:

1. Set **Filter** to **face**, **edge**, or **cell** as appropriate.
2. Select the two entities.

3. Click **Distance** (hot-key **Ctrl+D**).

For example, if an edge (or face or cell) and a node are selected, the distance between the centroid of the edge (or face or cell) and the node is computed and printed to the console window.

Projecting Nodes

To reconstruct features in the surface mesh that were not captured in the surface mesh generation, project selected nodes onto a specified line or plane.

The **Create Boundary Zone** dialog box will appear automatically when you create a new face zone (see [Modify Boundary Dialog Box \(p. 486\)](#)). You can specify the name and type of the new zone in this dialog box.

To project nodes, do the following:

1. Define the projection line or plane. For a projection line, select two entities and for a projection plane, select three entities. If edges, faces, or cells are selected, their centroidal locations will be used.
2. Click **Set** (hot-key **Ctrl+S**) and the projection line or plane will be shown in the graphics display.
3. Select the nodes to be projected.
4. Click **Project** (hot-key **Ctrl+P**).

The selected nodes will be projected onto the projection line or plane that you defined with the **Set** button.

Simplifying Boundary Modification

The following functions simplify the boundary modification process:

Finding the Worst/Marked Faces

You can display faces in the descending order of their quality as follows:

1. To find the face having the worst quality in the grid, select **Quality Limit** and click **First** or press the **F11** key on the keyboard.

The worst face will be displayed in the graphics window and its quality and zone ID are reported in the console. The **Next** button will replace the **First** button.

- The longest edge of the face and the node opposite it are selected, and the display is limited to the neighborhood of the highly skewed face.
 - If the grid has not been displayed, the worst face, its quality, and the zone in which it lies will be reported (in the console).
2. Click **Next** (hot-key right-arrow key).
- The face having the next highest quality will be displayed in the graphics window. When you subsequently click **Next**, the face having the next highest quality (after that of the previously displayed or reported face) will be displayed or reported.
3. Click **Reset** (hot-key left-arrow button) to reset the display to the worst quality element.

You can also find the worst face within a subset of zones by activating a group containing the required zones (using the **User Defined Groups** dialog box and then clicking **First**). When you click the **Next** button after activating a particular group, the face having the next highest quality within the active group will be displayed.

To display the marked faces in succession, do the following:

1. Select **Mark** and click **First** (hot-key **F11**) to find the first marked face.

The face will be displayed in the graphics window. The **Next** button will replace the **First** button.

2. Click **Next** (hot-key right-arrow key).

The next marked face will be displayed in the graphics window. When you subsequently click **Next**, the next face will be displayed or reported.

3. Click **Reset** (hot-key left-arrow button) to reset the display to the first marked face.

You can use the /bounday/unmark-selected-faces command (hot-key **Ctrl+U**) to unmark the faces.

To improve the quality of the face, use the following operations:

- Use the **Smooth** operation to smooth the node opposite the longest face.
- Use the **Merge** operation to collapse the shortest edge of the face, merging the other two edges together. The longer of the remaining two edges is retained, while the shorter one is merged with the other edge.
- Use the **Swap** operation to swap the selected edge.
- Use the **Split** operation to refine the face by bisecting the selected edge.

If the selected entities are not appropriate, clear them, choose the appropriate items, and perform the desired operations.

Deselecting a Selected Entity

If you select an inappropriate entity, you can click on it again in the graphics window to deselect it. You can also select it in the **Selections** list in the **Modify Boundary** dialog box and click **Clear**. You can use the hot-key **F2** to deselect all entities selected.

Warning

Deselect operations are performed only on the items selected in the **Selections** list.

Undoing an Operation

To undo an operation, click the **Undo** button or press the **F12** key on the keyboard. In some cases, a particular sequence of operations cannot be undone. Hence, make sure that you save the mesh periodically between the modifications.

Click **Undo** or press **F12** *n* times to undo the last *n* operations.

Warning

The **Undo** operation is limited to the operations in the **Modify Boundary** dialog box (or the /boundary/modify menu). If other operations/commands are interleaved, the **Undo** operation may cause unexpected results.

Note

You can also use the boundary modification operations to fix holes in the geometry. Refer to [Detecting and Filling the Holes Manually \(p. 179\)](#) for details.

9.3.3. Locally Remeshing a Boundary Zone or Faces

To remesh a single zone, select a zone in the graphics window and use the hot-key **Ctrl+Shift+r**.

The [Local Remesh Dialog Box \(p. 642\)](#) contains options for remeshing faces based on selections in the graphics window. Select the faces to be remeshed and use the hot-key **Ctrl+Shift+r** to open the **Local Remesh** dialog box. Specify the sizing source (**geometric** or **size-function**), the number of radial layers of faces to be remeshed, and the feature angle to be preserved while remeshing the selected faces. You can alternatively use the command /boundary/modify/local-remesh after selecting the faces in the graphics window.

9.3.4. Text Commands for Boundary Modification

The tools for modifying the boundary work in conjunction with the mouse. To use them,

1. Select the appropriate entity in the graphics window using the mouse or the selection commands.
2. Enter the command in the text window.
3. First set the selection probe to select.
Each entity that you pick is stored in a list.
4. Monitor this list of selected items using list-selections.

The list can be modified using clear-selections and deselect-last commands. After you perform an operation on the selected items, the selection list is cleared.

These text commands perform the same functions as the **Modify Boundary** dialog box (see [Using the Modify Boundary Dialog Box \(p. 128\)](#)). The following operations are available:

/boundary/modify/analyze-bnd-connectvty

finds and marks free edges and nodes and multiply-connected edges and nodes. This process is necessary if the boundary mesh has been changed with Scheme functions.

/boundary/modify/clear-selections

clears all selections.

/boundary/modify/clear-skew-faces

clears faces that were marked using the boundary/modify/mark-skew-face command.

/boundary/modify/collapse

collapses pairs of nodes, edge(s), or face(s). If a pair of nodes is selected, both the nodes are deleted and a new node is created at the midpoint of the two nodes. If a triangular face is selected, the complete face is collapsed into a single node at the centroid of the face.

/boundary/modify/create

creates a boundary face if the selection list contains 3 nodes and an optional zone. If the selection list contains positions, then nodes are created.

/boundary/modify/create-mid-node

creates a node at the midpoint between two selected nodes.

/boundary/modify/delete

deletes all selected faces and nodes.

/boundary/modify/delta-move

allows you to move the selected node by specified magnitude.

/boundary/modify/deselect-last

removes the last selection from the selection list.

/boundary/modify/hole-feature-angle

allows you to specify the feature angle for consideration of holes in the geometry.

/boundary/modify/list-selections

lists all the selected entities.

/boundary/modify/local-remesh

allows you to remesh faces based on selections in the graphics window. Select the faces to be remeshed and specify the sizing source (geometry or size-function), the number of radial layers of faces to be remeshed, and the feature angle to be preserved while remeshing the selected faces.

/boundary/modify/mark-skew-face

marks faces that should be skipped when the worst skewed face is reported using the **Modify Boundary** dialog box. This allows you to search for the next skewed face.

/boundary/modify/merge

merges pairs of nodes. The first node selected is retained, and the second is the duplicate that is merged.

/boundary/modify/move

moves the selected node to the specified position.

/boundary/modify/next-skew

finds the triangular face having the next highest skewness value (after that of the worst skewed face) in the mesh (or the active group). The face ID, its skewness, the ID of the longest edge, and the ID of the node opposite the longest edge are displayed in the console.

/boundary/modify/repair

repairs the selected zone(s) by filling all holes associated with free edges.

/boundary/modify/rezone

moves the selected faces from their current zone into the selected zone, if the selection list contains a zone and one or more faces.

/boundary/modify/select-filter

selects a filter. The possible filters are off, cell, face, edge, node, zone, position, object, and size. If off is chosen then when a selection is made it is first checked to see if it is a cell, then a face, an edge, and so on. When the node filter is used, if a cell or face is selected the node closest to the selection point is picked. Thus nodes do not have to be displayed to be picked.

/boundary/modify/select-probe

selects the probe function. The possible functions are off, label, select, and print. When the function is off, mouse probes are disabled. label prints the selection label in the graphics window, select adds the selection to the selection list, and print prints the information on the selection in the console window.

/boundary/modify/select-position

allows you to add a position to the selection list by entering the coordinates of the position.

/boundary/modify/select-entity

allows you to add a cell, face, or node to the selection list by entering the name of the simplex.

/boundary/modify/select-zone

allows you to add a zone to the selection list by entering the zone name or ID.

/boundary/modify/show-filter

shows the current filter.

/boundary/modify/show-probe

shows the current probe function.

/boundary/modify/skew

finds the face with the highest (worst) skewness, selects it in the graphics window, and reports its skewness and zone ID in the console window

/boundary/modify/skew-report-zone

allows you to select the zone for which you want to report the skewness. You can either specify the zone name or zone ID.

/boundary/modify/smooth

uses Laplace smoothing to modify the position of the nodes in the selection list. The new position is an average of the neighboring node coordinates and is not reprojected to the discrete surface.

/boundary/modify/split-face

splits the selected face into three faces.

/boundary/modify/swap

swaps boundary edges (of triangular faces) if the selection list contains edges.

/boundary/modify/undo

undoes the previous operation. When an operation is performed, the reverse operation is stored on the undo stack. For example, a create operation places a delete on the stack, and a delete adds a create operation.

The merge and collapse options cannot be undone. Theoretically if no merge or collapse operations are performed, you could undo all previous operations. In reality, certain sequences of operations are not reversible.

Note

The undo operation is limited to the operations in the **Modify Boundary** dialog box or the /boundary/modify menu. If other operations/commands are interleaved, the undo operation may cause unexpected results.

9.4. Improving Boundary Surfaces

The quality of the volume mesh is dependent on the quality of the boundary mesh from which it is generated. You can improve boundary surfaces to improve the overall mesh quality.

You can improve the boundary mesh by specifying an appropriate quality limit depending on the quality measure considered. You can also smooth and swap faces on the boundary surfaces to improve the mesh quality. You can use the **Boundary Improve** dialog box to improve the surfaces. You can diagnostically determine the boundary mesh quality using the **Check** and **Skew** buttons available when the **Swap** option is selected.

- [9.4.1. Improving the Boundary Surface Quality](#)
- [9.4.2. Smoothing the Boundary Surface](#)
- [9.4.3. Swapping Face Edges](#)
- [9.4.4. Text Commands for Improving Boundary Surfaces](#)

9.4.1. Improving the Boundary Surface Quality

You can improve the boundary surface quality using skewness, size change, aspect ratio, or area as the quality measure.

- For improving the boundary surface quality based on skewness, size change, and aspect ratio, specify the quality limit, the angle, and the number of improvement iterations. All the elements above the specified quality limit will be improved.
- For improving based on the area, collapse faces and then either swap the edges or smooth the surface. All faces having area smaller than the specified minimum absolute size will be collapsed.

You can also specify the minimum relative size (size of the neighboring entity) to be considered while using the **Collapse and Swap** option.

9.4.2. Smoothing the Boundary Surface

Smoothing of the surface mesh allows you to control the variation in the size of the mesh elements, thereby improving the accuracy of the numerical analysis. Smoothing is critical in regions of proximity or regions where surfaces intersect and the accuracy of the approximations used in numerical analysis techniques deteriorates with rapid fluctuations in the element size. The smoothing procedure involves relocating of the mesh nodes without changing the mesh topology.

9.4.3. Swapping Face Edges

Edge swapping can be used to improve the triangular surface mesh. The procedure involves checking each pair of faces that shares an edge and identifying the connecting diagonal that results in the most

appropriate configuration of faces within the resulting quadrilateral. For a face considered, if the unshared node on the other face lies within its minimal sphere, the configuration is considered to be a Delaunay violation and the edge is swapped. The procedure makes a single pass through the faces to avoid cyclic swapping of the same set of edges. Thus, the edge swapping process is repeated until no further improvement is possible. At this stage, even if a few Delaunay violations exist, the differences resulting from continual swapping are marginal.

Important

If the triangular boundary zone selected is the cap face zone of a prism layer, the edge swapping will automatically propagate through the prism layers.

9.4.4. Text Commands for Improving Boundary Surfaces

The text commands available for improving boundary surfaces are:

/boundary/improve/collapse-bad-faces

allows you to collapse the short edge of faces having a high aspect ratio or skewness in the specified face zone(s).

/boundary/improve/degree-swap

allows you to improve the boundary mesh by swapping edges based on a node degree value other than 6. The node degree is defined as the number of edges connected to the node.

/boundary/improve/improve

allows you to improve the boundary surface quality using skewness, size change, aspect ratio, or area as the quality measure.

/boundary/improve/smooth

allows you to improve the boundary surface using smoothing.

/boundary/improve/swap

allows you to improve the boundary surface using edge swapping.

9.5. Refining the Boundary Mesh

To use refinement regions for local refinement in some portion of the domain (e.g., to obtain a high mesh resolution in the wake of an automobile), you may refine the associated boundary zones as well. When you perform the local refinement, the boundary faces that border the refinement region will not be refined. It is therefore possible that you will have a jump in face size where a small cell touches a large boundary face. To improve the smoothness of the mesh, use the **Refine Boundary Zones** dialog box to appropriately refine the boundary zones that border the refinement region before performing the refinement of the volume mesh. Boundary refinement can be performed only on triangular boundary zones.

[9.5.1. Procedure for Refining Boundary Zone\(s\)](#)

[9.5.2. Text Commands for Boundary Zone Refinement](#)

9.5.1. Procedure for Refining Boundary Zone(s)

To refine boundary zones based on marked faces, do the following:

1. Open the **Refine Boundary Zones** dialog box.

Boundary → Mesh → Refine...

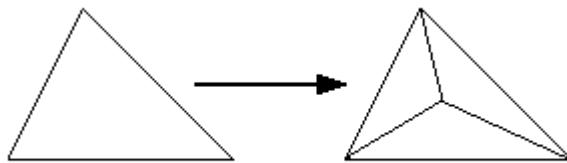
2. Select **Mark** in the **Options** list and define the refinement region. Click the **Local Regions...** button to open the **Boundary Refinement Region** dialog box. Define the refinement region as appropriate.
3. Select the zones to be refined in the **Tri Boundary Zones** list.
4. Select the region to be refined in the **Regions** list. The **Max Face Area** will be updated based on the value specified in the **Boundary Refinement Region** dialog box.
5. Click **Apply** to mark the faces to be refined.

The faces in the selected zones having face area greater than the **Max Face Area** specified will be marked.

6. Select **Refine** in the **Options** list and **Mark** in the **Refinement** group box.
7. Click **Apply**.

The marked faces are refined by dividing them into three faces (see [Figure 9.11: Refining a Triangular Boundary Face \(p. 140\)](#)).

Figure 9.11: Refining a Triangular Boundary Face



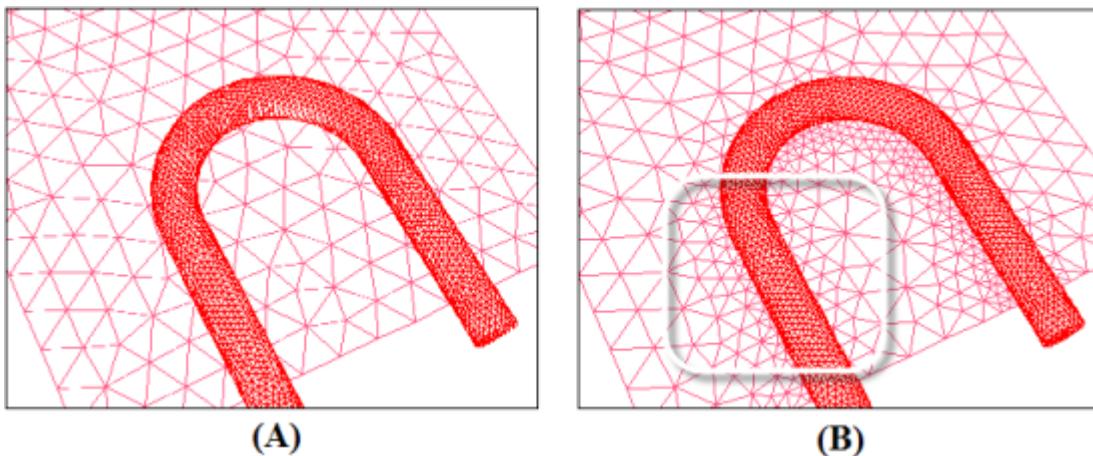
To refine boundary zones based on proximity, do the following:

1. Open the **Refine Boundary Zones** dialog box.

Boundary → Mesh → Refine...

2. Select **Refine** in the **Options** list and **Proximity** in the **Refinement** group box.
3. Select the zone from which the proximity is to be determined in the **Tri Boundary Zones** selection list.
4. Specify the **Relative Distance** and number of refinement iterations as appropriate.
5. Click **Apply**.

The faces in the proximity of the specified zone are refined as shown in [Figure 9.12: Boundary Mesh \(A\) Before and \(B\) After Refining Based on Proximity \(p. 141\)](#).

Figure 9.12: Boundary Mesh (A) Before and (B) After Refining Based on Proximity

To further improve the quality of the refined boundary mesh, do the following:

1. Select **Swap** in the **Options** list and specify the **Max Angle** and **Max Skew** as appropriate (see the description in [Refine Boundary Zones Dialog Box \(p. 513\)](#)).
2. Click **Apply**.
3. If the geometry of the boundary is close to planar, you can improve the mesh quality further by selecting the **Smooth** option, specifying the **Max Angle** and **Relax** parameters, as appropriate (see the description in [Refine Boundary Zones Dialog Box \(p. 513\)](#)), and clicking **Apply**.

Warning

If the geometry is far from planar, smoothing is not recommended, since it may modify the shape of the boundary.

If you wish to repeat the process for another refinement region, first select the **Clear** option and click **Apply** to clear all marks.

9.5.2. Text Commands for Boundary Zone Refinement

The text commands for boundary zone refinement are as follows:

/boundary/refine/auto-refine

automatically refines a face zone based on proximity. The original face zone is treated as a background mesh. Faces are refined by multiple face splitting passes, so that no face is in close proximity to any face in the current domain.

/boundary/refine/clear

clears all refinement marks from all boundary faces.

/boundary/refine/count

counts the number of faces marked on each boundary zone.

/boundary/refine/limits

prints a report of the minimum and maximum size of each specified zone. This report will also tell you how many faces on each zone have been marked for refinement.

/boundary/refine/local-regions/define

defines the refinement region according to the specified parameters.

/boundary/refine/local-regions/delete

deletes the specified region.

/boundary/refine/local-regions/init

creates a region encompassing the entire geometry.

/boundary/refine/local-regions/list-all-regions

lists all the refinement regions in the console.

/boundary/refine/mark

marks the faces for refinement.

/boundary/refine/refine

refines the marked faces.

9.6. Creating and Modifying Features

Geometric features, such as ridges, curves, or corners should be preserved while performing various operations (e.g., smoothing, remeshing) on the boundary mesh. You can create edge loops for a face zone and if required, you can also modify the node distribution on the edge loop. The **Feature Modify** dialog box contains options available for creating and modifying edge loops. You can also draw the edge loops to determine their direction (i.e., determine the start and the end points).

[9.6.1. Creating Edge Loops](#)

[9.6.2. Modifying Edge Loops](#)

[9.6.3. Using the Feature Modify Dialog Box](#)

[9.6.4. Text Commands for Creating and Modifying Features](#)

Important

You can also use the **Surface Retriangulation** dialog box for creating edge loops before remeshing the face zones. The **Surface Retriangulation** dialog box allows you to use the face zone approach only.

9.6.1. Creating Edge Loops

Edge loops can be created according to the specified combination of the edge loop creation approach and the angle criterion.

The angle criteria used for creating edge loops are as follows:

- Fixed angle criterion

This method considers the feature angle between adjacent faces when creating edge loops. You can specify the minimum feature angle between adjacent faces as a parameter for edge loop creation. The common edge thread between two faces will be created when the feature angle is greater than the value specified.

- Adaptive angle criterion

This method compares the angle at the edge with the angle at neighboring edges. If the relation between the angles matches the typical patterns of the angles in the neighborhood of the feature edge, the edge in question is considered to be a feature edge. You do not need to specify a value for the feature angle in this case.

The approaches available for edge loop creation are as follows:

- Face zone approach

The edge thread is created on the entire face zone based on the specified angle criteria. The face zone approach is useful when creating edge threads on common edges where two surfaces of the zone intersect each other. The common edge is considered to be a feature edge when the angle value specified (fixed angle criterion) is less than the feature angle. Alternatively, the edge thread at the common edge can be created by detecting the change in the feature angle automatically (adaptive angle criterion).

- Face seed approach

The edge thread is created surrounding the surface on which the seed face is defined based on the specified angle criteria. The common edge is considered to be a feature edge when the angle value specified (fixed angle criterion) is less than the feature angle. Alternatively, the edge thread at the common edge can be created by detecting the change in the feature angle automatically (adaptive angle criterion).

The **Face Seed** approach is available only when you use the **Feature Modify** dialog box for creating edge loops. If you use the **Surface Retriangulation** dialog box instead, the **Face Zone** approach will be used for creating the edge loops.

Figure 9.13: Surface Mesh—Feature Angle = 60

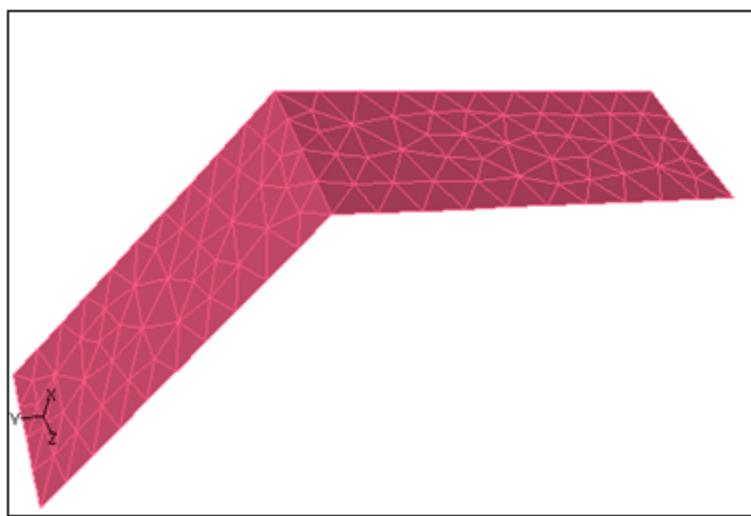
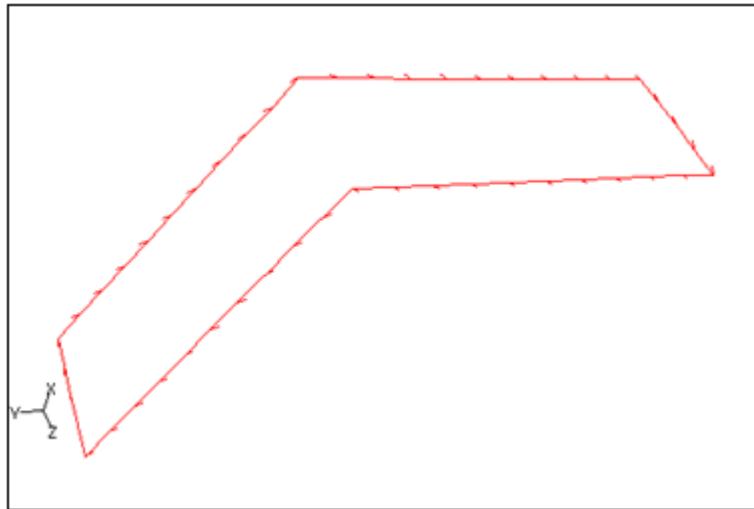


Figure 9.13: Surface Mesh—Feature Angle = 60 (p. 143) shows a surface mesh with two faces connected at a common edge and having a feature angle of 60 degrees. Both faces are in the same face zone.

Figure 9.14: Edge Loop for Face Zone Approach and Fixed Angle = 65 (p. 144)—Figure 9.17: Edge Loops for Face Seed Approach and Fixed Angle = 55 (or Adaptive Angle) (p. 145) show the edge loops created for different combinations of approach and angle criterion.

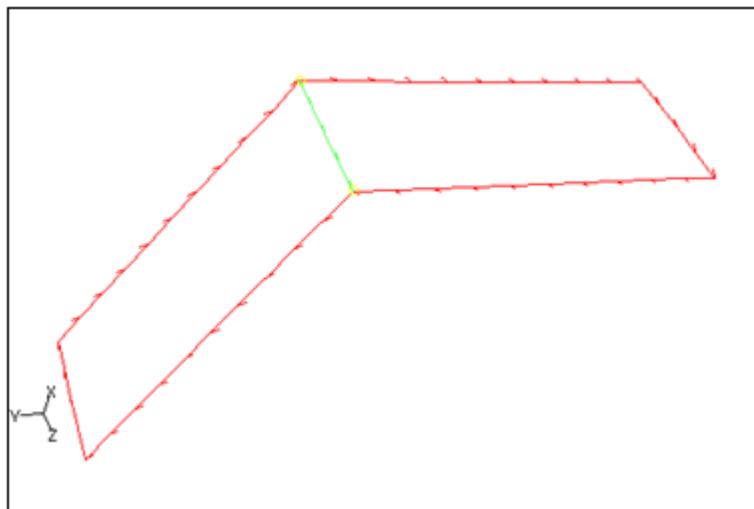
- Figure 9.14: Edge Loop for Face Zone Approach and Fixed Angle = 65 (p. 144) shows the single edge loop created by using the **Face Zone** approach and **Fixed** angle criterion, with the angle specified as 65 degrees. The edge thread at the common edge is not created since the specified value for **Angle** is greater than the feature angle.

Figure 9.14: Edge Loop for Face Zone Approach and Fixed Angle = 65

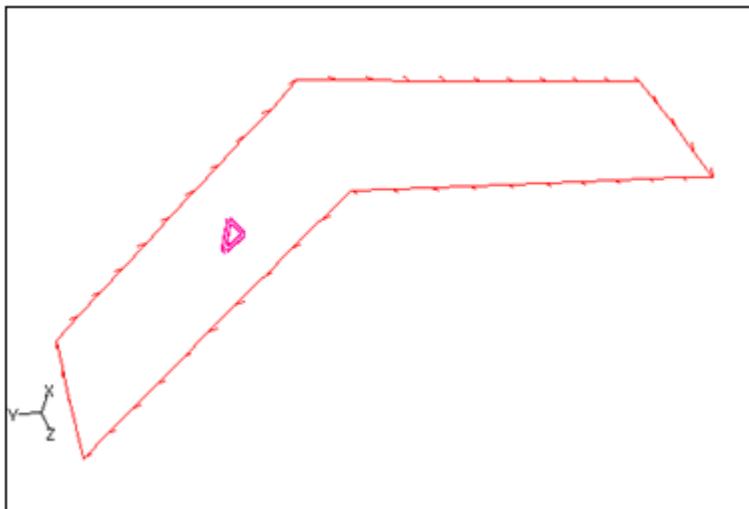


- Figure 9.15: Edge Loops for Face Zone Approach and Fixed Angle = 55 (or Adaptive Angle) (p. 144) shows the edge loops created by using the **Face Zone** approach and **Fixed** angle criterion, with the angle specified as 55 degrees. The interior edge thread at the common edge is created since the specified value for **Angle** is smaller than the feature angle. Alternatively, if you use the **Adaptive** angle criterion, the change in angle will be detected automatically and the interior edge thread will be created as shown in Figure 9.15: Edge Loops for Face Zone Approach and Fixed Angle = 55 (or Adaptive Angle) (p. 144).

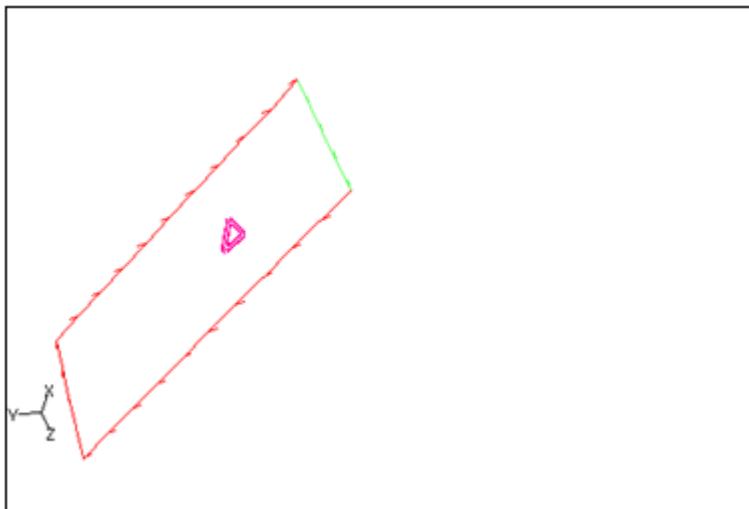
Figure 9.15: Edge Loops for Face Zone Approach and Fixed Angle = 55 (or Adaptive Angle)



- Figure 9.16: Edge Loop for Face Seed Approach and Fixed Angle = 65 (p. 145) shows the single edge loop created by using the **Face Seed** approach and **Fixed** angle criterion, with the angle specified as 65 degrees. The edge thread at the common edge is not created since the specified value for **Angle** is greater than the feature angle.

Figure 9.16: Edge Loop for Face Seed Approach and Fixed Angle = 65

- Figure 9.17: Edge Loops for Face Seed Approach and Fixed Angle = 55 (or Adaptive Angle) (p. 145) shows the edge loops created by using the **Face Seed** approach and **Fixed** angle criterion, with the angle specified as 55 degrees. The boundary edge thread is created based on the seed face selected. The interior edge thread at the common edge is created since the specified value for **Angle** is smaller than the feature angle. Alternatively, if you use the **Adaptive** angle criterion, the change in angle will be detected automatically and the boundary and interior edge threads will be created as shown in Figure 9.17: Edge Loops for Face Seed Approach and Fixed Angle = 55 (or Adaptive Angle) (p. 145).

Figure 9.17: Edge Loops for Face Seed Approach and Fixed Angle = 55 (or Adaptive Angle)

9.6.2. Modifying Edge Loops

The following edge modification options are available:

- Deleting edge loops.
- Copying existing edge loops (including the modes) to a new edge loop.
- Toggling the edge loop type between boundary and interior.

- Grouping and ungrouping edge loops.
 - Orienting the edges on the edge loop to point in the same direction.
 - Reversing the direction of the edge loop.
-

Note

The direction of a boundary edge loop determines the side from which new faces are formed. The direction of a boundary edge loop should be right-handed with respect to the average normal of the face zone to be remeshed. However, the direction is not so important in the case of interior edge loops since faces are always formed on both sides of the loop.

- Separating the edge loop based on the connectivity and feature angle specified.
 - Merging multiple edge loops into a single loop.
-

Note

Only edge loops of the same type (boundary or interior) can be merged.

- Remeshing the edge loops to modify the node distribution.
- Projecting the edges of the edge loop onto a face zone.

You can select the closest point method or specify the direction in which the edge should be projected onto the selected face zone.

- Intersecting edge loops to create a new edge loop comprising the common edges.

9.6.3. Using the Feature Modify Dialog Box

The **Feature Modify** dialog box can be used for creating edge loops as follows:

1. Select the required zone(s) from the **Boundary Zones** selection list.
 2. Select **Create** from the **Options** list.
 3. Select the appropriate option from the **Approach** drop-down list. Select the appropriate **Seed Face** when using the **Face Seed** approach.
 4. Select the appropriate option from the **Angle Criterion** drop-down list. Specify an appropriate value for the **Angle** when using the **Fixed** angle criterion.
 5. Click **Apply** to create the edge loops.
-

Important

You can also use the **Surface Retriangulation** dialog box to create edge loops using the **Face Zone** approach. Refer to [Surface Retriangulation Dialog Box \(p. 520\)](#) for details.

The **Feature Modify** dialog box can be used for modifying edge loops as follows:

- Operations such as deleting, copying, grouping/ungrouping, orienting, separating, and merging edge loops, toggling the edge loop type, and reversing the edge loop direction:

- Select the appropriate zone(s) in the **Edge Zones** selection list.

Warning

You can select only one edge zone when separating an edge loop.

- Click the appropriate button in the **Edge Modify** group box.

- Remeshing edge loops:

- Select **Remesh** from the **Options** list.
- Select the appropriate zone(s) from the **Edge Zones** selection list.
- Select an appropriate method from the **Method** drop-down list. You can specify a constant spacing of nodes or select either the arithmetic or the geometric method for node spacing. You can also select the **Size Function** option to use advanced size functions to remesh the edge loops.

For the **Constant**, **Arithmetic**, or **Geometric** methods, set the following parameters:

- Specify values for **First Spacing** and **Last Spacing** as required.

Note

For the **Constant** method, the value specified for **First Spacing** will be the constant node spacing. Also, the **Last Spacing** option is not relevant for the **Constant** method and will not be available.

- Specify an appropriate value for **Feature Angle**.

- Enable **Quadratic Reconstruct**, if required. The quadratic reconstruction option allows you to reconstruct the edge by fitting a quadratic polynomial between the original edge nodes.

Alternatively, for remeshing using size functions, make sure the size function(s) are defined as required (see [Defining Size Functions \(p. 93\)](#)).

- Click **Apply** to remesh the edge loop.

- Projecting edge loops:

- Select **Project** from the **Options** list.
- Select the appropriate zone(s) in the **Edge Zones** selection list.
- Select the appropriate face zone from the **Face Zones** selection list.

4. Select the appropriate projection method from the **Method** drop-down list. The **Closest Point** method specifies that the edge should be projected to the closest point on the face zone selected. The **Specific Direction** method allows you to project the edge on the face zone in a specific direction.
 5. Specify the direction in which the edge(s) should be projected when using the **Specific Direction** method.
 6. Click **Apply** to project the edge onto the selected face zone.
- Intersecting edge loops:
 1. Select **Intersect** from the **Options** list.
 2. Select the appropriate zone(s) in the **Edge Zones** selection list.
 3. Enable **Delete** in the **Overlapped Edges** group box if you want to automatically delete all the overlapping edges.

You can use the delete-overlapped-edges text command to delete individual overlapping edges.
 4. Specify an appropriate value for **Intersection Tolerance**.
 5. Click **Apply** to intersect the selected edge loops.

9.6.4. Text Commands for Creating and Modifying Features

The text commands for creating and modifying features are:

/boundary/feature/copy-edge-zones

copies the specified edge zone(s) to new edge zone(s).

/boundary/feature/create-edge-zones

extracts edge loops for the specified face zone(s) based on the feature method specified. You also need to specify an appropriate value for feature angle when using the `fixed-angle` method.

Note

The **Face Seed** approach cannot be used when creating edge loops using text commands.

/boundary/feature/delete-degenerated-edges

deletes degenerated edges (edges where the two end nodes are the same) for the edge zone(s) specified.

/boundary/feature/delete-edge-zones

deletes the specified edge zone(s).

/boundary/feature/edge-size-limits

reports the minimum, maximum, and average edge length for the specified edge zone(s) in the console.

/boundary/feature/group

associates the specified edge zone(s) with the specified face zone.

boundary/feature/intersect-edge-zones

intersects the specified edge loops to create a new edge loop comprising the common edges. You can enable automatic deleting of overlapped edges and specify an appropriate intersection tolerance.

/boundary/feature/list-edge-zones

lists the name, ID, type, and count for the specified edge zone(s).

/boundary/feature/merge-edge-zones

merges multiple edge loops of the same type into a single loop.

/boundary/feature/orient-edge-direction

orients the edges on the loop to point in the same direction.

/boundary/feature/project-edge-zones

projects the edges of the specified loop onto the specified face zone using the specified projection method.

/boundary/feature/remesh-edge-zones

remeshes the specified edge zones, modifying the node distribution according to the specified remeshing method (constant, arithmetic, geometric or size-function). For the constant, arithmetic, and geometric methods, you can specify spacing values and feature angle, and enable quadratic reconstruction, if required.

/boundary/feature/reverse-edge-direction

reverses the direction of the edge loop.

/boundary/feature/separate-delete-small-edges

separates the edge zones based on the feature angle specified, and then deletes the edges having a count smaller than the minimum count specified.

/boundary/feature/separate-edge-zones

separates the specified edge loop based on connectivity and the specified feature angle.

/boundary/feature/separate-edge-zones-by-seed

separates the edge loop based on the seed edge specified. The edge zone separation angle is used to separate the edge zone (default 40).

/boundary/feature/toggle-edge-type

toggles the edge type between boundary and interior.

/boundary/feature/ungroup

ungroups previously grouped edge zones.

9.7. Remeshing Boundary Zones

In some cases, you may need to regenerate the boundary mesh on a particular boundary face zone. You may find that the mesh resolution on the boundary is not high enough, or that you want to generate triangular faces on a boundary that currently has quadrilateral faces. Remeshing of boundary faces can be accomplished using the **Surface Retriangulation** dialog box.

You can remesh the boundary face zones based on edge angle, curvature, and proximity.

9.7.1. Creating Edge Loops

9.7.2. Modifying Edge Loops

9.7.3. Remeshing Boundary Face Zones

9.7.4. Using the Surface Retriangulation Dialog Box

9.7.5. Text Commands for Remeshing

9.7.1. Creating Edge Loops

To remesh a face zone, you first need to generate edge loops (or edge zones) on the borders of the face zones using the parameters available in the **Edge Create** group box in the **Surface Retriangulation** dialog box (see [Using the Surface Retriangulation Dialog Box \(p. 151\)](#)).

You can create the edge loops according to your requirement by specifying an appropriate combination of the edge loop creation approach and angle criteria (refer to [Creating Edge Loops \(p. 142\)](#) for details). Alternatively, you can use the **Feature Modify** dialog box to create the edge loops.

Important

The **Face Seed** approach is available only when you use the **Feature Modify** dialog box for creating edge loops. Click the **Feature Modify...** button to open the **Feature Modify** dialog box.

You can also draw the edge loops to determine their direction (i.e., the start point and the end point).

9.7.2. Modifying Edge Loops

You can modify the node distribution on the edge loops using the **Feature Modify** dialog box (opened using the **Feature Modify...** button in the **Surface Retriangulation** dialog box). If you want to assign different node distributions to two or more portions of an edge loop, you can separate the loop based on a specified feature angle between consecutive edges. Separation is performed automatically at multiply-connected nodes.

After creating edge loops using an appropriate combination of the edge loop creation approach and angle criteria, modify the edge loops as required. You can modify the edge loops using the options available in the **Feature Modify** dialog box. Refer to [Using the Feature Modify Dialog Box \(p. 146\)](#) for details on using the various options available in the **Feature Modify** dialog box.

It is also possible to modify the edges of the loop using the operations in the **Modify Boundary** dialog box. Any edges you create must have the same direction as the edge loop.

Important

You cannot remesh a continuous edge loop. You must first separate it into two or more non-continuous edge loops (i.e., edge loops with start and end points).

9.7.3. Remeshing Boundary Face Zones

If the mesh resolution on the boundary face zone is not enough, or you want to create triangular faces on a boundary face zone that currently has quadrilateral faces, you can remesh that boundary face zone. You can remesh the boundary face zone using the **Surface Retriangulation** dialog box (see [Using the Surface Retriangulation Dialog Box \(p. 151\)](#) for details).

9.7.4. Using the Surface Retriangulation Dialog Box

The generalized procedure for remeshing a boundary face zone using the Surface Retriangulation Dialog Box (p. 520) is as follows:

1. Create the edge loops as appropriate.
 - a. Select the boundary face zone for which you want to create edge loops in the **Boundary Face Zones** selection list.
 - b. Select the appropriate option from the **Angle Criterion** drop-down list.

By default, the **Face Zone** approach is used to create edge loops. Therefore, you can only specify the required **Angle Criterion** in the **Surface Retriangulation** dialog box. If however, you want to use **Face Seed** approach, you can use the **Feature Modify** dialog box to create the edge loops instead (see [Creating Edge Loops \(p. 142\)](#)).

- c. Click **Create**.

The edge loops created will now be available in the **Edge Zones** selection list.

- d. Select the appropriate zone(s) in the **Edge Zones** selection list and click **Draw** to display them.

The selected edge zone(s) will be displayed in the graphics window. If you are not satisfied with the edge loops and you want to modify them, open the **Feature Modify** dialog box.

2. Modify the edge loops as required using the options available in the **Feature Modify** dialog box. Click the **Feature Modify...** button to open the **Feature Modify** dialog box. Refer to [Modifying Edge Loops \(p. 145\)](#) for details.

When you are satisfied with the edge loops you can proceed to remesh the faces.

3. Select the zone to be remeshed in the **Boundary Face Zones** list.

You can select only a single boundary face zone for remeshing, unless the **Use Conformal Remesh** option is enabled.

4. Set the appropriate remeshing options in the **Face Remesh Options** group box.

- a. Enable **Size Function** if you want to use the advanced size functions to remesh the faces.

Note

Edge zone(s) associated with face zone(s) are not remeshed implicitly. If you have feature edge zone(s) associated with the surface being remeshed, you need to remesh them before remeshing the face zone(s).

- b. Select the appropriate options from the **Reconstruction (Order)** drop-down list in the **Face Remesh Options** group box.

- c. Enable **Replace Face Zone**, if required.
-

Important

Remeshing can be performed on both triangular and quadrilateral face zones. However, it will always result in a triangular face zone.

- d. Enable **Use Conformal Remesh** if you wish to conformally remesh multiple face zones connected along the shared boundary.
-

Note

- This option is available only when **Size Function** is enabled and **None** is selected in the **Reconstruction** drop-down list. You will be asked to compute the size field or read a size field file.
 - Periodic face zones cannot be remeshed using this option.
-
- Set the minimum **Corner Angle** to specify the minimum angle between feature edges that will be preserved during remeshing.
-

Note

The shared boundary between different zones will be remeshed only if all the face zones incident to it are selected for conformal remeshing.

5. Click **Remesh** to remesh the face zones.
-

Note

Edge loops are saved when the mesh file is written.

9.7.5. Text Commands for Remeshing

Text commands for remeshing face zones are:

/boundary/remesh/create-edge-loops
creates edge loops for a specified face zone, based on feature angle.

/boundary/remesh/create-intersect-loop
creates an interior edge loop at the intersection between two adjacent face zones.

/boundary/remesh/delete-overlapped-edges
deletes edges that overlap selected edge loops.

/boundary/remesh/remesh-face-zone

remeshes a specified face zone by automatically extracting edge loops. If edge loops are present in the current domain (e.g., if they were created using the `create-edge-loops` command), they are used to remesh the specified face zone.

You can specify whether the original face zone is to be deleted after remeshing. If the original face zone is to be retained, you can specify whether the remeshed zone should replace the original face zone. In this case, the remeshed face zone will be given the original face zone name while the original face zone will be renamed with `-orig-#` appended to the original name.

/boundary/remesh/remesh-face-zones-conformally

remeshes face zones using the current size field and keeping a conformal interface between them. If no size field is available, an error message will be generated.

This command will prompt for:

- Boundary Face Zones
- Boundary Edge Zones
- feature angle – used to determine the minimum angle between features that will be preserved during remeshing
- corner angle – used to specify the minimum angle between feature edges that will be preserved
- Replace Face Zone? – (default is yes) the remeshed face zone(s) will take the name of the original zones, and the original face zone(s) will have “orig” appended to their name. If No, the remeshed face zone(s) will have “retri” added postfix.

Note

Periodic face zones cannot be remeshed using this command.

/boundary/remesh/remesh-overlapping-zones

remeshes overlapping face zones. The non-overlapping region is remeshed using the edge loops created from the overlapping face zones.

/boundary/remesh/controls/delete-overlapped?

enables/disables the deleting of overlapped edges.

/boundary/remesh/controls/direction

specifies the direction of the edge loop projection.

/boundary/remesh/controls/project-method

specifies the method for projecting edge loops.

/boundary/remesh/controls/quadratic-recon?

enables/disables quadratic reconstruction of edge loops during remeshing.

/boundary/remesh/controls/remesh-method

specifies the method to be used for the node distribution on the edge loop.

/boundary/remesh/controls/spacing

sets the node spacing for the edge loop.

/boundary/remesh/controls/tolerance

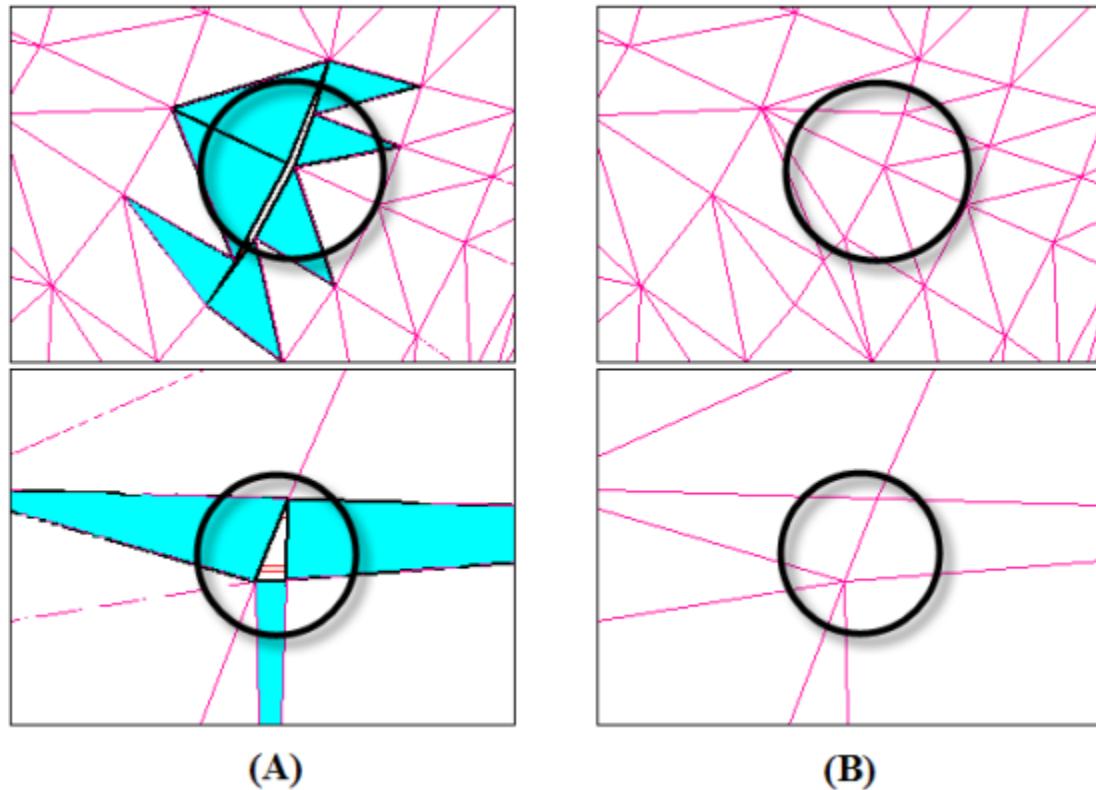
sets the tolerance for determining if two edges intersect.

9.8. Faceted Stitching of Boundary Zones

You can repair surfaces having internal cracks or free edges using the **Faceted Stitch** option. You can specify an appropriate tolerance value within which the free edges will be stitched. The **Self Stitch only** option allows you to stitch the edges within the same boundary zone. The faceted stitching operation is available only for triangular boundaries.

Figure 9.18: Mesh (A) Before and (B) After Using the Faceted Stitch Option (p. 154) shows the repair of a surface with internal cracks.

Figure 9.18: Mesh (A) Before and (B) After Using the Faceted Stitch Option



The command /boundary/remesh/faceted-stitch-zones allows you to perform the faceted stitching of zones.

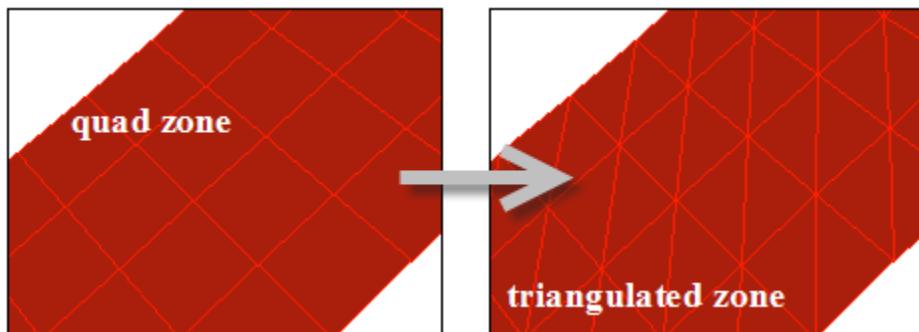
Note

Features may not be maintained when using the faceted stitching operation.

9.9. Triangulating Boundary Zones

Some operations like intersection, joining, stitching, and wrapping are limited only to triangular boundary zones. You can remesh a quadrilateral face zone with triangular faces as shown in [Figure 9.19: Triangulating a Boundary Zone \(p. 155\)](#). You can either copy the quad zone(s) and triangulate the copied zones or replace the original quad zone(s) with the triangulated zone.

Figure 9.19: Triangulating a Boundary Zone



9.10. Separating Boundary Zones

There are several methods available that allow you to separate a single boundary face zone into multiple zones of the same type. If your mesh contains a zone that you want to break up into smaller portions, you can make use of these options. For example, if you created a single wall zone when generating the mesh for a duct, but you want to generate different mesh shapes on specific portions of the wall, you will need to break that wall zone into two or more wall zones.

9.10.1. Methods for Separating Face Zones

9.10.2. Text Commands for Separating Face Zones

9.10.1. Methods for Separating Face Zones

There are six methods available for separating a boundary face zone. They are:

Separating Using Angle

For geometries with sharp corners, it is often easy to separate face zones based on the significant angle. Faces with normal vectors that differ by an angle greater than or equal to the specified angle value will be placed in different zones.

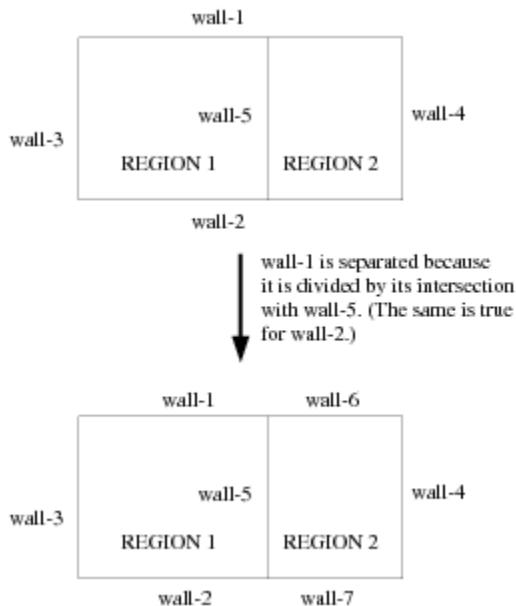
For example, if the mesh consists of a cube, and all 6 sides of the cube are in a single wall zone, you would specify a significant angle of 89°. Since the normal vector for each cube side differs by 90° from the normals of its adjacent sides, each of the 6 sides will be placed in a different wall zone.

Separating Using Regions

You can also separate face zones based on contiguous regions. For example, if you want to generate the mesh in different regions of the domain using different meshing parameters, you may need to split up a boundary zone that encompasses more than one of these regions. Separating based on region splits non-contiguous boundary face zones (i.e., zones that are separated into two or more isolated groups) into multiple zones.

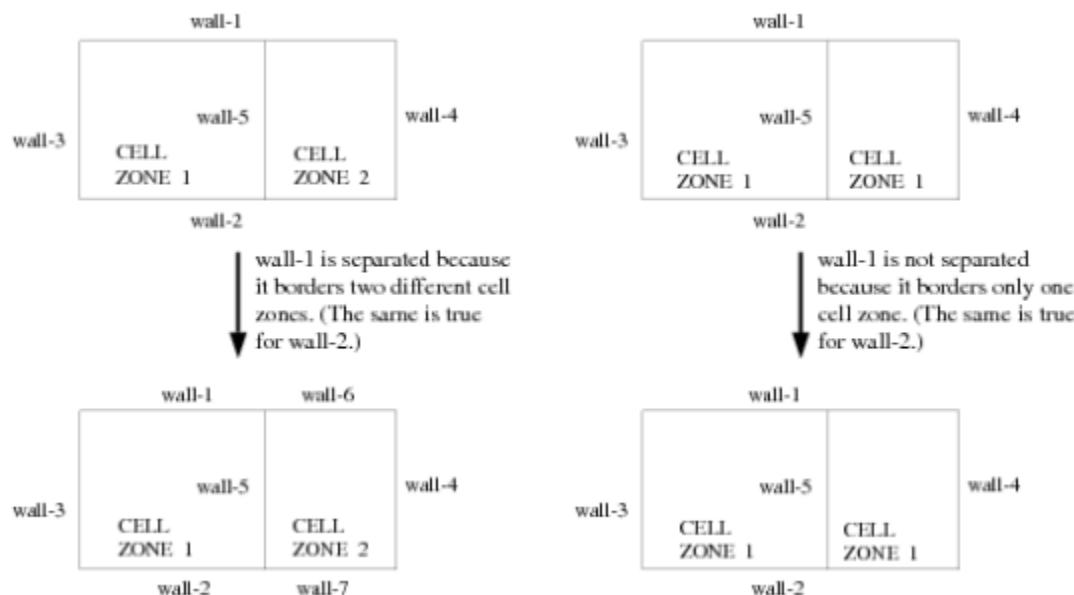
This command will also split zones that are divided by another face zone. An example could be two face zones touching in a "T". Using this command on the top zone (e.g., wall-1 in [Figure 9.20: Face Separation Based on Region \(p. 156\)](#)) would split it into two zones. However, individual faces in the corners at the "T" junction may be put in their own zones. To check for this problem, list the new face zones (using the **List** button in the **Boundary Zones** dialog box), looking for zones with a single face in them. You can then merge these faces into the appropriate zone.

Figure 9.20: Face Separation Based on Region



Separating Based on Neighboring Cell Zones

Region separation will split wall-1 in [Figure 9.20: Face Separation Based on Region \(p. 156\)](#) into two zones regardless of whether the two regions are in the same cell zone. However, neighbor-based separation will yield different results. If both regions are in the same cell zone, wall-1 will not be separated (see [Figure 9.21: Face Separation Based on Cell Neighbor \(p. 157\)](#)). If they are in different cell zones, the zone will be separated. Thus, when neighbor separation is used, wall-1 will be separated only if it is adjacent to more than one cell zone. If the two regions are in two different cell zones, then wall-1 has two different neighboring cell zones and therefore it will be separated into two wall zones.

Figure 9.21: Face Separation Based on Cell Neighbor

Separating Based on the Face/Element Shape

You can also separate face zones based on the shape of the faces. For example, if a face zone contains both triangular and quadrilateral faces, you can separate the zone into two zones (one containing the triangular faces, and the other containing the quadrilateral faces).

Separating Using a Seed Element

You can separate face zones by specifying a face element (in the face zone) as a seed face. You can also separate different faces of a single face zone using this method. The surface on which you define a seed face gets separated from rest of the face zone. You can separate face zones using the seed face based on the following criteria:

- Feature Angle Criteria

This method allows you to separate the surface on which you have defined a seed face from the surfaces around it based on the specified value of the feature angle. The feature angle is the angle between the normal vectors of the cells. To separate the face zones based on this criteria, do the following:

1. Select **Seed** in the **Options** list and **Angle** in the **Flood Fill Options** list.
2. Specify the seed element in the **Face Seed** text entry field. Right-click on the face you want to choose as a seed element in the graphics window. The **Face Seed** field will be updated automatically.
3. Specify the required feature angle in the **Angle** field.
4. Click **Separate**.

The surface on which you defined the seed face will be separated from other surfaces of the zone for which the feature angle change is greater than or equal to the specified value. For example, if the mesh consists of a cube, and all 6 sides of the cube are in a single wall zone, specify a significant angle of 89° and specify a seed face on any one of the walls. Since the normal vector for each cube side differs by 90° from the normals of its adjacent sides, the face on which you have defined a seed

cell will be placed in a different wall zone. Therefore, two zones will be created, one zone will have a face on which you defined a seed face and the second zone will have remaining faces.

- Edge Loop Criteria

This method allows you to separate the surface, on which you have defined a seed face, from the other faces in the zone based on the existing edge thread loops associated with it. You must create the edge thread loops for the given mesh to use this method.

To separate the face zones based on this criteria, do the following:

1. Select **Seed** in the **Options** list and **Edge Loop** in the **Flood Fill Options** list.
2. Specify the seed element in the **Face Seed** text entry field.

For this method, you will only specify the seed element. The **Angle** field will not be available.

3. Click **Separate**.

Important

Create edge threads on the surface zones again using the **Surface Retriangulation** dialog box after performing above operations.

Separating Based on Marked Faces

You can separate face zones by placing marked faces in a new zone. To use this option in the **Separate Face Zones** dialog box, explicitly define a subregion of the domain (using the **Boundary Refinement Region** dialog box), then separate face zones based on whether or not each face in the specified zone is in the selected local region.

You can also mark faces for separation using the following commands:

- /boundary/mark-faces-in-region
- /boundary/mark-face-proximity
- /boundary/mark-face-intersection

The command /boundary/separate/sep-face-zone-by-mark can also be used to separate zones based on the marked faces. The command /boundary/clear-marked-faces can be used to clear marked faces. These commands are described in [Additional Boundary Mesh Text Commands \(p. 172\)](#).

9.10.2. Text Commands for Separating Face Zones

Text commands for separating face zones are listed below:

/boundary/separate/local-regions/define
defines the refinement region according to the specified parameters.

/boundary/separate/local-regions/delete
deletes the specified region.

/boundary/separate/local-regions/init
creates a region encompassing the entire geometry.

/boundary/separate/local-regions/list-all-regions
lists all the refinement regions in the console.

/boundary/separate/mark-faces-in-region
marks the faces that are contained in a specified local refinement region.

/boundary/separate/sep-face-zone-by-angle
separates a face zone based on significant angle.

/boundary/separate/sep-face-zone-by-cnbor
separates a face zone based on its cell neighbors.

/boundary/separate/sep-face-zone-by-mark
separates a face zone by moving marked faces to a new zone.

/boundary/separate/sep-face-zone-by-region
separates a face zone based on contiguous regions.

/boundary/separate/sep-face-zone-by-shape
separates a face zone based on the shape of the faces (triangular or quadrilateral).

/boundary/separate/sep-face-zone-from-seed
separates a face zone by defining a seed face on the surface.

9.11. Projecting Boundary Zones

Another mesh refinement method involves projecting the nodes of one face zone onto another (possibly non-planar) face zone to create a new face zone that has the same connectivity as the original face zone. This new face zone is created after the projection, and no cell zones are created. The face zone that is projected is not modified in any way.

Projecting a face zone is used mainly to fill in gaps by extending the domain through the projection. The original connectivity is maintained after the projection, with the effect being that elements on the connected side zones will be stretched to cover the projection distance. Affected side zones should then be remeshed to obtain regular size elements on them. Such a remeshing results in a new side zone, after which you can (and should) delete the original side zone. Finally, you can mesh the domain to get the volume elements.

9.11.1. Text Commands for Projecting Boundary Zones

The text command for projecting boundary zones is:

/boundary/project-face-zone
allows nodes on a selected face zone to be projected onto a target face zone. Projection can be performed based on normal direction, closest point, or specified direction.

9.12. Creating Groups

You can create groups of faces and edges which will be available in all the dialog boxes along with the default groups (e.g., boundary, tri, quad, etc.). The face and edge zones are grouped separately. The

User Defined Groups dialog box allows you to define new face and/or edge groups, update existing groups, activate or delete a particular group. Although the dialog box is opened from the **Boundary** menu, it can be used with all dialog boxes that contain zone lists.

Note

When a user-defined group is activated, the wild-cards used for zone selection in all the text commands will return zones contained in the active group. For example, the command `/display/boundary-grid *` will display all the boundary zones contained in the active group.

For object based meshing (see [Object-Based Meshing \(p. 317\)](#)), you can create a face group and an edge group comprising the face zones and edge zones included in the specified objects using the options in the **Zone Group** group box in the **Operations** tab in the [Manage Objects Dialog Box \(p. 430\)](#). Additionally, a face zone group is automatically created when a mesh object is created using the **Sew** operation. This face zone group is prefixed by `_mesh_group`, and allows easy selection of mesh object face zones for various operations (improve, smooth, etc.).

For CutCell meshing, the mesher separates the face zones by cell neighbor and creates a face zone group for the face zones of each fluid cell zone. See [Generating the CutCell Mesh \(p. 295\)](#) for details.

Note

When an object is deleted along with the face and edge zones comprising the object, the corresponding groups will also be deleted.

9.12.1. Text Commands for User-Defined Groups

The text commands for manipulating user-defined groups are as follows:

/boundary/manage/user-defined-groups/activate
allows you to activate the specified user-defined group.

/boundary/manage/user-defined-groups/create
allows you to create a group of face or edge zones comprising the specified zones.

/boundary/manage/user-defined-groups/delete
deletes the specified group.

/boundary/manage/user-defined-groups/list
lists the groups in the console.

/boundary/manage/user-defined-groups/update
allows you to modify an existing group.

/objects/create-groups
creates a face group and an edge group comprising the face zone(s) and edge zone(s) included in the specified object(s), respectively.

/mesh/cutcell/objects/create-groups

creates a face group and an edge group comprising the face zone(s) and edge zone(s) included in the specified object(s), respectively.

9.13. Manipulating Boundary Zones

Boundary zones are groups of boundary faces. Usually the grouping collects boundary faces with the same boundary conditions, but further sub-groupings are often used to preserve a sharp edge in the surface mesh or simply as an artifact of the boundary mesh generation process.

The options in the [Manage Face Zones Dialog Box \(p. 528\)](#) can be used to find information about each zone, identify them, merge zones or delete them, change the boundary type of all faces in a zone, rename zones, and rotate, scale, or translate zones. Each zone has a unique ID, which must be a positive integer.

9.13.1. Text Commands for Manipulating Boundary Zones

The text commands with the same functions as the controls in the **Manage Face Zones** dialog box are as follows:

/boundary/manage/auto-delete-nodes?

specifies whether or not unused nodes should be deleted when their face zone is deleted.

/boundary/manage/change-prefix

allows you to change the prefix for the specified face zones.

/boundary/manage/copy

copies the nodes and faces of the specified face zones.

/boundary/manage/create

creates a new face zone.

/boundary/manage/delete

deletes the face zone.

/boundary/manage/flip

reverses the normal direction of the specified boundary zone(s).

/boundary/manage/id

specifies a new boundary zone ID. If a conflict is detected, the change will be ignored.

/boundary/manage/list

prints information about all boundary zones.

/boundary/manage/merge

merges face zones. You can use the alphabetic-order, first-zone, or larger-area as appropriate.

/boundary/manage/name

gives a face zone a new name.

/boundary/manage/orient

consistently orients the faces in the specified zones.

/boundary/manage/origin

specifies a new origin for the mesh, to be used for face zone rotation and for periodic zone creation.
The default origin is (0,0,0).

/boundary/manage/rotate

rotates all nodes of the specified face zone(s).

/boundary/manage/rotate-model

rotates all nodes of the model through the specified angle, based on the specified point and axis of rotation.

/boundary/manage/scale

scales all nodes of the specified face zone(s).

/boundary/manage/scale-model

scales all nodes of the model by multiplying the node coordinates by the specified scale factors (x, y, z).

/boundary/manage/translate

translates all nodes of the specified face zone(s).

Note

The translation offsets are interpreted as absolute numbers in meshing mode. In solution mode, however, the translation offsets are assumed to be distances in the length unit set. This may lead to differences in domain extents reported after translating the mesh in the respective modes.

/boundary/manage/translate-model

translates all nodes of the model by the specified translation offsets (x, y, z).

Note

The translation offsets are interpreted as absolute numbers in meshing mode. In solution mode, however, the translation offsets are assumed to be distances in the length unit set. This may lead to differences in domain extents reported after translating the mesh in the respective modes.

/boundary/manage/type

changes the boundary type of the specified face zone.

9.14. Manipulating Boundary Conditions

Case files read in the meshing mode also contain the boundary and cell zone conditions along with the mesh information. The [Boundary Conditions Dialog Box \(p. 532\)](#) allows you to copy or clear boundary conditions assigned to the boundary zones when a case file is read.

- You can copy the boundary conditions from the zone selected in the **With** list to those selected in the **Without** list using the **Copy** option.
- You can clear the boundary conditions assigned to the zones selected in the **With** list using the **Clear** option.

9.14.1.Text Commands for Manipulating Boundary Conditions

The following text commands allow you to manipulate boundary conditions assigned to the boundary zones when a case file is read:

/boundary/boundary-conditions/clear
clears the boundary conditions assigned to the specified face zones.

/boundary/boundary-conditions/clear-all
clears the boundary conditions assigned to all the face zones.

/boundary/boundary-conditions/copy
allows you to copy the boundary conditions from the face zone selected to the specified face zones.

9.15.Creating Surfaces

You can create specific types of surfaces within the existing geometry using one of the options available in the **Boundary/Create** menu. The following sections explain how to create surfaces.

- 9.15.1.Creating a Bounding Box
- 9.15.2.Creating a Planar Surface Mesh
- 9.15.3.Creating a Cylinder/Frustum
- 9.15.4.Creating a Swept Surface
- 9.15.5.Creating a Revolved Surface
- 9.15.6.Creating Periodic Boundaries
- 9.15.7.Text Commands for Creating Surfaces

9.15.1.Creating a Bounding Box

In some cases, you may want to create a box that encloses the input geometry (e.g., creating a wind tunnel around a geometry). You can create a bounding box around the input geometry or only the selected zones of the geometry using the **Bounding Box** dialog box. You can also specify the required clearance values of the bounding box from the boundaries of the geometry.

There are two methods available for creating bounding box:

Using Absolute Values

This method allows you to create the bounding box by specifying the minimum and maximum extents of the bounding box in X, Y, and Z directions.

Using Relative Values

This method allows you to create the bounding box by specifying the relative coordinate values with reference to the selected face zone.

9.15.1.1. Using the Bounding Box Dialog Box

The procedure for creating a bounding box is as follows:

1. Select the zone(s) around which you want to create a bounding box in the **Face Zones** list.
2. Select the appropriate method in the **Method** list.

- a. For the **Absolute** method, specify the bounding box extents (**X Min**, **X Max**, **Y Min**, **Y Max**, **Z Min**, and **Z Max**). If you click **Compute**, the extents will be computed such that the bounding box encloses the selected boundary zones.
- b. For the **Relative** method, specify the clearance values in the **Delta** entry fields (**Delta X Min**, **Delta X Max**, **Delta Y Min**, **Delta Y Max**, **Delta Z Min**, and **Delta Z Max**).

Initially, all the **Delta** entry fields will be set to 0. This implies that the bounding box will touch the boundaries of the selected face zones. Positive delta values indicate that the bounding box will be created outside the initial bounding box while negative values indicate that the bounding box will be created inside the initial bounding box.

3. Specify an appropriate value for **Edge Length**. When you click **Compute** for the **Absolute** method, the value will be automatically set to 1/10th that of the minimum length of the bounding box.
4. Enable **Create Object** if you need to create a wrap object based on the bounding box face zone created.

Note

Do not use the **Create Object** option if the box is to be used as a body of influence while setting up the size functions.

5. Click **Draw** to visualize the bounding box.
6. Click **Create** to create a bounding box based on the specified parameters.

9.15.2. Creating a Planar Surface Mesh

In some cases, you may need to create a plane surface mesh in the geometry (e.g., creating a baffle-like surface inside a hollow tube). You can create a plane surface and mesh the surface using triangular faces of the required size using the **Plane Surface** dialog box.

Warning

It is possible to create a planar surface of rectangular shape only. You cannot create a planar surface of any other shape.

There are two methods available for creating planar surface mesh:

- Axis Direction Method:

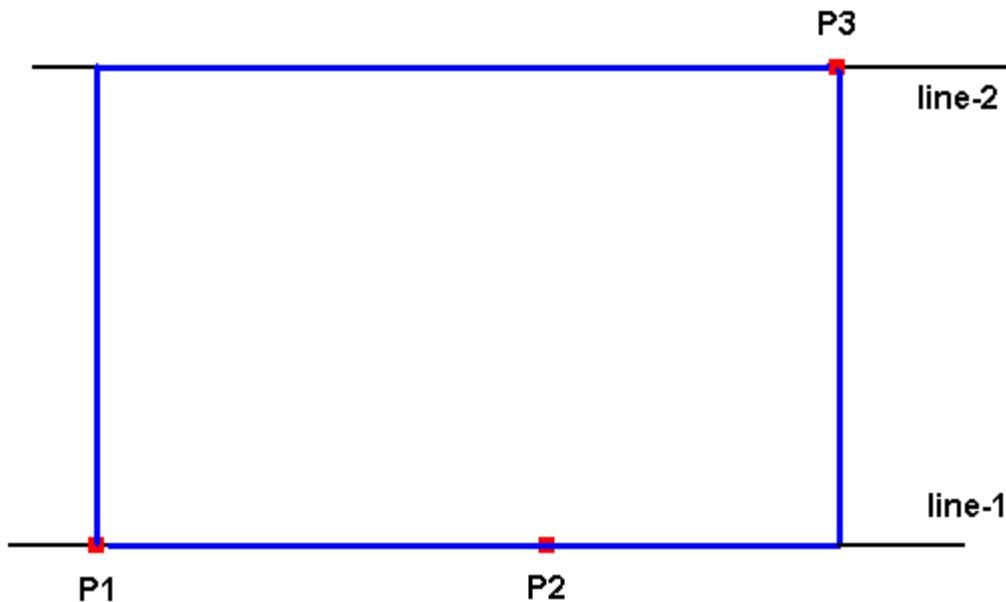
This method allows you to create the plane surface perpendicular to any of the coordinate axes. Select the axis perpendicular to which you want to create a planar surface mesh and then, specify the coordinates of the points that will form a rectangular surface perpendicular to the axis selected. You can also create a plane surface enclosing the boundaries of the selected face zone using this method.

- Planar Points Method:

This method allows you to create a plane surface mesh from three points in the geometry selected using the mouse.

The concept of the planar points method is shown in [Figure 9.22: Planar Points Method \(p. 165\)](#). After specifying the planar points, the first point (**P1**) and second point (**P2**) are connected to each other by a line (**line-1**). Another line (**line-2**) is drawn through the third point (**P3**) parallel to the first line. Perpendiculars are drawn from points **P1** and **P3** on **line-2** and **line-1** respectively.

Figure 9.22: Planar Points Method



This creates a rectangular surface which can be meshed as required.

9.15.2.1. Using the Plane Surface Dialog Box

The procedure for creating a surface mesh is as follows:

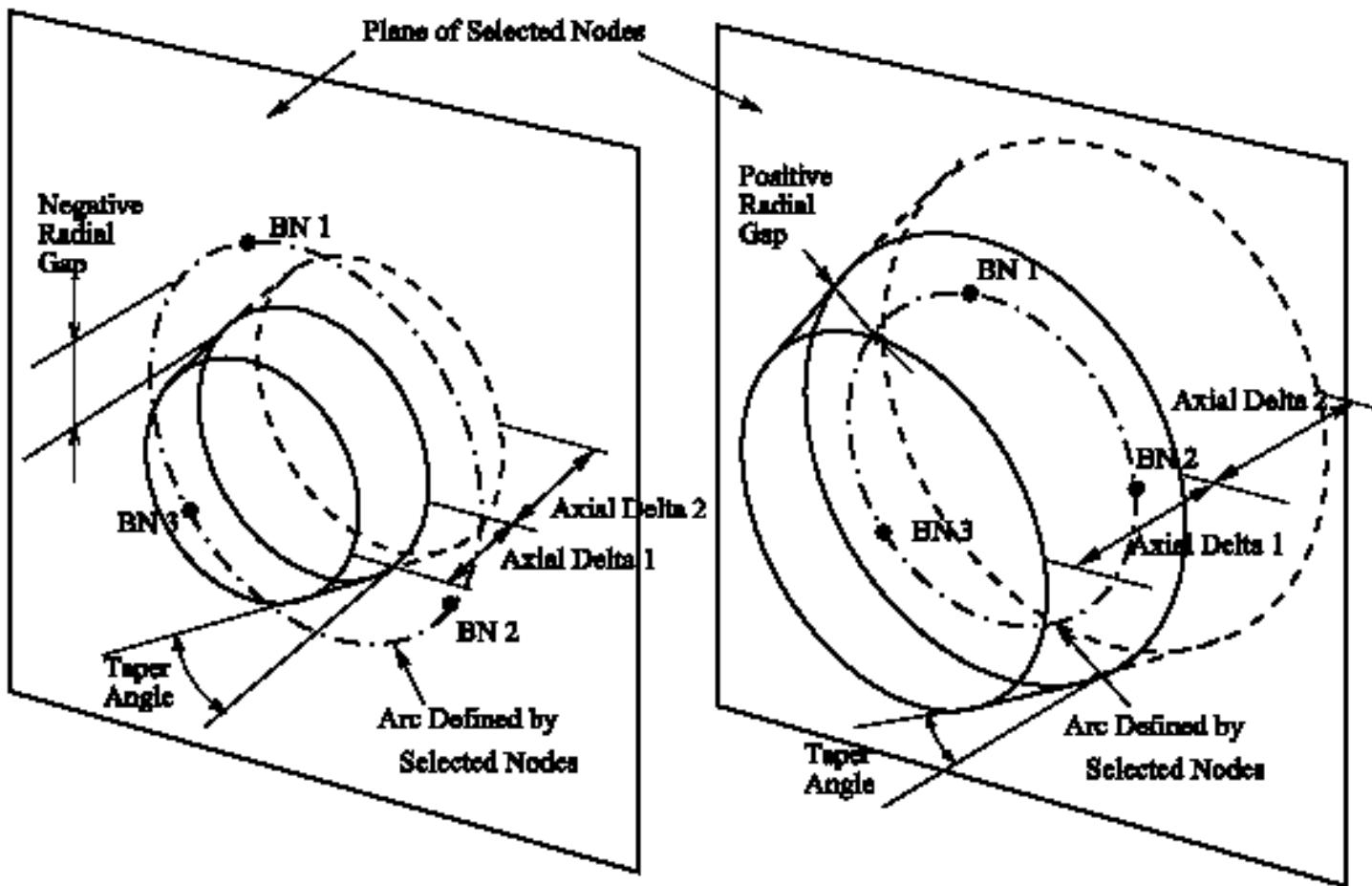
1. Select the appropriate method in the **Options** list.
 - a. For the **Axis Direction** method, select the appropriate face zone(s), direction, and specify the coordinates of the points perpendicular to the axis.
If you select **X Axis** then the entry box for specifying coordinates in X direction will not be accessible. This applies to the other two axes as well.
 - b. For the **Points** method, specify the coordinates for the three points defining the plane. You can click the **Select Points...** button and select the points using the mouse button.
2. Specify an appropriate value for **Edge Length**. If you click **Compute**, the **Edge Length** will be computed as 1/10th of the minimum distance along the coordinate axes.
3. Enable **Create Object** if you need to create a wrap object based on the plane surface face zone created.
4. Click **Draw** to visualize the surface.
5. Click **Create** to create the planar surface.

9.15.3. Creating a Cylinder/Frustum

In some cases, you may want to create a cylinder or frustum within the existing geometry (e.g., creating an MRF zone for problems involving moving parts such as rotating blades or impellers, creating a cylindrical surface to close a gap in the geometry, etc.). You can create a cylindrical surface and mesh it with a triangular surface mesh using the options available in the **Cylinder** dialog box.

- **Using 3 Arc Nodes** You can create a cylindrical surface using three nodes which lie on a circular arc. (see [Figure 9.23: Cylinder Defined by 3 Arc Nodes, Radial Gap, and Axial Delta \(p. 166\)](#)). Specify the radial gap and taper angle which will determine the actual radii of the cylinder/frustum to be created. You can specify a positive or negative radial gap value depending on the required size of the cylinder/frustum. A taper angle of zero will result in a cylinder. The **Caps** option allows you to create the circular capping surfaces along with the cylindrical surface.

Figure 9.23: Cylinder Defined by 3 Arc Nodes, Radial Gap, and Axial Delta



- **Using 3 Arc Nodes and a Height Node** You can create a cylindrical surface using three nodes which lie on a circular arc, and a fourth node to determine the height of the cylinder/frustum (see [Figure 9.24: Cyl-](#)

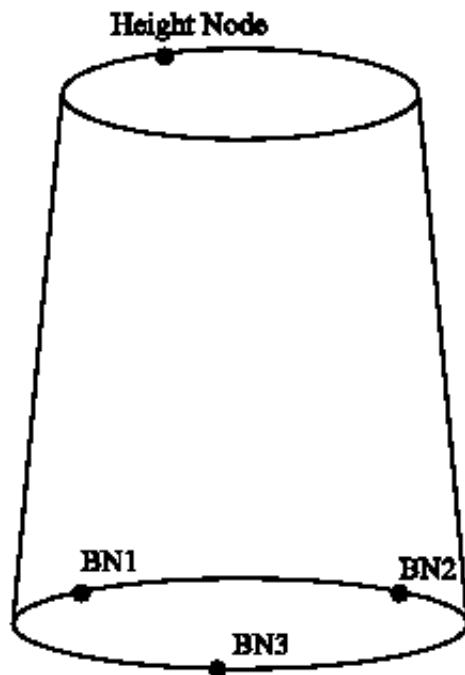
inder Defined by 3 Arc Nodes and a Height Node (p. 167)). The radii, height, and taper angle will be determined based on the nodes selected.

Note

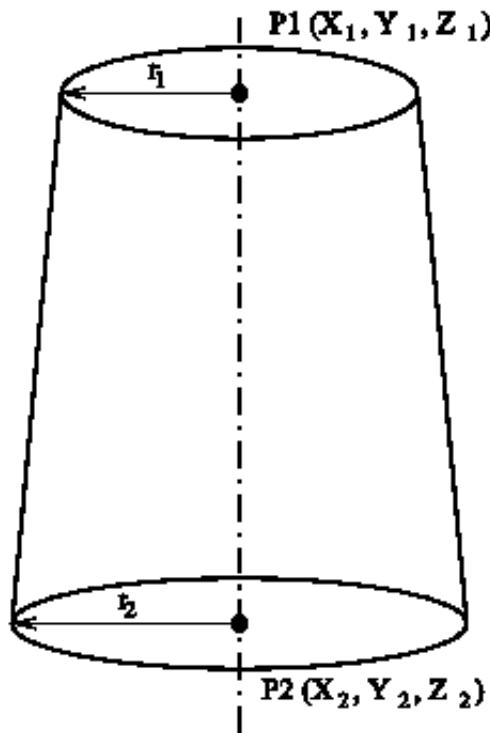
A planar annular surface will be created if the four nodes selected are in the same plane (i.e., the height is zero).

The **Caps** option allows you to create the circular capping surfaces along with the cylindrical surface.

Figure 9.24: Cylinder Defined by 3 Arc Nodes and a Height Node



- **Using 2 Axis Locations or 2 Axis Nodes** You can also create a cylindrical surface by specifying the radii (r_1, r_2) of the cylinder/frustum and two points (P1 and P2) defining the axis (see [Figure 9.25: Cylinder Defined by Axial Points and Radii \(p. 168\)](#)). Equal values of r_1 and r_2 will result in a cylinder. The axis can be defined by specifying the location (X, Y, Z) of the points or by specifying the appropriate boundary nodes corresponding to the axial points P1 and P2. The **Caps** option allows you to create the circular capping surfaces along with the cylindrical surface.

Figure 9.25: Cylinder Defined by Axial Points and Radii

9.15.3.1. Using the Cylinder Dialog Box

The procedure for creating a cylindrical surface is as follows:

1. Select the appropriate option for defining the cylinder.
 - a. For the **3 Arc Nodes** method, select the nodes on the circular arc. Enter appropriate values for **Axial Delta 1**, **Axial Delta 2**, **Taper Angle**, and **Radial Gap**.
 - b. For the **3 Arc, 1 Height Node** method, select the 3 nodes on the circular arc and the height node.
 - c. For the **2 Axis Locations** and **2 Axis Nodes** methods, specify the points defining the axis. You can specify the locations (or node IDs) manually. Alternatively, you can click the **Select Points...** (or the **Select Nodes...**) button and select the points using the mouse. Enter appropriate values for **Radius1** and **Radius2**.
2. Enter an appropriate value for **Edge Length**.
3. Enable **Caps** to create the circular capping surfaces along with the cylindrical surface.

4. Enable **Create Object** if you need to create a wrap object based on the cylinder/frustum face zone(s) created.

Note

Do not use the **Create Object** option if the cylinder/frustum is to be used as a body of influence while setting up the size functions.

5. Click **Preview** to preview the cylinder to be created.
6. When you are satisfied with the settings, click **Create** to create the cylindrical surface. Enter an appropriate zone name prefix in the **Object/Zone Prefix** dialog box and click **OK**.

9.15.4. Creating a Swept Surface

In some cases, you may want to create a swept surface by projecting an edge loop along a specified linear distance in a specified direction. You can create a swept surface using the options available in the **Swept Surface** dialog box.

9.15.4.1. Using the Swept Surface Dialog Box

The procedure for creating a swept surface is as follows:

1. Create the edge zone for the creating the swept surface.
 - Use either the **Feature Modify** dialog box or the **Surface Retriangulation** dialog box (see [Creating Edge Loops \(p. 142\)](#) for details).
 - Use the hot-key **Ctrl+Shift+I**. In node selection mode, select the nodes and use **Ctrl+I** to create the edge.

2. Open the **Swept Surface** dialog box.

Boundary → **Create** → **Swept Surface...**

3. Select the edge zone to be swept from the **Edge Zones** drop-down list.
4. Select the corresponding face(s) from the **Face Zones** selection list.
5. Specify the distance along which the edge is to be swept in the **Total Distance** field.
6. Specify the appropriate value in the **No. of Offsets** field.
7. Specify the **Vector** defining the direction in which the edge is to be swept.

Alternatively, you can click **Define** and select two nodes or positions to specify the vector. The **Total Distance** is also computed based on the nodes/positions selected.

8. Enable **Split Quad Faces**, if required.
9. Enable **Create Object** if you need to create a wrap object based on the swept surface face zone created.
10. Click **Create** to create the swept surface.

9.15.5. Creating a Revolved Surface

In some cases, you may want to create a revolved surface from specific edge zones. The revolved surface is created by revolving the selected edge zones through the angle specified using the pivot and axis of rotation defined. You can create a revolved surface using the options available in the **Revolved Surface** dialog box.

9.15.5.1. Using the Revolved Surface Dialog Box

The procedure for creating a revolved surface is as follows:

1. Create the edge zones to be used for creating the revolved surface.
 - Use either the **Feature Modify** dialog box or the **Surface Retriangulation** dialog box (see [Creating Edge Loops \(p. 142\)](#) for details).
 - Use the hot-key **Ctrl+Shift+I**. In node selection mode, select the nodes and use **Ctrl+I** to create the edge.

2. Open the **Revolved Surface** dialog box.

Boundary → **Create** → **Revolved Surface...**

3. Select the edge(s) to be revolved from the **Edge Zones** selection list.
4. Specify the appropriate value in the **Number of Segments** field.
5. Specify the angle through which the edge is to be revolved in the **Angle** field.
6. Specify an appropriate value for **Scale Factor** depending on the radius required for the revolved surface.
7. Specify the pivot point and the axis of revolution. Click **Define** and select 1-6 nodes to define the pivot and axis as follows:
 - If only 1 node is selected, the pivot point is at the node location and the axis of rotation is the global z-axis.
 - For 2 nodes, the pivot point is at the midpoint of the nodes selected and the axis of rotation is the global z-axis.
 - For 3 nodes, the pivot point is at the first node selected. The axis of rotation is the local z-axis normal to the plane defined by the three points, the positive direction is determined by the right-hand rule.
 - For 4, 5 or 6 nodes, the first 3 points define a circle. The pivot point is at the center of the circle. The axis of rotation is the local z-axis normal to the circular plane, the positive direction is determined by the right-hand rule.
8. Enable **Create Object** if you need to create a wrap object based on the revolved surface face zone created.
9. Click **Create** to create the revolved surface.

9.15.6. Creating Periodic Boundaries

If the preprocessor used to create the boundary mesh does not allow you to generate periodic boundaries that are identical and contain either face or node correspondence information, you can create the periodic boundaries in the meshing mode of ANSYS Fluent using the following sequence:

1. In the preprocessor, create only *one* of the (periodic) boundaries.
2. Assign any non-periodic boundary type to this boundary.
3. In Fluent meshing mode, change this boundary to a periodic boundary using the [Make Periodic Boundaries Dialog Box](#).
4. Create the corresponding periodic shadow boundary.

A zone type of periodic will be assigned to both the periodic and the periodic-shadow zones, and the face/node correspondence will be generated.

Note

This is the only way to create periodic boundaries in the meshing mode; it is not sufficient to simply set a zone type to be periodic.

You can specify either rotational or translational periodicity.

- For rotational periodicity, you need to enter an angle of rotation and the pivot and the axis of rotation. The pivot and axis of rotation can be defined by selecting 1-6 nodes as follows:
 - If only 1 node is selected, the pivot point is at the node location and the axis of rotation is the global z-axis.
 - For 2 nodes, the pivot point is at the midpoint of the nodes selected and the axis of rotation is the global z-axis.
 - For 3 nodes, the pivot point is at the first node selected. The axis of rotation is the local z-axis normal to the plane defined by the three points, the positive direction is determined by the right-hand rule.
 - For 4, 5 or 6 nodes, the first 3 points define a circle. The pivot point is at the center of the circle. The axis of rotation is the local z-axis normal to the circular plane, the positive direction is determined by the right-hand rule.
- For translational periodicity, you need to specify only a translational shift.

When the periodic-shadow boundary is created from the original (periodic) boundary, the nodes around the outer edges of the shadow zone will be duplicates of existing nodes. These duplicates will be marked as free, so they can be verified by counting them and drawing them. Before generating the initial mesh, you must merge these nodes.

Important

To ensure that the periodic-shadow boundary creation works properly, you must define the node distribution correctly in the preprocessor that generates the boundary mesh.

Ensure that the distribution of nodes on the boundaries that will be shared by the shadow zone and the surfaces adjacent to it is the same as the distribution on the boundaries shared by the original (periodic) zone and its adjacent surfaces.

Note

Pre release 15 files written in mesher mode will be automatically converted when read into mesher mode. These files may not contain sufficient information to properly set up periodic information in case of multiple periodic pairs.

9.15.7. Text Commands for Creating Surfaces

The text interface commands for creating surfaces are as follows:

/boundary/create-bounding-box

allows you to create the bounding box for the specified zones. You can specify the zone type, name, edge length, and the extents of the box, as required. You can also optionally create an object from the bounding box created.

/boundary/create-cylinder

allows you to create a cylinder by specifying the axis, radius, and edge length or three arc nodes, the axial delta, the radial gap, and the edge length. You can also specify the prefix for the zone being created, as required. You can also optionally create an object from the cylinder created.

/boundary/create-plane-surface

allows you to create a plane surface by specifying either the axis direction, axial location, and the extents of the surface or three points defining the plane. You can also optionally create an object from the plane surface created.

/boundary/create-revolved-surface

allows you to create a revolved surface by rotating the specified edge through the angle specified. Specify the number of segments, scale factor, and the pivot point and axis of rotation. You can also optionally create an object from the revolved surface created.

/boundary/create-swept-surface

allows you to create a surface by sweeping the specified edge in the direction specified. You need to specify the distance to sweep through and the number of offsets, as required. You can also optionally create an object from the swept surface created.

/boundary/make-periodic

allows you to make the specified boundaries periodic. You can specify the type of periodicity (rotational or translational), the angle, pivot, and axis of rotation, for rotational periodicity or the translational shift for translational periodicity.

For each of the zones specified, a corresponding periodic shadow boundary zone will be created.

9.16. Additional Boundary Mesh Text Commands

Some commands for checking and modifying the boundary mesh are not available in the GUI. The text interface commands for these operations are listed below:

/boundary/clear-marked-faces

clears faces that were marked with the /boundary/mark-face-proximity or /boundary/mark-face-intersection command.

/boundary/check-duplicate-geom

displays the names of the duplicate surfaces and prints maximum and average distance between them.

/boundary/clear-marked-nodes

clears nodes that were marked with the boundary/mark-duplicate-nodes command.

/boundary/count-unused-faces

lists the number of boundary faces that are not used by any cell.

/boundary/delete-duplicate-faces

searches for faces on a specified zone that have the same nodes and deletes the duplicates. Duplicate faces may be present if you generated your boundary mesh using a third-party grid generator, or if you used the slit-boundary-face command (described below) to modify the boundary mesh and then merged nodes. You can detect duplicate faces by displaying multiply-connected faces, using the **Display Grid** dialog box.

/boundary/delete-all-dup-faces

searches for faces on all boundary zones that have the same nodes and deletes the duplicates.

/boundary/delete-free-edge-faces

allows you to remove faces with the specified number of free edges from the specified boundary zones.

/boundary/delete-unused-faces

deletes all boundary faces that are not used by any cell. You should use this command after creating a mesh and deleting any dead zones. This command will remove any faces located between two deleted cell zones. Such faces should be removed if you are generating a mesh for ANSYS Fluent, but the solver will delete them if you do not.

/boundary/edge-limits

prints the length of the shortest and longest edges on the boundary. This information can be useful for setting initial mesh parameters and refinement controls.

/boundary/fix-mconnected-edges

resolves multi-connected edges/non-manifold configurations in the boundary mesh by deleting fringes and overlaps based on threshold values specified.

/boundary/jiggle-boundary-nodes

"randomly" perturbs all boundary nodes based on an input tolerance. Some nodes will be perturbed less than the tolerance value, while others will be perturbed by half of the tolerance value in all three coordinate directions.

/boundary/mark-duplicate-nodes

marks duplicate nodes (see [Manipulating Boundary Nodes \(p. 119\)](#) for information about duplicate nodes). The marked nodes will appear in the display when nodes are displayed. For a list of duplicate nodes, set the /report/verbosity level to 2 before using the mark-duplicate-nodes command.

/boundary/mark-face-intersection

marks intersecting faces. Intersection is detected if the line defined by any two consecutive nodes on a face intersects any face in the current domain. The marked faces will appear in the display when faces

are displayed. For a list of intersecting faces, set the `report/verbosity` level to 2 before using the `mark-face-intersection` command.

/boundary/mark-face-proximity

marks faces that are in proximity to each other. Face A is considered to be in proximity to face B if any of the nodes on face A are within the calculated proximity distance from face B. The proximity distance is calculated based on the specified relative distance and the sphere radius. The sphere radius is determined by the maximum distance from the centroid of the face to its nodes. The marked faces will appear in the display when faces are displayed. For a list of faces in proximity to each other, set the `report/verbosity` level to 2 before using the `mark-face-proximity` command.

/boundary/merge-small-face-zones

merges the face zones having area less than the specified minimum area with the adjacent zone.

/boundary/orient-faces-by-point

orients the normals based on the specified material point.

/boundary/print-info

prints information about the mesh in the text window. See [Printing Grid Information \(p. 406\)](#) for more information.

/boundary/reset-element-type

resets the element type (mixed, tri, or quad) of a boundary zone. If you have separated a mixed (tri and quad) face zone into one tri face zone and one quad face zone, for example, each of these will be identified as a "mixed" zone. Resetting the element type for each of these new zones will identify them as, respectively, a triangular zone and a quadrilateral zone.

/boundary/scale-nodes

applies a scaling factor to all node coordinates. You can use this command to change the units of the grid.

/boundary/slit-boundary-face

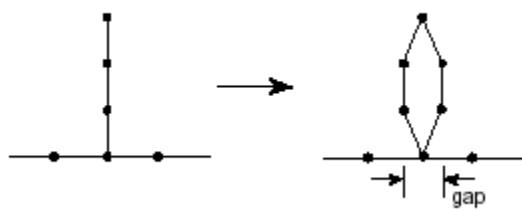
slits a boundary face zone by duplicating all faces and nodes, except those nodes that are located at the edges of the boundary zone. A displacement can be specified to provide thickness to the boundary (see [Figure 9.26: Slitting a Boundary Face Zone \(p. 175\)](#)). The slit command only works when it is possible to move from face to face using the connectivity provided by the cells.

Important

You should slit the boundary face *after* you generate the volume mesh so that the mesher does not place cells inside the gap. There may be some inaccuracies when you graphically display solution data for a grid with a slit boundary in ANSYS Fluent.

/boundary/auto-slit-faces

slits all boundary faces with cells on both sides (these cells must be in the same cell zone). A displacement can be specified to provide thickness to the boundary (see [Figure 9.26: Slitting a Boundary Face Zone \(p. 175\)](#)).

Figure 9.26: Slitting a Boundary Face Zone

Chapter 10: Wrapping Boundaries

Geometries migrated from various CAD packages often contain gaps and overlaps between the surfaces due to algorithm and tolerance differences of the CAD packages. Repairing such geometries manually is a tedious and time consuming process. The wrapper provides the ability to create reliable meshes for such geometries without extensive manual clean up and reduces the time required for preprocessing.

The wrapper can take account of gaps and overlaps at the expense of certain degree of geometry details of the model. It can handle unclean geometries and does not require a watertight representation of the geometry.

The wrapper is useful in the following industrial applications:

- Automotive
 - Underhood thermal management (engine only, front car, full car)
 - Cabin HVAC
 - External aerodynamics
 - Brake cooling and engine cooling
- Aerospace
 - Engine core compartment
 - Cockpit HVAC, cabin HVAC
 - Landing gear
- Drill bit applications
- Smoke and fire spread
- Bio-medical applications
- Other applications with bad input geometries

The following sections discuss the use of the wrapper utility and its associated parameters.

- [10.1. The Wrapping Process](#)
- [10.2. Examining and Repairing the Input Geometry](#)
- [10.3. Initializing the Cartesian Grid](#)
- [10.4. Examining the Cartesian Grid for Leaks](#)
- [10.5. Extracting the Wrapper Surface](#)
- [10.6. Checking the Quality of the Wrapper Surface](#)
- [10.7. Post Wrapping Improvement Operations](#)
- [10.8. Text Commands for the Wrapper](#)

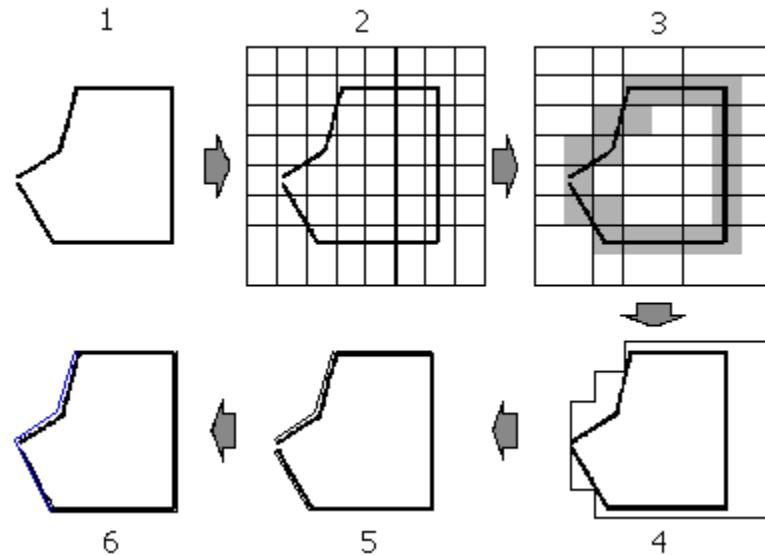
10.1. The Wrapping Process

The wrapping process is based on the Cartesian grid (or overlay grid) approach. The general procedure for creating a wrapper surface is as follows:

1. A coarse Cartesian grid is overlaid on the input geometry (including gaps and overlaps). This Cartesian grid is used to automatically clean the input geometry and to create the water-tight representation.
2. The intersection between the Cartesian grid and the input geometry is calculated and the intersecting cells are identified and marked.
3. A watertight faceted representation is created along the boundary of the intersecting cells.
4. The nodes on this faceted representation are projected on to the faces and feature edges of the input geometry which then results in a wrapper surface closely representing the input geometry.
5. The wrapper surface quality is improved by post-wrapping operations such as smoothing, swapping, etc.

[Figure 10.1: Schematic Representation of Wrapping Process \(p. 178\)](#) shows a schematic representation of the wrapping procedure.

Figure 10.1: Schematic Representation of Wrapping Process



Note

The dimensions of the distortion (e.g., hole, gap, etc.) in the input geometry should be smaller than that of the size of the Cartesian cells created by the wrapper. If there is significant distortion in the input geometry, repair it to the extent that the distortion becomes smaller in size. Large holes, if present in the initial geometry, should be filled. Otherwise such holes will be ignored in the wrapping process.

The **Boundary Wrapper** dialog box allows you to perform the wrapping operation. In addition to the **Boundary Wrapper** dialog box, you will also use the **Display Grid** dialog box during the wrapping process.

The wrapping process consists of the following tasks:

- Examining and repairing the input geometry
- Initializing the Cartesian grid
- Creating the wrapper surface
- Recovering boundary zones and features
- Using the post wrapping improve operations

10.2. Examining and Repairing the Input Geometry

Before performing the wrapping operation, you need to examine the input geometry and repair it to some extent so that the final wrapper surface is of good quality. The geometry repair operations include the following operations:

- Checking for duplicate nodes.
- Converting quad zones into tri zones.
- Detecting and filling the holes in the geometry.

Checking and Removing Duplicate Nodes

Check the geometry for free or duplicate nodes. If the mesh contains free (duplicate) nodes, delete (merge) them using the **Merge Boundary Nodes** dialog box before proceeding to the wrapping operation. Analyze the geometry and remove unnecessary details, if any.

Note

Free nodes may cause trouble during the wrapping process, because the wrapper assumes each edge to be a special feature of the geometry. Also, curvature calculations cannot be performed on disconnected edges.

Triangulate Quad Zones

Triangulate the quad zones present in the input geometry using the **Triangulate Zones** dialog box. You can also use the command /boundary/remesh/triangulate to triangulate the quad zones.

Important

The boundary wrapper does not support quad zones.

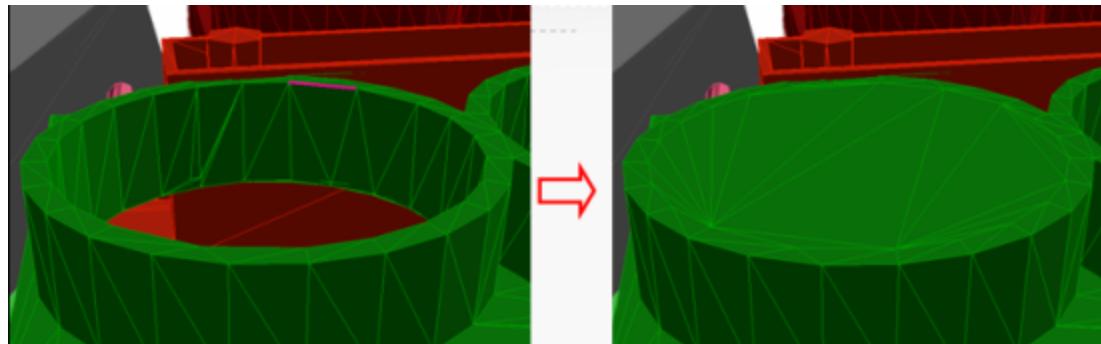
Detecting and Filling the Holes Manually

Examine the geometry and fill any unwanted holes in the geometry. A hole may be bounded by either free edges or feature edges. You will mainly try to locate and fix the major holes in the geometry. If you miss smaller holes, you can fix them later using the automatic hole detection feature.

There are several ways in which you can fix the holes in the geometry.

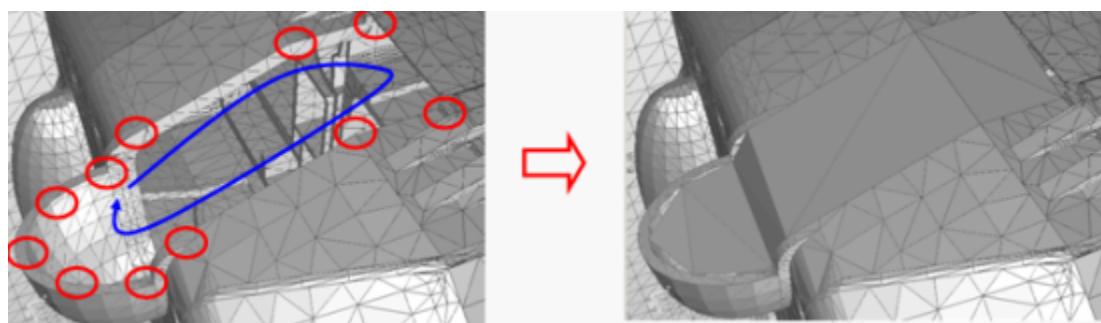
- Fixing holes by selecting edges ([Figure 10.2: Hole Filling Using Edge \(p. 180\)](#)).
 1. Select the edge filter (hot-key **Ctrl+E**).
 2. Select any edge on the hole boundary. You can select either free edges or feature edges.
 3. Press **F5** to create the surface that closes the hole.

Figure 10.2: Hole Filling Using Edge

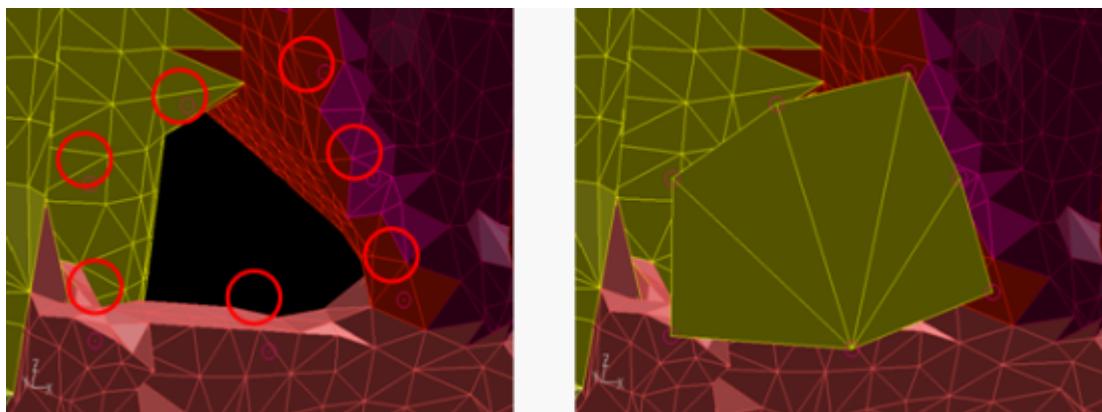


- Fixing holes by selecting nodes ([Figure 10.3: Hole Filling Using Nodes \(p. 180\)](#)).
 1. Select the node filter (hot-key **Ctrl+N**).
 2. Select the nodes around the hole.
 3. Press **F5** to create the surface that closes the hole.

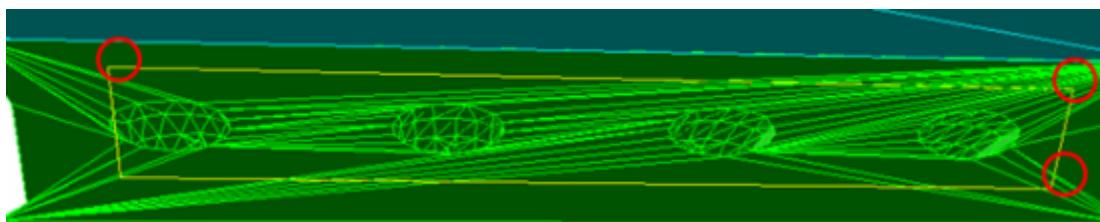
Figure 10.3: Hole Filling Using Nodes



- Fixing holes by selecting positions ([Figure 10.4: Hole Filling Using Positions \(p. 181\)](#)).

Figure 10.4: Hole Filling Using Positions

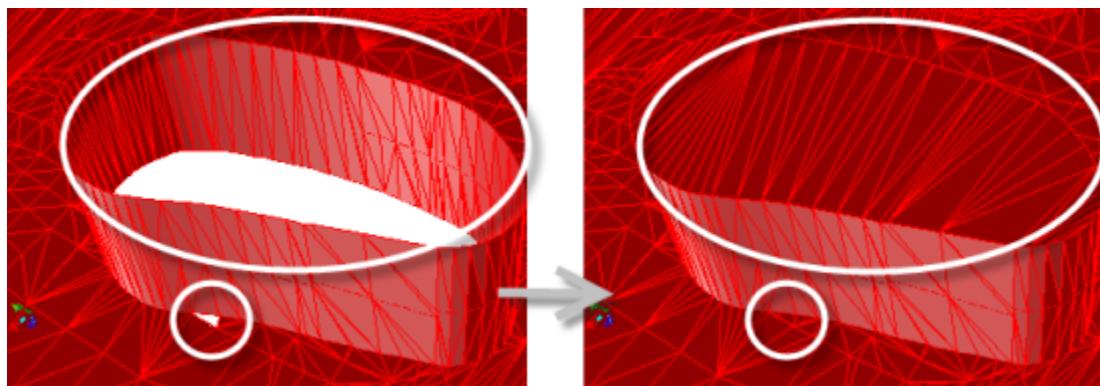
1. Select the position filter (hot-key **Ctrl+X**).
 2. Select the positions around the hole.
 3. Press **F5** to create the surface that closes the hole.
- Fixing multiple holes near each other by creating a planar surface.
 1. Open the **Plane Surface** dialog box.
Boundary → **Create** → **Plane Surface...**
 2. Select three positions.
 3. Specify appropriate mesh size and click **Create**. See [Creating a Planar Surface Mesh \(p. 164\)](#) for details.

Figure 10.5: Hole Filling Using a Plane Surface

- Fixing multiple holes in selected zone(s) ([Figure 10.6: Fixing Holes in Selected Zones \(p. 182\)](#)).
 1. Select the zone(s) to be repaired.
 2. Use the command `/boundary/modify/repair` or the hot-key **Ctrl+R**.

Note

This method will close all holes associated with free edges in the selected zone(s). This is useful for large geometries, where you can close multiple holes in a single operation.

Figure 10.6: Fixing Holes in Selected Zones

The command `/boundary/check-duplicate-geom` can be used to identify duplicate surfaces. The duplicate surfaces and the maximum and average distance between them will be printed in the console.

10.3. Initializing the Cartesian Grid

The first step in the wrapping procedure is to overlay a Cartesian grid onto the input geometry. The accuracy of the wrapping depends on the cell size distribution of the Cartesian grid. ANSYS Fluent Meshing allows you to specify the cell size of the Cartesian grid according to your requirement. Finer cells give better results, but require longer computational time. Therefore, it is very important to use a Cartesian grid of optimum cell size distribution. Before specifying these parameters, you need to identify the zones that require smaller cell sizes.

You can specify the parameters required to initialize the Cartesian grid in the **Face Size** tab of the **Boundary Wrapper** dialog box. The procedure for specifying the parameters required to initialize the Cartesian grid is as follows:

1. Open the **Boundary Wrapper** dialog box.

Boundary → **Wrap...**

The existing face zones in the geometry will be listed in the **Tri Boundary Zones** list. The zones listed in the **Tri Boundary Zones** list may be face zones in the input geometry (before wrapping) or the wrapper surface (after wrapping).

2. Specify an appropriate value for **Default Length**.

The **Default Length** is the largest allowable cell length in the Cartesian grid. Depending on the shape and size of the input geometry, ANSYS Fluent Meshing automatically computes an appropriate value for the **Default Length**. However, you can also specify the value manually.

3. Specify the **Global Size Function**.

- a. Enable **Proximity** and/or **Curvature**.

The **Proximity** and **Curvature** size functions calculate the required cell length on each facet of the input geometry within the upper (**Max Length**) and lower (**Min Length**) limit of the refinement. The Cartesian grid will be refined based on these sizes. The curvature size function

is applied whenever there is a change in angle between the two faces. The proximity size function is applied to all the faces within the zone as well as between two zones.

Enabling these options allows you to create a Cartesian grid adapted to the gaps and curvatures of the input geometry. The size functions are established while initializing the Cartesian grid.

Important

If you initialize the Cartesian grid without enabling **Proximity** or **Curvature**, a grid of uniform size (the default size) will be created throughout the domain.

You can control the sensitivity of the size functions using the following commands:

- /boundary/wrapper/set/curvature-factor
- /boundary/wrapper/set/proximity-factor

The default values of the curvature factor and the proximity factor are set to 0.5 and 0.25, respectively. If you reduce these values, the wrapper will generate a finer mesh in the affected regions.

Warning

The size functions can produce strange results if the input mesh contains cells with skewness greater than 0.99.

- b. Set **Min Length** and **Max Length** by changing the **Level** as appropriate.

The **Min Length** is the minimum allowable cell length in the Cartesian grid. A smaller size captures details more accurately and also reduces the crossover configurations, although it increases the number of holes in the geometry. Hence, it is recommended to specify a **Min Length** value less than half the required mesh size.

Panning the regions also helps in deciding the **Min Length** value. It is recommended to specify a value 3-4 times smaller than the size of an important narrow gap present in the geometry.

Refer to [Pan Regions Dialog Box \(p. 508\)](#) for details. **Max Length** is the same as the **Default Length**. If the resulting Cartesian grid is very fine, you can coarsen it at a later stage.

Cell lengths for each **Level** are determined relative to the **Default Length**. For example, if the **Default Length** is **76.46942**, then the first **Level** will have a cell length of **76.46942/2⁰**, the second will have a cell length of **76.46942/2¹**, and so on.

4. Specify the **Zone Specific Size**.

To have a finer (or coarser) mesh for a particular boundary zone in the domain, you can manage the cell size by using the parameters in the **Zone Specific Size** group box. This option creates a Cartesian grid of the specified length throughout the selected boundary zone.

- a. Select the zone(s) for which you want to specify a local cell length according to your requirement from the **Tri Boundary Zones** selection list.

You can specify the **Zone Specific Size** in terms of **Level** only.

- b. Increase (or decrease) the **Level** as required.

The cell length for each **Level** will be displayed in the **Length** box.

- c. Click **Apply** to apply the value of the **Length** specified in the **Zone Specific Size** group box to the selected boundary zone(s).
 - d. Click **Clear** to clear the previously saved size.
 - e. Click **List** to display the zone(s) and corresponding size(s) applied to them.
 - f. Click **Select** to select the boundary zones of the selected **Level** in the **Tri Boundary Zones** list. Click **Deselect** to deselect them.
5. Specify the **Local Size Function**.

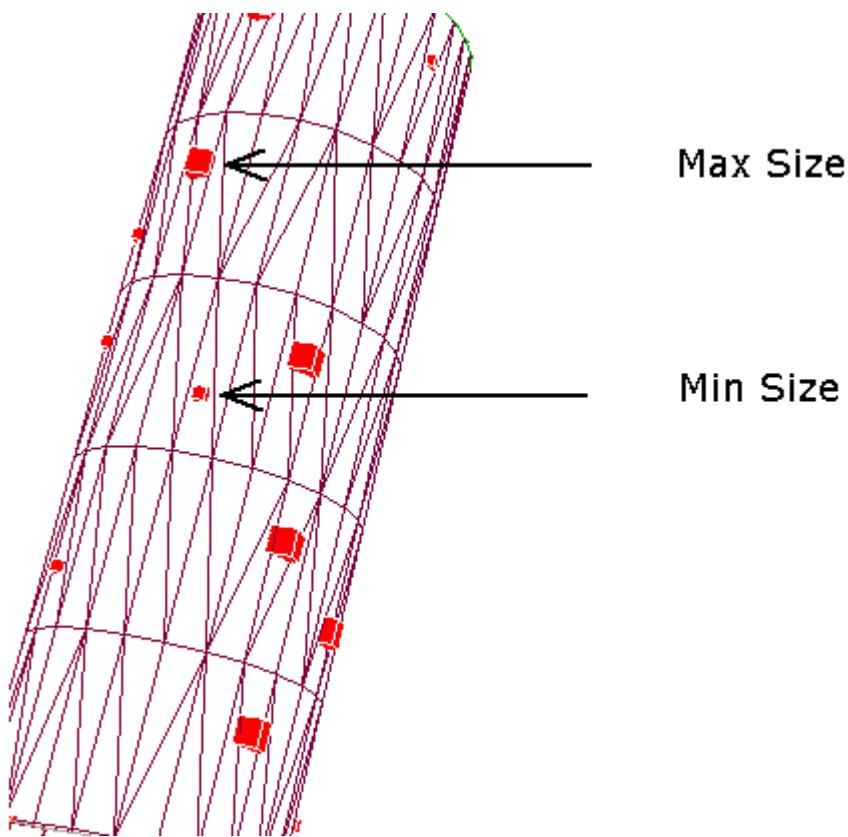
The parameters in this group box allow you to enable or disable proximity and curvature size functions locally to any boundary zone you want.

- a. Enable **Proximity** and/or **Curvature**.
 - b. Set the **Level** as appropriate.
 - c. Click **Apply**.
6. Click **Draw Sizes**.

This is to ensure whether or not the specified cell sizes are sufficient. ANSYS Fluent Meshing highlights the cells of the specified **Max Length** and **Min Length** on the selected face zone(s) in the graphics window.

- Two different sized red boxes representing the minimum and maximum cell length will be displayed on the boundary zones for which you applied **Global Size Function** and **Local Size Function**.
- Uniform sized red boxes will be displayed on the boundary zones for which you applied **Zone Specific Size**.

If the highlighted cell sizes in a zone are too large, reduce the values before creating the wrapper surface. See [Figure 10.7: Display of Min and Max Cell Sizes \(p. 185\)](#).

Figure 10.7: Display of Min and Max Cell Sizes

You can use the following command to change the number of cells to be highlighted:

```
/boundary/wrapper/set/number-of-size-boxes
```

The default value for the number of cells to be displayed is set to 20. You can increase or decrease the value as required.

7. Specify advanced options, if required.

By default, ANSYS Fluent Meshing considers self-proximity during the wrapping process. You can ignore self-proximity detection during wrapper initialization, if required. In order to have a smooth transition from fine to coarse cells, you can set the number of additional layers of cells to be included during refinement. When you specify the number of **Buffer Layers**, ANSYS Fluent Meshing marks additional layers of cells adjacent to those marked for refinement by the size requirement.

8. Select the face zone(s) you want to wrap from the **Tri Boundary Zones** list.

- If the input geometry does not have face zones of the **geometry** type and you do not select specific zone(s) in the **Tri Boundary Zones** list, all the available face zones will be considered for wrapping.
- If the input geometry has face zones of the **geometry** type and you do not select specific zone(s) in the **Tri Boundary Zones** list, only the face zones of the **geometry** type will be considered for wrapping.
- If you select specific zone(s) in the **Boundary Zones** list, irrespective of whether or not the input geometry has **geometry** type face zones, only the selected zone(s) will be considered for wrapping.

9. Click **Init** to initialize the Cartesian grid.

After creating the Cartesian grid, the wrapper virtually partitions all the cells into several volumetric regions based on the intersecting and isolated cells. These regions are ordered according to descending order of their sizes. The surface that you want to wrap (or recover) is **region:1**.

10. Display each region created during the Cartesian grid initialization process using the **Draw** button below the **Region** list in the **Region** tab of the **Boundary Wrapper** dialog box. Ensure that **region:1** is the largest region and forms a faceted watertight representation of the boundaries of the input geometry.
11. Examine **region:1** and make sure that the cell sizes are according to your requirement. If you are not satisfied with the cell size distribution of the Cartesian grid, specify appropriate values for **Zone Specific Size** and **Local Size Functions** for the individual (or required) zones to obtain an appropriate Cartesian grid in the corresponding zones.
12. Refine the Cartesian grid, if required.

Before wrapping the boundary regions, examine the configuration of the Cartesian grid around critical zones of the domain such as sharp curves and edges. If the cell size after the automated grid initialization is too large around such zones, reduce the cell sizes.

- a. Select the zone(s) for which you want to specify local size functions from the **Tri Boundary Zones** selection list.
- b. Specify appropriate **Zone Specific Size** parameters and **Local Size Function** parameters.

Important

If you specify **Zone Specific Size** as well as **Local Size Function** parameters for a particular boundary zone, the Cartesian grid will take the minimum cell size specified by both the options.

- c. Click **Refine** to update the Cartesian grid for the new set of parameters.
- d. Define local region(s) if required, and refine them. Currently, the only possible shape is a box, either aligned with the coordinate axes, or oriented as required. The region is defined by an x, y, and z range. You can specify the level of refinement required for each local region. You can visualize the regions defined using the **Draw** button in the **Wrapper Refinement Region** dialog box. The size of the elements in the defined region will be indicated in the graphics window.
- e. (optional) Refine regions for better representation of the geometry. Select the region(s) to be refined in the **Regions** selection list and click the **Refine** button.

10.4. Examining the Cartesian Grid for Leaks

When the Cartesian grid you are using to create the wrapper surface contains a hole or niche of geometry whose size is greater than the size of the Cartesian cells in that zone, a leakage or hole is created in the resulting wrapper surface. Examine the region you want to wrap for mesh density, leakages, etc. before extracting the wrapper surface. You can use either the automatic leak detection feature or fix the leaks manually and update the region accordingly.

The options available for leak detection are described in the following sections:

- 10.4.1. Automatic Leak Detection
- 10.4.2. Manual Leak Detection

10.4.1. Automatic Leak Detection

When you refine an existing region or specify additional sizing functions for better representation of the geometry, the minimum size may be reduced to a size smaller than some existing leakages. These leakages can be detected automatically using the automatic hole detection tool in ANSYS Fluent Meshing. The refinement of a region may result in the joining of previously separate regions through cells newly introduced by the refinement (region collision). While refining a region, ANSYS Fluent Meshing will automatically detect cells causing region collision and group them. Such groups of cells will be identified as holes in the region. The number of holes exposed by the current refinement will be reported in the console.

The following operations allow you to automatically detect holes:

- Refining a single region.
- Specifying local size function and additional zone-specific sizes after initialization.
- Refining local regions defined according to requirements.

The options for automatic detection and fixing of leaks are available in the **Region** tab of the **Boundary Wrapper** dialog box. The **Automatic** selection list in the **Fix Holes** group box lists the potential holes for all regions.

The generic procedure for detecting and fixing holes/leaks automatically is as follows:

1. Examine the holes for the region of interest.
 - a. Enter the appropriate value for **Region** (e.g., 1 for **region:1**) and click **Select** to select the potential holes for the region of interest.
 - b. Click **First** to zoom in to the first selected hole. You can then click **Next** repeatedly to traverse the list of selected holes.
 - c. After examining the holes of interest, click **Reset**.

Alternatively, you can select a single hole from the **Automatic** selection list and click **Zoom** to examine the hole.

2. Open all holes other than the holes for the region of interest.

The **Open** operation can be used when the identified group of cells does not represent an actual hole or is not relevant in the subsequent wrapping process. The regions associated with the hole will be merged and the hole information will be deleted.

- a. Enter the appropriate value for **Region** (e.g., 1 for **region:1**) and click **Select** to select the potential holes for the region of interest.
 - b. Click the  button to select all the holes other than those in **region:1**.
 - c. Click **Open**.
3. Select all the holes in the **Automatic** selection list and click **Fix**.

The fix operation automatically fills the holes using triangular patches. The patches created will be added into the geometry and cell-intersections will be updated accordingly. The newly added patches will serve as part of the geometry in the subsequent wrapping process.

10.4.2. Manual Leak Detection

The manual leak detection procedure involves using the **Pan Regions** dialog box to examine the region, the **Trace Path** dialog box to detect the exact location of the hole, and the various methods described in [Detecting and Filling the Holes Manually \(p. 179\)](#) to fill the hole.

Using the Pan Regions Dialog Box

The **Pan Regions** dialog box allows you to observe and analyze the region (to be wrapped) created during the Cartesian grid initialization. A plane can be passed through the selected region (or all available regions) along the X, Y, or Z direction, as required. The interior of the region(s) on every position of the plane is displayed on the plane. You can view the interior of the Cartesian grid on the plane.

You can also overlay the boundary surfaces while panning, and clip the boundary surfaces on either side of the cutting plane.

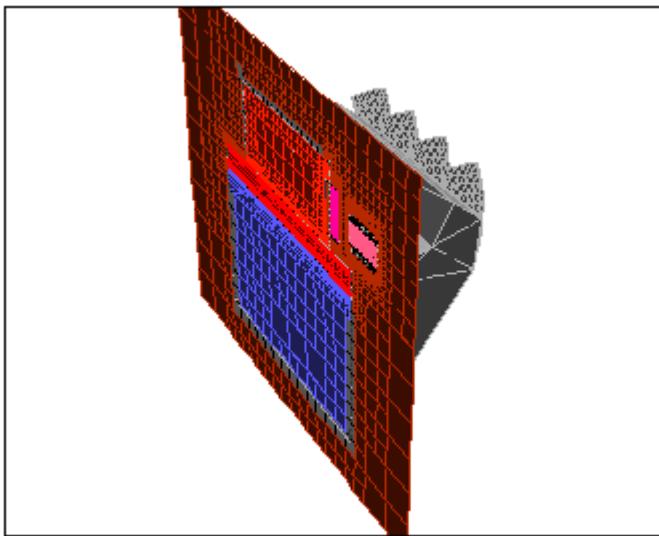
To pan through all the region(s) of your choice, do the following:

1. Display the required region (using the **Draw** button below the **Regions** list).
2. Click the **Pan Regions...** button to open the **Pan Regions** dialog box.
3. Select the appropriate axis along which you want to pan through the selected region(s) from the **Direction** list.

The **Start**, **End**, and **Increment** fields will be updated automatically based on the cell size distribution of the Cartesian grid. You can change these values as appropriate.

4. Enable **Overlay Graphics** if you want to see the geometry along with the pan plane.

Select **Positive** or **Negative** to clip the surfaces on the positive or negative side of the cutting plane. [Figure 10.8: Overlaid Geometry Clipped with the Pan Plane \(p. 189\)](#) shows the geometry on the positive side of the cutting plane.

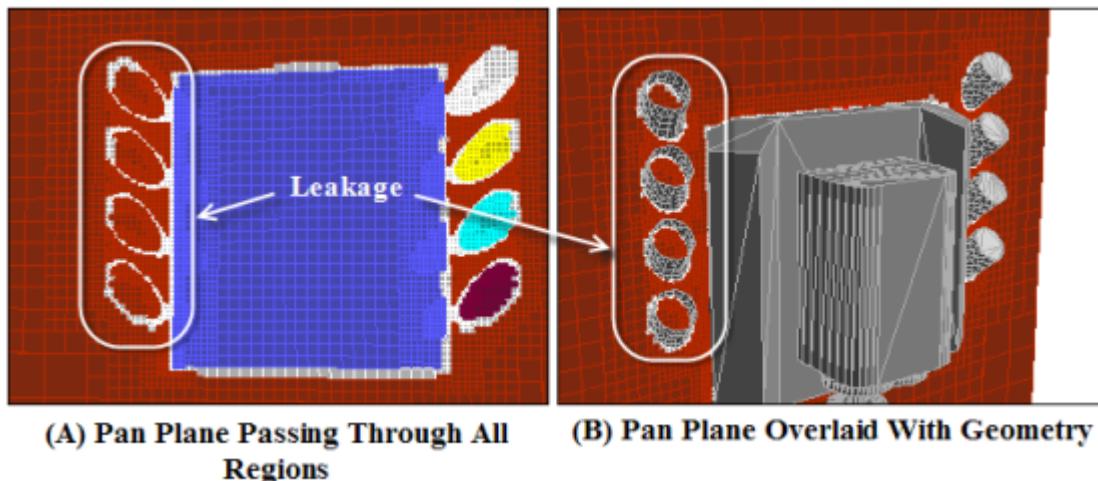
Figure 10.8: Overlaid Geometry Clipped with the Pan Plane

5. Enable **All** in the **Region** group box to pan through all regions. Alternatively, disable **All** and select the required region to pan.
6. Click **Pan** to start the plane movement through the selected region(s).

The interior of the Cartesian grid is displayed on every position of the plane during its movement through the region. Increase (or decrease) the **Increment** value to increase (or decrease) the speed of movement of the plane.

7. Use the and buttons to observe the interior of the region at a particular location. You can also move the plane to any position by moving the slider bar to any position using the mouse.

[Figure 10.9: Leak Detection Using the Pan Regions Dialog Box \(p. 190\)](#) shows how the leakage can be detected using the **Pan Regions** dialog box. When the plane passes through the Cartesian grid the interior of the Cartesian grid is displayed at every position on the plane. If at any position of the plane the color of the region is seen inside the geometry, there may be a leak or hole in the Cartesian grid.

Figure 10.9: Leak Detection Using the Pan Regions Dialog Box

8. Adjust the plane at the position where the leak is seen on the plane.

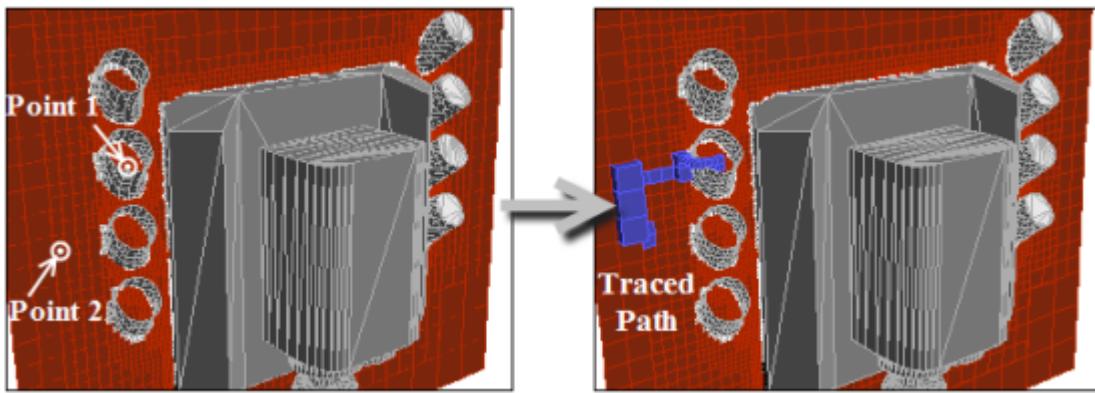
Warning

The **Pan Regions** dialog box only allows you to know whether or not there is a leak or hole in the geometry. If the geometry has a hole or leak, you need to find its exact location and fill it.

Using the Trace Path Dialog Box

To locate the hole or leak using the **Trace Path** dialog box, do the following:

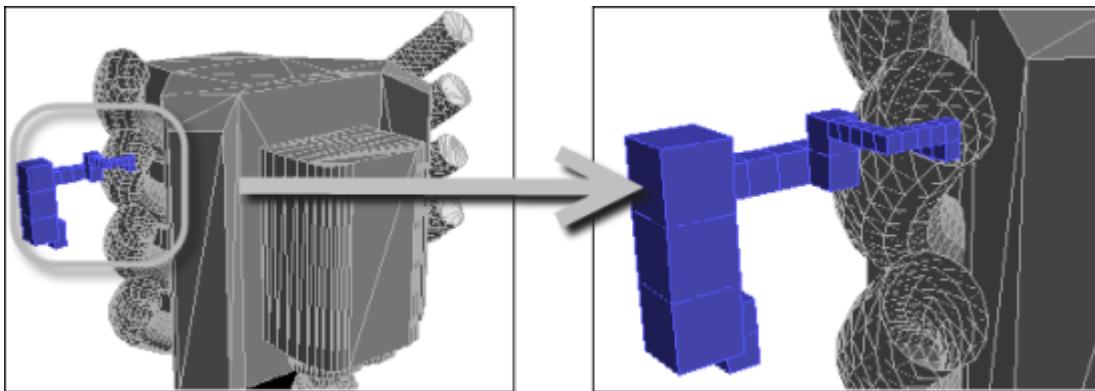
1. Display the pan plane at a position where there is an indication of a possible leak. You can overlay the geometry to allow easier location of the leak.
2. Click the **Trace Path...** button (in the **Region** tab of the **Boundary Wrapper** dialog box) to open the **Trace Path** dialog box.
3. Specify a pair of points, one “inside” and the other “outside” the geometry (see [Figure 10.10: Trace Path \(p. 191\)](#)). You can either enter the locations manually, or click the **Select Points...** button and select the points using the mouse.
4. Click **Trace** ([Figure 10.10: Trace Path \(p. 191\)](#)).

Figure 10.10: Trace Path

The path connecting the cells corresponding to the points specified will be highlighted. This traced path will always pass through the hole/leak.

5. Display the original geometry and click the **Trace** button in the **Trace Path** dialog box (Figure 10.11: Display of Geometry with Highlighted Trace Path (p. 191)).

The traced path will now be visible along with the original geometry. You can zoom in to the region where the trace path enters the geometry (Figure 10.11: Display of Geometry with Highlighted Trace Path (p. 191)).

Figure 10.11: Display of Geometry with Highlighted Trace Path

6. Close the hole or leak using the methods described in [Detecting and Filling the Holes Manually \(p. 179\)](#).
7. Select the surfaces in the **Tri Boundary Zones** selection list and click the **Update Regions** button in the **Manual** group box in the **Boundary Wrapper** dialog box.

The regions will be updated to account for the fixed hole. You can click the **Trace** button in the **Trace Path** dialog box to verify that the region has been updated for the fixed hole. ANSYS Fluent Meshing will report that there is no path between the two positions.

8. Examine the updated region for further leaks or holes. Fill any remaining leaks or holes before proceeding.

10.5. Extracting the Wrapper Surface

In this section you will learn how to extract the initial wrapper surface and imprint it on the feature edges of the input geometry.

1. Select the region to be wrapped and click the **Wrap** button (below the **Region** list) to extract the initial wrapper surface.

Alternatively, you can select the region to be wrapped and click the **Extract** button to extract the interface. The extracted interface will be available in the **Interface** list.

2. Select the appropriate interface in the **Interface** list and click **Wrap**.

You may want to clear the regions created during the initialization of the Cartesian grid before proceeding to extract the wrapper surface. Deleting the regions reduces the peak memory and improves the speed of further operations.

When creating the wrapper surface directly from the region, you will be asked if you want to delete all the regions. Click **Yes** in the **Question** dialog box to clear the regions. When creating the wrapper surface from an interface, click **Clear** (below the **Region** selection list) before extracting the initial wrapper surface from the interface.

ANSYS Fluent Meshing creates a wrapper surface having a face zone type **wrapper**. The newly created wrapper surface will be listed in the **Tri Boundary Zones** selection list. It takes the name, **wrapper-surf-xx** (where, **xx** is a random number). The wrapper surface can be displayed using the either the **Display Grid** dialog box or the **Boundary Wrapper** dialog box.

Warning

You can perform the imprinting and post wrapping improvement operations only on surfaces having the type **wrapper**. If the mesh file contains the wrapper surface created in TGrid 4.0, you need to change the type of the surface from **internal** to **wrapper** before performing these operations. An error will be reported if you perform operations like imprinting features or post wrapping improvements on a surface having type other than **wrapper**.

It is a good practice to save the mesh at this point.

3. Modify the wrapper surface to represent the input geometry more closely.

The initial wrapper surface created will not be the exact representation of the input geometry. Some of the distinctive features such as sharp edges between the zones will be ignored. In this step, you will recover the feature edges of the input geometry and imprint the wrapper surface onto these edge loops.

- a. Select the zones for which you want to recover the feature edges from the **Tri Boundary Zones** selection list.
- b. Select the method to be used for feature extraction from the **Feature Methods** list. Refer to [Creating Edge Loops \(p. 142\)](#) for details on the feature extraction methods.
- c. Click **Extract Features**.

The edge loops (edge zones) will be created for the selected boundary zones. These edge loops will be listed in the **Edge Zones** list. You can display each of them by selecting each one and clicking the **Draw** button.

- d. Modify/create feature edges using advanced options using the **Feature Modify** dialog box, if required. Click the **Feature Modify...** button to open the **Feature Modify** dialog box. Refer to [Creating Edge Loops \(p. 142\)](#) and [Modifying Edge Loops \(p. 145\)](#) details about using this dialog box.
- e. Determine the feature edges onto which the wrapper surface will be projected.

You can use the **Filter Features** dialog box to automatically delete feature edges which are beyond a specified distance from the wrapper surface. Only those features within the specified distance will be retained for future use.

- i. Click **Filter Features...** to open the **Filter Features** dialog box.
- ii. Specify appropriate values for **Absolute Distance** and **Relative Distance**.
- iii. Click **Filter**.

The larger of the local values calculated based on the absolute and relative distances specified will be used. All feature edges beyond the calculated value will be automatically deleted from the **Edge Zone(s)** list.

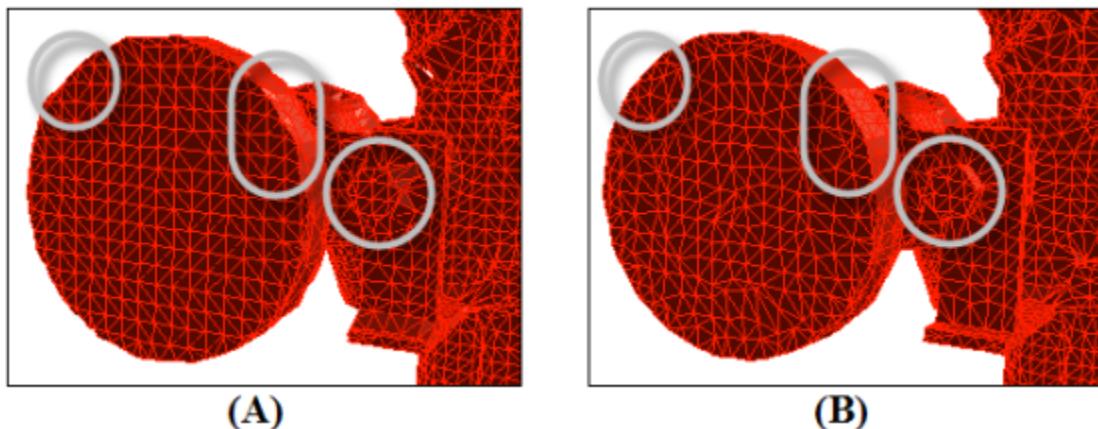
Alternatively, you can use the command /boundary/wrapper/post-improve/filterout-far-features and specify the appropriate values.

- f. Project the nodes of the wrapper surface on the edge loops.
 - i. Select the wrapper surface (**wrapper-surf-xx**) in the **Tri Boundary Zones** list.
 - ii. Select the edge loops on which you want to project nodes of the initial wrapper surface in the **Edge Zones** selection list.
 - Deselect the edges to be ignored during imprinting from the **Edge Zones** list.
 - iii. Click **Imprint**.

Imprinting is carried out by tracking a node path on the wrapper surface corresponding to a feature edge in the geometry. The nodes on the wrapper surface close to the feature edge are projected onto the edge loop. [Figure 10.12: Wrapper Surface \(A\) Before and \(B\) After Imprinting \(p. 194\)](#) shows the effect of imprinting on the wrapper surface.

Important

You may have to click **Imprint** more than once.

Figure 10.12: Wrapper Surface (A) Before and (B) After Imprinting

The nodes outside the critical range will not be projected because the projection of such nodes creates flipped or highly skewed triangles.

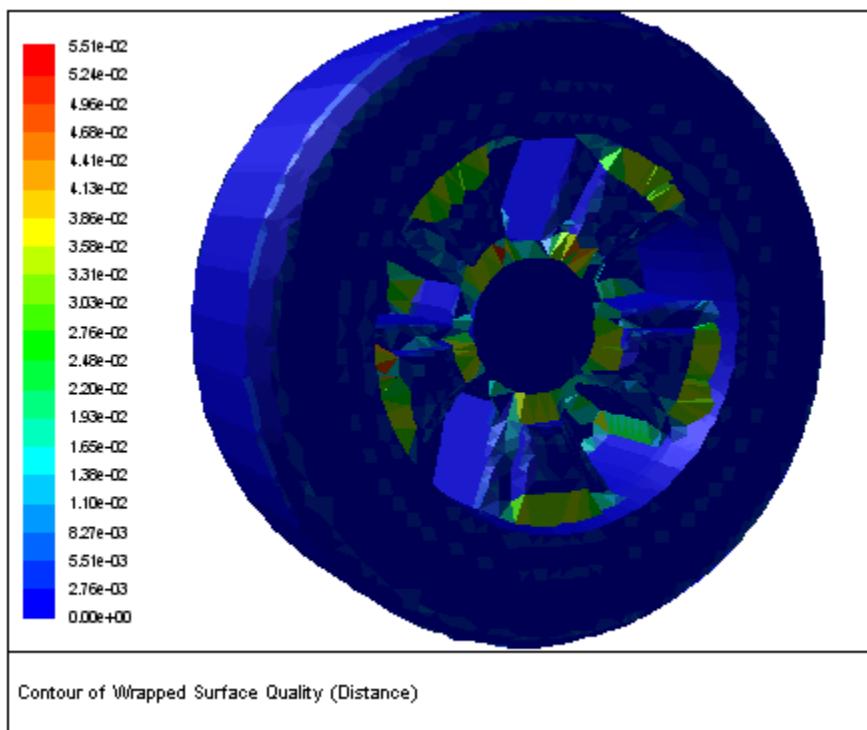
If the features are not extracted properly after the imprint operation, you can separate the wrapper surface zone by zone, and click **Imprint** by selecting all the separated zones at a time. This is useful when handling a large and complicated mesh.

10.6. Checking the Quality of the Wrapper Surface

The quality of the wrapper surface can be checked using the options available in the **Reports** tab. You can display the contours of **Distance**, **Normal**, or **Composite** and check the deviation of the wrapper surface from the original geometry. You can also identify problems like crossover configurations, insufficient geometry representation using the contours of quality.

1. Click the **Reports** tab and select the wrapper surface in the **Tri Boundary Zones** list.
2. Enable **Node Value** and/or **Auto Range** as appropriate in the **Contour Options** group box.
3. Select **Distance**, **Normal**, or **Composite** as appropriate in the **Contours of** drop-down list.
4. Set the **Levels** as appropriate and click **Draw Contours**.

[Figure 10.13: Contours of Distance of a Wrapper Surface \(p. 195\)](#) shows a sample display of contours of distance of the wrapper surface from the input geometry. The red patches in the display indicate that at such locations wrapper surface does not exactly represent the input geometry. You will use the available post wrapping operations to improve the wrapper surface at such locations.

Figure 10.13: Contours of Distance of a Wrapper Surface

10.7. Post Wrapping Improvement Operations

The wrapper surface created after imprinting is of good quality and it represents the input geometry very well in most regions. However, you can improve it further in some regions of the geometry such as sharp corners and curves. The post wrapping improvement operations allow you to improve the wrapper surface by performing various operations such as smoothing, swapping, inflating thin regions, removing crossovers, etc.

When you perform a particular post wrap operation, the completion of the operation performed will be indicated (in the console) along with the zone(s) on which the operation was performed.

The improvement operations are available in the **Post Improve** tab in the **Boundary Wrapper** dialog box. They are grouped as follows:

- [10.7.1. Coarsening Options](#)
- [10.7.2. Post Wrap Options](#)
- [10.7.3. Zone Options](#)
- [10.7.4. Expert Options](#)

10.7.1. Coarsening Options

The wrapper surface created may be finer than you require in some regions. You can coarsen the mesh in such regions or globally for the entire wrapper surface. This operation also reduces the cell count of the mesh, thereby reducing the computation time.

To coarsen the wrapper surface, do the following:

1. Select the boundary zone(s) that you want to coarsen.
2. Select **Coarsen** in the **Options** drop-down list.

3. Enter an appropriate value for **Edge Length Change**.

The **Edge Length Change** value specifies the allowable change in edge length during the coarsening operation.

4. Enter an appropriate value for **Max Angle Change**.

The **Max Angle Change** value specifies the maximum allowable change in the angle between adjacent faces during the coarsening operation. This allows you to prevent the appropriate curvatures from over coarsening.

5. Specify appropriate values for **Min Length** and **Max Length**.

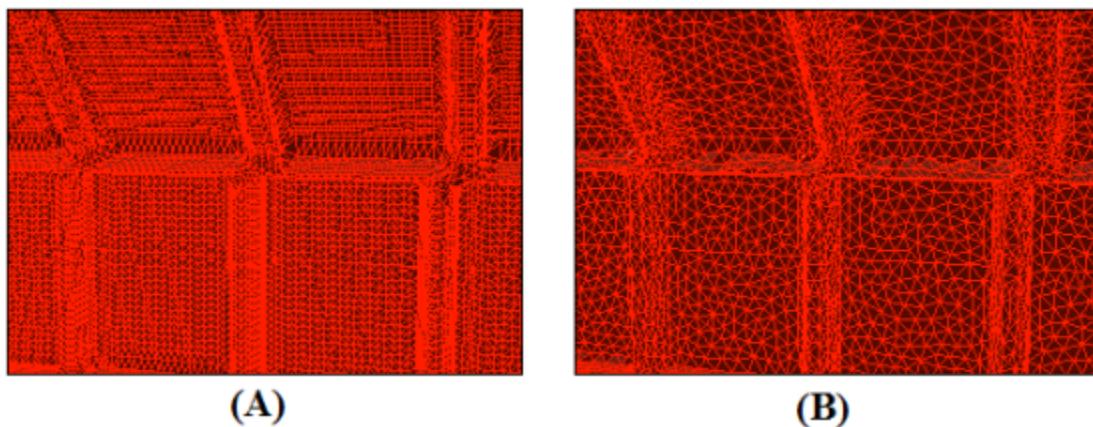
These values specify the minimum and maximum edge length. Only faces within this range will be considered for coarsening.

6. Enable **Preserve Boundary**, if required.

7. Click **Apply**.

[Figure 10.14: Wrapper Surface \(A\) Before and \(B\) After Coarsening \(p. 196\)](#) shows an example of the coarsening of the wrapper surface.

Figure 10.14: Wrapper Surface (A) Before and (B) After Coarsening



10.7.2. Post Wrap Options

The following post wrapping operations are available:

- Auto post wrap
- Removing duplicate nodes
- Smoothing
- Smoothing folded faces
- Inflating thin regions
- Removing self intersections
- Deleting islands

- Swapping
- Improving the quality

Auto Post Wrap

You can use a pre-defined sequence of post wrapping operations to improve the wrapper surface. The automatic post wrapping procedure performs the various post wrapping operations in a pre-defined sequence based on the parameters specified in the **Boundary Wrapper** dialog box. You can also use the individual post wrapping operations in the order required as described in the subsequent sections.

To use the automatic post wrapping option, do the following:

1. Select the wrapper surface in the **Tri Boundary Zones** list.
2. Select **Post Wrap** in the **Options** drop-down list and **Auto Post Wrap** in the **Sub Options** list, respectively.
3. Enter an appropriate value for **Distance Tolerance**.

The value specified for **Distance Tolerance** will be considered for the removal of duplicate nodes on the wrapper surface.

4. Enter an appropriate value for **Relative Thickness**.

The value specified for **Relative Thickness** will be considered for the removal of self-intersecting surfaces on the wrapper surface.

5. Enter an appropriate value for **Critical Thickness**.

The value specified for **Critical Thickness** will be considered for the inflating of thin regions on the wrapper surface.

6. Enter an appropriate value for **Max Improve Angle**.
7. Click **Apply**.

The operations performed and the quality at the end of the auto post wrapping will be reported in the console.

Removing Duplicate Nodes

The initial wrapper surface created after imprinting may contain duplicate nodes (two nodes having the same Cartesian coordinates). Multiple copies of the same node may cause problems in connectivity of the final wrapper surface. To remove the duplicate nodes, do the following:

1. Select the wrapper surface in the **Tri Boundary Zones** selection list.
2. Select **Post Wrap** in the **Options** drop-down list and **Remove Duplicate** in the **Sub Options** list, respectively.
3. Enter an appropriate value for **Distance Tolerance**.

The **Distance Tolerance** specifies the minimum distance between two distinct nodes on the wrapper surface. If the distance between two nodes is less than the value specified, the nodes will be considered to be duplicate nodes.

4. Click **Mark** to highlight the duplicate nodes on the wrapper surface. Alternatively, you can use the command /boundary/mark-duplicate-nodes to view the duplicate nodes.
5. Click **Apply**.

The nodes within the specified tolerance value will be removed.

Smoothing

This operation improves the skewness of the faces in the wrapper surface. Smoothing allows you to reposition the nodes to improve the cell connectivity and quality. To perform node smoothing, do the following:

1. Select the wrapper surface in the **Tri Boundary Zones** selection list.
2. Select **Post Wrap** in the **Options** drop-down list and **Smooth** in the **Sub Options** list.
3. Specify the number of smoothing attempts in the **Iteration** field.
4. Enter an appropriate value for **Relaxation**.
5. Enable **Reprojection**, if required.
6. Specify an appropriate value for **Rel Reproject Range**.

The smoothing operation repositions the nodes on the wrapper surface. The new position of the node may not be on the original geometry. This parameter specifies the maximum allowable distance between the new location of the node and the original geometry. The local mesh size (say **D**) is computed by averaging edge lengths at the node. If the new node position is within the value **Rel Reproject Range** $\times D$, the node will be projected back on the original geometry. Other nodes are left at their locations.

7. Enable **Preserve Boundary**, if required.

If you perform the smoothing operation after separating the zones of the wrapper surface, the nodes on the boundaries of the separated zones will not be moved or repositioned during smoothing.

8. Click **Apply**.

Smoothing Folded Faces

The presence of folded faces on the wrapper surface may cause problems in calculating the CFD solution. You can resolve folded faces by performing node smoothing and reprojection of nodes on the original geometry. To smooth the folded faces, do the following:

1. Select the wrapper surface in the **Tri Boundary Zones** selection list.
2. Select **Post Wrap** in the **Options** drop-down list and **Smooth Folded** in the **Sub Options** list.
3. Specify the number of attempts to be made to resolve the folded faces in the **Iteration** field.

4. Specify an appropriate value for **Critical Angle**.

The critical angle is the angle between adjacent faces. If the feature angle between the two faces is less than the value specified, the faces are considered as folded. The default value for this parameter is 30. You can specify an appropriate value for the critical angle, however, it is recommended that you not specify a very large value.

5. Enable **Reprojection** to allow projection of nodes onto the original geometry, if required.

6. Click **Apply**.

Inflating Thin Regions

If the geometry to be wrapped contains regions of very small thickness (e.g., zero-thickness walls), the wrapper surface around it forms a thin region configuration. Such type of configurations may cause problems in further volume meshing.

The **Inflate Thin Regions** option allows you to resolve such configurations by pushing apart the nodes of the faces very close to each other. To inflate thin regions, do the following:

1. Select the wrapper surface in the **Tri Boundary Zones** selection list.
2. Select **Post Wrap** in the **Options** drop-down list and **Inflate Thin Regions** in the **Sub Options** list.
3. Specify an appropriate value for **Critical Thickness**.

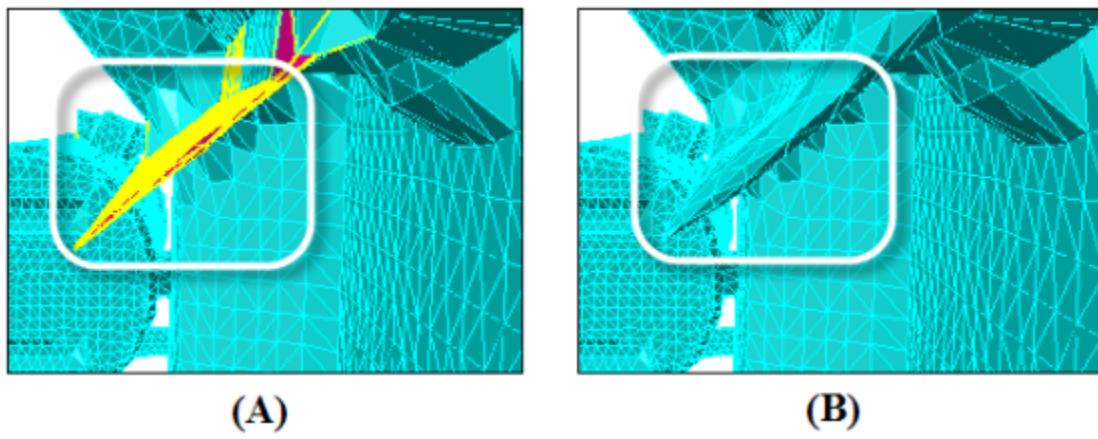
The critical thickness is the maximum distance the nodes can be moved away from each other.

4. Specify an appropriate value for **Critical Angle**.

The critical angle is the angle between the faces below which the faces are considered for inflating. Increasing this value will include more faces to be inflated.

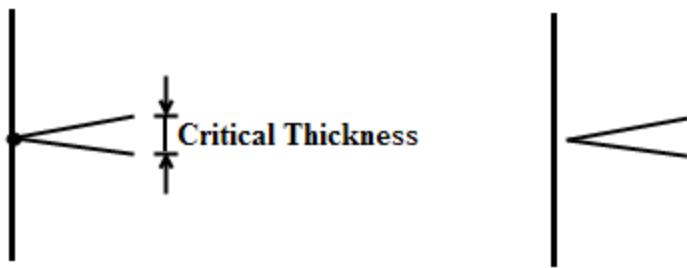
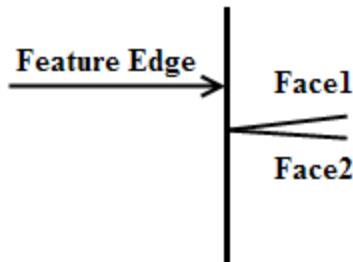
5. Enable **Preserve Boundary**, if required.
6. Click **Mark** to highlight the thin regions.
7. Click **Apply**.

[Figure 10.15: Wrapper Surface \(A\) Before and \(B\) After Inflating Thin Regions \(p. 200\)](#) shows the inflating of thin regions on the wrapper surface.

Figure 10.15: Wrapper Surface (A) Before and (B) After Inflating Thin Regions

In Figure 10.16: Preserving the Feature Edge While Inflating Thin Regions (p. 200) , **Face1** and **Face2** are the faces of the wrapper surface on either side of a thin region. The angle between the two faces is less than the **Critical Angle** specified.

- When inflating the thin region, the nodes of the faces are moved away from each other and the marked region will be inflated.
- If you enable **Preserve Boundary**, the node on the feature edge will not be moved.
- If you disable **Preserve Boundary**, the node on the feature edge will also be moved.

Figure 10.16: Preserving the Feature Edge While Inflating Thin Regions**(A) Using Preserve Boundary****(B) Without Preserve Boundary**

Removing Self Intersections

ANSYS Fluent Meshing removes most invalid configurations and improves the face quality when you smooth and swap the wrapper surface. However, configurations such as self-intersecting faces may remain. To remove self-intersecting faces, do the following:

1. Select the wrapper surface in the **Tri Boundary Zones** selection list.
2. Select **Post Wrap** in the **Options** drop-down list and **Remove Self Intersection** in the **Sub Options** list.
3. Specify the number of attempts for resolving self-intersecting configurations in the **Iteration** field.
4. Specify an appropriate value for **Relative Thickness**.

The relative thickness is the maximum distance by which the self-intersecting faces should be separated.

5. Click **Apply**.

ANSYS Fluent Meshing will check for the presence of self-intersecting faces and report the number of such faces in the console. The self-intersecting configurations will be removed from the wrapper surface. Most self-intersecting configurations will be removed, but there may be some faces that may not be resolved. Such faces will also be reported in the console.

Deleting Islands

The **Delete Islands** option allows you to delete faces in a non-contiguous region of a face zone. You can delete islands having a face count less than or equal to the specified critical face count. The critical face count is determined from the values specified for **Absolute Count** and **Relative Count**. The value for the critical face count is taken as the absolute count specified or the product of the relative count specified and the face count of the largest region, whichever is lower.

To delete island faces, do the following:

1. Select the wrapper surface in the **Tri Boundary Zones** list.
2. Select **Post Wrap** in the **Options** drop-down list and **Delete Island** in the **Sub Options** list.
3. Enter appropriate values for **Absolute Count** and **Relative Count** in the **Critical Face Count** group box.
4. Click **Apply** to delete the island faces.

Swapping

Swapping is another operation to improve the skewness of the faces in the wrapper surface. It allows you to swap the common edge of two adjacent triangles into the other diagonal directions. The swapping is carried out only when the two triangles form an angle close to planar and the swapped configuration gives better skewed cells.

To perform swapping on the wrapper surface, do the following:

1. Select the wrapper surface in the **Tri Boundary Zones** selection list.

2. Select **Post Wrap** in the **Options** drop-down list and **Swap** in the **Sub Options** list.
3. Specify an appropriate value for **Max Angle**.

The maximum angle specifies the maximum angle between two adjacent face normals. This restriction prevents the loss of sharp edges in the geometry while swapping wrapper surface.

4. Specify an appropriate value for **Max Skew**.

All the faces having a skewness value greater than the specified value will be swapped.

5. Click **Apply**.

Improving the Quality

This **Improve** operation allows you to further improve the quality of the wrapper surface.

To improve the wrapper surface, do the following:

1. Select the wrapper surface in the **Tri Boundary Zones** selection list.
2. Select **Post Wrap** in the **Options** drop-down list and **Improve** in the **Sub Options** list.
3. Select the appropriate option from the **Quality Measure** drop-down list.
4. Specify the number of attempts to improve the wrapper surface quality in the **Iteration** field.
5. Specify an appropriate value for **Max Angle**.

The maximum angle is the maximum allowable angle between two adjacent face normals. This restriction prevents the loss of sharp edges in the geometry while improving the wrapper surface.

6. Specify an appropriate value for **Quality Limit**.

The quality limit is the limit above which the wrapper surface will be improved.

7. Click **Apply**.

10.7.3. Zone Options

The wrapper surface created comprises only a single boundary zone. It is a good practice to separate the wrapper surface into several zones based on the zones of the original geometry. This is particularly useful when boundary conditions are to be applied for certain zones during the fluid simulation. You will then rezone the separated wrapper zones for better recovery of the zones in the geometry. You can also rename the zones as required.

Recovering Zones

The wrapper surface will be separated into boundary zones based on the zones in the original geometry. When you separate the wrapper surface into different zones, the boundary zones associated with the original geometry will be retained and the newly created boundary zones will be prefixed by "wrap". For example, for the boundary zone `rear-wing`, the corresponding zone created from the wrapper surface will be `wrap-rear-wing:xx` (where, `xx` is a random number). The original wrapper surface will be deleted.

To separate the zones of the wrapper surface, do the following:

1. Select **Zone** in the **Options** drop-down list and **Recover Zone** in the **Sub Options** list.
2. Click **Apply**.

This operation separates the wrapper surface into several zones based on the input geometry. The newly created zones will be listed in the **Tri Boundary Zones** selection list, prefixed by "wrap".

The boundaries of the separated zones are not smooth, you need to smooth them using the **Rezone** option.

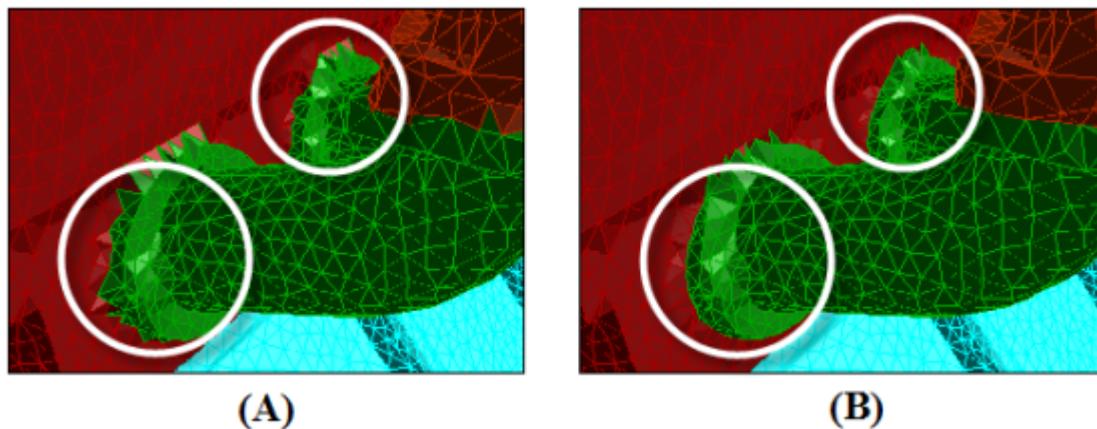
Rezoning Faces

The zones separated from the single wrapper surface can be smoothed by adjusting the node positions on the zone boundaries. Very small zones (islands) will be merged to bigger adjacent zones. To rezone the faces on the separated wrapper zones, do the following:

1. Select the boundary zone(s) to be rezoned in the **Tri Boundary Zones** selection list.
2. Retain the selection of **Zone** in the **Options** drop-down list and select **Rezone** in the **Sub Options** list.
3. Enable **Reprojection**, if required. This is useful in recovering the distinctive zone boundary in the original geometry.
4. Click **Apply**.

Multiple rezone operations may be required for better recovery of the boundaries of the wrapper zones. You may also need to rezone each wrapper zone separately.

Figure 10.17: Wrapper Surface (A) Before and (B) After Rezoning



Renaming Zones

By default, the separated wrapper zones are prefixed by "wrap- " to distinguish them from the original geometry. To rename the wrapper surfaces, do the following:

1. Select the surfaces to be renamed in the **Tri Boundary Zones** list.

2. Retain the selection of **Zone** in the **Options** drop-down list and select **Rename** in the **Sub Options** list.
3. Specify the prefix as appropriate in the **From** and **To** fields in the **Change Prefix** group box.
4. Click **Apply**.

10.7.4. Expert Options

The **Expert** options contain the following advanced post wrapping improvement options:

- Auto post improvement
- Recovering a single surface
- Removing crossovers

Auto Post Improvement

You can use a pre-defined sequence of improvement operations to improve the wrapper surface after recovering and rezoning the wrapper surface based on the original geometry. The improvement operations include the splitting and merging of nodes to remove small areas, improving the boundary surface quality, collapsing skewed faces, and removing duplicate and intersecting faces. The automatic improvement procedure performs these improvement operations in a pre-defined sequence based on the parameters specified in the **Boundary Wrapper** dialog box.

To use the automatic post improvement procedure, do the following:

1. Select **Expert** in the **Options** drop-down list and **Auto Post Improve** in the **Sub Options** list, respectively.
2. Enter an appropriate value for **Min Triangle Area**.
3. Enter appropriate values for **Max Aspect Ratio**, **Max Skewness**, and **Max Size Change**.
4. Click **Apply**.

The improvement operations performed and the quality at the end of the auto improvement will be reported in the console.

Recovering a Single Surface

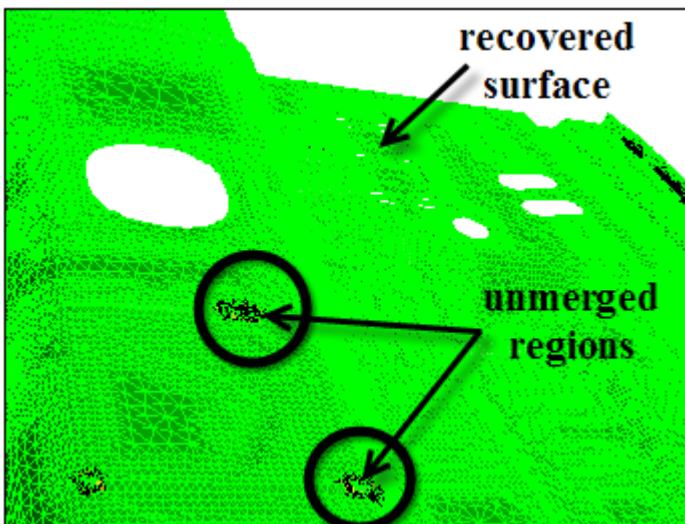
Certain geometries may have sheet metal components which have a small thickness and which are connected to other components. The post wrapping improvement options include the recovery of such thin sheets as a single surface after wrapping. You can either use the **Automatic** option to mark the thin parts or manually select the faces on opposite sides of the thin part. The marked faces will then be merged to obtain a single surface. You can also remove unmerged island regions after recovering the surfaces. To recover a single surface for thin parts, do the following:

1. Select **Expert** in the **Options** drop-down list and **Recover Single Surface** in the **Sub Options** list, respectively.
2. Select **Single Surface** in the **Recover Options** list.
3. Enable **Automatic** and click **Mark** to mark the faces to be merged in thin parts. Alternatively, disable **Automatic** and click the **Select** button in the **Parameters** group box. Select the seed faces as appropriate and click **Mark** to mark the faces to be merged.

4. Specify appropriate values for **Critical Thickness** and **Critical Angle**.
5. Click **Merge** to merge the thin surfaces into a single surface.

[Figure 10.18: Recovering a Single Surface \(p. 205\)](#) shows the recovered surface for a thin part. Enable the display of multiply-connected faces in the **Display Grid** dialog box to see the remaining unmerged regions.

Figure 10.18: Recovering a Single Surface



The unmerged regions can be removed using the **Post Single Surface** option.

6. Select **Post Single Surface** in the **Recover Options** list.
7. Enter an appropriate value for **Max Face Count** and click the **Auto** button in the **Parameters** group box to automatically remove the remaining unmerged regions.

Most unmerged faces will be deleted when the automatic option is used. Any remaining unmerged faces can be deleted manually. Select the seed face as appropriate and click the **Mark** button in the **Parameters** group box. The regions will be marked and the number of faces to be deleted will be reported. Click **Delete** to delete the marked faces.

Removing Crossovers

If the geometry has small gaps between two separate faces, the wrapper can create crossover configurations. The crossovers are unwanted faces that connect parts of the wrapper surface separated by a small gap. The crossover faces have one vertex on one side of the gap and the other two nodes on the other side of the gap. You need to remove the crossovers to make the wrapper surface represent the original geometry precisely.

You can remove crossover faces either automatically or manually. To remove crossover configurations, do the following:

1. Select the wrapper surface in the **Tri Boundary Zones** selection list.
2. Select **Expert** in the **Options** drop-down list and **Remove Crossover** in the **Sub Options** list.
3. Enable **Automatic** and click **Apply** to automatically remove crossovers.

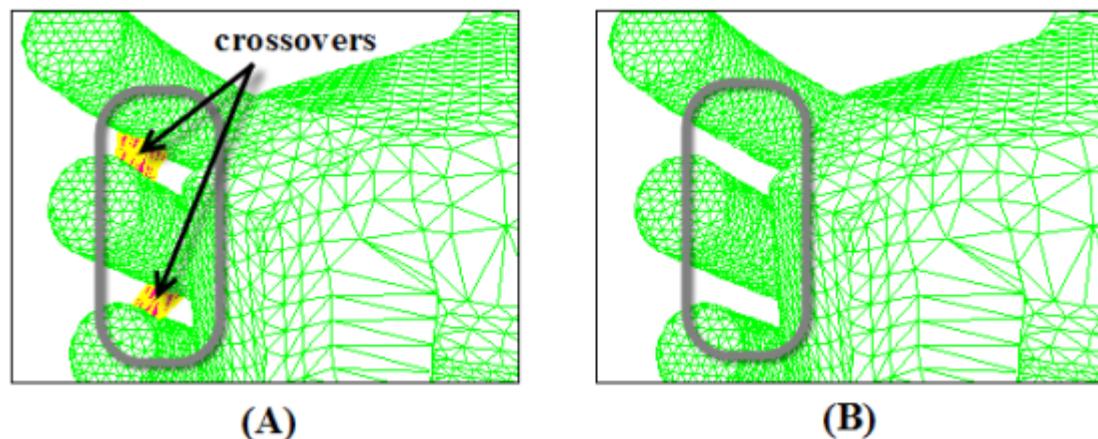
The automatic crossover removal resolves the crossover faces throughout the domain. You may need to manually resolve some configurations that could not be resolved during the automatic crossover removal.

4. Disable **Automatic** and specify the following crossover removal parameters.

- a. Specify the number of crossover removal attempts in the **Iteration** field.
- b. Select the seed face as appropriate.
- c. Enter appropriate values for **Max Faces** and **Relaxation**.
- d. Click **Apply**.

[Figure 10.19: Wrapper Surface \(A\) Before and \(B\) After Crossover Removal \(p. 206\)](#) shows crossovers connecting separate parts of the wrapper surface. These faces will be deleted during the crossover removal.

Figure 10.19: Wrapper Surface (A) Before and (B) After Crossover Removal



10.8. Text Commands for the Wrapper

The text interface commands for wrapping boundary zones are listed below:

/boundary(wrapper)/delete-all-cells?

allows you to delete the Cartesian mesh. This command is available only after initializing the Cartesian grid.

/boundary(wrapper)/imprint-edges

imprints the wrapper surface on recovered feature edges that you recover. This command is available only after creating the wrapper surface.

/boundary(wrapper)/initialize

creates a Cartesian mesh of the specified parameters.

/boundary(wrapper)/local-regions/define

allows you to define the local refinement region.

/boundary(wrapper)/local-regions/delete

deletes the specified refinement region.

- /boundary(wrapper/local-regions/init**
creates a region encompassing the entire geometry.
- /boundary(wrapper/local-regions/list-all-regions**
lists all the refinement regions in the console.
- /boundary(wrapper/local-regions/refine**
refines the specified region according to the refinement parameters specified.
- /boundary(wrapper/pre-smooth?**
allows you to enable or disable smoothing of nodes during wrapping. This command is available only after initializing the Cartesian grid.
- /boundary(wrapper/region/draw-holes**
draws the holes detected. See [Automatic Leak Detection \(p. 187\)](#) for details.
- /boundary(wrapper/region/extract-enclosing-region**
extracts the interface for the region enclosing the specified point.
- /boundary(wrapper/region/extract-interface**
extracts the interfaces for the specified regions.
- /boundary(wrapper/region/fix-holes**
fixes the specified hole(s). See [Automatic Leak Detection \(p. 187\)](#) for details.
- /boundary(wrapper/region/list-holes**
lists the existing holes.
- /boundary(wrapper/region/list-regions**
lists the region created when the Cartesian grid is initialized.
- /boundary(wrapper/region/merge-interior-regions**
merges all the interior regions. After using this command, two regions will remain, the exterior and the merged interior region.
- /boundary(wrapper/region/merge-regions**
merges the specified regions.
- /boundary(wrapper/region/modify-region-holes**
allows you to fix or open holes related to the specified region.
- /boundary(wrapper/region/open-holes**
opens the specified hole(s). See [Automatic Leak Detection \(p. 187\)](#) for details.
- /boundary(wrapper/region/refine**
refines the Cartesian grid based on the zone specific sizes and local size functions. This command is available only after initializing Cartesian grid.
- /boundary(wrapper/region/refine-enclosing-region**
refines the region enclosing the specified point.
- /boundary(wrapper/region/refine-region**
refines the specified region.

/boundary(wrapper/region/refine-zone-cells

refines the cells associated with the specified boundary zone. This command is available only after initializing the Cartesian grid.

/boundary(wrapper/region/update-regions

updates the regions to account for the changes made to the original geometry (during manual hole fixing).

/boundary(wrapper/region/wrap-enclosing-region

generates the wrapper surface for the region enclosing the specified point.

/boundary(wrapper/region/wrap-region

generates the wrapper surface for the specified region. This command is available only after initializing the Cartesian grid.

/boundary(wrapper/region/wrapper-region-at-location

reports the region at the specified location.

/boundary(wrapper/set/clear-size

clears all the zone specific size parameters.

/boundary(wrapper/set/curvature?

allows you to enable or disable the curvature size function.

/boundary(wrapper/set/curvature-factor

allows you to specify the curvature factor for initializing the wrapper.

/boundary(wrapper/set/default-face-size

allows you to specify the default face size for the Cartesian grid.

/boundary(wrapper/set/feature-threshold

allows you to specify the critical range within which the nodes of the wrapper will be projected onto the feature edges.

/boundary(wrapper/set/ignore-feature-skewness

specifies the critical skewness to preserve feature lines during the face improvement. Only faces with skewness greater than the specified value will be modified. The default value is such that no feature lines will be modified during the face improvement.

/boundary(wrapper/set/ignore-self-proximity?

allows you to enable or disable the self-proximity calculation during the refinement.

/boundary(wrapper/set/list-size

lists the current zone specific sizes of the domain.

/boundary(wrapper/set/local-size-function

allows you to set local size functions for refining the Cartesian grid.

/boundary(wrapper/set/max-refine-level

specifies the maximum allowable refinement level.

/boundary(wrapper/set/maximum-size-level

specifies the largest cell size in the Cartesian grid.

/boundary(wrapper)set/minimum-proximity-gap

specifies the minimum gap within which the proximity will be ignored.

/boundary(wrapper)set/number-of-size-boxes

specifies the number of boxes to be displayed when using the **Draw Sizes** button in the **Face Size** tab.

/boundary(wrapper)set/proximity?

allows you to enable or disable the proximity size function.

/boundary(wrapper)set/proximity-factor

allows you to specify the proximity factor for initializing the wrapper.

/boundary(wrapper)set/read-local-sizes

reads the zone-specific sizes written to a file.

/boundary(wrapper)set/refinement-buffer-layers

allows you to specify the specific number of additional cell layers that you want to refine.

/boundary(wrapper)set/relative-island-count

allows you to specify a critical cell count for island regions generated during wrapping.

/boundary(wrapper)set/write-local-sizes

writes the zone-specific sizes to a file.

/boundary(wrapper)set/zone-specific-size

allows you to specify zone specific sizes.

/boundary(wrapper/post-improve/auto-post-improve

improves the wrapper surface using a pre-defined sequence of operations.

/boundary(wrapper/post-improve/auto-post-wrap

performs a pre-defined sequence of post-wrapping operations on the wrapper surface.

/boundary(wrapper/post-improve/coarsen-wrapper-surf

allows you to coarsen the wrapper surface.

/boundary(wrapper/post-improve/filterout-far-features

allows you to delete feature edges beyond the specified distance from the wrapper surface.

/boundary(wrapper/post-improve/imprint-geom-surf

allows you to imprint the geometry on the wrapper surfaces (manual zone recovery).

/boundary(wrapper/post-improve/improve

allows you to improve the wrapper surface quality based on skewness, size change, or aspect ratio.

/boundary(wrapper/post-improve/inflate-thin-regions

allows you to push apart the overlapping faces in thin regions.

/boundary(wrapper/post-improve/post-single-surface

allows you to clean up unmerged island regions after recovering the single surface.

/boundary(wrapper/post-improve/recover-single-surface

allows you to recover thin surfaces as a single surface after wrapping.

/boundary(wrapper/post-improve/recover-zone)

separates the wrapper surface into zones based on the original geometry.

/boundary(wrapper/post-improve/remove-crossover-config)

allows you to remove crossover configurations.

/boundary(wrapper/post-improve/remove-duplicated-nodes)

removes the duplicate nodes on the wrapper surface.

/boundary(wrapper/post-improve/rename-wrapper-zones)

allows you to rename the wrapper zones by specifying an appropriate prefix instead of the default prefix (**wrap-**).

/boundary(wrapper/post-improve/resolve-nonmanifoldness)

allows you to resolve non-manifold configurations on the wrapper surface.

/boundary(wrapper/post-improve/resolve-self-intersection)

removes the self intersecting faces.

/boundary(wrapper/post-improve/rezone)

smoothes the zones separated from the wrapper surface for better representation of the geometry.

/boundary(wrapper/post-improve/smooth-folded-faces)

smoothes the folded faces on the wrapper surface.

/boundary(wrapper/post-improve/smooth-wrapper-surf)

smoothes the wrapper surface.

/boundary(wrapper/post-improve/swap-wrapper-surf)

swaps the nodes of the wrapper surface to improve its quality.

Chapter 11: Creating a Mesh

After reading the boundary mesh and performing the necessary modifications (e.g., merging duplicate nodes, edge swapping) you will create the volume mesh.

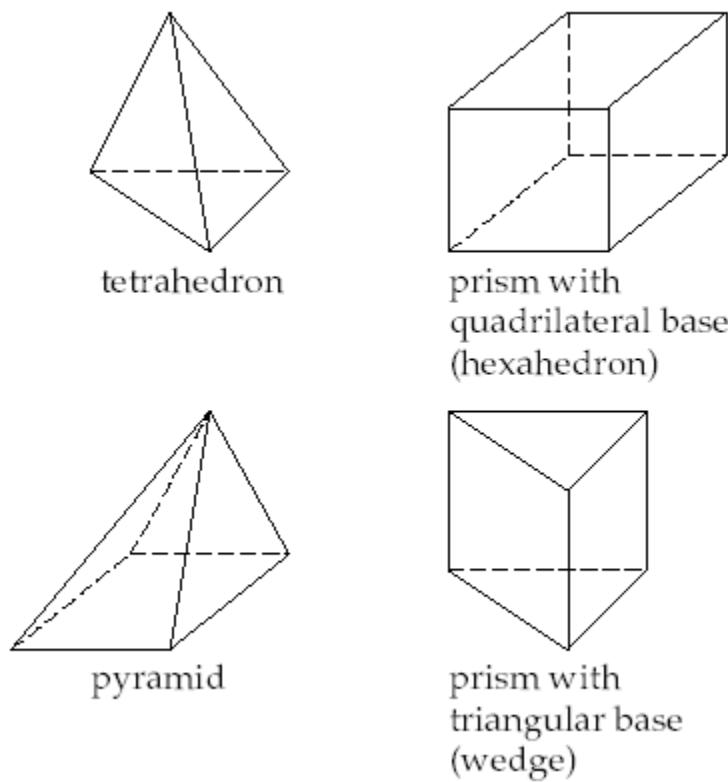
Depending on the type of mesh you are starting from (boundary mesh only, boundary mesh with hexahedral cells in one or more regions, etc.), you can automatically generate the mesh, manually generate the mesh step by step, or use a combination of manual and automatic commands.

You can create several types of meshes comprising different element types. The meshing strategy and the use of the **Auto Mesh** tool are described in this chapter. The quad-tet transition elements (pyramid and non-conformal meshing) and creation of a heat exchanger zone are also described. Detailed descriptions of meshing techniques such as prism meshing, tetrahedral and hexcore meshing options, etc. are described in subsequent chapters.

- 11.1. Choosing the Meshing Strategy
- 11.2. Using the Auto Mesh Dialog Box
- 11.3. Generating Pyramids
- 11.4. Creating a Non-Conformal Interface
- 11.5. Creating a Heat Exchanger Zone
- 11.6. Parallel Meshing

11.1. Choosing the Meshing Strategy

A variety of cell shapes that can be used to mesh the domain, e.g., tetrahedra, prisms, hexahedra, and pyramids. These shapes are shown in [Figure 11.1: Different Cell Shapes \(p. 212\)](#).

Figure 11.1: Different Cell Shapes

A mesh consisting entirely of tetrahedral elements is a tetrahedral mesh, and a mesh with any combination of cell shapes is referred to as a hybrid mesh .

Before generating a volume mesh, determine the shapes that are appropriate for the case you are solving. Then follow the instructions for creating the required cell types. Most cases will fall into one of the following categories:

- [11.1.1. Boundary Mesh Containing Only Triangular Faces](#)
- [11.1.2. Mixed Boundary Mesh](#)
- [11.1.3. Hexcore Mesh](#)
- [11.1.4. CutCell Mesh](#)
- [11.1.5. Additional Meshing Tasks](#)
- [11.1.6. Inserting Isolated Nodes into a Tet Mesh](#)

11.1.1. Boundary Mesh Containing Only Triangular Faces

If you require a high mesh resolution in some portion of the domain, such as a boundary layer, you can obtain an efficient and better quality mesh by meshing that portion with prisms (wedges) and then meshing the rest of the domain with tetrahedra (tets). The resulting mesh is referred to as a viscous hybrid mesh.

The procedure is as follows:

1. Build one or more layers of prisms, starting from the appropriate boundary (or boundaries). Refer to [Generating Prisms \(p. 229\)](#) for details.
2. Create a domain encompassing the region to be meshed with tetrahedra.

3. Generate the tets in the selected domain using either automatic, or manual tet mesh generation, or a combination of the two. Refer to [Generating Tetrahedral Meshes \(p. 267\)](#) for details.

[Figure 11.2: Mesh with Prisms in a Boundary Layer Region \(p. 213\)](#) shows several layers of prisms in a portion of a mesh created in this manner. The prisms extend throughout the entire region bounded by the quadrilateral faces, but only a few of them are shown here.

Figure 11.2: Mesh with Prisms in a Boundary Layer Region



The surface mesh originally contained only triangular faces. The quadrilateral faces are created automatically when the prisms are built on the triangular faces.

If the quadrilateral faces of the prisms do not lie on the external boundary of the domain (i.e., if the prism region begins and/or ends in the interior of the domain), create a layer of transitional pyramids between steps 1 and 2. Refer to [Generating Pyramids \(p. 220\)](#) for details.

If you have no special boundary layer resolution requirements, you can generate a mesh consisting entirely of tetrahedra (see [Figure 11.3: Surface Mesh Containing Only Tetrahedra \(p. 213\)](#)). You can use the automatic tetrahedral mesh generation procedure, the manual procedure, or a combination of both. Refer to [Generating Tetrahedral Meshes \(p. 267\)](#) for details.

Figure 11.3: Surface Mesh Containing Only Tetrahedra



11.1.2. Mixed Boundary Mesh

Start from a boundary mesh containing triangular and quadrilateral faces, as well as hexahedral cells in the quadrilateral face regions. The resulting mesh is referred to as a zonal hybrid mesh.

1. Add a layer of pyramids to the quadrilateral boundary face zone that lies between the hexahedral region and the adjacent region to be meshed with tetrahedra. This creates the triangular boundary face zone that is required to create tetrahedra in the adjacent region. Refer to [Generating Pyramids \(p. 220\)](#) for details.
2. Create a domain encompassing the region to be meshed with tetrahedra. Refer to [Using Domains to Group and Mesh Boundary Faces \(p. 368\)](#) for details.
3. Generate the tetrahedra in the selected domain using either automatic or manual tet mesh generation, or a combination of both. Refer to [Generating Tetrahedral Meshes \(p. 267\)](#) for details.

[Figure 11.4: Surface Mesh \(p. 213\)](#) shows the surface mesh for a portion of a grid containing hexahedra, pyramids, tetrahedra, and prisms that was created on a plenum feeding a valve-port cylinder.

Figure 11.4: Surface Mesh



- The less complicated plenum pipe on the left of the figure is meshed using hexahedral cells.
- The more complex valve port (valve not visible because it is inside the surrounding pipe) is meshed using tetrahedra.

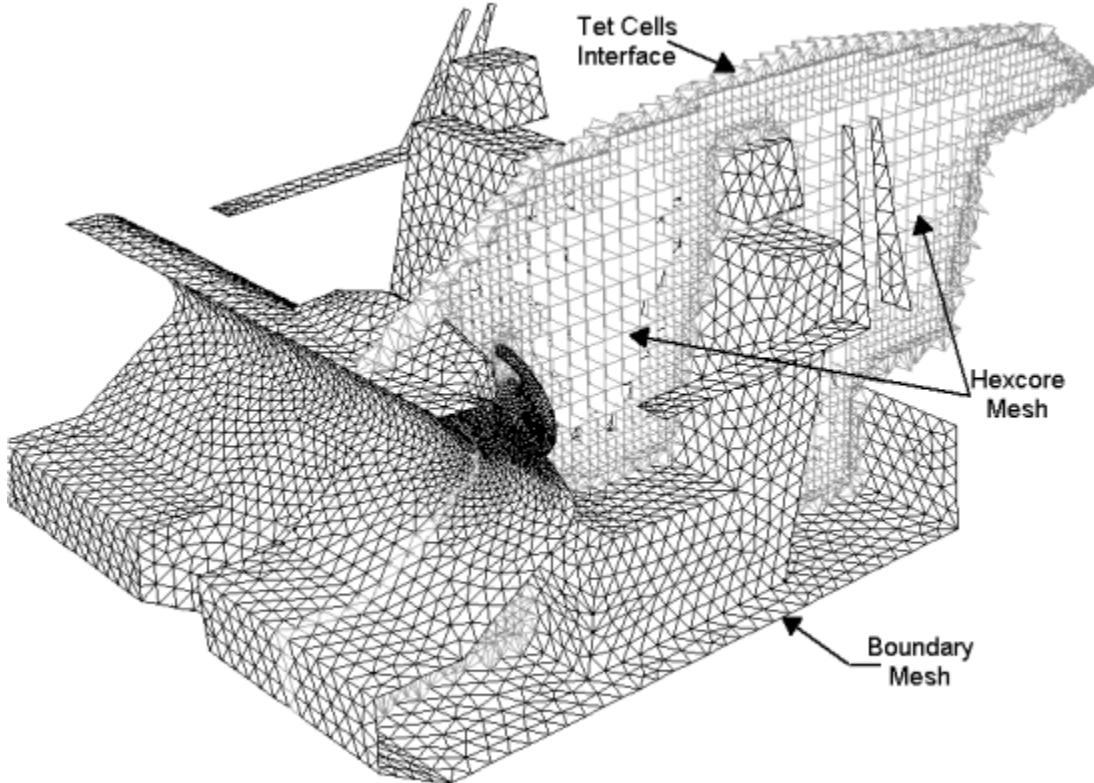
- Pyramids are used as a transition between the hexahedral grid for the plenum and the tetrahedral grid for the valve port. This transition occurs where the triangular and quadrilateral faces meet in the middle of the figure.
- Additionally, the quadrilateral faces produced by extending triangular faces in the cylinder (i.e., the quadrilateral sides of the resulting prism wedges) can be seen in the far right of the figure.

11.1.3. Hexcore Mesh

The hexcore mesh features a tetrahedral/hybrid mesh adjacent to walls and a Cartesian mesh in the core flow region. [Figure 11.5: Hexcore Mesh \(p. 214\)](#) shows the typical hexcore mesh. The hexcore meshing scheme creates a mesh consisting of two regions:

- An inner region composed of regular Cartesian cells.
- An outer region consisting of tetrahedral elements.

Figure 11.5: Hexcore Mesh



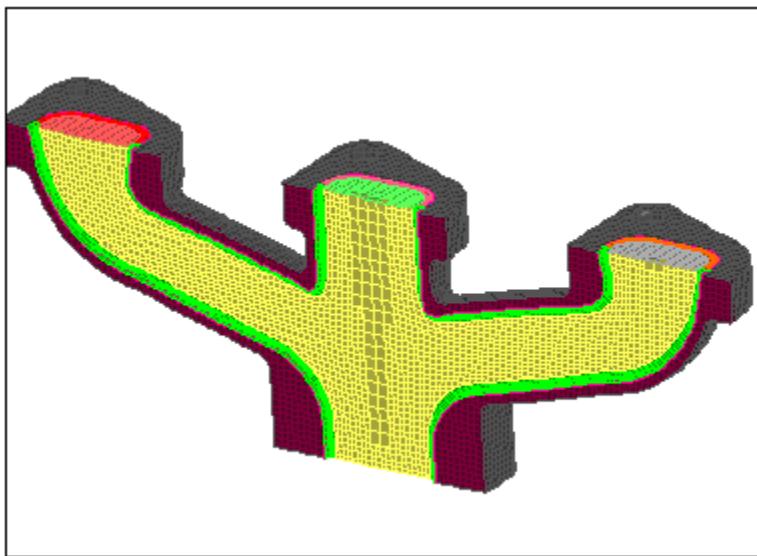
Wedge elements are created only when boundary layers are attached on faces pre-meshed with triangular elements. It combines the automation and geometric flexibility of tetrahedral/hybrid meshes with greatly reduced cell counts in many applications. The hexcore mesh is most beneficial in geometries with large open spaces as in automotive, aerospace, and HVAC applications. Refer to [Generating the Hexcore Mesh \(p. 285\)](#) for details.

11.1.4. CutCell Mesh

CutCell meshing is a general purpose hex-dominant meshing technique which can be used instead of tetrahedral or hexcore meshing, without requiring a very high quality surface mesh as a starting point. This method uses a direct surface and volume approach without the need of cleanup or decomposition,

thereby reducing the turnaround time required for meshing. A key feature is the large fraction of hex cells in the mesh.

Figure 11.6: CutCell Mesh



Refer to [Generating the CutCell Mesh \(p. 295\)](#) for details.

11.1.5. Additional Meshing Tasks

Additional meshing tasks that can be handled are:

- If you have a complete volume mesh and want to extend some portion of the domain (e.g., increase the length of an inlet pipe), you can grow one or more layers of prisms from the current external (quadrilateral or triangular) boundary to be extended.

[Figure 11.7: Extending an Existing Tetrahedral Mesh Using Prisms \(p. 215\)](#) shows a region of prisms (wedges) extended from the triangular face zone that bounds a tetrahedral region.

Figure 11.7: Extending an Existing Tetrahedral Mesh Using Prisms

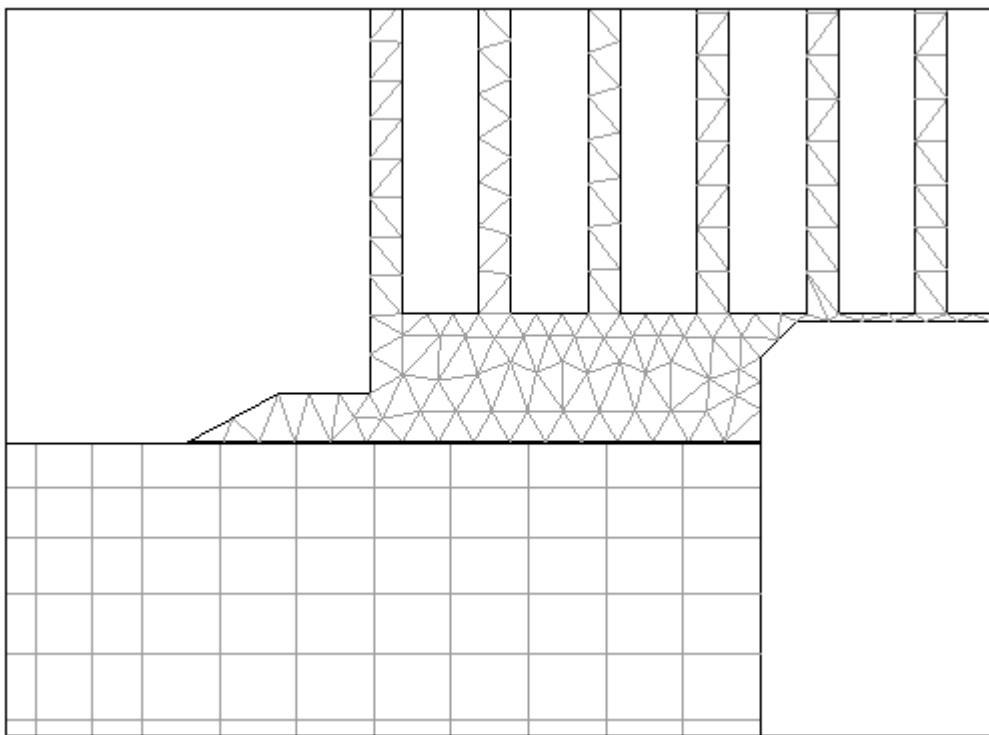


- Unless there is a reason to use hexahedral cells in the quad regions, it is preferable to convert a mixed tri/quad boundary mesh to an all-tri boundary mesh and then create a tetrahedral mesh for a 3D boundary mesh consisting of only triangular faces.

Use the **Triangulate Zones** dialog box or the command `/boundary/remesh/triangulate` to convert quad face zones to tri face zones.

You can then use the **Boundary Improve** dialog box to improve the skewness of the triangular boundary zone created.

- You may choose to use a non-conformal interface to define the computational domain. This type of interface allows you to relax the requirement for point-to-point matching at the interface between the meshes, as illustrated in [Figure 11.8: Example of a Non-Conformal Interface \(p. 216\)](#).

Figure 11.8: Example of a Non-Conformal Interface

This feature of relaxing the requirement for point-to-point matching at the interface between the meshes is particularly useful in parametric studies where you want to change an isolated region of the domain without changing the entire mesh. The procedure is as follows:

1. Read the meshes in meshing mode in ANSYS Fluent. These meshes need not share nodes, edges, faces, or cells.
2. Create the volume mesh using an appropriate meshing strategy. You can also create domains for each independent mesh region and mesh the individual domains separately.
3. Separate the region of non-conformal interface into new face zones using one of the following alternatives:
 - Use the command `/mesh/non-conformals/separate`. See [Separating the Non-Conformal Interface Between Cell Zones \(p. 225\)](#) for details.
 - Change the face zone type of the two surfaces that will be treated as non-conformal to **interface**.
4. Transfer the mesh to solution mode in ANSYS Fluent and create the non-conformal interface using the **Create/Edit Mesh Interfaces** dialog box. Refer to the [User's Guide](#) for details.

Note

If you chose to write a journal file when using the `/mesh/non-conformals/separate` command to separate the mesh interface zones, you can read the journal file to create the mesh interface automatically in solution mode.

11.1.6. Inserting Isolated Nodes into a Tet Mesh

To add nodes to the mesh without specifying the faces, you can create a boundary zone in the boundary mesh (in the program that created it) just for this purpose. Then you can introduce the nodes associated with these faces.

This feature is useful for clustering nodes (and therefore cells) in a controlled manner. [Figure 11.9: Mesh Generated Using Isolated Nodes to Concentrate Cells \(p. 218\)](#) shows a mesh that was generated using this method to cluster the nodes behind the wedge. A mesh that was generated for the same geometry without clustering is shown in [Figure 11.10: Mesh Generated Without Using Isolated Nodes \(p. 218\)](#). You can follow either of the two procedures to insert isolated nodes into a mesh:

- Delete the face zone before meshing.
- Introduce the nodes using face zones after meshing.

Deleting the Face Zone Before Meshing

This procedure is as follows:

1. Delete the face zone, but leave the associated nodes. Disable **Delete Nodes** in the **Manage Face Zones** dialog box to retain these nodes.

Note

By default, the unused nodes are deleted when the faces of the zone are deleted.

2. Disable **Delete Unused Nodes** in the **Tet** dialog box and generate the volume mesh.

Note

By default, unused nodes will be deleted during the automatic meshing.

The nodes will be introduced when you initialize the mesh. When you use this procedure, *all* nodes must be inserted into the mesh or the initialization will fail.

Warning

Do not place isolated nodes too close to the boundary or to other nodes.

Figure 11.9: Mesh Generated Using Isolated Nodes to Concentrate Cells

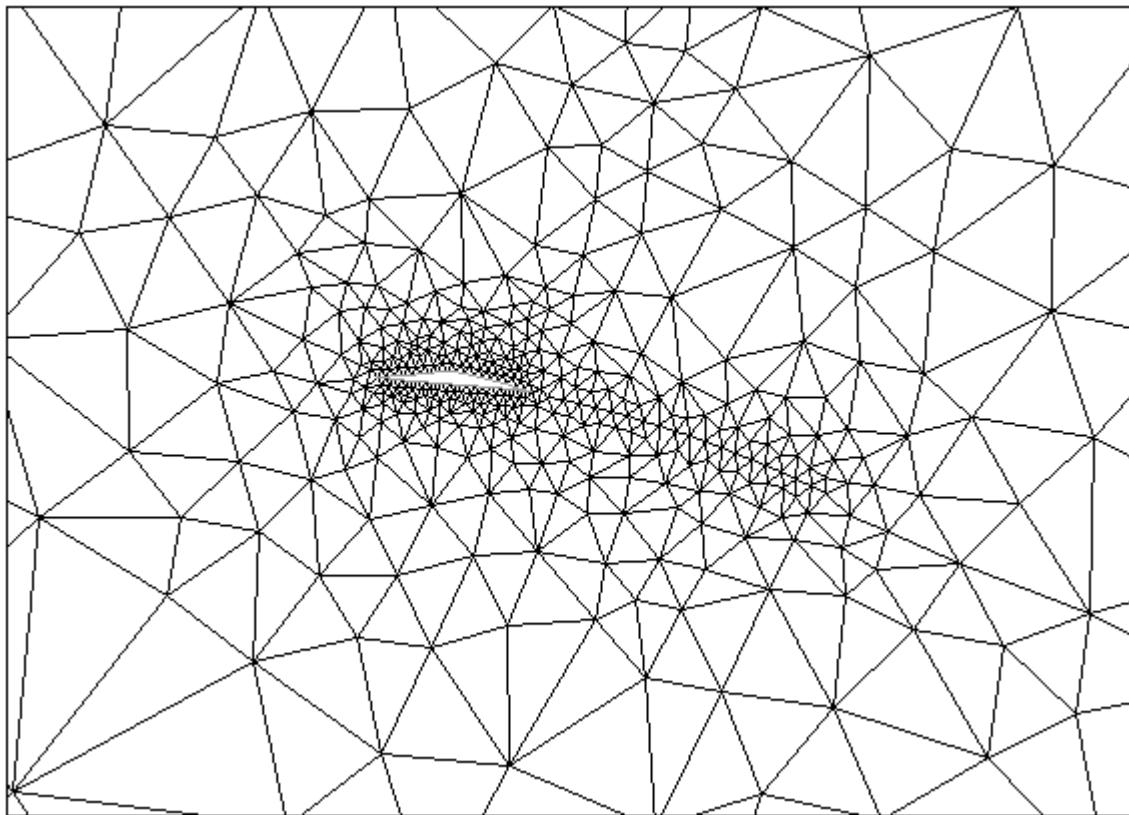
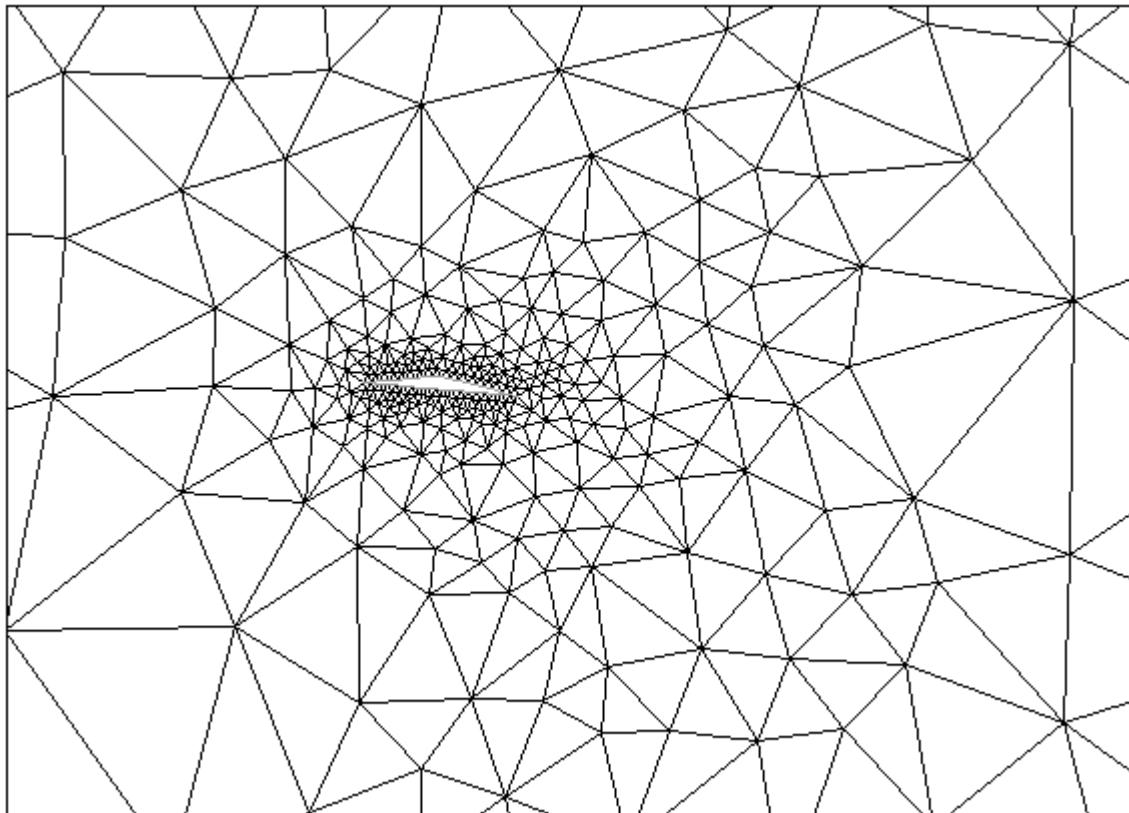


Figure 11.10: Mesh Generated Without Using Isolated Nodes



Introducing the Nodes Using Face Zones After Meshing

This procedure is as follows:

1. Create a subdomain that does not include the face zones used to control the mesh density.
2. Create the volume mesh using the subdomain.
3. Activate the global domain.
4. Introduce the additional nodes using the text command: /mesh/modify/mesh-nodes-on-zone.

Important

The /mesh/modify/mesh-nodes-on-zone command will delete the faces associated with the face zone.

11.2. Using the Auto Mesh Dialog Box

The **Auto Mesh** dialog box allows you to automatically create the volume mesh using the different mesh elements available. This dialog box can be used for generating the volume mesh based on face zones or based on a mesh object and relevant material points.

The generic procedure for using the [Auto Mesh Dialog Box \(p. 464\)](#) for creating the mesh comprises the following steps:

1. Determine the meshing approach (face zone based or object based) and the mesh elements required for the particular case.
2. Select the appropriate option in the **Object** drop-down list in the **Auto Identify Volume** group box.
 - a. For the face zone based meshing approach, ensure that **none** is selected in the **Object** drop-down list.
 - b. For the object based meshing approach, select the appropriate mesh object in the **Object** drop-down list. Select the material points in the **Material Points** selection list and enable **Keep Solid Cell Zones** if required.
3. If you need to grow prisms for the geometry under consideration, click the **Set...** button in the **Boundary Layer Mesh** group box to open the **Prisms** dialog box. Set the appropriate prism parameters and click **Apply** in the **Prisms** dialog box. Refer to [Prism Meshing Options \(p. 232\)](#) for details on the prism meshing options available. Verify that the **Prisms** check box is enabled in the **Boundary Layer Mesh** group box in the **Auto Mesh** dialog box.
4. Select the appropriate quad-tet transition elements from the **Quad Tet Transition** list. Click the **Set...** button to open the **Pyramids** dialog box or the **Non Conformals** dialog box (depending on the selection) and specify the appropriate parameters. Refer to [Creating Pyramids \(p. 221\)](#) and [Creating a Non-Conformal Interface \(p. 225\)](#) for details.
5. Select the appropriate option from the **Volume Fill** list. Click the **Set...** button to open the **Tet** dialog box or the **Hxcore** dialog box (depending on the selection) and specify the appropriate parameters.

Refer to [Initializing the Tetrahedral Mesh \(p. 274\)](#), [Refining the Tetrahedral Mesh \(p. 276\)](#), and [Controlling Hexcore Parameters \(p. 287\)](#) for details.

6. For face zone based meshing, enable **Merge Cell Zones** and **Auto Identify Topology**, if desired.

Note

The **Merge Cell Zones** and **Auto Identify Topology** options are not available for object based meshing.

7. Click **Mesh** to automatically create the mesh.

Alternatively, you can use the command `/mesh/auto-mesh` to generate the mesh automatically. Specify whether the volume mesh is to be generated based on mesh objects and material points or face zones in the domain. Specify the meshing parameters for the mesh elements (prisms, pyramids or non-conformals, tet or hex) using either the respective dialog boxes or the associated text commands prior to using the `auto-mesh` command. You will be prompted to specify the mesh elements required when the `auto-mesh` command is invoked. The previously set parameters will be used to create the mesh.

11.2.1. Text Commands for Auto Mesh

Text commands for the Auto Mesh option are as follows:

/mesh/auto-mesh

allows you to generate the mesh automatically. Specify whether the volume mesh is to be generated based on mesh objects and material points or face zones in the domain. Specify the mesh elements to be used when prompted. For face zone based meshing, specify whether to merge cell zones and automatically identify the domain to be meshed based on the topology information. You can specify the meshing parameters for the mesh elements (prisms, pyramids or non-conformals, tet or hex) using either the respective dialog boxes or the associated text commands prior to using the `auto-mesh` command.

/mesh/auto-prefix-cell-zones

allows you to specify a prefix for cell zones created during the auto mesh procedure.

Note

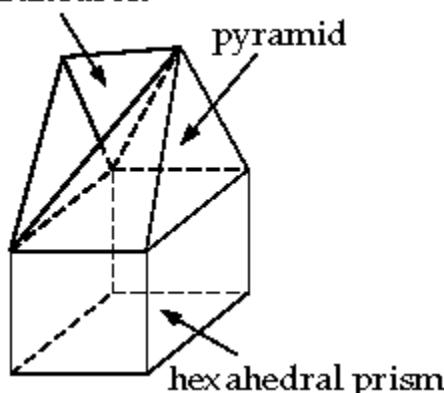
The `auto-prefix-cell-zones` command is not relevant for object-based meshing, where the cell zone names are generated based on the material point(s) and the wrap objects used to generate the mesh object.

11.3. Generating Pyramids

A pyramid has a quadrilateral face as its base and four triangular faces extending from the sides of the quadrilateral up to a single point above the base. See [Figure 11.11: Pyramid Cell—Transition from a Hexahedron to a Tetrahedron \(p. 221\)](#).

Figure 11.11: Pyramid Cell—Transition from a Hexahedron to a Tetrahedron

tetrahedron



To generate a conformal mesh with a region of tetrahedral cells adjacent to a region of hexahedral cells, you will first create a layer of pyramids as a transition from quadrilateral faces to triangular faces. After creating a single layer of pyramids, the resulting triangular faces will be used to create tetrahedra.

To create pyramids, you need to specify the boundary from which the pyramids will be built, the method for determining the top vertex of each pyramid, and the pyramid height.

[11.3.1. Creating Pyramids](#)

[11.3.2. Zones Created During Pyramid Generation](#)

[11.3.3. Text Commands for Generating Pyramids](#)

[11.3.4. Pyramid Meshing Problems](#)

11.3.1. Creating Pyramids

The procedure for creating a layer of pyramids from a quadrilateral boundary zone is as follows:

1. Check the aspect ratio limits of the boundary face zones on which you need to build pyramids.

Report → Face Limits...

The use of a high-aspect-ratio quadrilateral for the base of a pyramid produces skewed triangular faces that can cause problems during the tetrahedral mesh generation. If the maximum quadrilateral face aspect ratio is much greater than 10, you will need to regenerate them.

- If the faces were created in a different preprocessor, return to that application and try to reduce the aspect ratio of the faces in question.
- If the faces were created during the building of prism layers, rebuild the prisms using a more gradual growth rate.

2. Select the appropriate quadrilateral boundary zone(s) in the **Boundary Zones** selection list in the **Pyramids** dialog box.

Mesh → Pyramids...

Click **Draw** to view the selected zones.

If the quadrilateral zone you require does not appear in the list, use the /boundary/reset-element-type text command to update the type of the zone. It is possible that the quadrilateral zone may not be recognized due to changes made to the boundary mesh.

For example, if you separate a mixed (tri and quad) face zone into a tri face zone and a quad face zone, each of these will be identified as a mixed zone. You need to reset the element type for the quad zone for it to be recognized and included in the **Boundary Zones** selection list.

3. Select the appropriate method for determining the pyramid vertex location in the **Options** list. The **skewness** method is selected by default, and is appropriate for most cases.
4. Specify the height of the pyramids by setting the **Offset Scaling** value.
5. Click **Create**. The new pyramid cell zone and the new face zones created will be reported in the console. You can then use the **Display Grid** dialog box to view these new zones.

Display → Grid...

6. Change the boundary type of the quadrilateral base zone to the appropriate type (if necessary).

Boundary → Manage...

Note

Since you built cells next to the quadrilateral base zone, its original boundary type may no longer be correct.

Important

The pyramids should be automatically created on the appropriate side of the specified boundary zone(s). If the pyramids are on the wrong side, do the following:

1. Delete the newly created zones related to the pyramids.
 2. Reverse the normal direction on the quadrilateral boundary where the pyramids are being built (using the **Flip Normals** option in the [Manage Face Zones Dialog Box \(p. 528\)](#)).
 3. Recreate the pyramids.
-

11.3.2. Zones Created During Pyramid Generation

When pyramids are generated, at least two new zones are created: a cell zone containing the pyramid cells and a face zone containing the triangular faces of the pyramids.

The following zones will be created during pyramid generation:

- The cell zone containing the pyramids (`pyramid-cells-n`).
- The face zone containing the triangular faces of the pyramid cells (`base-zone-pyramid-cap-n`).

For example, if the pyramids were built from the quadrilateral face zone **wall-4**, they will be placed in a new zone called **wall-4-pyramid-cap-9** (where the 9 is the zone number assigned).

- The face zones containing the pyramid sides which use existing faces from the original boundary mesh (`base-zone-pyramid-side:n`), where, n is the zone number assigned.

For example, if triangular boundary faces from the zone **wall-3** are used, they will be placed in a new zone called **wall-3-pyramid-side-6** (where the 6 is the zone number assigned).

Important

If you include the pyramid-side boundary zone(s) when defining the domain in which you are going to generate a tetrahedral mesh, the tetrahedral meshing will fail.

11.3.3. Text Commands for Generating Pyramids

Text commands for creating pyramid cells are as follows:

/mesh/pyramid/controls/neighbor-angle

sets the threshold dihedral angle used to limit the neighboring faces considered for pyramid creation. For example, if the value is set to 110 degrees and the angle between a given quadrilateral face and the neighboring triangular face is greater than 110 degrees, the resulting pyramid will not include the triangular face.

/mesh/pyramid/controls/offset-factor

specifies the fraction of the computed pyramid height (offset) by which the pyramid heights will be randomly adjusted. The default value is 0, indicating that all pyramids will have the exact height computed. A value of 0.1, for example, will limit each adjustment to $\pm 10\%$ of the computed height.

/mesh/pyramid/controls/offset-method

specifies the method by which offset distances are determined.

mesh/pyramid/controls/offset-scaling

specifies the scaling (s) to be used to determine the height of the pyramid. See [Equation 21.1 \(p. 554\)](#).

mesh/pyramid/controls/vertex-method

specifies the method by which the location of the new vertex of the pyramid will be determined. The skewness method is used by default.

/mesh/pyramid/create

creates a layer of pyramid cells on the specified quadrilateral face zone.

11.3.4. Pyramid Meshing Problems

Most problems associated with creating pyramid layers manifest themselves in the subsequent process of generating the tetrahedral mesh.

Rapid Changes in Volume

Rapid changes in the sizes of cells have a negative influence on the convergence and accuracy of the numerical solution. The pyramid layer creation can produce rapid variations in cell volume in the following situations:

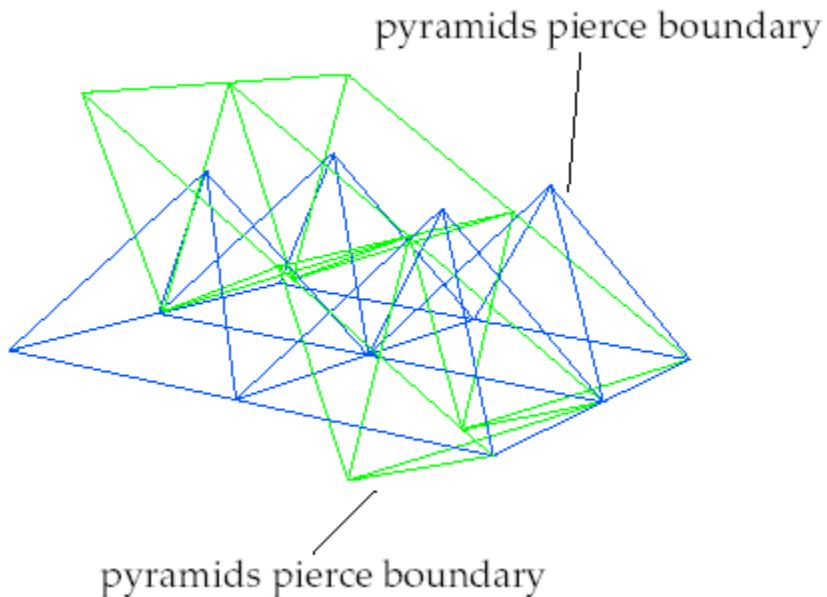
- If the quadrilateral surface mesh has faces with rapid changes in size.
- If there is great disparity between the sizes of the quadrilateral faces and the neighboring triangular faces used in the pyramid creation.

You can avoid the rapid variation in volume by creating quadrilateral and neighboring triangular grids with smooth variations in face size.

Intersecting Faces

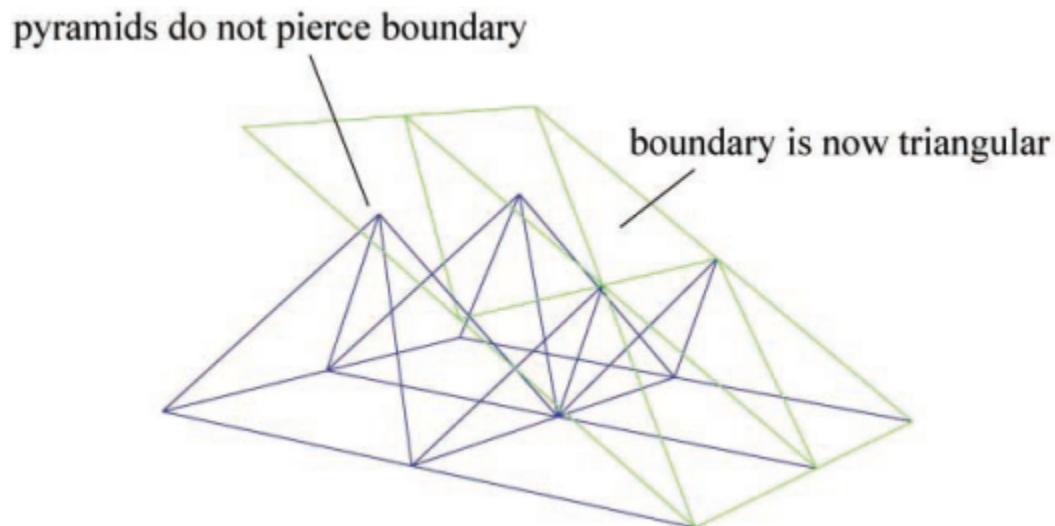
If the quadrilateral surface used to create pyramids has highly concave corners, the resulting pyramids may pierce each other and/or neighboring boundary faces (see [Figure 11.12: Pyramid Cells Intersecting Each Other and Boundary \(p. 224\)](#)).

Figure 11.12: Pyramid Cells Intersecting Each Other and Boundary



In such cases, you can either increase the resolution to prevent the intersections or alter the meshing strategy. An alternative is to separate the quadrilaterals in the concave corner into another zone (using the **Separate Face Zones** dialog box or the **Modify Boundary** dialog box, create triangular faces from the quadrilateral faces using the /boundary/remesh/triangulate text command, and then create pyramids.

The skewness-based pyramid creation will use the existing triangular faces and avoid the intersection problem (see [Figure 11.13: Fixed Intersecting Pyramid Cells Using Triangular Faces \(p. 225\)](#)).

Figure 11.13: Fixed Intersecting Pyramid Cells Using Triangular Faces

High Aspect Ratio

Creating pyramids on quadrilateral faces with very high aspect ratios results in highly skewed triangular faces. Subsequent attempts to create a tetrahedral mesh from these elements will produce a poor-quality mesh.

Irrespective of the method used to generate the quadrilaterals, modify the meshing strategy to reduce the aspect ratio using an external grid generation package or the prism layer capability.

11.4. Creating a Non-Conformal Interface

For meshes containing both hexahedral and tetrahedral elements, you can generate a non-conformal interface to avoid creating intermediate pyramids as transition elements between the quadrilateral and triangular surfaces. You can also choose to create a non-conformal interface when growing prisms from a boundary on a surface mesh, to avoid quad faces in the domain to be meshed. The surfaces containing quad elements will be copied and then triangulated while keeping the original surfaces intact. The free nodes of the triangulated surface will then be merged with the nodes on the original surface mesh. Both surfaces will then be converted to interface type.

The options in the [Non Conforms Dialog Box \(p. 555\)](#) allow you to create a non-conformal interface. You can also use the relevant text commands ([Text Commands for Creating a Non-Conformal Interface \(p. 226\)](#)).

[11.4.1. Separating the Non-Conformal Interface Between Cell Zones](#)

[11.4.2. Text Commands for Creating a Non-Conformal Interface](#)

11.4.1. Separating the Non-Conformal Interface Between Cell Zones

The command `/mesh/non-conforms/separate` allows you to separate the face zones comprising the non-conformal interface between the cell zones specified. Specify the cell zones where the interface is not conformal, an appropriate gap distance (absolute or relative), and the critical angle to be used for separating the face zones. The gap distance used for the separation is the larger value of the absolute gap distance specified and the relative gap distance times the average local edge length. You can also choose to orient the boundary face zones after separation and additionally write a journal file during the separation operation. This journal file can then be read in solution mode to create the mesh interfaces automatically.

The separated face zones will be named as follows: **cell zone 1:cell zone 2-orig face zone 1** for face zone 1 attached to cell zone 1 and **cell zone 2:cell zone 1-orig face zone 2** for face zone 2 attached to cell zone 2. If more than one face zone comprise the separated interface zone, the original face zone contributing a larger number of faces will be used for the name.

11.4.2. Text Commands for Creating a Non-Conformal Interface

The text commands for creating a non-conformal interface are as follows:

/mesh/non-conformals/controls/enable?

toggles the creation of a non-conformal interface.

/mesh/non-conformals/controls/retri-method

specifies the method to be used for retriangulating the quad faces on the non-conformal zones.

prism

remeshes the prism-side quad zones named **prism-side*** or ***-quad***.

quad-split

splits the quad faces diagonally into tri faces.

remesh

remeshes all the quad faces based on the edge and surface feature angle specified.

/mesh/non-conformals/create

creates the non-conformal interface on the specified face zone(s) using the specified retriangulation method.

/mesh/non-conformals/separate

allows you to separate the face zones comprising the non-conformal interface between the cell zones specified. Specify the cell zones where the interface is non-conformal, an appropriate gap distance, and the critical angle to be used for separating the face zones. You can also choose to orient the boundary face zones after separation and additionally write a journal file for the separation operation.

11.5. Creating a Heat Exchanger Zone

Many engineering systems, including power plants, climate control, and engine cooling systems typically contain heat exchangers. However, for most engineering problems, it is impractical to model individual fins and tubes of the heat exchanger core.

You can create a heat exchanger volume mesh using the **Mesh/Create/Heat Exchanger...** menu item.

The heat exchanger mesh created comprises prisms generated from a quad split surface mesh. You need to specify four points (either by selecting the locations or nodes) and the required intervals between the first selected point and each of the remaining points to create the heat exchanger mesh. A meshed plane is created using the first three specified points and the corresponding intervals. Prisms are created on the meshed plane using the fourth point and the corresponding interval.

Warning

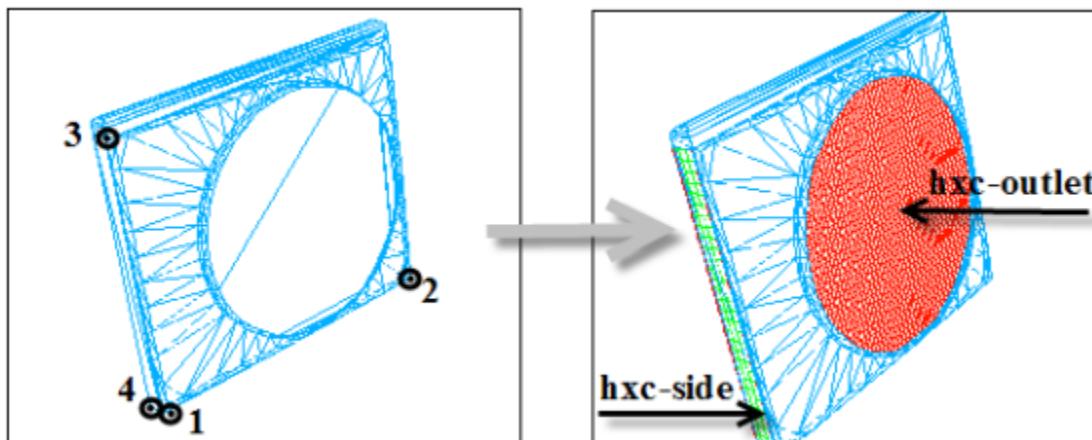
The order of selection of the points is important since the heat exchanger zone is created based on the intervals specified between the first selected point and each of the remain-

ing points. If the points are not specified in the correct order, you will get a heat exchanger zone that is different from the required one.

Alternatively, you can use the command `/mesh/create-neat-exchanger` to create the heat exchanger mesh. You need to specify the method for selecting the points (by location or by nodes), the points, the intervals, and the zone name prefix, as required.

You can preview the heat exchanger zones and modify the parameters if you are dissatisfied with the results. The heat exchanger zones (prefixed by **hxc-**) are created as shown in [Figure 11.14: Creating the Heat Exchanger Mesh \(p. 227\)](#). You can specify the prefix for the zones as required in the **Zone Prefix** dialog box.

Figure 11.14: Creating the Heat Exchanger Mesh



11.6. Parallel Meshing

If you launched Fluent with parallel processing for meshing enabled, you will be able to distribute the mesh data across the available compute nodes, or recombine the mesh data as required. For additional information on specifying parallel processing, see [Parallel Processing in Meshing Mode](#).

When the mesh file is initially loaded, all mesh data is associated with compute node 0. Combined mesh data is required for geometry and surface mesh repair.

Follow these steps to distribute the mesh across the available compute nodes. Distributed mesh data is required to take full advantage of parallel processing.

1. Section the boundary zones.

Any closed surface mesh between objects or bodies may serve to separate sections. The domain may be pre-partitioned (independent creation of separate regions), or you may take advantage of regions in the geometry, or you may maintain a surface mesh between objects or bodies.

Parallel meshing supports non-conformally connected bodies if they can be identified as separate bodies. Within a body, surfaces must be conformal.

2. Open the **Parallel** dialog box under **Parallel > Partition**.
3. Click **Compute Partitions**.

All the **Boundary Zones** will be assigned to the available partitions as groups of closed regions. Assignment is based on location with some consideration for load balancing.

If the number of closed regions is less than the number of compute nodes, then the number of regions is taken as the number of partitions and the other compute nodes will be idle with no mesh.

4. Use the **Compute Node Num, List**, and **Draw** controls to examine the expected mesh distribution.
5. Click **Distribute Mesh**.

Mesh data will be distributed to the available compute nodes based on the computed partitions.

Note

If you read in a case file (using the **File/Read/Case...** menu or the `/file/read-case` command) and then distribute the mesh, boundary conditions may not be preserved.

6. Create your volume mesh.

Note

- Available volume meshing options include Prisms, Non-conformal Quad-Tet transition, and Tetrahedral fill. Unavailable options will have their menu entry greyed out.
 - If including prism layers, consistent orientation must be done before computing the volume mesh.
 - The mesh created by parallel processing is valid although it may have some differences when compared to a serial processed mesh.
-

7. If necessary to perform geometry or surface mesh repair, use **Agglomerate Mesh**.

Distributed mesh data will be recombined into a single partition on compute node 0.

For additional detail on the **Parallel** menu, see [Parallel Menu](#)

11.6.1. Text Commands

Text command for parallel meshing options are as follows:

```
/parallel/agglomerate
    recombine distributed mesh data into a single partition on compute node 0.

/parallel/distribute
    allocate mesh to the compute nodes based on the computed partitions.

/parallel/print-partition-info
    display computed partition data to the console.

/display/set/colors/color-by-partition?
    allows you to view the partitions by color.
```

Chapter 12: Generating Prisms

This chapter describes the automatic and manual procedure for creating prisms. It also discusses the solution to some common problems that you may face while creating prisms.

This chapter comprises the following sections:

- 12.1. Overview
- 12.2. Procedure for Generating Prisms
- 12.3. Prism Meshing Options
- 12.4. Zones Created During Prism Generation
- 12.5. The Prism Generation Process
- 12.6. Using Adjacent Zones as the Sides of Prisms
- 12.7. Direction Vectors
- 12.8. Offset Distances
- 12.9. Improving Prism Quality
- 12.10. Text Commands for Generating Prisms
- 12.11. Prism Meshing Problems

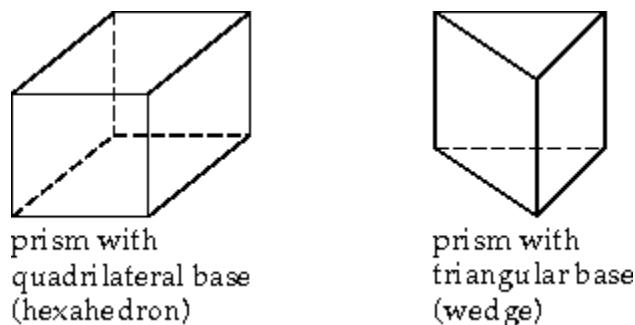
12.1. Overview

You can create prism cells starting from either quadrilateral or triangular boundary faces (or both). Prisms can be used to resolve a boundary layer region of a tetrahedral mesh. They can be used to extend some portion of a domain for which the volume mesh already exists (e.g., increase the length of an inlet pipe), or to create a volume mesh by extrusion. To create prisms, you need to specify the boundary zone(s) from which the prisms will be built, their heights, and the direction in which you want to build them.

You can also grow prisms on dangling walls. You can also specify different growth methods on adjacent walls in the domain. The mesher can automatically detect the collision between the prisms layers and avoid the collision by adjusting the height of the colliding prism layers.

Figure 12.1: Prism Shapes (p. 229) shows the types of prisms that can be generated.

Figure 12.1: Prism Shapes



Instructions for building prisms are provided in [Procedure for Generating Prisms \(p. 230\)](#). Details about how your inputs are used to generate the prisms are provided in [The Prism Generation Process \(p. 240\)](#). Problems related to prism generation are discussed in [Prism Meshing Problems \(p. 261\)](#).

12.2. Procedure for Generating Prisms

The procedure for creating prisms from boundary zones is as follows:

1. Check the boundary mesh to ensure that free nodes or faces with high skewness do not exist. See [Manipulating Boundary Nodes \(p. 119\)](#) and [Checking Face Distribution \(p. 380\)](#) for details.
2. Select the boundary zone(s) from which you want to grow prisms in the **Boundary Zones** selection list in the **Prisms** dialog box.

Mesh → Prisms...

Important

If you read in a mesh file created in a previous version, it is recommended that you reset all the prism parameters using the command `/mesh/prism/reset-parameters` before proceeding with setting the prism parameters.

3. Specify prism growth parameters in the **Growth** tab.
 - a. Select the appropriate **Offset Method** and **Growth Method** for growing prisms. See [Offset Distances \(p. 247\)](#) and [Prisms Dialog Box \(p. 539\)](#) for details.
 - b. Define the height of the first prism layer (**First Height/First Aspect Ratio**).
 - c. Set the growth method related parameter (**Slope**, **Rate**, or **Exponent**, as applicable) and the **Number of Layers**.
 - d. Specify options for prism growth, proximity detection, and splitting of prism layers (if needed) in the **Prisms Growth Options** dialog box (opened using the **Growth Options...** button), if required. Refer to [Prism Meshing Options \(p. 232\)](#) for details.
 - e. Click the **Plot** button to preview the height distribution.
- For information on setting different growth parameters for different zones, see [Growing Prisms Simultaneously from Multiple Zones \(p. 232\)](#).
4. Specify the direction for prism growth (see [Direction Vectors \(p. 244\)](#)).
 - a. Click the **Direction** tab in the **Prisms** dialog box to view the direction parameters.
 - b. Verify the orientation of the normals for the zones on which you want to grow prism layers. Ensure that the normals are pointing in the direction that you want to build the prisms.

You can use the **Color by Normal** option in the [Grid Colors Dialog Box \(p. 604\)](#) to verify the normal direction.

- If the boundary zone normals are incorrectly oriented, click the **Orient Normals...** button to open the **Orient Normals** dialog box. Specify the material point based on which the normals are to be oriented and click **Apply**.
- For mesh objects, use the options in the **Orient Mesh Object Face Normals** group box to orient the normals.
 - a. Select the mesh object in the **Object Name** drop-down list.
 - b. Select **Region** or **Material Point** as appropriate, and then select the region or material point in the drop-down list.
 - c. Ensure that **Select** is enabled, and enable **Select Walls** and/or **Select Baffles** as needed.
 - d. Click **Orient**.

Face boundary zone group(s) comprising the prism base zones will be created (prefixed by **_prisms**).

- c. Specify the method for determining the direction of the prisms:
 - Select **Normal** in the **Method** list to compute normal direction vectors. This method takes into account the change in direction required for a curved region. Specify the values for the maximum angle change.
 - Select **Uniform** in the **Method** list to use a constant, uniform direction for flat prism regions. Specify the appropriate direction vector.
5. Set the parameters for prism improvement and prism projection available in the **Improve** and **Project** tabs, respectively, as required.
 6. Click **Apply** to save your settings.
 7. Save the mesh.

File → Write → Mesh...

Important

It is a good practice to save the mesh at this point. If you are dissatisfied with the prisms generated, you can read in this mesh and modify the prism parameters to regenerate the prisms.

8. Click **Create** to generate the prisms.

Refer to [Prism Meshing Problems \(p. 261\)](#) for suggestions on solving problems occurring during prism generation.

9. Check the maximum face skewness value reported in the console when the final prism layer is created, to ensure that the value is acceptable.

This is especially important if you are going to generate tetrahedral or pyramidal cells using this new boundary zone.

The new zone should not have highly skewed triangles or quadrilaterals. For more information about the quality of quadrilateral faces used for creating pyramids, refer to [Creating Pyramids \(p. 221\)](#).

If the reported maximum skewness is too high, read the mesh file you saved after setting the prism parameters and try again with different parameters. Refer to [Prism Meshing Problems \(p. 261\)](#) for details.

10. A number of new zones are created when the prism generation is complete (see [Zones Created During Prism Generation \(p. 239\)](#)). If the prism-cap or prism-side face zones are *not* supposed to be walls (e.g., if they are supposed to be simple interior zones), change them to the appropriate boundary type.

Boundary → Manage...

For most cases, performing these steps will result in an acceptable mesh of prismatic cells, but for some complex geometries, you may have to modify the procedure. If you are not satisfied with the prisms generated, you can read in the mesh you saved before generating prisms and modify the prism parameters. Refer to subsequent sections for details on modifying the prism generation parameters to create a better mesh.

12.3. Prism Meshing Options

This section describes the various options available for creating a prism mesh in the domain.

[12.3.1. Growing Prisms Simultaneously from Multiple Zones](#)

[12.3.2. Growing Prisms on a Two-Sided Wall](#)

[12.3.3. Detecting Proximity and Collision](#)

[12.3.4. Ignoring Invalid Normals](#)

[12.3.5. Splitting Prism Layers](#)

[12.3.6. Preserving Orthogonality](#)

12.3.1. Growing Prisms Simultaneously from Multiple Zones

You can select multiple face zones for simultaneously growing prism layers. If the face zones are connected, the prisms grown will also be connected. A single zone of prism cells and a single zone of cap faces will be created for each set of simultaneously grown layers. To retain the individual prism cell zones and cap face zones, enable **Grow Individually** in the **Prisms Growth Options** dialog box (opened by clicking the **Growth Options...** button).

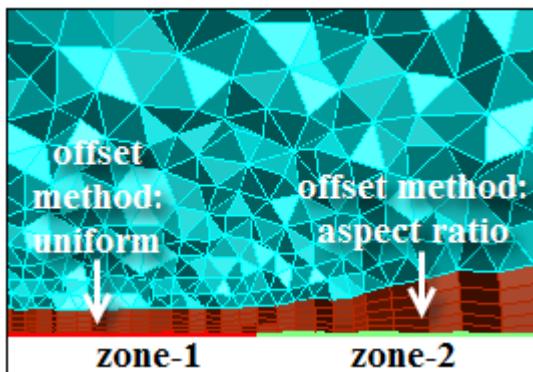
You can grow prisms from multiple zones with the same or different growth parameters.

- To use the same growth parameters for all zones, follow the steps in [Procedure for Generating Prisms \(p. 230\)](#). In this case, the same number of layers are grown from all zones, and all other parameters (growth method, offset method, direction method, etc.) also remain the same.
- To use different growth parameters for different zones, you can apply different growth methods, offset methods, first height, and other growth parameters (as required for the growth method selected) individually for each zone and click the **Apply** button in the **Zone Specific Growth** group box in the **Prisms** dialog box.

If you specify different growth parameters for different zones, all other parameters (direction method, offset method, etc.) can be set separately for each zone. In this case, the offset height of each node that is shared by multiple zones will be the average of the heights applied on the separate zones.

This produces a continuous transition between the zones ([Figure 12.2: Different Growth Parameters on Adjacent Zones \(p. 233\)](#)). Offset smoothing (see [Offset Smoothing \(p. 248\)](#)) is recommended in these cases to avoid sharp height changes at such edges.

Figure 12.2: Different Growth Parameters on Adjacent Zones



To assign different growth controls and different offset types to different face zones, replace steps 2 and 3 in [Procedure for Generating Prisms \(p. 230\)](#) with the following steps:

1. Select the zone(s) for which you want to specify a set of growth parameters in the **Boundary Zones** list in the **Prisms** dialog box.
2. Set the appropriate **Offset Method**, **Growth Method**, **First Height**, and the related growth parameters (as required) in the **Growth** tab.
3. Specify parameters for proximity/collision detection in the **Prisms Growth Options** dialog box (see [Detecting Proximity and Collision \(p. 235\)](#) for details).

Important

By default, a single zone of prism cells and cap faces is generated for the simultaneously grown prism layers. If you want to retain individual prism cell zones and cap faces, enable **Grow Individually** in the **General Options** group box in the **Prisms Growth Options** dialog box.

4. Click **Apply** in the **Prisms** dialog box.
5. Repeat this procedure for other zones for which you want to apply different growth controls.

6. Click **Create**.

Warning

Before you click the **Create** button, ensure that *all* the zones for which you want to grow prisms are selected in the **Boundary Zones** list.

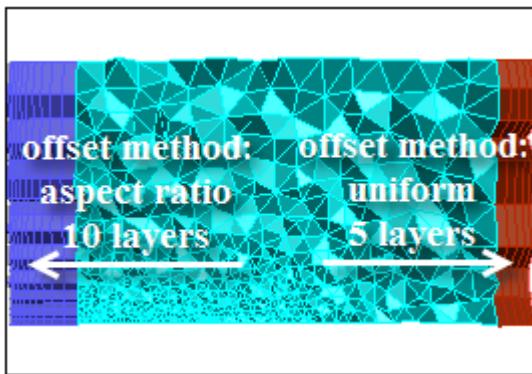
Note

When the **Prisms** dialog box is used to generate prism layers on multiple zones (adjacent or nonadjacent), the number of layers is determined by the chronologically last selected zone in the **Boundary Zones** selection list.

Important

- If you specify different number of layers to be grown on adjacent zones, the same number of layers (generally the smaller number of layers) will be grown on both the zones (see [Figure 12.2: Different Growth Parameters on Adjacent Zones \(p. 233\)](#)).
- If you specify different number of layers on multiple (nonadjacent) zones and use the **Prisms** dialog box to generate the prism layers, the same number of layers will be grown on the respective zones. The number of layers is determined by the chronologically last selected zone in the **Boundary Zones** selection list.
- If you specify different number of layers on multiple (nonadjacent) zones and use the **Auto Mesh** dialog box to generate the prism layers, the prism layers will be grown separately from each zone (see [Figure 12.3: Different Growth Parameters on Nonadjacent Zones—Using the Auto Mesh Option \(p. 234\)](#)).

Figure 12.3: Different Growth Parameters on Nonadjacent Zones—Using the Auto Mesh Option

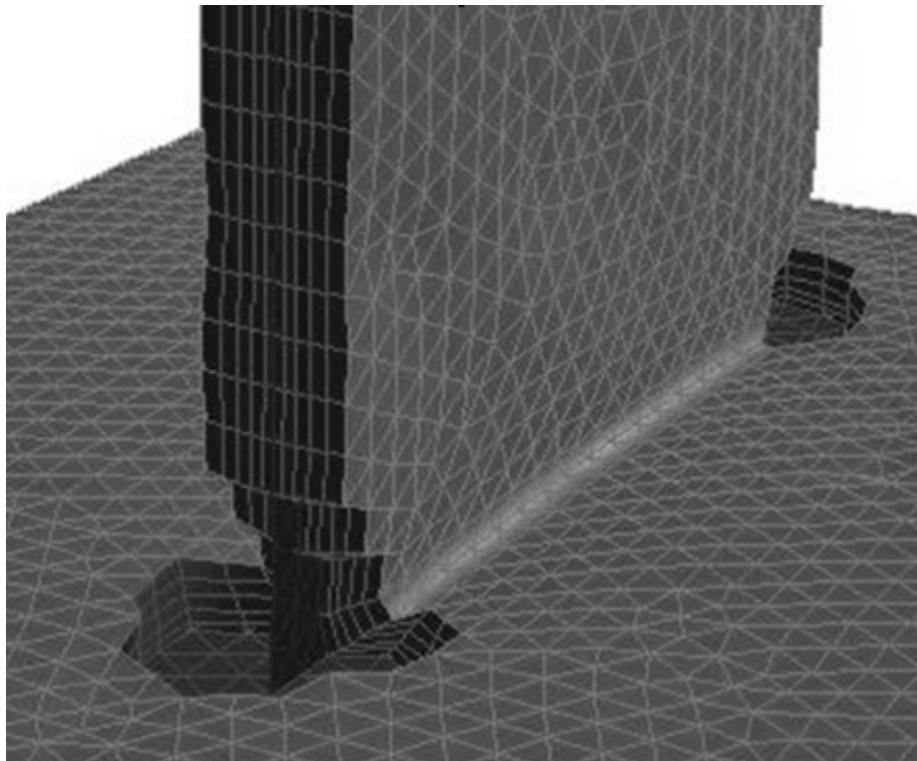


12.3.2. Growing Prisms on a Two-Sided Wall

Two-sided walls may be present in some geometries. To grow prisms on both sides of a two-sided wall (e.g., growing prisms on dangling walls), do the following:

1. Select the two-sided wall on which you want to grow prisms from the **Boundary Zones** selection list.
2. Specify the prism growth parameters as required.
3. Enable **Grow on Two Sided Wall** in the **Growth** tab of the **Prisms** dialog box.
4. Click **Create**. [Figure 12.4: Prism Growth on a Dangling Wall \(p. 235\)](#) shows an example of prisms grown on a dangling wall.

Figure 12.4: Prism Growth on a Dangling Wall



12.3.3. Detecting Proximity and Collision

If the zones on which you want to grow prisms are very close to each other, the prism layers from zones may intersect or collide with each other. This results in bad quality of prism layers.

For example, in [Figure 12.5: Collision of Prism Layers \(p. 236\)](#), the prism layers grown from proximal surfaces intersect each other. The **Allow Shrinkage** option allows you to avoid the intersection of prism layers by adjusting the height of the prism layers in closely placed zones. [Figure 12.6: Prism Layers Shrunk to Avoid Collision \(p. 236\)](#) shows the use of this option to avoid intersection of the prism layers by adjusting the prism layer height.

Figure 12.5: Collision of Prism Layers

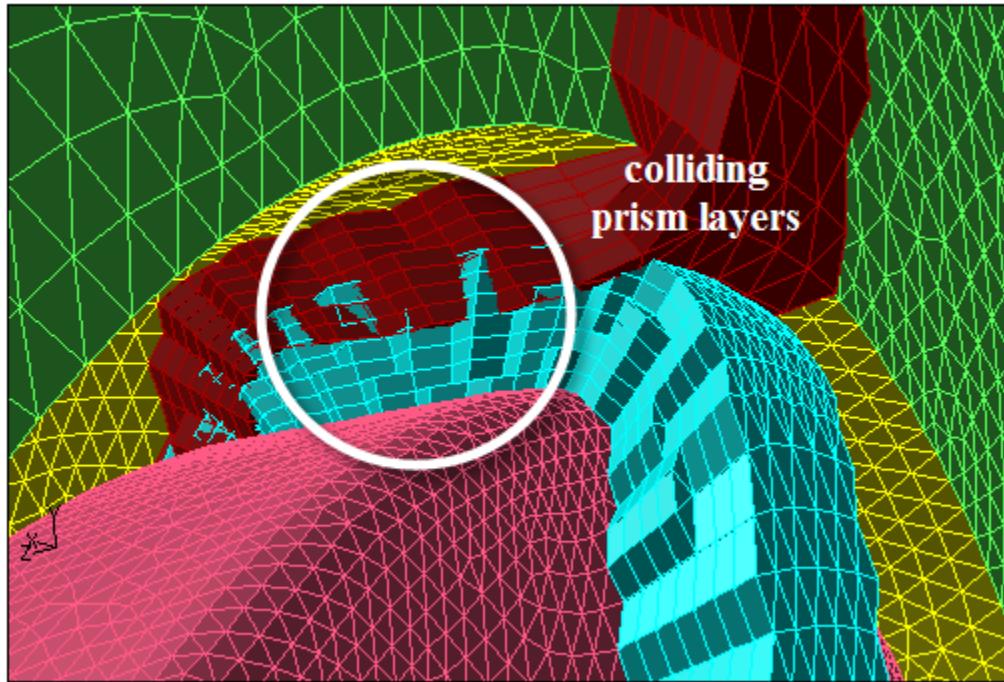
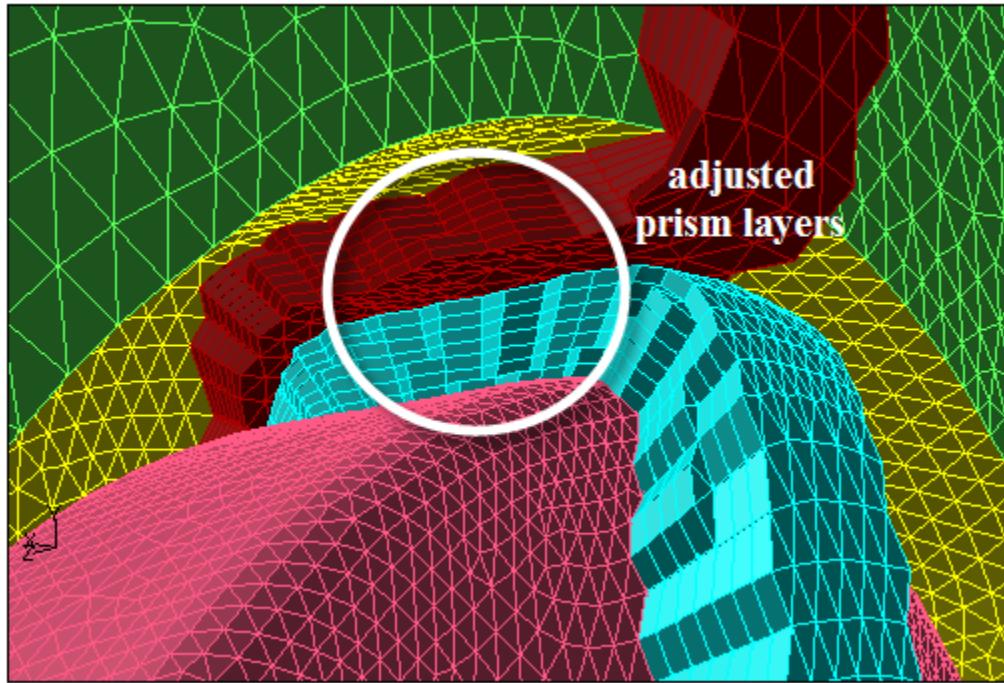


Figure 12.6: Prism Layers Shrunk to Avoid Collision



A value of n for **Gap Factor** implies that a gap equal to n times the maximum base edge length at the node in question will be maintained. Hence, a value of 1 implies that the gap maintained is equal to the maximum base edge length at the node considered.

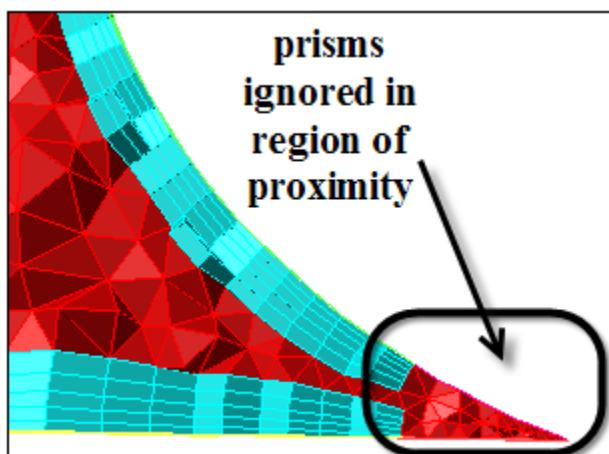
Important

Offset heights are scaled only in the regions where the intersection of prism layers takes place. For the remaining regions, the prisms are created according to the specified parameters.

You can also ignore prisms in regions of close proximity during prism generation. The area of proximity will be determined based on the shrink factor specified. Prism layers will be grown from the remaining geometry according to the specified parameters.

[Figure 12.7: Ignoring Areas of Proximity \(p. 237\)](#) shows a portion of the geometry where the region of proximity in a sharp corner has been ignored while creating prisms.

Figure 12.7: Ignoring Areas of Proximity



For automatic adjustment of intersecting/colliding prism layers while growing prisms, do the following:

1. Specify the prism growth parameters.
2. Click the **Growth Options...** button in the **Growth** tab to open the **Prisms Growth Options** dialog box.
3. Enable the appropriate options in the **General Options** group box.
4. Specify the parameters required for the proximity calculation and controlling the prism layer height.
 - a. Enable **Allow Shrinkage** to allow shrinkage of prism layers in areas of proximity. This option is enabled by default.
 - b. Enable **Keep First Layer Offsets**, if required.

This option allows you to keep the original first offset height and scale the offset heights for the remaining layers in case of intersecting prism layers.

- c. Specify an appropriate value for **Gap Factor**.
- d. Enable **Allow Ignore** to ignore the prism growth parameters in areas of proximity.

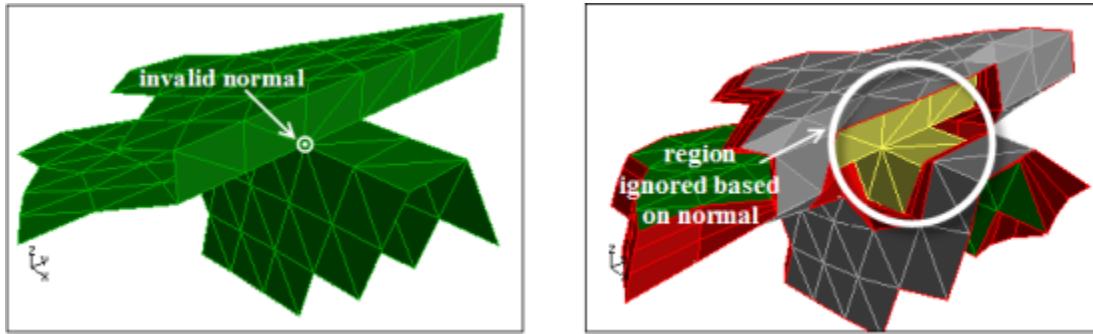
- e. Enter an appropriate value for **Max Aspect Ratio** or **Max Shrink Factor** as required.
 - f. Click **Apply** and close the **Prism Growth Options** dialog box.
5. Click **Create** in the **Prisms** dialog box.

12.3.4. Ignoring Invalid Normals

Some geometries may contain regions where the normals considered while growing prisms may be invalid. The normal at a particular node may be nearly tangential to the surrounding faces in the geometry and as a result, the prism generation may fail. For such cases, you can choose to ignore regions of the geometry where invalid normals exist, during prism generation. Prism layers will be grown from the remaining geometry according to the specified parameters.

[Figure 12.8: Ignoring Invalid Normals \(p. 238\)](#) shows a portion of the geometry where the normal at the highlighted node will be nearly tangential to some of the surrounding faces. You can see that the region around this node has been ignored while creating prisms.

Figure 12.8: Ignoring Invalid Normals



You can ignore regions based on normals while creating prisms as follows:

1. Specify the prism growth parameters.
2. Select the **Boundary Zone** containing the invalid normal and click the **Growth Options...** button to open the **Prisms Growth Options** dialog box.
 - a. Enable **Ignore Invalid Normals** in the **General Options** group box.
 - b. Click **Apply** and close the **Prism Growth Options** dialog box.
3. Grow prisms as required.

12.3.5. Splitting Prism Layers

You can generate fewer prism layers and then split them as part of the prism generation process to generate the total number of prism layers required. This option is faster than generating the same total number of prism layers.

You can specify the number of divisions per prism layer as follows:

1. Specify the prism growth parameters.

2. Click **Growth Options...** to open the **Prisms Growth Options Dialog Box** (p. 549).
3. Enable **Split** in the **Split Options** group box.
4. Specify the **Divisions Per Layer**.
5. Click **Apply** and close the **Prism Growth Options** dialog box.

12.3.6. Preserving Orthogonality

In some cases, it may be important for the prismatic mesh to be orthogonal near the original boundary. For such cases, there are two ways to preserve orthogonality:

- Decrease the value specified for the **Max. Angle Change** in the **Direction** tab of the **Prisms** dialog box to a lower value to limit the change in normal direction in each prism layer.

This parameter is set to 45 degrees by default. This means the normal direction at a node can change up to 45 degrees during normal, edge, and node smoothing. For example, if you reduce this parameter to 10 degrees, the change in normal direction will be more gradual, and the mesh near the original boundary will be nearly orthogonal.

- Specify an explicit number for **Orthogonal Layers** in the **Direction** tab of the **Prisms** dialog box.

When you specify orthogonal layers, you should decrease the **Max Angle Change** to approximately 10 degrees. This will prevent sudden sharp changes in normal direction at the first non-orthogonal layer.

For a non-orthogonal layer, edge swapping normal, edge, or node smoothing will be performed so that the layer is orthogonal to the original boundary. For example, if you specify 5 orthogonal layers, no smoothing will be performed on the first 5 layers, resulting in direction vectors that are normal to the original surface mesh faces. Full smoothing is used on the sixth and subsequent layers.

Important

If you are preserving orthogonality near the original boundary, make sure that the prism layers do not grow too quickly. Use small layer heights, relative to the sizes of the faces on the original boundary. If prism layers grow quickly, the nearly-orthogonal direction vectors are likely to cross at sharp concave corners, causing the prism generation to fail. The prism layer generation will be stopped at this point. See [Normal Smoothing \(p. 245\)](#).

Preserving orthogonality can affect skewness and lead to left-handed faces and/or negative volumes. See [Negative Volumes/Left-Handed Faces/High Skewness \(p. 262\)](#) for details.

12.4. Zones Created During Prism Generation

When prisms are generated, several new cell and face zones are created. You can identify the new zones by their default names.

- The cell zone containing the prisms will be named `prism-cells-#`.
- The face zone created at the end of the last prism layer will be named `prism-cap-#`.

- If the prism layers are bounded by one or more existing triangular face zones, each of these will be retriangulated so that the faces of the zone that are adjacent to the prisms will become quadrilateral faces, while the rest of the faces in the zone remain triangles.

If an original triangular face zone is called `symmetry`, the portion of it that still contains triangles will be retained as `symmetry`, while the portion containing quadrilateral faces will be named `symmetry-quad:#`.

- Any faces that were not projected to adjacent zones (see [Using Adjacent Zones as the Sides of Prisms \(p. 241\)](#)) are collected into a zone named `prism-side-n`.
 - For a zone `wall-#`, the zone created by ignoring prism growth in the region of proximity will be named as `wall-#:ignore`, where, # is the zone number assigned.
-

Important

All the ignored threads related to a base thread will be merged into a single thread by default. You can however change this using the command `/mesh/prism/controls/merge-ignored-threads?` which will generate more than one thread per base thread.

To merge one or more prism cell zones with other cell zones (and merge the corresponding face zones), use the **Merge** option in the [Manage Cell Zones](#) dialog box.

By default, the `prism-cap-#` zones will be wall zones. If you do not want to include these walls in the model, change them to interior zones (or any other type) using the [Manage Face Zones](#) dialog box.

12.5. The Prism Generation Process

The following process is used to build the prisms and to create a high-quality mesh.

1. Boundary mesh analysis:

Analyze the initial surface mesh to determine if any adjacent zones should be projected to and retriangulated (if triangular), or if their faces should be used as the sides of new prism cells (if quadrilateral).

For details on determining whether to use existing boundary zones or not, see [Using Adjacent Zones as the Sides of Prisms \(p. 241\)](#).

2. Prism layer growth.

Grow layers of prisms one by one, based on the layer height specification. This is a multistep procedure, and hence you can modify or skip some of the steps:

- Determine the initial direction vectors so that the direction in which to build the prisms is determined. Several advancement direction methods are available, as discussed in [Direction Vectors \(p. 244\)](#).
- Smooth the initial direction vectors. See [Normal Smoothing \(p. 245\)](#) for details.

- c. Determine the initial offset distances so that the mesher will know how far from the corresponding nodes on the previous layer to place the new nodes that define the prisms. See [Offset Distances \(p. 247\)](#) for details.
 - d. Smooth the offset distances are smoothed to eliminate spikes and dips in the new prism layer. See [Offset Smoothing \(p. 248\)](#) for details
 - e. Project new nodes along the outer edges of the prism layer to adjacent zones, based on the analysis in step 1.
 - f. Create prism faces and cells using the new nodes. The new faces at the top of the prism layer are referred to as cap faces.
 - g. The new cap faces of skewness value greater than the specified threshold are improved by edge swapping or edge node smoothing. See [Edge Swapping and Smoothing \(p. 249\)](#) for details.
 - h. Apply global smoothing across the new surface. This is also skewness-driven, but is based on the nodes not the faces. See [Node Smoothing \(p. 250\)](#) for details.
 - i. Check the quality of all new cells and faces.
 - j. If quality problems are detected, prism layer creation is stopped. If more layers are to be grown, the process is repeated starting from substep (a).
3. Zone clean up: After all the prism layers have been grown, the new faces along the outer sides of the layers are moved to zones of the same types as the zones to which their nodes are projected. Unprojected faces are collected in a single prism-side zone.
4. Retriangulation: Adjacent triangular zones that have been projected to are automatically retriangulated. Triangles that are overlapped by new quadrilateral faces from the prism layers are removed. Most of the original nodes are used in the new triangulation. The result is a conformal interface between the sides of the prism layers and the adjacent triangular zones.

12.6. Using Adjacent Zones as the Sides of Prisms

For each zone adjacent to the zone(s) from which you grow prisms, the angle between the prism growth direction and the adjacent zone will be checked. Depending on the size of this angle, the mesher will decide whether to use the adjacent zone for the prism sides, or create a new zone.

Using Adjacent Quadrilateral Face Zones

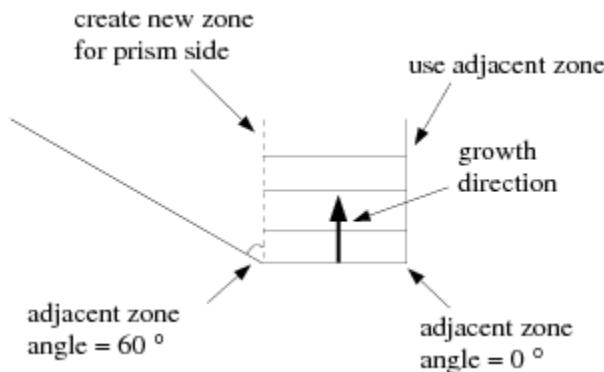
If an adjacent zone has quadrilateral faces, it will be used without modification as the prism-side boundary, and it satisfies the following requirements:

- It must share nodes with the boundary zone from which you are building the prisms i.e., there must be no free nodes (see [Free and Isolated Nodes \(p. 119\)](#)) where the zones touch.
- The angle between the adjacent zone and the prism growth direction must be less than the specified threshold, **Max. Adjacent Zone Angle** in the **Project** tab of the **Prisms** dialog box. The default threshold value is 45 degrees.

For example, if you are growing prisms from the bottom surface in [Figure 12.9: Effect of Adjacent Zone Angle \(p. 242\)](#) using the default maximum adjacent zone angle of 45 degrees, the zone on the left will

be excluded, but the zone on the right will be used as it is. A new zone will be created for the left sides of the prisms.

Figure 12.9: Effect of Adjacent Zone Angle



To fill in the gap on the left side of [Figure 12.9: Effect of Adjacent Zone Angle \(p. 242\)](#), create pyramids using the prism side faces, create a domain, and generate tetrahedral cells.

Projecting to Adjacent Triangular Face Zones

If an adjacent zone has triangular faces, two additional steps will be performed to incorporate the zone into the prism layers: projection and retriangulation. The mesher will project the outer nodes of the prisms onto the triangular faces of the adjacent zone, provided it satisfies the following requirements:

- It must share nodes with the boundary zone from which you are building the prisms. That is, there must be no free nodes (see [Free and Isolated Nodes \(p. 119\)](#)) where the zones touch.
- The angle between the adjacent zone and the prism growth direction must be less than the specified threshold, **Max. Adjacent Zone Angle** (in the **Project** tab of the **Prisms** dialog box). The default threshold value is 45 degrees.

In [Figure 12.9: Effect of Adjacent Zone Angle \(p. 242\)](#), the zone on the right will be projected to and retriangulated, while the zone on the left will be excluded.

Retriangulation

As shown in [Figure 12.1: Prism Shapes \(p. 229\)](#), the sides of the prism will always be quadrilateral faces, regardless of the type of face from which the prism is built (triangular or quadrilateral). If the prism-side nodes are projected to an adjacent triangular boundary face zone:

1. The prism-side faces on the shared boundary will be overlaid on the triangular faces of the existing boundary zone.
2. The triangular boundary zone will be retriangulated so that the triangular portion of the zone ends at the border of the portion now filled with quadrilateral faces. This is done to obtain a conformal mesh where all the nodes match up at face edges.
3. The triangles that were overlapping the quadrilateral prism-side faces will be deleted.

[Figure 12.10: Symmetry Zone and Car Wall Before Prism Generation \(p. 243\)](#) shows the initial boundary mesh for a car and the symmetry boundary beside it. If the prisms are generated from the car body *without* retriangulation of the triangular symmetry boundary, the resulting boundary mesh will be as

shown in [Figure 12.11: Symmetry Zone and Car Wall After Prism Generation Without Retriangulation \(p. 243\)](#). With retriangulation, the boundary mesh will appear as in [Figure 12.12: Symmetry Zone and Car Wall After Prism Generation and Retriangulation \(p. 244\)](#).

Figure 12.10: Symmetry Zone and Car Wall Before Prism Generation

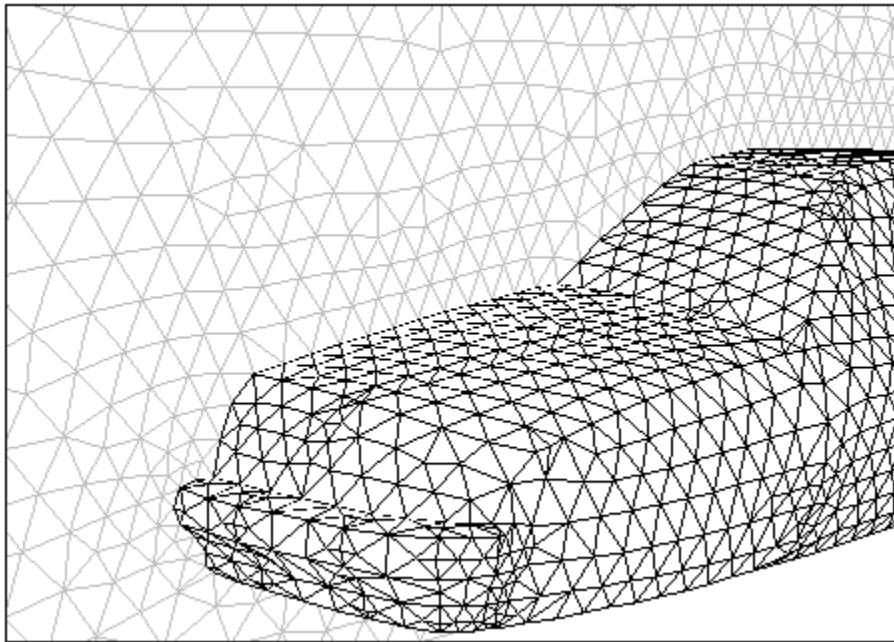


Figure 12.11: Symmetry Zone and Car Wall After Prism Generation Without Retriangulation

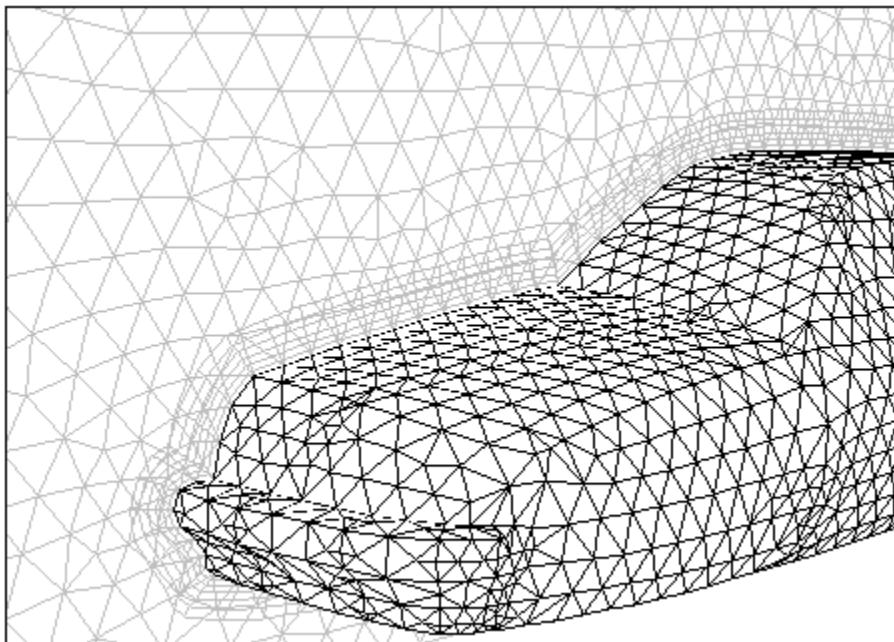
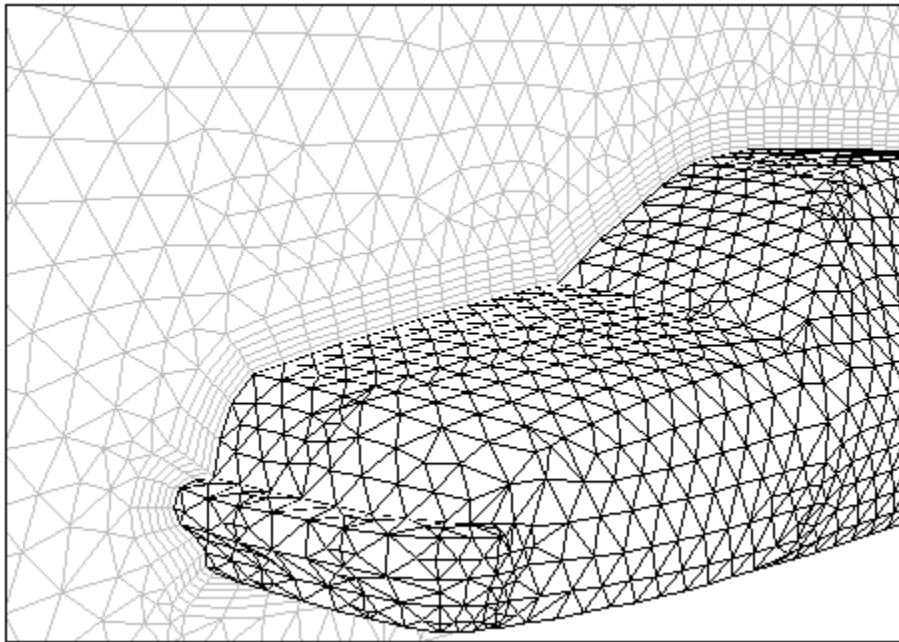


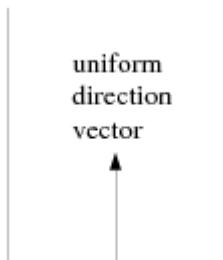
Figure 12.12: Symmetry Zone and Car Wall After Prism Generation and Retriangulation**Important**

Though it is possible to disable retriangulation (by disabling the **Retriangulate Adjacent Zones** option in the **Project** tab of the **Prisms** dialog box), it is not always recommended. Expert users experimenting with different settings may temporarily disable retriangulation.

12.7. Direction Vectors

As each layer of prisms is built, the mesher needs to know which direction to build in. There are two methods available for determining the direction vectors:

- **Extrusion Method:** For creating straight-sided prism regions without any curvature, you can specify a uniform direction vector for all prism layers. Select **Uniform** in the **Method** list in the **Direction** tab of the **Prisms** dialog box.

Figure 12.13: Uniform Direction Vector for a Straight-Sided Prism Region

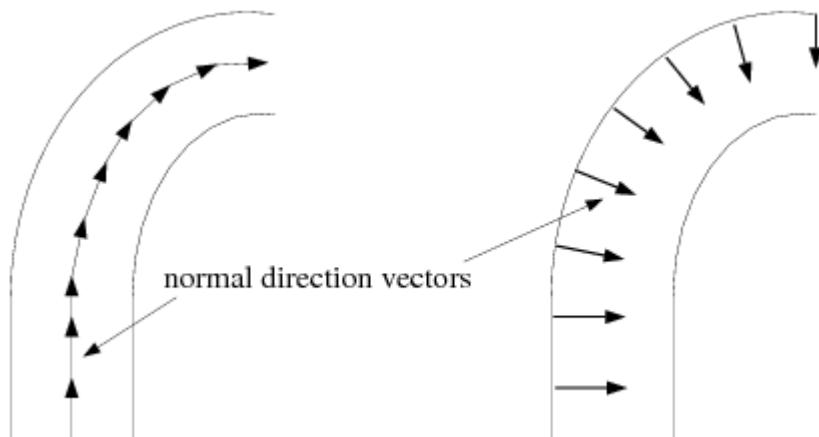
Specify the uniform direction vector or click **Compute** in the **Vector** group box. This constant vector will be used for all prism layers, instead of computing a new direction for each layer (see [Figure 12.13: Uniform Direction Vector for a Straight-Sided Prism Region \(p. 244\)](#)).

Note

The **Grow On Two Sided Wall** option cannot be used when the **Uniform** method is selected.

- **Normal Method:** For regions with curvature, the appropriate normal direction vector at each node will be determined, since it may be different for each node and each layer, see [Figure 12.14: Normal Direction Vectors for a Curved Prism Region \(p. 245\)](#). Select **Normal** in the **Method** group box in the **Direct** tab of the **Prisms** dialog box.

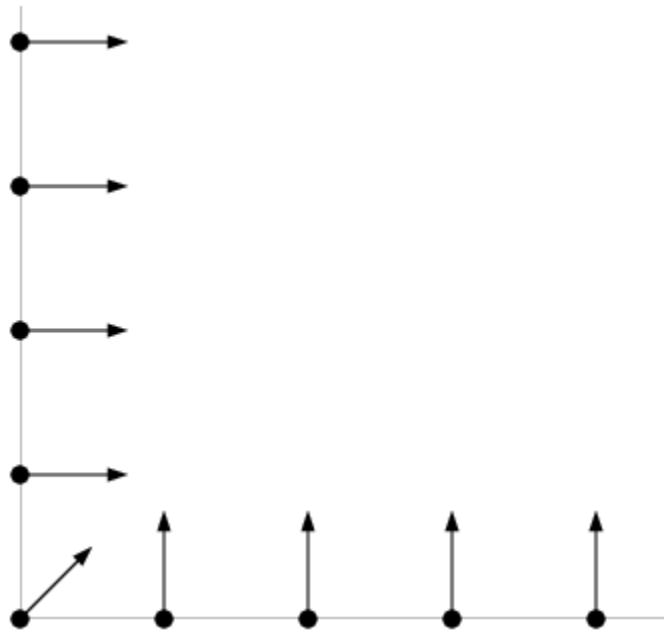
Figure 12.14: Normal Direction Vectors for a Curved Prism Region



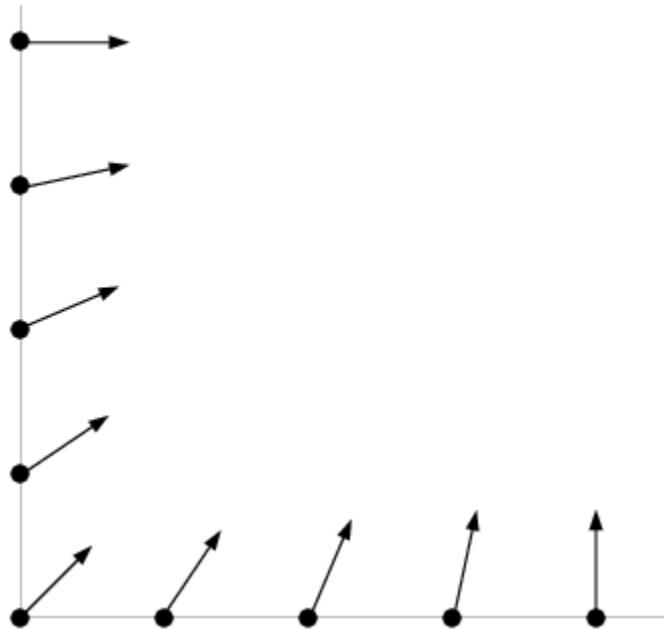
Normal Smoothing

The normal direction vectors obtained using the methods described in [Direction Vectors \(p. 244\)](#) are smoothed so that there is a gradual change in direction from one node to the next. This will reduce the chance of direction vector intersection, which causes the prism generation to fail. This step is not necessary when a uniform direction vector is used, since there is no change in direction from one node to the next.

[Figure 12.15: Normal Direction Vectors Before Smoothing \(p. 246\)](#) shows normal direction vectors in the vicinity of a sharp (90 degree) corner.

Figure 12.15: Normal Direction Vectors Before Smoothing

The direction vector is at an angle of 45 degree at the corner, while elsewhere it is 0 or 90 degrees. These direction vectors will intersect and prisms cannot be generated. [Figure 12.16: Normal Direction Vectors After Smoothing \(p. 246\)](#) shows the normal direction vectors near the corner after they have been smoothed. After smoothing, the normal direction changes more gradually from node to node.

Figure 12.16: Normal Direction Vectors After Smoothing

It is possible to create very thin prism layers using the direction vectors shown in [Figure 12.15: Normal Direction Vectors Before Smoothing \(p. 246\)](#), as long as the last layer lies below the point where the vectors cross.

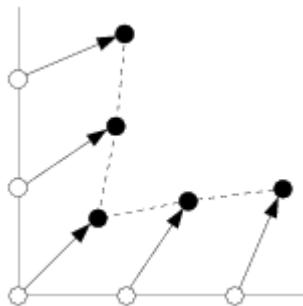
12.8. Offset Distances

The offset distance for a given node is the distance between adjacent layers at that node. This distance is based on the prism layer height computed from the specified prism growth parameters. The new node for a prism layer is placed at this distance along the direction vector.

There are four methods available for determining the offset distances:

- **Uniform Offset Distance Method:** In this method, every new node (child) is initially the same distance away from its parent node (i.e., the corresponding node on the previous layer, from which the direction vector is pointing).

Figure 12.17: Uniform Offset Distance Method

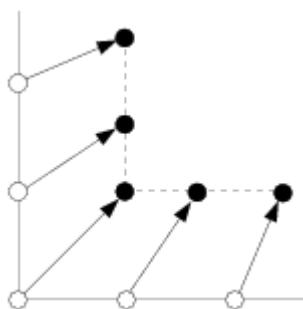


Uniform distances are fine for planar surface meshes, but they result in crevices at sharp corners, for example, the 90 degree corner in [Figure 12.17: Uniform Offset Distance Method \(p. 247\)](#). The direction vector at the corner node will be at an angle of 45 degrees, while the vectors at the adjacent nodes will be slightly more orthogonal to their faces. For uniform distances, this places the nodes for the first layer as shown in [Figure 12.17: Uniform Offset Distance Method \(p. 247\)](#).

The dashed line connecting the new nodes begins to pinch inward at the corner. The distances between parent and child nodes are still all the same, but the angles of the direction vectors introduce this pinching effect. Without some sort of correction such crevices can eventually collapse.

- **Minimum Height Offset Distance Method:** In this method, child nodes are guaranteed to be *at least* as far from their parent nodes as the distance computed for the current layer from the growth inputs. The mesher will also try to retain the shape of the original boundary.

Figure 12.18: Minimum-Height Offset Distance Method



For a perfect 90 degree corner, this results in a perfect 90 degree corner for the next layer (see [Figure 12.18: Minimum-Height Offset Distance Method \(p. 247\)](#)). Thus, in this method the shape is better preserved if no additional smoothing is applied.

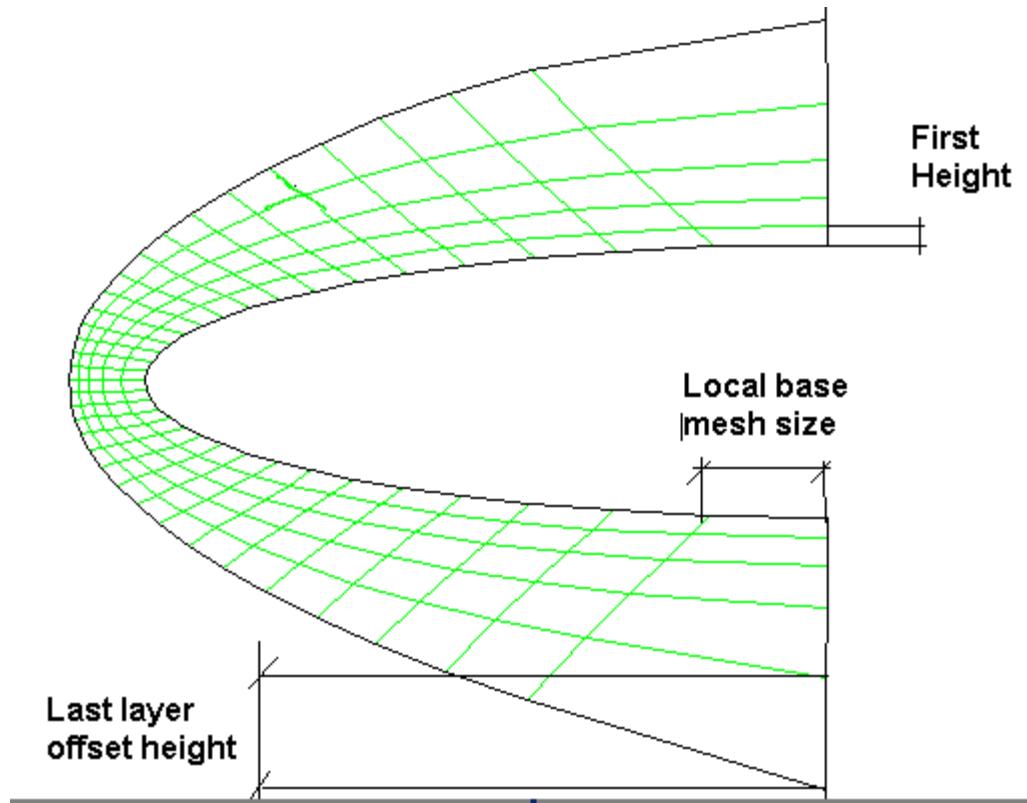
The minimum height method have some limitations. If the angle of the direction vector at the corner is not 45 degrees (i.e., if it does not exactly bisect the angle between the faces), the minimum-height method can increase skewness.

- **Aspect Ratio Method:** This method allows you to control the aspect ratio of the prism cells that are extruded from the base boundary zone. The *aspect ratio* is defined as the ratio of the prism base length to the prism layer height. The various growth methods (constant, linear, geometric, and exponential) can be used to specify the aspect ratio of the first layer.

When a non-constant growth method is used, the aspect ratio for subsequent layers will change accordingly. The heights of the prism cells will vary according to the local face sizes in the surface mesh, providing a convenient way to automatically vary prism heights across a zone.

- **Last Ratio Method:** This method also allows you to control the aspect ratio of the prism cells that are extruded from the base boundary zone. You can specify **First Height** for the first prism layer. If you select this method, the various growth methods will not be accessible. The last aspect ratio method is explained in [Figure 12.19: Last Ratio Method \(p. 248\)](#).

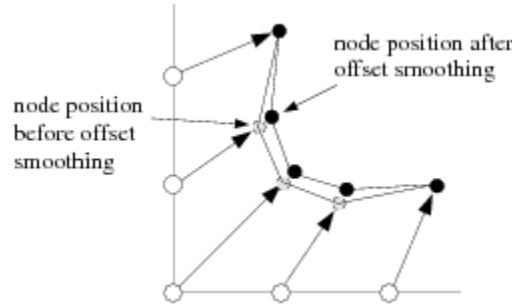
Figure 12.19: Last Ratio Method



Local base mesh size is used to find out the offset height for the last layer. For example, if you specify 80 as the **Last Percent** value the offset height of the last layer will be 0.8 times the local base mesh size. Local growth rate is used to calculate the other intermediate offset heights exponentially.

Offset Smoothing

The purpose of offset smoothing is to eliminate spikes and dips in the new surface layer. Smoothing is applied iteratively. [Figure 12.20: Effect of Offset Smoothing \(p. 249\)](#) shows how the nodes are moved during offset smoothing.

Figure 12.20: Effect of Offset Smoothing

12.9. Improving Prism Quality

The following operations are available for improving the quality of the prism faces:

- 12.9.1. Edge Swapping and Smoothing
- 12.9.2. Node Smoothing
- 12.9.3. Improving the Prism Quality
- 12.9.4. Removing Poor Quality Cells
- 12.9.5. Improving Warp

12.9.1. Edge Swapping and Smoothing

After the new cap faces are created for the new layer, the skewness is compared with the specified skewness threshold (**Skewness** in the **Swapping and Smoothing** group box in the **Improve** tab of the **Prisms** dialog box). If the skewness of a face is too high, the following improvement procedure is carried out:

1. Swap the longest edges of highly skewed faces.

When edge swapping occurs on a cap face, swapping propagates downward. Swapping must be performed on the corresponding faces on all prism layers, including the face on the original boundary mesh, so that the grid lines within the prism layers do not cross.

Important

Swapping is *not* performed if it will significantly alter the geometry defined by the original boundary mesh.

2. If swapping is not performed, the skewness will be improved by smoothing the nodes of the skewed faces using one of the following operations.
 - If the sharpest angle at the node (i.e., the most acute angle between adjacent faces that use the node) is nearly 180 degrees, smoothing will be performed (as described in [Node Smoothing \(p. 250\)](#)) to move the nodes.

- If the sharpest angle is far from 180 degrees, the smoothing will be limited so that the nodes are only allowed to move along the face's longest edge. This method will prevent the collapse of sharp corners in the mesh.

Note

Smoothing does *not* propagate downward, it is performed only on the skewed cap faces themselves.

Important

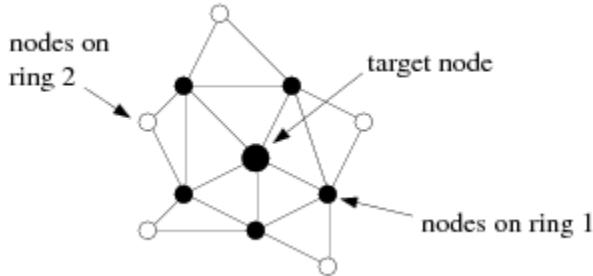
This edge swapping/smoothing procedure is not performed on quadrilateral cap faces.

12.9.2. Node Smoothing

If the edge swapping and smoothing are not sufficient to reduce the skewness of some faces, you can include an additional smoothing operation. In this method, if any of the faces surrounding a new node have a skewness greater than the specified threshold, the node is smoothed.

Instead of moving only the node, this method will also move the surrounding nodes to make space for the node to move.

Figure 12.21: Node Smoothing in Rings



The surrounding nodes are grouped and smoothed in rings (see [Figure 12.21: Node Smoothing in Rings \(p. 250\)](#)). The nodes on the outermost ring are smoothed first, etc., until the target node (on ring zero) is finally smoothed.

This type of smoothing is helpful in concave areas where the advancing layers continually decrease the available surface area, causing squashing of triangular faces. By smoothing outer rings of nodes first, more space is provided for the nodes that need to be moved in order to reduce the skewness. Thus, this method performs regional smoothing around faces of high skewness. The node smoothing procedure can be performed on both tri and quad faces.

12.9.3. Improving the Prism Quality

You can improve prism cell quality in a postprocessing step after all the required prism layers are created. The [Prism Improve Dialog Box \(p. 581\)](#) contains options that allow you to improve the prism cell quality based on the quality measure selected. The quality measures available are ICEM CFD quality, orthoskew, skewness, or squish. The following options are available:

- The **Smooth** option allows optimization based smoothing of prism cells. Poor quality cells can be identified based on the quality measure selected. The nodes of cells with quality worse than the specified **Max Cell Quality** value will be moved to improve quality. The cell aspect ratio will also be maintained based on the value specified for max-aspect-ratio.
- The **Improve** option collects and smooths cells in layers around poor quality cells. Poor quality cells can be identified based on the quality measure selected. Cells with quality worse than the specified **Max Cell Quality** value will be identified, and the nodes of the cells surrounding the poor quality cells will be moved to improve quality. The cell aspect ratio will also be maintained based on the value specified for max-aspect-ratio.
- The **Smooth and Improve** option uses a combination of node movement and optimized smoothing to improve the quality. This option is a combination of the **Smooth** and **Improve** options. The cell aspect ratio will also be maintained based on the value specified for max-aspect-ratio.

When the **Smooth and Improve** option is enabled in the **Post Operations** group box in the **Improve** tab of the **Prisms** dialog box, the prism cells will be improved based on the options selected in the [Prism Improve Dialog Box \(p. 581\)](#), after the required prism layers are created.

You can alternatively use the options in the [Prism Improve Dialog Box \(p. 581\)](#) to improve the prism cell quality in a postprocessing step after the mesh is created.

Mesh → Tools → Prism → Improve...

12.9.4. Removing Poor Quality Cells

In cases where the prism mesh is created separately (without using the **Merge Cell Zones** option in the **Auto Mesh** tool) and the **prism-cap** zone exists, you can use additional options to remove layers of poor quality cells in regions of poor quality and sharp corners.

- The [Prism Post Ignore Dialog Box \(p. 585\)](#) contains options that allow you to remove poor quality prism cells based on quality, intersection, interior warp, and feature edges. You can specify the number of cell rings to be removed around the marked cells. Cells will be marked for removal in regions of sharp corners based on the **Ignore Options** selected and then extended based on the number of cell rings specified. Additional cells will be marked for removal in regions of high aspect ratio and feature angle (if selected in the **Expand Ignore Options** group box) around the exposed prism side.

The **Feature Edges** tab contains options for manipulating feature edges to be used for the post-ignore operation. You can extract edge zones from the prism base zones when the prism cap zone is selected in the **Boundary Zones** list. The edge zones are extracted based on the **Feature Angle** specified. Other operations like merging or separating the edge zones are also available.

The boundary will be smoothed at feature corners after the prism cells have been removed. The prism-side faces exposed by the removal of the prism cells will be collected in a zone named `prism-side-#`, while for a zone `wall-#`, the faces corresponding to the ignored prism cells will be collected in a zone named `wall-# : ignore`. You can also optionally smooth the prism side nodes from the base node to the cap node to create better triangles for the non-conformal interface.

When the **Ignore** option is enabled in the **Post Operations** group box on the **Improve** tab of the **Prisms** dialog box, the prism cells will be removed based on the options selected in the [Prism Post Ignore Dialog Box \(p. 585\)](#), after the required prism layers are created.

Note

The **Feature Edge** option, under **Ignore Options** in the **Prism Post Ignore** dialog box, requires a prism-cap zone to be present so it cannot be used as a **Post Operation** control during prism generation.

You can alternatively use the options in the [Prism Post Ignore Dialog Box \(p. 585\)](#) to remove layers of poor quality cells in a postprocessing step after the mesh is created.

Mesh → Tools → Prism → Post Ignore...

- The [Prism Tet Improve Cavity Dialog Box \(p. 588\)](#) contains options for creating a cavity in regions where the prism quality is adequate, but the quality of adjacent tetrahedra is poor. The cavity is created based on the tetrahedral cell zone, the quality measure and the corresponding threshold value specified. Additional cells will be removed based on the number of expand cell rings specified. You can create a cavity comprising only tetrahedral cells or optionally include prism cells when the cavity is created. When prism cells are also included in the cavity, you can specify whether the non-conformal interface is to be created.

Mesh → Tools → Prism → Tet Improve Cavity...

12.9.5. Improving Warp

Improving warp in prisms is a postprocessing step which is carried out after all the required prism layers are created. The face warp in the generated prism faces is improved by moving the nodes of the face to make it planar.

When the **Improve Warp** option is enabled in the **Post Operations** group box in the **Improve** tab of the **Prisms** dialog box, the prism layer(s) having faces with warp ≥ 0.5 are identified. The nodes on the identified faces are moved and the new location of the node is updated only when:

1. The overall maximum warp of the faces connected to the node has decreased.
2. The overall maximum skewness of the cells connected to the node has decreased or remained the same.

This is useful in regions having complex sharp corners, where the nodes of a prism cell may have drastic normal change.

12.10. Text Commands for Generating Prisms

The text commands for generating prisms are listed below. Some of them allow additional control that is not available via the **Prisms** dialog box.

/mesh/prism/create

creates prism layers on the specified face zone(s) according to the specified parameters.

/mesh/prism/controls/adjacent-zone/project-adjacent-angle

sets the tolerance used to determine whether or not to use an adjacent zone. If a zone shares outer nodes with any of the zones from which the layers are being grown (the “base zones”), its angle with

respect to the growth direction is compared with this value. If the angle is less than or equal to this value, then the zone will be used. See [Using Adjacent Zones as the Sides of Prisms \(p. 241\)](#) for details.

/mesh/prism/controls/adjacent-zone/project-converged

sets the convergence criterion for iterative projection. This is non-dimensionalized by the offset height at each local node.

/mesh/prism/controls/adjacent-zone/project-iter

sets the maximum number of iterations to perform when projecting to multiple zones. Most projections converge in only a few iterations. There is normally no need to change this value.

/mesh/prism/controls/adjacent-zone/project?

enables/disables projection of outer nodes to adjacent zones.

/mesh/prism/controls/adjacent-zone/retri-feature-angle

allows you to specify the feature angle that should be prevented while generating prisms.

/mesh/prism/controls/adjacent-zone/retriangulate-adjacent?

specifies whether or not adjacent triangular face zones to which outer nodes have been projected will be automatically retriangulated. This option is on by default.

/mesh/prism/controls/adjacent-zone/side-feature-align-angle

specifies the angle used for aligning projected normals along a feature edge.

/mesh/prism/controls/adjacent-zone/side-feature-angle

specifies the angle used for computing the feature normals.

/mesh/prism/controls/adjacent-zone/side-topology-align-angle

specifies the angle used for aligning projected normals along a particular feature edge based on the topology. This is particularly useful when the side-feature-angle specified is not sufficient to decide the feature edge to align the projected normals.

/mesh/prism/controls/auto-separate-cells?

enables/disables automatic separation of the cells extruded from different face zones to different cell zones.

Note

This option is not considered when the auto-mesh option is used to generate the mesh.

/mesh/prism/controls/check-quality?

enables/disables the checking of volume, skewness, and handedness of each new cell and face.

/mesh/prism/controls/grow-individually?

specifies whether prisms should be grown from multiple zones individually so that the individual prism cell zones and the individual cap face zones are retained.

/mesh/prism/controls/improve/cell-quality-improve?

allows you to improve cell quality for every layer by smoothing normals in the current layer. In addition, perturbation smoothing will be performed to improve cell quality in the lower layer, when the quality measure is set to either skewness, squish, ICEM CFD quality, or orthoskew. The poor quality elements are identified based on the value set for max-allowable-cell-skew. The cell aspect ratio will also be maintained during the cell quality improvement.

/mesh/prism/controls/improve/check-allowable-skew?

allows you to check the skewness of the prism cap for every layer.

/mesh/prism/controls/improve/check-size?

enables the checking of cell size during the generation of each prism layer. An error will be reported if zero-area prism cells are generated and prism layer growth will be stopped.

When the check-size? option is disabled, the zero-area prism cells generated can be removed during the improvement smoothing operations performed at the end of the prism generation procedure.

/mesh/prism/controls/improve/corner-height-weight?

when enabled, the offset height at corners with large angles (e.g., 270°) is reduced to give a smoother prism cap.

/mesh/prism/controls/improve/edge-smooth?

enables/disables local smoothing of nodes of the longest edges of skewed faces.

/mesh/prism/controls/improve/edge-smooth-angle

specifies the maximum allowable angle between the normals of adjacent cap faces for skewness-driven edge smoothing.

/mesh/prism/controls/improve/edge-swap?

enables/disables edge swapping to decrease the skewness of highly skewed faces.

/mesh/prism/controls/improve/edge-swap-base-angle

specifies the maximum allowable angle between a pair of base face normals for skewness-driven edge swapping.

/mesh/prism/controls/improve/edge-swap-cap-angle

specifies the maximum allowable angle between a pair of cap face normals for skewness-driven edge swapping.

/mesh/prism/controls/improve/face-smooth?

allows you to enable face-driven smoothing to improve skewness.

/mesh/prism/controls/improve/face-smooth-converged

specifies the convergence criteria for cap face smoothing.

/mesh/prism/controls/improve/face-smooth-rings

specifies the number of rings to propagate skewness during smoothing of cap faces.

/mesh/prism/controls/improve/face-smooth-skew

specifies the minimum skewness to smooth cap faces.

/mesh/prism/controls/improve/identify-feature-line?

allows you to smooth the normal along the feature lines of the base face zones, during normal smoothing. This option is disabled by default.

/mesh/prism/controls/improve/improve-warp?

enables/disables improving of face warp during prism generation. This option is disabled by default.

/mesh/prism/controls/improve/left-hand-check

specifies checking for left-handedness of faces. A value of 0 implies face handedness will not be checked, 1 implies only cap faces will be checked, while 2 implies faces of all cells in current layer will be checked.

/mesh/prism/controls/improve/max-allowable-cap-skew

specifies the maximum allowable skewness for cap faces after smoothing. Prism layer growth will be stopped if any cap face has skewness greater than the specified value.

/mesh/prism/controls/improve/max-allowable-cell-skew

specifies the cell quality criteria for smoothing and quality checking.

/mesh/prism/controls/improve/node-smooth?

allows you to enable node-driven smoothing to improve skewness.

/mesh/prism/controls/improve/node-smooth-angle

refers to the maximum deviation of a node's sharpest angle (i.e., the most acute angle between adjacent faces that use the node) from 180 degrees. The node will be smoothed only if its sharpest angle falls within this range. For example, if the specified value is 45 degrees, nodes with sharpest angles between 135 and 225 degrees can be smoothed.

/mesh/prism/controls/improve/node-smooth-converged

sets the convergence criterion for node smoothing. If the node positions are changing by less than this value, smoothing iterations will stop.

/mesh/prism/controls/improve/node-smooth-iter

specifies the maximum number of node smoothing iterations to be performed for the nodes on each layer. These iterations will be performed until the convergence criterion is reached.

/mesh/prism/controls/improve/node-smooth-local?

allows you to enable node smoothing to converge locally. This is useful for large geometries.

/mesh/prism/controls/improve/node-smooth-rings

controls the locality of node smoothing by setting the number of rings around each node to be smoothed.

- If zero, only the node itself is smoothed.
- If one, the node and all of its neighbors are smoothed.
- If two, the neighbors of the neighbors are also smoothed, and so on.

/mesh/prism/controls/improve/post-improve?

allows you to perform prism height adjustment based on growth rate.

/mesh/prism/controls/improve/shrink-left-handed-cap?

allows you to enable shrinking of prism layers to get rid of left handed faces.

/mesh/prism/controls/improve/smooth-improve-prism-cells?

allows you to set the parameters for improving the prism cells after the required prism layers are created. You can select optimized smoothing (smooth), node movement (improve), or a combination of both to improve the quality. Specify the quality measure to be used, the cell quality threshold, the number of improvement iterations, and the minimum improvement required.

/mesh/prism/controls/improve/swap-smooth-skew

specifies the skewness threshold for edge swapping and edge and node smoothing. The faces with skewness greater than or equal to the specified value will be swapped and/or smoothed.

/mesh/prism/controls/merge-ignored-threads?

allows you to automatically merge all ignored zones related to a base thread into a single thread. This option is enabled by default. When this option is disabled, more than one thread per base thread will be generated.

/mesh/prism/controls/morph/improve-threshold

specifies the quality threshold used for improving the quality during the morphing operation.

/mesh/prism/controls/morph/morphing-convergence-limit

specifies the convergence limit for the morphing operation. The morpher uses an iterative solver. It is assumed to have converged when the relative residual is less than this number.

/mesh/prism/controls/morph/morphing-frequency

specifies the frequency of the morphing operation. The number specified denotes the number of prism layers after which the morpher is applied to the remainder of the mesh (e.g., a value of 5 indicates that the morpher is applied to the mesh after every 5 prism layers grown).

/mesh/prism/normal/bisect-angle

is required for growing prisms out of sharp interior corners. When the value of this angle is set, the normals are automatically projected onto the plane bisecting the angle between faces having an interior angle less than this angle.

/mesh/prism/normal/compute-normal

computes the normal vector for the specified boundary face zone.

/mesh/prism/normal/converge-locally?

specifies whether or not the normal smoothing at each node is frozen once convergence is satisfied at that node. If not, all normals are continuously smoothed until all of them have converged.

/mesh/prism/controls/normal/direction-method

specifies whether the prism layers should be grown normal to surfaces or along a specified direction vector.

/mesh/prism/controls/normal/ignore-invalid-normals?

allows you to ignore nodes which have poor normals.

/mesh/prism/controls/normal/intersect-detect?

enables the adjustment of normals to avoid their intersection.

/mesh/prism/controls/normal/intersect-detect-angle

specifies the angle based on which normal intersection is determined. The normal intersection is determined in cases when the feature angle of the manifold of a node is less than the value specified.

/mesh/prism/controls/normal/max-angle-change

specifies the maximum angle by which the normal direction at a node can change during smoothing.

/mesh/prism/controls/normal/normal-method

specifies the method to use for normal direction computation (see [Direction Vectors \(p. 244\)](#)).

acute-bisection

is a variation of that proposed by Kallinderis et al. [7 (p. 681)], [8 (p. 681)]. The faces surrounding a node are analyzed to find those whose planes form the most acute angle. The resulting normal lies on the plane bisecting those two planes.

This method is computationally more expensive than surface averaging, but is less robust at singular points where convex edges meet smooth surfaces. For example, if a box is sitting on the table, such a point would be where one of the box's corners meets the table top.

face-average

computes the normal at a node by averaging the normals of the surrounding faces.

hybrid-normal

computes the normal by all the methods available and uses the best quality normal obtained. This is the default method.

minimized-angle

is a variation of that proposed by Pirzadeh [14 (p. 681)], [15 (p. 681)]. An initial normal at a node is determined by averaging the normals of the surrounding faces (the face-average method). Iterations are then performed to minimize the largest angle between the node normal and the normals of the surrounding faces.

This method is a computationally more expensive than surface averaging, and can produce slightly better normals when the uniform offset method is used and the surface mesh does not contain sharp edges. When the minimum-height offset method is used, this direction method may introduce spikes and pits in the advancing layer surfaces.

surface-average

is similar to face-average, but handles sharp edges better.

/mesh/prism/controls/normal/orient-mesh-object-face-normals

allows you to orient the face normals for mesh object boundary zones. Specify the mesh object, region or material point as appropriate, and specify whether walls, baffles or both comprising the prism base zones are to be separated and oriented.

/mesh/prism/controls/normal/orthogonal-layers

specifies the number of layers to preserve orthogonality. All smoothing is deferred until after these layers have been generated.

/mesh/prism/normal-smooth?

enables or disables smoothing of normal direction vectors.

/mesh/prism/normal-smooth-converged

sets the convergence criterion (in degrees) for normal smoothing. If the normal directions are changing by less than this value, smoothing iterations will stop.

/mesh/prism/normal-smooth-iter

specifies the maximum number of normal smoothing iterations to be performed for the normal vectors on each layer. These iterations will be performed until the convergence criterion is reached.

/mesh/prism/controls/offset/first-aspect-ratio-min

specifies the minimum first aspect ratio (ratio of prism base length to prism layer height) for the prism cells.

/mesh/prism/controls/offset/min-aspect-ratio

specifies the minimum aspect ratio (ratio of prism base length to prism layer height) for the prism cells.

/mesh/prism/controls/offset/smooth?

enables/disables offset distance smoothing.

/mesh/prism/controls/offset/smooth-converged

sets the convergence criterion for offset smoothing. If the offset heights change by less than this value, smoothing iterations will stop.

/mesh/prism/controls/offset/smooth-iter

sets the maximum number of offset smoothing iterations to be performed. These iterations will be performed until the convergence criterion is reached.

/mesh/prism/controls/post-ignore/post-remove-cells?

allows you to set the parameters for removing poor quality prism cells after the required prism layers are created. You can remove cells based on quality, intersection, interior warp, and feature edges. Specify options for removing additional cells in regions of high aspect ratio and feature angle, the number of cell rings to be removed around the marked cells, and options for smoothing the prism boundary and prism side height.

/mesh/prism/controls/proximity/allow-ignore?

allows you to ignore nodes where specified maximum shrink factor cannot be maintained.

/mesh/prism/controls/proximity/allow-shrinkage?

allows you to enable shrinkage while growing prism layers.

/mesh/prism/controls/proximity/gap-factor

specifies the gap between prism layers in the proximity region.

/mesh/prism/controls/proximity/keep-first-layer-offsets?

allows you to retain first layer offsets while performing proximity detection.

/mesh/prism/controls/proximity/max-aspect-ratio

specifies the maximum allowable cell aspect ratio to determine the limit for the shrinkage of prism layers. This option is available only when the allow-ignore? option is disabled. The default value is 50.

Note

If the cell aspect ratio exceeds the specified maximum aspect ratio, a message will appear during the prism meshing process, indicating that shrinkage was limited by the maximum aspect ratio specified.

/mesh/prism/controls/proximity/max-shrink-factor

specifies the shrink factor determining the maximum shrinkage of the prism layers. This option is available only when the allow-ignore? option is enabled.

/mesh/prism/controls/proximity/smoothing-rate

specifies the rate at which shrinkage is propagated in lateral direction.

/mesh/prism/controls/remove-invalid-layer?

removes the last prism layer if it fails in the quality check.

/mesh/prism/controls/split?

allows you to set parameters for splitting the prism layers after the initial prism layers are generated, to generate the total number of layers required. Specify the number of divisions per layer.

/mesh/prism/controls/zone-specific-growth/apply-growth

applies the zone-specific growth parameters specified.

/mesh/prism/controls/zone-specific-growth/clear-growth

clears the zone-specific growth specified.

/mesh/prism/controls/zone-specific-growth/list-growth

lists the zone-specific growth parameters specified for individual zones in the console.

/mesh/prism/improve/improve-prism-cells

collects and smooths cells in layers around poor quality cells. Poor quality cells can be identified based on the ICEM CFD quality, orthoskew, skewness, or squish measures. Cells with quality worse than the specified threshold value will be identified, and the nodes of the cells surrounding the poor quality cells will be moved to improve quality. The cell aspect ratio will also be maintained based on the value specified for max-aspect-ratio.

/mesh/prism/improve/smooth-brute-force?

forcibly smooths cells if cell skewness is still high after regular smoothing.

/mesh/prism/improve/smooth-cell-rings

specifies the number of cell rings to be considered around the skewed cells (to be used by `improve-prism-cells`).

/mesh/prism/improve/smooth-improve-prism-cells

uses a combination of node movement and optimized smoothing to improve the quality. This command is a combination of the `smooth-prism-cells` and `improve-prism-cells` commands. The cell aspect ratio will also be maintained based on the value specified for `max-aspect-ratio`.

/mesh/prism/improve/smooth-prism-cells

allows optimization based smoothing of prism cells. Poor quality cells can be identified based on the ICEM CFD quality, orthoskew, skewness, or squish measures. The nodes of cells with quality worse than the specified threshold value will be moved to improve quality. The cell aspect ratio will also be maintained based on the value specified for `max-aspect-ratio`.

/mesh/prism/improve/smooth-sliver-skew

specifies the skewness above which prism cells will be smoothed.

/mesh/prism/list-parameters

lists all prism mesh parameters in the console.

/mesh/prism/mark-ignore-faces

allows you to mark the faces to be ignored during prism meshing.

/mesh/prism/mark-nonmanifold-nodes

allows you to mark the non-manifold prism base nodes. A list of the non-manifold nodes will be printed in the console. The faces connected to the non-manifold nodes will also be marked. You can use this command after specifying zone-specific prism settings, prior to generating the prisms to verify that non-manifold configurations do not exist.

/mesh/prism/post-ignore/create-cavity

creates a cavity in regions where prism quality is adequate, but the quality of adjacent tetrahedra is poor. The cavity is created based on the tetrahedral cell zone, the quality measure and the corresponding threshold value, and the additional number of cell rings specified. You can create a cavity comprising only tetrahedral cells or optionally include prism cells in the cavity created. When prism cells are also included in the cavity, you can specify whether the non-conformal interface is to be created.

/mesh/prism/post-ignore/mark-cavity-prism-cap

marks the prism cap faces and tetrahedral cell faces bounding the cavity to be created in regions where prism quality is adequate, but the quality of adjacent tetrahedra is poor. Specify the tetrahedral cell zone, the quality measure and the corresponding threshold value to be used, and the additional number of cell rings based on which the cavity will be created.

/mesh/prism/post-ignore/mark-prism-cap

marks the prism cap faces for ignoring prism cells in regions of poor quality cells and sharp corners. Specify the prism cell zone and the basis for ignoring prism cells and the relevant parameters. The prism cells can be ignored based on quality, intersection, (both enabled by default), warp, and features (both disabled by default). Specify the quality measure and threshold value to be used for ignoring cells based on quality and (if applicable) the feature edges for ignoring cells based on features. Additionally, specify whether cells are to be marked in regions of high aspect ratio and based on feature angle, and the additional number of cell rings based on which prism cells will be removed.

/mesh/prism/post-ignore/post-remove-cells

allows you to remove prism cells in layers around poor quality cells and sharp corners. Specify the prism cell zone, the basis for ignoring prism cells (quality, intersection, warp, features) and the relevant parameters. Specify the number of cell rings to be removed around the marked cells. Cells will be marked for removal in regions of sharp corners based on quality, intersection, warp, and features (as applicable) and then extended based on the number of cell rings specified. Additional cells will be marked for removal in regions of high aspect ratio and based on feature angle (if applicable) around the exposed prism side. The boundary will be smoothed at feature corners after the prism cells have been removed. The prism-side faces exposed by the removal of the prism cells will be collected in a zone named `prism-side-#`, while for a zone `wall-#`, the faces corresponding to the ignored prism cells will be collected in a zone named `wall-#:ignore`. You can also optionally smooth the prism side nodes from the base node to the cap node to create better triangles for the non-conformal interface.

/mesh/prism/quality-method

specifies the quality method used during prism generation.

/mesh/prism/reset-parameters

resets the prism parameters to their defaults.

Important

If you read in a mesh file created in a previous version, you can reset all the prism parameters using this command before proceeding with setting the prism parameters.

/mesh/prism/split/split

allows you to split the prism layers after the initial prism layers are generated, to generate the total number of layers required. Specify the prism cell zones to be split and the number of divisions per layer. You can also choose to use the existing growth rate (default) or specify the growth rate to be used while splitting the prism layers.

/mesh/manage/revolve-face-zone

generates prisms by revolving a specified face zone through a specified angle about a specified rotation origin and axis to produce a polar mesh.

12.11. Prism Meshing Problems

This section discusses a number of common problems that you may encounter when generating prismatic meshes. An appropriate solution is also recommended for each problem. If the prism generation fails or is unsatisfactory, you may read in the mesh you saved before starting the prism generation (in step 6 of [Procedure for Generating Prisms \(p. 230\)](#)) and repeat the process, incorporating the appropriate corrections.

The most common prism meshing problems are:

- Orientation
- Retriangulation failures
- Too many or too few nodes, or unknown face combinations
- Negative volumes, left-handed faces, or high skewness
- Growing too far
- Large jumps in prism height at the edges of layers

Orientation

If the faces of the boundary zone from which you are building prisms are not all oriented in the same direction, you will be alerted that the faces have been reoriented, and the prism generation process will proceed.

Retriangulation Failures

If nodes, where adjacent zones meet the zones (from which prisms are growing) are duplicated, the mesher will be unaware that the zones are connected. It will ignore such zones when considering nodes for projection, and hence retriangulation is likely to fail.

Solution: To verify this problem check the messages printed before the first prism layer. It should contain a list of all the zones connected to the zones from which you are growing the prisms.

1. Make sure there are no free nodes in the surface mesh before growing any prisms.
2. Use the **Count Free Nodes** button in the **Merge Boundary Nodes** dialog box to check for free nodes.
3. Enable the **Free** option in the **Faces** section of the **Display Grid** dialog box to see where the free nodes are located.

Too Many/Few Nodes or Unknown Face Combinations

If you receive the following type of messages, then you are most likely growing prisms into existing cells.

```
Warning: wedge cell c887 has too many nodes
Warning: wedge cell c1928 has too few nodes
Warning: base_mask = 7 (0x7), unknown face combinations
```

Either you have already grown prisms from the selected zones, or you are growing them in the wrong direction into cells on the other sides of the selected zones. In the latter case, reorient the normals and growing the layers. See step 4 in [Procedure for Generating Prisms \(p. 230\)](#).

Negative Volumes/Left-Handed Faces/High Skewness

The prism layer creation process will automatically stop if negative cell volumes, left-handed faces, or high skewness are detected.

Left-handed faces are faces that have collapsed into their cells. You can view them by enabling the **Left Handed** option in the **Faces** section of the **Display Grid** dialog box. Very high skewness occurs when the value of skewness is greater than the specified **Max. Allowable Skewness** in the **Improve** section of the **Prisms** dialog box). These problems can occur where prism layer fronts are advancing too quickly in areas of high curvature.

For example, the first layer height may be greater than the minimum edge length of your original surface mesh. You should compare the **First Height** with the results of clicking the **Edge Size Limits** button in the **Growth** section of the **Prisms** dialog box. The **First Height** should usually be smaller than the minimum edge size.

Solution: If the **First Height** is reasonable but you are unable to successfully generate your requested **Total Height**, try one of the following solutions:

- Reduce the **First Height** or the parameter used for the growth **Method** (i.e., **Slope**, **Rate**, or **Exponent**), and increase the **Number of Layers**. Extra layers allow smoothing to compensate better for sharp edges and corners.
- If you have specified a nonzero value for **Orthogonal Layers** (in the **Direct** section of the **Prisms** dialog box), change it to zero. Increasing orthogonality can reduce robustness.
- If you have reduced the **Max. Angle Change** (in the **Direct** section of the **Prisms** dialog box) from its default of 70 degrees to improve orthogonality, reset it to 70.
- If you have disabled edge swapping or normal, offset, or edge smoothing, turn them back on.
- If you are using the minimum-height offset method (enabled with the `mesh/controls/prism/offset-method` command), switch back to the uniform method.
- Lower the **Skewness for Swapping and Smoothing** in the **Improve** section of the **Prisms** dialog box.
- Increase the number of **Layers** in the **Improve** section of the **Prisms** dialog box if it is smaller than the number of layers being grown.
- Enable **Smooth Nodes** in the **Improve** section of the **Prisms** dialog box. You can also increase the number of smoothing rings, using the command:
`/mesh/controls/prism/node-smooth-rings`
- If you are growing simultaneously from multiple zones, use zone-specific growth controls. Use smaller initial heights and/or growth rates for zones that are having problems.

Growing Too Far

The problem here is growing too many layers. Ideally, to provide for a smooth transition to the nearly-equilateral tetrahedra, the volume of the tetrahedra should be the local growth rate times the volume of the last layer prism cells.

Solution: To ensure this does not happen, verify that the prism parameters satisfy the above condition.

Large Jumps in Prism Height

If the prisms use quadrilateral faces of adjacent zones, and the node spacings on these faces are very different from the layer heights, the prism heights jump at the outer edges.

Solution: To fix this problem, replace the adjacent quadrilateral face zones with triangular face zones, using the /boundary/remesh/triangulate text command. When you reattempt the prism generation, the zone will be projected to and retriangulated.

Since the outer nodes of the prisms are projected to the adjacent boundary zone, and the original nodes on that portion of the zone are discarded, the different node spacing on the adjacent zone will not affect the prism heights at the outer edges.

Chapter 13: Generating Thin Volume Mesh

This chapter describes the procedure for creating a swept mesh in a thin volume region.

This chapter comprises the following sections:

- [13.1. Overview](#)
- [13.2. Procedure for Generating a Thin Volume Mesh](#)
- [13.3. Text Command for Generating Thin Volume Mesh](#)

13.1. Overview

Thin Volume Mesh creates sweep-like mesh for a body occupying a thin gap. You define boundary face zones for source and target such that the source face normal should point to the target. The source face mesh may be triangles or quads. Four other controls are then used by Fluent Meshing to grow the volume mesh from the source zone to the target zone.

13.2. Procedure for Generating a Thin Volume Mesh

The steps to generate a Thin Volume Mesh are described below:

1. Check the source boundary mesh to ensure that free nodes or faces with high skewness do not exist. See [Manipulating Boundary Nodes \(p. 119\)](#) and [Checking Face Distribution \(p. 380\)](#) for details.
2. Open the [Thin Volume Mesh Dialog Box \(p. 551\)](#).

Mesh > Thin Volume Mesh

3. Select the **Source Boundary Face Zone(s)** from the list.
4. Select the **Target Boundary Face Zone(s)** from the list.
5. Specify the **Gap Thickness**. If set to 0, the mesher will automatically calculate the gap thickness. If non-zero, the Gap Thickness defines the maximum separation between source and target zones in the swept-mesh region.
6. Specify the **Number of Divisions** between source and target faces.
7. Specify the **Growth Rate** between adjacent layers.
8. Choose **Remesh overlap zones** behavior. Remesh overlap zones replaces any overlapped part of the surface mesh on the target and adjacent faces. Original meshes in these areas are replaced.
9. Click **Create** to calculate the thin solid mesh.
10. Click **Draw** to display the mesh in the graphics window of the GUI.

13.3. Text Command for Generating Thin Volume Mesh

The text command for generating a Thin Volume Mesh is:

mesh/thin-volume-mesh

create

Entering this command in the TUI will start a dialog in which you specify the source face(s), target face(s), gap thickness, number of divisions, growth rate, and remesh overlap zones. See the following example dialog.

```
Meshing/mesh/thin-volume-mesh/create
()
Source face zones(1) [() ] source*
Source face zones(2) [() ]
()
Target face zones(1) [() ] target
Target face zones(2) [() ]
Gap Thickness [0] 1
Number of Divisions [1] 5
Growth Rate [1]
Remesh Overlap Zones? [yes]
Start thin-gap mesh:
- Create and project nodes and faces on targets ...
- Project nodes on targets ...
- Create layer nodes ...
- Create side quad faces ...
- Create interior layer cells ...
- Retriangulate target face zones ...
- Retriangulate adjacent face zones ...
Done
```

Chapter 14: Generating Tetrahedral Meshes

This chapter describes the procedure to create the tetrahedral mesh in the domain. It also describes some of the common problems faced during tetrahedral meshing. It contains the following sections:

- 14.1. Automatically Creating a Tetrahedral Mesh
- 14.2. Manually Creating a Tetrahedral Mesh
- 14.3. Initializing the Tetrahedral Mesh
- 14.4. Refining the Tetrahedral Mesh
- 14.5. Additional Text Commands for Tetrahedral Mesh Generation
- 14.6. Common Tetrahedral Meshing Problems

You can use one of the following techniques to generate tetrahedral meshes:

- Automatic mesh generation (see [Automatically Creating a Tetrahedral Mesh \(p. 267\)](#))
- Manual mesh generation (see [Manually Creating a Tetrahedral Mesh \(p. 271\)](#))
- A combination of manual and automatic commands (see [Initializing the Tetrahedral Mesh \(p. 274\)](#), [Refining the Tetrahedral Mesh \(p. 276\)](#))

If the cells (e.g., prisms or pyramids) have already been created in some portion of the computational domain, you need to create a domain encompassing the region to be meshed with tetrahedral cells before starting with the tetrahedral mesh generation. See [Using Domains to Group and Mesh Boundary Faces \(p. 368\)](#) for details. In such cases, all automatic and/or manual meshing actions will apply only to the active domain.

Alternatively, instead of creating a new domain, you can specify the cell zone(s) to be preserved while generating the tetrahedral mesh, before proceeding with the mesh generation. You can use the command /mesh/tet/preserve-cell-zone to specify the cell zone(s) to be preserved.

14.1. Automatically Creating a Tetrahedral Mesh

The **Auto Mesh** submenu in the **Mesh** menu contains commands that encapsulate the recommended volume mesh generation strategy. The starting point for this procedure is a valid surface mesh.

- 14.1.1. Automatic Meshing Procedure for Tetrahedral Meshes
- 14.1.2. Using the Auto Mesh Tool
- 14.1.3. Automatic Meshing of Multiple Cell Zones
- 14.1.4. Automatic Meshing for Hybrid Meshes
- 14.1.5. Further Mesh Improvements

14.1.1. Automatic Meshing Procedure for Tetrahedral Meshes

The automatic tetrahedral mesh generation process is divided into two fundamental tasks: initialization and refinement. The automatic initialization procedure comprises the following steps:

1. Merging free nodes.
2. Deleting unused nodes.

3. Improving the surface mesh.
 4. Initializing the mesh.
 5. Generating and separating cell regions.
-

Important

If the mesh fails to initialize, typically this indicates a problem with the surface mesh. In some rare cases, the sliver size (which is automatically computed) may need to be changed. This may happen for domains in which the minimum boundary face size is very small as compared to the domain extent. When changing the sliver size, the rule of thumb is that the specified value should be of the order 1e-12 times the minimum face area.

The automatic refinement procedure comprises the following steps:

1. Sorting the boundary faces by size.
2. Refining the boundary cells.
3. Reverse sorting cells by skewness.
4. Refining the active cell zone(s).
5. Swapping cells based on skewness.
6. Reverse sorting cells by skewness.
7. Smoothing the mesh.
8. Removing boundary slivers.

You can select either the advancing front refinement, or the skewness-based refinement of the tetrahedral mesh. The refinement procedure is repeated a number of times.

You can control the number of repetitions for skewness-based refinement by setting the **Number of Levels** in the **Tet Refine Controls** dialog box. For each subsequent level of refinement, the cell skewness thresholds are lowered. Additional refinement levels increase the mesh resolution (and the number of cells) and decrease the average cell skewness.

Each refinement level actually consists of two sweeps through the refinement procedure:

- One sweep at the appropriate skewness threshold for that particular level of refinement.
- Another sweep with a high skewness threshold and a relaxed **Min Boundary Closeness** set in the **Tet Refine Controls** dialog box.

This sweep attempts to reduce highly skewed cells in confined regions of the geometry.

Boundary slivers will be removed after the refinement sweeps are complete if the **Remove Slivers** option is enabled in the **Refinement** tab of the **Tet** dialog box.

In some rare cases, it may happen that the first sweep of the first refinement level will never finish because more skewed cells are formed due to refinement, which will be further refined to create more

skewed cells, and so on. Such a problem may occur when there is a problem with the boundary mesh, such as rapid mesh transition in small gaps, etc. This problem can be avoided by using the incremental improvement option for the refinement parameters or by increasing the **Min Node Closeness** value in the **Tet Refine Controls** dialog box.

You can select the fast transition option for the refinement parameters in order to generate a mesh with a smaller number of cells. When this option is selected, appropriate refinement parameters will be used to generate fewer cells during meshing. The rate of change of the cell size is increased in order to reduce the number of cells generated. Alternatively, you can specify an appropriate cell sizing option with a sufficiently large value for the growth rate.

Important

Various steps in the automatic meshing process can be added or eliminated using the **Tet** dialog box. In addition, most aspects of the automatic meshing process can be changed dynamically through the Scheme interface. Contact your support engineer for details.

14.1.2. Using the Auto Mesh Tool

Mesh → Auto Mesh...

The **Auto Mesh** tool can be used to automatically do either of the following functions:

- Generate a tetrahedral volume mesh starting from a boundary mesh.
- Generate a tetrahedral volume mesh in an unmeshed domain of a mesh that contains other cell shapes.

The **Auto Mesh** tool allows you generate the tetrahedral volume mesh automatically. When you select **Tet** in the **Volume Fill** list and click the **Mesh** button in the **Auto Mesh** dialog box:

- If the volume mesh does not exist, all the steps listed in [Automatic Meshing Procedure for Tetrahedral Meshes \(p. 267\)](#) will be performed.
- If a volume mesh already exists, a **Question** dialog box will appear, asking if you want to clear the existing mesh. If you click **Yes**, the volume mesh will be cleared and the active steps in the automatic mesh generation process will be performed. If you click **No**, the operation will be canceled.

Important

You can use the command `/mesh/tet/preserve-cell-zone` to specify the cell zone(s) to be preserved during mesh generation and click the **Mesh** button to proceed with the automatic mesh generation.

14.1.3. Automatic Meshing of Multiple Cell Zones

The **Auto Mesh** tool allows you to mesh multiple cell zones automatically. This is useful when the mesh has multiple cell zones (e.g., the problem requires a fluid zone and one or more solid zones). You can refine the mesh using refinement parameters specific to individual zones, if required. The procedure for meshing multiple cell zones comprises the following steps:

1. Initialize the mesh.

2. Activate the appropriate zones using the **Manage Cell Zones** dialog box.
3. Set the zone-specific refinement parameters and refine the active zones.

If you want to mesh *all* the zones included in the mesh with the same refinement parameters, modify the **Non-Fluid Type** in the **Tet** dialog box.

Important

By default, the **Non-Fluid Type** is set to **dead** as the mesh is considered to comprise a single fluid region and one or more dead regions. The active zone is considered to be the fluid zone and only this fluid zone will be considered for refinement during the automatic meshing process.

When the **Non-Fluid Type** is set to a type other than **dead** (e.g., **solid**), all the zones will be active after the initialization is complete. Hence, all the zones will be considered for refinement. If required, you can change the zone type using the **Manage Cell Zones** dialog box.

Select the appropriate type from the **Non-Fluid Type** drop-down list in the **Tet Zones** group box in the **Tet** dialog box before initializing the mesh.

14.1.4. Automatic Meshing for Hybrid Meshes

The **Auto Mesh** tool can also create prisms and pyramids automatically, allowing automatic generation of hybrid meshes.

- If a mix of surface mesh types (quadrilateral and triangular) is present in the domain, pyramid cell zones will be created on the quadrilateral boundary faces before meshing the tetrahedral domain.
- If prism growth parameters have been attached to a boundary zone (using the **Prisms** dialog box), the prism layers will be automatically extruded before meshing the tetrahedral domain.

When pyramids or prisms are generated using the automatic mesh generation feature, the intermediate boundary zones will be merged automatically after the tetrahedral mesh generation is complete. This enhancement avoids the creation of additional boundary zones such as **prism-side** and **pyramid-cap** (see [Zones Created During Prism Generation \(p. 239\)](#) and [Zones Created During Pyramid Generation \(p. 222\)](#)).

- You can transition from quad to tri faces using the **Non Conformals** option. This option is useful when you want to avoid pyramids on quad faces when growing prisms from a boundary. For mixed surface mesh types (quadrilateral and triangular), you can select **Non Conformal** in the **Quad Tet Transition** list in the **Auto Mesh** dialog box. The surfaces containing quad faces will be copied and triangulated, keeping the original faces intact. The free nodes of the triangulated surface will then be merged with the nodes of the original surface mesh and both the surfaces will be converted to interface type.

14.1.5. Further Mesh Improvements

Examine the following after generating the mesh using the automatic mesh generation process:

- The cell distribution (see [Reporting Boundary Cell Limits \(p. 398\)](#))
- The cell limit reports (see [Reporting Cell Limits \(p. 398\)](#) and [Reporting Boundary Cell Limits \(p. 398\)](#))

- The mesh size (see [Reporting the Mesh Size \(p. 397\)](#))

It is typically possible to reduce the maximum skewness to the range 0.8–0.9. Skewness values higher than 0.9 are typically obtained due to constraints imposed by the surface meshes. If you still have highly skewed cells apply additional swapping and smoothing to improve the quality (see [Smoothing Nodes \(p. 347\)](#) and [Swapping \(p. 349\)](#)). To increase the density of the mesh locally, use refinement regions (see [Using Local Refinement Regions \(p. 277\)](#)).

Warning

Do not over-refine the mesh. The volume mesh produced should be of sufficient density to resolve the shape of the geometry. The mesh can be improved more effectively using the solution-adaptive mesh capability provided in the solution mode in ANSYS Fluent.

14.2. Manually Creating a Tetrahedral Mesh

In addition to the **Auto Mesh** tool, you can control the tetrahedral mesh generation process by modifying parameters at each step. The basic operations are described here, and the methods for modifying the associated parameters are described in [Initializing the Tetrahedral Mesh \(p. 274\)](#) and [Refining the Tetrahedral Mesh \(p. 276\)](#).

14.2.1. Manual Meshing Procedure for Tetrahedral Meshes

The basic components of the manual meshing process are examining and repairing the surface mesh, creating an initial mesh, refining the mesh, and improving the mesh.

The following steps describe the recommended meshing procedure.

Step 1: Examining the Surface Mesh

The first step in the mesh generation process is to examine the validity and quality of the surface mesh. The recommended approach includes the following steps:

- Checking for free and isolated nodes (see [Free and Isolated Nodes \(p. 119\)](#)).
- Checking and improving the surface mesh quality (see [Improving Boundary Surfaces \(p. 138\)](#)).
- Visually examining the surface mesh for free, multiply-connected, skewed, and/or close-proximity faces (see [Displaying the Grid \(p. 373\)](#)).
- Making local repairs if required (see [Modifying the Boundary Mesh \(p. 128\)](#)).

After obtaining a valid, (ideally) high-quality surface mesh, you can proceed to create the volume mesh.

Step 2: Creating the Initial Mesh

The first step in generating the volume mesh is creating the initial mesh. This process first creates a pre-meshed box encompassing the entire geometry, and then sequentially introduces each boundary node into the mesh. As the nodes of a boundary face are inserted into the mesh, any necessary mesh modifications needed to insert the face are performed, effectively inserting both boundary nodes and faces into the mesh simultaneously. For more information see [Initializing the Tetrahedral Mesh \(p. 274\)](#).

Activating Multiple Zones (for Multi-zone Meshes Only)

If the mesh has multiple regions (e.g., the problem requires a fluid zone and one or more solid zones), you can refine the mesh using refinement parameters specific to individual zones, if required. The procedure for meshing multiple cell zones comprises the following steps:

1. Initialize the mesh.
2. Activate the appropriate zones using the **Manage Cell Zones** dialog box.
3. Set the zone-specific refinement parameters and refine the active zones.

If you want to mesh *all* the zones included in the mesh with the same refinement parameters, modify the **Non-Fluid Type** in the **Tet** dialog box.

Select the appropriate type from the **Non-Fluid Type** drop-down list in the **Tet Zones** group box in the **Tet** dialog box.

Important

By default, the **Non-Fluid Type** is set to **dead** as the mesh is considered to comprise a single fluid region and one or more dead regions. The active zone is considered to be the fluid zone and only this fluid zone will be considered for refinement during the automatic meshing process.

When the **Non-Fluid Type** is set to a type other than **dead** (e.g., **solid**), all the zones will be active after the initialization is complete. Hence, all the zones will be considered for refinement. If required, you can change the zone type using the **Manage Cell Zones** dialog box.

Step 3: Refining the Mesh

You will refine the initial mesh by adding cells at the boundary and in the interior. The refinement methods available are the skewness-based method and the advancing front method. You can also define local refinement regions using the **Tet Refinement Region** dialog box. You can modify the parameters in the **Refinement** tab of the **Tet** dialog box and select additional refinement options for skewness-based refinement in the **Tet Refine Controls** dialog box, if required.

Step 4: Improving the Mesh

You can improve the skewness by smoothing, swapping, and refining the mesh further to improve the quality of the volume mesh. At any point in the mesh generation process, you can compute and/or plot the cell skewness distribution using the **Cell Distribution** dialog box. This will give you an idea of the improvement required. It is typically possible to reduce skewness to 0.8–0.85 (for simple geometries) or 0.9–0.95 (for more complex geometries).

Step 4a: Swapping and Smoothing without Refining

Swapping and smoothing improve the mesh by manipulating the nodes and faces without increasing the total number of cells. Refinement, on the other hand, improves the mesh by adding nodes, which typically increases the number of cells. To get the best possible mesh with the minimum number of cells, perform only smoothing and swapping before refining the mesh any further.

Smoothing repositions interior nodes to lower the maximum skewness of the mesh. For swapping, given $n+2$ nodes in dimension n , there are at most two triangulations of the nodes depending on the configuration of the nodes. In cases where two triangulations exist, the two alternatives are examined, and the one that has the lowest maximum skewness is selected. Smoothing and swapping are automatically performed to improve the mesh when the **Improve Mesh** option is enabled in the **Refinement** tab of the **Tet** dialog box.

Refer to [Smoothing Nodes \(p. 347\)](#) and [Swapping \(p. 349\)](#) for details on improving the mesh by swapping and smoothing.

Step 4b: Further Refinement

You can further refine the cells in the active zones by changing parameters such as **Max Cell Volume**, and **Max Cell Skew** and **Max Boundary Cell Skew** (available only for skewness-based refinement).

Important

Changing mesh size controls other than **Max Cell Volume** will have no effect on a mesh that has already been refined. Alternately, use local refinement regions. Refer to [Using Local Refinement Regions \(p. 277\)](#) for details.

Refining with skewness parameters reduced to values less than 0.5 will rarely improve the mesh and will be very time-consuming.

Important

Do not over-refine the mesh, the volume mesh produced should be of sufficient density to resolve the shape and any intuitive flow features (e.g., wall boundary layers). Additional resolution can be more effectively produced using the solution-adaptive mesh capability provided in the solution mode inANSYS Fluent.

At this step you will have an acceptable mesh for most geometries.

Step 4c: Boundary Slivers and Other Sources of Lingering High Skewness

When viewing the cell distribution plot, if you find that there are a few cells with very high skewness, check to see where they are located (i.e., on the boundary or in the interior). To do so, use the **Report Boundary Cell Limits** dialog box.

Refer to [Removing Slivers from a Tetrahedral Mesh \(p. 350\)](#) and [Moving Nodes \(p. 356\)](#) for details on removing slivers and other mesh improvement options.

You can initialize and refine the tetrahedral mesh in a single step by clicking the **Init & Refine** button in the **Tet** dialog box after setting the appropriate parameters in the **Initialization** and **Refinement** tabs. Alternatively, you can set the appropriate parameters in the **Initialization** tab and click the **Init** button to create the initial mesh. You can then refine the initial mesh by setting the appropriate parameters in the **Refinement** tab and clicking the **Refine** button.

If you need to refine a particular region, you can define the region to be refined using the **Tet Refinement Region** dialog box.

14.3. Initializing the Tetrahedral Mesh

The first step in the volume mesh generation process is initialization. The initial mesh consists of the nodes and triangles of the boundary surface mesh. For some geometries, it is not possible to create a tetrahedral mesh from the boundary nodes alone. In such cases, a small number of nodes are automatically added in the interior of the domain. Interior nodes may also have to be added to resolve numerical problems associated with the Delaunay criterion.

14.3.1. Using the Tet Dialog Box

14.3.2. Text Commands for Initializing the Mesh

14.3.1. Using the Tet Dialog Box

You can initialize the mesh using the options available in the **Initialization** tab of the **Tet** dialog box.

Mesh → Tet...

1. Select the appropriate actions to be performed during initialization from the **Options** group box.

The **Merge Free Nodes** and **Delete Unused Nodes** options are enabled by default. You can also include improving the surface mesh in the mesh initialization process by enabling the **Improve Surface Mesh** option.

2. Select the appropriate option from the **Non-Fluid Type** drop-down list. Enable **Delete Dead Zones**, if required.

When the initial mesh is generated, all the cells are grouped into contiguous zones separated by boundaries. The mesh is considered to comprise a single fluid zone and one or more dead regions. The zone just inside the outer boundary is set to be active and is labeled a fluid zone. All other non-fluid zones will be inactive. Only active zones will be considered for refinement during the mesh generation process.

You can refine different groups of zones using different refinement parameters for each group by toggling the zones between active and inactive. If however, you need to use the same refinement parameters for all the zones, you can change the specification of **Non-Fluid Type** to a type other than **dead** (e.g., **solid**). When the **Non-Fluid Type** is set to a type other than **dead**, all the zones will be activated after initialization. Hence, you can set the appropriate refinement parameters without setting all the zones to be active.

3. Specify additional initialization parameters, if required. These parameters are available in the **Tet Init Controls** dialog box. Click the **Controls...** button to open the **Tet Init Controls** dialog box.
4. Click **Init** to initialize the mesh.

A **Working** dialog box will appear, informing you that the initialization is in progress. Click the **Cancel** button in the **Working** dialog box to abort the mesh initialization process. Canceling the initialization will leave the mesh incomplete.

- If you try to initialize the mesh after canceling an initial attempt, you will be asked (in a **Question** dialog box) if it is OK to clear the incomplete mesh.

After you approve, the initialization process will begin again.

- If you try to initialize the mesh when duplicate nodes exist, the initialization will fail. You must clear the mesh and merge the duplicate nodes before attempting the initialization again.

- If you try to initialize the mesh when a volume mesh already exists, a **Question** dialog box (see [Question Dialog Box \(p. 33\)](#)) will appear, asking if you want to clear the existing mesh. If you click **Yes**, the volume mesh will be cleared and then the mesh will be initialized. If you click **No**, the operation will be canceled.
- If you need to preserve the existing mesh during the meshing process, use the command `/mesh/tet/preserve-cell-zone` to specify the cell zone(s) to be preserved and click the **Init** button to proceed with the initialization.
- If the initial mesh generation fails, check the validity of the surface mesh. In some rare cases, you may need to change the sliver size parameter (see [Tet Init Controls Dialog Box \(p. 559\)](#)).

Refer to [Common Tetrahedral Meshing Problems \(p. 282\)](#) for more information on meshing problems.

14.3.2. Text Commands for Initializing the Mesh

The text commands for initializing the mesh have the same functionality as the initialization parameters available in the **Initialization** tab of the **Tet** dialog box.

/mesh/tet/init

meshes boundary nodes, generating the initial Delaunay mesh.

/mesh/tet/controls/delete-dead-zones?

toggles the automatic deleting of dead zones during mesh initialization.

/mesh/tet/controls/delete-unused-nodes?

toggles the deleting of unused nodes during mesh initialization.

/mesh/tet/controls/improve-surface-mesh?

toggles the improving of the surface mesh during mesh initialization.

/mesh/tet/controls/merge-free-nodes?

toggles the merging of free nodes during mesh initialization.

/mesh/tet/controls/non-fluid-type

selects the non-fluid cell zone type.

/mesh/tet/controls/advanced/circumsphere-tolerance

defines the thickness of the circumsphere boundary.

In a Delaunay meshing algorithm, the most basic operation is checking to see if a node is inside or outside of the circumsphere of a cell. Due to numerical errors, it is not possible to get a definitive answer. The tolerance is used to define a thickness for the boundary of the circumsphere. If a node is on this boundary, it is neither inside nor outside of the circumsphere and hence will be treated specially.

/mesh/tet/controls/advanced/defaults?

toggles the use of the default values for the initialization parameters.

/mesh/tet/controls/advanced/node-tolerance

defines the smallest distance between two distinct nodes. This command is available only when you disable the use of the default values for the initialization parameters.

/mesh/tet/controls/advanced/sliver-size

is the smallest cell whose size can be determined accurately. This command is available only when you disable the use of the default values for the initialization parameters.

14.4. Refining the Tetrahedral Mesh

After initializing the mesh, you can improve the quality and density of the mesh using global and local refinement. Global refinement allows you to refine all cells in the active zones, while local refinement allows you to refine cells within a specified region.

In most applications, you will use only global refinement to create an acceptable discretization of the volume. Local refinement is used to modify the grading away from boundaries or increase the resolution of an interior region of the mesh. You need to define and activate the regions to be refined before proceeding.

The refinement process comprises a series of sweeps through a sequence of sorting, refining, reverse sorting, and swapping and smoothing of the cells. You can specify the number of refinement levels to be performed for skewness-based refinement. Each refinement level consists of two iterations through the refinement procedure. For each subsequent level of refinement, the cell skewness thresholds are lowered. Additional refinement levels will increase the mesh resolution (and the number of cells) and decrease the average cell skewness. The preset refinement control parameters are available in the **Tet Refine Controls** dialog box. Alternatively, you can select the advancing front refinement method.

Specify whether the mesh has to be improved or if the slivers have to be removed during refinement.

- The **Improve Mesh** option performs additional smoothing and swapping with lowered skewness thresholds, attempting to improve the average skewness of the mesh.
- The **Remove Slivers** option attempts to lower the maximum skewness by improving highly skewed (sliver) cells. There are two approaches for sliver removal: fast and aggressive. Both methods use the same controls and give similar results for good quality surface meshes. In case of poor surface meshes (meshes with large size difference between neighboring elements, narrow gaps with widely varying mesh sizes on either side of the gap, etc.), the aggressive method will typically succeed in improving the mesh to a greater extent, but it may be slower than the default fast method.

The aggressive method corresponds to the improve option in the **Tet Improve** dialog box.

The boundary mesh is fixed during the mesh improvement and sliver removal operations.

Refinement Controls

The refinement controls for the skewness method are available in the **Tet Refine Controls** dialog box. Click the **Controls...** button in the **Refinement** tab of the **Tet** dialog box to open the **Tet Refine Controls** dialog box.

The refinement controls for the advancing front method are available in the **/mesh/tet/controls/adv-front-method** menu. The **/mesh/tet/controls/adv-front-method/skew-improve/target?** command is important as it allows you to enable targeted skewness-based refinement for the advancing front method. The refinement process will attempt to reach the targeted skewness (specified by the command **/mesh/tet/controls/adv-front-method/skew-improve/target-skew**). Though the default values work well in most situations, it may be advantageous to increase the target-skew value in some cases (e.g., cases where a combination of a poor surface

mesh and narrow gaps will result in poor cells in the mesh) to increase the meshing speed. Refer to [Text Commands for Setting Refinement Controls \(p. 279\)](#) for the list of commands available.

14.4.1. Using Local Refinement Regions

14.4.2. Using the Tet Dialog Box

14.4.3. Text Commands for Setting Refinement Controls

14.4.1. Using Local Refinement Regions

A refinement region limits refinement to specific cells inside the domain. When you use the **Refine** option in the **Tet Refinement Region** dialog box, the cells within the activated region(s) are refined. The primary use of refinement regions is to reduce the cell size in the region to less than it would be normally. Currently, the only possible region shape is a box, which can be oriented as required.

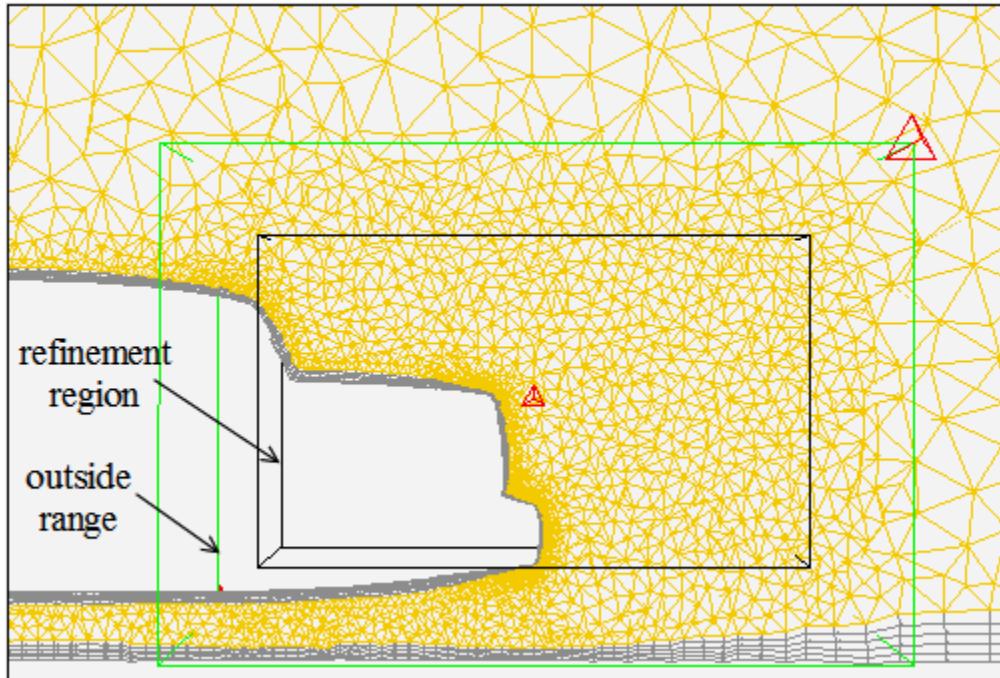
The region is defined by an *x*, *y*, and *z* range and its orientation about the *x*, *y*, and *z* axes. Any cell within the region or which intersects the region will be subjected to the refinement criteria. You can control the concentration of interior nodes by specifying the maximum cell volume in the refinement region. It is possible to save different values for the **Maximum Cell Volume** when you save each region. You can also specify an outer region with the required volume growth to have a smooth transition between the original and the refined cells.

Note

In addition to the use of refinement regions for refinement during the mesh generation process, they can also be used as a postprocessing tool to refine a region of an existing mesh.

You can create multiple regions that overlap each other and the geometry. To create additional regions, you can copy an existing region and then modify the parameters as required. The default region includes the entire geometry.

[Figure 14.1: Local Refinement Region for the Tetrahedral Mesh \(p. 278\)](#) shows the refinement region and outer region. The maximum cell size for the refinement and outer regions is displayed based on the maximum cell volume and outside volume growth specified. For information on refining triangular boundary faces in anticipation of local refinement, refer to [Refining the Boundary Mesh \(p. 139\)](#).

Figure 14.1: Local Refinement Region for the Tetrahedral Mesh

14.4.2. Using the Tet Dialog Box

You can refine the mesh using the options available in the **Refinement** tab of the **Tet** dialog box.

Mesh → Tet...

1. Select the appropriate refinement method from the **Refine Method** drop-down list.
2. Enable the appropriate options in the **Options** group box.
3. Specify an appropriate value for **Max Cell Volume**.
4. Select the appropriate option from the **Cell Sizing** drop-down list.

Ensure that the cells in the interior are not larger than the size required by selecting the appropriate option in the **Cell Sizing** list. The following options are available:

none

specifies that the cell size is determined based on the **Max Cell Volume** and skewness. This option is available only for the skewness-based refinement method.

linear

specifies that the cell size in the interior is linearly interpolated from the size of the boundary triangles.

geometric

specifies that the cell size in the interior of the domain is obtained by a geometric growth from the closest boundary according to the growth rate specified.

size-function

specifies that the cell size is determined based on the current size functions or size-field.

5. Modify the refinement control parameters, if required.
 - For the skewness method, click the **Controls...** button to open the **Tet Refine Controls** dialog box. Select the appropriate option from the **Preset Parameters** drop-down list. Verify the parameter values in the **Tet Refine Controls** dialog box and modify the values as appropriate.
 - For the advancing front method, use the commands available in the /mesh/tet/controls/adv-front-method menu. You can also specify the targeted skewness for the refinement process (see [Refinement Controls \(p. 276\)](#) for details).
6. To refine specific regions of the mesh, click the **Local Regions...** button to open the **Tet Refinement Region** dialog box. Define the refinement region(s) and activate them. The number of activated regions will be reported in the **Message** field in the **Refinement** tab of the **Tet** dialog box.
7. Click **Refine** to refine the mesh.

14.4.3. Text Commands for Setting Refinement Controls

The text interface commands for setting refinement parameters are as follows:

/mesh/tet/refine

refines the initialized mesh.

/mesh/tet/controls/adv-front-method/first-improve-params

defines the refining front improvement parameters for the advancing front method.

/mesh/tet/controls/adv-front-method/second-improve-params

defines the cell zone improvement parameters for the advancing front method.

/mesh/tet/controls/adv-front-method/refine-parameters

defines the refinement parameters for the advancing front method.

/mesh/tet/controls/adv-front-method/skew-improve/attempts

specifies the number of overall improvement attempts for the advancing front method.

/mesh/tet/controls/adv-front-method/skew-improve/boundary-sliver-skew

specifies the boundary sliver skewness for the advancing front method. This parameter is used for removing sliver cells along the boundary.

/mesh/tet/controls/adv-front-method/skew-improve/iterations

specifies the number of improvement iterations in each attempt for the advancing front method.

/mesh/tet/controls/adv-front-method/skew-improve/sliver-skew

specifies the sliver skewness for the advancing front method. This parameter is used for removing sliver cells in the interior.

/mesh/tet/controls/adv-front-method/skew-improve/target-low-skew

specifies the targeted skewness threshold above which cells will be improved. The improve operation will attempt to improve cells with skewness above the target-low-skew value specified, but there will be no attempt to reduce the skewness below the specified value. A limited set of improve operations will be used as compared to the operations required for the target-skew value-based improvement. The value specified could be approximately 0.1 lower than the target-skew value.

/mesh/tet/controls/adv-front-method/skew-improve/target-skew

specifies the targeted skewness during improvement for the advancing front method. The improve operation will attempt to reduce the maximum skewness below the target-skew value specified.

/mesh/tet/controls/adv-front-method/skew-improve/target?

allows you to enable targeted skewness-based refinement for the advancing front method. This option allows you to improve the mesh until the targeted skewness value is achieved.

/mesh/tet/controls/advanced/max-cells

sets the maximum number of cells in the mesh.

/mesh/tet/controls/advanced/max-nodes

sets the maximum number of nodes in the mesh.

Warning

Be careful when you set upper limits on the number of cells or nodes. Nodes or cells will not be added when the prescribed limit is reached, and this may leave the mesh unevenly refined and less than adequate.

/mesh/tet/controls/cell-sizing

controls the cell sizing for refinement. You can select size-function, geometric, linear, or none, as appropriate.

/mesh/tet/controls/compute-max-cell-volume

computes the maximum cell volume for the current mesh.

/mesh/tet/controls/max-cell-volume

specifies the maximum allowable cell volume.

/mesh/tet/controls/refine-method

allows you to select the refinement method. You can select either skewness-based refinement or the advancing front method.

/mesh/tet/controls/skewness-method/levels

specifies the number of refinement levels for skewness-based refinement.

/mesh/tet/controls/skewness-method/max-skew-improve?

toggles the skewness-based improvement during refinement.

/mesh/tet/controls/skewness-method/must-improve-skewness?

toggles the modification of the default auto refinement parameters in order to improve skewness after refinement.

/mesh/tet/controls/skewness-method/refine-boundary-cells?

toggles the automatic refinement of boundary cells during refinement.

/mesh/tet/controls/skewness-method/refine-cells?

toggles the automatic refinement of cells during refinement.

/mesh/tet/controls/skewness-method/smooth-mesh?

toggles the automatic smoothing of the mesh during refinement.

/mesh/tet/controls/skewness-method/sort-boundary-faces?
toggles the automatic sorting of boundary faces by size during refinement.

/mesh/tet/controls/skewness-method/sort-cells?
toggles the automatic reverse sorting of cells by skewness during refinement.

/mesh/tet/controls/skewness-method/swap-faces?
toggles the automatic swapping of faces during refinement.

/mesh/tet/controls/skewness-method/type
allows you to select the appropriate pre-defined skewness refinement parameters. You can select default, fast-transition, or incremental-improve as required.

/mesh/tet/local-regions/activate
activates the specified region(s) for refinement.

/mesh/tet/local-regions/deactivate
deactivates the specified region(s) for refinement.

/mesh/tet/local-regions/define
defines the refinement region according to the specified parameters.

/mesh/tet/local-regions/delete
deletes the specified refinement region.

/mesh/tet/local-regions/ideal-vol
reports the volume of an ideal tetrahedron for the edge length specified.

/mesh/tet/local-regions/init
defines the default refinement region encompassing the entire geometry.

/mesh/tet/local-regions/list-all-regions
lists all refinement region parameters and the activated regions in the console.

/mesh/tet/local-regions/refine
refines live cells inside the specified region based on refinement parameters.

14.5. Additional Text Commands for Tetrahedral Mesh Generation

/mesh/tet/init-refine
generates the tetrahedral mesh.

/mesh/tet/controls/advanced/freeze-boundary-cells
freezes the boundary cells during mesh improvement and sliver removal operations.

/mesh/tet/controls/advanced/keep-virtual-entities?
toggles the automatic deleting of virtual entities after mesh initialization.

/mesh/tet/controls/improve-mesh/improve?
toggles the automatic improvement of the mesh during the meshing process.

/mesh/tet/controls/improve-mesh/laplace-smooth
sets the parameters for Laplacian smoothing in order to improve mesh quality.

/mesh/tet/controls/improve-mesh/skewness-smooth

sets the parameters for skewness-based smoothing in order to improve mesh quality.

/mesh/tet/controls/improve-mesh/swap

sets the parameters for swapping in order to improve mesh quality.

/mesh/tet/controls/remove-slivers/angle

specifies the maximum dihedral angle for considering the cell to be a sliver

/mesh/tet/controls/remove-slivers/attempts

specifies the number of attempts overall to remove slivers.

/mesh/tet/controls/remove-slivers/iterations

specifies the number of iterations to be performed for the specific sliver removal operation.

/mesh/tet/controls/remove-slivers/low-skew

specifies the targeted skewness threshold above which cells will be improved. The improve operation will attempt to improve cells with skewness above the low-skew value specified, but there will be no attempt to reduce the skewness below the specified value. A limited set of improve operations will be used as compared to the operations required for the skew value-based improvement.

/mesh/tet/controls/remove-slivers/method

allows you to select the method for sliver removal. The default method used is the fast method. The fast and the aggressive methods use the same controls and give similar results for good quality surface meshes. In case of poor surface meshes, the aggressive method will typically succeed in improving the mesh to a greater extent, but it may be slower than the fast method.

The aggressive method corresponds to the improve option in the **Tet Improve** dialog box.

/mesh/tet/controls/remove-slivers/remove?

toggles the automatic removal of slivers.

/mesh/tet/controls/remove-slivers/skew

specifies the skewness threshold for sliver removal. The improve operation will attempt to reduce the maximum skewness below the skew value specified.

/mesh/tet/delete-virtual-cells

deletes virtual cells created due to the use of the keep-virtual-entities? option.

/mesh/tet/preserve-cell-zone

allows you to specify the cell zones to be preserved during the meshing process.

14.6. Common Tetrahedral Meshing Problems

Most problems with the mesh generation process become manifest in the failure to generate an initial mesh. There are two sources of such problems:

- An invalid surface mesh
- Incorrect sliver size specified

Some of the common problems and suggestions for checking and fixing the mesh are described in this section.

To look at the nodes and faces that have not been meshed, enable **Unmeshed** in the **Display Grid** dialog box.

Duplicate Nodes or Faces

Some codes create multiple copies of the same node where two boundary curves or surfaces meet, resulting in a connectivity problem. This problem can be detected by reporting the number of free nodes or by drawing the free edges. If the geometry does not contain infinitely thin walls then there should not be any free nodes or edges. You can remove duplicate nodes using the `/boundary/delete-duplicate-nodes` command described in [Additional Boundary Mesh Text Commands \(p. 172\)](#).

The presence of duplicate faces can be detected in one of two ways:

- If the nodes of the duplicate faces are not shared by the non-duplicate faces, the duplicate faces may have free edges that can be displayed.
- If the nodes are not distinct or have been merged, there will be multiply-connected edges.

Note

Duplicate faces can be handled, it is not necessary to remove them.

Extra Nodes or Faces

Nodes that are not used by any face are called unused nodes. The unused nodes can easily be found and deleted using the operations described in [Merge Boundary Nodes Dialog Box \(p. 482\)](#) if they are not required. However, there are cases where these additional nodes may be useful (see [Inserting Isolated Nodes into a Tet Mesh \(p. 217\)](#)). Extra faces can be identified and removed using the method described for duplicate faces.

Intersecting Faces

If there are intersecting faces in the mesh, you will not be able to generate a mesh until the problem faces are removed. Most intersecting faces can be located using the `/boundary/mark-face-intersection` command. If intersecting boundary faces are encountered during initialization, a message will be displayed.

After the meshing fails, the unmeshed faces can be drawn to see the problem area. Usually, since the intersecting face will have been meshed, it will not be drawn and some additional sleuthing will be required. Draw all the faces near the unmeshed faces by setting display bounds (see [Display Grid Dialog Box \(p. 596\)](#)). Click the **Mark** button in the **Intersect Boundary Zones** dialog box to highlight the face intersection. Alternatively, you can use the command `/boundary/mark-face-intersection` to highlight the face intersection.

Poor Boundary Node Distribution

Some meshing problems can be caused by a poor quality surface mesh. The surface mesh may have highly skewed faces, or two or more boundaries in close proximity may have very different face sizes. In the latter case, the boundaries can be connected (e.g., in a corner) or completely separate. Either way, the boundary mesh cannot be created without considering the impact of other nearby boundaries.

If the gap between two boundaries is **D**, the largest edge on a face should not exceed **D**. The mesh quality is extremely important when a coarse mesh is required in a small gap.

Non-Closed Boundaries

You can generate an initial mesh even if there is a hole in a boundary either due to missing faces or a gap between two zones. In this case, instead of having two separate zones on opposite sides of the boundary, you will have just one combined zone. This situation is evident because you will have very few zones.

- If the hole is in the outer boundary, the cells outside the boundary will be combined with the cells inside, resulting in an error. The edges around the hole will be marked as free edges and therefore can be displayed.
- If the problem is caused by a small gap between boundaries and the duplicate nodes have already been merged, increase the tolerance and perform additional merging.

The following command can be used to detect holes in the geometry by tracing the path between the two specified cells:

```
/mesh/tet/trace-path-between-cells
```

A path between the selected cells will be highlighted.

Interior Node Near Boundary

During refinement, a node may be placed too close to a boundary face, resulting in a highly skewed cell. You can detect this situation by displaying the highly skewed cells. You can then smooth the mesh to eliminate the problem.

If the interior node is constrained by the boundary faces, increase the **Min Boundary Closeness** (in the **Tet Refine Controls** dialog box) and regenerate the mesh. Refer to [Tet Improve Dialog Box \(p. 571\)](#) and [Auto Node Move Dialog Box \(p. 577\)](#) for details.

Chapter 15: Generating the Hexcore Mesh

Hexcore meshing is a hybrid meshing scheme that generates Cartesian cells inside the core of the domain and tetrahedral cells close to the boundaries. Hanging-node (or H-) refinements on the Cartesian cells allow efficient cell size transition from boundary to interior of the domain. This results in fewer cells with full automation and can handle complex geometries, internal walls and gaps.

The hexcore meshing scheme is applicable to all volumes but is useful mainly for volumes with large internal regions and few internal boundaries such as intrusions or holes. The **Hexcore** submenu contains options to control the hexcore mesh generation. The starting point is a valid surface mesh.

- 15.1. Hexcore Meshing Procedure
- 15.2. Using the Hexcore Dialog Box
- 15.3. Controlling Hexcore Parameters
- 15.4. Text Commands for Hexcore Meshing

15.1. Hexcore Meshing Procedure

The hexcore mesh generation process comprises the following steps:

1. Generating initial Cartesian cells inside a bounding box (or the region specified) around the volume to be meshed.
2. Marking the Cartesian cells that intersect the boundary mesh and those cells having sizes larger than the average size of the faces they intersect.
3. Marking additional buffer-layer cells adjacent to the cells marked in Step 2 (if specified).
4. Marking additional cells to enforce one-level refinement difference between adjacent cells.
5. Subdividing all marked (in steps 2–4) cells and repeating steps 2–5 until the local face size criteria is met.
6. Deleting cells that intersect or are within some distance to the closest face on the surface mesh.
7. Triangulating the external surface of the hexcore by converting quads into tri interface faces. The quad faces become parents (with type parent-face) of the interface faces.

Note

Pyramids are not used for the transition from the hexcore to the tetrahedral cell regions.

-
8. Smoothing the interface faces (if interface smoothing is enabled).

9. Initializing and refining the tetrahedral cells between the interface faces and the boundary mesh.
-

Note

The maximum skewness reported at the end of the refinement process is not necessarily the final maximum skewness. Sliver cells on the interfaces can be removed at a later stage.

10. Removing sliver cells on the interface faces.

11. Merging Cartesian cells with tetrahedral (and wedge, if present) cells to form contiguous cell zones.
-

Important

If prism parameters have been attached to boundary zones, the prism layers will first be generated. The hexcore meshing procedure will then be applied to the resulting prism caps, along with the other boundary zones that were not involved in the prism mesh generation.

You can control the general shape, size, and density of the core, as well as the size of elements created at the outer boundary of the core using the parameters in the **Hexcore** dialog box and the commands in the /mesh/hexcore/ menu.

15.2. Using the Hexcore Dialog Box

To create a hexcore mesh using the **Hexcore** dialog box, do the following:

1. Read in a boundary mesh and check and improve its quality, if necessary.
2. (optional) Split any quad faces in the boundary mesh into triangular faces using the /boundary/remesh/triangulate command.

By default, quad faces are not allowed while initializing the hexcore mesh.

Alternatively, use the command /mesh/non-conformals/controls/enable? to enable the creation of a non-conformal interface.

If this option is enabled, all the surfaces having quad elements will be copied and remeshed with triangular faces. The free nodes of the triangular mesh will be merged with the original surface mesh. The method to be used for retriangulation can be specified using the command /mesh/non-conformals/controls/retri-method.

The quad-split method is the default method used for retriangulation. You can select prism, quad-split, or remesh as appropriate.

3. Specify the hexcore parameters in the **Hexcore** dialog box.

The parameters (e.g., maximum cell length) required during the hexcore meshing are automatically calculated. These parameters can be changed using the **Hexcore** dialog box or using the commands in the /mesh/hexcore text menu.

- Click **Create** in the **Hexcore** dialog box to generate the hexcore mesh.

15.3. Controlling Hexcore Parameters

The parameters that control the hexcore mesh generation can be changed using the **Hexcore** dialog box. The following options are available to control the hexcore mesh generation in the domain:

- 15.3.1. Defining Hexcore Extents
- 15.3.2. Hexcore to Selected Boundaries
- 15.3.3. Only Hexcore
- 15.3.4. Maximum Cell Length
- 15.3.5. Buffer Layers
- 15.3.6. Peel Layers
- 15.3.7. Local Refinement Regions

15.3.1. Defining Hexcore Extents

You can optionally create the hexcore mesh until the boundary of the domain instead of creating a tetrahedral mesh at the boundary. This feature is useful for the hexcore mesh generation for external flow domains, e.g., external aerodynamics cases where the boundary conformity is not needed for far-field boundaries. This also helps increase the count of hexahedral cells in the mesh.

Hexcore extents can be defined by specifying a box in which the hexcore mesh is to be generated and/or a set of axis-aligned planar boundaries to which the hexahedral core is to be extended. The extents can be defined as follows:

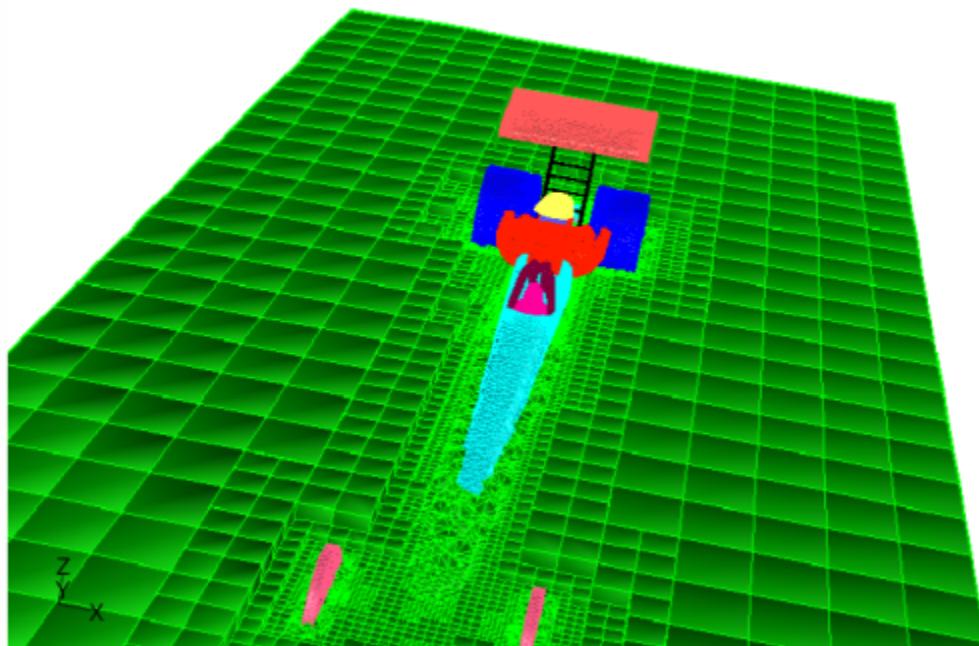
- Enable **Define Hexcore Extents** in the **Hexcore** dialog box.
- Click the **Specify...** button to open the **Outer Domain Parameters** dialog box.
- Enable **Coordinate Extents** in the **Outer Domain Parameters** dialog box. The minimum and maximum domain extents will be automatically updated with values slightly greater than the bounding box of the surface mesh in the domain. The coordinate extents of the hexcore outer box can also be specified as required.

Important

The hexcore mesh will extend to the defined domain extents. A warning will be displayed if the gap between the user-defined domain extents and geometric boundaries is less than 20 % of size of the bounding box which contains the given geometry.

- Enable **Draw Outer Box** and click **Draw** to verify that the domain extents are correctly defined.
- Click **Apply** in the **Outer Domain Parameters** dialog box.
- Create the hexcore mesh.

In [Figure 15.1: Hexcore to the Far-Field Boundary \(p. 288\)](#), the hexcore mesh has been extended until the domain boundary and does not have a tetrahedral mesh at the far-field boundary.

Figure 15.1: Hexcore to the Far-Field Boundary

15.3.2. Hexcore to Selected Boundaries

The **Boundary Extents** option in the **Outer Domain Parameters** dialog box can be used to imprint quad faces on selected axis-aligned planar boundaries. The selected boundary will be replaced with a mix of quad and tri face zones. You can generate the hexcore mesh to selected boundaries as follows:

1. Enable **Define Hexcore Extents** in the **Hexcore** dialog box.
2. Click the **Specify...** button to open the **Outer Domain Parameters** dialog box.
3. Enable **Boundary Extents** in the **Outer Domain Parameters** dialog box and select the appropriate planar, axis-aligned boundaries to which the hexcore mesh is required.

Note

The planar boundaries must be split into separated boundary zones.

Click **Apply** in the **Outer Domain Parameters** dialog box.

The hexcore box coordinates will automatically snap to the selected boundaries.

Enable **Draw Outer Box**. Draw the outer box along with the selected boundaries to verify that the box snaps to the selected boundaries.

If the box does not snap to a selected boundary, it indicates that the hexcore mesh cannot be grown to the boundary. If the selected boundary zone is a plane which is misaligned with the axis, you can align it using the **Auto Align** option as follows:

- a. Select the zone(s) to be aligned in the **Boundary Zones** selection list.

The **Auto Align** group box will now be activated.

- b. Enable **Auto Align**.
- c. The recommended auto align tolerance value can be computed for the selected set of boundary zones. Click **Compute** to determine the recommended tolerance or enter an appropriate tolerance value.
- d. Click **Apply** in the **Auto Align** group box to align the selected zones which are within the tolerance specified.

Warning

The **Auto Align** option may deform the geometry permanently. Take care while selecting the boundary zones.

- e. Draw the outer box along with the selected boundaries to verify that the box has snapped to the selected boundaries.
4. Enable **Delete Old Face Zones**, if required and click **Apply** in the **Outer Domain Extents** dialog box.
5. Create the hexcore mesh.

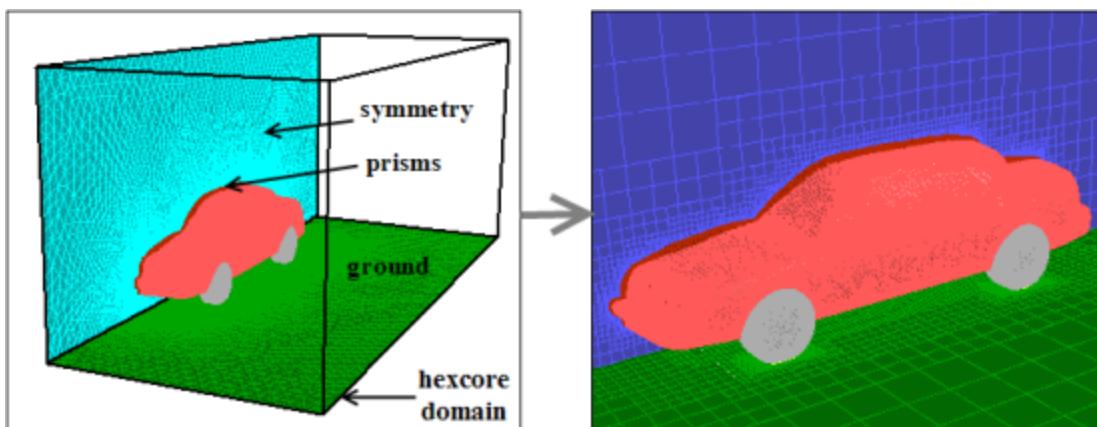
The newly created quad face zones will be named with the original zone names.

The hexcore mesh will extend to the selected boundaries, aligned with the X, Y, and Z axes. An example is shown in Figure 15.2: Hexcore to Boundaries (p. 289).

Warning

An error will be reported if you attempt to use this option when the geometry is completely within the domain.

Figure 15.2: Hexcore to Boundaries



15.3.3. Only Hexcore

The **Only Hexcore** option allows you to prevent the automatic creation of the tetrahedral mesh after hexcore generation (see [Figure 15.3: Only Hexcore \(p. 290\)](#)). However, the tetrahedral mesh domain is

created and activated during the hexcore meshing procedure. You can manually create the tetrahedral mesh in a separate step.

Figure 15.3: Only Hexcore



Hex cell islands may be created when the **Only Hexcore** option is used. You can delete small islands which may be created by setting the minimum allowable size for the hex cell islands using the command /mesh/hexcore/controls/post-relative-island-count. This command is available only when the **Only Hexcore** option has been enabled in the **Hexcore** dialog box. You can also use the command /mesh/hexcore/controls/only-hexcore? to enable only hexcore meshing.

The default value for post-relative-island-count is 10. All hex cell islands whose size is less than the specified percentage (in the default case, 10%) of the largest hex cell zone will be automatically deleted.

No islands will be removed if you set the value to 0.

Warning

You may need to verify that hex cells are created only in the domain of interest since the mesher cannot identify whether the cells are within the domain without tetrahedral meshing. Also, all the non-fluid type zones are created as separate fluid zones. You need to retain the fluid zone(s) of interest and delete the others.

15.3.4. Maximum Cell Length

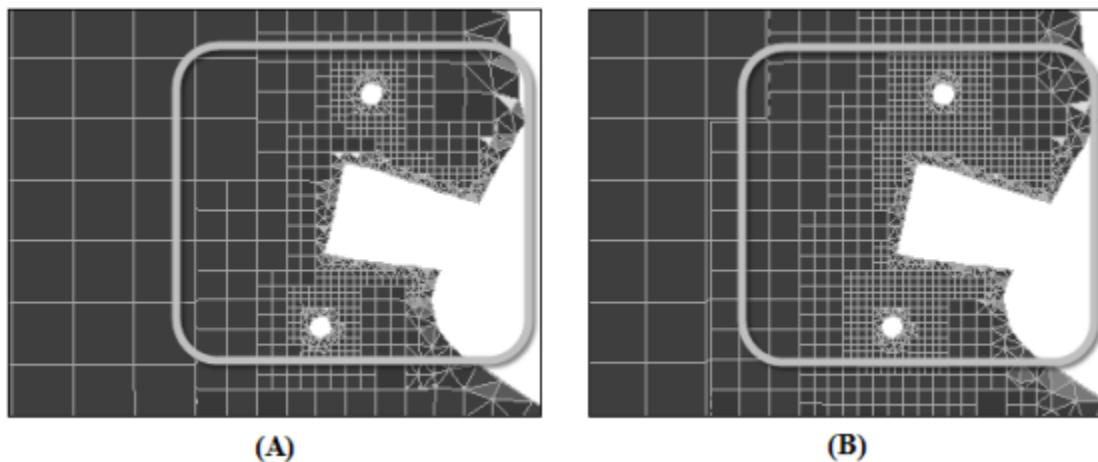
An optimal maximum cell length can be automatically calculated for a particular mesh. Click the **Compute** button next to the **Max Cell Length** field to obtain the optimal maximum cell length for the mesh. Alternatively, specify the maximum cell length as required.

The initial Cartesian cells will be generated based on the maximum cell length.

15.3.5. Buffer Layers

The Cartesian cells are marked (and subsequently subdivided) to satisfy the size requirement on the boundary mesh. When there is large disparity in size distribution between the boundary mesh and the initial Cartesian cells, there will be a rapid transition from fine to coarser cells. To avoid this, additional layers of cells are marked adjacent to those marked by the size requirement (see [Figure 15.4: Hexcore Mesh Using \(A\) Buffer Layers = 1 \(B\) Buffer Layers = 2 \(p. 291\)](#)). You can control the number of additional layers by setting the **Buffer Layers** in the **Hexcore** dialog box. The default number of buffer layers is set to 1. Setting the number of buffer layers to zero may result in a poor quality mesh.

Figure 15.4: Hexcore Mesh Using (A) Buffer Layers = 1 (B) Buffer Layers = 2

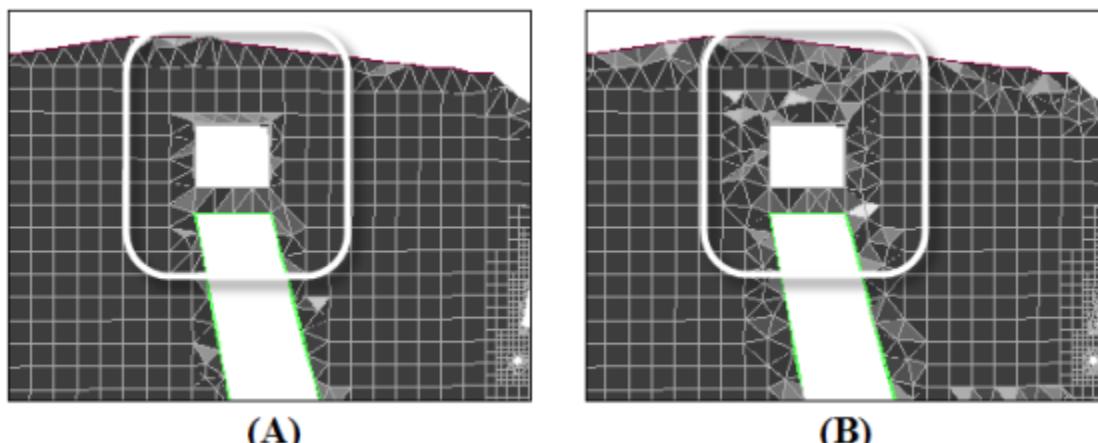


15.3.6. Peel Layers

The peel layers control the gap between the hexahedra core and the geometry. After the Cartesian cells are subdivided to meet the size requirement, the cells intersected by boundary mesh and those within some distance to the closest face on the boundary mesh are deleted. The default value for **Peel Layers** is 1, hence this distance is assumed to be the height of an ideal tetrahedral cell on the boundary face. If **Peel Layers** is set to 0, the gap size can be smaller than the ideal height. The resulting hexcore mesh will contain the maximum number of Cartesian cells possible for the chosen parameters.

[Figure 15.5: Hexcore Mesh Using \(A\) Peel Layers = 0 \(B\) Peel Layers = 2 \(p. 291\)](#) shows the hexcore mesh generated for different values of **Peel Layers**.

Figure 15.5: Hexcore Mesh Using (A) Peel Layers = 0 (B) Peel Layers = 2



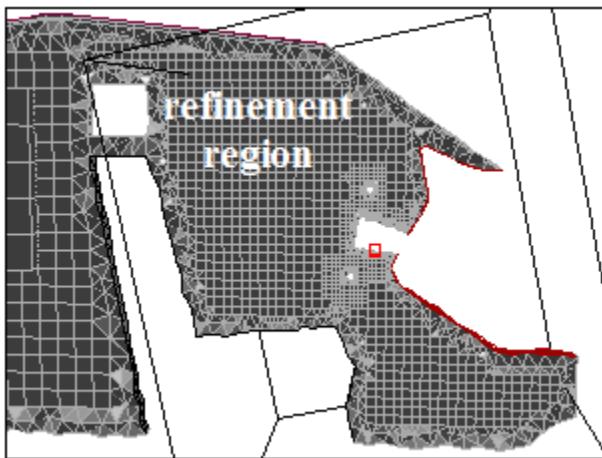
15.3.7. Local Refinement Regions

You can use local refinement regions to refine specific cells within the domain. During the hexcore meshing procedure, the cells within the activated local refinement region(s) will be refined. Currently, the only possible region shape is a box, either aligned with the coordinate axes, or oriented as required.

The region extents are defined by the center and the length of the region. The cell size within the region can be manipulated by setting the level of refinement relative to the maximum cell length in the hexcore domain. You can also orient the region as required. You can create multiple regions that overlap each other and the geometry. To create additional regions, you can copy an existing region and then modify the parameters as required. The default region includes the entire geometry.

You can use the **Draw** button in the **Hexcore Refinement Region** dialog box to display the defined region. [Figure 15.6: Local Refinement Region for the Hexcore Mesh \(p. 292\)](#) shows a refinement region defined. A sample Cartesian cell with the specified **Max Length** is also displayed at the center of the refinement box.

Figure 15.6: Local Refinement Region for the Hexcore Mesh



15.4. Text Commands for Hexcore Meshing

Text commands with the same functionality as the controls in the **Hexcore** dialog box along with additional text commands related to hexcore meshing are as follows:

/mesh/hexcore/controls/buffer-layers

sets the number of additional cells to mark for subdivision.

/mesh/hexcore/controls/delete-dead-zones?

toggles the deleting of dead zones during hexcore meshing.

/mesh/hexcore/controls/keep-hex-tet-separate?

toggles the merging of Cartesian cells with the tetrahedral (and wedge) cells at the end of the hexcore meshing process.

/mesh/hexcore/controls/maximum-cell-length

sets the maximum cell length for the hex cells in the domain.

/mesh/hexcore/controls/maximum-initial-cells

specifies the maximum number of cells in the initial Cartesian mesh.

/mesh/hexcore/controls/maximum-subdivisions

specifies the number of changes in size (hanging node subdivisions) allowed in the hex mesh region. The default value is 50.

/mesh/hexcore/controls/mesh-rel-island-count

specifies the threshold cell count (relative to the total cell count) for islands to be deleted when separating cells by region during hexcore meshing. The default value is 1% of the total cell count.

/mesh/hexcore/controls/non-fluid-type

allows you to set the cell zone type for the non-fluid cell zones.

/mesh/hexcore/controls/only-hexcore?

allows you to create only the hexcore mesh and activates the tetrahedral mesh domain (without tetrahedral mesh generation). This option is disabled by default.

/mesh/hexcore/controls/peel-layers

specifies the distance for the hexcore interface to peel back from the boundary. The default value is 0. The higher the value of peel layer, the greater the distance between the hexcore interface and the boundary.

/mesh/hexcore/controls/define-hexcore-extents?

allows you to extend the hexcore mesh to specified domain extents and/or selected planar boundaries. When enabled, the outer-domain-params sub-menu will be available.

/mesh/hexcore/controls/outer-domain-params/specify-coordinates?

allows you to specify the extents of the hexcore outer box using the coordinates command.

/mesh/hexcore/controls/outer-domain-params/coordinates

specifies the extents (min and max coordinates) of the hexcore outer box. This command is available when the specify-coordinates? option is enabled.

/mesh/hexcore/controls/outer-domain-params/specify-boundaries?

allows you to specify selected boundaries to which the hexcore mesh is to be generated using the boundaries command.

/mesh/hexcore/controls/outer-domain-params/boundaries

specifies the boundaries to which hexcore mesh is to be generated when the specify-boundaries? option is enabled. After specifying the boundaries, the auto-align?, delete-old-face-zones?, and list options will also be available.

/mesh/hexcore/controls/outer-domain-params/auto-align?

allows you to axis-align non-aligned planar boundaries to which hexcore mesh is to be generated. This option is available only when the specify-boundaries? option is enabled and the boundaries are specified.

/mesh/hexcore/controls/outer-domain-params/auto-align-tolerance

specifies the tolerance for aligning boundary zones when auto-align? is enabled.

/mesh/hexcore/controls/outer-domain-params/auto-align-boundaries

aligns the boundary zones specified (using the boundaries command) with the tolerance specified (using the auto-align-tolerance command) when auto-align? is enabled.

/mesh/hexcore/controls/outer-domain-params/delete-old-face-zones?

allows you to delete the original tri face zones which have been replaced during the hexcore meshing process. This option is available only when the `specify-boundaries?` option is enabled and the boundaries are specified.

/mesh/hexcore/controls/outer-domain-params/list

lists the boundaries to which the hexcore mesh is to be generated. This option is available only when the `specify-boundaries?` option is enabled and the boundaries are specified.

/mesh/hexcore/controls/post-relative-island-count

specifies the threshold cell count (relative to the cell count of the largest zone containing hex cells) for islands to be deleted after generating the hexcore mesh using the **Only Hexcore** option. The default value is 10.

/mesh/hexcore/controls/smooth-interface?

enables smoothing of the hexcore interface.

/mesh/hexcore/controls/smooth-iterations

sets the number of smoothing iterations on the hexcore.

/mesh/hexcore/controls/smooth-relaxation

sets the smoothing under relaxation on the hexcore interface.

/mesh/hexcore/create

allows you to create the hexcore mesh according to the specified parameters.

/mesh/hexcore/local-regions/activate

allows you to activate the specified local region(s) for refinement.

/mesh/hexcore/local-regions/deactivate

allows you to deactivate the specified local region(s).

/mesh/hexcore/local-regions/define

defines the local region according to the specified parameters.

/mesh/hexcore/local-regions/delete

deletes the specified refinement region.

/mesh/hexcore/local-regions/ideal-hex-vol

reports the ideal hex volume for the given edge length.

/mesh/hexcore/local-regions/init

creates a default region encompassing the entire geometry.

/mesh/hexcore/local-regions/list-all-regions

lists the defined and active regions in the console.

Chapter 16: Generating the CutCell Mesh

CutCell meshing is a general purpose hex-dominant meshing technique. The CutCell meshing algorithm is suitable for a large range of applications, and due to the large fraction of hex cells in the mesh, often produces better results than regular tetrahedral meshes. This method can be used instead of tetrahedral or hexcore meshing, without requiring a very high quality surface mesh as a starting point. Also, this method uses a direct surface and volume approach without the need of cleanup or decomposition, thereby reducing the turnaround time required for meshing.

The following sections are described in this chapter:

- 16.1. The CutCell Meshing Process
- 16.2. Using the CutCell Dialog Box
- 16.3. Improving the CutCell Mesh
- 16.4. Post CutCell Mesh Generation Cleanup
- 16.5. Generating Prisms for the CutCell Mesh
- 16.6. The Cut-Tet Workflow
- 16.7. Text Commands for CutCell Meshing

16.1. The CutCell Meshing Process

The CutCell meshing process involves the following approach:

1. Objects, material points (optional), and size functions are defined.
2. The initial size of the Cartesian grid is computed based on the minimum and maximum size set for the size functions.
3. A uniform Cartesian grid is created within the bounding box for the geometry.

The base size for the Cartesian grid is computed from the minimum and maximum size specified in the **Size Functions** dialog box as follows:

$$\text{Base Size} = 2^n \times \text{Min Size}$$

such that

$$\text{Base Size} \leq \text{Max Size}$$

where n is the number of refinement levels.

Note

It is strongly recommended to maintain the ratio between the base size and the global minimum size such that **Base Size = $2^n \times \text{Min Size}$** . This will ensure that the correct minimum size is used during the CutCell meshing.

Warning

During initialization, the Cartesian grid created will contain the maximum number of Cartesian cells possible for the computed base size. If the cell count of the initial Cartesian grid exceeds the limit, use the command `/mesh/cutcell/set/max-initial-cells` to set a more appropriate number.

- The size function values are computed and the grid is then adaptively refined based on the local size function values.

Figure 16.1: Schematic Representation of the Cartesian Grid Refinement Using Size Functions (p. 296) shows a schematic representation of the refinement using size functions. The size source specified via the size functions will be assimilated into the size function octree. The final mesh at respective locations will reflect an interpolated size based on the size functions.

Figure 16.1: Schematic Representation of the Cartesian Grid Refinement Using Size Functions

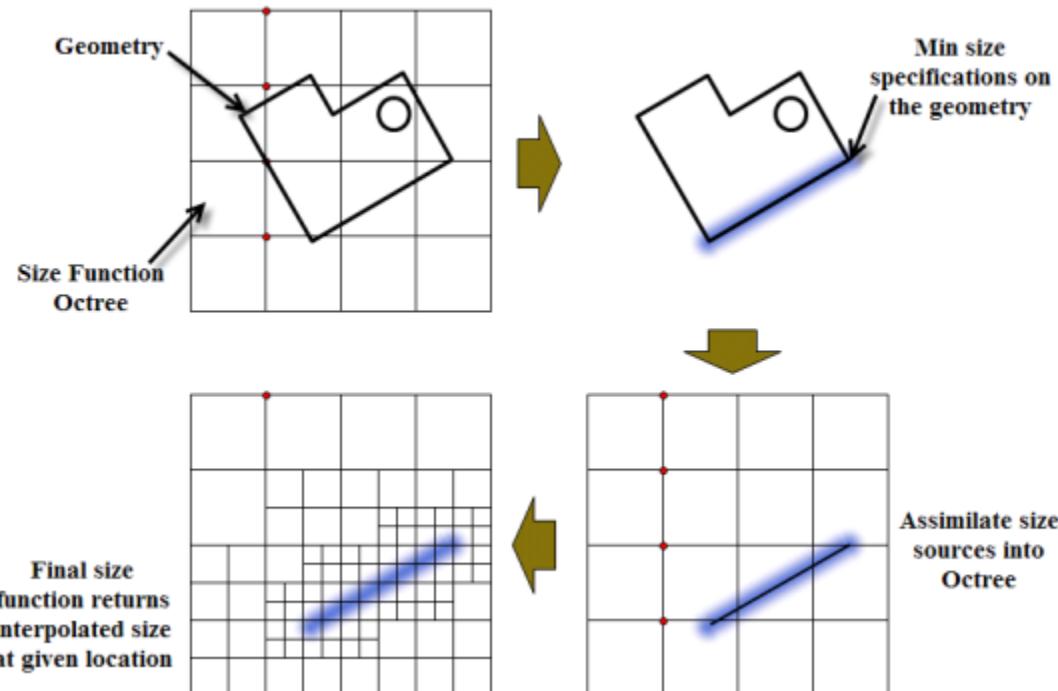
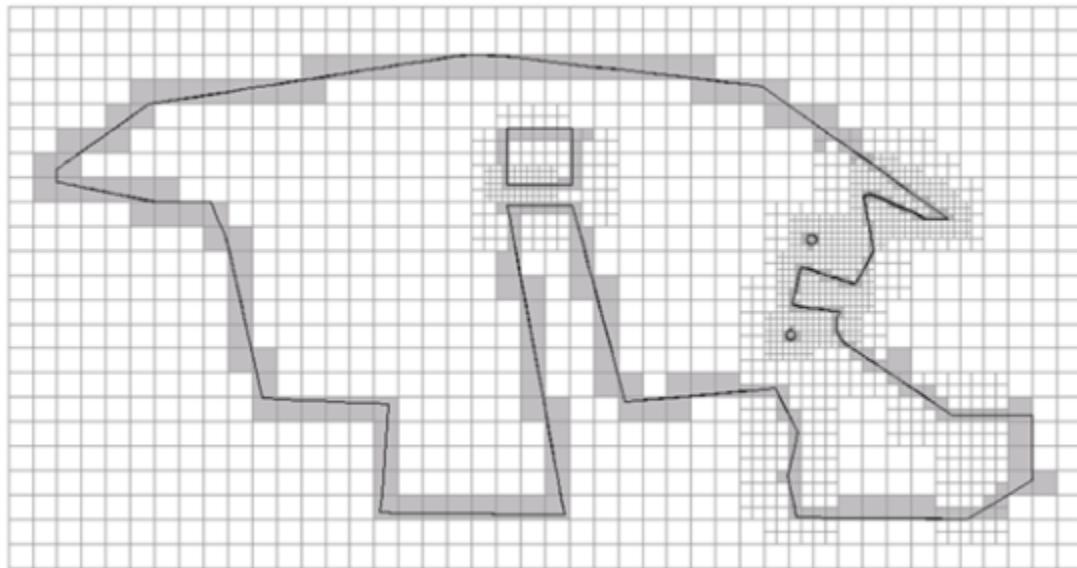
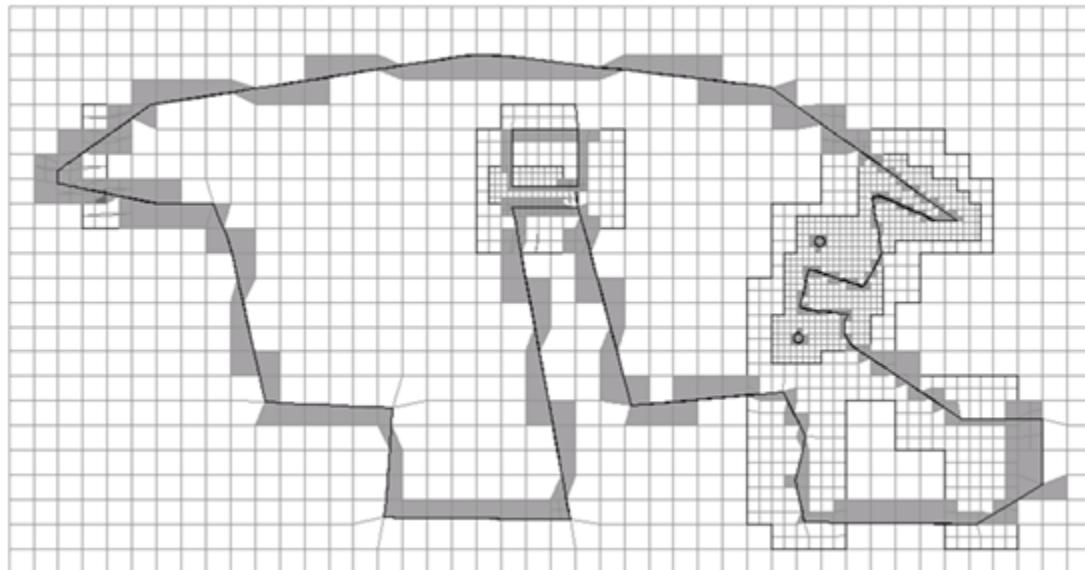


Figure 16.2: Mesh After Refinement (p. 297) shows the mesh after refining the initial grid based on size functions.

Figure 16.2: Mesh After Refinement

5. The cells intersected by the geometry are marked. Only nodes on marked cells are considered for projection. The nodes are projected to the geometry (corner, edge, and face in order of reducing priority).

Figure 16.3: Mesh After Projection (p. 297) shows the mesh after node projection.

Figure 16.3: Mesh After Projection

6. The edges intersected by the geometry are identified. Mesh edges to be preserved/recovered are determined, and are used to construct mesh faces. Once the mesh faces are identified, cells are decomposed to recover these faces. The cells are decomposed based on a number of templates.

Note

The CutCell mesher may have problems capturing features like acute internal and external face angles (e.g., trailing edges of fins, wheel-ground intersections). If such features are not recovered properly, the prisms generated at such locations are most likely to have bad quality.

In such cases, you can use the `set-thin-cut-edge-zones` command (see [Resolving Thin Regions \(p. 303\)](#)) and specify the edges where the feature capturing fails, and then regenerate the mesh.

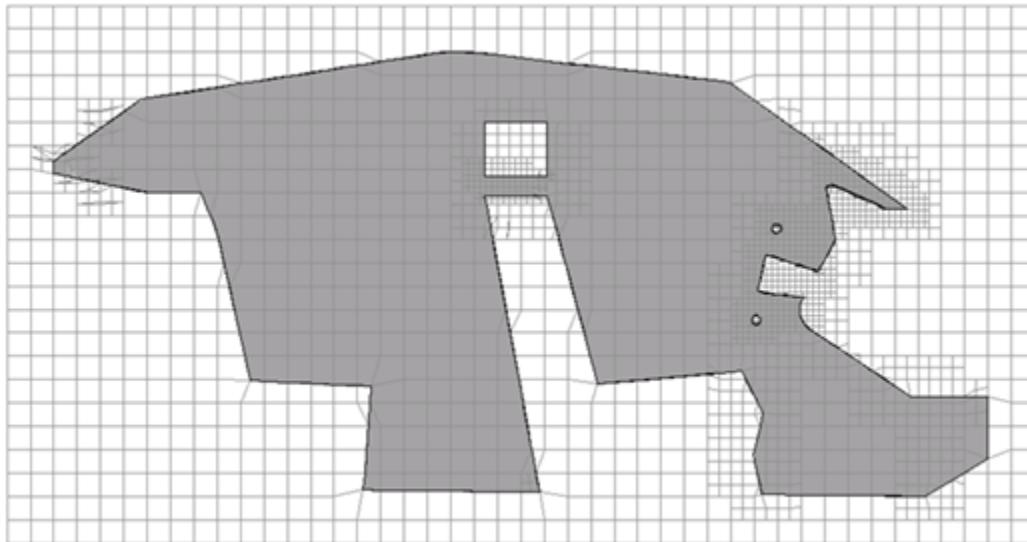
7. The quality of the cells thus generated is improved.

You can set the default parameters to be used for improving the CutCell mesh using the command `/mesh/cutcell/set/set-post-snap-parameters`. This command sets the quality limits and other parameters relevant to the node movement and cavity remeshing which are performed to improve quality.

8. Cells are separated into cell zones based on the respective objects and material point(s) (if any). When a cell has a vertex that lies out of the object while the other lies within the object, the cell will be decomposed further to represent the boundary crossing the cell. A cell included in multiple objects will be included with the object having the highest priority.

[Figure 16.4: Cells Separated After Decomposition \(p. 298\)](#) shows the cells separated into respective cell zones after decomposition.

Figure 16.4: Cells Separated After Decomposition



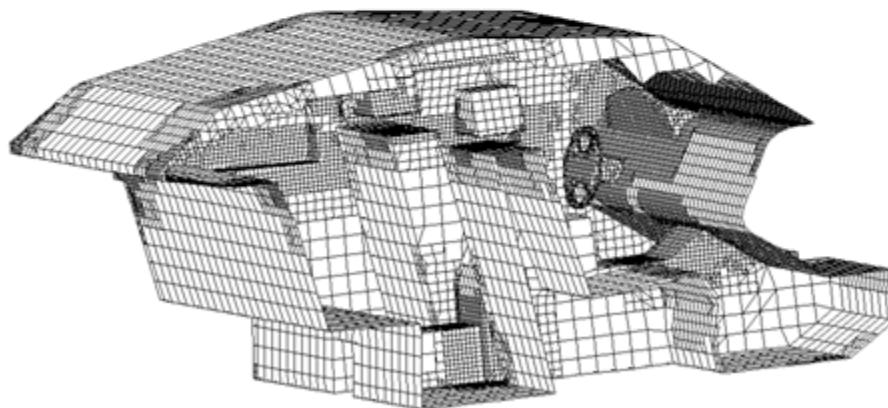
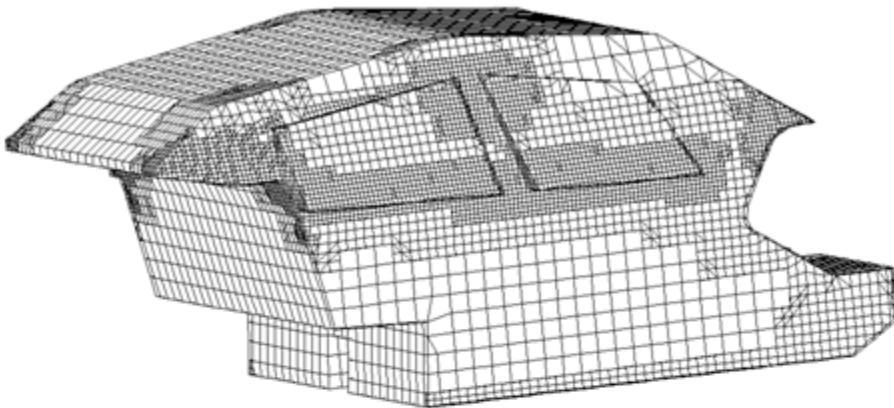
9. Dead zones are generally deleted (`auto-delete-dead-zones?` is enabled by default). Solid zones will be retained or deleted, depending on the setting of `auto-delete-solid-zones?` (disabled by default).
10. The boundary mesh is recovered and separated based on the underlying geometry.
 - Faces whose adjacent neighboring cells are in different cell zones automatically constitute the boundary mesh.
 - The neighboring cells of a face on an internal baffle are in the same cell zone. In such cases, faces close to and nearly parallel to the baffle surface are recovered to represent the baffle surface.
 - As each cell zone is a closed region, the mesh boundary is conformal.

The boundary zone types are assigned based on the underlying geometry zone type. [Figure 16.5: CutCell Mesh After Boundary Recovery \(p. 299\)](#) shows the CutCell mesh after the boundary mesh is recovered.

Note

The CutCell mesher will assign the type **wall** on the surface recovered over a geometry zone of the type **internal** (e.g., baffles) and the type **geometry**.

Figure 16.5: CutCell Mesh After Boundary Recovery



16.2. Using the CutCell Dialog Box

The **CutCell** dialog box and the commands in the /mesh/cutcell menu allow you to perform various tasks related to generating the CutCell mesh.

You can click **Create** in the **CutCell** dialog box or use the command /mesh/cutcell/create to generate the CutCell mesh based on the object(s) and material point(s) selected.

The generic procedure for generating the CutCell mesh is as follows:

1. Define the objects.
 - a. Click the **Objects...** button in the **Create** group box in the **Mesh Generation** task page to open the **Manage Objects** dialog box.
 - b. Make sure the objects are appropriately defined (see [Using the Manage Objects Dialog Box \(p. 107\)](#) for details).
 - c. Create capping surfaces if required (see [Creating Capping Surfaces \(p. 320\)](#) for details).

Important

The objects defined include the corresponding edges which are used for capturing features during the CutCell mesh generation. If you are starting from an earlier setup, you need to use the options in the **Edge Zones** group box in the **Operations** tab of the [Manage Objects Dialog Box \(p. 430\)](#) to add the appropriate edge zones to the object before proceeding.

2. Define the material point(s), if needed.
 - a. Click the **Material Point...** button in the **Create** group box in the **Mesh Generation** task page to open the **Material Point** dialog box.
 - b. Make sure the material point(s) are appropriately defined (see [Using the Material Point Dialog Box \(p. 117\)](#) for details).
3. Define the size functions as appropriate (see [Defining Size Functions \(p. 93\)](#) for details).
 - a. Click the **Size Functions...** button in the **Create** group box in the **Mesh Generation** task page to open the **Size Functions** dialog box.
 - b. Make sure the size functions are appropriately defined.
4. Select the object(s) and material point(s) to be used for the CutCell mesh generation in the **CutCell** dialog box.
 - a. Click the **CutCell...** button on the **Volume Mesh** tab in the **Mesh Generation** task page to open the **CutCell** dialog box.
 - b. Select the appropriate geometry or wrap objects in the **Objects** selection list.
 - c. Select the appropriate material point(s) in the **Material points** selection list.

- d. Enable **Keep Solids Cell Zones** in the **Options** group box, if required.
5. Click **Create** in the **CutCell** dialog box.

The face zones are separated by cell neighbor and normals on face zones connected to the fluid cell zones are oriented into the fluid zone. A face zone group is created for the face zones of each fluid cell zone. Additionally, the defaults for post volume mesh prism generation will be set (see [Generating Prisms for the CutCell Mesh \(p. 305\)](#) for details).

6. Click **Cleanup** button on the **Volume Mesh** tab in the **Mesh Generation** task page.
Operations such as deleting dead zones, deleting geometry/wrap objects, deleting edge zones, removing face/cell zone name prefixes and/or suffixes, deleting unused faces and nodes are performed during the cleanup operation.
7. Generate prism layers, if required.
 - a. Click **Create Prisms...** to open the **Prisms** dialog box.
 - b. Examine the face zone group created for the face zones of the fluid cell zones and determine the face zones for which prism meshing parameters are to be specified.
 - c. Specify the prism meshing parameters as appropriate and click **Apply**.
 - d. Click **Create** in the **Prisms** dialog box.
8. Verify the quality of the CutCell mesh and perform quality improvement operations, if required.

Note

Use the **File/Write/Case...** menu item with the **Write As Polyhedra** option enabled to write the case file in the format that can be read in solution mode in ANSYS Fluent.

16.2.1. Handling Zero-Thickness Walls

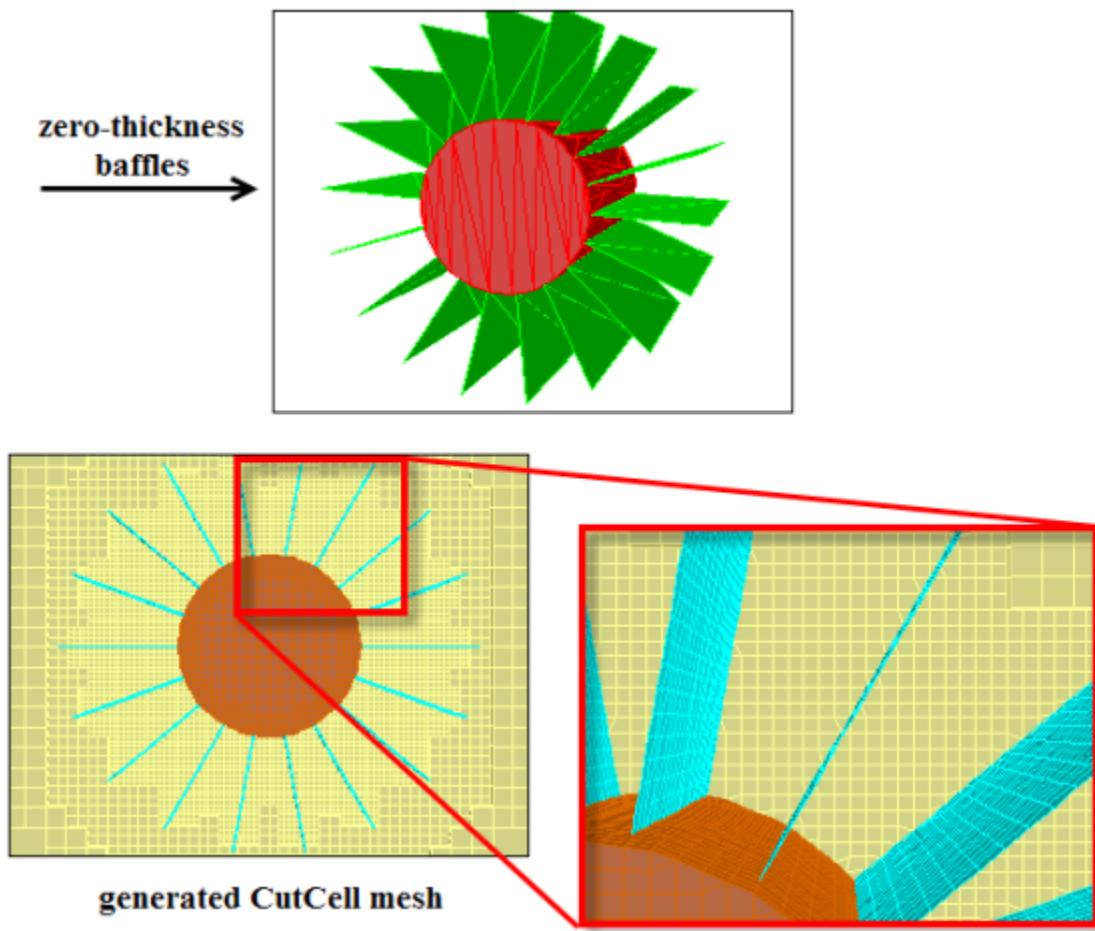
Certain geometries may have components which have zero-thickness. Such configurations can be handled during the CutCell meshing process. There are two types of zero-thickness walls:

- Baffles (zero-thickness walls having the same fluid/solid zone on either side)
- Interior walls (zero-thickness walls having different fluid/solid zones on either side)

To allow the recovery of baffles, any such surface must be of the type **internal**, which will be recovered as a **wall**. For jump conditions, the surface must be of one of the types **fan**, **radiator**, or **porous-jump**, which will be recovered based on the type defined. All such surfaces should be included in the object defined. If not, the surface will not be recovered.

The recovered surface for the zero-thickness baffles will be prefixed by **cutcell-two-sided** in the generated CutCell mesh.

[Figure 16.6: Mesh Generated for Geometry Having Zero-Thickness Baffles \(p. 302\)](#) shows an example where the CutCell mesh has been generated for a stirrer geometry having zero-thickness baffles.

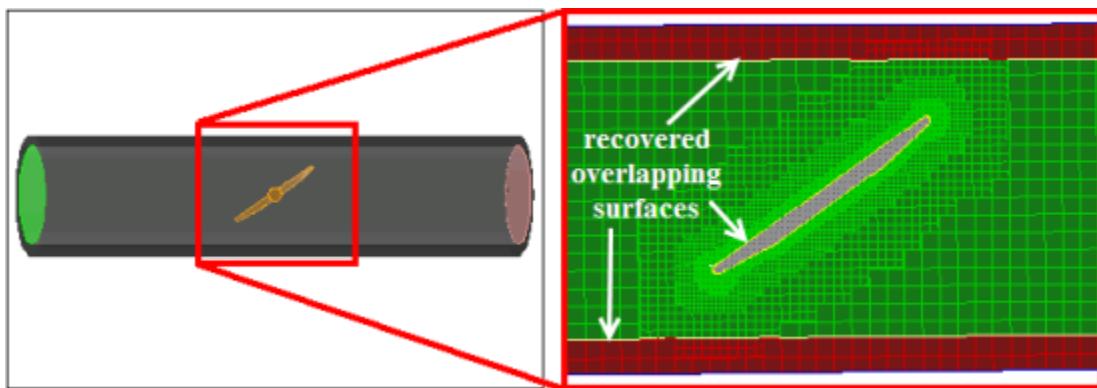
Figure 16.6: Mesh Generated for Geometry Having Zero-Thickness Baffles

For interior walls, you can define two objects, each including the zones comprising the respective domains. Hence, the interior wall will appear in both objects defined, and will be recovered properly.

16.2.2. Handling Overlapping Surfaces

Overlapping surfaces are surfaces from independently defined objects which partially or fully overlap. To allow the recovery of overlapping surfaces as a separate surface, the overlapping walls must be included in both defined objects. The distance between such surfaces must be at least ten times smaller than the minimum size set to avoid "trapped" cell zones. The overlapping surface will be included with the recovered boundary from the object having a higher priority value.

[Figure 16.7: Recovering Overlapping Surfaces \(p. 303\)](#) shows an example where the CutCell mesh has been generated for a butterfly valve. The overlapping surfaces between the valve and flow region as well as the pipe walls and flow region are recovered.

Figure 16.7: Recovering Overlapping Surfaces

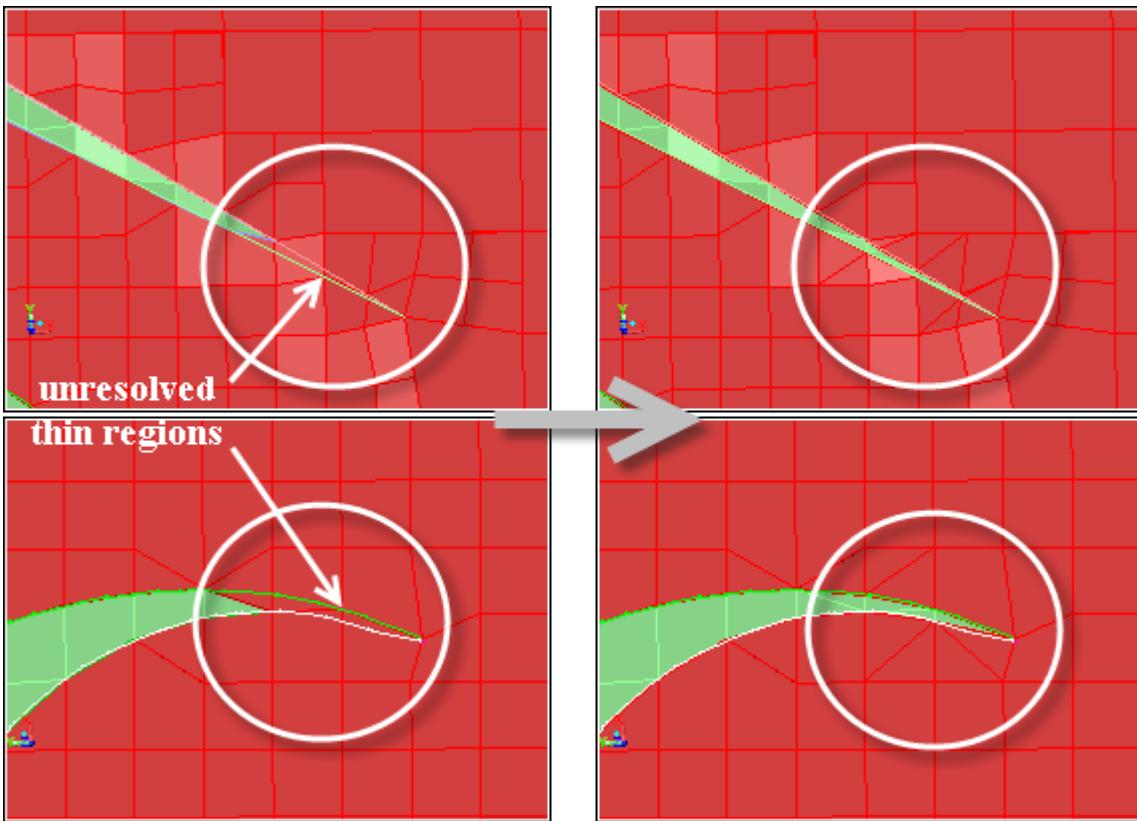
16.2.3. Resolving Thin Regions

Surfaces in close proximity constitute thin regions in the mesh. Examples of thin regions include sharp corners, trailing edge configurations, etc. Such configurations may not be recovered accurately enough by the CutCell mesher, and surface elements may span between nodes on the proximal surfaces.

You can explicitly define thin regions which need to be resolved when the CutCell mesh is generated.

- The command `/mesh/cutcell/set/set-thin-cut-face-zones` allows you to specify the face zones constituting the thin regions to be recovered.
- The command `/mesh/cutcell/set/set-thin-cut-edge-zones` allows you to specify edge zones defining the features to be recovered in thin regions.

[Figure 16.8: Resolving Thin Regions \(p. 304\)](#) shows an example where the thin regions have been resolved during the CutCell meshing process.

Figure 16.8: Resolving Thin Regions

16.3. Improving the CutCell Mesh

The following text commands allow you to improve the CutCell mesh:

/mesh/cutcell/modify/auto-node-move

allows you to use the **Auto Node Move** utility to improve the CutCell mesh quality. For details on the options available, refer to [Auto Node Move Dialog Box \(p. 577\)](#).

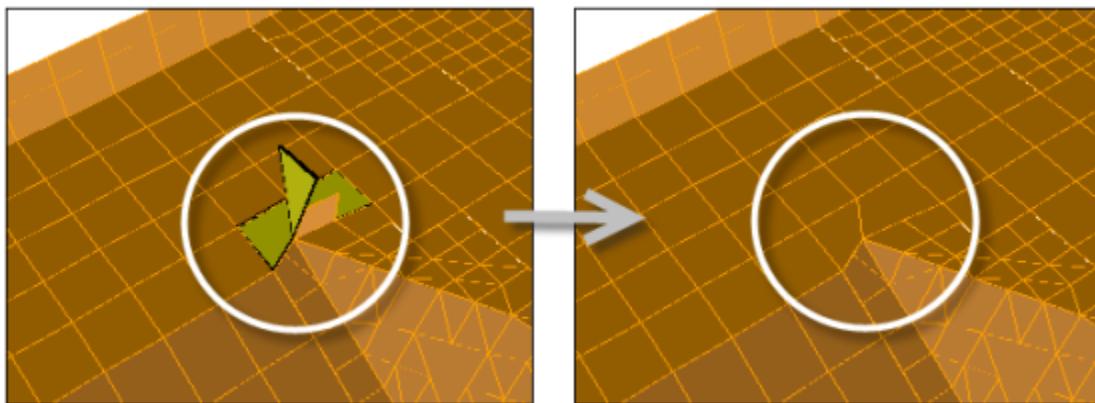
/mesh/cutcell/modify/cavity-remeshing

allows you to use the **Cavity Remeshing** utility to improve the CutCell mesh quality near the boundary. Specify the cell zones to be improved and the quality limit as appropriate. For details on the options available, refer to [Cavity Remeshing \(p. 358\)](#).

/mesh/cutcell/modify/rezone-multi-connected-faces

allows you to resolve multi-connected configurations on the CutCell boundary. Specify an appropriate value for the critical count for contiguous manifold faces.

An example is shown in [Figure 16.9: Rezoning Multiply Connected Faces \(p. 305\)](#) where the multiply connected faces around the surface are removed.

Figure 16.9: Rezoning Multiply Connected Faces

16.4. Post CutCell Mesh Generation Cleanup

After generating the CutCell mesh, you can perform cleanup operations such as deleting dead zones, deleting geometry/wrap objects, deleting edge zones, removing face/cell zone name prefixes and/or suffixes, deleting unused faces and nodes. These operations can be performed by clicking **Cleanup** in the **Mesh Generation** task page.

Note

It is recommended that you use the **Delete** option in the [Manage Cell Zones Dialog Box \(p. 590\)](#) or the command `/mesh/manage/delete` to delete cell zones in the CutCell mesh, instead of the **Mesh/Clear** option or the `/mesh/clear-mesh` command.

16.5. Generating Prisms for the CutCell Mesh

After generating the CutCell mesh, the face zones are separated by cell neighbor and normals on face zones connected to the fluid cell zones are oriented into the fluid zone. A face zone group is created for the face zones of each fluid cell zone. Additionally, the defaults for post volume mesh prism generation will be set. These include the following prism controls to reduce stair-stepping of prism layers:

- Disable edge swapping.

```
/mesh/prism/controls/improve/edge-swap? no
```

- Disable face smoothing.

```
/mesh/prism/controls/face-smooth? no
```

- Set the skewness threshold for edge swapping and edge and node smoothing to 0.95.

```
/mesh/prism/controls/improve/swap-smooth-skew 0.95
```

- Ensure that shrinkage for prism layers is enabled.

```
/mesh/prism/controls/proximity/allow-shrinkage? yes
```

- Set the smoothing rate (rate at which shrinkage is propagated laterally) to 1.2.

```
/mesh/prism/controls/proximity/smoothing-rate 1.2
```

- Disable ignoring of nodes which have poor normals.

```
/mesh/prism/controls/normal/ignore-invalid-normals? no
```

- Set the cell quality criterion for smoothing and quality checking to 0.999.

```
/mesh/prism/controls/improve/max-allowable-cell-skew 0.999
```

- Enable smoothing of normals along the feature lines of the base face zones.

```
/mesh/prism/controls/improve/identify-feature-line? yes
```

- Set the maximum allowable skewness for cap faces after smoothing to 0.999.

```
/mesh/prism/controls/improve/max-allowable-cap-skew 0.999
```

- Set the quality method to **Orthoskew**.

```
/mesh/prism/quality-method orthoskew
```

- Enable the improvement of cell quality for every prism layer. This will involve smoothing of normals in the current layer and perturbation smoothing to improve cell quality in the lower layer.

```
/mesh/prism/controls/improve/cell-quality-improve? yes
```

- Enable the adjustment of prism heights at prism cap corners to improve cell quality.

```
/mesh/prism/controls/improve/corner-height-weight? yes
```

- Disable forcible smoothing of cells if cell quality remains bad after regular smoothing.

```
/mesh/prism/improve/smooth-brute-force? no
```

You can generate prism layers on the appropriate face zones as follows:

1. Click **Create Prisms...** in the **CutCell** dialog box to open the **Prisms** dialog box.
2. Examine the face zone group created for the face zones of the fluid cell zones and determine the face zones for which prism meshing parameters are to be specified.
3. Specify the prism meshing parameters as appropriate and click **Apply**.

Note

Hanging-node cells on a boundary for which prism generation has been assigned, will be triangulated before the prism generation starts.

Important

Attempting to grow thicker prism layers in areas where the aspect ratio of the base to the prism cap is very large may result in an invalid mesh. In such cases, (e.g., external flow problems) it is recommended to use aspect ratio based growth to avoid problems with invalid meshes.

4. Click **Create** in the **Prisms** dialog box.

As the volume mesh already exists, a **Question** dialog box will appear, asking if you want to morph the existing volume mesh. Click **Yes** to generate the prism layers.

Alternatively, use the command `/mesh/cutcell/create-prism` to create the prism layers. Specify the cell zone(s) into which the prism layers are to be grown. The gap factor controls the number of elements in regions of proximity.

Note

If the cell aspect ratio exceeds the specified maximum aspect ratio, a message will appear during the prism meshing process, indicating that shrinkage was limited by the maximum aspect ratio specified.

You could also reduce the gap factor to avoid the cell aspect ratio exceeding the specified maximum aspect ratio. Reducing the gap factor (e.g., a value of 0.5) may improve the quality, but could have a negative impact on the robustness of the morphing. A higher value (e.g., 1.5) is generally more robust, but may not result in the best mesh quality.

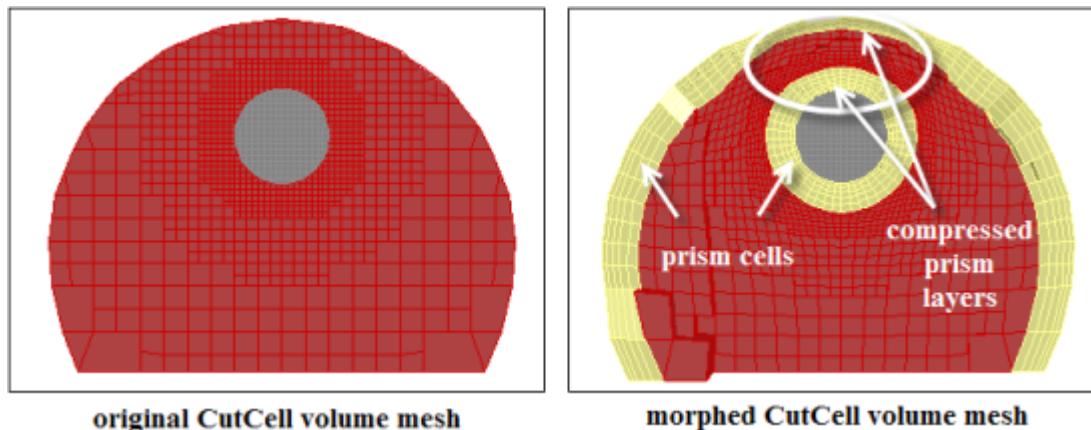
Note

During the surface mesh morphing, only the boundaries of face threads will be treated as features.

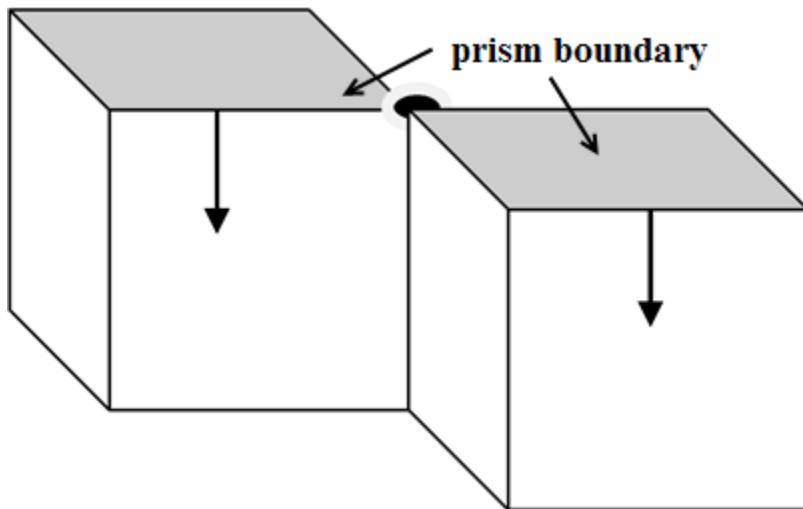
When prism layers are grown into cell zones sharing a face, the prisms will be imprinted on the shared face.

[Figure 16.10: Generating Prisms for the CutCell Mesh \(p. 307\)](#) shows the prism layers created for the CutCell mesh. The prism layers will be compressed in regions of proximity and bad normals. Note that local stair-stepping may occur in areas of poor quality.

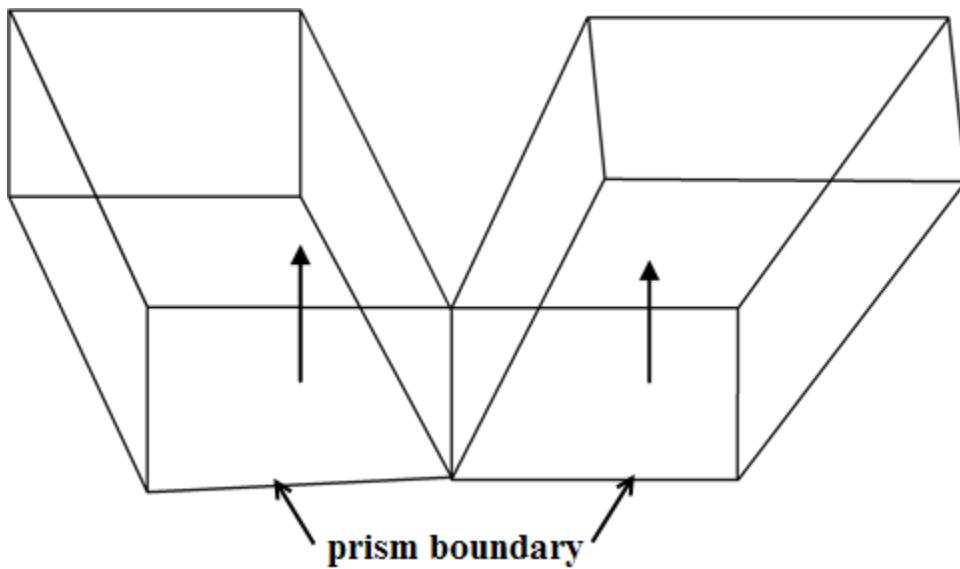
Figure 16.10: Generating Prisms for the CutCell Mesh



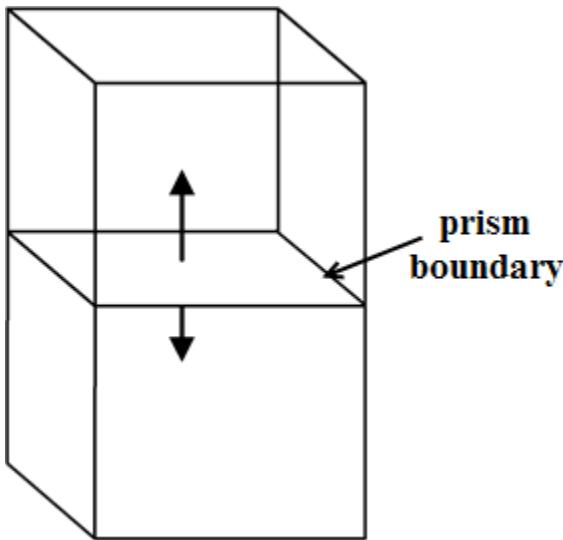
- When prism layers are grown into two volumes sharing an edge, stair-stepping will occur at the common vertex between the volumes ([Figure 16.11: Prism Growth Limitations—Volumes Sharing an Edge \(p. 308\)](#)).

Figure 16.11: Prism Growth Limitations—Volumes Sharing an Edge

- When prism layers are grown into two volumes sharing an edge, stair-stepping will occur at the nodes on the common boundary ([Figure 16.12: Prism Growth Limitations—Volumes Sharing an Edge \(p. 308\)](#)).

Figure 16.12: Prism Growth Limitations—Volumes Sharing an Edge

- Prism layers cannot be grown on both sides of a surface shared by adjacent volumes ([Figure 16.13: Prism Growth Limitations—Volumes Sharing the Prism Base \(p. 309\)](#)).

Figure 16.13: Prism Growth Limitations—Volumes Sharing the Prism Base

A combination of node movement and cavity remeshing is carried out to improve the CutCell mesh quality after the prism layers have been generated. The parameters for improving the CutCell mesh quality can be specified using the command `/mesh/cutcell/set/set-post-morph-parameters` prior to creating the prism layers. The quality method considered is that set by the `/mesh/cutcell/set/set-cutcell-quality-method` command.

5. Ensure that the quality of the prisms created is appropriate. If the quality of the prism cells is low, you can use post-prism smoothing to improve the quality. Use the options in the [Prism Improve Dialog Box \(p. 581\)](#) or text commands such as `/mesh/prism/improve/smooth-improve-prism-cells` to improve the prism cell quality.
6. To further improve the CutCell mesh using the command `/mesh/cutcell/modify/post-morph-improve`, modify the relevant parameters using the command `/mesh/cutcell/set/set-post-morph-parameters`. The quality considered is that set by the `/mesh/cutcell/set/set-cutcell-quality-method` command.

Note

This operation uses a combination of node movement and cavity remeshing to improve the CutCell mesh quality. For details on the relevant parameters, refer to [Moving Nodes \(p. 356\)](#) and [Cavity Remeshing \(p. 358\)](#).

Note

It is recommended that you use the `/mesh/cutcell/modify/post-morph-improve` command for cell zones other than the prism cells.

16.6. The Cut-Tet Workflow

The Cut-Tet workflow allows you to create a tetrahedral, hexcore, or prism mesh based on a triangulated and improved CutCell surface mesh. The initial requirement is the generated CutCell mesh.

The generic workflow is as follows:

1. Make the CutCell boundaries conformal to remove hanging-nodes.

```
/mesh/cutcell/modify/split-boundary cutcell-* ,
```

The command `/mesh/cutcell/modify/split-boundary` creates a copy of the specified CutCell boundary zone(s) and makes the boundary mesh conformal at the hanging-nodes on the copied zone(s). The new zone(s) will be named based on the original zone name(s) prefixed by **split-**.

2. Clear the volume mesh.

```
/mesh/clear-mesh
```

3. Triangulate the split CutCell boundary zones.

```
/boundary/remesh/triangulate split-* , yes
```

The split CutCell boundary zones will be replaced by the corresponding triangulated boundary zones.

4. Improve the boundary mesh by swapping edges based on a node degree value other than 6. The node degree is defined as the number of edges connected to the node.

```
/boundary/improve/degree-swap
```

Use boundary smoothing operation(s) to further improve the boundary mesh quality (see subsequent step).

Note

Do not use the `/boundary/improve/swap` operation immediately after the `/boundary/improve/degree-swap` operation, as this will restore the original degree configurations.

5. Improve the triangulated boundary mesh further using any of the following options available:

- a. Use wrapper smoothing operations to improve the boundary mesh quality.

- i. Change the type of the triangulated boundary mesh to **wrapper**.

- ii. Use the wrapper post improve operations to improve the boundary mesh quality. Refer to [Post Wrapping Improvement Operations \(p. 195\)](#) for detailed descriptions of the options available.
-

Note

Make sure the original geometry is retained when the CutCell mesh is generated, this required for reprojection when using the post improve operations.

- b. Use operations like improving based on boundary mesh quality, smoothing, and swapping to improve the boundary mesh quality. Refer to [Improving Boundary Surfaces \(p. 138\)](#) for the detailed descriptions of the options available.
- c. Use surface remeshing with the size functions defined for the CutCell mesh.
 - i. Extract edge zones from the triangulated boundary mesh.

This allows you to maintain the boundary node locations on the remeshed faces and facilitates the connection between the remeshed faces using simple node merge operations.

Split the edges having sizes bigger than the size function defined using the Scheme command (ti-refine-edge-threads-by-sf (get-edge-zones-of-filter 'split-*))

- ii. Remesh the triangulated boundary mesh using the defined size functions using the Scheme command (ti-remesh-multiple-threads (get-face-zones-of-filter 'split-*) #f #t "none" #t)

This allows you to remesh multiple surfaces in a single operation.

Tip

You can also add a meshed size function on the edge zones extracted from the triangulated boundary mesh to improve the quality.

- iii. Merge duplicate boundary nodes on the remeshed boundary zones.
- 6. Generate the tetrahedral/hexcore/prism mesh. Refer to [Generating Tetrahedral Meshes \(p. 267\)](#), [Generating the Hexcore Mesh \(p. 285\)](#), and [Generating Prisms \(p. 229\)](#) for the detailed descriptions of the meshing options available. You can also use the **Auto Mesh** option (see [Using the Auto Mesh Dialog Box \(p. 219\)](#)) for generating the mesh.

Note

To generate the tetrahedral mesh using the defined size functions, use (tgsetvar ! 'impose/cell-size-method 4). This will also respect the body of influence size functions defined for the CutCell meshing.

You can also use additional operations during the volume meshing process, as appropriate. e.g., /mesh/manage/merge-dead-zones can be used to merge dead zones having a cell count lower than the specified threshold value, with the adjacent cell zone.

- 7. Improve the mesh quality using the various improvement options available. Refer to [Improving the Mesh \(p. 347\)](#) for detailed descriptions of the options available.

16.7. Text Commands for CutCell Meshing

/mesh/cutcell/create

creates the CutCell mesh based on the objects and material points selected.

/mesh/cutcell/create-prism

creates the prism layers on the recovered boundary based on the zone-specific prism parameters set. Specify the cell zone(s) into which the prism layers are to be grown and the gap factor as appropriate.

/mesh/cutcell/modify/auto-node-move

allows you to use the **Auto Node Move** utility to improve the CutCell mesh quality.

/mesh/cutcell/modify/cavity-remeshing

allows you to use the **Cavity Remeshing** utility to improve the CutCell mesh quality near the boundary.

/mesh/cutcell/modify/post-morph-improve

improves the quality of the CutCell mesh post-prism generation.

/mesh/cutcell/modify/rezone-multi-connected-faces

allows you to resolve multiply-connected faces on the CutCell boundary.

/mesh/cutcell/modify/split-boundary

creates a copy of the specified CutCell boundary zone(s) and makes the boundary mesh conformal at the hanging-nodes on the copied zone(s). The new zone(s) will be named based on the original zone name(s) prefixed by **split-**.

/mesh/cutcell/objects/change-object-type

allows you to change the object type (geom, wrap, or mesh).

/mesh/cutcell/objects/create

creates the object based on the priority, cell zone type, face zone(s), edge zone(s), and object type specified. You can specify the object name or retain the default blank entry to have the object name generated automatically.

/mesh/cutcell/objects/create-and-activate-domain

creates and activates the domain comprising the face zone(s) from the object(s) specified.

/mesh/cutcell/objects/create-groups

creates a face group and an edge group comprising the face zone(s) and edge zone(s) included in the specified object(s), respectively.

/mesh/cutcell/objects/create-intersection-loops

allows you to create intersection loops for objects.

- The **collectively** option creates an interior edge loop at the intersection between two adjacent face zones included in the same object and between multiple objects.
- The **individually** option creates an interior edge loop at the intersection between two adjacent face zones included in the same object.

/mesh/cutcell/objects/create-mesh-object-from-wrap

allows you to create a mesh object from an existing wrap object. This command is not relevant for CutCell meshing.

/mesh/cutcell/objects/create-multiple

creates multiple objects by creating an object per face zone specified. The objects will be named automatically based on the prefix and priority specified.

/mesh/cutcell/objects/delete

deletes the specified object(s).

/mesh/cutcell/objects/delete-all

deletes all the defined objects.

/mesh/cutcell/objects/delete-all-geom-and-wrap

deletes all the defined objects of type geom and wrap.

/mesh/cutcell/objects/delete-unreferenced-faces-and-edges

deletes all the faces and edges which are not included in any defined objects.

/mesh/cutcell/objects/extract-edges

extracts the edge zone(s) from the face zone(s) included in the specified object(s), based on the edge-feature-angle value specified (/mesh/cutcell/objects/set/set-edge-feature-angle).

/mesh/cutcell/objects/improve-feature-capture

allows you to imprint the edges comprising the object on to the object face zones to improve feature capture for wrap or mesh objects. You can specify the number of imprinting iterations and additional aggressive imprinting iterations to be performed.

/mesh/cutcell/objects/improve-mesh-object

improves the mesh object specified. This command is not relevant for CutCell meshing.

/mesh/cutcell/objects/list

lists the defined objects, indicating the respective cell zone type, priority, face zone(s) and edge zone(s) comprising the object, object type, and object reference point in the console.

/mesh/cutcell/objects/merge

merges the specified objects into a single object.

/mesh/cutcell/objects/merge-edges

merges all the edge zones in an object into a single edge zone.

Note

If the object comprises edge zones of different types (boundary and interior), the edge zones of the same type (boundary or interior) will be merged into a single edge zone.

/mesh/cutcell/objects/merge-voids

allows you to merge voids in the mesh object after the sewing operation. This command is not relevant for CutCell meshing.

/mesh/cutcell/objects/merge-walls

merges all the face zones of type wall in an object into a single face zone.

/mesh/cutcell/objects/remove-gaps/

contains options for removing gaps between wrap objects. The commands in this sub-menu are not relevant for CutCell meshing.

/mesh/cutcell/objects/separate-faces-by-angle

separates the face zone(s) comprising the object based on the angle specified.

/mesh/cutcell/objects/separate-faces-by-seed

separates the face zone(s) comprising the object based on the seed face specified.

/mesh/cutcell/objects/set/set-edge-feature-angle

sets the edge feature angle to be used for extracting edge zone(s) from the face zone(s) included in the object(s).

/mesh/cutcell/objects/set/show-edge-zones?

displays the edge zone(s) comprising the object(s) drawn in the graphics window.

/mesh/cutcell/objects/set/show-face-zones?

displays the face zone(s) comprising the object(s) drawn in the graphics window.

/mesh/cutcell/objects/sew/

contains options for sewing wrap objects. The commands in this sub-menu are not relevant for CutCell meshing.

/mesh/cutcell/objects/update

allows you to update the objects defined when the face and/or edge zone(s) comprising the object have been deleted.

/mesh/cutcell/objects/wrap/

contains options for the object wrapping operation. The commands in this sub-menu are not relevant for CutCell meshing.

/mesh/cutcell/set/auto-delete-dead-zones?

controls the automatic deleting of the dead zones in the CutCell mesh. This option is enabled by default.

/mesh/cutcell/set/auto-delete-solid-zones?

controls the automatic deleting of the solid zones in the CutCell mesh. This option is disabled by default.

/mesh/cutcell/set/create-material-point

allows you to define a material point. Specify the fluid zone name and the location to define the material point.

/mesh/cutcell/set/delete-all-material-points

allows you to delete all defined material points.

/mesh/cutcell/set/delete-material-point

deletes the specified material point.

/mesh/cutcell/set/list-material-points

lists all the defined material points.

/mesh/cutcell/set/max-initial-cells

specifies the maximum number of cells in the initial Cartesian grid.

/mesh/cutcell/set/set-cutcell-quality-method

allows you to set the quality measure for the improve operations. The default measure used is the ortho skew metric.

/mesh/cutcell/set/set-post-morph-parameters

allows you to set parameters for improving the CutCell mesh post-prism generation using the /mesh/cutcell/modify/post-morph-improve command.

/mesh/cutcell/set/set-post-snap-parameters

allows you to set parameters for improving the CutCell mesh quality.

/mesh/cutcell/set/set-thin-cut-face-zones

allows you to specify face zones constituting the thin regions to be recovered during the CutCell meshing process.

/mesh/cutcell/set/set-thin-cut-edge-zones

allows you to specify edge zones defining the features in thin regions to be recovered during the CutCell meshing process.

/mesh/cutcell/size-functions/create

defines the size function based on the specified parameters.

/mesh/cutcell/size-functions/create-defaults

creates default size functions based on face and edge curvature and proximity.

/mesh/cutcell/size-functions/delete

deletes the specified size function(s).

/mesh/cutcell/size-functions/delete-all

deletes all the defined size functions.

/mesh/cutcell/size-functions/list

lists all the defined size functions and the corresponding parameter values defined.

/mesh/cutcell/size-functions/reset

allows you to reset the size function background grid in order to have the mesh distribution recomputed based on all defined size functions.

/mesh/cutcell/size-functions/reset-global-controls

resets the values for the global controls to the defaults.

/mesh/cutcell/size-functions/set-global-controls

sets the values for the global minimum and maximum size, and growth rate.

/mesh/cutcell/size-functions/set-prox-gap-tolerance

sets the tolerance relative to minimum size to take gaps into account. Gaps whose thickness is less than the global minimum size multiplied by this factor will not be regarded as a proximity gap.

Chapter 17: Object-Based Meshing

Object-based meshing is an alternative meshing approach, where you can generate a tetrahedral, hexcore, or hybrid volume mesh based on meshing objects from the imported geometry (from CAD or the .tgc format from ANSYS Meshing). In this case, you need to create a conformally connected surface mesh using the object wrapping and sewing operations before generating the volume mesh.

Note

Ensure that the model is suitably scaled during import and the global minimum size is at least 0.01 to avoid numerical problems during mesh generation.

You can alternatively use the CutCell mesher to directly create a hex-dominant volume mesh for the geometry based on meshing objects created. See [Generating the CutCell Mesh \(p. 295\)](#) for details on generating the CutCell mesh.

Refer to [Meshing Objects and Material Points \(p. 103\)](#) for details on meshing objects.

- [17.1. Object-Based Meshing Workflow](#)
- [17.2. Preparing the Geometry](#)
- [17.3. Defining Objects](#)
- [17.4. Diagnostics](#)
- [17.5. Fixing Holes in Objects](#)
- [17.6. Wrapping Objects](#)
- [17.7. Sewing Objects](#)
- [17.8. Improving the Mesh Objects](#)
- [17.9. Build Topology](#)
- [17.10. Generating the Volume Mesh Based on Mesh Objects](#)
- [17.11. Text Commands for Object Based Meshing](#)

17.1. Object-Based Meshing Workflow

The generic procedure for generating the volume mesh based on meshing objects is as follows:

1. Prepare the geometry as required (create capping surfaces, close gaps and patch holes, define the far-field domain, etc.).
2. Define size functions.
3. Define the material points as appropriate.
4. Define the objects.
5. Use the diagnostic tools to identify and fix any problems in the geometry.
6. Wrap the objects to create the conformal surface mesh on the relevant surfaces of the objects.
7. Repair the wrap objects if required.

8. Sew the objects together to obtain a conformal, triangular surface mesh between bodies.
9. Improve the surface mesh quality.
10. Fill the volume using the meshing options available.

The **Mesh Generation** task page contains options that allow you to generate the volume mesh based on meshing objects.

17.1.1. Using the Mesh Generation Task Page

The **Mesh Generation** task page can be used for generating the mesh as follows:

1. Prepare the geometry as required. Refer to [Preparing the Geometry \(p. 319\)](#) for the options available.
2. Define the size functions as appropriate (see [Defining Size Functions \(p. 93\)](#) for details).
 - a. Click the **Size Functions...** button in the **Create** group box in the **Mesh Generation** task page to open the **Size Functions** dialog box.
 - b. Make sure the size functions are appropriately defined.
3. Define the material points, if needed.
 - a. Click the **Material Point...** button in the **Create** group box in the **Mesh Generation** task page to open the **Material Point** dialog box.
 - b. Make sure the material points are appropriately defined (see [Using the Material Point Dialog Box \(p. 117\)](#) for details).
4. Define the objects.
 - a. Click the **Objects...** button in the **Create** group box in the **Mesh Generation** task page to open the **Manage Objects** dialog box.
 - b. Make sure the objects are appropriately defined (see [Using the Manage Objects Dialog Box \(p. 107\)](#) for details).
5. Obtain conformal, well-connected representations of geometry objects using the options in the [Wrap Dialog Box \(p. 454\)](#).
 - a. Click the **Wrap...** button in the **Objects** group box in the **Mesh Generation** task page to open the **Wrap** dialog box.
 - b. Select the geometry objects in the **Objects** selection list.
 - c. Select the appropriate object wrapping options and click **Wrap**. Refer to [Wrapping Objects \(p. 327\)](#) for details on the options available.

Alternatively, if conformal tessellation options were selected during import, you may convert geometry objects to wrap or mesh type directly using the **Change Type** option in the [Manage Objects Dialog Box \(p. 430\)](#) (available when multiple objects are selected) or the change-object-type command. When you change the object type to wrap or mesh, it is assumed that all selected objects

- have conformal faceting and that the quality of the surface mesh triangles is similar to that in a CFD surface mesh.
6. Repair the wrap objects if needed. See [Repairing Wrap Objects \(p. 331\)](#) for details on the options available.
 7. Sew the appropriate wrap objects using the options in the [Sew Dialog Box \(p. 457\)](#).

Note

The sewing operation is performed only on wrap objects.

- a. Click the **Sew...** button in the **Surface Mesh** group box in the **Mesh Generation** task page to open the **Sew** dialog box.
 - b. Select the wrap objects in the **Objects** selection list.
 - c. Enter an appropriate name in the **New Object Name** field.
 - d. Click **Sew**.
- Alternatively, for a single wrap object or flow volume, change the object type to mesh and use the options in the [Improve Dialog Box \(p. 459\)](#).
8. Improve the quality of the mesh object using the options in the [Improve Dialog Box \(p. 459\)](#).
 - a. Select the mesh object in the **Objects** list.
 - b. Specify the improvement parameters and click **Improve**.
 9. Fill the volume using the meshing options available in the [Auto Mesh Dialog Box \(p. 464\)](#). Refer to [Generating the Volume Mesh Based on Mesh Objects \(p. 342\)](#) for details.
 10. Click **Cleanup** in the **Mesh Generation** task page. Operations such as deleting dead zones, deleting geometry/wrap objects, deleting edge zones, removing face/cell zone name prefixes and/or suffixes, deleting unused faces and nodes are performed during the cleanup operation.

17.2. Preparing the Geometry

When the geometry is imported from CAD, there may be a number of gaps and the faceted geometry may be disconnected. Though operations such as merging nodes and faceted stitching can be used to partially connect the model, some gaps may remain and features may be lost.

You may want to perform tasks like creating a wind tunnel or far-field domain, close annular gaps or create capping surfaces for inlets or outlets, define material points or create groups of zones for models with a large number of zones. The options in the **Create** group box in the **Mesh Generation** task page allow you to perform such operations to prepare the geometry for further operations producing a good-quality surface mesh.

[17.2.1. Using a Bounding Box](#)

[17.2.2. Closing Annular Gaps in the Geometry](#)

[17.2.3. Creating Capping Surfaces](#)

[17.2.4. Defining Material Points](#)

[17.2.5. Defining Size Functions](#)

[17.2.6. Using User-Defined Groups](#)

17.2.1. Using a Bounding Box

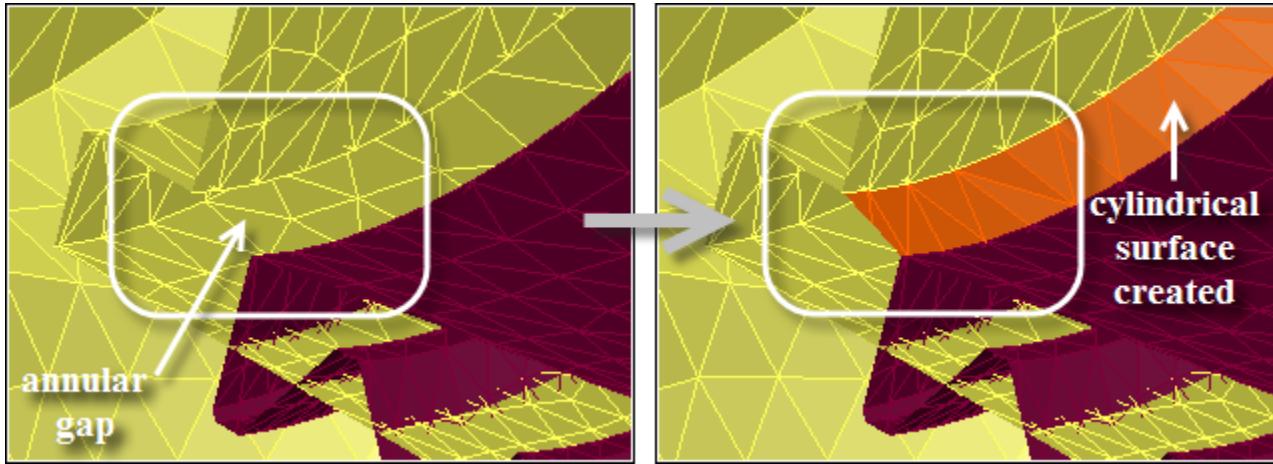
The bounding box tool can be used to create a wind tunnel or far-field domain for the imported geometry. You can also use the bounding box tool to create a body of influence to be used for defining size functions. See [Using the Bounding Box Dialog Box \(p. 163\)](#) for details on creating a bounding box.

17.2.2. Closing Annular Gaps in the Geometry

The **Cylinder** dialog box allows you to create a cylindrical or annular surface to close radial gaps in the geometry. A wrap object is created for the surface created. The options are available in the [Cylinder Dialog Box \(p. 417\)](#).

[Figure 17.1: Closing a Radial Gap \(p. 320\)](#) shows an example where a radial gap is closed using a cylindrical surface.

Figure 17.1: Closing a Radial Gap



1. Select **3 Arc, 1 Height Node** in the **Options** list.
2. Click **Select Nodes...** and select 3 nodes on one circle and one (height node) on the other across the radial gap.
3. Enter an appropriate value for **Edge Length**.
4. Enable **Create Object** and disable **Caps**.
5. Click **Create**.

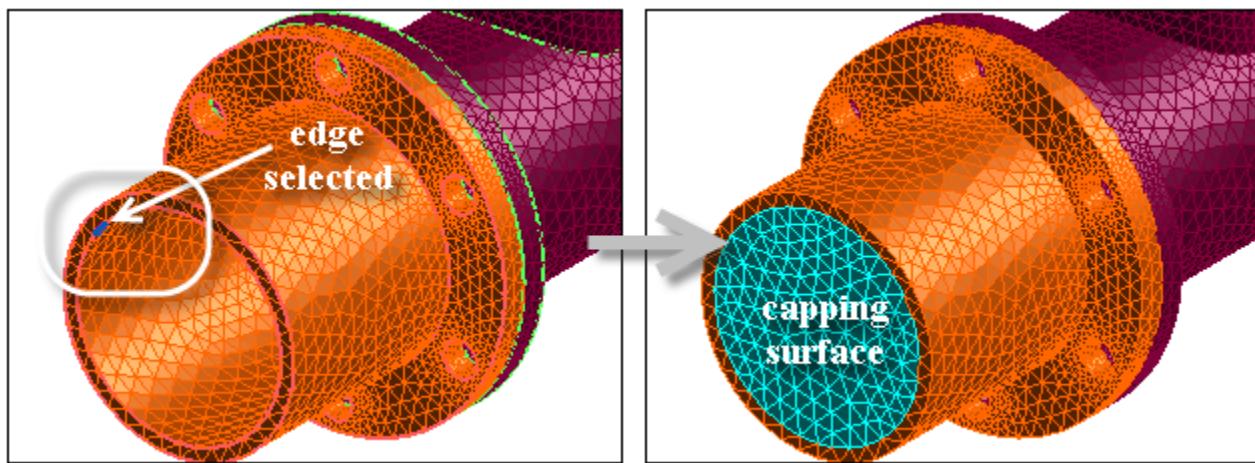
17.2.3. Creating Capping Surfaces

The capping surfaces tool allows you to cap an inlet/outlet and assign the appropriate zone type. Additionally, you can patch other complex shapes to close gaps including sharp angles, small pockets in the geometry.

The **Capping Surfaces** dialog box allows you to create a cap surface for an inlet or outlet based on entities selected in the graphics window. The appropriate zone type can also be assigned when the capping surface is created. A wrap object is created for the capping surface created. The options are available in the [Capping Surface Dialog Box \(p. 420\)](#).

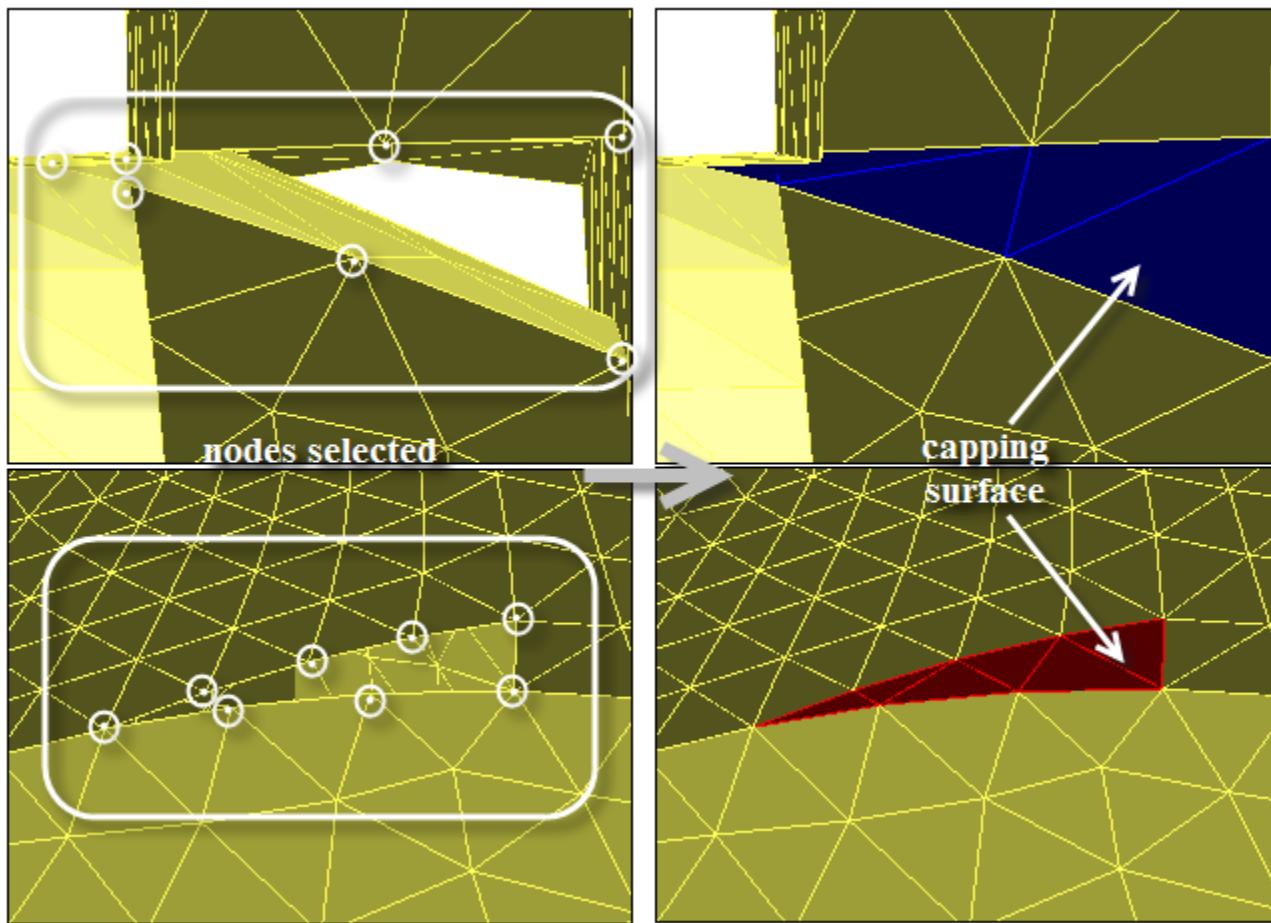
Figure 17.2: Creating a Capping Surface Using an Edge (p. 321) shows an example where the circular capping surface is created by selecting an edge on the existing object.

Figure 17.2: Creating a Capping Surface Using an Edge



You can also select multiple edges and create multiple capping surfaces in a single operation. A single object will be created which comprises the individual capping surface zones created based on the edges selected.

Figure 17.3: Creating a Capping Surface Using Nodes (p. 322) shows an example where the capping surfaces are created by selecting nodes on the existing object.

Figure 17.3: Creating a Capping Surface Using Nodes

1. Specify an appropriate name for the capping object in the **Object Name** field.
2. Select the zone type in the **Type** drop-down list.
3. Select the appropriate entity type in the **Entity Type** list.
Edges can be used for reasonably regular shapes, while you can select nodes for more complex patch shapes.
4. Enable **Create Edges** if you want to extract edges for the cap surface created.
5. Enable **Remesh** to create a more uniform distribution on the cap surface. This is required if the original geometry has many bad quality or skinny facets.
6. Click **Create**.

17.2.4. Defining Material Points

Material points can be defined in addition to objects to extract the internal fluid surface, using the object wrapping operation. Material points can be created using the options in the [Material Points Dialog Box \(p. 426\)](#).

17.2.5. Defining Size Functions

Size functions allow you to control the mesh size distribution. You can set up size functions using the options in the [Size Functions Dialog Box \(p. 421\)](#). The **Create Defaults** option allows you to create default size functions based on face and edge curvature and proximity. Alternatively, you can read in a size-field file using the **File/Read/Size Field...** menu item, or specify the size-field file when using **Conformal Tessellation** options for CAD import.

The **Draw Sizes** option in the **Global Controls** group box allows you to visually examine the defined sizes to ensure that they are adequate for the mesh to be generated. After defining the size functions, click **Compute** to compute the size field based on the defined size functions. You can then specify appropriate **Min** and **Max** values and draw the contours of size on the selected face zone(s). A visual indication of mesh size is also available using the mouse probe if the select-filter is set to size (**Ctrl+Y**).

17.2.6. Using User-Defined Groups

You can define user-defined groups to better handle large models. You can create a face group and an edge group comprising the face zones and edge zones included in the specified objects using the options in the **Zone Group** group box in the **Operations** tab in the [Manage Objects Dialog Box \(p. 430\)](#). You can activate a particular group using the **Activate** option in the [User Defined Groups Dialog Box \(p. 428\)](#).

Additionally, a face zone group is automatically created when a mesh object is created using the **Sew** operation. This face zone group is prefixed by **_mesh_group**, and allows easy selection of mesh object face zones for various operations (improve, smooth, etc.).

Note

When an object is deleted along with the face and edge zones comprising the object, the corresponding groups will also be deleted.

17.3. Defining Objects

Objects define the domain to be meshed. Objects can be defined and manipulated in the **Manage Objects** dialog box (see [Defining Objects \(p. 107\)](#) and [Object Manipulation Operations \(p. 108\)](#)).

When the face zones and/or edge zones comprising an object are deleted, the object definition will be updated. If all the zones associated with an object are deleted, the empty object will be deleted as well.

17.4. Diagnostics

The options in the [Diagnostic Tools](#) dialog box allow you to find and fix problems in boundary meshes on objects.

Diagnostics is divided into three sections:

- Find and fix assembly problems such as gaps or intersections between objects on the **Geometry** tab.
- Ensure mesh validity by finding and fixing problems such as free or multi-connected edges, overlapping or intersecting faces, etc. on the **Face Connectivity** tab.
- Improve surface mesh quality using a combination of up to three criteria on the **Quality** tab.

On the desired tab,

17.4.1. Geometry Issues

17.4.2. Face Connectivity Issues

17.4.3. Quality Checking

1. Select the desired **Issue** (or **Quality** measure), and then set the relevant options in the dialog box.
2. Click **Mark** to identify and provide a count of the **Unvisited** problems in your boundary mesh.
3. Click **First (Next)** to step through the problems individually. At each step, the graphics window will highlight the identified problem. You may then take the necessary action to correct the identified problem, if necessary.

17.4.1. Geometry Issues

On the **Geometry** tab, choose from the following **Issues**:

- Self Intersections
- Cross Intersections
- Self Face Proximity
- Cross Face Proximity
- Self Edge Proximity

For a full description, refer to the [Diagnostic Tools](#) dialog box page of the ANSYS Fluent Meshing User's Guide

17.4.2. Face Connectivity Issues

On the **Face Connectivity** tab, choose from the following **Issues**:

- Free
- Multi
- Self Intersections
- Self Proximity
- Duplicate
- Spikes
- Islands
- Steps
- Non-manifold
- Invalid Normals
- Leaks
- Deviation

For a full description, refer to the [Diagnostic Tools](#) dialog box page of the ANSYS Fluent Meshing User's Guide

17.4.3. Quality Checking

On the **Quality** tab, use the scroll buttons to specify up to three quality measures to be used. Then select from the following quality measure options.

- Skewness
- Size Change
- Edge Ratio
- Area
- Aspect Ratio
- Warp
- Dihedral Angle
- Ortho Skew

For more information on how ANSYS Fluent calculates the quality and adjusts the mesh, see the [Quality Measures](#) page and the [Boundary Improve](#) page.

For a full description, refer to the [Diagnostic Tools](#) dialog box page of the ANSYS Fluent Meshing User's Guide

17.5. Fixing Holes in Objects

The region/volume of interest should be well connected before the volume mesh can be generated. Any holes/leaks need to be located and fixed before proceeding with volume meshing. Holes/leaks can be located using a material point. When you select the material point and set the minimum and maximum limits for the hole size, the locations at which a path traces back to the material point through the geometry and which are within the given hole size limits, are recognized as holes or leaks.

You can locate such holes and fix them using the options in the [Fix Holes Dialog Box \(p. 448\)](#). The generic procedure for locating holes is as follows:

1. Select the appropriate objects in the **Objects** selection list and click **Draw**.
2. Select the appropriate material point from the **Material Point** drop-down list.

The default material point (**external**) is a suitable point external to the objects selected. Click **Reset** to change the material point to be used.

3. Enter the minimum and maximum limits for the hole sizes under consideration under **Max. Size** and **Min. Size**.

The size values allow you to limit the search for holes to a relevant subset based on the size range. As you locate the respective holes and fix them, the possibility of false holes is reduced.

You can click **Draw Sizes** to check the sizes on the geometry.

4. Click **Find Holes** to locate all the holes based on the specified parameters.

The number of holes is reported in the **Count** field in the **Wetted Holes** group box.

The options available for fixing holes are as follows:

- The options in the **Wetted Holes** group box allow you to traverse the holes detected and patch or open all the holes.
 - Click **Draw All** to view all the wetted holes detected.
 - Click **First** to view the first hole. The display will be limited to the region of the hole.
Click **Next** repeatedly to traverse all the wetted holes and examine them individually.
 - Click **Patch All** to automatically patch all the wetted holes detected.
 - Click **Open All** to open all the wetted holes detected (when the holes identified do not represent actual holes).
- The options in the **Selected Holes** and **Create Patch** group boxes allow you to fix the detected holes individually.
 - Click **Patch** to create a patch to fix the currently displayed hole.
 - Click **Open** if the current hole identified does not represent an actual hole. Opening the hole allows you to indicate that the approximated wetted region should propagate through the configuration detected as a hole.
 - Click **Ignore** if the current hole identified is not relevant for the object wrapping/sewing operation.
 - Click **Cylinder...** to open the [Cylinder Dialog Box \(p. 417\)](#), for creating a cylindrical surface to fix the hole.
 - Click **Caps...** to open the [Capping Surface Dialog Box \(p. 420\)](#), for creating capping surfaces to fix the hole.
- The **Trace to Points** group box allows you to locate holes/leaks by tracing a path from the **Material Point** to the selected **Target Point** through all the objects. Even after you fix the holes/leaks, you can use the **Trace to Points** options to verify that no tiny leaks remain.
- The options in the **Wetted Surface** group box allow you to view the approximate representation of the region of volume/interest based on the objects and material point selected. You can also create a wrap object using the shrink-wrap method.
 - Click **Show** to view the approximate representation of the wetted surface, based on the objects and material point selected. Enable **Overlay Graphics** to view the wetted surface along with the objects selected.
Click **Hide** to hide the wetted surface in the graphics window.
 - Click **Update** to update the wetted surface representation after any hole fixing operations (patch, open, creating caps, etc.) are performed.
 - Specify an appropriate **New Object Name** and select the appropriate option in the **Geometry Recovery** drop-down list. Click **Shrink Wrap** to create a wrap object using the shrink-wrap method.

The generic procedure for locating holes is as follows:

1. Examine the wetted surface representation and the wetted holes individually by traversing through all the wetted holes.
2. Fix the holes using the options in the **Wetted Holes**, **Selected Holes**, or **Create Patch** group box, as appropriate.
3. Click **Update** in the **Wetted Surface** group box to update the wetted surface representation.
4. Examine the updated region for further leaks or holes. Fill any remaining leaks or holes before proceeding.
 - a. Select a point from the **Target Points** selection list and click **Trace**.
The path connecting the cells corresponding to the points specified will be highlighted and will pass through the hole/leak.
 - b. Fix the holes using the options in the **Wetted Holes**, **Selected Holes**, or **Create Patch** group box, as appropriate.
 - c. Click **Update** in the **Wetted Surface** group box to update the wetted surface representation.
 - d. Click **Update** in the **Trace to Points** group box and then click **Trace** again to locate any remaining holes/leaks.
 - e. Repeat the hole fixing steps as needed to ensure no holes/leaks remain.

After all the holes/leaks are fixed you can proceed to wrap the objects. Specify an appropriate **New Object Name** and select the appropriate option in the **Geometry Recovery** drop-down list. Click **Shrink Wrap** to create a wrap object for the selected objects using the shrink-wrap method. For detailed options for object wrapping, use the [Wrap Dialog Box \(p. 454\)](#).

17.6. Wrapping Objects

The object wrapping operation extracts a conformal, well connected mesh on the relevant surfaces of the geometry objects.

The wrapping operation uses an appropriate reference point to simplify the geometry objects.

Note

When conformal tessellation options are used for CAD import or a good quality surface mesh is read in, the mesh is already well-connected and conformal. In such cases, you may choose to skip the object wrapping operation and directly convert the geometry objects to wrap type using the **Change Type** option in the [Manage Objects Dialog Box \(p. 430\)](#). When you change the object type to wrap, it is assumed that all selected objects have conformal faceting and that the quality of the surface mesh triangles is similar to that in a CFD surface mesh.

The following methods are available:

Shrink-Wrap

The **Shrink-Wrap** method uses a specialized version of the boundary wrapper utility to extract a well connected wrap object on the relevant surfaces of the geometry object. The shrink-wrap method is useful for cases involving defeaturig or when you need to walk over features.

A Cartesian grid is overlaid on each geometry object and contiguous regions are created. The Cartesian grid is then refined based on the size functions to better represent the geometry object. The interface is extracted on the boundary of the non-intersecting Cartesian volume region that encloses the reference point. A well-connected **wrap** object will be created.

The following **Geometry Recovery** options are available for the shrink-wrap method:

Low

allows you to create a rough wrapped representation of the geometry object.

Medium

performs additional refinement, imprinting and aggressive imprinting iterations to improve the feature recovery. Individual zones are recovered based on the original geometry object and then rezoned.

High

allows better feature capture and surface remeshing based quality improvement. Use the **Resolution Factor** to set sampling coarser or finer than the final surface mesh. Degenerate and island edges are deleted, and intersected and remeshed as appropriate. The edges are imprinted on the wrapped zones and individual zones are recovered based on the original geometry object and then rezoned. Surfaces are remeshed based on size functions/size field.

Note

This option requires clean geometry.

The command `/objects/wrap/set/shrink-wrap-rezone-parameters` allows you to set the parameters for improving the wrap object surface quality using rezoning. The geometry object zones will be separated based on the separation angle specified to improve the feature imprinting on the wrap object.

Note

The shrink-wrap option approximates the geometry using a stairstep like Cartesian grid without projection. It requires finer cells to resolve thin gaps. In cases when a gap area is not curved and not aligned to the Cartesian axes, you may need to refine 3–4 times finer than the gap thickness.

This should be taken into account while setting the global and local minimum size for size functions and the cells per gap for the proximity size function being used.

Cut-Wrap

The **Cut-Wrap** method uses the CutCell mesher to extract a well connected wrap object on the relevant surfaces of the geometry object. The cut-wrap method offers good feature and zero-thickness (baffle) capture and ensures a well-connected surface mesh. However, the surface mesh quality is dependent on cleanliness of the objects imported.

A uniform Cartesian grid is created with a base size computed from the minimum and maximum values specified for the size functions. The Cartesian grid is then refined based on the size functions. The cells intersected by the geometry object are marked and the nodes on the marked cells are then projected to the geometry object. The cells are decomposed to construct faces based on the feature edges of the geometry object. The recovered CutCell boundary is made conformal and then triangulated to obtain the **wrap** object.

You can use the command /objects/wrap/wrap to create the wrap object. Specify the objects to be wrapped, the method, and other relevant parameters.

17.6.1. Object Wrapping Options

The following object wrapping options are available:

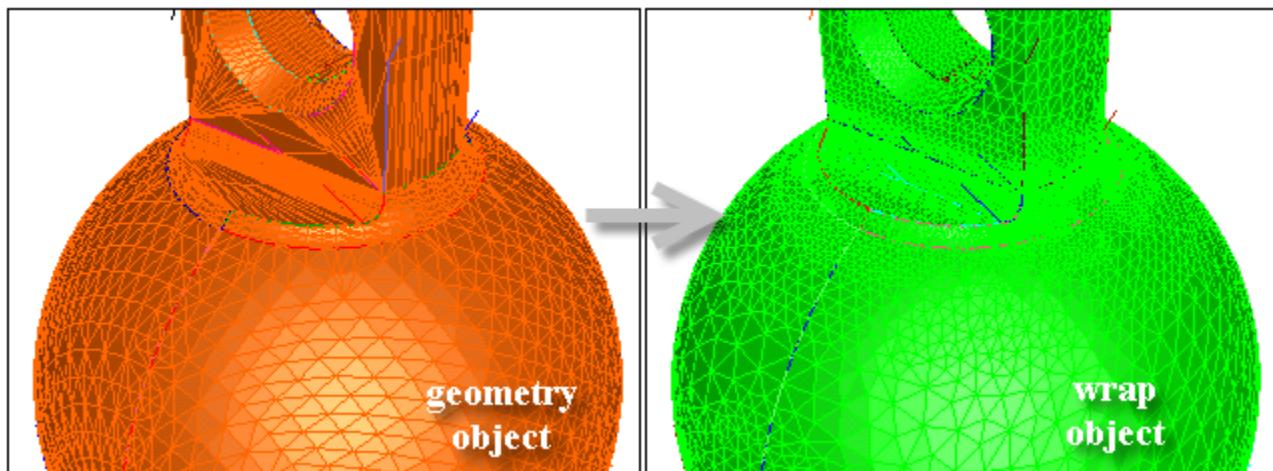
- 17.6.1.1. Wrapping Selected Objects
- 17.6.1.2. Creating a New Wrap Object
- 17.6.1.3. Resolving Thin Regions During Object Wrapping

17.6.1.1. Wrapping Selected Objects

The **Wrap Selected Objects** option allows you create a conformal surface mesh for each geometry object selected. A well-connected wrap object and corresponding zones will be created for each geometry object selected. The object name is the same as the original geometry object name. This operation uses a suitable material point which is external to the objects selected. Hence, any internal voids or features will be eliminated. The wrap objects created are suitable for repair operations like gap or thickness removal, hole fixing, etc. or as the final surface mesh.

[Figure 17.4: Wrapping Individual Objects \(p. 329\)](#) shows the wrap object created for a geometry object when the **Wrap Selected Objects** option is used.

Figure 17.4: Wrapping Individual Objects



17.6.1.2. Creating a New Wrap Object

The **Create New Object** option allows you to create a single, well-connected wrap object and corresponding zones, based on the objects selected. You can specify an appropriate name for the wrap object created.

When you select **external** in the **Material Point** drop down list, a suitable reference point external to the objects is selected for the object wrapping operation. Hence, any internal voids or features will be

eliminated. [Figure 17.5: Multiple Solids \(p. 330\)](#) shows an example with multiple solid objects. The aim of the object wrapping operation is to mesh the solids conformally and create a single solid cell zone in the final mesh. [Figure 17.6: Single Solid Surface \(p. 330\)](#) shows the wrap object created, where the multiple solids are unified.

Figure 17.5: Multiple Solids

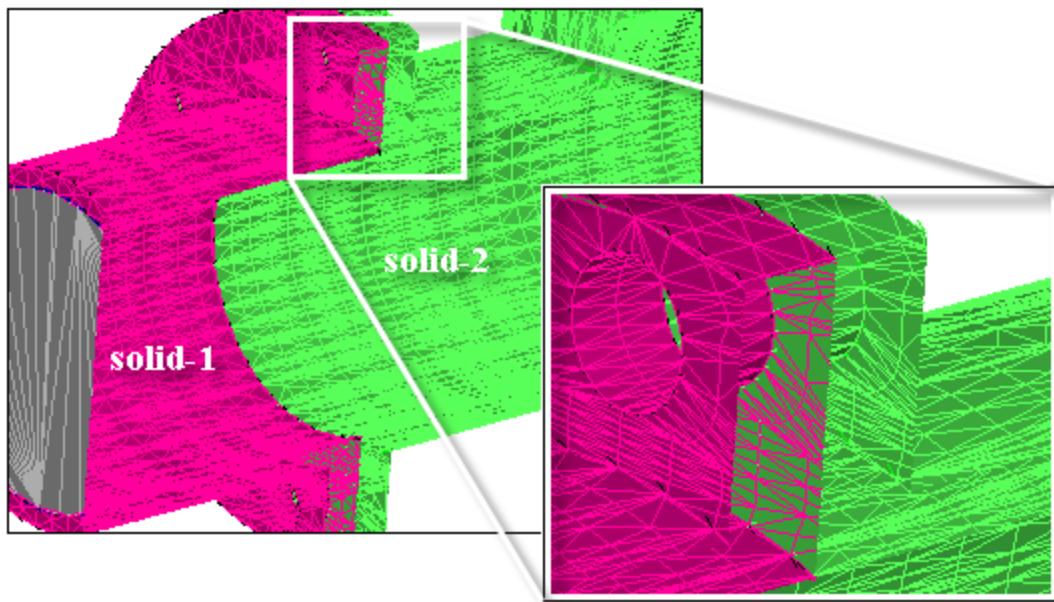
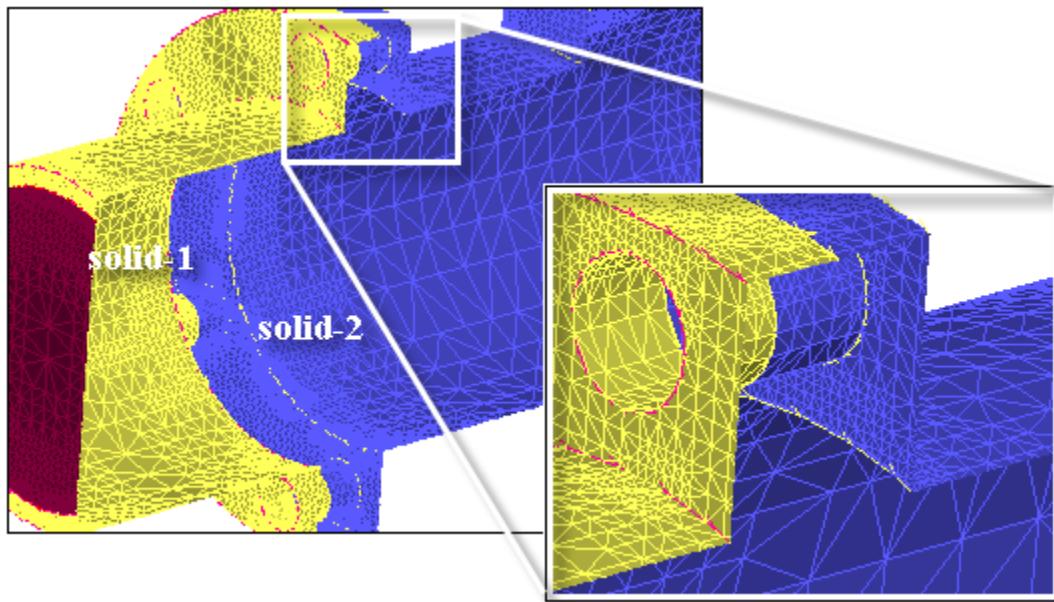
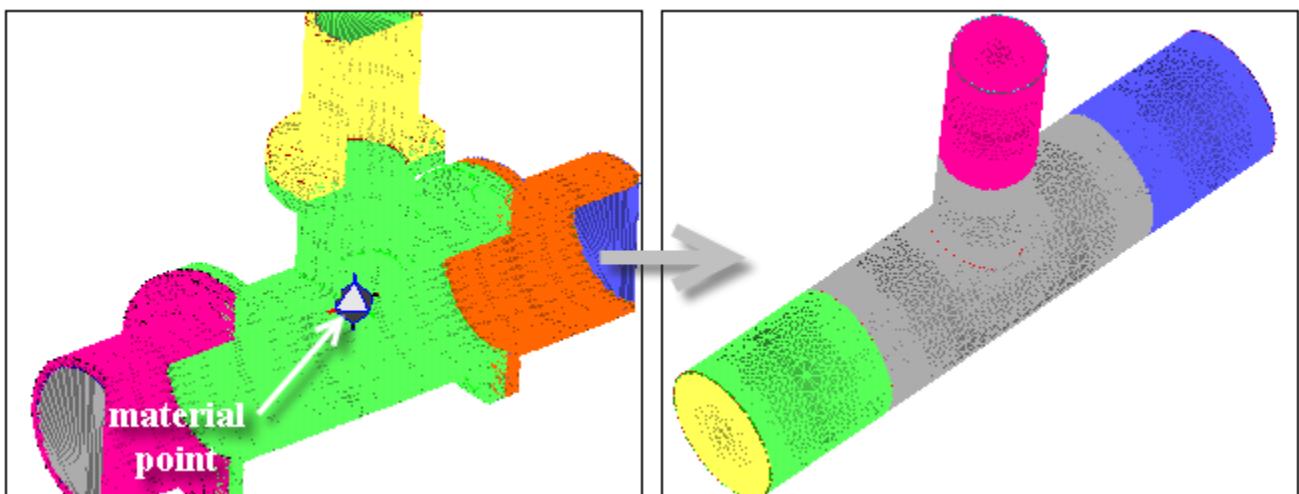


Figure 17.6: Single Solid Surface



Alternatively, you can use this option to create the flow volume, when the surrounding solids are not needed in the final mesh. All geometry objects bounding the flow volume should be selected for the wrapping operation. Select the appropriate material point needed to identify the “wetted” region that comprises the flow volume. The material point can be defined in the [Material Points Dialog Box \(p. 426\)](#).

[Figure 17.7: Extracting the Flow Volume \(p. 331\)](#) shows the extraction of the internal flow volume for a T-junction by specifying a material point and using the **Fluid Surface** option.

Figure 17.7: Extracting the Flow Volume

17.6.1.3. Resolving Thin Regions During Object Wrapping

Surfaces in close proximity constitute thin regions in the mesh. Examples of thin regions include sharp corners, trailing edge configurations, etc., which may not be recovered accurately enough during the object wrapping operation and surface elements may span between nodes on the proximal surfaces.

You can use the command `/objects/wrap/set/include-thin-cut-edges-and-faces` to allow better recovery of such configurations during the object wrapping operation.

17.6.2. Repairing Wrap Objects

The following options are available for repairing wrap objects:

- 17.6.2.1. Detecting Holes in the Wrap Object
- 17.6.2.2. Removing Gaps Between Wrap Objects
- 17.6.2.3. Removing Thickness in Wrap Objects
- 17.6.2.4. Improving Feature Capture For Wrap Objects

17.6.2.1. Detecting Holes in the Wrap Object

The command `/objects/wrap/set/report-holes?` allows you to check for holes in the wrap object created. Holes, if any will be reported at the end of the object wrapping operation.

The command `/objects/wrap/set/max-free-edges-for-hole-patching` allows you to set the maximum number of free edges in a loop to fill the holes.

The command `/objects/wrap/check-holes` allows you to check for holes in the wrap objects. The number of hole faces marked will be reported.

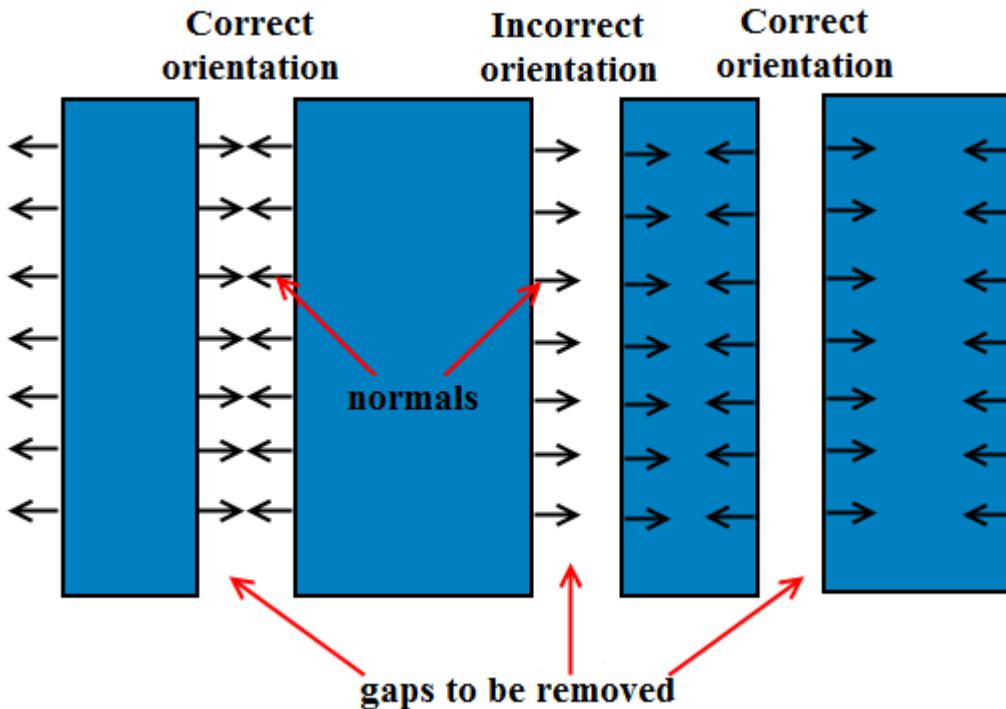
17.6.2.2. Removing Gaps Between Wrap Objects

Gaps between wrap objects can be removed using the options in the [Remove Gaps Dialog Box \(p. 452\)](#). The gaps can be between surfaces (face-face) or between a surface and an edge (face-edge).

1. Select the objects for the gap removal operation in the **Objects** selection list.
2. Select **Remove Gaps Between Objects** in the **Operation** list.

3. Specify an appropriate value for the **Min. Gap Distance**, **Max. Gap Distance**, and **Percentage Margin**.
4. Specify an appropriate value for **Critical Angle**. The critical angle is the maximum angle between the faces constituting the gap to be removed.
5. **Ignore Orientation** is enabled by default. If the thickness of any of the object selected is less than the **Max. Gap Distance**, then you can disable **Ignore Orientation**. In this case the orientations of the normals will be considered. Ensure that, the normals in the gaps to be removed are appropriately oriented (Figure 17.8: Orientation of Normals in Gap (p. 332)).

Figure 17.8: Orientation of Normals in Gap



6. Select the appropriate option for feature edge extraction (**none**, **feature**, or **all**) and specify the **Extract Angle** to be used.
7. Select the type of gap in the **Gap Type** list (**Face-Face** or **Edge-Face**).
 - For face-face gap removal, select the appropriate option for projection in the **Order** list. You can choose to project the faces to the object of higher priority (**Low To High Priority**) or to the object of lower priority (**High to Low Priority**).
8. Click **Mark** to see the faces marked for projection.
9. Click **Remove** to remove the gaps between the objects selected.

Figure 17.9: Removing Gaps Between Objects—Face-Face Option (p. 333) shows an example where a face-face gap between wrap objects has been removed.

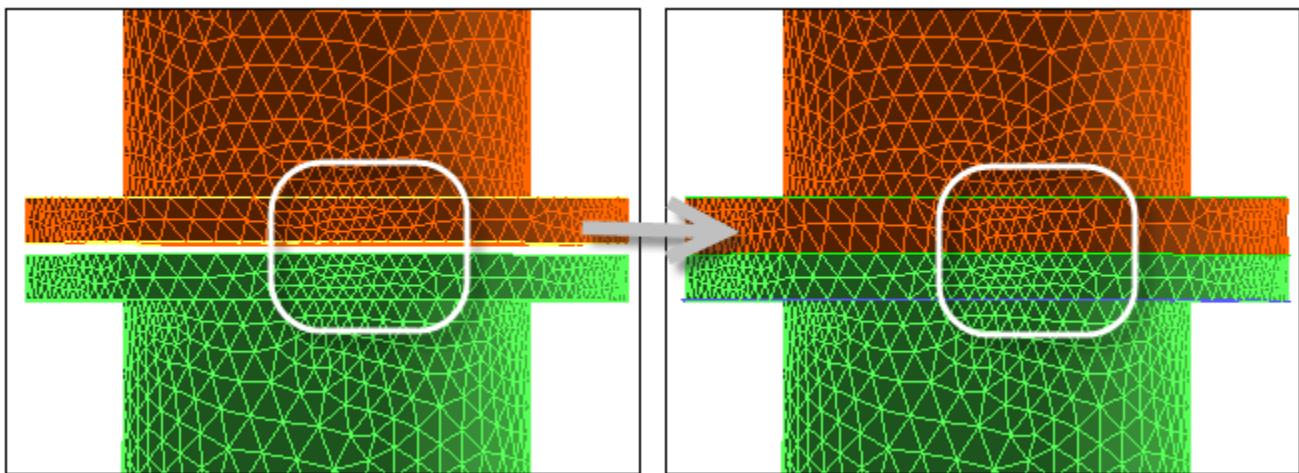
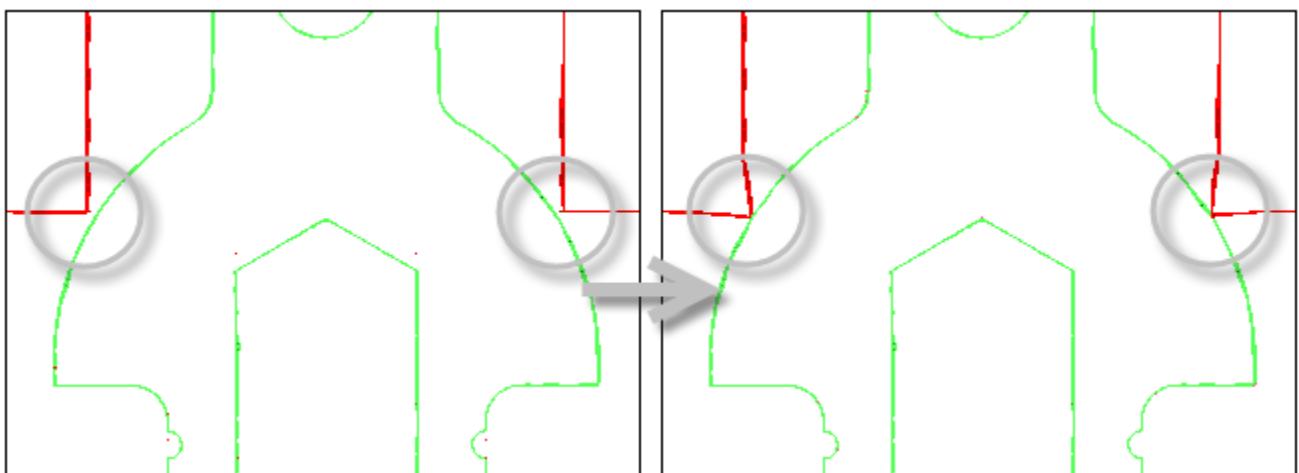
Figure 17.9: Removing Gaps Between Objects—Face-Face Option

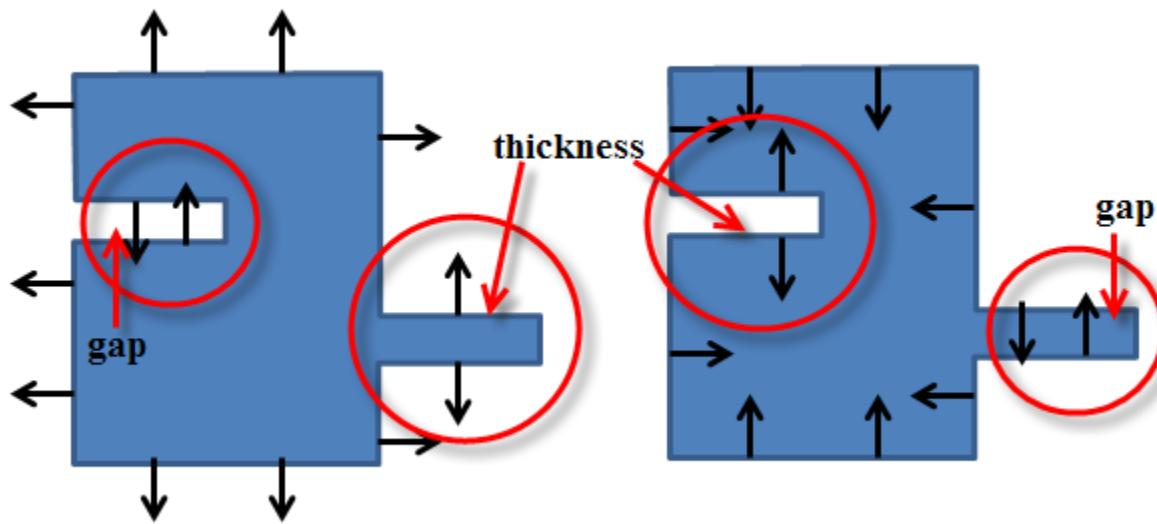
Figure 17.10: Removing Gaps Between Objects—Face-Edge Option (p. 333) shows an example where a face-edge gap between wrap objects has been removed.

Figure 17.10: Removing Gaps Between Objects—Face-Edge Option

17.6.2.3. Removing Thickness in Wrap Objects

The thickness across a wrap object can be removed by projecting the close surfaces to a mid-surface. During the thickness removal operation, the object face zones will be separated in order to project the close surfaces to the mid-surface and the separated zones will be merged back after the projection. The options for thickness removal are available in the [Remove Gaps Dialog Box \(p. 452\)](#).

Configurations can be distinguished as gaps or thicknesses based on the orientation of the normals on the wrap object ([Figure 17.11: Gap and Thickness Configurations \(p. 334\)](#)). The normals across a gap configuration point toward each other, while for a thickness configuration, the normals point away from each other.

Figure 17.11: Gap and Thickness Configurations

When using the option for thickness removal, ensure that the wrap object normals are appropriately oriented depending on the configurations to be removed (see [Figure 17.11: Gap and Thickness Configurations \(p. 334\)](#)).

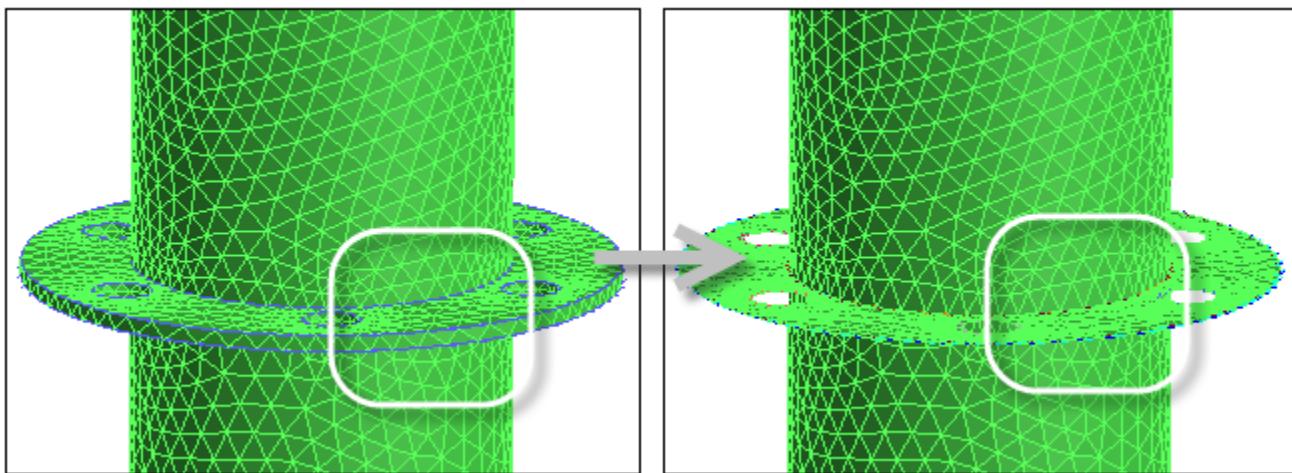
The generic procedure for removing thickness in objects using the [Remove Gaps Dialog Box \(p. 452\)](#) is as follows:

1. Select the objects for the thickness removal operation in the **Objects** selection list.
2. Select **Remove Thickness In Objects** in the **Operation** list.
3. Specify an appropriate value for the **Max. Gap Distance** and **Percentage Margin**.
4. Specify an appropriate value for **Critical Angle**. The critical angle is the maximum angle between the faces constituting the thickness to be removed.
5. Select the appropriate option for feature edge extraction (**none**, **feature**, or **all**) and specify the **Extract Angle** to be used.
6. Click **Remove** to remove the thickness in the objects selected.

Note

The thickness removal operation involves separation and merging back of face zones, which may affect the mesh quality. It is recommended that you save the mesh before proceeding.

[Figure 17.12: Removing Thickness in Objects \(p. 335\)](#) shows an example where the thickness in the wrap object has been removed.

Figure 17.12: Removing Thickness in Objects

17.6.2.4. Improving Feature Capture For Wrap Objects

The command `/objects/improve-feature-capture` allows you to imprint the edges comprising the object on to the object face zones to improve feature capture for wrap objects. You can specify the number of imprinting iterations to be performed.

Note

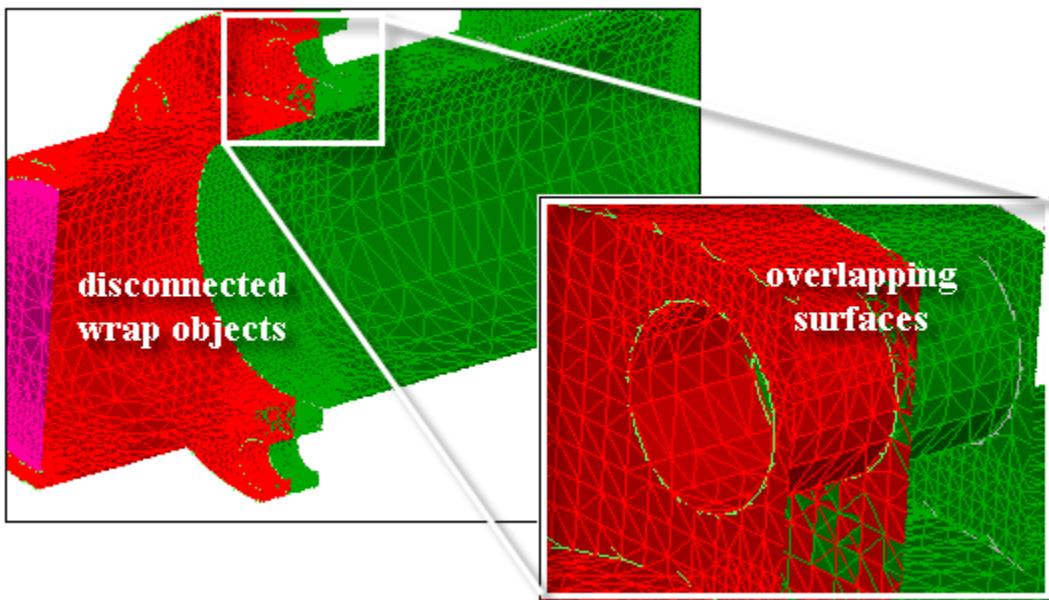
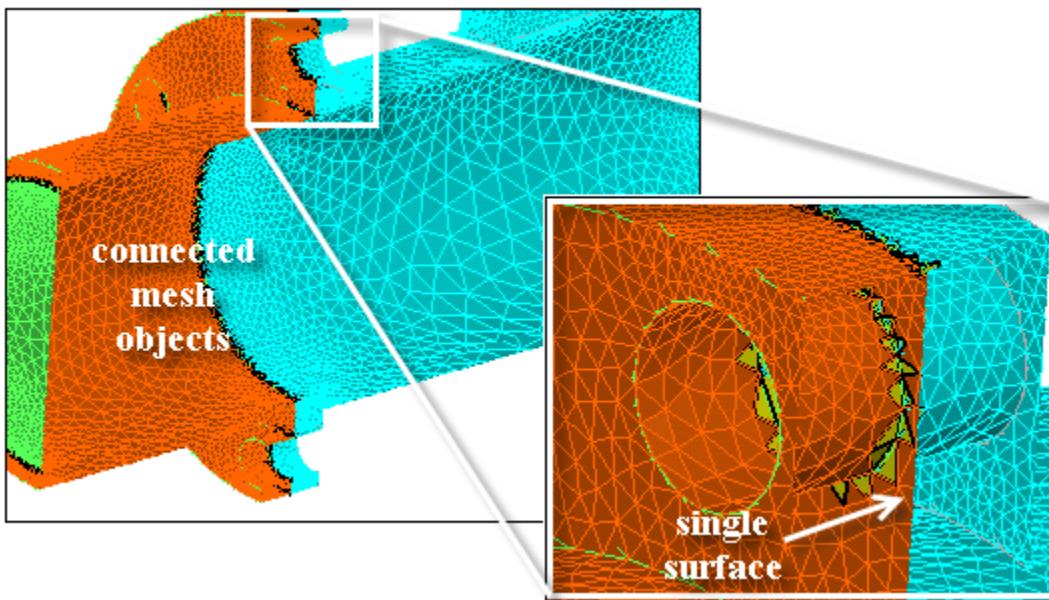
The geometry objects used to create the wrap object should be available when the `improve-feature-capture` command is invoked. Additionally, the face zones comprising the objects should be of type other than geometry.

17.7. Sewing Objects

The sewing operation is a face connecting operation of all the repaired/wrapped objects that creates a conformal, triangular surface mesh by connecting the specified wrap objects. This operation creates conformal mesh between bodies, and produces a topology-verified model. Disconnected assemblies are sewn together to create the conformal surface mesh. Normals are also reoriented suitably for further prism meshing.

You can also improve the surface mesh quality of the mesh object created by the sewing operation. This option is enabled by default. When the **Improve** option is disabled, you may need to improve the surface mesh quality of the mesh object created using the options in the [Diagnostic Tools Dialog Box \(p. 437\)](#) and [Improve Dialog Box \(p. 459\)](#).

You can select multiple wrap objects for the sew operation in the [Sew Dialog Box \(p. 457\)](#). Specify the name for the mesh object to be created for the selected wrap objects. [Figure 17.13: Wrap Objects to be Connected \(p. 336\)](#) shows an example with disconnected wrap objects. The sewing operation creates the conformal surface mesh by connecting the individual wrap objects into a single mesh object ([Figure 17.14: Mesh Object Created \(p. 336\)](#)).

Figure 17.13: Wrap Objects to be Connected**Figure 17.14: Mesh Object Created**

You can use the command `/objects/sew/sew` to connect the wrap objects. Specify the objects to be connected and the name for the mesh object to be created.

A face zone group is automatically created for the mesh object created by the **Sew** operation. This face zone group is prefixed by **_mesh_group**, and allows easy selection of mesh object face zones for various operations (improve, smooth, etc.).

Note

The sew operation is not needed for a single wrap object, or flow volume (only) extraction type problems. You can use the options in the [Diagnostic Tools Dialog Box \(p. 437\)](#) and then

use the [Improve Dialog Box \(p. 459\)](#) or the command `/objects/improve-object-quality` to improve the surface mesh (see [Improving Wrap Objects \(p. 337\)](#)).

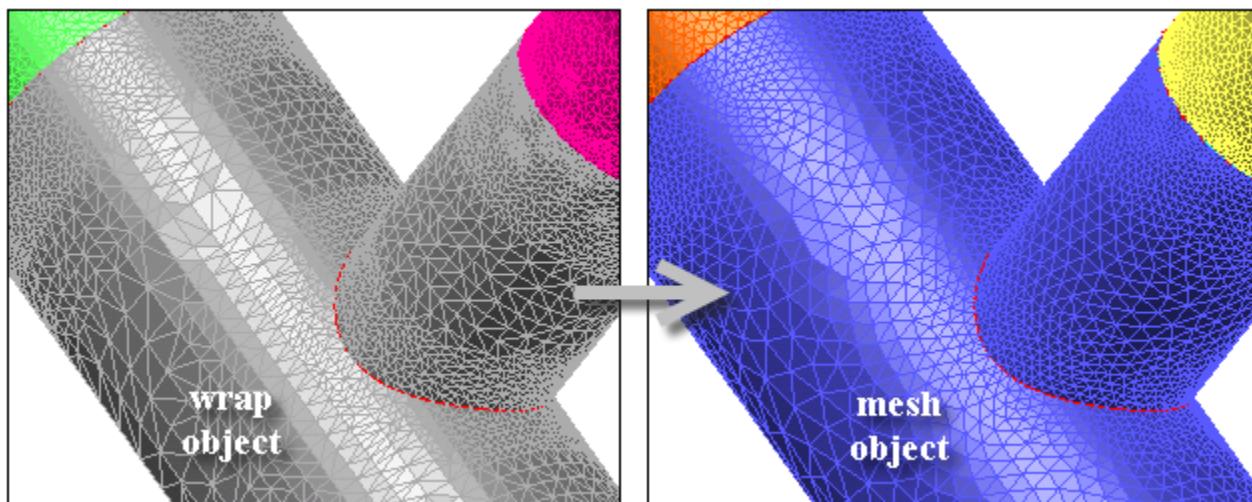
17.7.1. Improving Wrap Objects

You can use the options in the [Improve Dialog Box \(p. 459\)](#) or the command `/objects/improve-object-quality` to improve the surface mesh quality for wrap objects. When these options are used to improve wrap objects, the selected wrap objects will be converted to mesh objects, assuming that any face connectivity problems have been diagnosed and fixed. It is recommended that you use the options in the [Diagnostic Tools Dialog Box \(p. 437\)](#) before using these options to improve the wrap objects.

- The **Smooth and Improve** option improves the mesh by a combination of smoothing, swapping, and improvement operations, along with correcting the orienting of the object normals and deleting of island faces. You can optionally coarsen the surface mesh by specifying a suitable coarsening factor. Additional imprinting can be done to improve feature capture on the surface mesh.
- The **Surface Remesh** option improves the mesh by remeshing based on the current size functions/size field. Object normals are correctly oriented and island faces are also deleted.

[Figure 17.15: Improving the Surface Mesh Quality for Wrap Objects \(p. 337\)](#) shows the improved surface mesh obtained by using the options in the **Improve** dialog box for an existing wrap object.

Figure 17.15: Improving the Surface Mesh Quality for Wrap Objects



17.7.2. Resolving Thin Regions

Surfaces in close proximity constitute thin regions in the mesh. Examples of thin regions include sharp corners, trailing edge configurations, etc., which may not be recovered accurately enough during the sewing operation and surface elements may span between nodes on the proximal surfaces.

You can use the command `/objects/sew/set/include-thin-cut-edges-and-faces` to allow better recovery of such configurations during the sewing operation.

17.7.3. Processing Slits

In cases containing baffles, when the **shrink-wrap** method is used for the object wrapping operation, the wrap object is created with nearly overlapping surfaces representing the baffle. Though the surfaces

are nearly overlapping, there is a numerically small angle between them (parallel face angle). Such configurations constitute slits in the wrap object.

The command `/objects/sew/set/process-slits-as-baffles?` allows you to collapse the nearly overlapping surfaces corresponding to the baffle when the sew operation is performed to create the mesh object. Specify the maximum slit thickness relative to the minimum size specified and the parallel face angle between the faces comprising the slit when `process-slits-as-baffles` is enabled.

Note

When `process-slits-as-baffles` is enabled for the **Sew** operation, it is recommended that you check the mesh object created for voids or pockets. Use the command `/objects/merge-voids` to remove any voids or pockets created in the mesh object (see [Removing Voids \(p. 338\)](#) for details).

17.8. Improving the Mesh Objects

The options in the [Improve Dialog Box \(p. 459\)](#) allow you to improve the surface mesh quality of wrap and mesh objects. You can alternatively use the command `/objects/improve-object-quality`.

- The **Smooth and Improve** option improves the mesh by a combination of smoothing, swapping, and improvement operations, along with correcting the orientation of the object normals and deleting of island faces. You can optionally coarsen the surface mesh by specifying a suitable **Coarsening Factor**. Additional **Imprinting** can be done to improve feature capture on the surface mesh.
- The **Surface Remesh** option improves the mesh by remeshing based on the current size functions/size field. Object normals are correctly oriented and island faces are also deleted.

17.8.1. Removing Voids

The command `/objects/merge-voids` allows you to remove voids or pockets created in the mesh object.

17.8.2. Improving Feature Capture For Mesh Objects

The command `/objects/improve-feature-capture` allows you to imprint the edges comprising the object on to the object face zones to improve feature capture for mesh objects. You can specify the number of imprinting iterations to be performed.

Note

The wrap objects used to create the mesh object should be available when the `improve-feature-capture` command is invoked. Additionally, the face zones comprising the objects should be of type other than geometry.

17.9. Build Topology

The topology building operation connects mesh objects by joining overlapping faces or intersecting face zones. For best results, it is recommended that faces be of similar size where the join or intersect

operation is occurring. The process is interactive, fast, scriptable, and allows direct control over local shape and quality, and volumetric and surface overlaps.

Build Topology may be employed as a bottom-up strategy that allows you to build multiple sub-assemblies individually, and then connect the sub-assemblies to create the final assembly. It may also be used to connect all the mesh objects into the final assembly in one operation. Support for part replacement without global remeshing is inherent in the process.

For best results when using Build Topology, the following practices are highly recommended:

- Prepare clean input prior to using **Build Topology**:
 - Resolve free, duplicate and sliver faces using the **Diagnostics** tools. See [Diagnostics \(p. 323\)](#).
 - Identify self-proximity locations and separate it into different zones using the **Diagnostics** tools.
 - Remove gaps between the face zones to be joined to get clean contacts (overlaps). Alternatively, choose **Absolute Tolerance** with a value greater than or equal to the known gap if joining over smaller gaps.
 - Have similar mesh sizes (density) on face zones to be joined.
- If wrapped meshes are used with **Join**, use the **High** option together with a scaled **Size Function** to produce a very fine mesh. After Build Topology, all the surfaces may be remeshed with the default size function.

The **Build Topology** dialog box contains the controls for generating cell zone regions from mesh objects.

1. Open the [Build Topology Dialog Box](#).
2. If there are no mesh objects in the drop-down list, choose **Create New**.

The **New Mesh Object** dialog box opens.

- a. Select from the list of available wrap objects.
- b. Specify a name for the new mesh object.
- c. Click **Create**.

The new mesh object now shows up in the **Mesh Object** drop-down list in the [Build Topology Dialog Box](#).

Also, on the **Mesh Generation** task page, individual wrap object names are removed from the **Objects** list and replaced with the mesh object name.

3. Select a **Mesh Object** in the drop-down list and its **Sub Objects** are listed.

Select a **Sub Object** and its **Face Zones** are highlighted.

Note

"Sub object" represents either the wrap objects that were used to create the mesh object or, in the case of conformal faceted import with one object per part, the bodies of the part.

4. Choose **Join** or **Intersect** in the **Operation** group box.
-

Note

Always use the join operation first, then intersect.

5. In the **Parameters** group box, specify the decision thresholds for **Angle** (default is 40 degrees) and **Tolerance** (default is relative tolerance of 0.05, or 5% of local triangle size).

Check **Absolute Tolerance** to specify tolerance is in the same dimensional units as the geometry.

6. Global **Join (Intersect)**, under the **Face Zones** list, will perform the selected **Operation** on selected **Face Zones** using the specified **Parameters**.
-

Note

- The join operation separates the two overlapping face zones and keeps only one separated face zone. It is possible that the separated zone may overlap with a third zone, hence it is recommended to repeat the operation iteratively until no overlaps are found. The intersection operation does not separate the face zones, still it is a good practice to repeat the operation iteratively until no intersections are found
 - The surface should be inspected for self-intersections, duplicates, etc. after **Join** or **Intersect** operations using the **Diagnostics** tools.
 - Quads are not supported by **Join** or **Intersect**.
-

7. Use **Local** controls to perform the operations on *pairs* of overlapping or intersecting face zones.

Local is recommended for geometry where you do not know the tolerances.

Note

Length scales of geometric features should be smaller than the local triangle size.

For information on the function of each control in the Local group, see the [Build Topology Dialog Box](#).

8. Open the **Regions** tab.

- a. Select the **Material Point**, if desired.
- b. Click **Compute**.

All regions are now available and listed. The selected **Material Point** (if any) will be used to identify the region which will be named after the **Material Point** while computing.

Note

"Region" is a finite contiguous volumetric space bounded by face zones.

Note

When complete, all **Sub Objects** should appear as **Regions**. See the **Region Count**. If a **Region** is not computed correctly, it is likely caused by self-intersections and/or leaks. These problems may be found and fixed using the **Diagnostics** tools.

Warning

Regions must be recomputed if the face zones of the mesh object are modified in a way that leads to a change in region definition. For example, merging, separating or deleting face zones of a mesh object will require regions to be recomputed.

Add follows a similar sequence, but starts with one existing mesh object.

1. Select an existing mesh object from the drop down list.
2. Click **Add**.

The **Add Objects** dialog box opens.

- a. Select from the list of available mesh and wrap objects.
- b. Click **Add**.

The names of added objects appear in the **Sub Objects** list and are removed from the **Objects** list on the **Mesh Generation** task page.

17.9.1. Text commands for Build Topology

Text commands related to defining and manipulating objects are listed in [Text Commands for Objects \(p. 110\)](#). Text commands specific to **Build Topology** are as follows:

objects/build-topology

contains options for connecting overlapping and intersecting face zones.

objects/build-topology/add-objects-to-mesh-object

allows you to specify one or more wrap or mesh objects to be added to an existing mesh object.

objects/build-topology/change-region-type

allows you to select a cell zone type (solid, fluid or dead) for a specific region.

objects/build-topology/compute-regions

closed cell zone regions are computed from the specified mesh object. You may include a material point, if desired.

objects/build-topology/create-mesh-object

allows you to specify one or more wrap objects to be connected in one mesh object.

objects/build-topology/delete-region

removes a closed cell zone region and all of its face zones, except those which are shared by other regions, from the specified mesh object.

objects/build-topology/intersect

connects two crossing face zones within specified angle and tolerance.

objects/build-topology/join

connects two overlapping face zones within specified angle and tolerance.

objects/build-topology/merge-regions

specified regions are joined into a single region.

objects/build-topology/rename-region

allows you to specify a new name for a specified region.

17.10. Generating the Volume Mesh Based on Mesh Objects

The face zone group automatically created for the mesh object created by the **Sew** operation (prefixed by **_mesh_group**) allows easy segregation of the mesh object face zones from those associated with non-mesh objects. You can optionally activate this face zone group using the [User Defined Groups Dialog Box \(p. 428\)](#) to view only the mesh object face zones in other dialog boxes.

The [Auto Mesh Dialog Box](#) contains options for generating the volume mesh based on mesh objects. The volume mesh can be generated as follows:

1. Select the appropriate mesh object in the **Object** drop-down list.
2. To grow prisms for the geometry under consideration, click the **Set...** button in the **Boundary Layer Mesh** group box to open the **Prisms** dialog box.
 - a. Examine the face zone group created for the mesh object and determine the face zones for which prism meshing parameters are to be specified.
 - b. Verify that the normals are oriented in the correct direction for prism growth.
 - i. Ensure that the appropriate mesh object is selected in the **Object Name** drop-down list in the **Orient Mesh Object Face Normals** group box in the **Direction** tab.
 - ii. Select **Region** or **Material Point** and then select the appropriate region or material point in the drop-down list.

Regions may be computed using **Build Topology**, if necessary.

- iii. Ensure that **Select** is enabled and **Select Walls** and/or **Select Baffles** are enabled as required.
- iv. Click **Orient**. A new face zone group comprising the prism base zones will be created (prefixed by **_prism**).
- v. Select the group comprising the prism base zones and set the appropriate prism parameters.

- vi. Click **Apply** in the **Prisms** dialog box. Refer to [Prism Meshing Options \(p. 232\)](#) for details on the prism meshing options available.
- c. Verify that the **Prisms** check box is enabled in the **Boundary Layer Mesh** group box in the **Auto Mesh** dialog box.
- 3. Select the appropriate quad-tet transition elements from the **Quad Tet Transition** list. Click the **Set...** button to open the **Pyramids** dialog box or the **Non Conformals** dialog box (depending on the selection) and specify the appropriate parameters. Refer to [Creating Pyramids \(p. 221\)](#) and [Creating a Non-Conformal Interface \(p. 225\)](#) for details.
- 4. Select the appropriate option from the **Volume Fill** list. Click the **Set...** button to open the **Tet Dialog Box (p. 556)** or the **Hexcore Dialog Box (p. 563)** (depending on the selection). Specify the appropriate parameters. Refer to [Initializing the Tetrahedral Mesh \(p. 274\)](#), [Refining the Tetrahedral Mesh \(p. 276\)](#), and [Controlling Hexcore Parameters \(p. 287\)](#) for details.

Note

When you click the **Compute** button in the **Tet** or **Hexcore** dialog box, you will be asked if the maximum cell size is to be computed based on the mesh object selected. Click **Yes** to recompute the cell sizes based on the mesh object.

- 5. Click **Mesh** in the **Auto Mesh** dialog box.

Note

The **Merge Cell Zones** and **Auto Identify Topology** options are not available when a mesh object is selected for volume meshing.

17.11. Text Commands for Object Based Meshing

The text commands related to defining and manipulating objects are listed in [Text Commands for Objects \(p. 110\)](#). The text commands for object based meshing are as follows:

/objects/improve-feature-capture

allows you to imprint the edges comprising the object on to the object face zones to improve feature capture for wrap or mesh objects. You can specify the number of imprinting iterations to be performed.

Note

The geometry/wrap objects used to create the wrap/mesh objects should be available when the `improve-feature-capture` command is invoked. Additionally, the face zones comprising the objects should be of type other than geometry.

/objects/improve-object-quality

allows you to improve the surface mesh quality for the wrap or mesh objects specified.

Note

When this command is used to improve wrap objects, they will be converted to mesh objects, assuming that any face connectivity problems have been diagnosed and fixed.

/objects/list

lists the defined objects, indicating the respective cell zone type, priority, face zones and edge zones comprising the object, object type, and object reference point in the console.

/objects/merge-voids

allows you to merge voids in the mesh object after the sewing operation.

/objects/remove-gaps/ignore-orientation?

allows you to set whether the orientation of the normals should be taken into account while identifying the gap to be removed.

/objects/remove-gaps/remove-gaps

allows you to remove gaps between the wrap objects specified or remove the thickness in the wrap objects specified. Select the appropriate repair option and specify the other parameters required.

/objects/remove-gaps/show-gaps

marks the faces at the gap between wrap objects based on the min.gap distance, max. gap distance, and percentage margin specified.

/objects/sew/sew

connects the wrap objects to generate the conformal surface mesh.

/objects/sew/set/include-thin-cut-edges-and-faces

allows better recovery of thin region configurations during the sewing operation.

/objects/sew/set/process-slits-as-baffles?

allows you to collapse the nearly overlapping surfaces corresponding to the baffle when the sew operation is performed to create the mesh object. Specify the maximum slit thickness relative to the minimum size specified and the parallel face angle between the faces comprising the slit when process-slits-as-baffles is enabled.

/objects/sew/set/zone-name-prefix

allows you to specify a prefix for the zones included in the mesh object created using the sew operation.

/objects/wrap/check-holes

allows you to check for holes in the wrap objects. The number of hole faces marked will be reported.

/objects/wrap/recover-periodic-surfaces

allows you to make the periodic and shadow face zones on the wrap object identical. You will be prompted to identify the source and shadow face zones, and specify whether to project the face zone boundaries or not. Specify the periodicity source:

zone

enables you to specify the periodic face zone on the underlying geometry.

manual

enables you to specify the angle of rotational periodicity, and the origin and axis of rotation.

Note

This command supports only a single zone on symmetry or shadow zones. If multiple face zones exist on the symmetry or shadow side, merge them before invoking the `recover-periodic-zones` command.

Only rotational periodicity is supported, translational periodicity is not supported currently.

/objects/wrap/set/report-holes?

allows you to check for holes in the wrap object created. Holes, if any will be reported at the end of the object wrapping operation.

/objects/wrap/set/include-thin-cut-edges-and-faces

allows better recovery of thin region configurations during the object wrapping operation.

/objects/wrap/set/max-free-edges-for-hole-patching

allows you to set the maximum number of free edges in a loop to fill the holes.

/objects/wrap/set/shrink-wrap-rezone-parameters

allows you to set the parameters for improving the wrap object surface quality using rezoning. The geometry object zones will be separated based on the separation angle specified to improve the feature imprinting on the wrap object.

/objects/wrap/set/zone-name-prefix

allows you to specify a prefix for the zones included in the wrap object created using the object wrapping operation.

/objects/wrap/wrap

creates the wrap objects based on the geometry objects selected and other object wrapping parameters specified.

/mesh/cutcell/objects/improve-feature-capture

allows you to imprint the edges comprising the object on to the object face zones to improve feature capture for wrap or mesh objects. You can specify the number of imprinting iterations to be performed.

Note

The geometry/wrap objects used to create the wrap/mesh objects should be available when the `improve-feature-capture` command is invoked. Additionally, the face zones comprising the objects should be of type other than geometry.

/mesh/cutcell/objects/improve-object-quality

allows you to improve the surface mesh quality for the wrap or mesh objects specified. This command is not relevant for CutCell meshing.

/mesh/cutcell/objects/list

lists the defined objects, indicating the respective cell zone type, priority, face zones and edge zones comprising the object, object type, and object reference point in the console.

/mesh/cutcell/objects/merge-voids

allows you to merge voids in the mesh object after the sewing operation. This command is not relevant for CutCell meshing.

/mesh/cutcell/objects/remove-gaps/remove-gaps

contains options for removing gaps between wrap objects. The commands in this sub-menu are not relevant for CutCell meshing.

/mesh/cutcell/objects/sew/

connects the wrap objects to generate the conformal surface mesh. The commands under this sub-menu are not relevant for CutCell meshing.

/mesh/cutcell/objects/wrap/check-holes

allows you to check for holes in the wrap objects. The number of hole faces marked will be reported.

/mesh/cutcell/objects/wrap/recover-periodic-surfaces

allows you to make the periodic and shadow face zones on the wrap object identical. This command is not relevant for CutCell meshing.

/mesh/cutcell/objects/wrap/set/report-holes?

allows you to check for holes in the wrap object created. Holes, if any will be reported at the end of the object wrapping operation.

/mesh/cutcell/objects/wrap/set/include-thin-cut-edges-and-faces

allows better recovery of thin region configurations during the object wrapping operation.

/mesh/cutcell/objects/wrap/set/max-free-edges-for-hole-patching

allows you to set the maximum number of free edges in a loop to fill the holes.

mesh/cutcell/objects/wrap/set/shrink-wrap-rezone-parameters

allows you to set the parameters for improving the wrap object surface quality using rezoning. The geometry object zones will be separated based on the separation angle specified to improve the feature imprinting on the wrap object.

/mesh/cutcell/objects/wrap/set/zone-name-prefix

allows you to specify a prefix for the zones included in the wrap object created using the object wrapping operation.

/mesh/cutcell/objects/wrap/wrap

creates the wrap objects based on the geometry objects selected and other object wrapping parameters specified.

Chapter 18: Improving the Mesh

A volume mesh created from a high quality surface mesh may contain some high skewness cells. The poor cells may result from unsuitable mesh size distribution over the domain or more often are caused by constraints imposed by the boundaries.

After creating a tetrahedral, or hybrid mesh, you can improve the quality of the mesh by smoothing nodes and swapping faces. Smoothing and face swapping are tools that help to improve the quality of the final numerical mesh. You can also use the improve command which combines operations like collapsing cells, node smoothing, face swapping, and inserting nodes. This chapter describes the options available for improving the mesh quality by removing highly skewed cells.

[18.1. Smoothing Nodes](#)

[18.2. Swapping](#)

[18.3. Improving the Mesh](#)

[18.4. Removing Slivers from a Tetrahedral Mesh](#)

[18.5. Modifying Cells](#)

[18.6. Moving Nodes](#)

[18.7. Cavity Remeshing](#)

[18.8. Manipulating Cell Zones](#)

[18.9. Manipulating Cell Zone Conditions](#)

[18.10. Using Domains to Group and Mesh Boundary Faces](#)

[18.11. Checking the Mesh](#)

[18.12. Checking the Mesh Quality](#)

[18.13. Clearing the Mesh](#)

18.1. Smoothing Nodes

Smoothing repositions the nodes to improve the mesh quality. The smoothing methods available are:

- [Laplace smoothing](#)
- [Variational smoothing](#)
- [Skewness-based smoothing](#)

18.1.1. Laplace Smoothing

Laplace smoothing is used to improve (reduce) the average skewness of the mesh. In this method, a Laplacian smoothing operator is applied to the unstructured grid to reposition nodes. The new node position is the average of the positions of its node neighbors.

The relaxation factor (a number between 0.0 and 1.0) multiplies the computed node position increment. A value of zero results in no movement of the node and a value of unity results in movement equivalent to the entire computed increment.

This node repositioning strategy improves the skewness of the mesh, but usually relaxes the clustering of node points. In extreme circumstances, the unchecked operator may create grid lines that cross over

the boundary, creating negative cell volumes. To prevent such crossovers, the skewness of the resulting cells is checked before the node is repositioned. This makes the smoothing operation time-consuming.

The smoothing operator can also be applied repeatedly, but as the number of smoothing iterations increases, the node points have a tendency to pull away from boundaries and the mesh tends to lose any clustering characteristics.

18.1.2. Variational Smoothing

Variational smoothing is available only for tetrahedral meshes. It can be considered as a variant of Laplace smoothing. The new node position is computed as a weighted average of the circumcenters of the cells containing the node. The variational smoothing method is provided as a complement to Laplace smoothing.

18.1.3. Skewness-Based Smoothing

Skewness-based smoothing is available only for tetrahedral meshes. When you use skewness-based smoothing, a smoothing operator is applied to the mesh, repositioning interior nodes to lower the maximum skewness of the mesh. Interior nodes will be moved to improve the skewness of cells with skewness greater than the specified minimum skewness. You can also specify an appropriate value for the minimum improvement, if required. This allows you to stop performing the smoothing iterations when the maximum change in cell skewness is less than or equal to the value specified for minimum improvement.

The maximum change in cell skewness will be compared with the specified value for minimum improvement. When the maximum change in cell skewness is less than or equal to the value specified for minimum improvement, further smoothing iterations will no longer yield appreciable improvement in the mesh. The smoothing will be stopped at this point even if the requested number of iterations has not been completed.

This skewness-based smoothing process can be very time-consuming, so it is advisable to perform smoothing only on cells with high skewness. Improved results can be obtained by smoothing the nodes several times. There are internal checks that will prevent a node from being moved if moving it causes the maximum skewness to increase, but it is common for the skewness of some cells to increase when a cell with a higher skewness is being improved. Hence, you may see the average skewness increase while the maximum skewness is decreasing.

You should consider whether the improvements to the mesh due to a decrease in the maximum skewness are worth the potential increase in the average skewness. Performing smoothing only on cells with very high skewness (e.g., 0.8 or 0.9) may decrease the adverse effects on the average skewness.

18.1.4. Text Commands for Smoothing

Text commands for smoothing the grid are as follows:

/mesh/laplace-smooth-nodes

applies a Laplacian smoothing operator to the grid nodes. This command can be used for smoothing of all cell types, including prismatic cells.

/mesh/tet/improve/skew-smooth-nodes

applies skewness-based smoothing to nodes on the tetrahedral cell zones to improve the mesh quality.

/mesh/tet/improve/smooth-nodes

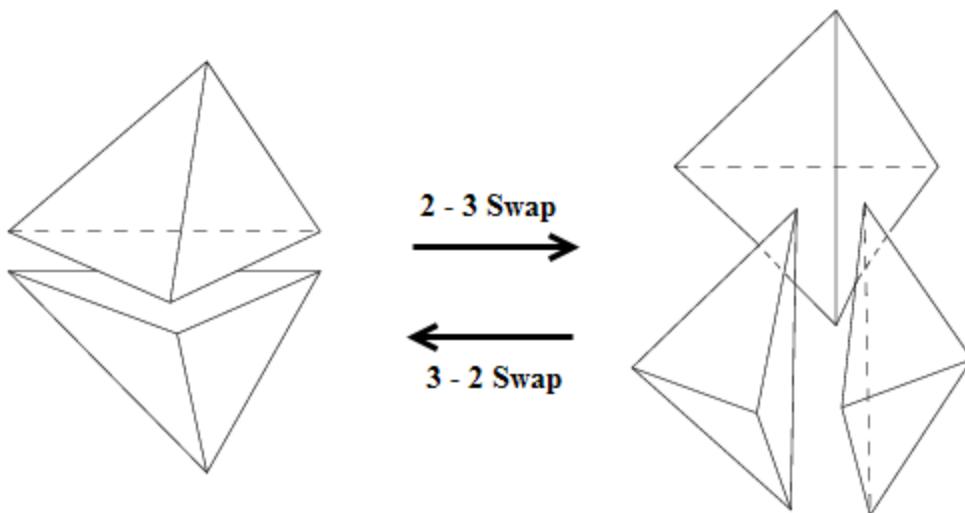
allows you to apply either Laplacian or variational smoothing to nodes on the tetrahedral cell zones to improve the mesh quality.

18.2. Swapping

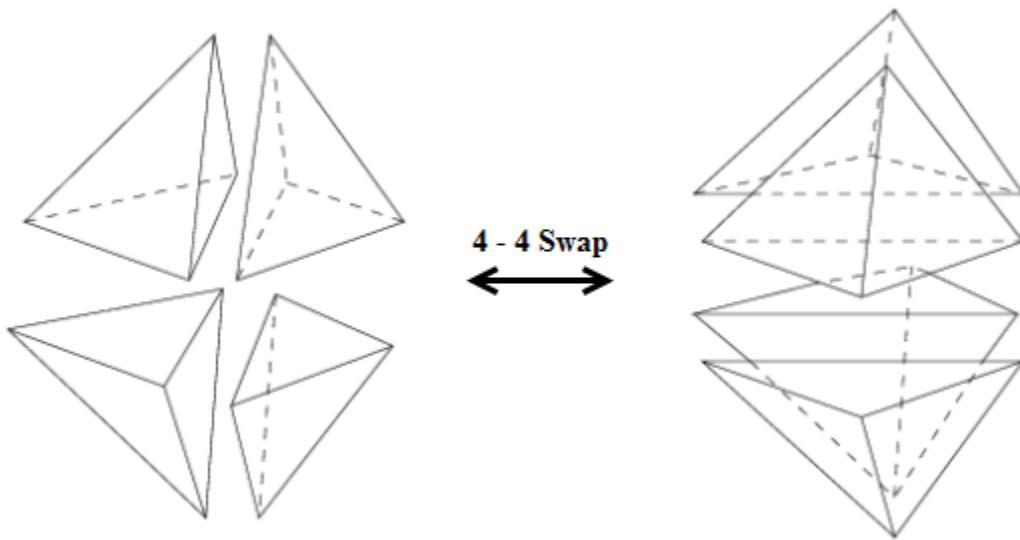
For tetrahedral meshes, swapping involves searching for a specific configuration of cells and replacing it by an alternative configuration. The default option is a 3–2 swap configuration where three tetrahedra are replaced by two tetrahedra after swapping. The other possible combinations are the 2–3 swap configuration (replacing two tetrahedra by three) and the 4–4 swap configuration (replacing four existing tetrahedra with four alternate tetrahedra).

[Figure 18.1: 2–3 and 3–2 Swap Configurations \(p. 349\)](#) shows the 2–3 and 3–2 swap configurations. The two tetrahedra (on the left) having a common interior face can be replaced by three tetrahedra having a common interior edge. The common face will be replaced by three interior faces and an interior edge during swapping. Conversely, the three tetrahedra (on the right) have two interior faces each and share a common interior edge. During swapping for a 3–2 configuration, three interior faces and the common interior edge will be replaced by a single face. This results in two tetrahedra having a common interior face.

Figure 18.1: 2–3 and 3–2 Swap Configurations



Another possible swap configuration is the 4–4 swap configuration where four tetrahedra will be replaced by four alternate tetrahedra. In [Figure 18.2: 4–4 Swap Configuration \(p. 350\)](#), either the common interior edge or two common faces can be replaced, resulting in four alternate tetrahedra.

Figure 18.2: 4-4 Swap Configuration

18.2.1. Text Commands for Swapping

The text command for swapping in the mesh is as follows:

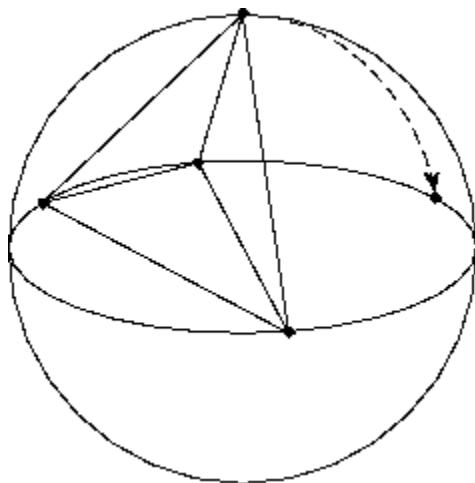
```
/mesh/tet/improve/swap-faces
    performs interior face swapping to improve cell skewness.
```

18.3. Improving the Mesh

The **Improve** operation is an automated procedure for sliver removal or for reducing the maximum skewness in the mesh. The improvement is carried out by removing cells above the specified skewness threshold by collapsing cells, swapping faces, smoothing nodes, and inserting new nodes iteratively. Each operation is specialized, and the mesh will be modified only if the mesh is noticeably improved. The skewness before and after an operation is taken into account to determine the improvement. Hence, the lower the skewness of the cells involved, the larger the improvement will have to be in order for the mesh to be modified. The improve operation is a more elaborate version of the **Remove Slivers** option invoked from the **Refinement** tab in the **Tet** dialog box.

18.4. Removing Slivers from a Tetrahedral Mesh

A sliver typically denotes a flat tetrahedral cell. Figure 18.3: Sliver Formation (p. 351) shows an acceptable tetrahedron.

Figure 18.3: Sliver Formation

If the top node of the tetrahedron were to travel along the path of the dotted line in the direction of the arrow, as it approached the end of the line the resulting cell would be a degenerate tetrahedron, or a sliver.

In the following sections, the term sliver is used to denote all types of poorly shaped cells. There are several commands for removing slivers or to reduce the maximum skewness of the mesh.

18.4.1. Automatic Sliver Removal

18.4.2. Removing Slivers Manually

18.4.3. Text Commands for Sliver Removal

18.4.1. Automatic Sliver Removal

Sliver removal operations can be invoked during the tetrahedral mesh refinement process. The **Remove Slivers** option in the **Refinement** tab of the **Tet** dialog box controls the removal of slivers during the meshing process. The **Remove Slivers** option comprises operations such as collapsing cells (to remove nodes), face swapping, smoothing, and point insertion, which are invoked iteratively.

Usually the sliver cells will be removed during refinement, but occasionally a few might be left behind. In such cases, you can use the options in the **Slivers** tab in the **Tet Improve** dialog box to remove the slivers manually. The **Improve** command uses an automated procedure to remove slivers or to improve the mesh quality in general.

18.4.2. Removing Slivers Manually

The **Tet Improve** dialog box contains options for removing slivers manually. The operations available for sliver removal are as follows:

Smoothing Boundary Slivers

This operation involves smoothing nodes on sliver cells having at least one node on the boundary. During smoothing, the nodes will be repositioned so long as the skewness of the surrounding cells is improved. The nodes on features will also be smoothed, but will not be projected on to the original geometry. However, nodes at branch points (more than two feature edges at the node) and end points (one feature edge at the node) will be fixed. The nodes will be smoothed until the skewness value is less than the specified value. The default values for the skewness threshold, minimum dihedral angle between boundary faces, and feature angle are 0.985, 10, and 30, respectively.

Smoothing Interior Slivers

This operation involves smoothing non-boundary nodes on sliver cells having skewness greater than the specified threshold value. The default value for the skewness threshold is 0 . 985.

Swapping Boundary Slivers

A flat boundary cell containing two boundary faces can be removed by moving the boundary to exclude the cell from the zone in which it is located, effecting a minor change in the geometry. However, if there is another live zone on the other side of the boundary, this operation will result in the cell being moved to the other zone. In such cases (e.g., conjugate heat transfer problems), you can decide which live zone is least critical, and then move the boundary sliver to that zone.

The default values for the skewness threshold and the minimum dihedral angle between faces are 0 . 95 and 10, respectively.

Refining Boundary Slivers

This operation attempts to increase the volume of boundary slivers to create a valid tetrahedral cell. Tetrahedra having one or two faces on the boundary are identified and then the edge opposite the boundary face(s) is split. The edge opposite the face pair with the largest dihedral angle will be split for a tetrahedron with one boundary face, while the edge opposite the boundary faces will be split for a tetrahedron having two boundary faces. The split node is then smoothed such that the volume of the tetrahedron increases, thereby creating a valid tetrahedral cell.

Refining Interior Slivers

This operation attempts to remove the sliver by placing a node at or near the centroid of the sliver cell. Swapping and smoothing are then performed to improve the skewness.

Collapsing Slivers

This operation attempts to collapse the edge of a skewed sliver cell on any one of its neighbors. The default skewness threshold is 0 . 985.

Notes:

- If you are not using a two-sided wall condition for the boundary, you can slit the face zone containing the sliver (using the /boundary/slit-boundary-face command) and then perform the sliver removal operation.
- Multiple slivers may exist on top of each other thus, requiring multiple operations to remove them all.

18.4.3. Text Commands for Sliver Removal

The text interface commands for removing boundary slivers are:

/mesh/tet/improve/collapse-slivers

attempts to collapse the nodes of a skewed sliver cell on any one of its neighbors.

/mesh/tet/improve/refine-boundary-slivers

attempts to increase the volume of boundary slivers to create a valid tetrahedral cell. Tetrahedra having one or two faces on the boundary are identified and then the appropriate edge is split. The split node

is then smoothed such that the volume of the tetrahedron increases, thereby creating a valid tetrahedral cell.

/mesh/tet/improve/refine-slivers

attempts to remove the sliver by placing a node at or near the centroid of the sliver cell. Swapping and smoothing are then performed to improve the skewness. You can also specify whether boundary cells are to be refined. Refining the boundary cells may allow you to carry out further improvement options such as smoothing, swapping, and collapsing slivers.

/mesh/tet/improve/sliver-boundary-swap

removes boundary slivers by moving the boundary to exclude the cells from the zone.

/mesh/tet/improve/smooth-boundary-sliver

smooths nodes on sliver cells having all four nodes on the boundary until the skewness value is less than the specified value. The default values for the skewness threshold, minimum dihedral angle between boundary faces, and feature angle are 0.985, 10, and 30, respectively.

/mesh/tet/improve/smooth-interior-sliver

smooths non-boundary nodes on sliver cells having skewness greater than the specified threshold value. The default value for the skewness threshold is 0.985.

18.5. Modifying Cells

Additional tools are available for you to perform primitive operations on the cells such as smoothing nodes, swapping cells, splitting cells, etc. This section describes the generic procedure for modifying the cells using the **Modify Cells** dialog box. You will also use the **Display Grid** dialog box during the modification of the cells.

18.5.1. Using the Modify Cells Dialog Box

1. Display the cell(s) or cell zone(s) to be modified using the options in the **Display Grid** dialog box.
2. Select the type of entity (**cell**, **face**, **node**, etc.) you want in the **Filter** list in the **Modify Cells** dialog box.
3. Select the objects to be modified in the graphics window.
4. Click the appropriate button in the **Operation** group box.
5. Repeat the procedure to perform different operations on the cells.

Warning

Save the mesh periodically since not all operations are reversible.

The operations available in the **Modify Cells** dialog box for modifying cells are as follows:

Smoothing Nodes

During node smoothing, the selected node will be repositioned based on the average of the surrounding nodes. Select the node(s) to be smoothed and click **Smooth** in the **Operation** group box to smooth nodes.

Splitting Cells

During splitting, the selected cell will be refined by the addition of a node at the centroid of the cell.

Moving Nodes

You can move the selected node either to a specified position or by a specified magnitude.

Do the following to move a node to a particular position:

1. Select **node** in the **Filter** list and select the node to be moved.
2. Select **position** in the **Filter** list and select the appropriate position.
3. Click **Move To** in the **Operation** group box.

Do the following to move a node by a specified magnitude:

1. Select **node** in the **Filter** list and select the node to be moved.
2. Enter the magnitude by which you want to move the node in the **Enter Selection** field and press **Enter**.
The increment will now be selected in the **Selections** list.
3. Click **Move By** in the **Operation** group box.

Swapping Cells

You can perform either a 3–2 configuration swap or a 2–3 configuration swap. Refer to [Swapping \(p. 349\)](#) for details on the swap configurations. Do the following to swap cells:

1. Select the appropriate option in the **Filter** list and select the entities to be swapped.
 - Select 3 cells or the common edge in order to perform a 3–2 swap.
 - Select 2 cells or the common face in order to perform a 2–3 swap.
2. Click **Swap** in the **Operation** group box.

Determining the Coordinates of the Centroid

Do the following to determine the centroid:

1. Select the appropriate option in the **Filter** list (**cell**, **face**, **edge**, or **node**) and select the required entity.
2. Click **Centroid** in the **Operation** group box.

The centroid coordinates will be printed in the console.

Determining the Distance Between Entities

Do the following to determine the distance between entities:

1. Select the appropriate option in the **Filter** list and select the two entities between which the distance is to be determined.
2. Click **Distance** in the **Operation** group box.

The distance between the selected entities will be printed in the console.

Projecting Nodes

You can project nodes onto a specified projection line or plane. Do the following to project nodes:

1. Define the projection line or projection plane, as appropriate.
 - a. Select the appropriate option in the **Filter** list.
 - b. Select two entities to define the projection line or three entities to define the projection plane.
 - c. Click **Set** in the **Operation** group box to define the projection line or plane.

The line coordinates or plane position will be reported in the console.

2. Select the node(s) to be projected.

The projection line/plane will be highlighted in the display window.

3. Click **Project** in the **Operation** group box.

18.5.2. Text Commands for Modifying Cell Zones

The text interface commands for modifying cell zones are as follows:

/mesh/modify/clear-selections
deselects the selected entities.

/mesh/modify/deselect-last
deselects the last selected entity.

/mesh/modify/extract-unused-nodes
extracts the unused nodes into a separate interior node zone.

/mesh/modify/list-selections
lists the selected entities in the console.

/mesh/modify/list-skewed-cells
lists the cells which are within the specified skewness limits.

/mesh/modify/mesh-node
introduces a new node into the existing mesh.

/mesh/modify/mesh-nodes-on-zone

inserts the nodes associated with a particular face or node thread into the volume mesh.

Note

For a specified face thread, the faces will be deleted before introducing the nodes into the mesh.

/mesh/modify/neighborhood-skew

reports the skewness of all cells using the specified node.

/mesh/modify/refine-cell

refines the cells specified.

/mesh/modify/select-entity

selects the specified entity.

/mesh/modify/smooth-node

performs Laplace smoothing on the specified nodes.

18.6. Moving Nodes

Highly skewed meshes can be improved by moving the nodes of the cells. Moving nodes manually is a time consuming process. The node movement process for improving the mesh quality is automated. You can specify quality improvement based on the quality measure specified or based on the warp (quadrilateral elements). You can also choose between the automatic correction procedure and the semiautomatic correction procedure for the quality-based correction. You can also repair negative volume cells by moving nodes using the automatic correction procedure.

18.6.1. Automatic Correction

18.6.2. Semi-Automatic Correction

18.6.3. Repairing Negative Volume Cells

18.6.4. Text Commands for Moving Nodes

18.6.1. Automatic Correction

The automatic correction allows you to improve all the cells in the selected cell zone based on the specified criteria. You can also improve the cells based on the warp values.

Quality-Based Improvement

For the quality-based correction, you can specify the quality limit based on the quality measure selected, dihedral angle, and the number of iterations per node to be moved. The cells which have a quality above the specified threshold limit will be selected.

For boundary nodes, you can restrict the node movement in the plane containing each of the boundary faces sharing the node being moved. Nodes on sharp features and free edges will not be considered for movement.

The node to be moved for a particular cell will be selected based on the selection of zones in the **Boundary Zones** selection list and an alternative position for the node will be determined. The node will be moved to the new position only if the quality of the cell and its neighbors is improved by the change in node position. The procedure is repeated for the specified iterations per node. You can also

set the number of repetitions through the automatic correction procedure as required. By default, the correction procedure is performed only once.

Warp-Based Improvement

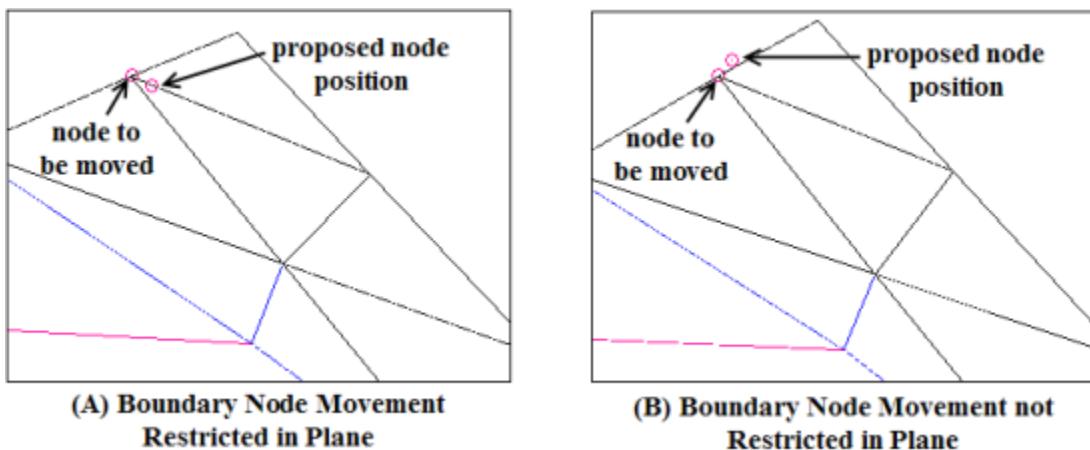
For the warp-based correction, you can specify the maximum warp and the number of iterations per face to be improved. The quadrilateral faces which have a warp value greater than the specified maximum warp will be selected. The ideal position for the node to be moved will be determined based on the remaining three nodes. The node will be moved to the new position only if the warp of the face decreases and the cell quality does not deteriorate by the change in node position. The procedure is repeated for the specified iterations per face. You can also set the number of repetitions through the automatic correction procedure as required. By default, the correction procedure is performed four times.

18.6.2. Semi-Automatic Correction

The semi-automatic correction is available only for quality-based improvement. The generic procedure for using the semiautomatic correction is as follows:

1. Select the appropriate zones in the **Cell Zones** drop-down list and the **Boundary Zones** selection list.
2. Select the quality measure and specify values for **Quality Limit**, **Iterations/Node**, and **Dihedral Angle** as appropriate.
3. Enable **Restrict Boundary Nodes Along Surface** if required. When this option is enabled, the movement of the boundary node will be limited to the plane containing the boundary faces sharing the boundary node (see [Figure 18.4: Movement of Boundary Nodes \(p. 357\)](#)).

Figure 18.4: Movement of Boundary Nodes



4. Click **Skew** to display the cell with the worst quality depending on the quality limit specified. The cell having the worst quality and cells/faces within a pre-defined range of the cell will be displayed.
5. Click **Propose**. The node to be moved and the alternative position will be highlighted. The improvement in the skewness will also be reported in the console.
6. Click **Accept** if the proposed position is appropriate. Else, click **Refuse** and then **Propose** to obtain the next suggestion.

7. Click **Next Skew** to proceed with the node correction for the next cell.

18.6.3. Repairing Negative Volume Cells

You can also repair negative volume cells by moving nodes. Specify the appropriate boundary zones, dihedral angle, the number of iterations per node to be moved and the number of iterations of the automatic node movement procedure (default, 1). You can also choose to restrict the movement of boundary nodes along the surface.

18.6.4. Text Commands for Moving Nodes

The text commands for improving the mesh by moving nodes are as follows:

/mesh/modify/auto-node-move

allows you to improve the mesh quality by node movement. Specify the appropriate cell zones and boundary zones, the quality limit based on the quality measure selected, dihedral angle, the number of iterations per node to be moved and the number of iterations of the automatic node movement procedure (default, 1). You can also choose to restrict the movement of boundary nodes along the surface.

/mesh/modify/auto-improve-warp

allows you to improve face warp by node movement. Specify the appropriate cell zones and boundary zones, the maximum warp, the number of iterations per face to be improved, and the number of iterations of the automatic node movement procedure (default, 4).

/mesh/modify/repair-negative-volume-cells

allows you to improve negative volume cells by node movement. Specify the appropriate boundary zones, dihedral angle, the number of iterations per node to be moved and the number of iterations of the automatic node movement procedure (default, 1). You can also choose to restrict the movement of boundary nodes along the surface.

18.7. Cavity Remeshing

Cavity remeshing is useful in parametric studies since it allows you add, remove, and replace different parts of the existing mesh. You can compare alternative designs by creating a cavity around the object to be replaced and then, inserting the new object and connecting it to the existing mesh. Prisms can be grown using appropriate parameters and the cavity can be filled with cells. You can also improve the quality of the volume mesh by creating an appropriate cavity around the skewed cells and remeshing it.

You can create a cavity in an existing mesh by removing cells in a defined bounded region. The cells intersecting the bounded region will be marked and the cavity boundaries will be extracted from the marked cells. The marked cells will then be deleted to create the cavity.

The various options available are:

- Removing zones: This option allows you to specify zones to be removed from the existing volume mesh.
- Adding zones: This option allows you to add new zones to the existing volume mesh.
- Replacing zones: This option allows you to remove a set of zones and replace them with a new set of zones.
- Improving a region: This option allows you to define a cavity around skewed cells in the existing mesh. You can modify the mesh as appropriate and then remesh the cavity.

The following sections describe the cavity remeshing options:

- 18.7.1.Tetrahedral Cavity Remeshing
- 18.7.2.Hexcore Cavity Remeshing
- 18.7.3.Text Commands for Cavity Remeshing

18.7.1.Tetrahedral Cavity Remeshing

The generic procedure for remeshing a cavity with tetrahedra using the **Cavity Remesh** dialog box is as follows:

1. Select the appropriate zones from the **Remove Boundary Zones** and **Add Boundary Zones** selection lists.
 - **Removing Zones** : Select the zone(s) to be removed in the **Remove Boundary Zones** selection list. Make sure that no zones have been selected in the **Add Boundary Zones** list.
 - **Adding Zones** : Select the zone(s) to be added in the **Add Boundary Zones** selection list. Make sure that no zones have been selected in the **Remove Boundary Zones** list.
 - **Replacing Zones** : Select the zone(s) to be removed in the **Remove Boundary Zones** selection list and the zone(s) to be added in the **Add Boundary Zones** selection list.
 - **Improving a Region** : Make sure that no zones have been selected in the **Remove Boundary Zones** and **Add Boundary Zones** selection lists.
2. Enable **Create Face Group** if you want to create a user-defined group (UDG) comprising the zones defining the cavity domain.

The **cavity** UDG and the corresponding **cavity** domain will be created, but the **global** domain will be retained as active when the **Create Face Group** option is enabled. When this option is disabled, the **cavity** domain will be activated when the cavity is created.

Important

The **Create Face Group** option is useful when using the cavity remeshing feature for large cases. For such cases, you can avoid frequent switching between domains by enabling the **Create Face Group** option when creating the cavity. The basic procedure is as follows:

- a. Create a UDG for the cavity and save the boundary mesh for the cavity group defined using the **File/Write/Boundaries...** option (see [Writing Boundary Mesh Files \(p. 63\)](#)).
- b. Read this boundary mesh in a separate session and create the volume mesh in the cavity as appropriate.
- c. Save the volume mesh and read it back into the previous session using the **Append File(s)** option.
- d. Connect the meshed cavity to the parent mesh and merge the cavity domain with the parent domain.

3. Enter an appropriate value for **Scale** and click **Compute**. The extents of the bounding box will be computed (based on the zones selected in the **Remove Boundary Zones** and **Add Boundary Zones** selection

lists, and the scale factor specified) and reported in the **Cavity Remesh** dialog box. Alternatively, you can specify the extents of the bounding box as required.

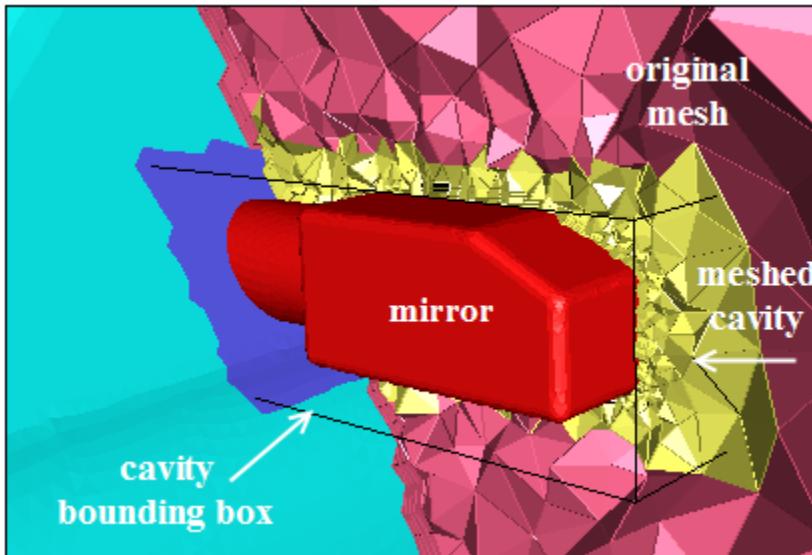
Warning

You need to manually specify the extents of the bounding box when using the cavity remeshing feature to improve a region of skewed cells.

4. Specify the orientation of the bounding box in the **Orient** group box.
5. Click **Draw** to preview the cavity domain to be created.
6. Click **Create** to create the cavity domain. The boundary zones touching the cavity bounding box will be separated and included in the cavity domain along with the new zone(s) to be added. Any zone(s) to be removed will not be included in the cavity domain. The existing volume mesh in the cavity will be removed and boundary zone(s) extracted from the interior zone(s) (if any) will be changed to **internal** type and included in the cavity domain.
7. Connect the new zone(s) with the boundaries of the cavity domain using node merging or intersect operations when removing, adding, or replacing zones in the volume mesh. Refer to [Manipulating Boundary Nodes \(p. 119\)](#) and [Intersecting Boundary Zones \(p. 121\)](#) for details. Modify the boundary mesh (if required) when improving a region in the volume mesh. Refer to [Manipulating the Boundary Mesh \(p. 119\)](#) for details on the various mesh improvement options.
8. Activate the cavity domain and create the volume mesh as appropriate.

[Figure 18.5: Cavity Around a Mirror Remeshed With Tetrahedra \(p. 360\)](#) shows a cavity created around the rear-view mirror of a car. You can see the original mesh and the cavity created around the mirror and remeshed with tetrahedra. The bounding box defined for the cavity is also shown.

Figure 18.5: Cavity Around a Mirror Remeshed With Tetrahedra



9. Activate the global domain and delete the boundary zone(s) which are no longer required. Merge the cavity domain with the parent domain using the command `/mesh/cavity/merge-cavity`.

During the merging operation, the cavity cell zone(s) will be merged with the cell zone(s) in the parent domain. The boundaries extracted from the interior zones will be converted to **interior** type and merged with the corresponding zone(s) in the parent domain. Other boundary zones included in the cavity domain will be merged with the parent face zones.

18.7.2. Hexcore Cavity Remeshing

The generic procedure for remeshing a cavity with a hexcore mesh using the **Cavity Remesh** dialog box is as follows:

1. Select the appropriate zones from the **Remove Boundary Zones** and **Add Boundary Zones** selection lists.
 - **Removing Zones** : Select the zone(s) to be removed in the **Remove Boundary Zones** selection list. Make sure that no zones have been selected in the **Add Boundary Zones** list.
 - **Adding Zones** : Select the zone(s) to be added in the **Add Boundary Zones** selection list. Make sure that no zones have been selected in the **Remove Boundary Zones** list.
 - **Replacing Zones** : Select the zone(s) to be removed in the **Remove Boundary Zones** selection list and the zone(s) to be added in the **Add Boundary Zones** selection list.
 - **Improving a Region** : Make sure that no zones have been selected in the **Remove Boundary Zones** and **Add Boundary Zones** selection lists.

Note

- Hexcore cavity remeshing is not supported for baffles/dangling walls.
 - Hexcore cavity remeshing is not supported if the mesh contains any dead cell zones.
-

2. Enable **Hexcore Cavity** to remesh the cavity with hexcore mesh.

Note

If this option is disabled, only the tetrahedral cells in the cavity will be replaced during the cavity remeshing.

3. Enter an appropriate value for **Scale** and click **Compute**. The extents of the bounding box will be computed (based on the zones selected in the **Remove Boundary Zones** and **Add Boundary Zones** selection

lists, and the scale factor specified) and reported in the **Cavity Remesh** dialog box. Alternatively, you can specify the extents of the bounding box as required.

Warning

You need to manually specify the extents of the bounding box when using the cavity remeshing feature to improve a region of skewed cells.

Note

The **Orient** group box will be disabled when the **Hexcore Cavity** option is enabled.

4. Click **Draw** to preview the cavity domain to be created.

Note

Any dead cells which intersect the cavity bounding box should be converted to fluid or solid type.

5. Click **Create** to create the cavity domain. The boundary zones touching the cavity bounding box will be separated and included in the cavity domain along with the new zone(s) to be added. Any zone(s) to be removed will be deleted from the global domain. The existing volume mesh in the cavity will be removed and boundary zone(s) extracted from the interior zone(s) (if any) will be changed to **internal** type and included in the cavity domain.
6. Connect the new zone(s) with the boundaries of the cavity domain using node merging or intersect operations when removing, adding, or replacing zones in the volume mesh. Ensure that the new boundaries are properly connected with the existing boundaries. Refer to [Manipulating Boundary Nodes \(p. 119\)](#) and [Intersecting Boundary Zones \(p. 121\)](#) for details. Modify the boundary mesh (if required) when improving a region in the volume mesh. Refer to [Manipulating the Boundary Mesh \(p. 119\)](#) for details on the various mesh improvement options.
7. If required, create the prism layers as appropriate.

Important

To facilitate easier scripting of design changes in Hexcore meshes with prism layers using the Cavity Remeshing utility, the prism base boundary zones separated during the cavity creation are suffixed by **_cavity-prism:#**. This allows you to easily identify the zones to be assigned prism growth settings while remeshing.

8. Click **Remesh** to create the volume mesh in the cavity.

During the remeshing operation, the cavity cell zone(s) will be merged with the cell zone(s) in the parent domain. The boundaries extracted from the interior zones will be converted to **internal** type and merged with the corresponding zone(s) in the parent domain. If the remeshing involves hexcore

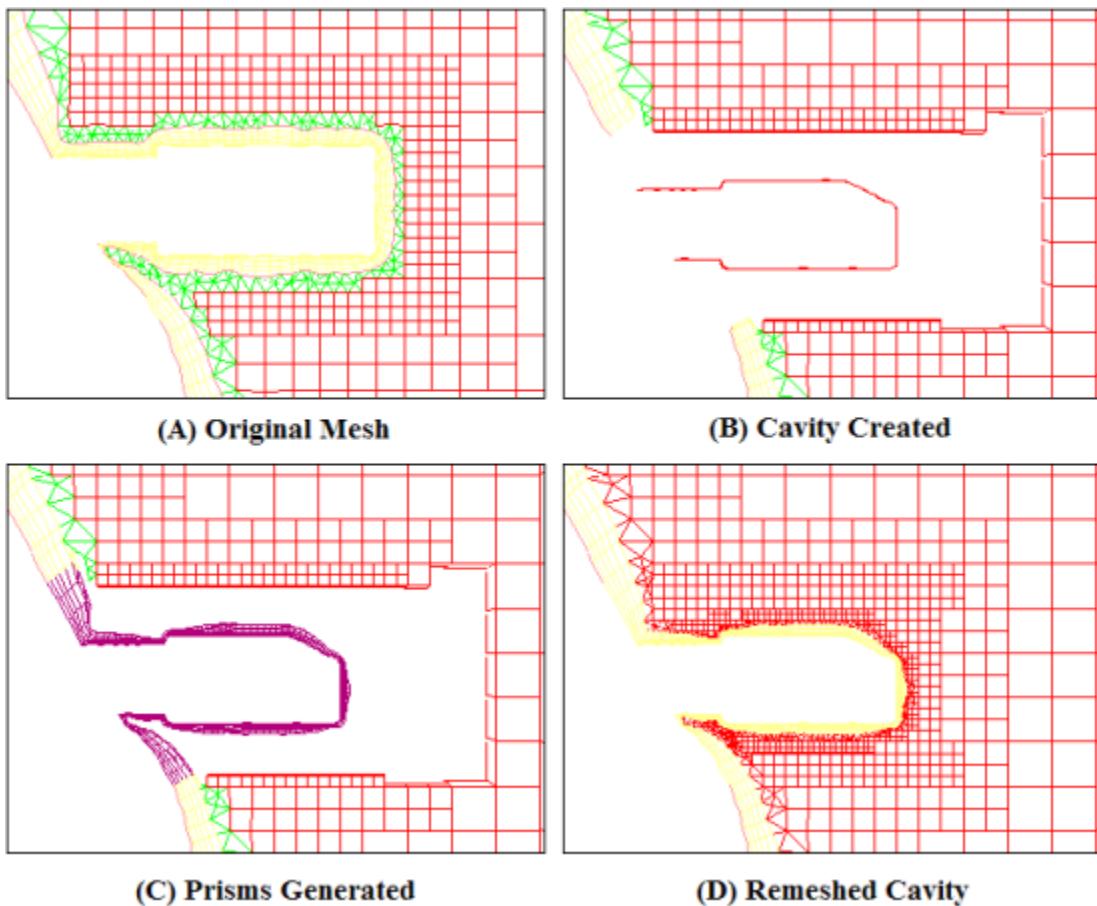
mesh generated to selected boundaries, the old boundaries will be removed and replaced during the remeshing operation.

Note

Do not change domains between the **Create** and **Remesh** operations.

Figure 18.6: Cavity Around a Mirror Remeshed With Hexcore Mesh (p. 363) shows the procedure for creating a cavity to replace the rear-view mirror of a car. You can see the original mesh and the cavity created around the mirror. Prisms are then generated and the cavity is remeshed with hexcore mesh.

Figure 18.6: Cavity Around a Mirror Remeshed With Hexcore Mesh



18.7.3. Text Commands for Cavity Remeshing

The text interface commands for cavity remeshing are as follows:

/mesh/cavity/add-zones

allows you to create a cavity for adding new zone(s) to the existing volume mesh.

/mesh/cavity/create-hexcore-cavity-by-region

creates the cavity in the hexcore mesh based on the zones and bounding box extents specified.

/mesh/cavity/create-hexcore-cavity-by-scale

creates the cavity in the hexcore mesh based on the zones and scale specified.

/mesh/cavity/merge-cavity

merges the specified tetrahedral cavity domain with the specified parent domain.

/mesh/cavity/region

allows you to create a cavity to improve the existing volume mesh in the specified region.

/mesh/cavity/remesh-hexcore-cavity

remeshes the cavity in the hexcore mesh with hexcore mesh.

/mesh/cavity/remove-zones

allows you to create a cavity for removing zone(s) from an existing volume mesh.

/mesh/cavity/replace-zones

allows you create a cavity for removing a set of zones from an existing volume mesh and replacing them with a new set of zones.

18.8. Manipulating Cell Zones

When a volume mesh is created (of any cell shape) from a boundary mesh, these cells will be grouped into cell zones (contiguous zones separated by boundaries). You can manipulate these zones to control further mesh generation or to duplicate an existing volume mesh to model a repeated geometry.

18.8.1. Active Zones and Cell Types

18.8.2. Copying and Moving Cell Zones

18.8.3. Text Commands for Manipulating Cell Zones

18.8.1. Active Zones and Cell Types

After the initial mesh is generated, all the cells are grouped into contiguous zones separated by boundaries. An artifact of the meshing algorithm is that a virtual zone is created outside the outer boundary, and it is always given a cell type of dead. This zone is automatically deleted upon completion of the initial mesh generation. If the initial mesh generation is interrupted for some reason, this zone will remain in the mesh until the initialization is completed. Other zone types available are fluid and solid.

The zone just inside the outer boundary is automatically set to be active and labeled a fluid zone, although you can change this type later. When refining the mesh, only the active zones are refined. By toggling the zones between active and inactive, you can refine different groups of zones independently, using different mesh parameters for the different groups.

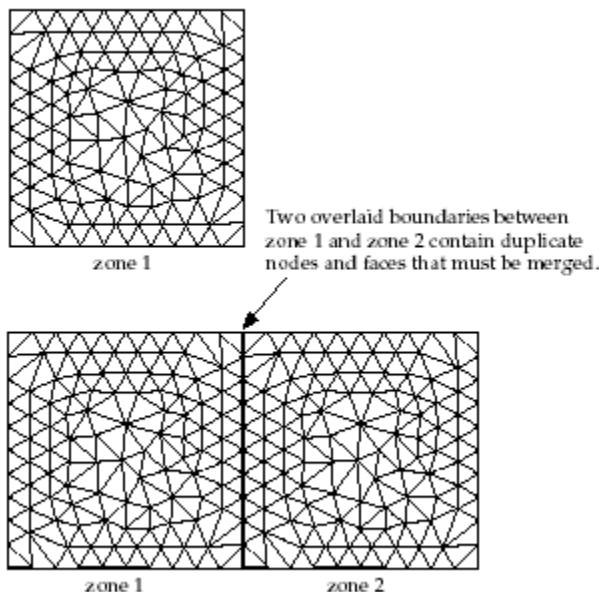
If you plan to refine all cell zones using the same refinement parameters, change the **Non-Fluid Type** in the **Tet** dialog box before initializing the mesh. If you change the **Non-Fluid Type** to any type other than **dead**, all zones will be set active automatically after the initialization occurs. This eliminates the need for you to set all zones to be active in the **Cell Zones** dialog box. See also the automatic activation method available through the text interface, described in [Text Commands for Manipulating Cell Zones \(p. 365\)](#).

18.8.2. Copying and Moving Cell Zones

If you are creating a mesh for a geometry that repeats periodically, you can simplify the meshing tasks. To do this, create the boundary and volume mesh for just one of the repeated sections. Copy the appropriate cell zone(s) to the required location(s). If the copy shares a boundary with the original zone (i.e., if the two zones are connected), ensure that the distribution of nodes is the same on the two overlaid boundaries.

A simplified case is illustrated in [Figure 18.7: Copying and Translating a Cell Zone \(p. 365\)](#). Here, the volume mesh was created for zone 1, and then copied and translated to create zone 2. The node distribution on the left boundary of zones 1 and 2 is the same as the distribution on the right boundary. Since the left boundary of zone 2 is overlaid on the right boundary of zone 1, there will be duplicate nodes. It is important that you merge these duplicate nodes.

Figure 18.7: Copying and Translating a Cell Zone



The procedure for doing this is as follows:

1. Open the **Merge Boundary Nodes** dialog box.
2. Compare all nodes on both boundaries. To do this, select the two zones in both the **Compare...** and **With...** group boxes.
3. Disable **Only Free Nodes** for both the zones.
4. Click the **Merge** button to merge the duplicate nodes.

After the duplicate nodes on the two boundaries have been merged, one of the two boundary face zones will be deleted automatically. Since duplicate faces are merged when the duplicate nodes are merged, one zone will no longer have any faces.

18.8.3. Text Commands for Manipulating Cell Zones

Text interface commands for manipulating cell zones are:

```
/mesh/manage/active-list
lists all active zones.
```

```
/mesh/manage/adjacent-face-zones
lists all face zones that refer to the specified cell zone.
```

/mesh/manage/auto-set-active

sets the active zone(s) based on points that are defined in an external file. For each zone you want to activate, specify the coordinates of a point in the zone, the zone type (e.g., fluid), and a new name. This command is valid only for tetrahedral meshes.

A sample file is as follows:

```
((1550.50 -466.58 896.41) fluid heater-#)
((1535.83 -643.14 874.71) fluid below-heater-#)
((1538.73 -444.28 952.69) fluid above-heater-#)
((1389.18 -775.51 825.97) fluid plenum-#)
```

Here, four fluid zones are identified, renamed, and activated. Any zone that you identify in the file will automatically be activated. The # (hash sign) indicates that the appropriate ID number for the zone should be appended.

/mesh/manage/change-prefix

allows you to change the prefix of the cell zone.

/mesh/manage/copy

copies all nodes and faces of specified cell zones.

/mesh/manage/delete

deletes a cell zone, along with its associated nodes and faces.

/mesh/manage/get-material-point

prints the coordinates of the material point for the specified cell zone.

Note

If the cell zone is non-contiguous, the get-material-point command will print a list of material points, one for each contiguous region.

/mesh/manage/id

specifies a new cell zone ID. If a conflict is detected, the change is ignored.

/mesh/manage/list

prints information on all cell zones.

/mesh/manage/merge

merges two or more cell zones.

/mesh/manage/merge-dead-zones

allows you to merge dead zones having a cell count lower than the specified threshold value, with the adjacent cell zone. The result of the merge operation is determined by the type of the adjacent cell zone and the shared face area. The priority for merging with the adjacent cell zone based on type is fluid > solid > dead (i.e., merging with an adjacent fluid zone takes priority over merging with an adjacent solid zone, which in turn takes priority over merging with a dead zone). Also, if the adjacent zones are of the same type (e.g., fluid), the zone will be merged with the zone having the largest shared face area.

/mesh/manage/name

allows you to rename a cell zone.

/mesh/manage/origin

specifies a new origin for the mesh, to be used for cell zone rotation. The default origin is (0,0,0).

/mesh/manage/revolve-face-zone
generates cells by revolving a face thread.

/mesh/manage/rotate
rotates all nodes of specified cell zones by a specified angle.

/mesh/manage/scale
scales all nodes of specified cell zones by a specified factor.

/mesh/manage/set-active
sets the specified cell zones to be active.

/mesh/manage/translate
translates all nodes of specified cell zones by a specified vector.

Note

The translation offsets are interpreted as absolute numbers in meshing mode. In solution mode, however, the translation offsets are assumed to be distances in the length unit set. This may lead to differences in domain extents reported after translating the mesh in the respective modes.

/mesh/manage/type
allows you to changes the type of the cell zone.

/mesh/separate/local-regions/define
allows you to define the local region.

/mesh/separate/local-regions/delete
deletes the specified local region.

/mesh/separate/local-regions/init
creates a region encompassing the entire geometry.

/mesh/separate/local-regions/list-all-regions
lists all the local regions defined.

/mesh/separate/separate-cell-by-face
separates cells that are connected to a specified face zone into another cell zone. This separation method applies only to prism cells.

/mesh/separate/separate-cell-by-mark
separates cells within a specified local region into another cell zone.

/mesh/separate/separate-cell-by-region
separates contiguous regions within a cell zone into separate cell zones.

/mesh/separate/separate-cell-by-shape
separates cells with different shapes (pyramids, tetrahedra, etc.) into separate cell zones.

/mesh/separate/separate-cell-by-size
separates cells based on the specified minimum and maximum cell sizes.

```
/mesh/separate/separate-cell-by-skew  
separates cells based on the specified cell skewness.
```

18.9. Manipulating Cell Zone Conditions

Case files read in the meshing mode also contain the boundary and cell zone conditions along with the mesh information. The [Cell Zone Conditions Dialog Box \(p. 593\)](#) allows you to copy or clear cell zone conditions when a case file is read.

- You can copy the cell zone conditions from the zone selected in the **With** list to those selected in the **Without** list using the **Copy** option.
- You can clear the cell zone conditions assigned to the zones selected in the **With** list using the **Clear** option.

18.9.1. Text Commands for Manipulating Cell Zone Conditions

18.9.1.1. Text Commands for Manipulating Cell Zone Conditions

The following text commands allow you to manipulate cell zone conditions when a case file is read:

```
/mesh/cell-zone-conditions/clear  
clears the cell zone conditions assigned to the specified cell zones.  
  
/mesh/cell-zone-conditions/clear-all  
clears the cell zone conditions assigned to all the cell zones.  
  
/mesh/cell-zone-conditions/copy  
allows you to copy the cell zone conditions from the cell zone selected to the specified cell zones.
```

18.10. Using Domains to Group and Mesh Boundary Faces

Domains allow you to group different boundary zones together so that you can create tetrahedral meshes in the region they enclose, or you can limit the zones available for a display or report to only those zones in a selected subset of the domain, rather than the entire domain.

- 18.10.1. Using Domains
- 18.10.2. Defining Domains
- 18.10.3. Text Commands for Domains

18.10.1.1. Using Domains

If you are generating a hybrid mesh containing hexahedra, tetrahedra, and pyramids, you can identify a domain of the global mesh as the region in which you want to generate tetrahedral cells. You can also use domains to group boundary zones so that you can perform diagnostics on them or display them. When you display the grid, the zones available for display will be only those zones that are included in the active domain. Similarly, diagnostic reports will report information about only those zones.

- If you want to check a subset of the global domain, you can create and activate a domain that includes the desired zones, and then proceed with the display or report.
- If you want your grid display or report to include all zones in the mesh, activate the **global** domain in the **Domains** dialog box.

18.10.2. Defining Domains

The procedure for defining a new domain is as follows:

1. Deselect all zones in the **Boundary Zones** list and click **Create**. It is quicker to create an empty domain and then add the zones you want, instead of creating a domain with many zones and then removing those you do not want.
2. In the **Boundary Zones** list, select the zones you want to include in the new domain. If you are not sure about the zones, click **Draw** to display the zones that are currently selected in the **Boundary Zones** list.
It is possible to select all triangular or quadrilateral boundary face zones by choosing **tri** or **quad** in the **Boundary Zone Groups** list.
 - If you are creating a domain within a hybrid mesh to create tetrahedra, make sure that the domain contains all zones required to enclose the region that is to be meshed with tetrahedra, and *only* those zones.
 - If the zones you select do not completely enclose the region, or if you include additional zones that do not bound this region, the tetrahedral meshing is likely to fail or be incorrect.
3. Click **Change**. The dialog box will be updated so that the node, face, and cell zones highlighted in their respective lists are those that are affiliated with the boundary zones in the domain.
4. Select the domain that you want to mesh (or display or report on) in the **Domains** list, and then click the **Activate** button. This domain is then considered to be the active domain, and the **Activate** button is disabled until you select another domain.

By default, the most recently created domain is automatically set to be the active domain, so you only need to explicitly set the active domain if it is not the one you just created.

Note

If you are dissatisfied with the domain definition, delete it using the **Delete** button and start over, or modify it by selecting it and using the **Change** button.

18.10.3. Text Commands for Domains

Text commands for domain creation and activation are as follows:

/mesh/domains/activate

activates the specified domain for meshing/reporting operations.

/mesh/domains/create

creates a new domain based on the specified boundary face zones.

/mesh/domains/create-by-cell-zone

creates a new domain based on the specified cell zone.

/mesh/domains/create-by-point

creates a new domain based on the specified material point.

Note

The create-by-point option works only for cases with no overlapping face zones.

/mesh/domains/delete

deletes the specified domain.

/mesh/domains/draw

displays the boundary face zones of the specified domain.

/mesh/domains/print

prints the zones comprising the specified domain.

18.11. Checking the Mesh

When you complete the mesh generation process, you need to check the mesh before saving it.

The mesh checking capability will check the mesh connectivity and the orientation of the faces (face handedness, which should be right-handed for all faces because the solvers use a right-handed system). The domain extents, volume statistics, and face area statistics will be reported along with the results of other checks on the mesh.

You can obtain this information by selecting the **Mesh/Check** menu item.

Mesh → Check

Alternatively, you can use the command `/mesh/check` to check the mesh.

When you select the **Mesh/Check** menu item, the mesh check information will be printed in the console.

The sample output is as follows:

```
Domain extents.  
  x-coordinate: min = -2.500000e+00, max = 2.500000e+00.  
  y-coordinate: min = -4.357625e-15, max = 2.000000e+00.  
  z-coordinate: min = -1.111022e-04, max = 2.000000e+00.  
Volume statistics.  
  minimum volume: 2.297312e-09.  
  maximum volume: 7.856795e-03.  
  total volume: 1.953600e+01.  
Face area statistics.  
  minimum face area: 1.258676e-06.  
  maximum face area: 4.944555e-02.  
  average face area: 1.939640e-04.  
Checking number of nodes per cell.  
Checking number of faces per cell.  
Checking cell faces.  
Checking isolated cells.  
Checking face handedness.  
Checking periodic face pairs.  
Checking face children.  
Checking face zone boundary conditions.  
Checking for invalid node coordinates.  
Checking poly cells.  
Done.
```

The domain extents list the x, y, and z coordinates in meters. The volume statistics include the maximum, minimum, and total cell volume in m³. A negative value for the minimum volume indicates that one or more cells have improper connectivity. The face area statistics will also be reported.

The topological information verified includes the number of nodes and faces per cell. A tetrahedral cell should have 4 faces and 4 nodes while a hexahedral cell should have 6 faces and 8 nodes.

Next, the face-handedness and face node order for each zone will be checked. The zones should contain all right-handed faces and all faces should have the correct node order. If any problems are reported, you need to repair the mesh.

18.12. Checking the Mesh Quality

It is important to check the quality of the mesh to evaluate whether it is sufficient for the problem you are modeling. It is recommended that you check the mesh quality before transferring the mesh data to solution mode or writing out the mesh/case file.

You can obtain information about the mesh quality by selecting the **Mesh/Check Quality** menu item.

Mesh → Check Quality

Alternatively, you can use the command /mesh/check-quality to check the mesh quality.

When you select the **Mesh/Check Quality** menu item, the quality information will be printed in the console. The sample output is as follows:

```
Mesh Quality:  
Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.  
Minimum Orthogonal Quality = 6.07960e-01  
Maximum Aspect Ratio = 5.42664e+00
```

Note

The information reported corresponds to that reported using the **Report Quality** option in solution mode (refer to the [User's Guide](#) for details).

For information about additional quality metrics, set the /mesh/check-quality-level to 1 prior to using the **Mesh/Check Quality** option. In addition to the orthogonal quality and Fluent aspect ratio, additional metrics like cell squish and skewness will be reported when the check-quality-level is set to 1.

18.13. Clearing the Mesh

If you are dissatisfied with the volume mesh generated, you can choose to clear the mesh and start again from the boundary mesh. When the mesh is cleared, all interior nodes and faces, and all cells both live and dead are deleted. Only the boundary nodes and faces will be left. After the mesh is cleared, you can generate a new mesh.

This feature is available via the **Clear** menu item in the **Mesh** pull-down menu.

Mesh → Clear

You can also use the text command `/mesh/clear-mesh`. To delete the boundary mesh, use the text command `mesh/reset-mesh`. When you use either of these commands you will be asked to confirm that you want to clear or reset the mesh.

Important

If you have used domains to generate the mesh or grouped zones for reporting (as described in [Using Domains to Group and Mesh Boundary Faces \(p. 368\)](#)), only the mesh in the active domain will be cleared.

Chapter 19: Examining the Mesh

The following sections explain how to examine the mesh. You can specify various parameters that control the display environment, display the mesh, view histograms of cell and face distribution, modify the mouse button functions, and manipulate the lighting and view in the graphics window.

- 19.1. Displaying the Grid
- 19.2. Checking Face Distribution
- 19.3. Checking Cell Distribution
- 19.4. Modifying the Attributes of the Plot Axes
- 19.5. Controlling Display Options
- 19.6. Modifying the View
- 19.7. Adding Lights
- 19.8. Composing a Scene
- 19.9. Controlling the Mouse Buttons
- 19.10. Controlling the Mouse Probe Functions
- 19.11. Annotating the Display
- 19.12. Shortcuts for Selecting Zones
- 19.13. Setting Default tgvars

19.1. Displaying the Grid

You can manipulate the display of grid entities and modify the way they appear in the graphics window. You can draw selected boundary zones, unmeshed faces, free or multiply-connected faces, and unused nodes.

You can also control the display of items using intrinsic parameters such as skewness. When you specify a range for a particular parameter, the entities that satisfy that condition will be displayed. If more than one range limitation is specified, then only those entities that match all specified range limitations are drawn.

You can also specify information regarding color, display of solid (filled) faces, and shrinkage of faces and cells. These features can help you visualize your mesh effectively and quickly determine the cause of any problems in the mesh. You can perform these operations using the **Display Grid** dialog box and the associated set of dialog boxes or using the associated text commands.

The **Display Grid** dialog box consists of a series of “tabbed frames” within the main dialog box. Each tab has a set of related controls and the active tab is indicated by the labeled button which appears depressed.

The **Nodes**, **Faces**, and **Cells** tabs contain controls and zone lists specific to those entity types. The **Bounds** tab contains controls allowing you to limit the display within specific coordinate ranges, or within a distance to a selected entity. The **Attributes** tab contains controls allowing you to change how the entities are displayed, add labels, and display normal vectors.

The **Mesh Generation** task page also contains tools that allows you to interactively crop the display of the mesh. The **Bounds** group contains a tool to limit the display within specific coordinate ranges or

within a specified distance to a selected entity and a clipping slider to set limits based on the displayed mesh or all mesh.

The **Objects Toolbar** contains drop-down lists for **Face Options** and **Object Options**, which allow you to control the display of face zones and objects, respectively.

[19.1.1. Generating the Grid Display Using the Display Grid Dialog Box](#)

[19.1.2. Grid Display Options](#)

[19.1.3. Text Commands for Displaying the Grid](#)

[19.1.4. Text Commands for Grid Colors](#)

[19.1.5. Text Commands for Style Attributes](#)

[19.1.6. Displaying Objects](#)

19.1.1. Generating the Grid Display Using the Display Grid Dialog Box

The procedure for generating the grid display is as follows:

1. Select the entities to be displayed in the zone lists in the **Nodes**, **Faces**, or **Cells** tabs as appropriate. You can also select several entities of the same group by selecting the required group in the respective zone groups lists in the **Nodes**, **Faces**, or **Cells** tabs.
2. Select the display options as appropriate from the **Options** group box in the **Nodes**, **Faces**, or **Cells** tabs.
3. Set the range for limiting the display to a specific coordinate range or within a specific distance from a selected entity, if required.
4. Set the display options as appropriate in the **Attributes** tab.
5. Click **Display** to draw the grid in the active graphics window.

19.1.2. Grid Display Options

The grid display options include modifying the grid colors, adding the outline of important features to a display, displaying normals, shrinking the faces and/or cells in the display, adding lights for filled grid displays, etc.

Modifying the Grid Colors

You can control the colors used to render the grid for each zone type or entity. You can modify the colors using the **Grid Colors** dialog box.

1. Click the **Colors...** button in the **Display Grid** dialog box to open the **Grid Colors** dialog box.

By default, the **Color by Type** option is selected, allowing you to assign colors based on zone type.

2. To change the color assigned to a particular zone type, select the zone type in the **Types** list.
3. Select the required color in the **Colors** list.

4. If you prefer to assign colors based on the zone ID, select the **Color by ID** option.

Important

You can set colors individually for the grid displayed on each of the zones using the **Scene Description** dialog box.

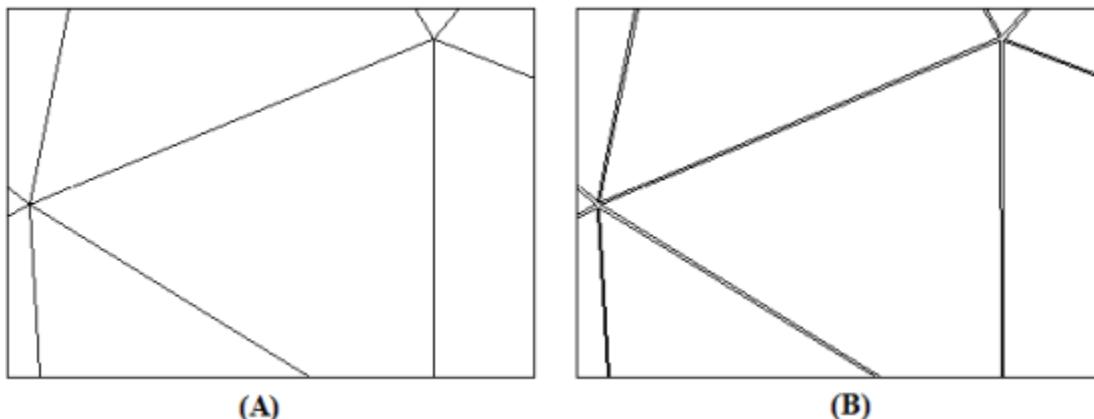
Adding Features to an Outline Display

For closed 3D geometries, the standard outline display may not show sufficient details to accurately depict the shape. This is because for each boundary, only the edges on the outside of the geometry (i.e., those used by only one face on the boundary) are drawn. You can capture additional features using the **Feature** option in the **Display Grid** dialog box. Specify an appropriate value for the **Feature Angle** to obtain the outline display required.

Shrinking Faces and Cells in the Display

To distinguish individual faces or cells in the display, you may want to enlarge the space between adjacent faces or cells by increasing the **Shrink Factor** in the **Display Grid** dialog box. The default value of zero produces a display in which the adjacent faces or cells overlap. A value of 1 creates the opposite extreme, where each face or cell is represented by a point and there is a considerable space between entities. A small value such as 0.01 may be sufficient to allow you to distinguish a face or cell from its neighbor. [Figure 19.1: Grid Display \(A\) With Shrink Factor = 0 \(B\) With Shrink Factor = 0.01 \(p. 375\)](#) shows displays with different **Shrink Factor** values.

Figure 19.1: Grid Display (A) With Shrink Factor = 0 (B) With Shrink Factor = 0.01



19.1.3. Text Commands for Displaying the Grid

The following text commands are similar in functionality to the controls in the **Display Grid** dialog box:

/display/all-grid

displays the grid according to the currently set parameters.

/display/boundary-cells

displays boundary cells attached to the specified face zones.

/display/boundary-grid

displays only boundary zones according to the currently set parameters.

/display/draw-cells-using-faces
draws cells that are neighbors for the selected faces.

/display/draw-cells-using-nodes
draws cells that are connected to the selected nodes.

/display/draw-face-zones-using-entities
displays the neighboring face zones of the selected entity.

/display/draw-zones
draws the boundary/cell zones using the zone ID as input.

/display/redisplay
redraws the grid in the graphics window.

/display/set-grid/all-cells?
toggles the display of all cells.

/display/set-grid/all-faces?
toggles the display of all faces.

/display/set-grid/all-nodes?
toggles the display of all nodes.

/display/set-grid/free?
toggles the drawing of faces/nodes that have no neighboring face on at least one edge.

/display/set-grid/left-handed?
toggles the display of left-handed faces.

/display/set-grid/multi?
toggles the display of those faces/nodes that have more than one neighboring face on an edge.

/display/set-grid/refine?
toggles the display of those faces that have been marked for refinement.

/display/set-grid/unmeshed?
toggles the display of nodes and faces that have not been meshed.

/display/set-grid/unused?
toggles the display of unused nodes.

/display/set-grid-marked?
toggles the display of marked nodes.

/display/set-grid/tagged?
toggles the display of tagged nodes.

/display/set-grid/cell-quality
sets the lower and upper limits of quality for cells to be displayed. Only cells with a quality measure value (e.g., skewness) within the specified range will be drawn.

/display/set-grid/face-quality
sets the lower and upper limits of quality for faces to be displayed. Only faces with a quality measure value (e.g., skewness) within the specified range will be drawn.

/display/set-grid/neighborhood

sets the x, y, and z range to be within a specified neighborhood of an entity (node, edge, face, cell) or position "x y z" or zone, specified in symbol or string format.

/display/set-grid/x(or y or z)-range

limits the display to the specified range.

/display/set-grid/normals?

toggles the display of face normals.

/display/set-grid/normal-scale

sets the scale factor for face normals.

/display/set-grid/labels?

toggles the display of labels.

/display/set-grid/label-alignment

sets the alignment of labels that appear in the graphics window. By default, the label is centered on the node, cell, etc. to which the label refers. You can specify

* , ^ , v , < , > , for center, top, bottom, left, or right, respectively. You can also combine symbols—for example, "*v" for bottom center.

/display/set-grid/label-font

sets the label font. By default, all labels appear in "sans serif" font. Some other choices are Roman, typewriter, and stroked.

/display/set-grid/label-scale

scales the size of the label.

/display/set-grid/list

lists all the grid display settings.

/display/set-grid/default

resets the grid display parameters to their default values.

/display/center-view-on

sets the camera target to be the center (centroid) of an entity.

/display/set/edges?

turns the display of face/cell edges on and off.

/display/set/filled-grid?

turns the filled grid option on and off. When a grid is not filled, only its outline is drawn.

/display/set/shrink-factor

sets shrinkage of both faces and cells. A value of zero indicates no shrinkage, while a value of one would shrink the face or cell to a point.

/display/set-list-tree-separator

sets the separator character to be used to determine the common prefix for items listed in the selection lists, when the tree view is used.

19.1.4. Text Commands for Grid Colors

The following text interface commands are similar in functionality to the controls in the **Grid Colors** dialog box:

/display/set/colors/color-by-type?

allows you to specify that the entities should be colored by their zone types. The individual zone type colors can be set using the various text-menu options:

- axis-faces
- far-field-faces
- free-surface-faces
- inlet-faces
- interface-faces
- interior-faces
- internal-faces
- outlet-faces
- periodic-faces
- rans-les-interface-faces
- surface
- symmetry-faces
- traction-faces
- wall-faces

/display/set/colors/list

lists the colors available for the selected zone type.

/display/set/colors/reset-colors

resets the individual grid surface colors to the defaults.

19.1.5. Text Commands for Style Attributes

The following text interface commands perform functions similar to those in the **Style Attributes** dialog box.

/display/set-grid/node-size

sets the node symbol scaling factor.

/display/set-grid/node-symbol

specifies the node symbol.

/display/set/styles/

contains commands for setting the display style for the different types of nodes and faces that can be displayed.

19.1.6. Displaying Objects

Objects can be displayed by selecting them in the **Objects** selection list in the **Mesh Generation** task page and clicking **Draw**. The **Objects Toolbar** contains drop-down lists for **Face Options** and **Object Options**, which allow you to control the display of face and edge zones and objects, respectively.

The **Mesh Generation** task page also contains tools that allows you to interactively crop the display of the mesh. The **Bounds** group contains a tool to limit the display within specific coordinate ranges or within a specified distance to a selected entity and a clipping slider to set limits based on the displayed mesh or all mesh.

19.1.6.1. Text Commands for Displaying Objects

The following text commands can be used to display objects.

/display/objects/display-neighborhood

displays the objects which are in the neighborhood of the selected object. The neighboring objects have to be in contact, or intersecting the selected object.

/display/objects/display-similar-area

displays the objects with similar area to the selected object area.

/display/objects/explode

explodes the objects in the geometry. (This command is valid only when the geometry is an assembled mode.)

/display/objects/hide-objects

hides the selected objects in the display.

/display/objects/implode

implodes or assembles the objects in the geometry. (This command is available only when the geometry is an exploded mode.)

/display/objects/isolate-objects

displays only the selected objects.

/display/objects/make-transparent

makes the geometry transparent so that internal objects are visible. This command works as a toggle undoing the transparency of the previously selected objects.

/display/objects/select-all-visible

selects all the visible objects in the graphics window.

/display/objects/show-all

unhides all the objects in the geometry and displays them.

/display/objects/toggle-color-mode

toggles the colors of the geometry. In one mode geometry is colored object-wise while in the other mode it is colored zone-wise.

```
/display/objects/toggle-color-palette
```

toggles the color palette of the geometry.

19.2. Checking Face Distribution

When checking the quality of the mesh you may find it useful to look at a histogram of boundary face quality. You can generate such a plot or report using the **Face Distribution** dialog box or the associated text commands. When a histogram plot is displayed in the graphics window, you can use any of the mouse buttons to add text annotations to the plot. See [Controlling the Mouse Buttons \(p. 391\)](#) and [Annotating the Display \(p. 393\)](#) for more information about using the mouse buttons and annotation features.

19.2.1. Text Commands for Face Distribution

19.2.1.1. Text Commands for Face Distribution

Text commands with functionality similar to that of the controls in the **Face Distribution** dialog box are:

```
/display/xy-plot/face-distribution
```

plots a histogram of face quality.

```
/boundary/face-distribution
```

reports the distribution of face quality in the text window.

```
/display/xy-plot/set/xy-percent-y?
```

toggles whether the y-coordinate should be scaled to show a percent of total values being plotted.

19.3. Checking Cell Distribution

You can check cell size or quality by using the controls in the **Cell Distribution** dialog box (or the associated text commands) to generate a histogram plot or report. When a histogram plot is displayed in the graphics window, you can use any of the mouse buttons to add text annotations to the plot. For more information about the annotation features, see [Controlling the Mouse Buttons \(p. 391\)](#) and [Annotating the Display \(p. 393\)](#).

19.3.1. Text Commands for Cell Distribution

Text commands with functionality similar to that of the controls in the **Cell Distribution** dialog box are:

```
/display/xy-plot/cell-distribution
```

plots a histogram of cell quality.

```
/report/cell-distribution
```

reports the distribution of cell quality in the text window.

```
/display/xy-plot/set/xy-percent-y?
```

toggles whether the y-coordinate should be scaled to show a percent of total values being plotted.

19.4. Modifying the Attributes of the Plot Axes

You can modify the appearance of the axes by changing the parameters that control the labels, scale, range, numbers, and tick marks. These operations are available in the **Axes** dialog box and in the associated text interface menu.

19.4.1. Text Commands for Modifying Axes Attributes

19.4.1.1. Text Commands for Modifying Axes Attributes

Text commands with functionality similar to that of the controls in the **Axes** dialog box are:

/display/xy-plot/set/auto-scale?

sets the range for the x- and y-axis. If auto-scaling is not activated for a particular axis, you are prompted for the minimum and maximum data values.

/display/xy-plot/set/background-color

sets the color of the field within the abscissa and ordinate axes.

/display/xy-plot/set/key

sets the visibility and title of the description box that displays the markers and/or lines with their associated data. The key can be positioned and resized using the left mouse button.

/display/xy-plot/set/labels

sets the strings that define the x- and y- axis labels.

/display/xy-plot/set/lines

sets the pattern, weight, and color of the plot lines.

/display/xy-plot/set/log?

toggles the log scaling on the x- and y-axis.

/display/xy-plot/set/markers

sets parameters for the data markers.

/display/xy-plot/set/numbers

sets the format and precision of the data numbers displayed on the x- and y-axis.

/display/xy-plot/set/rules

sets the visibility, line weight and color of the major and minor rules in the x- and y-axis directions.

19.5. Controlling Display Options

When examining graphics, you may want to change some of the attributes of the display window. For example, you can open a number of windows to look at several different displays simultaneously, control the lighting attributes in a scene, or modify rendering parameters.

These and other options are available in the **Display Options** dialog box or with the associated text commands.

19.5.1. Modifying the Display Options

19.5.2. Text Commands for Controlling Display Options

19.5.1. Modifying the Display Options

You can modify some of the rendering parameters available in the **Rendering** group box in the [Display Options Dialog Box \(p. 614\)](#). After making a change to any of these rendering parameters, click **Apply** to re-render the scene in the active graphics window with the new attributes.

Line Width

By default, all lines drawn in the display have a thickness of 1 pixel. If you want to increase the thickness of the lines, increase the value of **Line Width**.

Point Symbol

By default, nodes displayed on surfaces are represented in the display by a + sign inside a circle. If you want to modify this representation (for example, to make the nodes easier to see), you can select a different symbol in the **Point Symbol** drop-down list.

Animation Options

There are two animation options that you can choose from:

All

uses a solid-tone shading representation of all geometry during mouse manipulation.

Wireframe

uses a wireframe representation of all geometry during mouse manipulation. If your computer has a graphics accelerator, you may not want to use this option; otherwise, the mouse manipulation may be very slow.

Double Buffering

Enabling the **Double Buffering** option can dramatically reduce screen flicker during graphics updates. Note, however, that if your display hardware does not support double buffering and you turn this option on, double buffering will be done in software. Software double buffering uses extra memory.

Outer Face Culling

This option enables you to turn off the display of outer faces in wall zones. This is useful for displaying both sides of a slit wall. By default, when you display a slit wall, one side will “bleed” through to the other. When you enable the **Outer Face Culling** option, the display of a slit wall will show each side distinctly as you rotate the display. This option can also be useful for displaying two-sided walls (that is, walls with fluid or solid cells on both sides).

Hidden Line Removal

If you use hidden line removal, ANSYS Fluent will try to determine which lines in the display are behind others. If you do not use hidden line removal, all lines will be displayed, and a cluttered display will result for most 3D mesh displays. This option is not available when the **Workbench** color scheme is used.

Hidden Surface Removal

If you use hidden surface removal, ANSYS Fluent will try to determine which surfaces in the display are behind others. If you do not use hidden surface removal, all surfaces will be displayed, and a cluttered display will result for most 3D mesh displays.

You can choose one of the following methods for performing hidden surface removal in the Hidden Surface Method drop-down list. These options vary in speed and quality, depending on the device you are using.

Hardware Z-buffer

is the fastest method if supported by your hardware. The accuracy and speed of this method is hardware dependent.

Painters

will show fewer edge-aliasing effects than hardware-z-buffer. This method is often used instead of software-z-buffer when memory is limited.

Software Z-buffer

is the fastest of the accurate software methods available (especially for complex scenes), but it is memory intensive.

Z-sort only

is a fast software method, but it is not as accurate as software-z-buffer.

19.5.2. Text Commands for Controlling Display Options

Text commands with functionality similar to the controls in the **Display Options** dialog box are listed below:

/display/open-window
opens a graphics window.

/display/close-window
closes a graphics window.

/display/set-window
sets the specified window as the active graphics window.

/display/set/line-weight
sets the line width factor.

/display/set/reset-graphics
resets the graphics system.

/display/set/title
sets the problem title.

/display/set/rendering-options/
contains the commands that allow you to set options that determine how the scene is rendered.

/display/set/rendering-options/animation-option/
allows you to specify the animation option as appropriate.

all

uses a solid-tone shading representation of all geometry during mouse manipulation.

wireframe

uses a wireframe representation of all geometry during mouse manipulation. This is the default option.

/display/set/rendering-options/auto-spin?

enables mouse view rotations to continue to spin the display after the button is released.

/display/set/rendering-options/device-info

prints information about the graphics driver in the console.

/display/set/rendering-options/double-buffering?

allows you to enable/disable double buffering. Double buffering dramatically reduces screen flicker during graphics updates. If your display hardware does not support double buffering and you enable this option, double buffering will be done in the software. Software double buffering uses extra memory.

/display/set/rendering-options/driver/

contains commands for changing the current graphics driver.

/display/set/rendering-options/face-displacement

sets the face displacement (in Z-buffer units along the camera Z-axis) for the displayed geometry when both faces and edges are displayed simultaneously.

Quite often, faces and edges may be displayed at the same location causing bad edges known as edge-stitching. Parts of an edge become invisible due to overlapping portions of the face. Setting the face displacement can help alleviate the problem by pushing the faces back by a small amount so that the complete edge becomes visible. A value of 0 for the face-displacement does not displace the faces at all. If the value is set too high, it may cause the hidden edges to show up in front of the displayed faces. You can adjust the display by specifying an appropriate value for the face displacement. The default value of face displacement is set to 8 Z-buffer units.

/display/set/rendering-options/hidden-lines?

allows you to enable/disable hidden line removal.

Note

This option is not available when the **Workbench** color scheme is used (/display/set/colors/color-scheme workbench).

/display/set/rendering-options/hidden-surfaces?

enables/disables hidden surfaces removal.

/display/set/rendering-options/hidden-surface-method/

allows you to choose from among the hidden surface removal methods that are supported. These options (listed below) are display hardware dependent.

hardware-z-buffer

is the fastest method if your hardware supports it. The accuracy and speed of this method is hardware dependent.

painters

will show less edge-aliasing effects than hardware-z-buffer. This method is often used instead of software-z-buffer when memory is limited.

software-z-buffer

is the fastest of the accurate software methods available (especially for complex scenes), but it is memory intensive.

z-sort

is a fast software method, but it is not as accurate as software-z-buffer.

list

lists the current hidden surface method.

/display/set/rendering-options/outer-face-cull?

enables/disables the display of outer faces.

/display/set/rendering-options/surface-edge-visibility

controls whether or not the mesh edges are drawn.

/display/set/rendering-options/wireframe-animation?

turns the use of wireframes during mouse manipulation on or off.

/display/set/windows/

contains commands that allow you to customize the relative positions of sub-windows inside the active graphics window. The menu structure for the axes, main, scale, text, and xy submenus is similar. The * in the following descriptions stands for any one of these submenus.

/display/set/windows/*/

contains commands for changing the location of the sub-window (*).

/display/set/windows/*/border?

draws a border around the sub-window.

/display/set/windows/*/bottom

sets the bottom boundary of the sub-window.

/display/set/windows/*/left

sets the left boundary of the sub-window.

/display/set/windows/*/right

sets the right boundary of the sub-window.

/display/set/windows/*/top

sets the top boundary of the sub-window.

/display/set/windows/*/visible?

sets the visibility of the sub-window.

/display/set/windows/axes/clear?

sets the transparency of the axes window.

/display/set/windows/video/

contains options for modifying a video. This command is not relevant in the meshing mode.

Text interface commands that control the display and are not found in the GUI are as follows:

/display/clear

clears the active graphics window. This option is useful when you redo an overlay.

/display/set/re-render

re-renders the current window after modifying the variables in the set menu.

/display/set/colors/background

sets the background (window) color.

/display/set/colors/foreground

sets the foreground (text and window frame) color.

/display/set/colors/user-color
allows you to change the color for the specified zone.

/display/xy-plot/file
allows you to choose a file from which to create an xy plot.

/display/xy-plot/set/plot-to-file
allows you to write xy plot values to a file.

19.6. Modifying the View

You can control the view of the scene that is displayed in the graphics window. You can modify the view by scaling, centering, or rotating the scene. You can also save a view that you have created, or restore or delete a view that you saved earlier. These operations are performed in the **Views** dialog box or with the associated text commands.

19.6.1. Text Commands for Modifying the View

Text interface commands with the same functionality as the controls in the **Views** dialog box are:

/display/view/auto-scale
scales and centers the current scene without changing its orientation.

/display/view/default-view
resets the view to front and center.

/display/view/delete-view
deletes a named view from the list of stored views.

/display/view/last-view
returns to the camera position before the last manipulation.

/display/view/list-views
lists all predefined and saved views.

/display/view/restore-view
sets the current view to one of the stored views.

/display/view/read-views
reads views from an external view file.

/display/view/save-view
saves the currently displayed view into the list of stored views.

/display/view/write-views
writes views to an external view file.

/display/view/camera/dolly-camera
allows you to move the camera left, right, up, down, in, and out.

/display/view/camera/field
allows you to set the field of view (width and height) of the scene.

/display/view/camera/orbit-camera

allows you to move the camera around the target. Gives the effect of circling around the target.

/display/view/camera/pan-camera

gives you the effect of sweeping the camera across the scene. The camera remains at its position but its target changes.

/display/view/camera/position

sets the camera position.

/display/view/camera/projection

lets you switch between perspective and orthographic views.

/display/view/camera/roll-camera

lets you adjust the camera up-vector.

/display/view/camera/target

sets the point the camera will look at.

/display/view/camera/up-vector

sets the camera up-vector.

/display/view/camera/zoom-camera

adjusts the camera's field of view. This operation is similar to dollying the camera in or out of the scene. Dollying causes entities in front to move past you. Zooming changes the perspective effect in the scene (and can be disconcerting).

19.7. Adding Lights

You can add lights with a specified color and direction to your display. These lights can enhance the appearance of the display, especially for 3D geometries.

19.7.1. Enabling Lighting Effects

19.7.2. Text Commands for Adding Lights

19.7.1. Enabling Lighting Effects

- To enable the effect of lighting using the [Display Options Dialog Box \(p. 614\)](#), enable **Lights On** in the **Lighting Attributes** group box and click **Apply**. You can also choose the method to be used in lighting interpolation. Select **Flat**, **Gouraud**, or **Phong** in the **Lighting** drop-down list. **Flat** is the most basic method: there is no interpolation within the individual polygonal facets. **Gouraud** and **Phong** have smoother gradations of color because they interpolate on each facet.
- The [Lights Dialog Box \(p. 621\)](#) contains options for creating lights and then enabling/disabling individual lights as needed. In this way, you can retain lights that you have defined previously but do not want to use at present.

The default light, **light 0** is defined to be dark gray with a direction of (1,1,1). To define additional lights, do the following:

1. Increase **Light ID** to a new value (for example, 1).
2. Enable the **Light On** check box.

3. Define the light color by entering a descriptive string (for example, `lavender`) in the **Color** field, or by moving the **Red**, **Green**, and **Blue** sliders to obtain the desired color. The default color for all lights is dark gray.
4. Specify the light direction by doing one of the following:
 - Enter the (X, Y, Z) Cartesian components under **Direction**.
 - Click the middle mouse button in the desired location on the sphere under **Active Lights**. (You can also move the light along the circles on the surface of the sphere by dragging the mouse while holding down the middle button.) You can rotate the sphere by pressing the left mouse button and moving the mouse (like a trackball).
 - Use your mouse to change the view in the graphics window so that your position in reference to the geometry is the position from which you would like a light to shine. Then click **Use View Vector** to update the X,Y,Z fields with the appropriate values for your current position and update the graphics display with the new light direction. This method is convenient if you know where you want a light to be, but you are not sure of the exact direction vector.
5. Repeat steps 1–4 to add more lights.
6. When the lights have been defined, click **Apply** in the **Lights** dialog box to save the definitions.

To remove a light, enter the ID number of the light to be removed in the **Light ID** field and then clear the **Light On** check box. When a light is disabled, its definition is retained, so you can easily add it to the display again at a later time by selecting the **Light On** check box.

If you have made changes to the light definitions, but you have not yet clicked **Apply**, you can reset the lights by clicking **Reset**. All lighting characteristics will revert to the last saved state (that is, the lighting that was in effect the last time you opened the dialog box or clicked on **Apply**).

19.7.2. Text Commands for Adding Lights

Text interface commands with the same functionality as the controls in the **Lights** dialog box are:

/display/set/lights/headlight-on?
turns the light that moves with the camera on/off.

/display/set/lights/lights-on?
turns all lights on/off.

/display/set/lights/lighting-interpolation
sets the lighting interpolation method to be used. You can choose `flat`, `gouraud`, or `phong`. The first one is the most basic method, and the others are more sophisticated and provide smoother gradations of color.

/display/set/lights/set-ambient-color
sets the ambient color for the scene. The ambient color is the background light color in a scene.

/display/set/lights/set-light
adds or modifies a directional, colored light.

19.8. Composing a Scene

After displaying the mesh or parts of the mesh in your graphics window, you may want to overlay an additional display or move entities around and change their characteristics to increase the effectiveness of the scene displayed.

You can use the [Scene Description Dialog Box \(p. 623\)](#), and the [Display Properties Dialog Box \(p. 624\)](#) and the [Transformations Dialog Box \(p. 626\)](#) which are opened from within it, to rotate, translate, and scale each entity individually, as well as change the color and visibility of each entity. You can make geometric entities visible and invisible, thereby adding or deleting entities from the scene one at a time.

19.8.1. Changing the Display Properties

19.8.2. Transforming Geometric Entities in a Scene

19.8.3. Using the Scene Description Dialog Box

19.8.4. Text Commands for Scene Description

19.8.1. Changing the Display Properties

To enhance the scene in the graphics window, you can change the color, visibility, and other display properties of each geometric entity in the scene.

- You can specify different colors for displaying the edges and faces of a grid entity to show the underlying mesh (edges) when the faces of the grid are filled and shaded.
- You can also make a selected entity temporarily invisible. If, for example, you are displaying the entire grid for a complicated problem, you can make entities visible or invisible to display only certain boundary zones of the grid without regenerating the grid display using the **Display Grid** dialog box.
- You can also use the visibility controls to manipulate geometric entities for efficient graphics display.

These features along with several others are available in the **Display Properties** dialog box.

19.8.2. Transforming Geometric Entities in a Scene

When composing a scene in your graphics window, it is helpful to move a particular entity from its original position or to increase or decrease its size. For example, you may want to temporarily move an interior portion of the grid outside the grid boundaries where it can be seen and interpreted more easily.

You can also move an entity by rotating it about a specified point. If you want to display one entity more prominently than the others, scale its size. All these capabilities are available in the **Transformations** dialog box.

19.8.3. Using the Scene Description Dialog Box

You can use the [Scene Description Dialog Box \(p. 623\)](#) and its associated dialog boxes as follows:

1. Select the mesh entities in the **Names** selection list in the **Scene Description** dialog box.

The **Names** list is a list of the mesh entities that currently exist in the scene (including those that are presently invisible). If you select more than one entity, any operation (color specification, transformation, etc.) will apply to all the selected entities. You can also select the entities by clicking on them in the graphics window using the mouse probe button (right mouse button, by default).

2. Set the color, visibility, and other display properties for the mesh entities selected using the [Display Properties Dialog Box \(p. 624\)](#).
 - You can specify different colors for individual mesh entities. You can specify different colors for displaying the edges and faces of a grid entity to show the underlying mesh (edges) when the faces of the grid are filled and shaded.

To modify the color of faces, edges, or lines, choose **face-color**, **edge-color**, **line-color**, or **node-color** in the **Color** drop-down list. The **Red**, **Green**, and **Blue** color scales will show the RGB components of the face, edge or line color, which you can modify by moving the sliders on the color scales. When you are satisfied with the color specification, click **Apply** to save it and update the display.

 - You can control the visibility of individual mesh entities using the **Visible** option. Hence, you can make an entity visible or invisible to display only certain boundary zones, without regenerating the entire mesh display.
 - To enable the effect of lighting for the selected entities on or off, use the **Lighting** check box. You can choose to have lighting affect only certain entities instead of all of them.
 - To toggle the filled display of faces for the selected zones, use the **Faces** option. Enabling **Faces** on here has the same effect as turning it on for the entire mesh in the [Display Grid Dialog Box \(p. 596\)](#).
 - To enable the display of outer edges, use the **Outer Faces** option. This option is useful for displaying both sides of a slit wall. By default, when you display a slit wall, one side will “bleed” through to the other. When you disable the **Outer Faces** option, the display of a slit wall will show each side distinctly as you rotate the display. This option can also be useful for displaying two-sided walls (that is, walls with fluid or solid cells on both sides).
 - To enable/disable the display of interior and exterior edges of the zones, use the **Edges** option.
 - To enable/disable the display of the outline of the zones, use the **Perimeter Edges** check box.
 - To enable/disable the display of feature lines, if any, use the **Feature Edges** option.
 - To enable/disable the display of the lines (if any), use the **Lines** check box.
 - To enable/disable the display of nodes (if any), use the **Nodes** check box.
3. To overlay one display over another, select the mesh entities in the **Names** selection list, enable **Overlays** in the **Scene Composition** group box and click **Apply**. Once overlaying is enabled, subsequent graphics that you generate will be displayed on top of the existing display in the active graphics window. To generate a plot without overlays, you must disable the **Overlays** option and click **Apply**.

Note

To overlay bounded cell zones over face zones in the display, do the following:

1. Display only the bounded cell zones (face zones should not be selected).
2. Select the cell zones in the **Names** selection list in the [Scene Description Dialog Box \(p. 623\)](#), enable **Overlays** and click **Apply**.

3. Display the face zones with the appropriate bounds specified.

If you select the bounded face zones first and enable **Overlays**, you will need to deselect the face zones in the [Display Grid Dialog Box \(p. 596\)](#) before displaying the bounded cell zones.

4. To transform the entities displayed, select the mesh entities in the **Names** selection list in the **Scene Description** dialog box and click **Transform...** to open the [Transformations Dialog Box \(p. 626\)](#) for the selected entities.
 - To translate the selected entities, enter the translation distance in each direction in the X, Y, and Z real number fields under **Translate**.
 - To rotate the selected entities, enter the number of degrees by which to rotate about each axis in the X, Y, and Z integer number fields under **Rotate By**. You can enter any value between -360 and 360. By default, the rotation origin will be (0,0,0). If you want to spin an entity about its own origin, or about some other point, specify the X, Y, and Z coordinates of that point under **Rotate About**.
 - To scale the selected entities, enter the amount by which to scale in each direction in the X, Y, and Z real number fields under **Scale**. To avoid distortion of the shape, be sure to specify the same value for all three entries.

19.8.4. Text Commands for Scene Description

The following text commands are related to scene description:

/display/update-scene/draw-frame?
enables/disables the drawing of the bounding frame.

/display/update-scene/overlays?
enables/disables the overlays option.

/display/update-scene/select-geometry
allows you to select the geometry to be updated.

/display/update-scene/set-frame
allows you to change the frame options.

19.9. Controlling the Mouse Buttons

A convenient feature of ANSYS Fluent is that it allows you to assign a specific function to each of the mouse buttons. According to your specifications, clicking a mouse button in the graphics window will cause the appropriate action to be taken.

Important

3DConnexion Space products (Ball, Mouse, Pilot, and Navigator) are not supported with ANSYS Fluent.

19.9.1. Button Functions

19.9.2.Text Commands for Selecting Mouse Buttons

19.9.1.Button Functions

The predefined button functions available are as follows:

mouse-rotate

allows you to rotate the view by dragging the mouse across the screen. Dragging horizontally rotates the entity about the y-axis of the screen, vertical mouse movement rotates the entity about the x-axis of the screen. The function completes when the mouse button is released or the cursor leaves the graphics window.

mouse-dolly

allows you to translate the view by dragging the mouse while holding down the button. The function completes when the mouse button is released or the cursor leaves the graphics window.

mouse-zoom

allows you to draw a zoom box, anchored at the point at which the button is pressed, by dragging the mouse with the button held down. When you release the button:

- If the dragging was from left to right, a magnified view of the area within the zoom box will fill the window.
- If the dragging was from right to left, the area of the window shrinks to fit into the zoom box, resulting in a “zoomed out” view.
- If the mouse button is clicked (not dragged), the selected point becomes the center of the window.

mouse-roll-zoom

allows you to zoom in or out of the view by rolling the mouse while the button is pressed.

mouse-probe

allows you to perform the specified mouse probe function. The mouse probe function can be set in the **Mouse Probe** dialog box.

mouse-annotate

allows you to insert text into the graphics window. If you drag the mouse button an attachment line is drawn. When you release the button (after dragging or clicking), a cursor is displayed in the graphics window, and you can enter the text. Press **Enter** or move the cursor out of the graphics window.

To remove annotated text and attachment lines, use the **Clear** button in the [Annotate Dialog Box \(p. 630\)](#) or the `/display/clear-annotation` command, as described in [Annotating the Display \(p. 393\)](#).

These commands will delete all annotated text from the window.

19.9.2.Text Commands for Selecting Mouse Buttons

Text interface commands with the same functionality as the controls in the **Mouse Buttons** dialog box are:

/display/set/mouse-buttons

prompts you to select a function for each of the mouse buttons. The options available are:

mouse-rotate

allows you to rotate the view by dragging the mouse across the screen.

mouse-dolly

allows you to translate the view by dragging the mouse across the screen.

mouse-zoom

allows you to zoom in or out on the display.

mouse-roll-zoom

allows you to zoom in or out of the view by rolling the mouse while the button is pressed.

mouse-probe

allows you to perform the specified mouse probe function. The mouse probe function can be set to one of the values described for the /boundary/modify/select-probe command in [Text Commands for Boundary Modification \(p. 135\)](#).

mouse-annotate

allows you to insert text into the graphics window.

19.10. Controlling the Mouse Probe Functions

ANSYS Fluent allows you to assign a specific function to the mouse probe button. Click the mouse probe in the graphics window for the appropriate action to take.

[19.10.1. Text Commands for Mouse Probe Selection](#)

Text interface commands with the same functionality as the controls in the **Mouse Probe** dialog box are /boundary/modify/select-probe and /boundary/modify/select-filter.

19.11. Annotating the Display

Text annotations with optional attachment lines may be added to the graphics windows. The text is added to the window at a location chosen with the mouse, using the mouse-probe button (see [Controlling the Mouse Buttons \(p. 391\)](#) for information on setting the mouse buttons). Dragging with the mouse-probe button pressed will draw an attachment line from the point where the mouse was first clicked to the point where it was released. The annotation text will be placed at the point where the mouse button was released.

The annotation text is associated with the active graphics window; it is removed only when the annotations are explicitly cleared. You can edit the text in the graphics window's caption block by the left mouse button in the desired location. When a cursor appears, you can type the new text or delete the existing text. Text in the caption block will *not* be deleted when you clear annotations.

[19.11.1. Text Commands for Text Annotation](#)

Text commands related to annotation are:

/display/annotate

adds annotation text to a graphics window. It will prompt you for a string to use as the annotation text. A dialog will prompt you to select a screen location using the mouse-probe button on your mouse. For more information on setting the mouse buttons see [Controlling the Mouse Buttons \(p. 391\)](#).

/display/clear-annotations

removes all annotations and attachment lines from the active graphics window.

19.12. Shortcuts for Selecting Zones

The dialog boxes opened from the **Display** menu and some other dialog boxes contain lists of zones present in the model. You can choose one or more face, cell, node, or edge zones from these lists. If a large number of zones are present in the model, it becomes difficult to find and select the required zone.

The **Zone Selection Helper** dialog box allows you to easily find and select the zone(s) you want. You can either select the required zones based on their naming pattern, or by number of entities present in them, or using the minimum or maximum face zone area. Although the dialog box is opened from the **Display** menu, it can be used with *all* dialog boxes that contain zone lists (e.g., **Cell Zones** and **Boundary Zones** dialog boxes).

Text commands related to zones are:

/display/zones/display-neighbourhood

displays the zones which are in the neighborhood of the selected zone. The neighboring zones have to be in contact, or intersecting the selected zone.

/display/zones/display-similar-area

displays the zones with similar area to the selected zone area.

/display/zones/hide-zones

hides the selected zones in the display.

/display/zones/isolate-zones

isolates the selected zones in the display.

/display/zones/make-transparent

makes the geometry transparent so that internal zones are visible. This command works as a toggle undoing the transparency of the previously selected zones.

/display/zones/select-all-visible

selects all the visible zones in the graphics window.

/display/zones/show-all

unhides all the zones in the geometry and displays them.

/display/zones/toggle-color-mode

toggles the colors of the geometry. In one mode geometry is colored object-wise while in the other mode it is colored zone-wise.

/display/zones/toggle-color-palette

toggles the color palette of the geometry.

19.13. Setting Default tgvars

The **Display/Controls...** menu item opens the **Controls** dialog box to set file-specific variables called tgvars.

See [Display/Controls...](#) for additional information.

Chapter 20: Reporting Mesh Statistics

The quality of a mesh is determined more effectively by looking at various statistics, such as maximum skewness, rather than just performing a visual inspection. Unlike structured grids, unstructured grids are nearly impossible to comprehend with only a graphical plot.

The types of mesh information that can be reported include minimum, maximum, and average values of cell size, skewness, aspect ratio, or change in size, warp, squish, edge ratio, and the total number of each element type. These reporting operations are described in the following sections.

[20.1. Reporting the Mesh Size](#)

[20.2. Reporting Face Limits](#)

[20.3. Reporting Cell Limits](#)

[20.4. Reporting Boundary Cell Limits](#)

[20.5. Mesh Quality](#)

[20.6. Printing Grid Information](#)

[20.7. Additional Text Commands for Reporting](#)

20.1. Reporting the Mesh Size

The **Report Mesh Size** dialog box and associated text commands allow you to check the size of your mesh by reporting the number of nodes, faces, and cells in it.

[20.1.1. Text Commands for Reporting Mesh Size](#)

20.1.1. Text Commands for Reporting Mesh Size

Text commands for reporting the size of the mesh are as follows:

/report/mesh-size

reports the number of nodes, faces, and cells.

/report/number-meshed

reports the number of elements that have been meshed.

20.2. Reporting Face Limits

Before generating a volume mesh, check the quality of the faces to get an indication of the overall mesh quality. The default quality measure is skewness, but you can also report the aspect ratio limits or the range in size change for a face zone. For 3D meshes, a maximum less than 0.9 and an average of 0.4 are good. The lower the maximum skewness, the better the mesh. See [Mesh Quality \(p. 399\)](#) for information on face and cell quality measures available.

You can check face size and quality limits using the **Report Face Limits** dialog box or the associated text commands. Before creating a layer of pyramid cells, check the aspect ratio of the quadrilateral faces that will form the base of the pyramids. This aspect ratio should be less than 8 else, the triangular faces of the pyramids will be highly skewed. If the aspect ratio is greater than 8, regenerate the quadrilateral faces.

- If they were created in a different preprocessor, return to that application and try to reduce the aspect ratio of the faces in question.
- If they were created during the building of prism layers, rebuild the prisms using a more gradual growth rate.

Check the skewness of triangular faces on a boundary from which you are going to build prisms, to ensure that the quality of the prism cells will be good. After you create the prisms, check the skewness of the triangular faces that were created during the prism generation.

20.2.1.Text Commands for Reporting Face Limits

Text commands with the same functionality as the controls in the **Report Face Limits** dialog box are:

/report/face-size-limits

reports the face size limits.

/report/face-quality-limits

reports the face quality limits.

20.3. Reporting Cell Limits

You can report the quality of the cells in your volume mesh. The default quality measure is skewness, but you can also report the aspect ratio limits or the range of change in cell size. You can check cell size and quality limits using the **Report Cell Limits** dialog box or the associated text commands.

20.3.1.Text Commands for Reporting Cell Limits

Text commands with the same functionality as the controls in the **Report Cell Limits** dialog box are listed below:

/report/cell-size-limits

reports the cell size limits.

/report/cell-quality-limits

reports the cell quality limits.

/report/neighborhood-quality

reports the maximum skewness, aspect ratio, or size change of all cells using a specified node.

20.4. Reporting Boundary Cell Limits

For boundary layer flows, boundary cells with low skewness are very important.

For tetrahedral meshes, the maximum skewness will not be lower than the maximum face skewness, because the boundary cell skewness is limited by the boundary face skewness.

Highly skewed cells with two boundary faces can be removed using the **Tet Improve** dialog box.

To check the skewness of boundary cells, use the **Report Boundary Cell Limits** dialog box or the associated text commands.

20.4.1.Text Commands for Reporting Boundary Cell Limits

20.4.1. Text Commands for Reporting Boundary Cell Limits

The following command is associated with reporting boundary cell limits:

/report/boundary-cell-quality

reports the quality of boundary cells. (If you specify zero for number of boundary faces, you will be prompted for number of boundary nodes.)

20.5. Mesh Quality

The quality of the mesh guarantees the best analysis results for the problem, minimizes the need for additional analysis runs, and improves one's predictive capabilities.

Mesh quality is determined by some measures which are described briefly in this section. The basic measures include clustering, smoothness, skewness, and aspect ratio. The default quality measure is skewness.

You can check the quality of faces and cells using the following dialog boxes:

- **Report Face Limits** dialog box
- **Report Cell Limits** dialog box
- **Report Boundary Cell Limits** dialog box

To specify the quality measure (skewness, aspect ratio, change in size, etc.), use the **Quality Measure** dialog box.

[20.5.1. Quality Measures](#)

[20.5.2. Text Commands for Selecting the Quality Measure](#)

20.5.1. Quality Measures

Clustering

The requirement for clustering is that the mesh be fine enough to resolve the primary features of the flow being analyzed. The resolution depends on the boundary mesh that you start from and also the parameters controlling the generation of the interior mesh.

Smoothness

In a high-quality mesh, the change in size from one face or cell to the next should be gradual (smooth). Large differences in size between adjacent faces or cells will result in a poor computational grid because the differential equations being solved assume that the cells shrink or grow smoothly.

Aspect Ratio

The aspect ratio of a face or cell is the ratio of the longest edge length to the shortest edge length. The aspect ratio applies to triangular, tetrahedral, quadrilateral, and hexahedral elements and is defined differently for each element type.

The aspect ratio can also be used to determine how close to ideal a face or cell is.

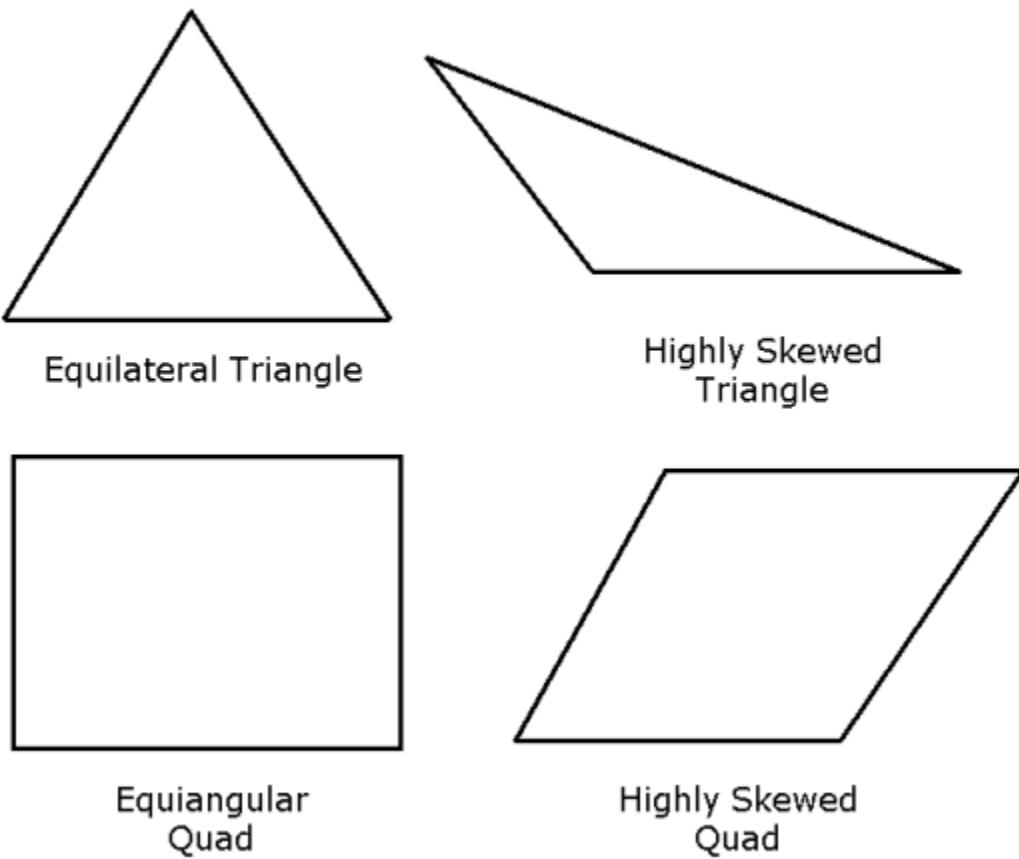
- For an equilateral face or cell (e.g., an equilateral triangle or a square, etc.), the aspect ratio will be 1.

- For less regularly-shaped faces or cells, the aspect ratio will be greater than 1, since the edges differ in length.
- For triangular faces and tetrahedral cells and for pyramids, you can usually focus on improving the skewness, and the smoothness and aspect ratio will consequently be improved as well.
- For prisms, it is important to check the aspect ratio and/or the change in size in addition to the skewness, because it is possible to have a large jump in cell size between two cells with low skewness or a high-aspect-ratio low-skew cell.

Skewness

Skewness is one of the primary quality measures for a mesh. Skewness determines how close to ideal (i.e., equilateral or equiangular) a face or cell is (see [Figure 20.1: Ideal and Skewed Triangles and Quadrilaterals \(p. 400\)](#)).

Figure 20.1: Ideal and Skewed Triangles and Quadrilaterals



[Table 20.1: Skewness Ranges and Cell Quality \(p. 400\)](#) lists the range of skewness values and the corresponding cell quality.

Table 20.1: Skewness Ranges and Cell Quality

Skewness	Cell Quality
1	degenerate
0.9-<1	bad (sliver)

Skewness	Cell Quality
0.75–0.9	poor
0.5–0.75	fair
0.25–0.5	good
>0–0.25	excellent
0	equilateral

According to the definition of skewness, a value of 0 indicates an equilateral cell (best) and a value of 1 indicates a completely degenerate cell. Degenerate cells (slivers) are characterized by nodes that are nearly coplanar. Cells with a skewness value above 1 are invalid.

Highly skewed faces and cells should be avoided since they can lead to less accurate results than when relatively equilateral/equiangular faces and cells are used.

Two methods for measuring skewness are:

- Based on the equilateral volume (applies only to tetrahedra).
- Based on the deviation from a normalized equilateral angle. This method applies to all cell and face shapes, e.g., pyramids and prisms.

The default skewness method for tetrahedra is the equilateral volume method. But you can change to the angle deviation method using the **Quality Measure** dialog box.

Equilateral-Volume-Based Skewness

In the equilateral volume deviation method, skewness is defined as

$$\text{Skewness} = \frac{\text{Optimal Cell Size} - \text{Cell Size}}{\text{Optimal Cell Size}} \quad (20.1)$$

where, the optimal cell size is the size of an equilateral cell with the same circumradius.

Quality 3D meshes have a skewness value of approximately 0.4. [Table 20.1: Skewness Ranges and Cell Quality \(p. 400\)](#) provides a general guide to the relationship between cell skewness and quality.

In 3D meshes, most cells should be good or better, but a small percentage will generally be in the fair range and there are usually even a few poor cells. The presence of poor cells can indicate poor boundary node placement. You should try to improve your boundary mesh as much as possible, because the quality of the overall mesh can be no better than that of the boundary mesh.

Normalized Equiangular Skewness

In the normalized angle deviation method, skewness is defined (in general) as

$$\max \left[\frac{\theta_{\max} - \theta_e}{180 - \theta_e}, \frac{\theta_e - \theta_{\min}}{\theta_e} \right] \quad (20.2)$$

where

θ_{\max} = largest angle in the face or cell

θ_{\min} = smallest angle in the face or cell

θ_e = angle for an equiangular face/cell (e.g., 60 for a triangle, 90 for a quad, etc.)

For a pyramid, the cell skewness will be the maximum skewness computed for any face. An ideal pyramid (skewness = 0) is one in which the 4 triangular faces are equilateral (and equiangular) and the quadrilateral base face is a square. The guidelines in Table [Table 20.1: Skewness Ranges and Cell Quality \(p. 400\)](#) apply to the normalized equiangular skewness as well.

Size Change

Size change is the ratio of the area (or volume) of a cell in the geometry to the area (or volume) of each neighboring face (or cell). This ratio is calculated for every face (or cell) in the domain. The minimum and maximum values are reported for the selected zones.

Squish

Squish is a measure used to quantify the non-orthogonality of a cell with respect to its faces. It is defined as follows:

$$1 - (A \cdot r_c) / |r_c| \quad (20.3)$$

where

A = face unit area vector

r_c = the vector connecting the adjacent cell centroids (for face squish) or the cell and face centroid (for cell squish)

Edge Ratio

The edge ratio is defined as the ratio of maximum length of the edge of the element to the minimum length of the edge of the element.

By definition, edge ratio is always greater than or equal to 1. The higher the value of the edge ratio, the less regularly shaped is its associated element. For equilateral element shapes, the edge ratio is always equal to 1.

Warp

Face warp applies only to quadrilateral elements and is defined as the variation of normals between the two triangular faces that can be constructed from the quadrilateral face. The actual value is the maximum of the two possible ways triangles can be created.

Mathematically, it is expressed as follows:

$$\frac{Z}{\min [a, b]} \quad (20.4)$$

where

Z = the deviation from a best-fit plane that contains the element

a, b = the lengths of the line segments that bisect the edges of the element

The value of face warp ranges between 0 and 1. A value of 0 specifies an equilateral element and a value of 1 specifies a highly skewed element.

Dihedral Angle

Dihedral angle applies only to the faces and not cells. It highlights the faces with the dihedral angle between them in the range specified in the **Cell Quality Range** group box in the **Cells** tab of the **Display Grid** dialog box.

You can specify a range of values from 0–180. This quality measure is useful in locating the sharp corners in complicated geometries.

ICEM CFD Quality

The ICEM CFD quality measure calculates element quality based on the quality values in ANSYS ICEM CFD. This measure is only available for cells and not faces.

- Tetrahedra: The quality is calculated as the skewness of the tetrahedral element.

The remaining quality values are based on various quality metrics computed in ANSYS ICEM CFD.

$$\text{Quality} = 1 - \text{ANSYS ICEM CFD Quality} \quad (20.5)$$

- Hexahedra: The quality is based on the Determinant, Max Orthogls, and Max Warppls metrics in ANSYS ICEM CFD.

- The determinant is the ratio of the smallest and the largest determinant of the Jacobian matrices, where a Jacobian matrix is computed at each node of the element.
- The max orthogls metric calculates the maximum deviation of the internal angles of the element from 90 degrees. Angles between 180 and 360 degrees (deviation up to 270 degrees) will also be considered.
- The warp for each face is calculated as the maximum angle between the triangles connected at the diagonals of the face. The max warppls metric is calculated as the maximum warp of the faces comprising the element.

These values are normalized and the minimum value of the three normalized diagnostics will be used.

- Pyramids: The quality is based on the Determinant computed in ANSYS ICEM CFD.

The determinant is the ratio of the smallest and the largest determinant of the Jacobian matrices, where a Jacobian matrix is computed at each node of the element.

- Prisms: The quality is based on the Determinant and Warp computed in ANSYS ICEM CFD.

- The determinant is the ratio of the smallest and the largest determinant of the Jacobian matrices, where a Jacobian matrix is computed at each node of the element.
- The warp for each face is calculated as the maximum angle between the respective edges and a plane containing the mid-points of the edges comprising the face. The warp for the element is then calculated as the maximum warp of the faces comprising the element.

The quality is calculated as the minimum of the determinant and warp.

For further details on the quality measures available in ANSYS ICEM CFD, refer to the ANSYS ICEM CFD Help Manual.

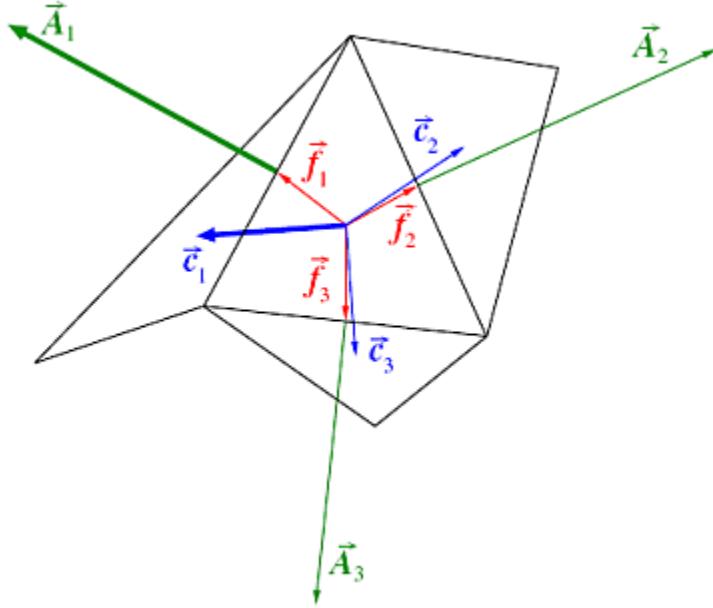
The range of quality values reported is between 0–2 (the range in ANSYS ICEM CFD is -1 to 1, and a value of 2 corresponds to -1 in ANSYS ICEM CFD, 1 to 0, and 0 to 1, respectively). A value of 0 indicates

a perfect, non-distorted element, 1 indicates a degenerate element, and values above 1 indicate an invalid (e.g., concave, dihedral angle > 180 degree, or negative volume) element.

Ortho Skew

The ortho skew quality for cells is computed using the face normal vector, the vector from the cell centroid to the centroid of each of the adjacent cells, and the vector from the cell centroid to each of the faces. [Figure 20.2: Vectors Used to Compute Ortho Skew Quality for a Cell \(p. 404\)](#) illustrates the vectors used to determine the ortho skew quality for a cell.

Figure 20.2: Vectors Used to Compute Ortho Skew Quality for a Cell



The ortho skew for a cell is computed as the maximum of the following quantities computed for each face i :

$$1 - \frac{\vec{A}_i \cdot \vec{f}_i}{|\vec{A}_i| |\vec{f}_i|} \quad (20.6)$$

where \vec{A}_i is the face normal vector and \vec{f}_i is a vector from the centroid of the cell to the centroid of that face

and

$$1 - \frac{\vec{A}_i \cdot \vec{c}_i}{|\vec{A}_i| |\vec{c}_i|} \quad (20.7)$$

where \vec{A}_i is the face normal vector and \vec{c}_i is a vector from the centroid of the cell to the centroid of the adjacent cell that shares the face.

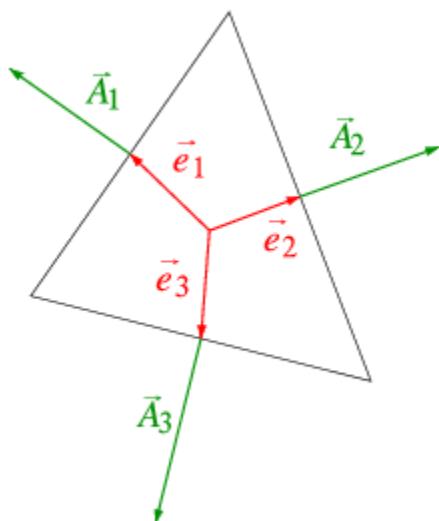
Note

- When the cell is located on the boundary, the vector \vec{c}_i across the boundary face is ignored during the quality computation.

- When the cell is separated from the adjacent cell by an internal wall, the vector \vec{c}_i across the internal boundary face is ignored during the quality computation.
- When the adjacent cells share a parent-child relation, the vector \vec{f}_i is the vector from the cell centroid to the centroid of the child face while the vector \vec{c}_i is the vector from the cell centroid to the centroid of the adjacent child cell sharing the child face.

The ortho skew quality for faces is computed using the edge normal vector and the vector from the face centroid to the centroid of each edge. [Figure 20.3: Vectors Used to Compute Ortho Skew Quality for a Face \(p. 405\)](#) illustrates the vectors used to determine the ortho skew quality for a face.

Figure 20.3: Vectors Used to Compute Ortho Skew Quality for a Face



The ortho skew for a face is computed as the maximum of the following quantity computed for each edge i :

$$1 - \frac{\vec{A}_i \cdot \vec{e}_i}{|\vec{A}_i| |\vec{e}_i|} \quad (20.8)$$

where \vec{A}_i is the edge normal vector and \vec{e}_i is a vector from the centroid of the face to the centroid of the edge.

Note

The ortho skew measure is equivalent to the orthogonal quality in ANSYS Fluent solution mode and ANSYS Meshing, except that the scale is reversed:

Ortho Skew = 1 – Orthogonal Quality

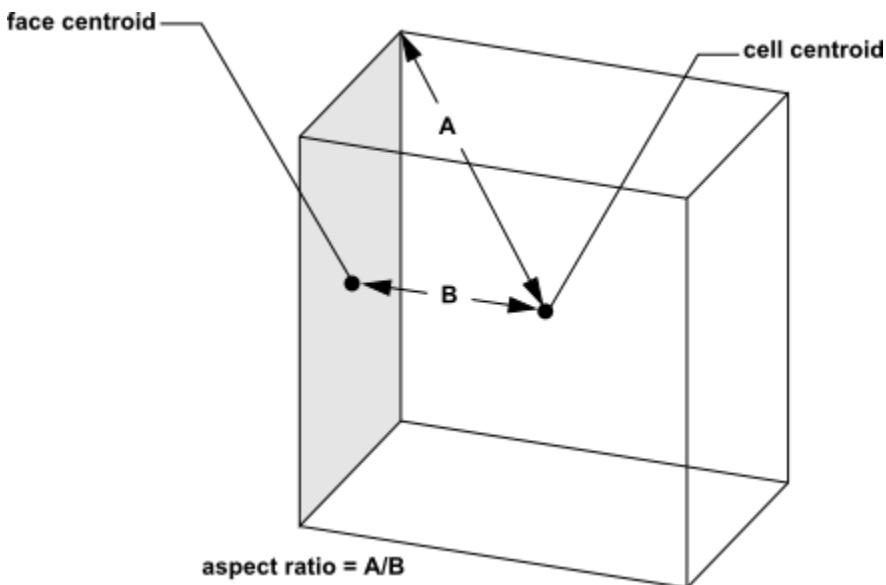
The ortho skew values may not correspond exactly with the orthogonal quality values in solution mode as the computation depends on boundary conditions on internal surfaces (**wall** vs. **interior / fan / radiator / porous-jump**). Also, for CutCell meshes, the elements are

"traditional" (hex/tet/wedge/pyramid) elements while in solution mode, meshes contain polyhedral cells.

Fluent Aspect Ratio

The Fluent aspect ratio is a measure of the stretching of a cell. It is computed as the ratio of the maximum value to the minimum value of any of the following distances: the normal distances between the cell centroid and face centroids, and the distances between the cell centroid and nodes. For a unit cube (see [Figure 20.4: Calculating the Fluent Aspect Ratio for a Unit Cube \(p. 406\)](#)), the maximum distance is 0.866, and the minimum distance is 0.5, so the aspect ratio is 1.732. This type of definition can be applied on any type of mesh, including polyhedral.

Figure 20.4: Calculating the Fluent Aspect Ratio for a Unit Cube



20.5.2. Text Commands for Selecting the Quality Measure

The following text command can be used for specifying the quality measure:

/report/quality-method
specifies the method to be used for reporting face and cell quality.

20.6. Printing Grid Information

The `/report/print-info` text command allows you to obtain information about individual components of the mesh. This command also appears in the boundary menu. When you use this command, you will be prompted for an "entity" (i.e., a node, face, or cell). An entity name consists of a prefix and an index.

The valid prefixes are: `bn` (boundary node), `n` (node), `bF` (boundary face), `F` (face), and `c` (cell). Hence, the name of the first boundary node would be `bn1`. If the first node is a boundary node, then both `bn1` and `n1` refer to the same node.

The output for each type of entity is as explained in this section.

20.6.1. Boundary Node

20.6.2. Node

[20.6.3. Boundary Face](#)

[20.6.4. Cell](#)

[20.6.5. Face](#)

20.6.1. Boundary Node

```
/report > print-info
entity [ ] bnl
bnl = (3 (0 0 0) 0.1 (bf2099 bf1093 bf193))
```

The following information is listed for the boundary node:

- associated zone ID
- Cartesian coordinates of the node
- node radius
- faces using the node.

For boundary nodes, the node radius is the average distance to neighboring boundary nodes.

20.6.2. Node

```
/report > print-info
entity [ ] n30
n30 = (5 (2.433799 0.078610075 0.52846689) 0.12702106)
```

The following information is listed for a node:

- associated zone ID
- Cartesian coordinates of the node
- the node radius

For interior nodes, the node radius is the distance-weighted average of the surrounding nodes.

20.6.3. Boundary Face

```
/report > print-info
entity [ ] bf17
bf17 = (2 (bn1308 bn1314 bn1272) (( c1533) () 0.014393554))
```

The following information is listed for a boundary face:

- associated zone ID
- the nodes forming the face
- the neighboring cells
- the periodic shadow (null if not periodic)
- the quality of the face

The default measure of quality is skewness, but you can use the **Quality Measure** dialog box to specify aspect ratio or change in size instead.

20.6.4. Cell

```
/report > print-info
entity [ ] c26
c26 = (1 (n262 n34 bn204 bn205) (f4743 f5372 f1822 f3426)
0.00020961983 (2.7032523 0.32941867 0.072823988)
0.0081779587 0.44769606)
```

The following information is listed for a cell:

- zone ID
- the nodes forming the cell
- the faces forming the cell
- the size
- the coordinates of the circumcenter (relevant for tetrahedral cells only)
- square of the circumcenter radius (relevant for tetrahedral cells only)
- the quality of the cell

The default measure of quality is skewness, but you can use the **Quality Measure** dialog box to specify aspect ratio or change in size instead.

20.6.5. Face

```
/report > print-info
entity [ ] f32
f32 = (4 (n95 bn197 n90) (c4599 c1279))
```

The following information is listed for a face:

- associated zone ID
- the nodes forming the face
- the neighboring cells

Due to the manner in which the algorithm maintains memory, not all indices will have values—that is, an empty slot can be caused by a delete operation. Empty slots are reused in subsequent operations, so the actual entity at a particular index may change as the mesh is generated.

20.7. Additional Text Commands for Reporting

Text interface commands related to the **Report** menu that are not available in the GUI are listed below:

```
/report/cell-distribution
    reports the distribution of cell quality.

/report/cell-zone-at-location
    reports the cell zone at the specified location.

/report/cell-zone-volume
    reports the volume of the specified cell zone in the console.
```

/report/edge-size-limits

reports the edge size limits.

/report/face-distribution

reports the distribution of face quality.

/report/face-zone-area

reports the area of the specified face zone in the console.

/report/face-zone-at-location

reports the face zone at the specified location.

/report/list-cell-quality

reports a list of cells with the specified quality measure within a specified range.

/report/memory-usage

reports the amount of memory used for all nodes, faces, and cells, and the total memory allocated.

/report/mesh-statistics

writes mesh statistics (such as range of quality, range in size, and number of cells, faces, and nodes) to an external file.

/report/unrefined-cells

reports the number of cells that have not been refined.

/report/update-bounding-box

updates the extents of the bounding box.

/report/verbosity-level

specifies how much information should be displayed during mesh initialization and refinement and other operations. Changing the value to 2 from the default value of 1 will result in more messages while changing it to 0 will disable all messages.

Chapter 21: Task Page, Menu, and Dialog Box Reference Guide

This reference guide provides detailed information about the task page, menus, and dialog boxes available in meshing mode in ANSYS Fluent.

- [21.1. Mesh Generation Task Page](#)
- [21.2. File Menu](#)
- [21.3. Boundary Menu](#)
- [21.4. Mesh Menu](#)
- [21.5. Display Menu](#)
- [21.6. Report Menu](#)
- [21.7. Parallel Menu](#)
- [21.8. View Menu](#)
- [21.9. Help Menu](#)
- [21.10. Hot Key Activated Dialog Boxes](#)

21.1. Mesh Generation Task Page

The **Mesh Generation** task page allows you to perform various operations related to mesh generation and display.



Controls

Create

contains controls for creating bounding box, cylinder, or capping surfaces; and defining size functions, material points, user-defined groups, and objects for generating the mesh.

Box...

opens the [Bounding Box Dialog Box \(p. 416\)](#), where you can create bounding box surfaces.

Cylinder...

opens the [Cylinder Dialog Box \(p. 417\)](#), where you can create cylindrical surfaces.

Caps...

opens the [Capping Surface Dialog Box \(p. 420\)](#), where you can create capping surfaces of the appropriate type as wrap objects.

Size Functions...

opens the [Size Functions Dialog Box \(p. 421\)](#), where you can define the size functions for generating the mesh. You can also compute the size field based on the parameters.

Material Point...

opens the [Material Points Dialog Box \(p. 426\)](#), where you can define material points.

Groups...

opens the [User Defined Groups Dialog Box \(p. 428\)](#), where you can create groups of surfaces or edges, and activate or update them.

Objects...

opens the [Manage Objects Dialog Box \(p. 430\)](#), where you can define and manipulate the objects for generating the mesh.

Objects

contains a list of the objects available.

Object Types

contains the list of the object types. If you select an object type from the list, all objects of that type will be selected in the **Objects** list. You can also select multiple types to select all objects of different types (e.g., **geom** and **wrap**).

Draw

displays the selected object(s) in the graphics window.

List

reports (in the console) the object name, cell zone type, priority, object type, and the face and edge zones comprising the object.

Delete

deletes the object(s) selected in the **Objects** selection list.

Surface Mesh

contains controls for manipulating the surface mesh comprising the wrap objects.

Diagnostics...

opens the [Diagnostic Tools Dialog Box \(p. 437\)](#) allowing you to look for problems caused by geometry, face connectivity, or mesh quality issues.

Fix Holes...

opens the [Fix Holes Dialog Box \(p. 448\)](#) where you can locate and fix holes in the objects.

Remove Gaps...

opens the [Remove Gaps Dialog Box \(p. 452\)](#), where you control the behavior across areas of discontinuity, for example close gaps or remove thickness in the wrap objects.

Wrap...

opens the [Wrap Dialog Box \(p. 454\)](#), where you can create the wrap object from the existing geometry objects. The wrap object created is a well-connected, conformal representation of the geometry, suitable for further repair operations, if needed.

Sew...

opens the [Sew Dialog Box \(p. 457\)](#), where you can combine multiple wrap objects to create a well-connected boundary mesh. The boundary mesh zones created by the sew operation will comprise a mesh object.

Improve...

opens the [Improve Dialog Box \(p. 459\)](#), where you can create a mesh object from an existing wrap object and improve the surface mesh quality of the mesh object.

Build Topology...

opens the [Build Topology Dialog Box \(p. 460\)](#) where you can perform local sew (i.e., join/intersect) operations to produce a fully connected mesh for volume meshing.

Volume Mesh

contains options for generating the volume mesh.

Auto Mesh...

opens the [Auto Mesh Dialog Box \(p. 464\)](#), where you can select the mesh element types and set the relevant parameters.

CutCell...

opens the [CutCell Dialog Box \(p. 466\)](#), where you can perform operations relevant to generating the CutCell mesh.

Cleanup

performs a cleanup operation after the volume mesh has been generated. Operations such as deleting dead zones, deleting geometry/wrap objects, deleting edge zones, removing face/cell zone name prefixes and/or suffixes, deleting unused faces and nodes are performed during the cleanup operation.

Overlay

allows you to superimpose the selected object(s) and zone(s) on the object(s) and zone(s) currently displayed in the graphics window.

Bounds

contains controls which allow you to crop the display of your model in the graphics window.

You may insert up to six **cutplanes** (two in each of x-, y-, and z- direction), and control their location, using the controls in this panel

Selection

is used to specify an entity on which the Cutplane ranges are centered. You can select entities with the mouse in the graphics window if the mouse probe function (see [Controlling the Mouse Probe Functions \(p. 393\)](#)) is currently **select** and the **filter** is set to **node, edge, face, cell, zone, position, or object**.

+/- Delta

specifies the upper and lower limit of the displayed range (offset of the cutplanes) relative to the selected entity.

X Range

inserts Cutplanes in the X-coordinate direction.

Y Range

inserts Cutplanes in the Y-coordinate direction.

Z Range

inserts Cutplanes in the Z-coordinate direction.

Set Ranges

computes the location of the cutplanes as specified by **Selection** and **+/- Delta**, and then sets the **X Range**, **Y Range**, and **Z Range** check boxes.

Note

X Range, Y Range, and Z Range may be selected or deselected independently.

Reset

removes any cutplanes and restores the controls to their default settings (i.e., no limits on the X, Y, or Z range).

Use Cutplanes as bounds?

If checked, the cutplane ranges are linked to the values in the **Bounds** tab in the [Display Grid Dialog Box \(p. 596\)](#), and may be set in either location. If unchecked, the cutplane ranges are not linked to the **Bounds** ranges in the Display Grid dialog box.

Insert Clipping Planes

allows you to insert clipping planes on only one of x-, y-, or z-directions.

Note

The **Use Cutplanes as bounds** box must be unchecked to select **Insert Clipping Planes**.

Clipping Planes

is activated when the **Insert Clipping Planes** box is checked, allowing you to control the location of the clipping planes using the slide control. The extent of the display range is as set on the **Bounds** tab in the [Display Grid Dialog Box \(p. 596\)](#).

Limit in X

Slide to clip the graphics display in the X direction. The slider extents are the minimum and maximum **Bounds** in the X direction.

Limit in Y

Slide to clip the graphics display in the Y direction. The slider extents are the minimum and maximum **Bounds** in the Y direction.

Limit in Z

Slide to clip the graphics display in the Z direction. The slider extents are the minimum and maximum **Bounds** in the Z direction.

Note

Only one Limit radio button may be selected at one time.

Flip Clipping Planes?

Reverses the direction of movement of the clipping plane relative to the slider motion, allowing you to inspect the other side of the clipped geometry.

Tree View for List

contains options for specifying a separator symbol to determine the common prefix for items listed in the selection lists, when the tree view is used.

Separator

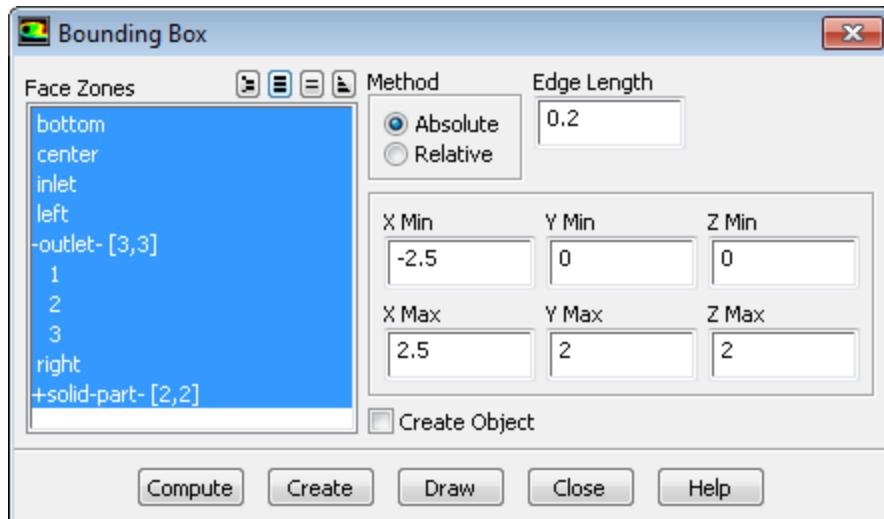
specifies the separator symbol to be used for selection list items.

Set

sets the specified character as the separator for selection list items.

21.1.1. Bounding Box Dialog Box

The **Bounding Box** dialog box allows you to create a box that encloses the input geometry.

**Face Zones**

contains a list of existing face zones in the geometry.

Method

contains options for selecting the method for creating bounding box.

Absolute

allows you to create a bounding box by specifying the extents along the coordinate axes.

Relative

allows you to create a bounding box by specifying the clearance values from the boundaries of the selected face zones.

Edge Length

allows you to specify the maximum size of the triangular cells that you want to create for the bounding box.

X Min, Y Min, Z Min, X Max, Y Max, Z Max

specify the extents of the bounding box when the **Absolute** method is selected.

Delta X Min, Delta Y Min, Delta Z Min, Delta X Max, Delta Y Max, Delta Z Max

specify the clearance values for the bounding box extents when the **Relative** method is selected.

Create Object

allows you to create a wrap object based on the bounding box face zone created. The wrap object created comprises the separated six face zones of the bounding box and the corresponding edges. The object cell zone type will be set to **fluid** and the priority will be set to $p-1$, where p is the lowest priority specified for the current object(s).

Compute

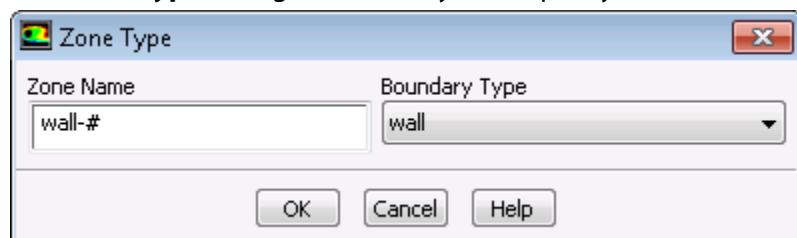
computes the bounding box extents based on the face zone(s) selected when the **Absolute** method is used.

Create

opens the [Zone Type Dialog Box \(p. 417\)](#), where you can specify the zone name and type for the bounding box zone created.

21.1.1.1. Zone Type Dialog Box

The **Zone Type** dialog box allows you to specify the zone name and type for the boundary zone created.

**Controls****Zone Name**

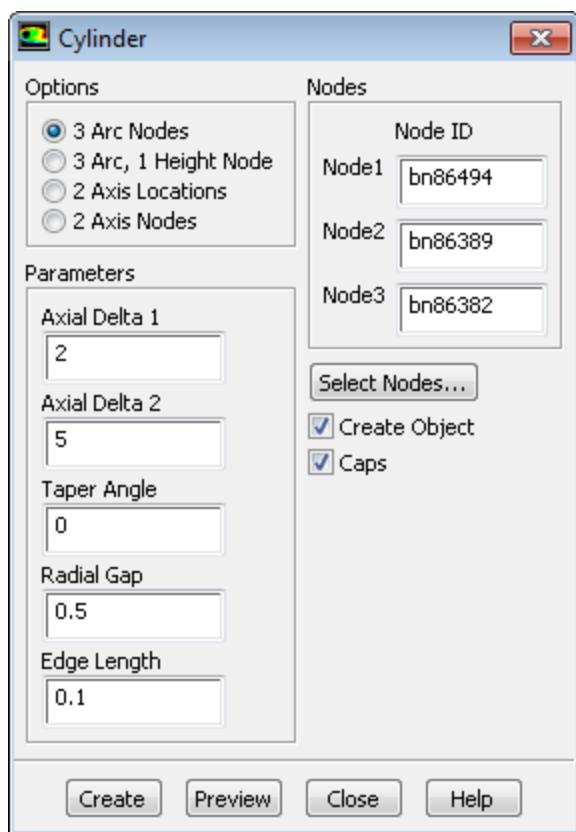
specifies the name for the boundary zone created. The default name that appears is the selected **Boundary Type** followed by the assigned zone ID number. For example, if the selected **Boundary Type** is **wall**, the default **Zone Name** will be **wall-#**, indicating that the new boundary zone ID number should be appended to "wall" to name the zone (e.g., wall-12). You can enter a name that does not include the ID if you prefer (e.g., **inside-wall**).

Boundary Type

allows you to set the type for the boundary zone created. The drop-down list comprises all available boundary zone types. When you select an item from this list, the **Zone Name** will be updated to reflect the new **Boundary Type**.

21.1.2. Cylinder Dialog Box

The **Cylinder** dialog box allows you to create a cylindrical surface mesh.



Controls

Options

contains options for defining the cylinder.

3 Arc Nodes

allows you to create the cylinder/frustum by specifying three arc nodes, along with the axial length, taper angle, radial gap, and edge length to be used.

3 Arc, 1 Height Node

allows you to create the cylinder/frustum by specifying three arc nodes and a node to determine the height. The radii and taper angle will be determined based on the nodes selected.

2 Axis Locations

allows you to create the cylinder/frustum by specifying the locations (**X Pos**, **Y Pos**, **Z Pos**) of the points **P1** and **P2**, defining the axis, along with the radii and edge length to be used.

2 Axis Nodes

allows you to create the cylinder/frustum by specifying the nodes corresponding to the points defining the axis along with the radii and edge length to be used.

Nodes

specifies the nodes selected for the **3 Arc Nodes**, **3 Arc, 1 Height Node**, and **2 Axis Nodes** methods.

Select Nodes...

allows you to select the nodes for the **3 Arc Nodes**, **3 Arc, 1 Height Node**, and **2 Axis Nodes** methods using the mouse. When you click the **Select Nodes...** button, a **Working** dialog box will appear, prompting you to select the nodes.

Locations

specifies the locations selected for the **2 Axis Locations** method.

Select Points...

allows you to select the points defining the axis for the **2 Axis Locations** method using the mouse. When you click the **Select Points...** button, a **Working** dialog box will appear, prompting you to select the points.

Parameters

contains parameters to be specified for creating the cylindrical surface mesh.

Axial Delta 1, Axial Delta 2

determine the axial length of the cylinder for the **3 Arc Nodes** method.

Taper Angle

specifies the taper angle, which is used to determine the radii of the cylinder/frustum for the **3 Arc Nodes** method.

Radial Gap

determines the radii of the cylinder/frustum to be created using the **3 Arc Nodes** method. The actual radii will be determined based on the radial gap and taper angle specified.

Radius1, Radius2

specifies the radii of the cylinder/frustum to be created by the **2 Axis Locations** and **2 Axis Nodes** methods.

Edge Length

specifies the size of the surface mesh to be created for the cylinder.

Create Object

allows you to create a wrap object based on the cylinder face zone(s) created. The wrap object created comprises the cylindrical surface and the cap surfaces (if **Caps** is enabled) of the cylinder and the corresponding edges. The object cell zone type will be set to fluid and the priority will be set to $p-1$, where p is the lowest priority specified for the current object(s).

Caps

allows you to create the circular capping surfaces along with the cylindrical surface.

Preview

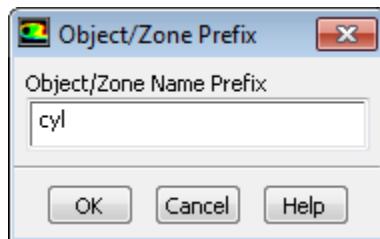
allows you to preview the cylinder/frustum to be created.

Create

opens the [Object/Zone Prefix Dialog Box \(p. 419\)](#), where you can specify the prefix for the zones to be created.

21.1.2.1. Object/Zone Prefix Dialog Box

The **Object/Zone Prefix** dialog box allows you to specify the prefix for the object/zones created (e.g., **cyl**) using the [Cylinder Dialog Box \(p. 417\)](#).



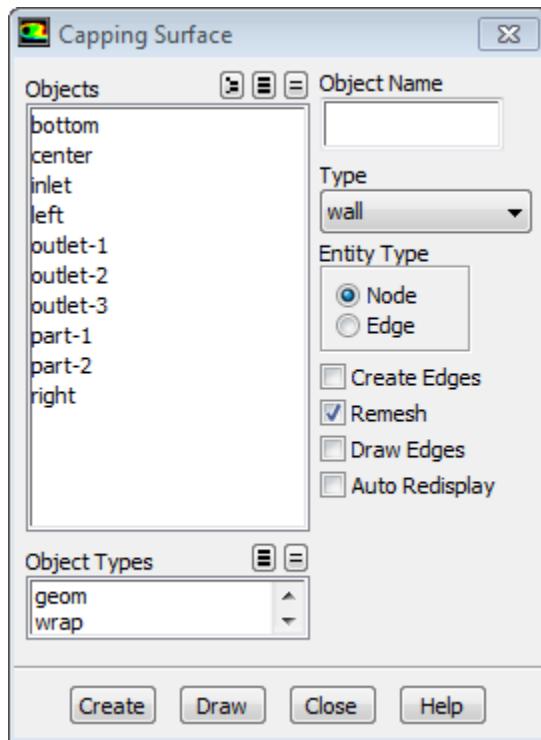
Controls

Object/Zone Name Prefix

specifies the prefix to be used for the object/zones being created.

21.1.3. Capping Surface Dialog Box

The **Capping Surface** dialog box contains options that allow you to create capping surfaces of the appropriate type as wrap objects.



Controls

Objects

contains a list of existing objects.

Object Types

contains the list of the object types. If you select an object type from the list, all objects of that type will be selected in the **Objects** list. You can also select multiple types to select all objects of different types (e.g., **geom** and **wrap**).

Object Name

specifies the name for the capping object to be created.

Type

specifies the boundary zone type to be assigned to the capping surface created.

Entity Type

allows you to select the appropriate entity for creating the capping surface.

Node

creates the capping surface based on the nodes selected in the graphics window.

Edge

creates the capping surface based on the edge selected in the graphics window.

Create Edges

allows you create edges from the capping surface created. The extracted edges will be included in the wrap object created for the cap surface.

Remesh

remeshes the capping surface to obtain better mesh quality.

Draw Edges

allows you to display the edges included in the object(s) to be drawn.

Auto Redisplay

toggles the automatic update of the display after an operation is performed. If the **Auto Redisplay** option is enabled, the mesh will be automatically displayed after you make a change to the boundary, allowing you to immediately see the effect of the operation performed. If **Auto Redisplay** is disabled, click **Draw** to see of the operation performed.

Create

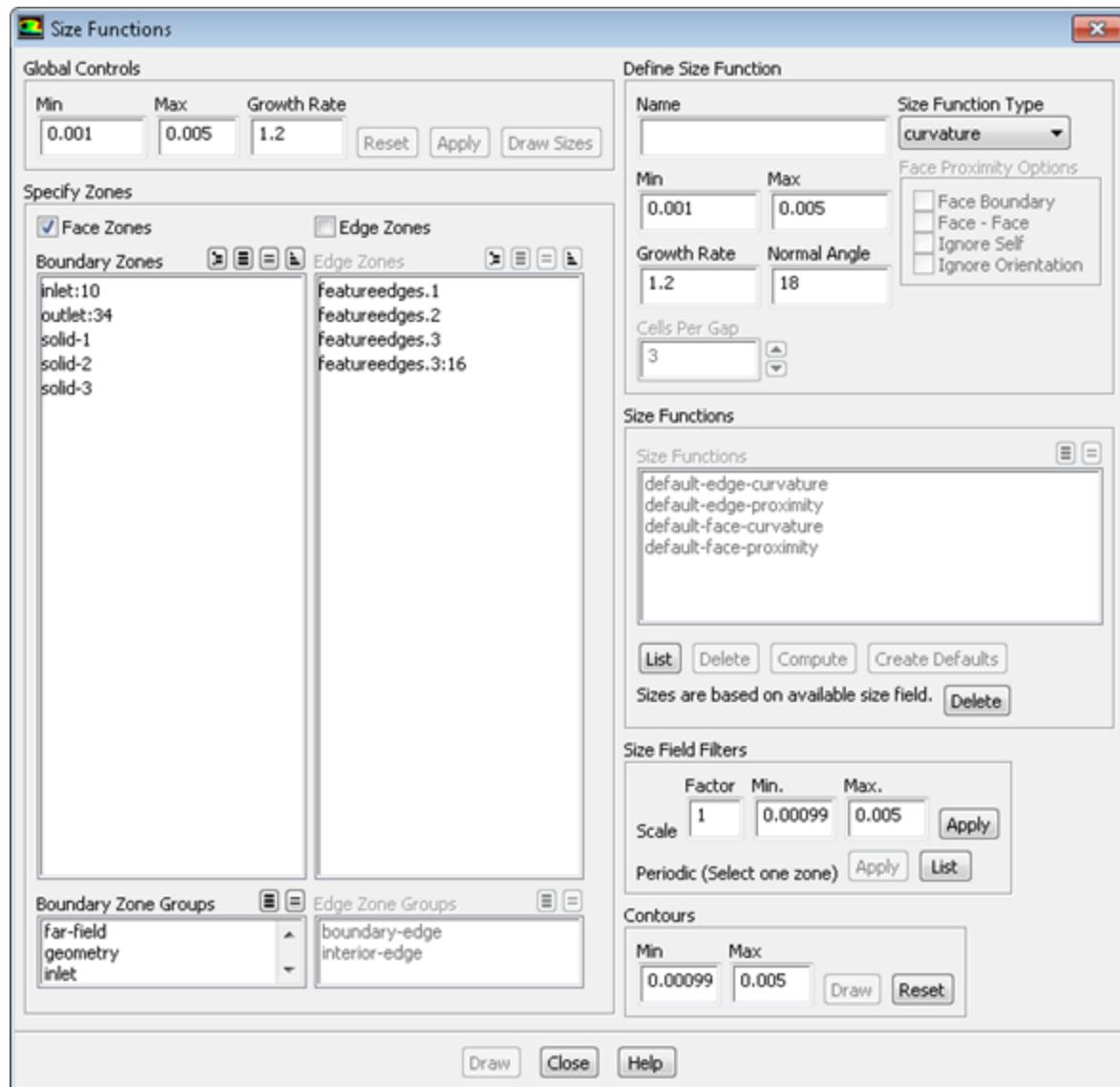
creates a capping surface of the appropriate type as a wrap object based on the entities selected.

Draw

displays the selected object(s) in the graphics window.

21.1.4. Size Functions Dialog Box

The **Size Functions** dialog box allows you to define advanced size functions for greater control of mesh size distribution.



Controls

Global Controls

specifies the global controls.

Min, Max

specify the global minimum and maximum size.

Note

If a proximity, curvature, or hard size function is defined using a local size smaller than the current global minimum size, the global minimum size will be reset to the smaller local size specified.

If you set the global minimum size to a value greater than the local minimum size defined for existing proximity, curvature, or hard size functions, a warning will appear, indicating that the global minimum size cannot be greater than the specified local minimum size.

Growth Rate

specifies the global growth rate.

Reset

resets the global controls to the default values.

Apply

sets the global controls to the specified values.

Draw Sizes

displays red boxes of the specified global minimum and maximum sizes over the selected face or edge zone(s).

Face Zones

when enabled, allows you to define size functions based on face zones.

Boundary Zones

contains the list of boundary zones from which you can select the zone(s) based on which the size function is to be defined.

Boundary Zone Groups

contains the list of the boundary zone types. If you select a zone type/group from the list, all edge zones of that type/group will be selected in the **Boundary Zones** list. You can also select multiple types to select all zones of different types (e.g., **inlet** and **outlet**).

Edge Zones

when enabled, allows you to define size functions based on edge zones.

Edge Zones

contains the list of edge zones from which you can select the zone(s) based on which the size function is to be defined.

You can also select the edge zone(s) associated with specific face zone(s) by selecting the appropriate face zone(s) in the **Face Zones** list. The associated edge zone(s) will be selected in the **Edge Zones** list, and will be used to define the size function.

Edge Zone Groups

contains the list of the edge zone types. If you select a zone type from the list (e.g., **boundary-edge**) all edge zones of that type will be selected in the **Edge Zones** list. This shortcut allows you to select all zones of a certain type without having to select each zone individually.

Define Size Function

contains options for defining the size function.

Name

specifies the name for the size function. You can enter an appropriate name or have the name generated automatically by leaving the **Name** field blank. In this case, the size function will be named based on the zone type (face or edge) and the selected size function type.

Size Function Type

allows you to select the type of size function to be defined.

curvature

allows you to define the curvature size function.

proximity

allows you to define the proximity size function.

meshed

allows you to define the meshed size function.

soft

allows you to define the soft size function.

hard

allows you to define the hard size function.

boi

allows you to define the body of influence size function.

Min

specifies the minimum edge length applicable to the curvature, proximity, and hard size functions.

Max

specifies the maximum edge length applicable to the curvature, proximity, soft, and body of influence size functions.

Growth Rate

represents the increase in element size with each succeeding layer of elements. For example, a growth rate of 1.2 results in a 20% increase in element edge length with each succeeding layer of elements.

Normal Angle

controls the size computation for the curvature size function and is the maximum allowable angle that one element edge is allowed to span. For example a value of 5 implies that a division will be made when the angle change along the curve is 5 degree. Hence, a 90 degree arc will be divided into approximately 18 segments.

Cells Per Gap

is applicable to the proximity size function and is the number of layers of elements to be generated in the gap.

Note

In case of CutCell meshes, the value of cells per gap is approximate. For example, if a value of 3 is specified for cells per gap on a narrow face, the final CutCell mesh may have between 2–4 cells across the gap, depending on the orientation of the face with respect to the global X, Y, Z axes.

Face Proximity Options

contains additional options for defining the face proximity size function.

Face Boundary

allows you to compute the shell proximity (edge-edge proximity within each face).

Face - Face

allows you to compute the proximity between two faces in the face zone(s) selected.

Ignore Self

allows you to ignore self proximity when the proximity size function is processed. This option is available only when the **Face - Face** option is enabled.

Ignore Orientation

allows you to ignore the orientation of face normals when the proximity size function is processed. This option is available only when the **Face - Face** option is enabled.

Size Functions

lists the defined size functions.

Note

When you select a particular size function in the **Size Functions** selection list, the zone(s) for which the size function is defined will be highlighted and the corresponding defined parameter values will also be displayed. If you select multiple size functions, the zone lists and **Define Size Function** controls will be automatically deactivated (greyed out).

List

lists all the defined size functions and the corresponding parameter values defined.

Delete

deletes the size functions selected in the **Size Functions** selection list.

You can also delete the size field when a size field file has been read or computed.

Important

If a size field file has been read in the current session, sizes will be based on the size field read. You cannot define additional size functions or modify the current sizes without deleting the size field.

Compute

computes the size field based on the defined parameters.

Important

If the size field file has been computed in the current session, sizes will be based on the computed size field. You cannot define additional size functions or modify the current sizes without deleting the size field.

Create Defaults

creates default size functions based on face and edge curvature and proximity.

Size Field Filters

contains options for applying a scale factor to temporarily filter the size output from the size field and specifying periodicity.

Scale

contains options for temporarily filtering the size output from the size field.

Factor

specifies the scale factor for temporarily modifying the size output.

Min, Max

specifies the minimum and maximum size values for the filtered size.

Apply

applies the scaling filter to the current size field.

Periodicity

allows you to set periodicity for a size field.

Note

Ensure that periodicity is previously defined in the **Make Periodic Boundaries** dialog box.

Only rotational periodicity is supported, translational periodicity is not supported currently.

Apply (Remove)

Select one periodic zone and then click **Apply** to set periodicity for the size field.

If periodicity is applied, this button changes to **Remove**.

List

displays the existing periodic size field information.

Contours

shows the contours of size on the selected face zone. You can set **Min** and **Max** values to define a range for the contours to be drawn.

Note

A visual indication of mesh size is also available using the Mouse Probe (right-click) if the select-filter is set to size (Ctrl+y).

Create

defines the size function based on the specified parameters.

Change

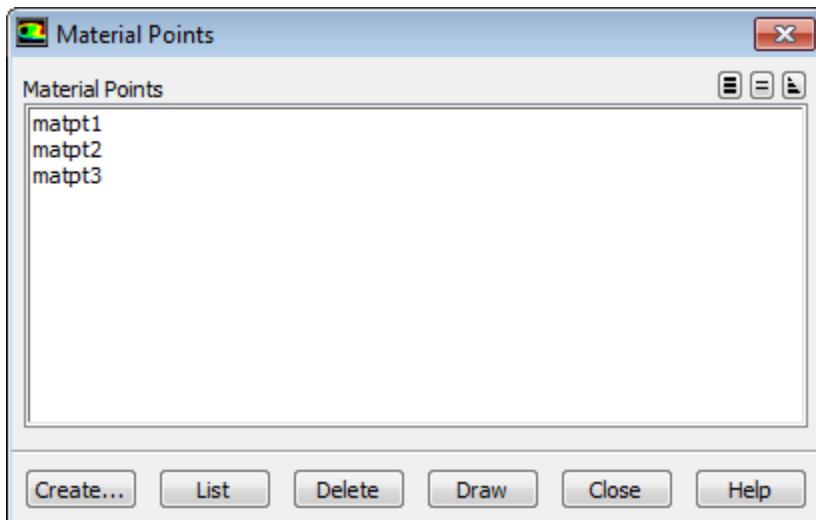
modifies the size function definition to include the modified parameters.

Draw

draws the selected zones in the graphics window.

21.1.5. Material Points Dialog Box

The **Material Points** dialog box contains options for managing material points.



When opened, the panel lists existing material points, and includes buttons to **Create**, **List**, **Delete**, or **Draw** material points.

Create...

opens the [Create Material Point Dialog Box \(p. 427\)](#).

List

lists the material points along with their x-, y-, z-coordinates in the console.

Delete

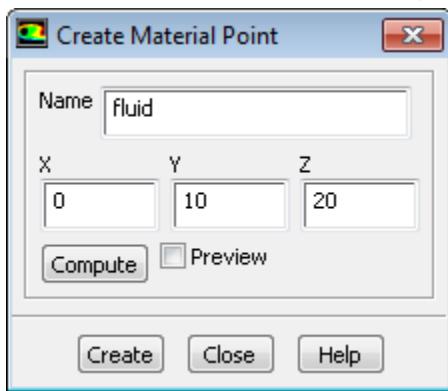
removes the selected material point. Its name is removed from the displayed list.

Draw

displays the material point(s) in the graphics window.

21.1.5.1. Create Material Point Dialog Box

The **Create Material Point** dialog box contains options for creating a material point.



Controls

Name

specifies the material point name.

X, Y, Z

specifies the material point location.

Compute

determines a suitable material point location based on the entities selected in the graphics window.

Preview

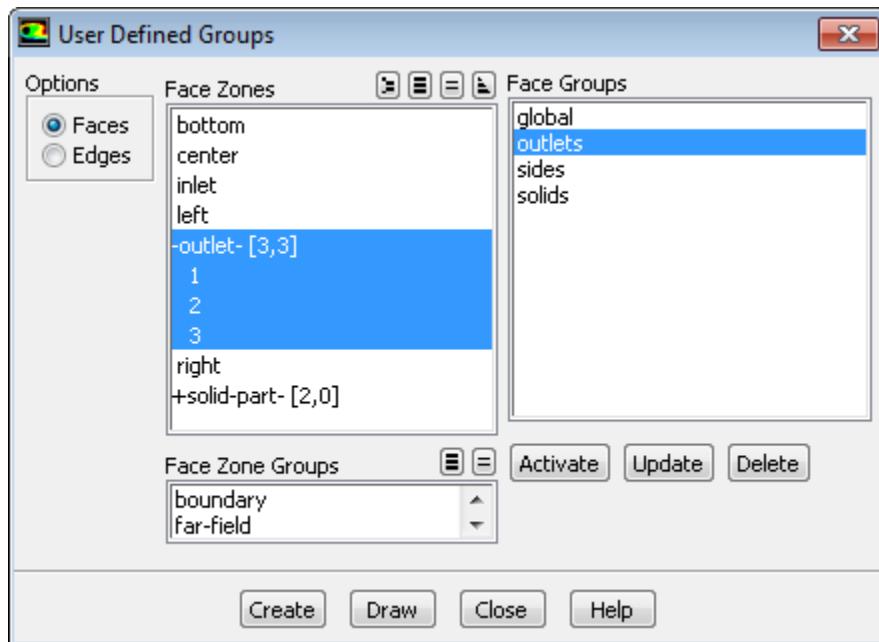
toggles a marker at the location where the material point will be created.

Create

creates the material point based on the name and location specified. The new material point name is added to the list in the **Material Points** dialog box.

21.1.6. User Defined Groups Dialog Box

The **User Defined Groups** dialog box allows you to create groups of surfaces or edges, which will be available in all dialog boxes.

**Controls****Options**

contains options for creating groups.

Faces

allows you to create a group containing one or more face zones.

Edges

allows you to create a group containing one or more edge zones.

Face Zones

contains a list of available face zones from which you can select one or more zones to create a group. The **Face Zones** list is replaced by the **Edge Zones** list when **Edges** is selected in the **Options** group box.

Face Zone Groups

contains a list of the default face zone types and user-defined groups. If you select a zone type/group from this list, all zones of that type/group will be selected in the **Face Zones** list. You can also select

multiple types to select all the zones of different types (e.g., **inlet** and **outlet**). The **Face Zone Groups** list is replaced by the **Edge Zone Groups** list when **Edges** is selected in the **Options** group box.

Face Groups

contains a list of all existing face groups. The **Face Groups** list is replaced by the **Edge Groups** list when **Edges** is selected in the **Options** group box.

A **global** group containing all the respective zones, is created by default. The **global** group cannot be updated or deleted.

Activate

activates the zone group selected in the **Face Groups** (or **Edge Groups**) selection list. Only the zones from the active group will be available in all dialog boxes.

Important

You need to ensure that the **global** group is activated to have all the zones available in the dialog boxes.

Update

updates the selected group according to the current selections in the **Face Zones** (or **Edge Zones**) selection list.

Delete

deletes the selected group from the **Face Groups** (or **Edge Groups**) list.

New

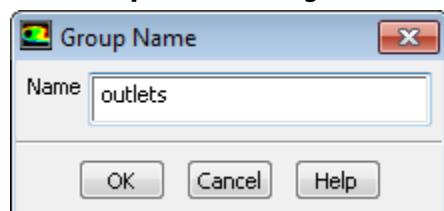
opens the **Group Name** dialog box in which you can enter the name for the group to be created.

Draw

displays the group selected in the **Face Zones** (or **Edge Zones**) list.

21.1.6.1. The Group Name Dialog Box

The **Group Name** dialog box allows you to specify the name for the group to be created.



Controls

Name

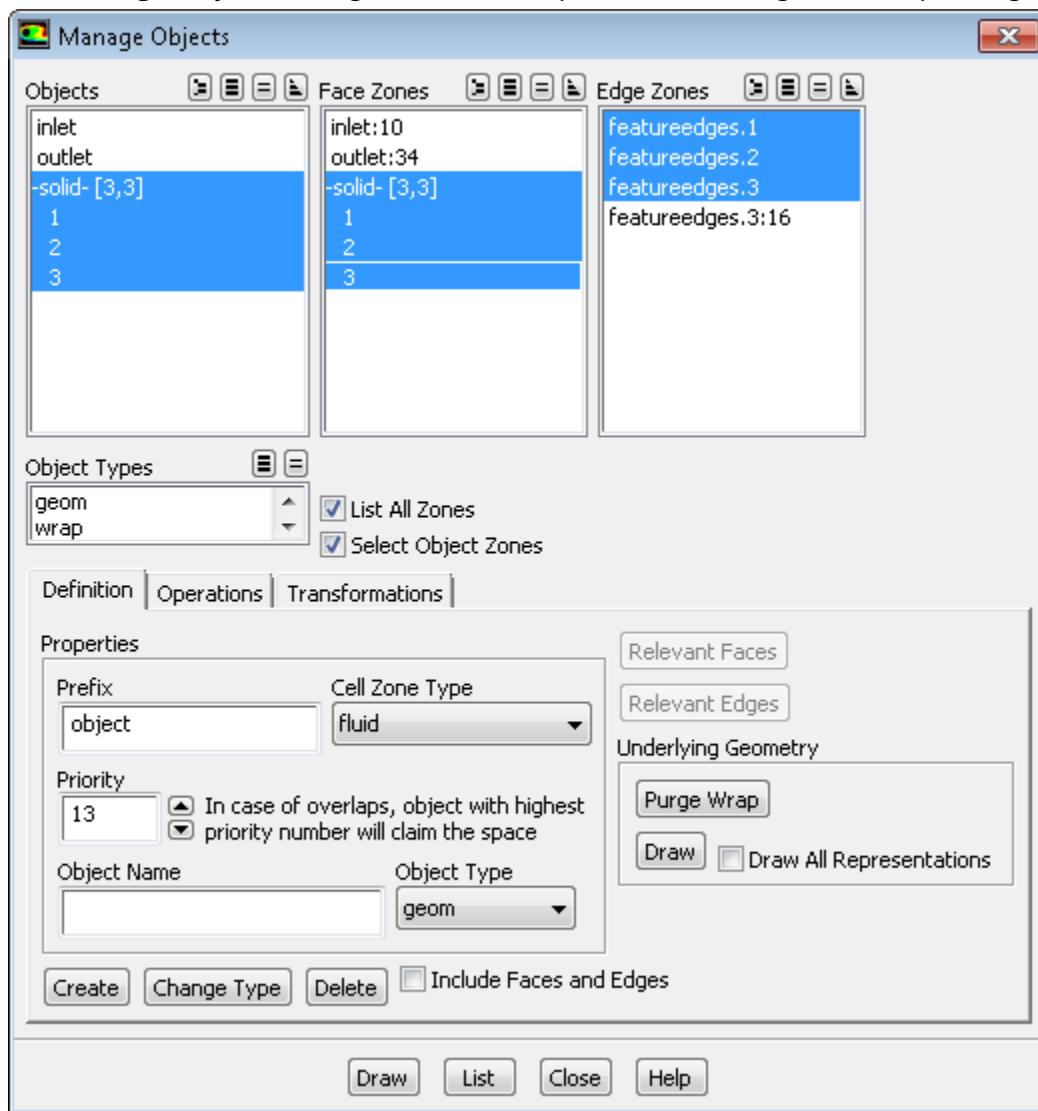
specifies the name of the group to be created.

Important

You cannot create a new group having the name **global**, or having the same name as one of the default groups. You also cannot create a new group having the same name as an existing group.

21.1.7. Manage Objects Dialog Box

The **Manage Objects** dialog box contains options for defining and manipulating objects.



Controls

Objects

contains a list of the defined objects. When you select an object in the list, the face and edge zones used to define the object will be highlighted in the **Face Zones** and **Edge Zones** lists when the **Select Object Zones** option is enabled.

Face Zones

contains a list of face zones. The number of face zones displayed in the list can be controlled using the **List All Zones** check box. When the **List All Zones** option is enabled (default), all available face zones will be listed in the **Face Zones** list. When this option is disabled, the **Face Zones** list will be populated with face zone(s) that are not included in the existing objects. When you select an object in the **Objects**

list, the face zone(s) used to define the object will be highlighted in the **Face Zones** list when the **Select Object Zones** option is enabled.

Note

Only tri face zones can be selected for defining objects.

Edge Zones

contains a list of edge zones. The number of edge zones displayed in the list can be controlled using the **List All Zones** check box. When the **List All Zones** option is enabled (default), all available edge zones will be listed in the **Edge Zones** list. When this option is disabled, the **Edge Zones** list will be populated with edge zone(s) that are not included in the existing objects. When you select an object in the **Objects** list, the edge zone(s) used to define the object will be highlighted in the **Edge Zones** list when the **Select Object Zones** option is enabled.

Object Types

contains the list of the object types. If you select an object type from the list, all objects of that type will be selected in the **Objects** list. You can also select multiple types to select all objects of different types (e.g., **geom** and **wrap**).

List All Zones

controls the number of face and edge zone(s) listed in the **Face Zones** and **Edge Zones** lists. When **List All Zones** is enabled (default), all available face and edge zones will be listed in the **Face Zones** and **Edge Zones** lists. When this option is disabled, the **Face Zones** and **Edge Zones** lists will be populated with face and edge zone(s) that are not included in the existing objects. When object(s) are selected, the face and edge zone(s) included in the object(s) will be highlighted in the **Face Zones** and **Edge Zones** lists.

Select Object Zones

when enabled, the face and edge zones used to define the selected object(s) will be highlighted in the **Face Zones** and **Edge Zones** lists.

Definition

contains options for defining objects.

Properties

allows you to specify the object attributes (see [Object Attributes \(p. 103\)](#)).

Prefix

specifies the prefix to be used for the object name when the names are to be generated automatically. The default prefix is **object-**. You can leave the **Prefix** field blank if you do not want to specify a prefix for the object name.

Cell Zone Type

allows you to select the cell zone type for the object.

Priority

sets the priority for the object created.

Object Name

specifies the name for the object to be created. When this field is blank, the object name will be generated automatically based on the prefix, cell zone type, and priority specified.

Object Type

allows you to specify the object type. The default object type is **geom**.

Create

creates the object based on the zone(s) selected, the cell zone type, the priority, and the object type specified.

Change

modifies the existing object definition based on the changes specified.

Change Type

allows you to change the object type (**geom**, **wrap**, or **mesh**) when multiple objects are selected in the **Objects** list.

Delete

deletes the object(s) selected in the **Objects** selection list.

Include Faces and Edges

allows you to include the faces and edges defining the object(s), when the selected object(s) are renamed or deleted.

Relevant Faces

shows additional face zone(s) in the vicinity of those included in the selected object. You can use this option to verify that the necessary face zone(s) are included in the object definition.

Relevant Edges

shows additional edge zone(s) in the vicinity of those included in the selected object. You can use this option to verify that the necessary edge zone(s) are included in the object definition.

Underlying Geometry

contains options for displaying the geometry representation of the selected wrap objects.

Purge Wrap

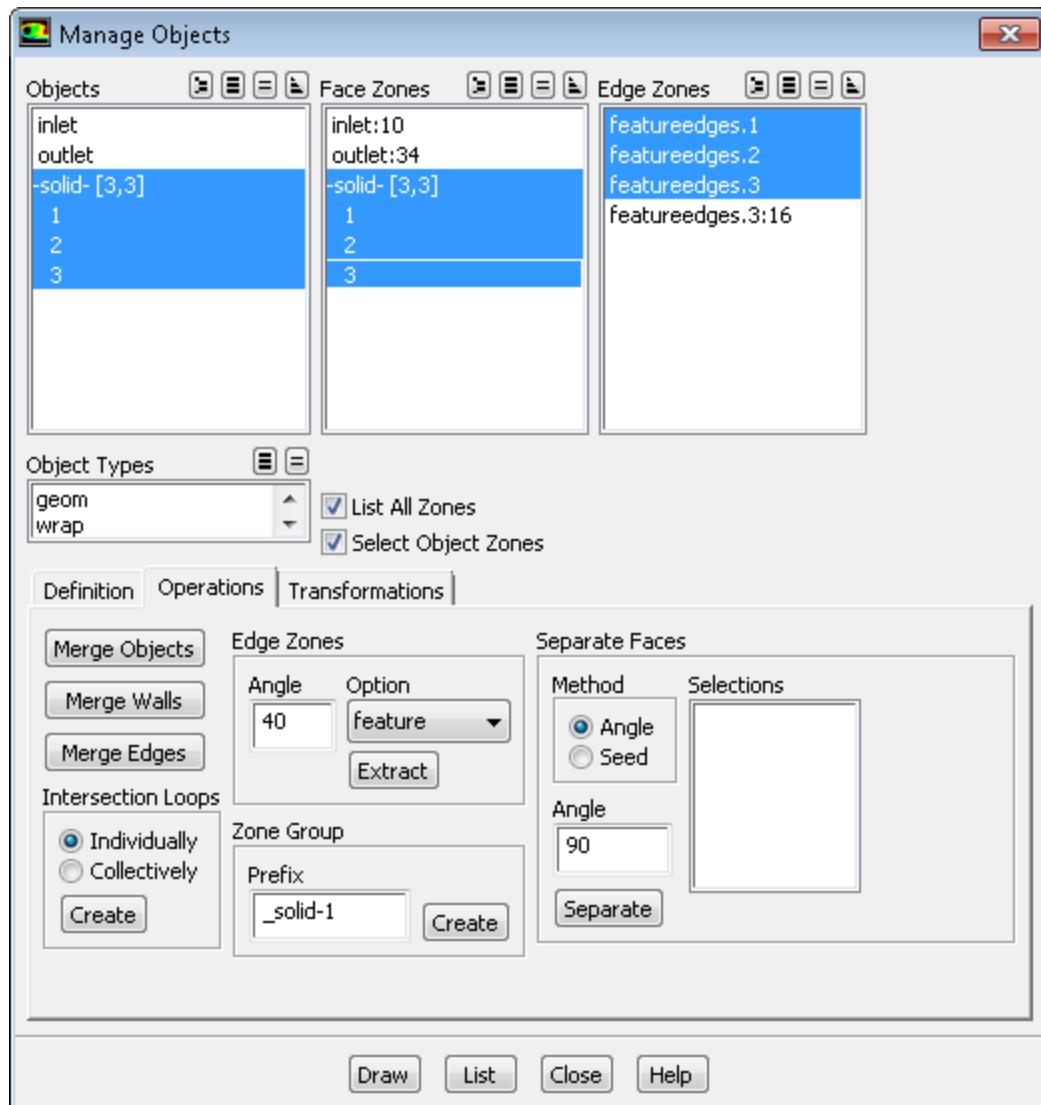
deletes the face zones associated with the wrap object(s) selected and reverts the object(s) to their geometry representations.

Draw

displays the geometry representation of the selected wrap object(s) in the graphics window.

Draw All Representations

displays the wrapped and geometry representations of the selected object(s) in the graphics window.



Operations

contains options for operations to be performed on objects.

Merge Objects

merges multiple objects into a single object.

Merge Walls

merges all the face zones of type **wall** in an object into a single face zone.

Merge Edges

merges all the edge zones in an object into a single edge zone.

Note

If the object comprises edge zones of different types (boundary and interior), the edge zones of the same type (boundary or interior) will be merged into a single edge zone.

Intersection Loops

contains options for creating intersection loops for objects.

Individually

creates an interior edge loop at the intersection between two adjacent face zones included in the same object.

Collectively

creates an interior edge loop at the intersection between two adjacent face zones included in the same object and between multiple objects.

Edge Zones

contains options for extracting the edge zone(s) from the face zone(s) included in the selected object(s).

Angle

specifies the edge feature angle to be used for extracting edge zone(s) from the face zone(s) included in the object(s).

Option

allows you to select whether only feature edges or all edges are to be extracted for the object(s) selected.

Extract

extracts the edge zone(s) from the face zone(s) included in the selected object, based on the angle value specified.

Zone Group

contains options for creating a face group and edge group based on the selected object(s). The face group created will be available in all the dialog boxes along with the default groups (e.g., boundary, tri, quad, etc.).

Prefix

specifies the prefix for the face and edge groups to be created.

Note

The use of the underscore (_) in the prefix allows the groups created to be listed at the top of the **Face Zone Groups/Edge Zone Groups** lists.

Create

creates a face group and an edge group comprising the face zone(s) and edge zone(s) included in the selected object(s), respectively.

Separate Faces

contains options for separating the face zone(s) comprising the selected object(s).

Method

specifies the method for separating the face zone(s) comprising the object.

Angle

allows you to specify the angle used to separate the face zones comprising the object. Faces with normal vectors that differ by an angle greater than or equal to the specified value will be placed in different zones.

Seed

allows you to specify the seed face(s) used to separate the face zones comprising the object.

Note

A single face zone will be created for each separate operation, and not per seed face selected.

Angle

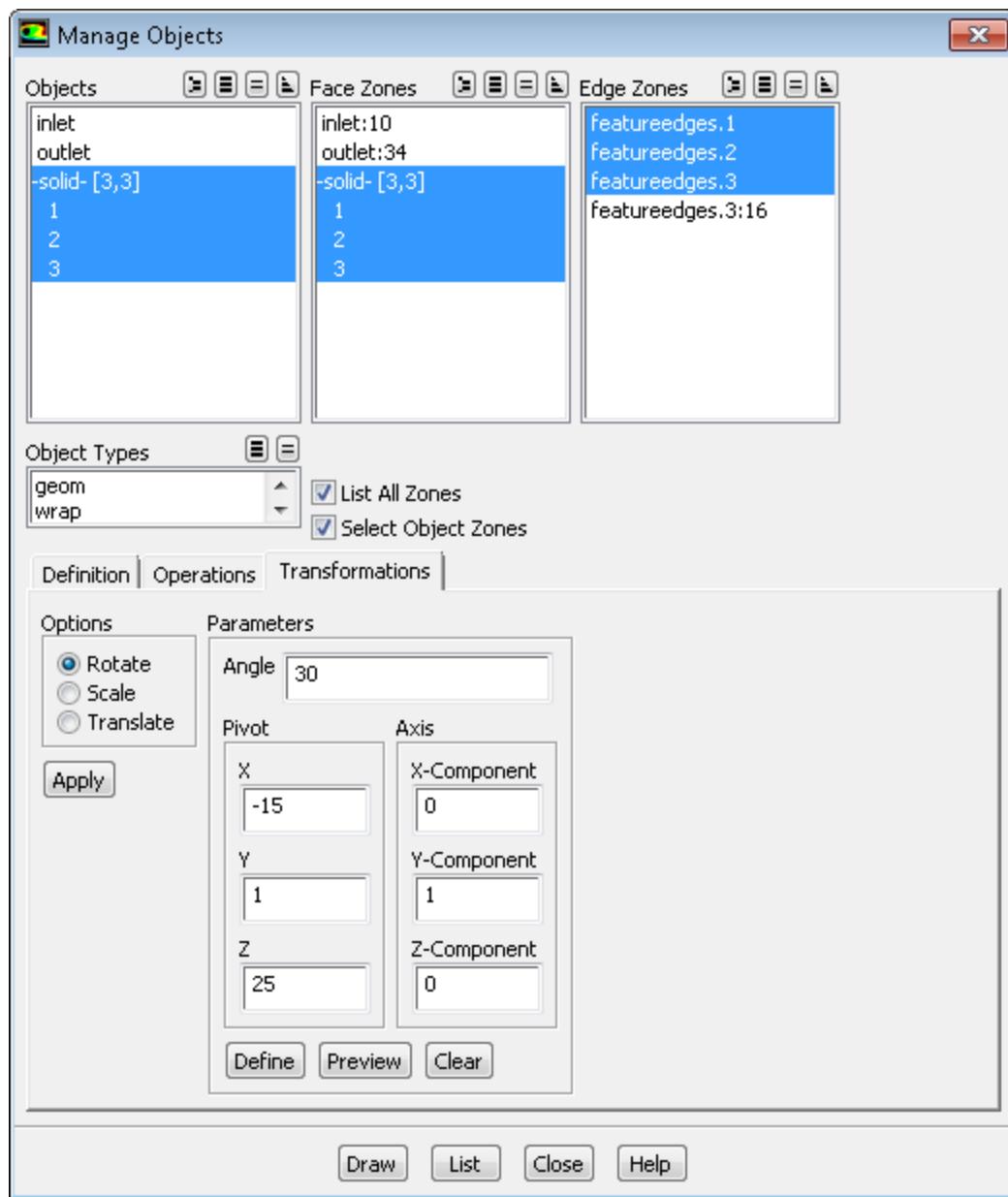
specifies the edge feature angle to be used for extracting edge zone(s) from the face zone(s) included in the object(s).

Selections

specifies the labels of the face element that you have selected as seed for separating the face zone(s).

Separate

separates the face zone(s) comprising the object based on the angle or seed face specified.



Transformations

contains options for object transformation.

Options

contains a list of object transformation operations, each of which has distinct input requirements.

Rotate

Parameters

contains options for specifying parameters for the object **Rotate** operation.

Angle

specifies the angle of rotation for the rotate operation.

Pivot

specifies the pivot point for the rotate operation.

Axis

specifies the axis of rotation for the rotate operation.

Define

allows you to select 1–6 nodes to define the pivot point and axis of rotation.

Preview

displays a triad at the pivot point, with the local z-axis indicating the axis of rotation.

Clear

clears the preview triad from the graphics window.

Scale**Scale Factors**

specifies the scale factors applied in each of the Cartesian coordinate directions (X, Y, and Z).

Translate**Translation**

specifies the translation offsets (X, Y, and Z) to be added to the Cartesian coordinate of every node in the selected zone(s).

Define

allows you to define the direction vector and total distance based on two nodes or positions selected in the graphics window.

Note

The translation offsets are interpreted as absolute numbers in meshing mode. In solution mode, however, the translation offsets are assumed to be distances in the length unit set. This may lead to differences in domain extents reported after translating the mesh in the respective modes.

Apply

performs the object transformation operation on the object(s) selected.

Draw

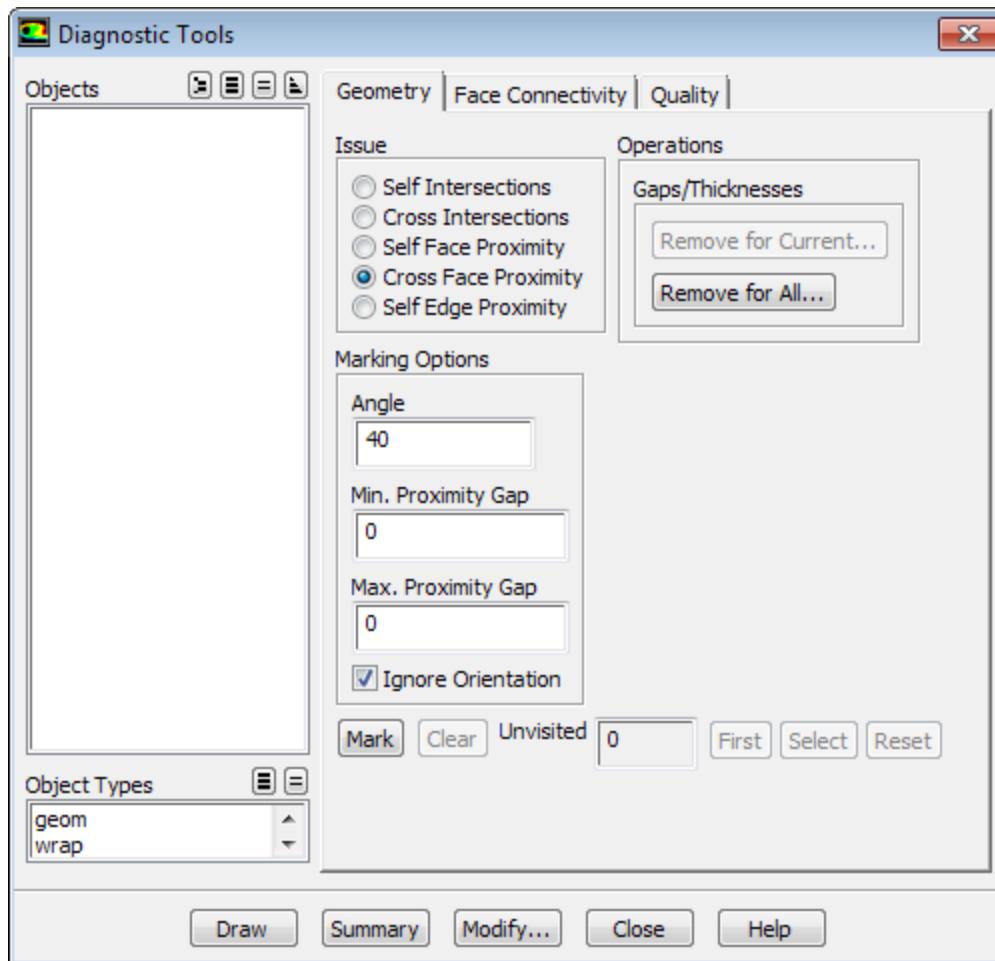
displays the selected object(s) in the graphics window.

List

reports (in the console) the object name, cell zone type, priority, object type, and the face and edge zones comprising the object.

21.1.8. Diagnostic Tools Dialog Box

The **Diagnostic Tools** dialog box gives you lists of tools for finding and fixing boundary mesh problems on objects.



Common Controls

These controls exist for all three tab options (**Geometry**, **Face Connectivity** or **Quality**).

Objects

contains a list of all available objects and selection tools.

Object Types

contains the list of the object types. If you select an object type from the list, all objects of that type will be selected in the **Objects** list. You can also select multiple types to select all objects of different types (e.g., **geom** and **wrap**).

Issue

contains the list of issues that can be found and fixed.

Operations

contains the group of issue-dependant options that you may use to repair the identified problems.

Mark

searches for instances of the selected **Issue** within the selected **Objects**.

If problems are found, the afflicted object(s) is (are) identified in the **Objects** list, and the **Objects** selection tools are disabled (greyed out).

Clear

clears **Marked** faces, zeroes the **Unvisited** value and re-enables the **Object** selection tools.

Unvisited

displays the number of problems found and that have yet to be examined. How the **Unvisited** value is determined depends on the **Issue** selected.

First

highlights a subset of problems in the graphical display, and changes the button label to **Next**. Click **Next** to cycle through all subsets of connected problems.

Each time **First (Next)** is clicked, the **Unvisited** value is reduced.

Select

Re-selects (highlights) the current subset of problems in the graphical display without cycling through the identified subsets. Use this control to select the current subset if selections were modified.

Reset

Deselects the subset in the graphical display and resets **Unvisited** value to show the number of **Marked** problems. **Next** button label is reset to **First**.

Draw

displays the selected object(s) in the graphics window.

Summary

displays an overall summary of issues for the selected objects in the console.

Modify...

opens the [Modify Boundary Dialog Box \(p. 486\)](#) to manually modify your boundary mesh.

Geometry Tab Controls

Tools to find and fix boundary mesh problems arising from Geometry issues.

Self Intersections

finds intra-object intersecting faces.

Note

Unvisited value is the number of affected objects and is reduced by one each time **First (Next)** is clicked.

Intersection Loops**Create For Current**

creates intra-object intersection loop(s) for only the current object.

Create For All

creates all intra-object intersection loops for all selected object(s)

Cross Intersections

finds inter-object intersecting faces.

Note

Unvisited value is the number of affected pairs of objects and is reduced by one each time **First (Next)** is clicked.

Intersection Loops

Create For Current

creates inter-object intersection loops for only the current pair of objects.

Create For All

creates all inter-object intersection loops for all selected object(s).

Self Face Proximity

finds regions of thin material (thickness is between **Min.** and **Max. Proximity Gap**).

Note

Unvisited value is the number of affected objects and is reduced by one each time **First (Next)** is clicked.

Gaps/Thicknesses

Remove For Current...

opens the [Remove Gaps Dialog Box \(p. 452\)](#), propagating only the current **Object** and **Marking Options**, where you may perform the **Remove Thickness in Objects** operation.

Remove For All...

opens the [Remove Gaps Dialog Box \(p. 452\)](#), propagating all selected **Objects** and **Marking Options**, where you may perform the **Remove Thickness in Objects** operation.

Marking Options

Angle

sets the upper limit on the angle between two edges.

Min. Proximity Gap

sets the lower limit for identifying thin material.

Max. Proximity Gap

sets the upper limit for identifying thin material.

Ignore Orientation

If unchecked, the search is limited using opposite outward normals.

Cross Face Proximity

finds locations where small gaps exist between objects.

Note

Unvisited value is the number of affected pairs of objects and is reduced by one each time **First (Next)** is clicked.

Gaps/Thicknesses

Remove For Current...

opens the [Remove Gaps Dialog Box \(p. 452\)](#), propagating only the current pair of **Objects** and **Marking Options**, where you may perform the **Remove Gaps Between Objects** operation.

Remove For All...

opens the [Remove Gaps Dialog Box \(p. 452\)](#), propagating all selected **Objects** and **Marking Options**, where you may perform the **Remove Gaps Between Objects** operation.

Marking Options**Angle**

sets the upper limit on the angle between two faces forming the gap.

Min. Proximity Gap

sets the lower limit for identifying gaps.

Max. Proximity Gap

sets the upper limit for identifying gaps.

Ignore Orientation

If unchecked, the search is limited using opposite outward normals.

Self Edge Proximity

marks edges within the proximity limit. You will have to manually fix the geometry to remove these issues.

Note

- The identified edges are grouped into clusters of connected edges, and then the clusters are sorted by the number of edges. **First (Next)** traverses the clusters from largest to smallest.
- Unvisited** is the total number of affected edges and is reduced by the number of connected edges in the displayed cluster each time **First (Next)** is clicked.

Marking Options**Max. Proximity Gap**

sets the upper proximity limit. Lower proximity limit is zero.

Note

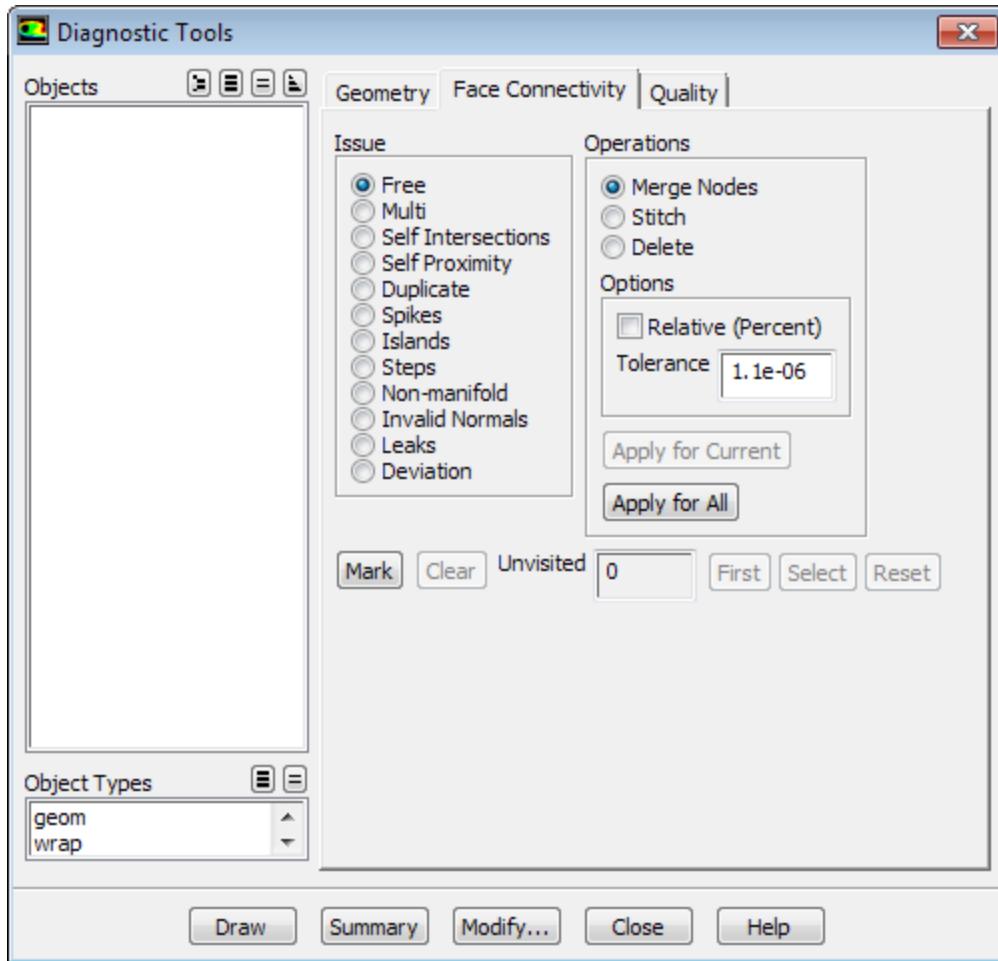
The **Max. Proximity Gap** must be less than the local edge length.

Face Connectivity Tab Controls

Tools to find and fix boundary mesh problems arising from Face Connectivity issues.

Note

- The identified faces are grouped into clusters of connected faces, and then the clusters are sorted by the number of faces. **First (Next)** traverses the clusters from largest to smallest.
- Unvisited** is the total number of faces exhibiting the selected **Issue** and is reduced by the number of connected faces in the displayed cluster each time **First (Next)** is clicked.



Free

identifies faces with at least one edge not connected to an adjacent face.

Merge Nodes

Free nodes within the specified **Tolerance** will be combined.

Stitch

Free edges within the specified **Tolerance** will be joined.

Delete

Faces with free edges will be removed if the **Options** threshold (free edge count, face count, or skewness, depending on selection) is met.

Multi

identifies faces with at least one edge with more than two connected faces.

Marking Options

Max Edge Count is the threshold for determining if a group of connected faces is a problem. If a connected group of faces has more than this threshold, the group is not counted. For example,

a hanging baffle may have many **Multi** faces along one edge where it connects to the rest of the model, and should not be marked.

Note

Set to -1 to mark all multi-faces without any restrictions.

Delete Fringes

Hanging faces, in group sizes smaller than the specified **Max Fringe Faces** threshold, are removed.

Delete Overlaps

Overlapping regions, in group sizes smaller than the specified **Max Overlap Faces** threshold, are joined.

Disconnect

Face zones boundaries, in group sizes smaller than the specified **Max Multi-Edges** threshold, will be separated.

All Above

allows you to delete fringes and overlaps, and disconnect the multiconnected faces.

Self Intersections

identifies faces that cross other faces.

Marking Options

Folds If enabled, also checks for sharp angles between faces.

Fix Self Intersections

This operation fixes only Intersections, and works by moving boundaries apart.

Fix Folded Faces

This operation fixes **Folds** as well as Intersections. Select the **Imprint Features** option if the fold is to follow geometry features.

Note

This operation is best for wrapped surfaces.

On Marked Faces

Provides options to **Smooth All** or only Smooth Current.

Smoothing moves node(s) of the Marked Faces to eliminate the intersecting face(s).

Self Proximity

identifies thin material within an object.

Marking Options

Relative

If checked, the maximum separation to be marked is proportional to the face size

Tolerance

sets the proportion used by **Relative** marking option, or the absolute separation if **Relative** is unchecked.

Ignore Orientation

If unchecked, the search is limited using opposite outward normals.

Angle

sets the upper limit on the angle between two faces forming the thickness.

Separate Current

applies the separation operation to only the selected faces.

Separate All

applies the separation operation to all identified faces.

Duplicate

identifies faces which share all the same nodes.

Remove Duplicate

Click to activate. This operation keeps one of the duplicate faces and deletes all other duplicates.

Spikes

identifies faces connected to nodes that deviate significantly from a surface.

Marking Options

Spike Angle is the sum of all edge angles at the node (360 for a flat node, 0 for an infinite spike).

On Marked Faces

Provides options to **Smooth All** or only Smooth Current.

Smoothing moves node(s) of the Marked Faces to eliminate the spike(s).

Islands

identifies groups of faces forming a volume (pocket) separate from the model.

Marking Options**Absolute Count**

the minimum number of faces which are considered a viable island.

Remove Islands

Click to delete identified islands.

Steps

identifies situations where edges of one or more faces are offset orthogonally from the edges of proximal, and often parallel, faces. Click **Mark** to find the connecting faces filling the space between the offset edges.

Marking Options**Angle**

Specifies the maximum angle between the normals of the faces along the edge offset to be considered as a step.

Step Width

Specifies the maximum size of the connecting faces that will be **Marked** as a step.

Smooth Current

Use the hotkey to maintain the connecting faces. The nodes are moved to a position computed from the average of its node neighbors.

Collapse Current

Use the hotkey to remove connecting faces. The nodes along the offset edges are merged along the midpoint.

Smooth All

Click to apply the smooth operation to all identified steps.

Collapse All

Click to apply the collapse operation to all identified steps.

Non-manifold

identifies faces where prism layer growth would cause mesh errors.

It is recommended to **Set Prism Settings** before using this tool.

Remove Point Contacts

Click to activate.

Invalid Normals

identifies faces where the orientation of the normal would cause errors when growing prism layers.

It is recommended to **Set Prism Settings** before using this tool.

On Marked Faces

Provides options to **Smooth All** or only Smooth Current.

Smoothing moves node(s) of the Marked Faces to eliminate the invalid normals.

Leaks

instructions are displayed to trace a potential leak path between faces.

Deviation

identifies locations where the relative distance between the wrapper surface and the associated geometry exceeds a specified threshold.

Note

If a new object is created during wrap, make sure that the original objects and face zones are not deleted or modified for this to work correctly.

Marking Options**Min. Deviation**

The threshold above which faces will be **Marked**.

Relative

The allowed deviation is proportional to the face size.

Feature Imprint

Imprint Iterations

This operation will attempt to capture the edge of a feature at the expense of smoothing the boundary mesh.

Aggressive Imprint Iterations

As above with more emphasis on capturing the feature.

On Marked Faces

No. of Layers

Specifies the number of faces surrounding the **Marked** face(s) to be included in the **Smooth** operation.

Smooth Aggressively

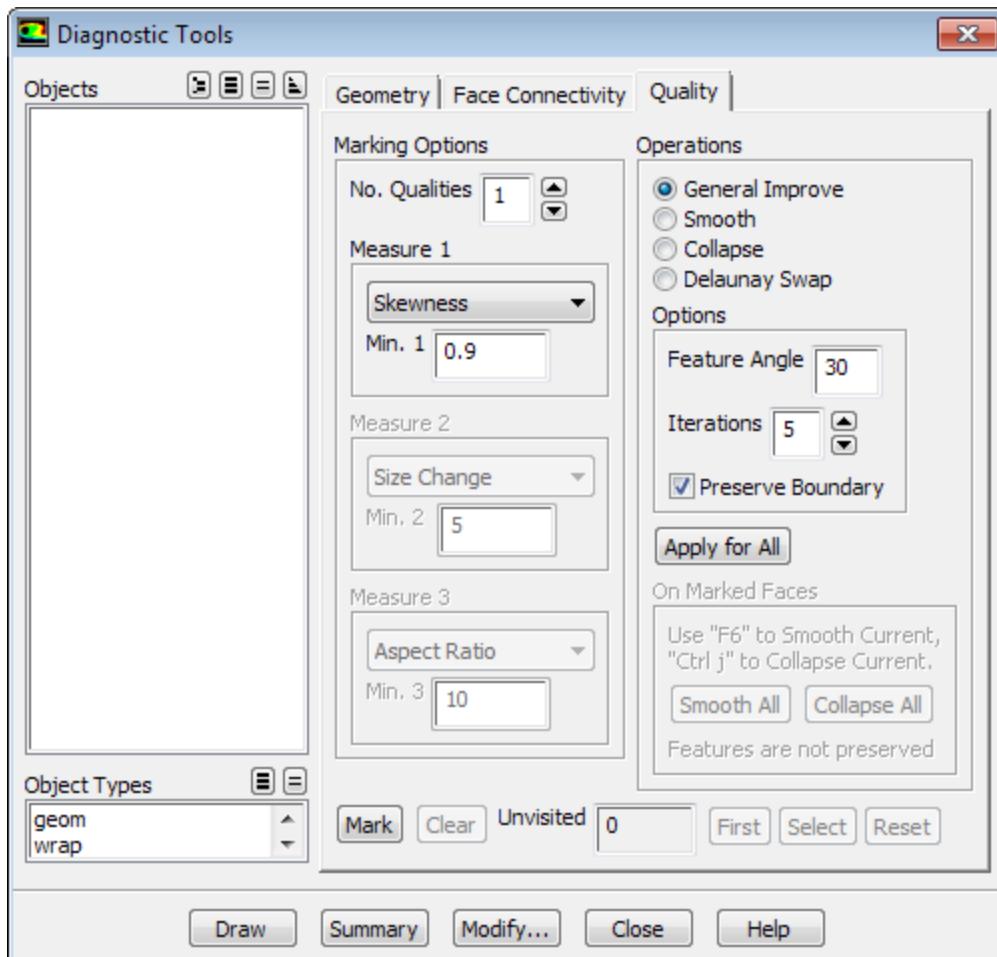
May increase deviation or lose features by reducing the angle between adjacent face normals.

Quality Tab Controls

Use this tab to identify problem faces based on up to three **Quality** measures.

Note

- The identified problem faces are grouped into clusters of connected faces, and then the clusters are sorted by their number of faces. **First (Next)** traverses the clusters from largest to smallest.
 - **Unvisited** is the total number of problem faces and is reduced by the number of connected faces in the displayed cluster each time **First (Next)** is clicked.
-



No. Qualities

You may choose 1, 2 or 3 simultaneous measures. A face will be counted if it fails at least one of the specified quality measures (OR operation).

Measure 1

(also **Measure 2** and **Measure 3** if exist) Choose a measure using the drop down list provided, and then set specific values for that measure. For more information on how ANSYS Fluent calculates the quality and adjusts the mesh, see the [Quality Measures](#) page and the [Boundary Improve](#) page.

The **Operations** group is disabled if more than one measure is selected.

General Improve

Uses multiple techniques to attempt to meet the quality measure criterion.

Smooth

moves all nodes to a position computed from the average of its node neighbors.

Collapse

collapses pairs of nodes, edges, or faces. If a pair of nodes is selected, both the nodes are deleted and a new node is created at the midpoint of the two nodes. If a triangular face is selected, the face is collapsed into a single node at the centroid of the face..

Delauney Swap

Checks each pair of faces that shares an edge and identifies the connecting diagonal that results in the most appropriate configuration of faces within the resulting quadrilateral.

Options

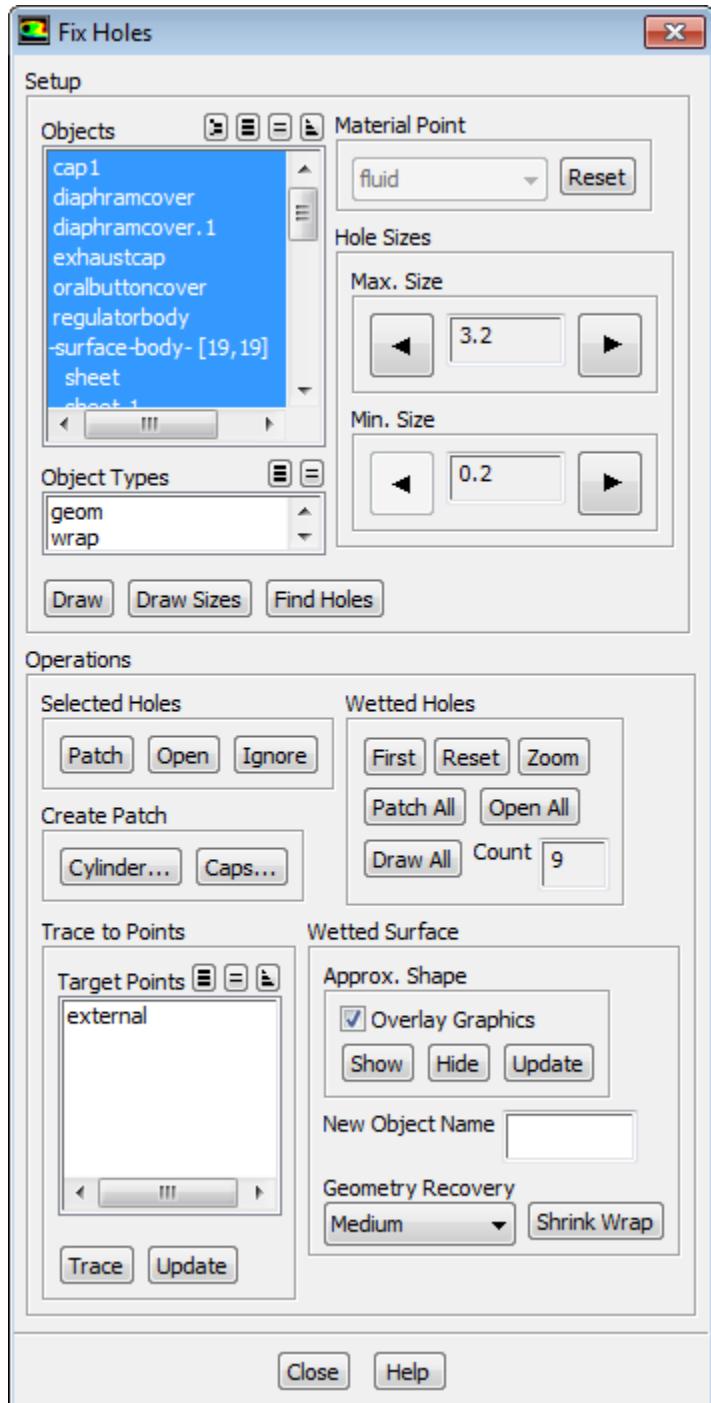
Set parameters on the Operation indicated to improve the quality. Available options depend on the **Measure** selected.

On Marked Faces

Click the button or use the specified hotkey to **Smooth All**, Smooth Current, **Collapse All** or Collapse Current.

21.1.9. Fix Holes Dialog Box

The **Fix Holes** dialog box contains options for locating and fixing holes in the objects.



Controls

Setup

contains options for specifying parameters to locate holes in the objects selected.

Objects

contains a list of the defined objects

Object Types

contains the list of the object types. If you select an object type from the list, all objects of that type will be selected in the **Objects** list. You can also select multiple types to select all objects of different types (e.g., **geom** and **wrap**).

Material Point

allows you to select the material point to determine the volume/region of interest. You can select a material point from the drop-down list. The default is an external material point.

Reset

allows you to change the material point to be used to locate the holes.

Hole Sizes

allows you to specify the minimum and maximum limits for the hole size. The size values allow you to limit the search for holes to a relevant subset based on the size range. As you locate the respective holes and fix them, the possibility of false holes is reduced.

Max. Size

specifies the maximum limit for the hole size.

Min. Size

specifies the minimum limit for the hole size.

Draw

displays the selected objects along with the **Material Point** in the graphics window

Draw Sizes

displays boxes on the geometry indicating the minimum and maximum sizes.

Find Holes

finds the holes in the objects selected based on the specified parameters.

Operations

contains options for operations to be performed on the holes and also for wrapping the geometry.

Selected Holes

contains operation which can be performed on the holes selected on graphics.

Patch

patches the selected hole.

Open

allows you to open the hole to allow the wetted surface to propagate through the selected hole.

Ignore

allows you to ignore the hole if it is not relevant for the wrapping operation.

Create Patch

contains options to manually create a patch to close the holes.

Cylinder...

opens the [Cylinder Dialog Box \(p. 417\)](#), which contains options for creating a cylinder.

Caps...

opens the [Capping Surface Dialog Box \(p. 420\)](#), which contains options for creating a cap surface.

Wetted Holes

contains options for traversing through the holes and fixing the holes connected to the wetted surfaces.

First, Next

allow you to traverse the list of wetted holes. When you click **First**, the first selected hole will be displayed in the graphics window and the hole ID will be reported in the console. You can then click **Next** repeatedly to traverse the list of selected holes.

Reset

resets traversal.

Zoom

zooms in on the current hole of traversal

Patch All

patches all the wetted holes

Open All

allows wetted surface to propagate through all the wetted holes and new list of wetted holes connected to the propagated wetted surface will be listed as wetted holes

Draw All

displays all the wetted holes.

Count

displays the number of wetted holes to be patched or opened.

Trace to Points

allows you to check if there is a path from the material point selected to the chosen target point.

Target Points

contains a list of the material points you have created and an external point, other than the material point selected for locating the holes. You can choose one or multiple points from the list which will act as target point(s) for tracing a path(s) from the **Material Point**.

Trace

draws a path from the **Material Point** to the chosen target point through all the selected objects. It also allows you to verify if the caps, cylinders, or patches created have closed the holes properly. It will take into consideration the minimum limit of the hole size.

Update

updates the trace to include any new objects created by the hole fixing operations.

Wetted Surface

contains options for displaying the wetted surface and creating the wrap object using the shrink-wrap method.

Approx. Shape

allows you to view the approximate shape of the wetted surface.

Overlay Graphics

allows you to display the geometry along with the wetted surface.

Show

displays an approximate shape of the wetted surface.

Hide

hides the wetted surface from the display in the graphics window.

Update

updates the selected wetted surface region with selected objects or the modifications made.

New Object Name

specifies a name for the wrap object to be created.

Geometry Recovery

contains options for geometry recovery on the wrap object.

Low

allows you to create a rough wrapped representation of the geometry object.

Medium

performs additional refinement, imprinting and aggressive imprinting iterations to improve the feature recovery. Individual zones are recovered based on the original geometry object and then rezoned.

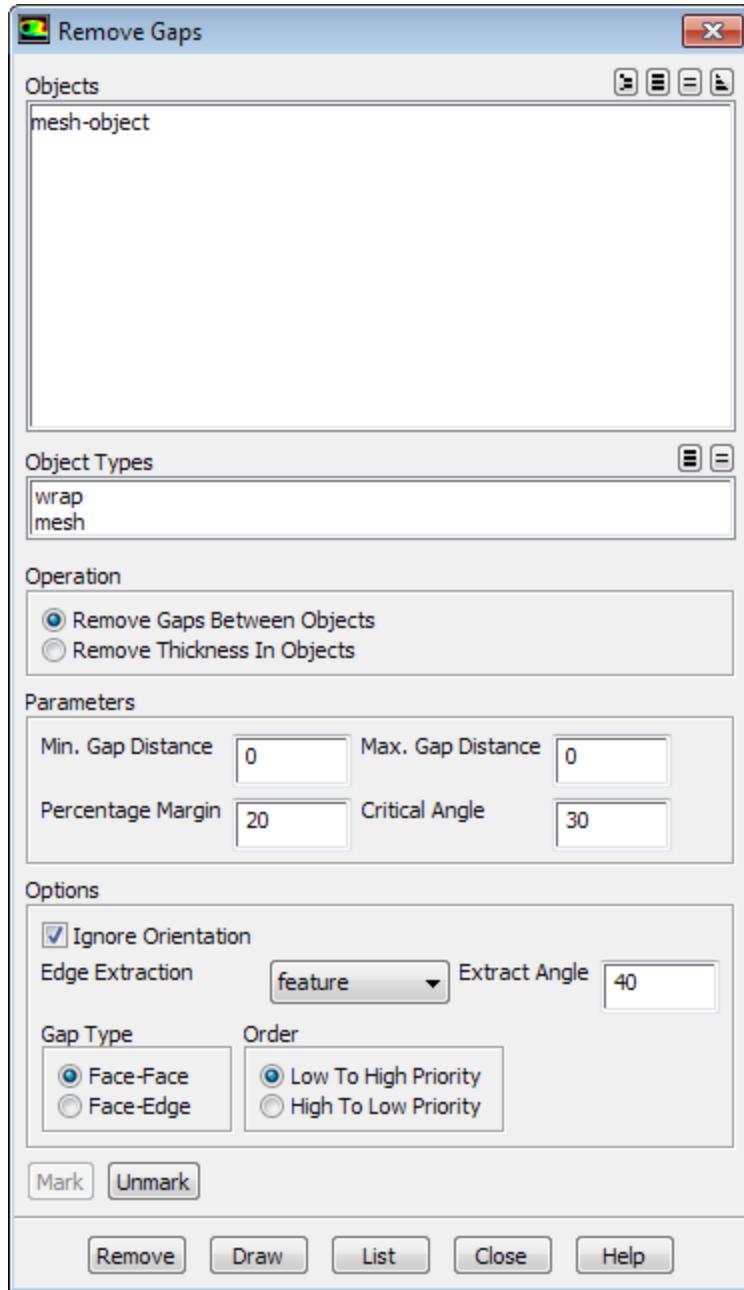
High

allows better feature capture and surface remeshing based quality improvement. Degenerate and island edges are deleted, intersected and remeshed as appropriate. The edges are imprinted on the wrapped zones, and individual zones are recovered based on the original geometry object and then rezoned. Surfaces are remeshed based on size functions/size field.

Shrink Wrap

creates a wrap object based on the selected objects, using the shrink-wrap method. The default geometry recovery option (**Medium**) will be used.

21.1.10. Remove Gaps Dialog Box



Controls

Objects

contains the list of wrap objects or mesh objects available.

Object Types

contains the list of the object types. If you select an object type from the list, all objects of that type will be selected in the **Objects** list. You may also select multiple types simultaneously to select all objects of different types.

Operation

allows you to select the repair operation.

Remove Gaps Between Objects

allows you to remove gaps between the objects selected.

Remove Thickness In Objects

allows you to project close thin sections to the mid-surface, thereby removing the thickness.

Parameters

specifies the parameters controlling the gap/thickness.

Min. Gap Distance

specifies the minimum distance across the gap which will be considered for gap removal.

Max. Gap Distance

specifies the maximum distance across the gap/thickness which will be considered for gap/thickness removal.

Percentage Margin

specifies a tolerance margin for determining the gap/thickness distance.

Critical Angle

specifies the maximum angle between the opposite faces constituting the gap/thickness to be removed.

Options

contains options controlling the removal of the gap/thickness.

Ignore Orientation

if enabled, does not take into account the normals in the gaps.

Edge Extraction

contains options for edge extraction. You can choose **none**, or create **feature** edges, or **all** boundary and feature edges.

none

does not extract edges during the operation.

feature

extracts feature edges based on the angle specified.

all

extracts all boundary edges and feature edges based on the angle specified.

Extract Angle

specifies the feature angle for edge extraction.

Gap Type

allows you to select whether the gap between the objects is between face zones (**Face-Face**) or faces and edges (**Face-Edge**). This option is available only when the **Remove Gaps Between Objects** option is selected.

Order

determines the direction of projection for gap removal. This option is only valid for **Face-Face** gap removal. You can project faces from the object of lower to higher priority (**Low to High Priority**) or higher to lower priority (**High to Low Priority**).

Mark

marks the faces that will be used for projection for the gap removal operation.

Unmark

unmarks the faces marked for projection for the gap removal operation.

Remove

removes the gap between objects or the thickness based on the operation selected.

Draw

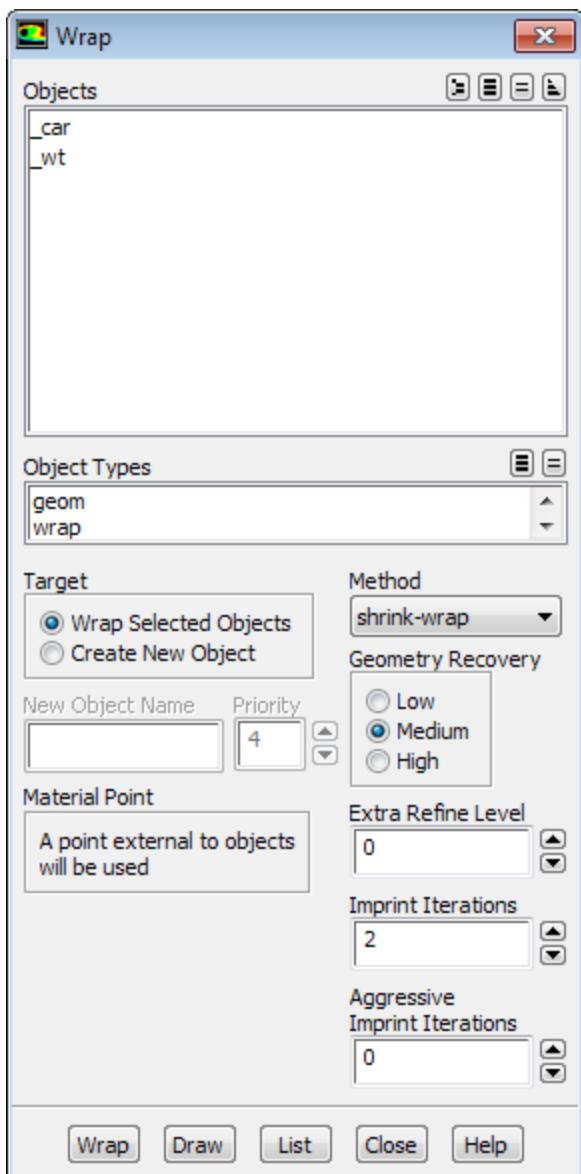
displays the selected object(s) in the graphics window.

List

reports (in the console) the object name, cell zone type, priority, object type, and the face and edge zones comprising the object.

21.1.11. Wrap Dialog Box

The **Wrap** dialog box contains options for the object wrapping operation, which allows you to obtain good quality, well-connected representations of each topological body.



Controls

Objects

contains a list of the defined geometry and wrap objects.

Object Types

contains the list of the object types. If you select an object type from the list, all objects of that type will be selected in the **Objects** list. You can also select multiple types to select all objects of different types (e.g., **geom** and **wrap**).

Target

contains options for the wrap object to be created.

Wrap Selected Objects

allows you to create a conformal, well-connected surface mesh (wrap object) for each of the objects selected.

Note

If you wrap a **geom** object with this option, the **wrap** objects created will have the same name as the corresponding **geom** objects. The face zones associated with the **wrap** object are those created by the wrapping operation. But you will have **geom** as well as **wrap** face zones. The original **geom** face zones are no longer associated with the object. To revert to the original geometry representation, use the **Purge Wrap** option in the [Manage Objects Dialog Box \(p. 430\)](#).

Create New Objects

allows you to extract the flow volume, when the surrounding solids are not needed in the surface mesh.

New Object Name

specifies the name of the wrap object to be created when the **Create New Object** option is used.

Priority

specifies the priority of the wrap object to be created when the **Create New Object** option is used.

Material Point

specifies the material point to be used for the object wrapping operation. An external material point is used for the **Wrap Selected Objects** option. You can choose to use an external point or specify an appropriate material point when the **Create New Object** option is selected. Select the appropriate point in the drop-down list.

Draw

displays the material point in the graphics window.

Create...

opens the [Create Material Point Dialog Box \(p. 427\)](#), where you can define material points.

Method

allows you to select the method used for the object wrapping operation.

cut-wrap

uses the CutCell mesher to extract a well connected, triangulated surface mesh for the geometry object(s) selected.

shrink-wrap

uses a specialized version of the boundary wrapper utility to extract a well connected surface mesh for the geometry object(s) selected.

Geometry Recovery

contains options for recovering the geometry when creating the wrap object by the shrink-wrap method.

The quality of the resulting wrap object is affected by the quality of the input geometry as well as your choice of geometry recovery option.

Low

allows you to create a rough wrapped representation of the geometry object. You can optionally choose to rezone the recovered zones.

Medium

performs additional refinement, imprinting and aggressive imprinting iterations to improve the feature recovery. Individual zones are recovered based on the original geometry object and then optionally rezoned.

Extra Refine Level

allows you to produce a very fine capturing of details by reducing the size function minimum and maximum values by half per extra level specified.

Imprint Iterations

allows you to specify the number of imprinting iterations for feature capturing. The default number of iterations is 2, but you can increase the number up to a maximum limit of 5 for detailed features to be captured.

Aggressive Imprint Iterations

allows you to specify additional iterations for aggressive imprinting to improve feature capture. You can increase the number up to a maximum limit of 5.

High

allows better feature capture and surface remeshing based quality improvement. You can also use sampling coarser/finer than the final surface mesh by specifying an appropriate resolution factor. Degenerate and island edges are deleted, and intersected and remeshed as appropriate. The edges are imprinted on the wrapped zones and individual zones are recovered based on the original geometry object and then rezoned. Surfaces are remeshed based on size functions/size field.

Resolution Factor

specifies the sampling size to be used by the wrapper utility. A value greater than 1 indicates that the sampling size is coarser while a value less than 1 indicates that the sampling size is finer than the sizes for remeshing.

Wrap

creates the wrap object(s) for the object(s) selected in the **Objects** selection list based on the specified parameters and the method selected.

Draw

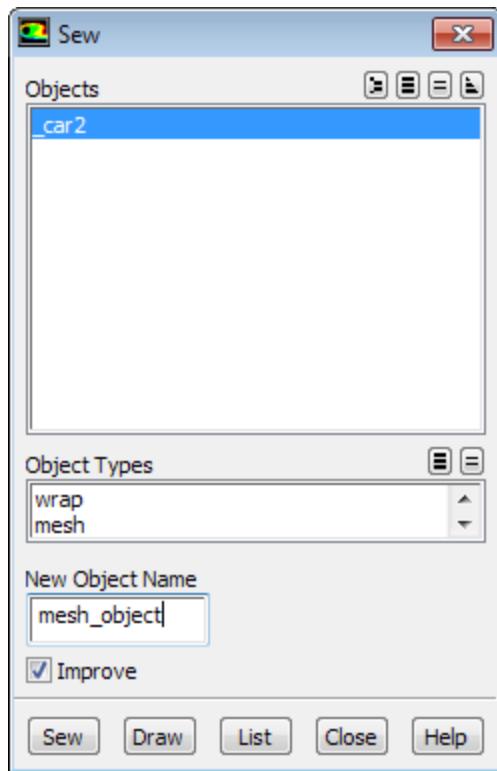
displays the selected object(s) in the graphics window.

List

reports (in the console) the object name, cell zone type, priority, object type, and the face and edge zones comprising the object.

21.1.12. Sew Dialog Box

The **Sew** dialog box contains options for creating the conformal, triangular surface mesh (mesh object) for the selected wrap objects.



Controls

Objects

contains a list of wrap and mesh objects.

Object Types

contains the list of the object types. If you select an object type from the list, all objects of that type will be selected in the **Objects** list. You can also select multiple types to select all objects of different types.

New Object Name

specifies the name of the mesh object to be created by the sew operation.

Improve

allows you to improve the surface mesh quality of the mesh object created by the sewing operation. This option is enabled by default.

When this option is disabled, you may need to improve the surface mesh quality of the mesh object created. You can use the options in the [Diagnostic Tools Dialog Box \(p. 437\)](#) and [Improve Dialog Box \(p. 459\)](#) to improve the surface mesh quality.

Sew

creates the conformal, triangular surface mesh (mesh object) for the selected wrap objects.

Draw

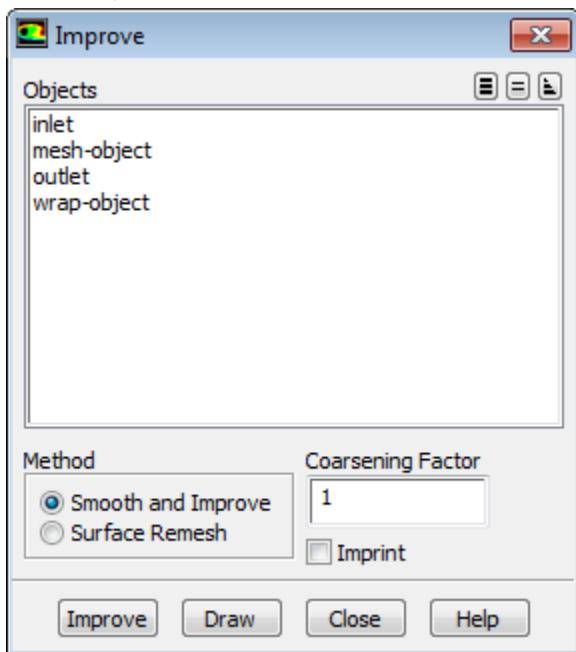
displays the selected object(s) in the graphics window

List

reports (in the console) the object name, cell zone type, priority, object type, and the face and edge zones comprising the object.

21.1.13. Improve Dialog Box

The **Improve** dialog box contains options that allow you to improve the surface mesh quality of the mesh objects.



Controls

Objects

contains a list of wrap and mesh objects.

Method

contains options for improving the wrap and mesh object surface mesh.

Smooth and Improve

improves the mesh by a combination of smoothing, swapping, and improvement operations, along with correcting the orienting of the object normals and deleting of island faces. You can optionally coarsen the surface mesh by specifying a suitable coarsening factor. Additional imprinting can be done to improve feature capture on the surface mesh.

Coarsening Factor

allows you to coarsen the surface mesh compared to that of the wrap object. The coarsening will be limited by the current global maximum size.

Imprint

allows additional imprinting of feature edges to improve the surface mesh quality. This option is disabled by default.

Surface Remesh

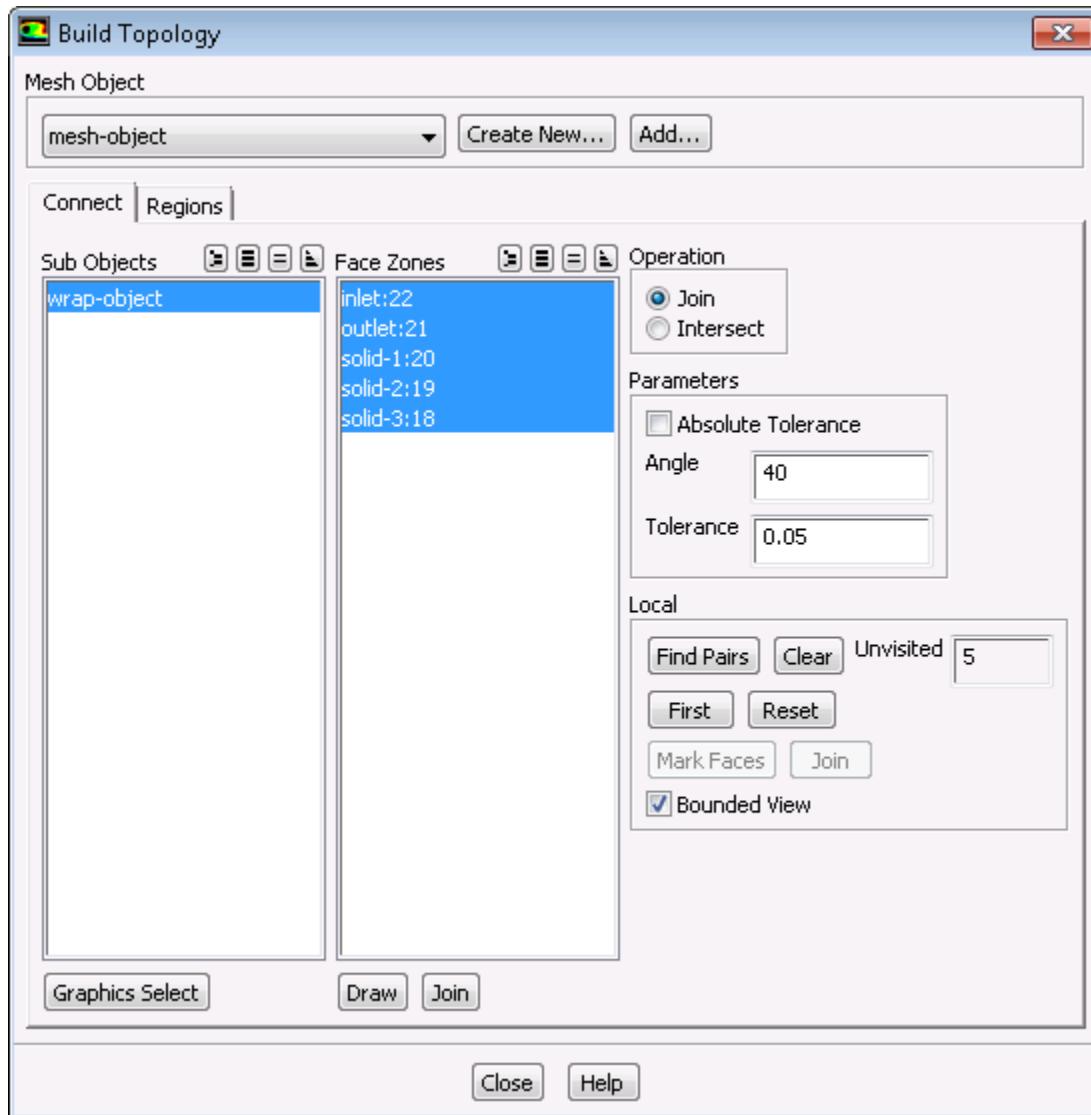
improves the mesh by remeshing based on the current size functions/size field. Object normals are correctly oriented and island faces are also deleted.

Improve

improves the surface mesh quality of the selected wrap/mesh objects, based on the method selected.

21.1.14. Build Topology Dialog Box

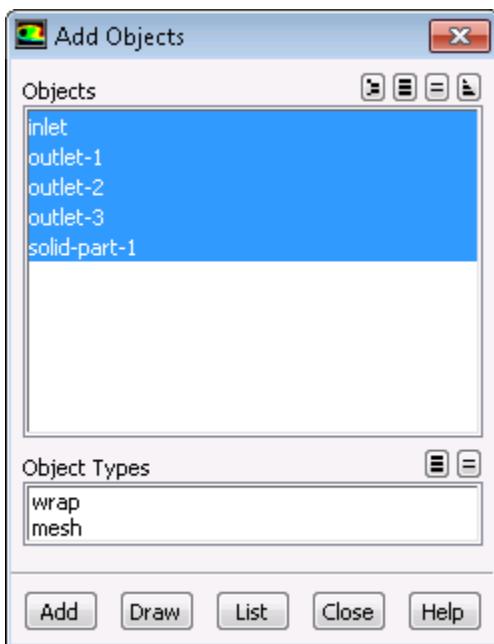
The **Build Topology** dialog box allows you to connect wrap objects into closed Regions for creation of cell zones.



Mesh Object

A drop-down list of available mesh objects.

Choose a mesh object from the list and click **Add..** to open the **Add Objects** dialog box to append wrap or mesh objects to an existing mesh object.



Or click **Create New...** to open the **New Mesh Object** dialog box and create a new mesh object.



Connect

Use this tab to connect (join or intersect) wrap objects.

Sub Objects

A list of wrap objects included in the selected **Mesh Object**.

Graphics Select

allows you to select **Sub Object**(s) associated with the face zone(s) selected in graphics window.
Select a zone, or zones, in the graphics window, and then click **Graphics Select**.

Face Zones

A list of **Face Zones** included in the selected **Sub Objects**.

Draw

displays the selected **Face Zones**.

Join (Intersect)

Global **Join (Intersect)**, under the **Face Zones** list, performs the selected **Operation** on all selected **Face Zones**.

Operation**Join**

used to connect overlapping pairs of face zones within or between sub-objects.

Intersect

used to connect crossing pairs of face zones within or between sub-objects.

Parameters

allows you to specify the **Parameters**, which control the size of the connecting region, to be used for the selected **Operation**.

Angle

specifies the maximum angle between normals of faces being connected.

Tolerance

specifies the maximum separation between faces being connected.

Local

Controls to manually step through the possible **Join (Intersect)** pairs of **Sub Objects**.

Find Pairs

searches for pairs of face zones that meet the **Angle** and **Tolerance** parameters. The number of pairs is displayed as the **Unvisited** value.

Clear

clears earlier found pairs, zeroes the **Unvisited** value, and re-enables the **Find Pairs** button.

Unvisited

displays the number of overlapping (intersecting) pairs within the **Parameters** for the selected **Operation**.

First (Next)

draws the first pair of overlapping (intersecting) face zones in the graphics window and changes the button label to **Next**. Click **Next** to cycle through all pairs.

Reset

restarts the pair traversal and changes the button label **Next** to **First**. Also resets the **Unvisited** value to show the number of disconnected pairs.

Join (Intersect)

Click to perform the selected **Operation** on only the drawn pair.

Mark Faces

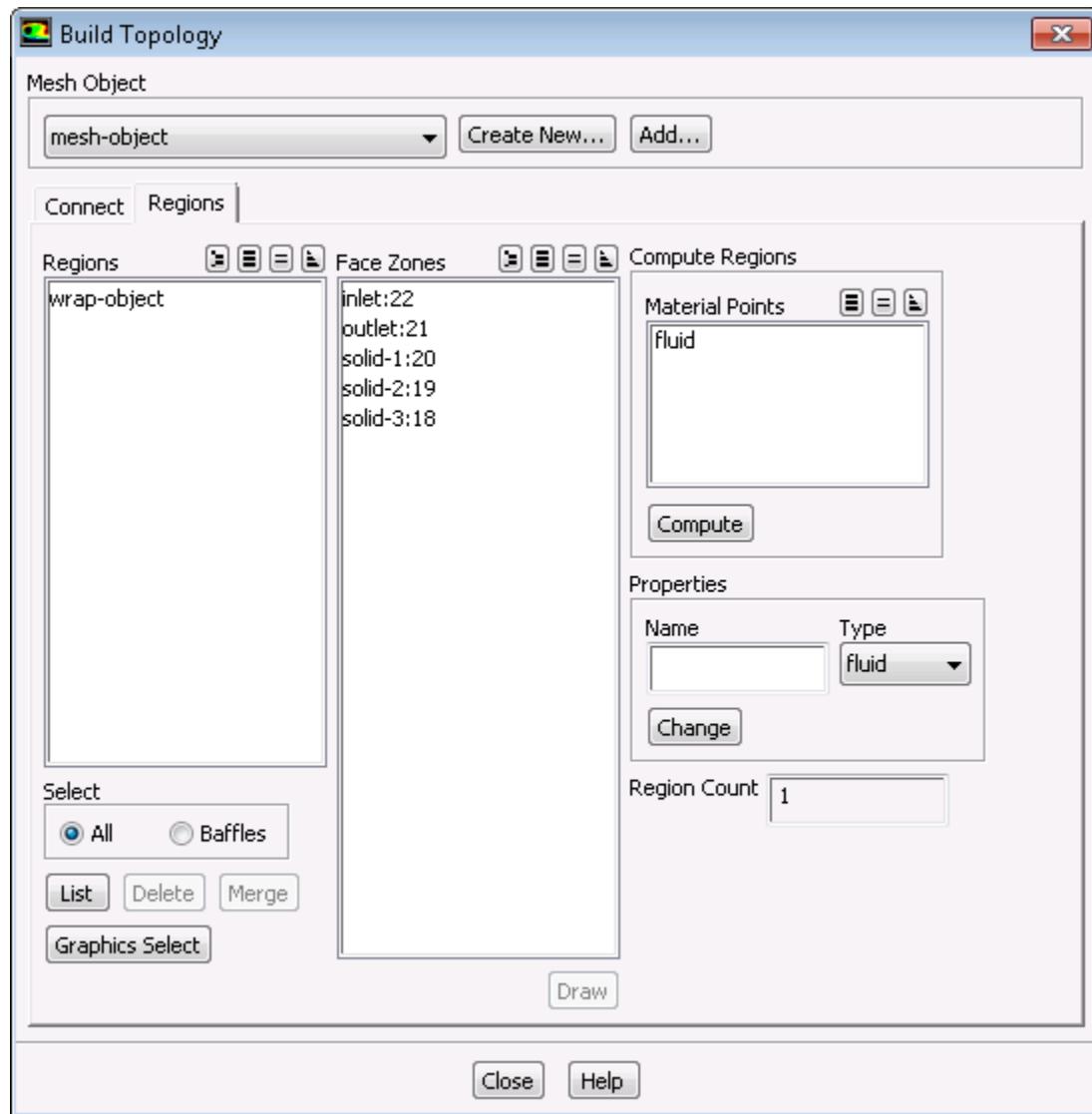
marks the faces graphically to provide a preview of the selected **Operation** for the currently selected pair.

Bounded View

allows you to set or remove the graphical bounds to show cropped or complete view of drawn pair.

Regions

Use this tab to manage the cell zone Regions.

**Regions**

The list of computed cell zones.

Select

allows you to select **All** face zones or only **Baffles** in the **Face Zones** selection list.

All

selects all face zones included in the selected regions.

Baffles

selects only baffle face zones included in the selected regions.

List

Information about the **Regions**, including region name, type, material-point and face-zones, is displayed in the console sorted by volume.

Delete

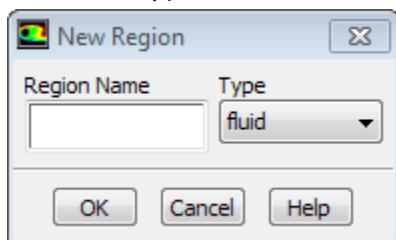
The selected **Region** and all of its **Face Zones**, except those which are shared by other **Regions**, is deleted from the **Mesh Object**.

Merge

Selected **Regions** are joined into a single **Region**.

Note

When merging more than one named region, or merging into the region claimed by the material point, a **New Region** dialog box opens where you specify the name and type.

**Graphics Select**

allows you to select **Regions** associated with the **Face Zone(s)** selected in graphics window. Select a **Face Zone** in the graphics window, and then click **Graphics Select** to select the **Region** from the list.

Face Zones

A list of Face Zones included in the selected **Region**

Draw

The selected **Face Zones** are displayed in the graphics window.

Compute Regions

Select a **Material Point**, if desired, and then click **Compute** to identify the cell zones.

Properties

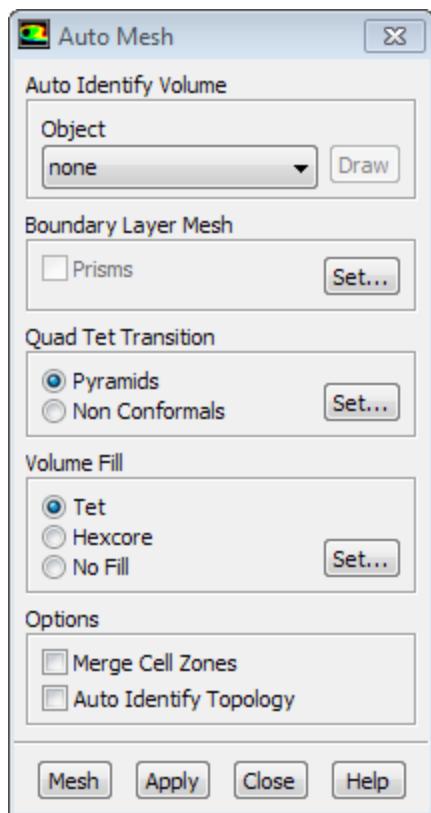
Enter a name and choose a cell zone **Type** (solid, fluid or dead) from the drop-down list, if desired.

Region Count

displays the number of computed **Regions**. This number should be greater than or equal to the number of **Sub Objects**.

21.1.15. Auto Mesh Dialog Box

The **Auto Mesh** dialog box allows you to automatically create the volume mesh. You can specify the mesh elements to be used and set appropriate parameters for the same.



Controls

Auto Identify Volume

contains options for selecting a mesh object and material point(s) for generating the volume mesh.

Object

contains a list of mesh objects available. You can select the appropriate mesh object for generating the volume mesh.

The default selection in the **Object** list is **none**. In this case, the volume mesh will be generated for the face zones in the current domain.

Draw

draws the mesh object selected in the **Object** drop-down list in the graphics window.

Boundary Layer Mesh

contains options for creating the boundary layer (prism) mesh.

Prisms

allows you to create prism layers in the geometry.

Set...

opens the **Prisms** dialog box.

Note

The **Prisms** option in the **Auto Mesh** dialog box will be visible only after applying zone-specific growth (by clicking **Apply** in the **Zone Specific Growth** group box in the **Prisms** dialog box).

Quad Tet Transition

contains options for specifying the quad-tet transition elements for a hybrid mesh.

Pyramids

specifies that pyramids will be created as transitional elements between the quadrilateral and triangular elements.

Non Conforms

specifies that a non-conformal interface will be created as a transition between the quadrilateral and triangular elements.

Set...

opens the **Pyramids** dialog box or the **Non Conforms** dialog box as appropriate.

Volume Fill

contains options for volume meshing.

Tet

allows you to mesh the geometry with tetrahedral/hybrid mesh.

Hexcore

allows you to mesh the geometry with hexahedral mesh in the core flow region.

No Fill

allows you to mesh the boundary layers with prisms and pyramids (or non-conforms) as specified, without creating tetrahedral/hexahedral cells. The **No Fill** option also creates the domain for the volume mesh but does not activate the domain created.

Set...

opens the **Tet** dialog box or the **Hexcore** dialog box, as appropriate.

Options

contains additional meshing options.

Merge Cell Zones

toggles the merging of cell zones during meshing.

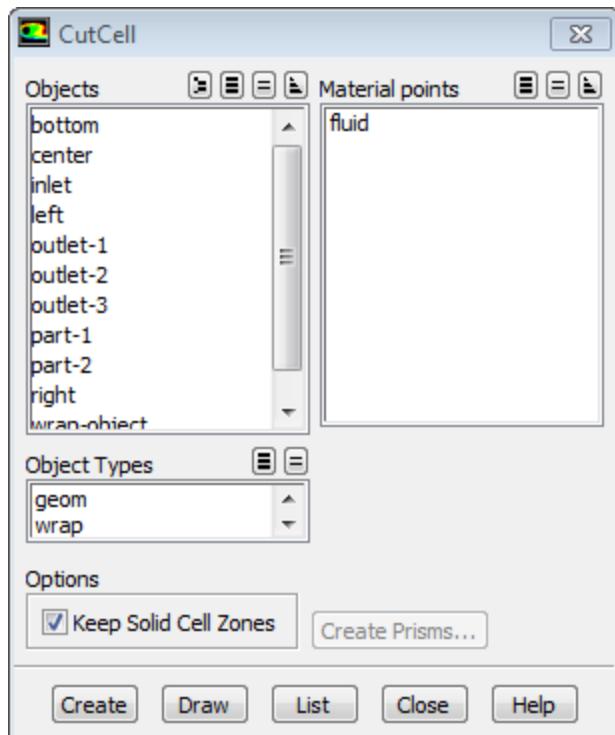
Auto Identify Topology

enables improved domain identification based on topology information. When enabled, face zones to be meshed on both sides will be identified and domains will be created based on the topology information extracted. The face normals will also be oriented properly (into the closed domain rather than outside).

These options are not available when a mesh object is selected for volume meshing.

21.1.16. CutCell Dialog Box

The **CutCell** dialog box allows you to create the CutCell mesh based on the object(s) and material point(s) selected.



Controls

Objects

contains a list of objects (geom or wrap) from which you can select the object(s) for creating the CutCell mesh.

Material points

contains a list of material points defined from which you can select the material point(s) for creating the CutCell mesh.

Options

contains additional CutCell meshing options.

Keep Solid Cell Zones

allows you to retain the solid cell zones when the CutCell mesh is generated.

Create Prisms...

opens the [Prisms Dialog Box \(p. 539\)](#), where you can set the parameters for post CutCell meshing prism generation.

Create

creates the CutCell mesh based on the object(s) and material point(s) selected.

Draw

draws the object(s) selected in the graphics window.

List

reports (in the console) the object name, cell zone type, priority, object type, the face and edge zones comprising the object, and the object reference point (if any).

21.2. File Menu

21.2.1. File/Read/Mesh...

The **File/Read/Mesh...** menu item opens the [Select File Dialog Box \(p. 33\)](#), which allows you to select the appropriate mesh file(s) to be read.

21.2.2. File/Read/Case...

The **File/Read/Case...** menu item opens the [Select File Dialog Box \(p. 33\)](#), which allows you to select the appropriate case file(s) to be read.

21.2.3. File/Read/Boundary Mesh...

The **File/Read/Boundary Mesh...** menu item opens the [Select File Dialog Box \(p. 33\)](#), which allows you to select the appropriate boundary mesh file(s) to be read.

21.2.4. File/Read/Size Field...

The **File/Read/Size Field...** menu item opens the [Select File Dialog Box \(p. 33\)](#), which allows you to select the appropriate size field file to be read.

21.2.5. File/Read/Scheme...

The **File/Read/Scheme...** menu item opens the [Select File Dialog Box \(p. 33\)](#), which allows you to select the appropriate scheme file to be read.

21.2.6. File/Read/Journal...

The **File/Read/Journal...** menu item opens the [Select File Dialog Box \(p. 33\)](#), which allows you to select the appropriate journal file to be read.

21.2.7. File/Read/Domains...

The **File/Read/Domains...** menu item opens the [Select File Dialog Box \(p. 33\)](#), which allows you to select the appropriate domain file to be read.

21.2.8. File/Write/Mesh...

The **File/Write/Mesh...** menu item opens the [Select File Dialog Box \(p. 33\)](#), which allows you to save the mesh file with a name of your choice.

21.2.9. File/Write/Case...

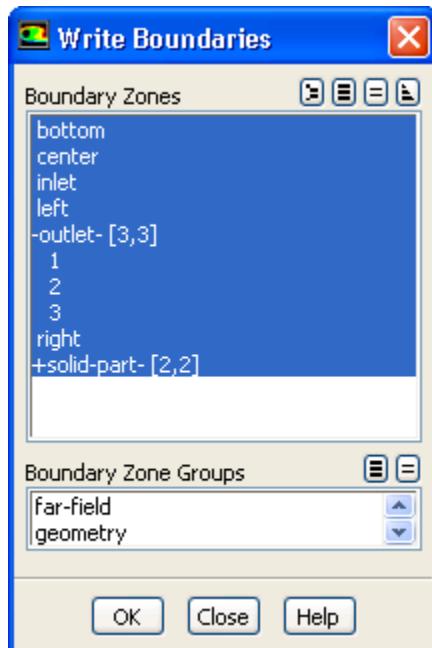
The **File/Write/Case...** menu item opens the [Select File Dialog Box \(p. 33\)](#), which allows you to save the case file with a name of your choice.

21.2.10. File/Write/Boundaries...

The **File/Write/Boundaries...** menu item opens the [The Write Boundaries Dialog Box \(p. 469\)](#), which allows you to select the boundary zones to be written to a mesh file.

21.2.10.1. The Write Boundaries Dialog Box

The **Write Boundaries** dialog box allows you to specify the boundaries to be written to a mesh file.



Controls

Boundary Zones

contains a list of the boundary zones available.

Boundary Zone Groups

contains a list of the boundary zone groups and user-defined groups available.

21.2.11. File/Write/Domains...

The **File/Write/Domains...** menu item opens the [Select File Dialog Box \(p. 33\)](#), which allows you to save the domain file with a name of your choice.

21.2.12. File/Write/Size Field...

The **File/Write/Size Field...** menu item opens the [Select File Dialog Box \(p. 33\)](#), which allows you to specify the name of the size field file to be written.

21.2.13. File/Write/Start Journal...

The **File/Write/Start Journal...** menu item is used to start the recording of subsequent commands to a journal file. You can read this journal file back into the application later (using the **File/Read/Journal...** menu item) to automate the execution of the recorded commands. See [Creating and Reading Journal Files \(p. 66\)](#) for details.

21.2.14. File/Write/Stop Journal

The **File/Write/Stop Journal** replaces the **File/Write/Start Journal...** menu item after the recording of a journal file has begun. The **File/Write/Stop Journal** menu item is used to end the journal recording. See [Creating and Reading Journal Files \(p. 66\)](#) for details.

21.2.15. File/Write/Start Transcript...

The **File/Write/Start Transcript...** menu item is used to start the recording of a transcript file containing all input to and output from the application. (You cannot read a transcript file back into the application.) See [Creating Transcript Files \(p. 68\)](#) for details.

21.2.16. File/Write/Stop Transcript

The **File/Write/Stop Transcript** menu item replaces the **File/Write/Start Transcript...** menu item after the recording of a transcript file has begun. The **File/Write/Stop Transcript** menu item is used to end the transcript recording. See [Creating Transcript Files \(p. 68\)](#) for details.

21.2.17. File/Import/ANSYS prep7/cdb/Surface...

The **File/Import/ANSYS prep7/cdb/Surface...** menu item allows you to import an ANSYS Prep7 surface file.

21.2.18. File/Import/ANSYS prep7/cdb/Volume...

The **File/Import/ANSYS prep7/cdb/Volume...** menu item allows you to import an ANSYS Prep7 volume file.

21.2.19. File/Import/CGNS/Surface...

The **File/Import/CGNS/Surface...** menu item allows you to import a CGNS surface file.

21.2.20. File/Import/CGNS/Volume...

The **File/Import/CGNS/Volume...** menu item allows you to import a CGNS volume file.

21.2.21. File/Import/FIDAP neutral/Surface...

The **File/Import/FIDAP neutral/Surface...** menu item allows you to import a FIDAP neutral surface file.

21.2.22. File/Import/FIDAP neutral/Volume...

The **File/Import/FIDAP neutral/Volume...** menu item allows you to import a FIDAP neutral volume file.

21.2.23. File/Import/GAMBIT neutral/Surface...

The **File/Import/GAMBIT neutral/Surface...** menu item allows you to import a GAMBIT surface file.

21.2.24. File/Import/GAMBIT neutral/Volume...

The **File/Import/GAMBIT neutral/Volume...** menu item allows you to import a GAMBIT volume file.

21.2.25. File/Import/HYPERMESH Ascii/Surface...

The **File/Import/HYPERMESH Ascii/Surface...** menu item allows you to import a HYPERMESH surface file.

21.2.26. File/Import/HYPERMESH Ascii/Volume...

The **File/Import/HYPERMESH Ascii/Volume...** menu item allows you to import a HYPERMESH volume file.

21.2.27. File/Import/IDEAS universal/Surface...

The **File/Import/IDEAS universal/Surface...** menu item allows you to import an IDEAS universal surface file.

21.2.28. File/Import/IDEAS universal/Volume...

The **File/Import/IDEAS universal/Volume...** menu item allows you to import an IDEAS universal volume file.

21.2.29. File/Import/NASTRAN/Surface...

The **File/Import/NASTRAN/Surface...** menu item allows you to import a NASTRAN surface file.

21.2.30. File/Import/NASTRAN/Volume...

The **File/Import/NASTRAN/Volume...** menu item allows you to import a NASTRAN volume file.

21.2.31. File/Import/PATRAN neutral/Surface...

The **File/Import/PATRAN neutral/Surface...** menu item allows you to import a PATRAN surface file.

21.2.32. File/Import/PATRAN neutral/Volume...

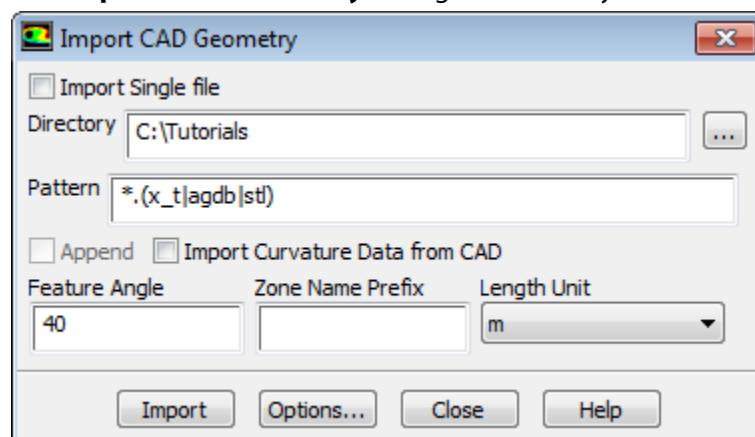
The **File/Import/PATRAN neutral/Volume...** menu item allows you to import a PATRAN volume file.

21.2.33. File/Import/CAD...

The **File/Import/CAD...** menu item allows you to import CAD models using the CAD readers or associative geometry interfaces (via plug-ins).

21.2.33.1. Import CAD Geometry Dialog Box

The **Import CAD Geometry** dialog box allows you to set the basic options for importing CAD files.



Controls**Import Single File**

allows you to import a single file as opposed to multiple files from the same directory. When this option is disabled, you can import multiple files from the specified directory.

File

specifies the CAD file to be imported when the **Import Single File** option is enabled.

Note

Ensure that the file path contains the appropriate platform-specific separators (e.g., **C:\Tutorials** on Windows systems, **Home/Tutorials** on Linux systems)

The following special characters are supported in the file name:

On Windows systems– + \$ ^ () [] { } @ # % _ - = , . ; '	~ ` !
On Linux systems– + \$ ^ () [] { } @ # % _ - = , . : ; '	> " ~ ` !

Directory

specifies the directory containing the CAD file(s) to be imported when the **Import Single File** option is disabled.

Note

Ensure that the path contains the appropriate platform-specific separators (e.g., **C:\Tutorials** on Windows systems, **Home/Tutorials** on Linux systems).

Pattern

specifies the file name pattern(s) or file type(s) to be imported from the directory. You can specify multiple patterns or file types to be imported separated by (e.g., *. (x_t | agdb | stl)).

Append

allows you to append the imported mesh information to the existing mesh information in the current session. This option is disabled by default and is available only when a mesh has been read in the current session.

Import Curvature Data from CAD

allows you to import the curvature data from the nodes of the facets.

Feature Angle

allows you to specify the feature angle to determine the features to be imported. The default value is 40.

Zone Name Prefix

specifies the zone name prefix to be added on import. The default prefix is a blank entry, where the zone names from the CAD file are retained.

Note

Valid characters include all alphanumeric characters as well as the following special characters:

_ + - : .

All other characters, including spaces, are invalid. If an invalid character is specified, it is replaced by a hyphen (-) upon import.

Length Unit

specifies the length unit to scale the mesh to on import. Models created in other units will be scaled accordingly. The default is meters (**m**).

Important

The imported CAD models are scaled based on the length unit selected for the meshing mode session only. When the model is transferred to solution mode, the model units are reverted to the original CAD units. Refer to [Scaling the Mesh](#) in the [Fluent User's Guide](#) for details on scaling the mesh in solution mode.

Options...

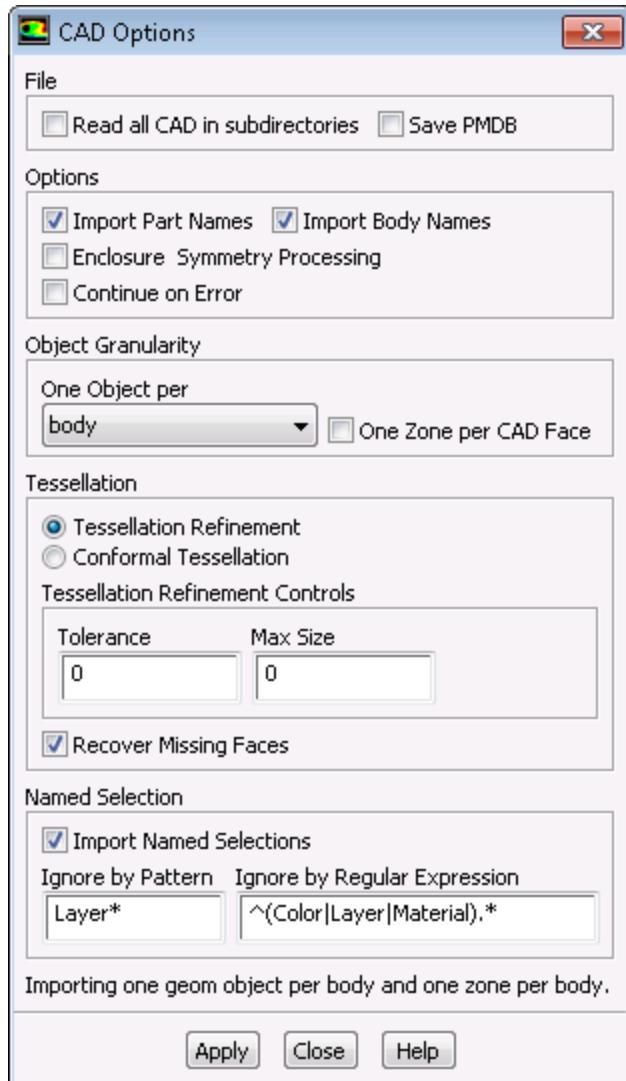
opens the [CAD Options Dialog Box \(p. 473\)](#) where you can set additional options for importing CAD files.

Import

imports the CAD file(s) based on the options set.

21.2.33.2. CAD Options Dialog Box

The **CAD Options** dialog box allows you to set additional options for importing CAD files.



Controls

File

contains options related to file processing during import.

Read all CAD in subdirectories

when enabled, all CAD files in the specified directory as well as in its subdirectories will be imported.
This option is disabled by default.

Save PMDB

saves a PMDB (*.pmdb) file in the directory containing the CAD files imported. You can use this file to import the same CAD file(s) again with different options set, for a quicker import than the full import. This option is disabled by default.

Note

Some options will not be available any more once the model is imported from a PMDB file (e.g., **Enclosure Symmetry Processing**), since they are processed before the PMDB file is created.

Options

contains miscellaneous options for the CAD import.

Import Part Names

allows import of Part names from the CAD file(s). This option is enabled by default.

Note

Any renaming of Part names in ANSYS Mechanical/ANSYS Meshing prior to the export of the mechdat/meshdat files is ignored during import. Only original Part names will be imported.

Import Body Names

allows import of Body names from the CAD file(s). This option is enabled by default.

Note

Any renaming of Body names in ANSYS Mechanical/ANSYS Meshing prior to the export of the mechdat/meshdat files is ignored during import. Only original Body names will be imported.

Enclosure Symmetry Processing

allows processing of enclosure and symmetry named selections during import. This option is disabled by default. This option is applicable only to ANSYS DesignModeler (*.agdb) files.

Continue on Error

continues the import of the CAD file(s), despite errors or problems creating the faceting on certain surfaces, or other issues. This option is disabled by default.

Object Granularity

contains options for controlling the granularity of objects imported from the CAD files.

One Object per

allows you to create one object per body/part/file to be imported.

One Zone per CAD Face

allows you to create one zone per CAD face imported. This option is available only when you choose to import one object per body and is disabled by default.

The object granularity is indicated in the **CAD Options** dialog box. Refer to [Importing CAD Files \(p. 72\)](#) for details on the object granularity.

Tessellation

contains options that allow you to control the tessellation (faceting).

Tessellation Refinement

allows you to control the tessellation (faceting) refinement during the file import.

Tessellation Refinement Controls

contains options that allow you to control the tessellation refinement.

Tolerance

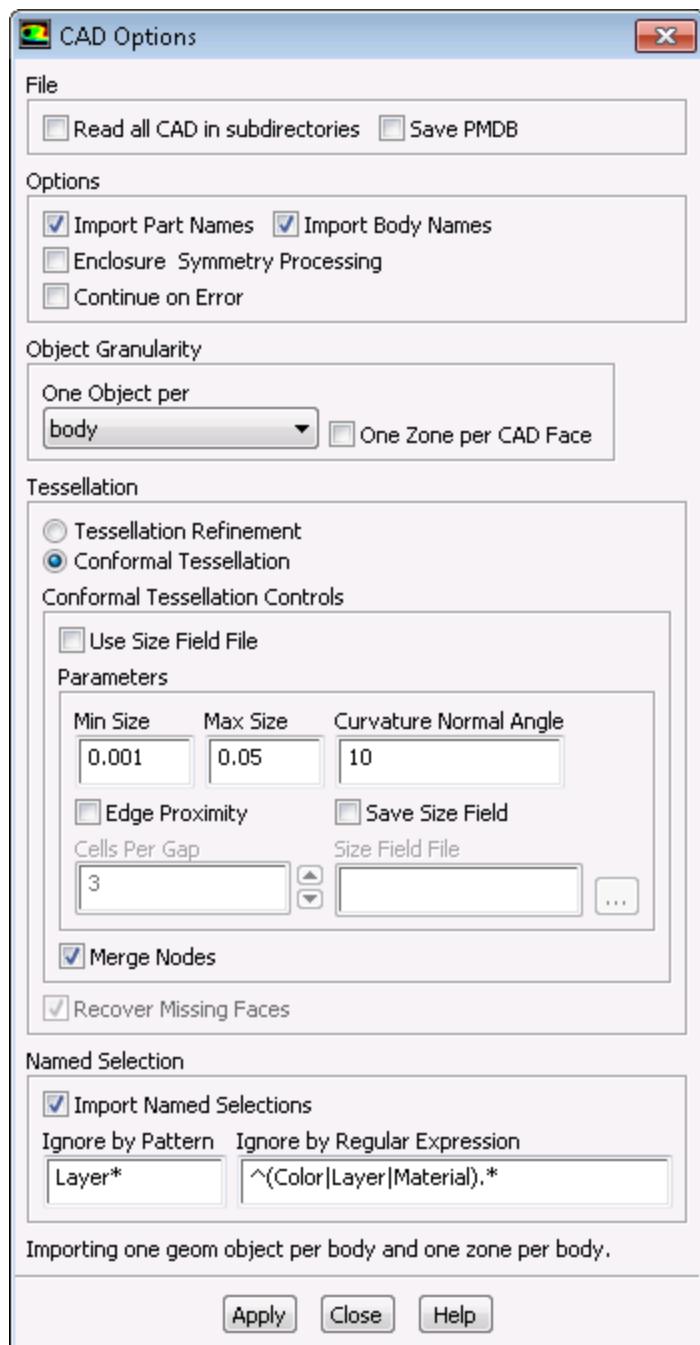
specifies the tolerance for the tessellation (faceting) refinement when **Tessellation Refinement** is selected. The default value is 0, which implies no tessellation (faceting) refinement during import.

Note

It is recommended that you use the default value of 0 for an initial (diagnostic) import. You can then determine the minimum size you intend to use for the mesh and import the file(s) again using a **Tolerance** value 1/10th the intended minimum size.

Max Size

specifies a maximum facet size for the imported model to avoid very large facets.



Conformal Tessellation

allows you to obtain conformal tessellation (faceting) using the curvature size function based on the minimum and maximum facet sizes, and the facet curvature normal angle specified and the edge proximity size function.

Note

For STL type models, the **Conformal Tessellation** option will not remesh the original facets. You can use the options in the [Wrap Dialog Box \(p. 454\)](#) to obtain conformal

faceting instead. The STL file should originate from supported CAD formats for better results.

Tip

It is recommended that you check for unexpected free faces when the conformal tessellation option is used for import. Free faces may indicate failure to generate conformal faceting for the corresponding geometry faces.

Conformal Tessellation Controls

contains options that allow you to control the conformal tessellation created.

Use Size Field File

if enabled, you can give a **Size Field File** to obtain conformal faceting.

Min Size

specifies the minimum facet size to be used for creating conformal faceting.

Max Size

specifies the maximum facet size to be used for creating conformal faceting.

Note

The **Max Size** value is limited to 1/10th the bounding box diagonal.

Curvature Normal Angle

specifies the curvature normal angle to be used for refining the conformal faceting based on the underlying curve and surface curvature.

Tip

In multipart cases where there are overlapping curved surfaces, the default curvature normal angle may result in faceting unlike the required conformal tessellation on adjacent (shared) geometry. This may create artificial gaps between such overlapping surfaces which can cause problems during further operations like sewing with face-face proximity specified. For such cases, it is recommended that you reduce the value of the curvature normal angle to 5 or less to obtain similar tessellation on the overlapping surfaces and avoid the creation of artificial gaps.

It is also recommended to increase the relative tolerance (size-functions/set-prox-gap-tolerance) to avoid false proximity computed at such artificial gaps.

Edge Proximity

enables the use of the edge proximity size function for creating the conformal tessellation, based on the number of cells per gap specified.

Cells Per Gap

specifies the number of layers of elements to be generated in the gap for the edge proximity size function.

Save Size Field

enables you to save the size field in a file, which is computed based on the defined parameters i.e. **Min Size**, **Max Size**, **Curvature Normal Angle**, **Cells Per Gap**.

Size Field File

specifies the file name for the size field file.

Merge Nodes

allows you to merge free nodes after import. Nodes will be merged at the object level with a tolerance of 1e-10.

Recover Missing Faces

allows you to recreate missing face tessellations during the file import.

Note

This option is always enabled when **Conformal Tessellation** is selected.

Named Selection

contains options for processing **Named Selections** from the CAD file(s), including Named Selections from ANSYS DesignModeler, publications from CATIA, etc.

Note

Named Selections defined in ANSYS Meshing cannot be imported.

Import Named Selections

imports the **Named Selections** set up in the CAD file(s). This option is enabled by default.

Ignore by Pattern

allows you to ignore import of certain Named Selections based on the pattern specified (e.g., Layer* to ignore layer Named Selections from CATIA).

Ignore by Regular Expression

allows you to ignore certain Named Selections by specifying multiple wild cards (e.g., ^ (Col- or |Layer|Material) . * to remove color, layer, and material Named Selections from CATIA).

21.2.34. File/Import/Fluent 2D Mesh...

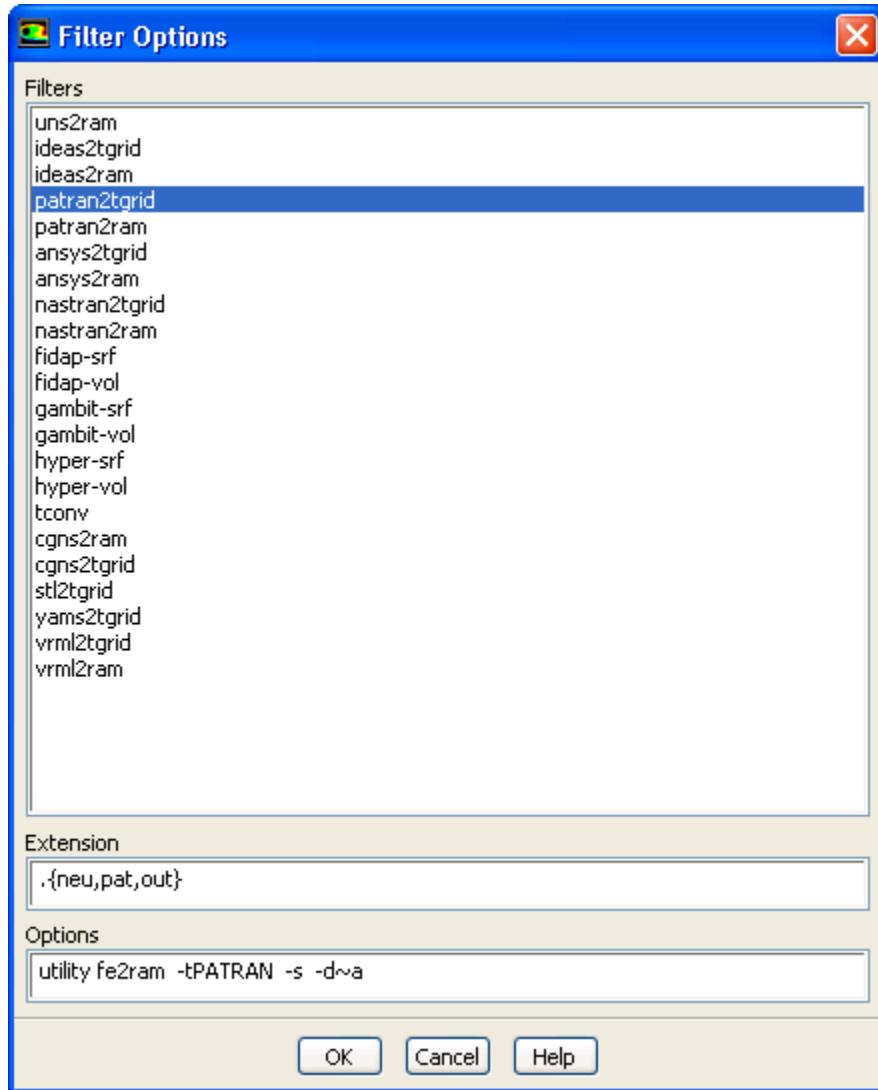
The **File/Import/Fluent 2D Mesh...** menu item allows you to read a 2D mesh file from ANSYS Fluent in the 3D version.

21.2.35. File/Import/Options...

The **File/Import/Options...** menu item opens the [Filter Options Dialog Box \(p. 480\)](#), which allows you to control import arguments and the extensions of the files to be converted.

21.2.35.1. Filter Options Dialog Box

The **Filter Options** dialog box allows you to change the extension (e.g., .cas, .msh, .neu) and arguments used with a specified filter.



Controls

Filters

contains a list of the available file converters. When you select a name from this list, the corresponding **Extension** and **Options** will be displayed.

A single filter (**fe2ram**) is used for files from all CAD packages. The arguments of the **fe2ram** filter will indicate the CAD package used to create the file. The name selected in the **Filters** list (e.g., **ideas2tgrid**) identifies the CAD package that created the file, and whether the file is a boundary mesh or volume mesh.

For example, **ideas2tgrid** is used for importing a boundary mesh from I-deas, and **ideas2ram** is used for importing a volume mesh from I-deas. When you select a filter name in the **Filters** list, the appropriate file extension and arguments for the **fe2ram** filter will appear under **Extension** and **Options**.

Extension

is the extension of the third-party mesh file to be imported.

Options

are the arguments used by the filter selected in the **Filters** list (or by the **fe2ram** filter).

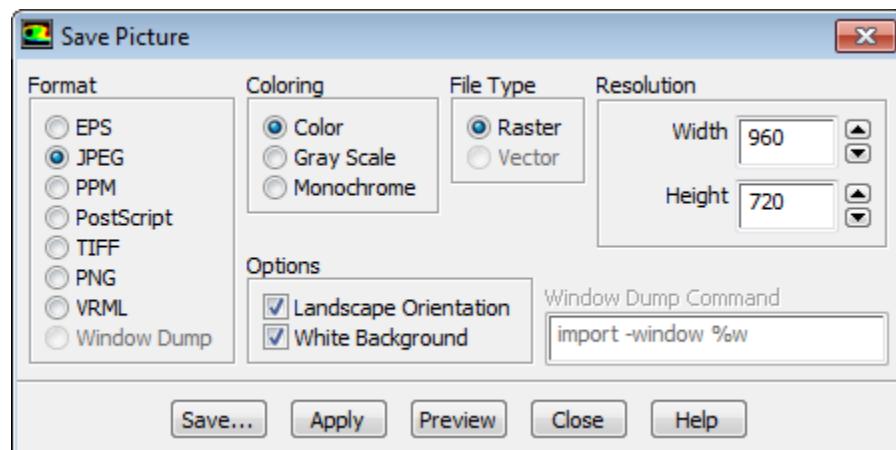
For some filters, one of the arguments will be the dimensionality of the grid. For such filters, a default dimensionality argument of `-d~ais` is shown. The dimensionality of the grid will be automatically determined, so you needn't substitute `~a`. For information about the import filters and their related arguments, see [Appendix A \(p. 647\)](#).

21.2.36. File/Save Picture...

The **File/Save Picture...** menu item opens the [Save Picture Dialog Box \(p. 481\)](#).

21.2.36.1. Save Picture Dialog Box

The **Save Picture** dialog box allows you to set graphics picture parameters and save files of graphics windows.

**Format**

allows you to select the format of picture files.

Coloring

specifies the color mode for picture files. Images may be made in **Color**, **Gray Scale**, or **Monochrome** (black and white). Most monochrome PostScript devices will render **Color** images in shades of gray.

Select **Gray Scale** to ensure that the color ramp is rendered as a linearly-increasing gray ramp.

File Type

specifies whether a **Raster** or **Vector** type file is to be saved. You can choose either of these if you are saving a PostScript or EPS file. See [Choosing the File Type \(p. 80\)](#) for details.

Resolution

specifies the size of raster files in pixels. If the **Width** and **Height** are both zero, the picture is generated at the same resolution as the active graphics window. For PostScript and EPS files, specify the resolution in dots per inch (DPI).

Options

specifies whether or not to use the following options:

Landscape Orientation

specifies the orientation of the picture. If selected, the image is made in landscape mode; otherwise, it is made in portrait mode.

White Background

specifies that the white background should be used in the picture. This feature allows you to make images with a white background and a black foreground.

Save...

opens the **Select File** dialog box, where you can specify a name for the picture file and then save it. The resulting file will contain an image of the active graphics window.

Apply

saves the current settings. These settings will be used when making subsequent images.

Preview

applies the saved settings to the currently active graphics window so the effects of different options may be investigated interactively.

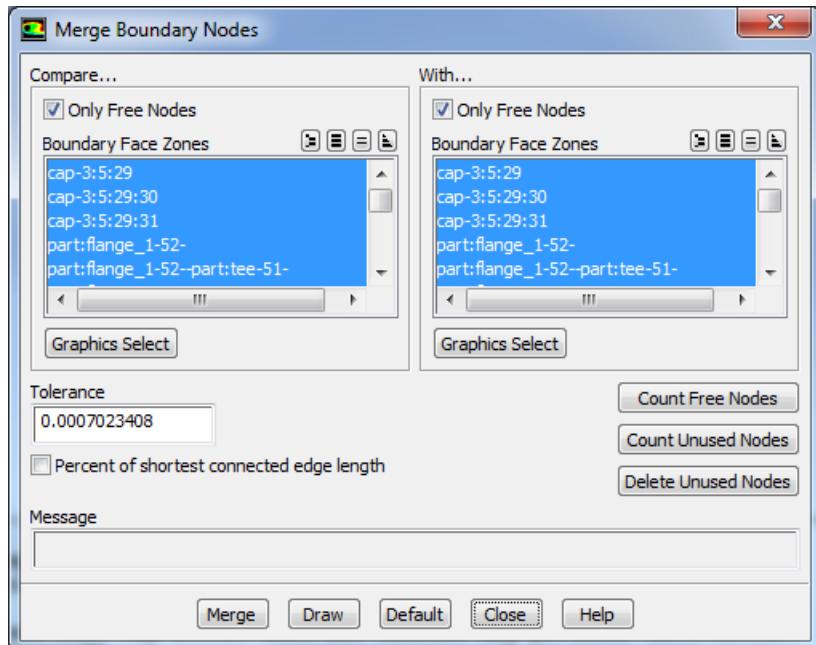
21.3. Boundary Menu

21.3.1. Boundary/Merge Nodes...

The **Boundary/Merge Nodes...** menu item opens the [Merge Boundary Nodes Dialog Box \(p. 482\)](#).

21.3.1.1. Merge Boundary Nodes Dialog Box

The **Merge Boundary Nodes** dialog box reports free or isolated boundary nodes and contains options for removing duplicate and/or isolated nodes.



Controls

Compare..., With...

contain control parameters used for merging duplicate nodes (using the **Merge** button). When searching for duplicate nodes, node A defined in the **Compare...** group box will be compared with node B, defined in the **With...** group box.

Only Free Nodes

allows you to limit the search to free nodes.

- If you enable **Only Free Nodes** only in the **Compare...** group box, node A will be compared with node B only if node A is a free node.
- If you enable **Only Free Nodes** only in the **With...** group box, node A will be compared with node B only if node B is a free node.
- If you enable **Only Free Nodes** in both the **Compare...** and **With...** group boxes (the default setting), the two nodes will be compared only if both are free nodes. If you disable this option in both group boxes, all nodes will be compared.

Since free nodes are a small subset of all nodes, it is much faster to compare free nodes only with other free nodes.

To merge nodes that are not on a free edge, disable **Only Free Nodes** in the **Compare...** or **With...** group box or in both group boxes. For example, if you read a hybrid mesh containing hexahedral cells (and quadrilateral boundary faces) and triangular boundary faces, there may be some duplicate nodes on adjacent quadrilateral and triangular boundary zones. To merge them, compare free nodes on both zones with *all* nodes on both zones (i.e., select both zones in the **Compare...** and **With...** group boxes, and disable **Only Free Nodes** in one of them).

Boundary Face Zones

is a selection list comprising all boundary zones in the domain. You can limit the search for free nodes by selecting a subset of these zones. By default, all boundary zones are selected in both the **Compare...** and **With...** group boxes, so node A on any boundary will be compared with node B on any boundary.

For example, if you want to compare the nodes on only the inlet and top, select **inlet** in the **Compare...** group box and **top** in the **With...** group box. The nodes on these two boundaries will be compared according to the specification of **Only Free Nodes** for the selected boundary zones. Selecting a subset of all boundary zones for node comparison will save time. The search for duplicate nodes and merging them will be quicker if there is no need to check the nodes on all boundaries.

Graphics Select

allows you to select zones graphically.

Select a zone, or zones, in the graphics window, and then click **Graphics Select**.

Tolerance

specifies the tolerance for finding duplicate nodes. If the position of two nodes differs by less than this **Tolerance** value, the nodes are considered duplicate nodes and will be merged when you click the **Merge** button. The default tolerance value is computed by dividing the shortest boundary edge by 1000.

Percent of shortest connected edge length

is used as an alternative to the **Tolerance** for determining whether two nodes will be merged. If this option is enabled, the distance between two nodes is compared against the shortest attached edge

length. If the separation distance is less than the specified percentage (specified in the **Tolerance** field) times the shortest attached edge length, then the nodes will be merged. The allowable maximum value of this parameter is 90%.

Count Free Nodes

reports (in the **Message** field) the number of free nodes.

Count Unused Nodes

reports (in the **Message** field) the number of nodes that are not used by any boundary faces (i.e., the number of isolated nodes).

Delete Unused Nodes

deletes all unused nodes.

Message

shows the information reported when the **Count Free Nodes** or **Count Unused Nodes** button is clicked.

Merge

finds and merges duplicate nodes according to the parameters specified in the **Compare...** and **With...** group boxes. If two nodes of a face are merged, the face is deleted.

Note

The **Merge** button will be activated only when boundary zones are selected in the **Boundary Face Zones** selection lists in both the **Compare...** and **With...** group boxes.

Draw

draws all zones that are selected in the **Compare...** and/or **With...** group boxes.

Default

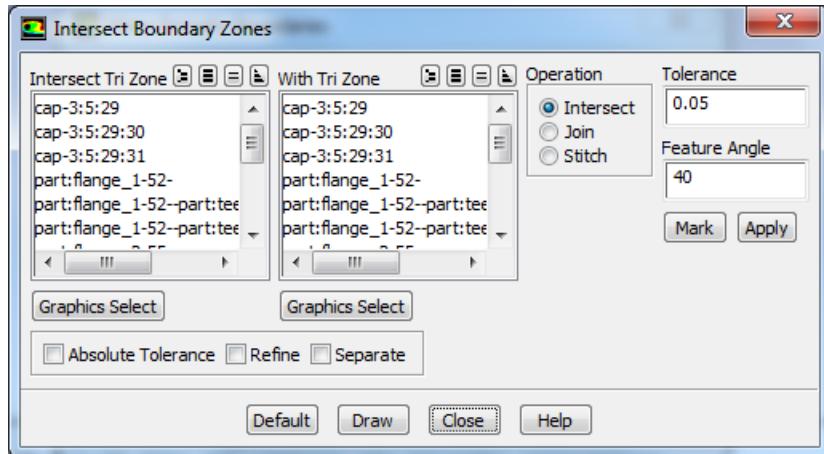
resets all controls in the dialog box to their default settings.

21.3.2. Boundary/Intersect...

The **Boundary/Intersect...** menu item opens the [Intersect Boundary Zones Dialog Box \(p. 484\)](#).

21.3.2.1. Intersect Boundary Zones Dialog Box

The **Intersect Boundary Zones** dialog box allows you to connect two triangular boundary zones. See [Intersecting Boundary Zones \(p. 121\)](#) for details.



Intersect Tri Zone

contains a list of triangular boundary zones from which you can select the zone(s) to be intersected.

Graphics Select

allows you to select zones graphically.

Select a zone, or zones, in the graphics window, and then click **Graphics Select**.

With Tri Zone

contains a list of triangular boundary zones from which you can select the zone(s) with which the zone(s) selected in the **Intersect Tri Zone** list are to be intersected.

Graphics Select

allows you to select zones graphically.

Select a zone, or zones, in the graphics window, and then click **Graphics Select**.

Operation

contains a list where you can select the appropriate connect operation.

Intersect

performs the intersect operation.

Join

performs the join operation.

Stitch

performs the stitch operation.

Tolerance

specifies the tolerance value for the surfaces to be intersected. Faces within the specified tolerance value are considered for the operation selected. The default value of **Tolerance** is 0.05.

By default, the relative tolerance is assumed, but you can change it to absolute tolerance by enabling **Absolute Tolerance** in the **Intersect Boundary Zones** dialog box. Alternatively, use the command /boundary/remesh/controls/intersect/absolute-tolerance? to use the absolute tolerance.

Feature Angle

specifies the minimum angle between the feature edges that should be preserved during retriangulation. All the edges in the zone having feature angle greater than the specified **Feature Angle** are retained. This option is useful for preserving the shape of the intersecting boundary zones. The default value of **Feature Angle** is 40, however, a value in the range of 10–50 degrees is recommended. A large value may distort the shape of the intersecting boundary zones.

Mark

highlights the triangles in the neighborhood of the line of intersection. This helps you to ensure whether or not all faces along the line of intersection are considered for intersection. This also helps you to be sure about the specified tolerance value.

Apply

executes the operation selected in the **Operations** list.

Absolute Tolerance

toggles the use of absolute tolerance.

Refine

toggles the refinement of the intersecting face zones after the intersection is performed. This option is useful for intersecting surfaces having a discrepancy in mesh sizes. Enabling **Refine** yields a smoother mesh having better gradation.

Important

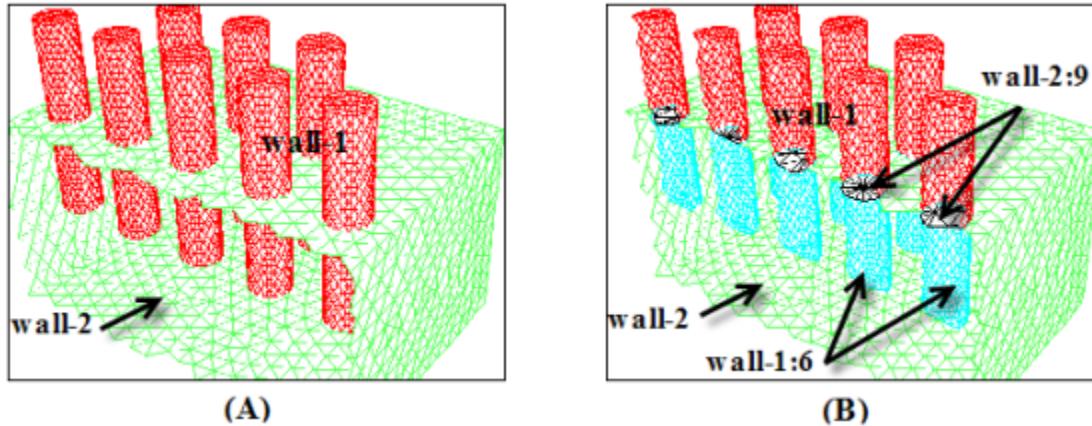
For complicated geometries with highly skewed elements the refinement can make overall mesh quality bad.

Separate

allows you to separate the intersecting zones at the edge loop of the intersection while performing any one of the three intersection operations. The intersecting zones are separated based on the edge loop criteria.

For example, the mesh shown in [Figure 21.1: Boundary Mesh \(A\) Before and \(B\) After Intersection Using the Separate Option \(p. 486\)](#) has two boundary zones (**wall-1** and **wall-2**). Enabling the **Separate** option while intersecting these boundary zones results in the separation of **wall-1** into two zones (**wall-1** and **wall-1:6**), one on either side of **wall-2** and the separation of **wall-2** into two zones, a circular zone inside the pipe section of **wall-1** (**wall-2:9**) and a zone with the remaining part of **wall-2**.

Figure 21.1: Boundary Mesh (A) Before and (B) After Intersection Using the Separate Option

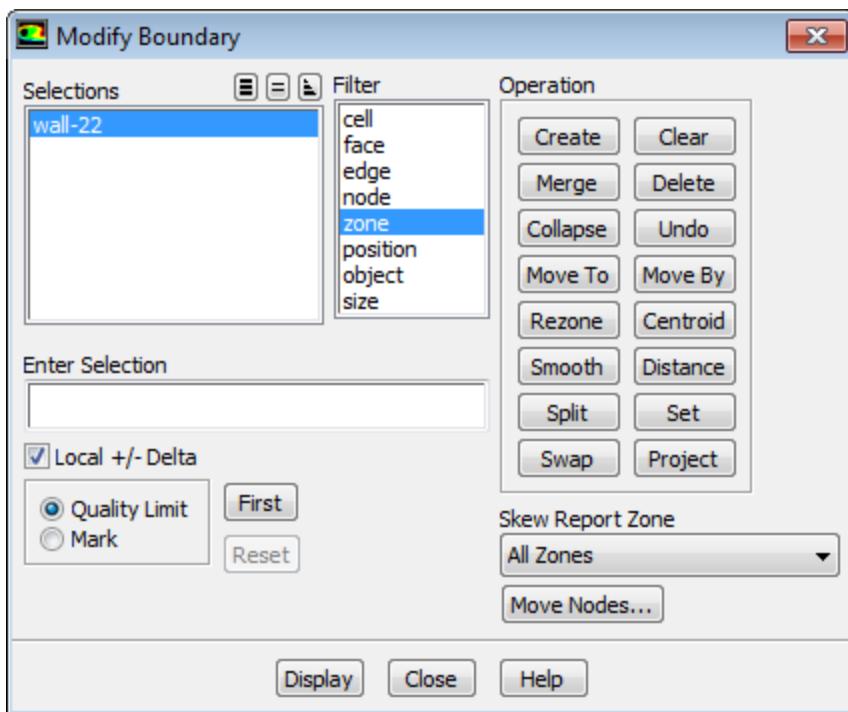


21.3.3. Boundary/Modify...

The **Boundary/Modify...** menu item opens the [Modify Boundary Dialog Box \(p. 486\)](#).

21.3.3.1. Modify Boundary Dialog Box

The **Modify Boundary** dialog box is used in conjunction with the mouse probe. The **Filter** list allows you to specify the type of entity you want to select using the mouse (node, zone, face, etc.). Each selected entity is added to the **Selections** list. You can perform the required operation on the selected items (e.g., merge nodes, swap faces, or move a node) using one of the **Operation** buttons after which the selection list is cleared. The mesh is automatically re-displayed after the operation is performed, allowing you to immediately see the effect of your change.



Controls

Selections

lists the entities (nodes, faces, zones, etc.) that have been selected. To deselect an entity, either click on it once more in the graphics window, or select it in the **Selections** list and then click **Clear** under **Operation**.

Filter

lists the types of entities (nodes, faces, zones, etc.) that can be selected. Only entities of the type highlighted in this list can be selected. You can select one or more entities of one type, then change the **Filter** and continue selecting entities of a different type.

Changing the **Filter** automatically changes the mouse probe function to **Select**. It is also possible to select a group of entities by defining a rectangular or polygonal selection region. See [Mouse Probe Dialog Box \(p. 629\)](#) for details.

Operation

contains the buttons that perform the boundary modification operations.

Create

creates nodes if the **Selections** list contains positions; creates boundary faces if the list contains 3 or 4 nodes and (optionally) a zone; or creates a new zone if **Filter** is first set to **zone**.

- If you create a face without selecting a zone, the new face will be in the same zone as an existing face that uses one of the specified nodes.
- If you select a zone as well as the nodes, the new face will be in the selected zone.
- If the nodes used to create a face are used by faces in different zones, select a zone, to ensure that the new face is in the desired zone.

Merge

merges pairs of nodes or face edges.

- If a pair of nodes is selected, the first node selected is retained, and the second is the duplicate that is merged.
- If a triangular face is selected, its shortest edge is collapsed, merging the other two edges together. The longer of the remaining two edges is retained, and the shorter one is merged with it.

Collapse

collapses pairs of nodes, edge(s), or face(s). If a pair of nodes is selected, both the nodes are deleted and a new node is created at the midpoint of the two nodes. If a triangular face is selected, the face is collapsed into a single node at the centroid of the face.

Important

The **Merge** and **Collapse** operations are irreversible. Save the mesh before performing a merge or collapse, in case you want to return to the previous boundary mesh.

Move To

moves the selected node to the specified position.

Move By

moves the selected node by the specified magnitude of the deviation.

Rezone

moves the selected faces from their current zone into the selected zone. No physical change is made to the mesh; just the zone of the selected face is changed.

Smooth

moves the selected node to a position computed from the average of its node neighbors. The new position is an average of the neighboring node coordinates and is not reprojected to the discrete surface.

Split

refines a triangular face by bisecting a selected edge or by adding a node at the center of a selected face.

Swap

performs swapping on the selected edge(s) of a triangular face. Edge swapping is not available for quadrilateral faces.

Clear

removes the selected entities from the **Selections** list.

Delete

deletes all selected faces and nodes.

Undo

undoes the previous operation. When an operation is performed, the reverse operation is stored on the “undo stack”. For example, a create operation places a delete on the stack, and a delete adds a create operation. The exceptions are merge and collapse, which cannot be undone.

Theoretically if no merge or collapse operations are performed, you can undo all previous operations. In reality, certain sequences of operations are not reversible. The undo operation requires

that the name of the entity exist when the action is undone. If the name does not exist, then the undo will fail.

Important

Usually you can undo the last few operations, but if many operations are being performed it is recommended that you save the mesh periodically, particularly before the merge and collapse operations.

Note

The **Undo** operation is limited to the operations in the **Modify Boundary** dialog box (or the /boundary/modify menu). If other operations/commands are interleaved, the **Undo** operation may cause unexpected results.

Centroid

reports the coordinates of a node or the centroid of a face or cell.

Distance

calculates the distance between any two selected entities (face, node, cell, etc.).

Set

defines a reference line or plane for the **Project** operation.

Project

projects selected nodes onto the projection line or plane defined by the **Set** operation.

Enter Selection

allows you to type in the name of an entity or zone, or the coordinates of a position. This provides an alternative to selecting entities, zones, and locations in the graphics window.

For example, if you have created a new zone and want to use **Rezone** to move some faces into it, there is no way to display the new zone. In such cases, enter the zone name in this field. To do so, enter the name of the desired entity or zone, or the coordinates of the desired location, and press the **Return** (or **Enter**) key on your keyboard. The entry is verified and added to the **Selections** list if it is valid.

To select cells, faces, edges, or nodes, enter their simplified names (e.g., bf213 for the boundary face numbered 213). To select zones, enter their names (e.g., wall-7). To enter positions, type their coordinates (e.g., 1.5 2.4 5.6).

Local +/- Delta

toggles the automatic limiting of the grid display to a neighborhood around the skewed face. If the **Local +/- Delta** option is enabled, the skewed face will be displayed along with some of the faces around it, allowing you to locate the exact position of the skewed cell in the geometry. You can also use the hot-keys up-arrow and down-arrow to increase or decrease the bound limits in the display, respectively.

Quality Limit

allows you to find the face with the worst quality in the entire grid.

Mark

allows you to find the marked faces.

First

finds the triangular face with the worst quality in the entire grid (or the active group) when **Quality Limit** is selected. The worst face will be displayed in the graphics window, and the quality and zone ID will be reported in the console. The selected node will appear in the **Selections** list, along with the longest edge of the face and its opposing vertex node.

When **Mark** is selected, the first of the marked faces will be displayed in the graphics window. The selected node will appear in the **Selections** list, along with the longest edge of the face and its opposing vertex node.

Modify the display region in the **Bounds** section of the **Display Grid** dialog box. If the zone in which the face lies is not currently included in the display, include it by selecting the zone in the **Face Zones** selection list in the **Faces** tab of the **Display Grid** dialog box.

If you have not yet displayed the grid, the worst face, its skewness, and the zone in which it lies will be reported (in the console).

Next

finds the triangular face having the next highest quality value (after that of the worst face) in the grid (or the active group) when **Quality Limit** is selected. Every time you click this button, the triangular face having the next highest quality value will be displayed in the graphics window.

This option will help you to view the locations of the skewed cells in the descending order of their quality.

When **Mark** is selected, the next marked face will be displayed in the graphics window.

Reset

allows you to reset the quality values displayed in the graphics window when **Quality Limit** is selected. When **Mark** is selected, the display will reset to the first marked face.

When you click this button, only the **First** button will be accessible. Click **First** and then **Next** to start viewing faces in descending order of their quality.

Skew Report Zone

contains a drop-down list to choose the particular zone in the geometry for which you want to find the skewed cells. This option allows you to find the skewed cells only in the zone of your interest, instead of finding it for the whole mesh.

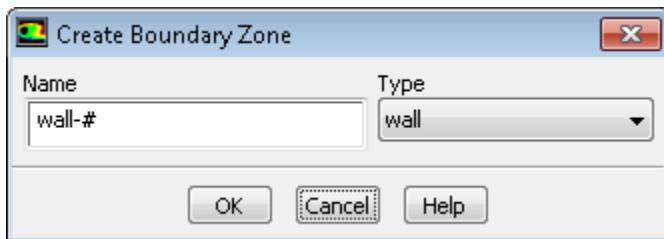
Display

displays the mesh entities in the **Selections** list.

For details about using the **Modify Boundary** dialog box, see [Using the Modify Boundary Dialog Box \(p. 128\)](#).

21.3.3.2. Create Boundary Zone Dialog Box

The **Create Boundary Zone** dialog box will appear automatically when you create a new face zone ([Using the Modify Boundary Dialog Box \(p. 128\)](#)). You can specify the name and type of the new zone in this dialog box.



Controls

Name

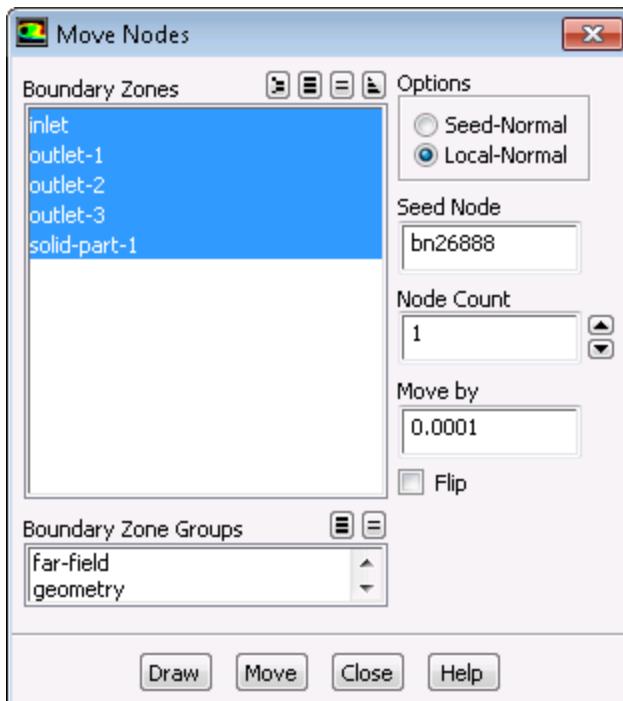
specifies the name of the newly created face zone. The default name that appears is the selected **Type** followed by the assigned zone ID number. For example, if the selected **Type** is **wall**, the default **Name** will be **wall-#**, indicating that the new zone ID number should be appended to "wall" to name the zone (e.g., **wall-12**). You can enter a name that does not include the ID if you prefer (e.g., **inside-wall**).

Type

contains a drop-down list of all available boundary zone types. When you select an item from this list, the **Name** will be updated to reflect the new **Type**.

21.3.3.3. Move Nodes Dialog Box

The **Move Nodes** dialog box allows you to move the selected node(s) by a specified distance either in the **Seed-Normal** or **Local-Normal** directions.



Controls

Boundary Zones

contains a list from which you can select individual boundary zone(s).

Boundary Zone Groups

contains a list of the default boundary zone types and user-defined groups. If you select a zone type/group from this list, all zones of that type/group will be selected in the **Boundary Zones** list. You can also select multiple types to select all the zones of different types (e.g., **inlet** and **outlet**).

Options

contains options for defining the direction of node movement.

Seed-Normal

specifies node movement in the direction of the selected seed node normal.

Local-Normal

specifies node movement in the direction of the individual node normals.

Seed Node

specifies the seed node selected.

Node Count

determines the nodes to be moved.

Move by

specifies the distance by which the node is to be moved.

Flip

flips the normal direction.

Draw

displays the direction and distance of the node movement.

Move

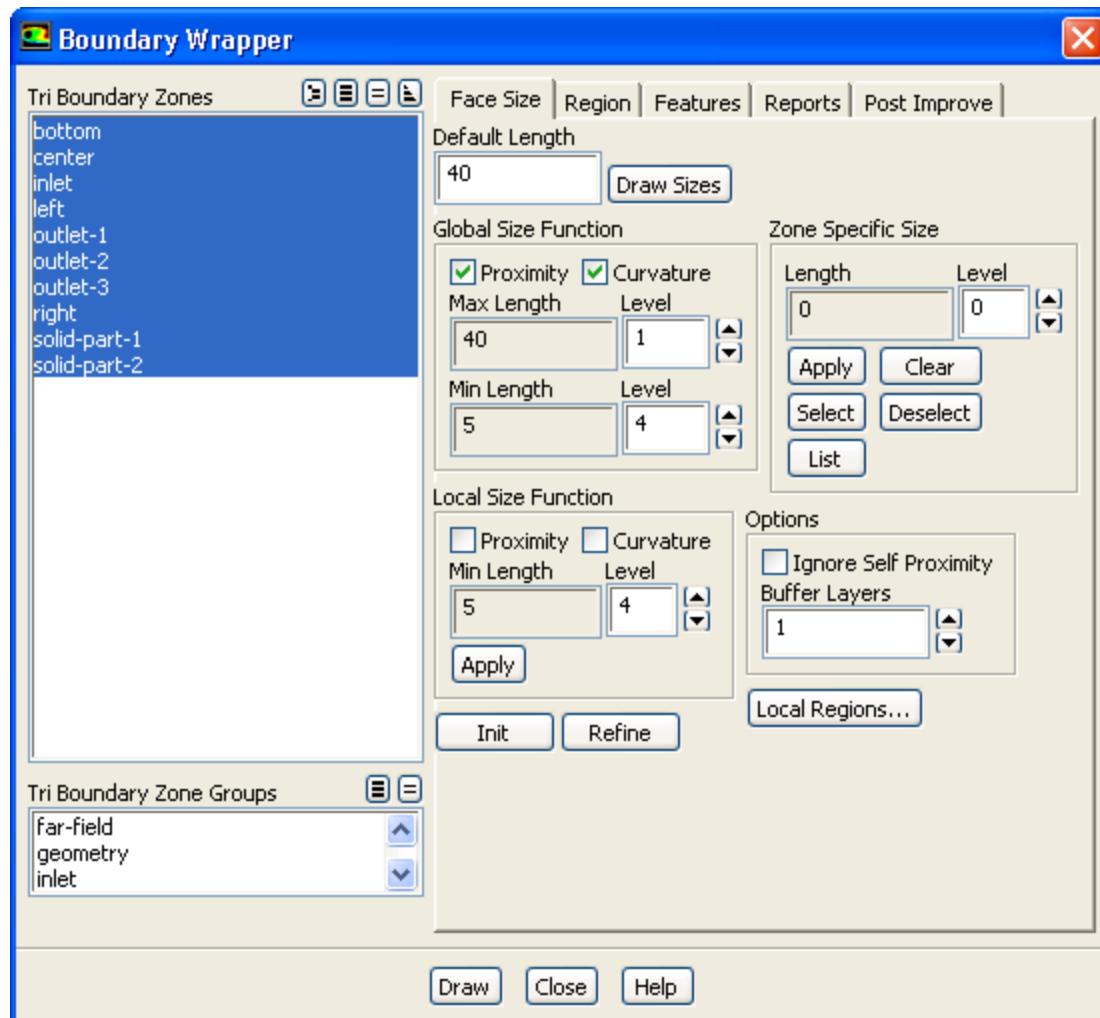
moves the node(s) according to the parameters specified.

21.3.4. Boundary/Wrap...

The **Boundary/Wrap...** menu item opens the [Boundary Wrapper Dialog Box \(p. 492\)](#).

21.3.4.1. Boundary Wrapper Dialog Box

The **Boundary Wrapper** dialog box allows you to preprocess geometries that contains gaps and overlaps. See [The Wrapping Process \(p. 178\)](#) for details about using the **Boundary Wrapper** dialog box.



Controls

Tri Boundary Zones

contains a list of existing tri face zones. The zones may be face zones in the input geometry or the wrapper surface (generated after wrapping).

Face Size

contains parameters for creating and controlling the initial Cartesian grid.

Default Length

specifies the size of the largest cells in the Cartesian grid.

Draw Sizes

displays red boxes of the specified default size over the selected boundary zone.

Global Size Function

allows you to specify the Cartesian grid initialization size function parameters to be applied globally.

Proximity

allows you to automatically refine the Cartesian cells based on the proximity of the zone to other faces in the current domain.

Curvature

allows you to automatically refine the Cartesian cells based on the curvature of the zone in the current domain.

Max Length, Min Length

specify the maximum and minimum size of the cells in the Cartesian grid, respectively. These are only informative boxes and you cannot make any change to the values displayed here.

Level

allows you to select the level of refinement. The cell size for each level is determined relative to the **Default Length**.

Zone Specific Size

contains the parameters for modifying the Cartesian grid cell configuration for each zone.

Length

specifies the size of the cells in the Cartesian grid that you want to create for the zone(s) selected in the **Tri Boundary Zones** list. The value in the **Length** field can only be modified by changing the **Level**.

Level

allows you to specify the cell length for the zones selected in the **Tri Boundary Zones** list.

Apply

applies the specified level of refinement to the selected boundary zone.

Clear

clears the specified level of refinement to the selected boundary zone.

Select

selects the geometry zones on which specified zone sizes are defined.

Deselect

deselects the geometry zones on which specified zone sizes are defined.

List

reports the name, level and cell size of the zone(s) selected in the **Tri Boundary Zones** list.

Local Size Function

allows you to define local size functions and to refine local cells based on the size functions.

Proximity

allows you to automatically refine the local cells based on the proximity of the zone to other faces in the domain.

Curvature

allows you to automatically refine the local cells based on the curvature of the zone in the domain.

Min Length

specifies the minimum allowable size of the cells in the Cartesian grid.

Level

allows you to specify the minimum cell length for local refinement.

Apply

computes required sizes based on the input geometry zones as well as parameters and updates size functions.

Options

contains additional options for wrapping.

Ignore Self Proximity

allows you to ignore self-proximity during wrapping. This option is disabled by default.

Buffer Layers

specifies the number of additional layers to be added in order to have a smooth transition from fine to coarse cells.

Local Regions...

opens the **Wrapper Refinement Region** dialog box.

Init

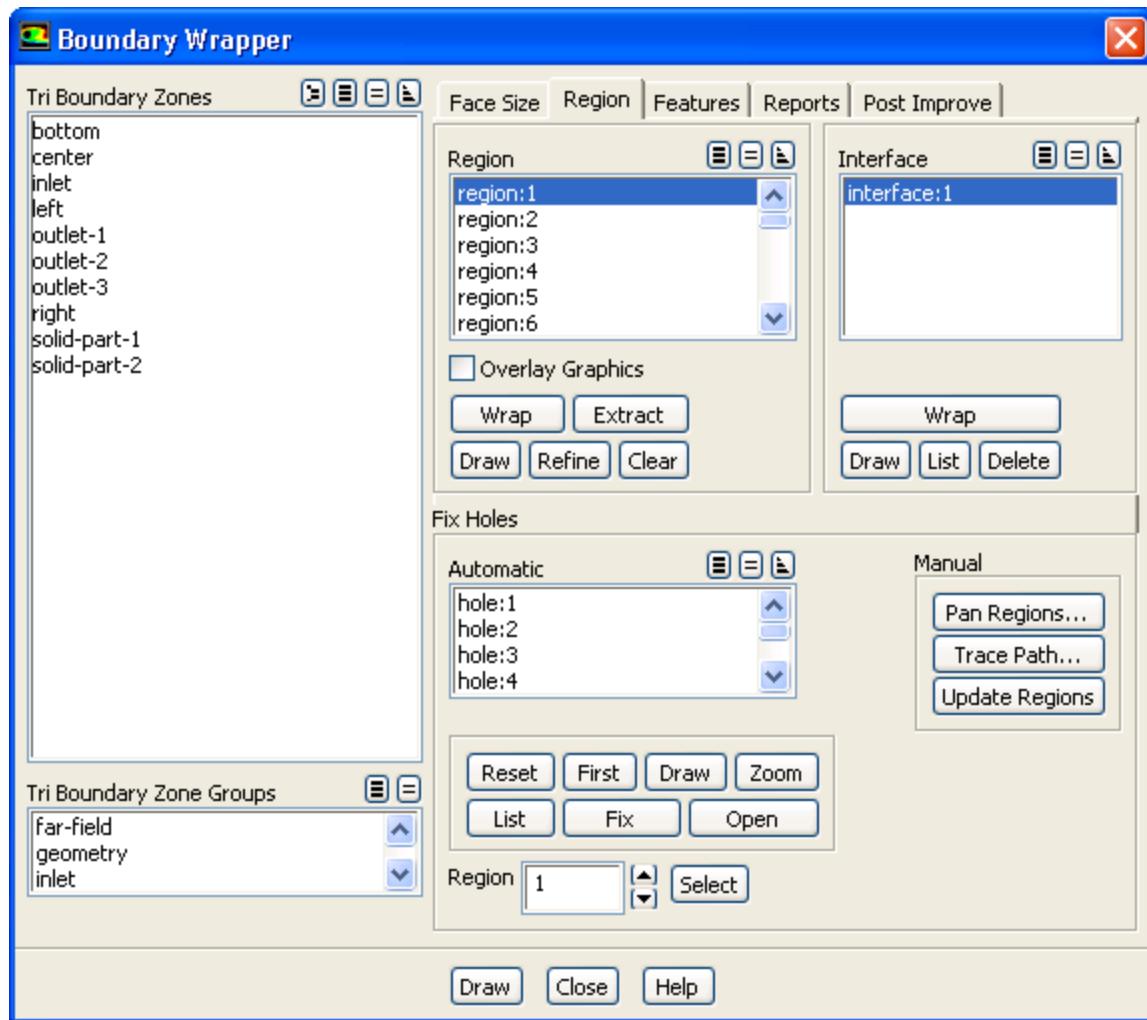
allows you to initialize the Cartesian grid.

Refine

refines the Cartesian grid according to the specified **Zone Specific Size** and/or **Local Size Function** parameters.

Region

contains the parameters to examine the region of interest among the several regions created during the wrapper initialization process.



Region

contains a list of the regions generated during the wrapping process.

Overlay Graphics

displays the Cartesian grid overlaid with the current display.

Wrap

wraps the selected region(s).

Extract

extracts the interface(s) from the selected region(s).

Draw

displays the selected region(s).

Refine

refines the selected region(s).

Clear

clears all the regions.

Deleting the regions reduces the peak memory and improves the speed of further operations. You can not recover the regions after they have been deleted.

Interface

contains a list of the interface surfaces extracted from the regions.

Note

The **Interface** list is available only after you have extracted an interface from any region.

Wrap

creates the wrapper surface from the selected interface(s).

Draw

displays the selected interface(s).

List

lists the face count for all extracted interface(s) in the console.

Delete

deletes the selected interface(s).

Fix Holes

contains options for fixing holes in the Cartesian grid.

Automatic

lists the holes detected for all regions.

Reset

resets the list of selected holes.

First, Next

allow you to traverse the list of selected holes. When you click **First**, the first selected hole will be displayed in the graphics window and the hole ID will be reported in the console. You can then click **Next** repeatedly to traverse the list of selected holes.

Draw

draws the hole(s) selected in the **Automatic** selection list.

Zoom

zooms into the hole selected in the **Automatic** selection list.

List

lists the ID, location, and the region for the hole(s) selected in the **Automatic** selection list.

Fix

closes the holes selected in the **Automatic** selection list.

Open

allows you to ignore holes in regions other than the region of interest.

Region

specifies the region to be considered for automatic hole fixing.

Select

selects the holes detected for the region specified in the **Region** field.

Manual

contains options for manual hole fixing.

Pan Regions...

opens the **Pan Regions** dialog box, using which you can examine the Cartesian grid.

Trace Path...

opens the **Trace Path** dialog box, using which you can trace the leaks or holes in the existing geometry.

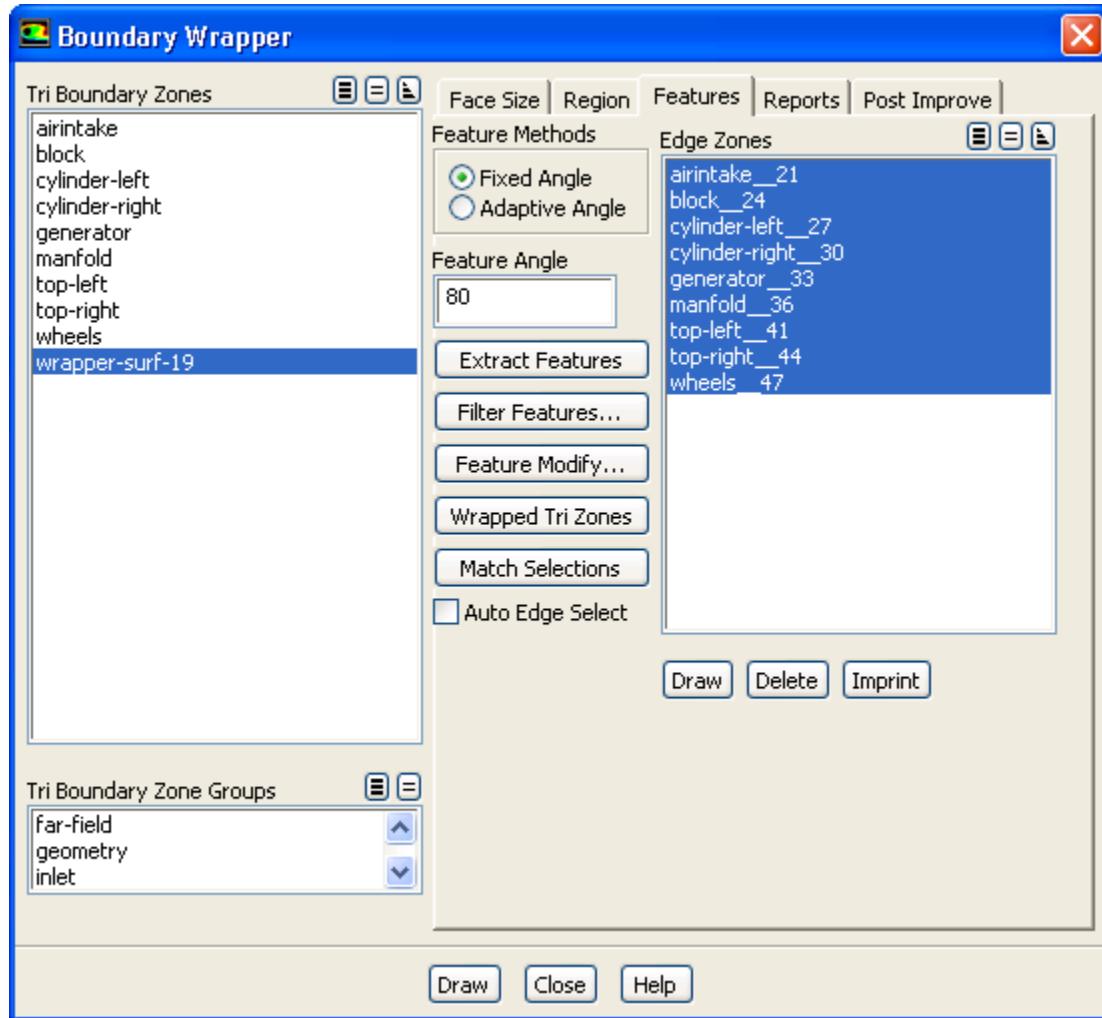
Update Regions

updates the selected region for the modifications made to the zones selected in the **Tri Boundary Zones** list.

Refer to the [Manual Leak Detection \(p. 188\)](#) section for details on the manual hole fixing procedure.

Features

contains the parameters to recover the distinctive feature lines in the geometry.

**Feature Methods**

specifies the methods available to create edge loops.

Fixed Angle

specifies the feature edge should be preserved only if the angle between adjacent triangular faces is greater than the specified value.

Feature Angle

specifies the minimum feature angle that should be prevented while remeshing operation.

Adaptive Angle

specifies the feature edge should be preserved using adaptive angle criteria.

Extract Features

extracts the edge loops for the zones selected in the **Tri Boundary Zones** selection list and lists them in the **Edge Zones** list.

Filter Features...

opens the **Filter Features** dialog box, using which you can retain the feature edges located within a specified distance from the wrapper surface.

Feature Modify...

opens the **Feature Modify** dialog box, using which you can modify/remesh edge loops.

Wrapped Tri Zones

selects the wrapped zones in the **Tri Boundary Zones** selection list.

Match Selections

selects the edge loop(s) associated with the selected zone(s) in **Tri Boundary Zones** list. This feature can be used only if the **Auto Edge Select** option is disabled.

Auto Edge Select

automatically selects edge loop(s) associated with the zones selected in the **Tri Boundary Zones** selection list.

Edge Zones

contains a list of edge loops that have been created for one or more face zones.

Draw

displays the edge loops selected in the **Edge Zones** list.

Delete

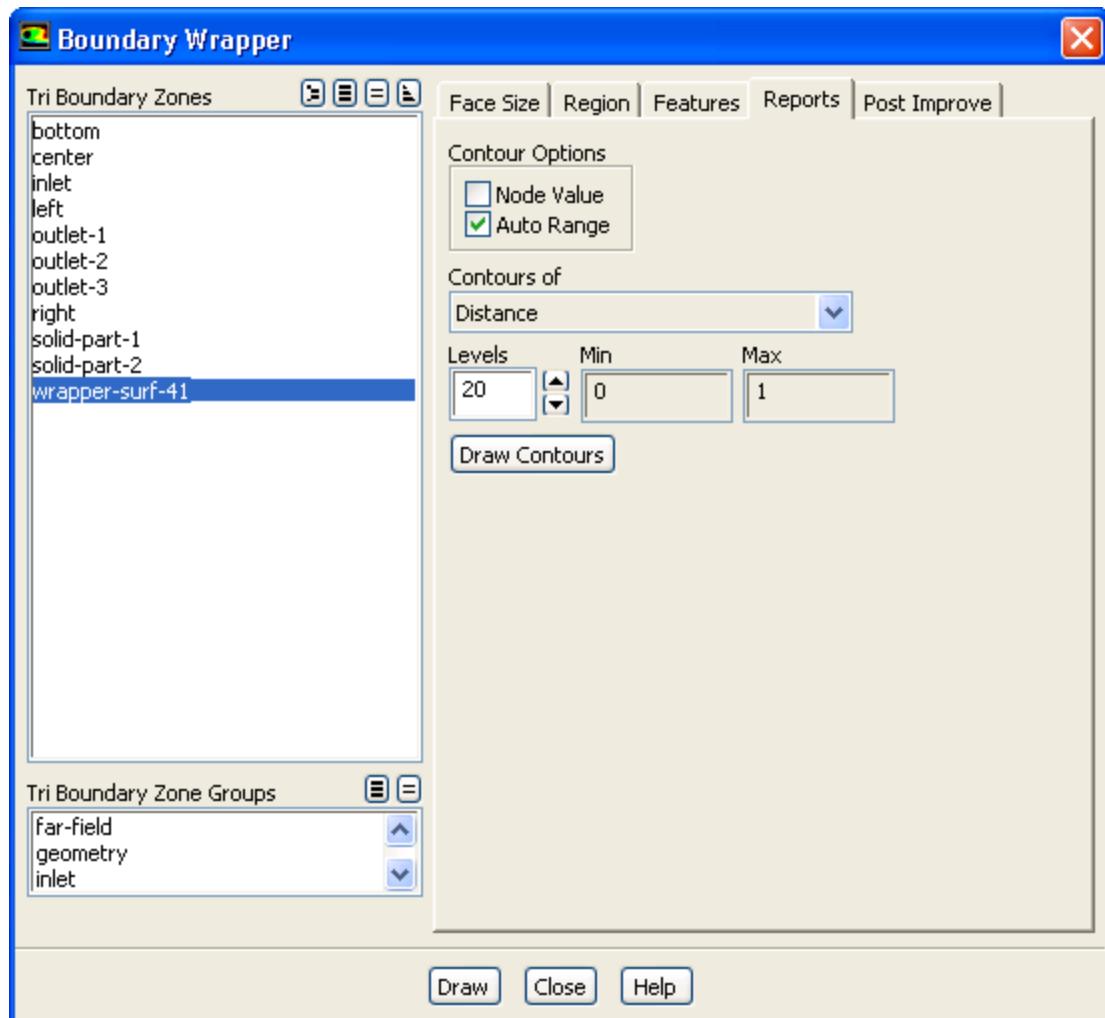
deletes the edge loops selected in the **Edge Zones** list.

Imprint

projects the nodes of the wrapper surface onto the selected **Edge Zones**.

Reports

contains the parameters to check the quality of the wrapper surface.



Contour Options

contains the various contour display options.

Node Values

toggles between using scalar field values at nodes and at face centers for computing the contours.

Auto Range

toggles between automatic and manual setting of the contour range.

Contours of

contains a list from which you can select the parameters to be contoured.

Distance

specifies the distance of face center to the original geometry.

Normal

specifies the direction of the face with respect to the original geometry.

Composite

specifies the value calculated by linearly combining above two values.

Levels

sets the number of contour levels that are displayed.

Min

shows the minimum value of the scalar field.

Max

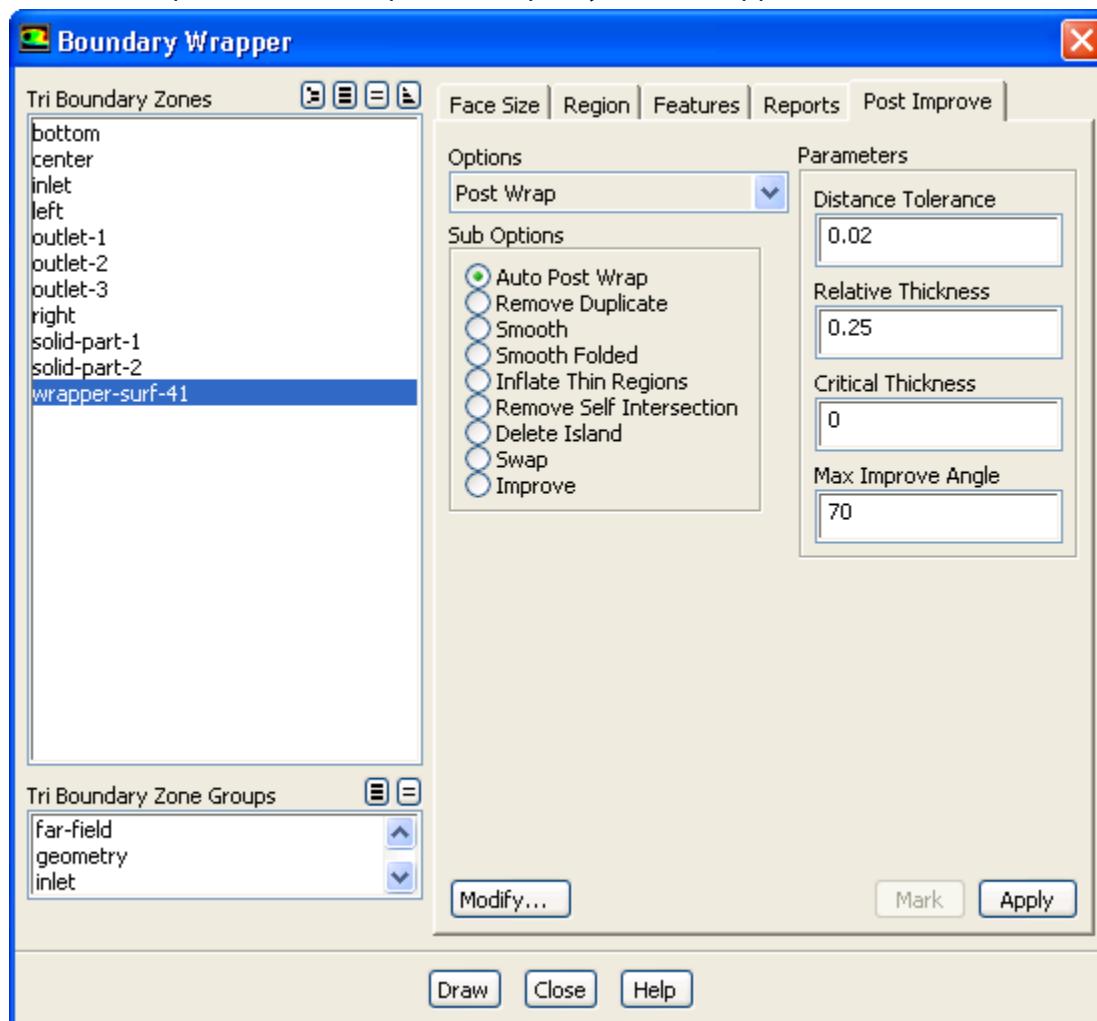
shows the maximum value of the scalar field.

Draw Contours

displays the contours in the active graphics window.

Post Improve

contains the parameters to improve the quality of the wrapper surface.

**Options**

contains various options that are available to improve the quality of the wrapper surface.

Coarsen

allows you to coarsen the wrapper surface. This option can also be used for wrapper boundary zones after zone recovery.

Total Face Count

reports the face count for the zone selected in the **Tri Boundary Zones** list. The value in this field is informational only.

Edge Length Change

specifies the maximum allowable edge length change.

Max Angle Change

specifies the maximum allowable angle change.

Min Length, Max Length

specify the minimum and maximum values (range) for the average length of faces to be considered for coarsening.

Preserve Boundary

allows you to preserve the original locations of the zone boundary nodes.

Post Wrap

contains options for the post wrapping operations available.

Auto Post Wrap

allows you to perform a pre-defined sequence of post wrapping operations on the wrapper surface.

Distance Tolerance

specifies the tolerance value for removing duplicate nodes.

Relative Thickness

specifies the distance for the separation of self-intersecting faces.

Critical Thickness

specifies the critical thickness for inflating thin regions.

Max Improve Angle

specifies the maximum angle considered for improving the wrapper surface.

Remove Duplicate

deletes the duplicate nodes within the proximity of the tolerance value.

Distance Tolerance

specifies the tolerance value to be considered while removing duplicate nodes.

Smooth

allows you to smooth the wrapper surface.

Iteration

specifies the number of smoothing attempts.

Relaxation

specifies the node smoothing relaxation factor.

Reprojection

enables the projection of nodes onto the geometry after smoothing.

Rel Reproject Range

specifies the maximum relative distance of the nodes from geometry within which the nodes are projected onto the geometry.

Preserve Boundary

allows you to preserve the original locations of the zone boundary nodes.

Smooth Folded

allows you to fix the folded faces on the wrapper surface.

Iteration

specifies the number of node smoothing attempts for resolving folded configurations.

Critical Angle

specifies an angle between the pair of adjacent faces. If the feature angle between the two faces is smaller than the specified value, the faces are assumed to be folded.

Relaxation

specifies the relaxation factor for node smoothing.

Reprojection

enables the projection of the adjusted nodes back on the geometry edges after smoothing.

Inflate Thin Regions

improves the wrapper surface by pushing apart the overlapping faces to a certain thickness.

Critical Thickness

specifies the critical distance within which the faces are assumed to be overlapped.

Critical Angle

specifies the critical angle between the faces below which the faces are assumed to be overlapped

Preserve Boundary

allows you to preserve the original locations of the zone boundary nodes.

Remove Self Intersection

moves apart the intersecting faces in the wrapper surface.

Iteration

specifies number of attempts to remove self intersecting faces.

Relative Thickness

specifies the distance up to which the separation of self-intersecting faces takes place.

Delete Island

deletes islands faces having a face count less than or equal to the specified critical face count.

Critical Face Count

specifies the critical face count for deleting the island faces.

Absolute Count

specifies the absolute face count below which the non-contiguous region can be treated as an island configuration.

Relative Count

specifies the face count relative to the largest region below which the non-contiguous region can be treated as an island configuration.

Swap

allows you to perform node swapping on the wrapper surface.

Max Angle

specifies the maximum angle between two adjacent faces allowed for performing swapping.

Max Skew

specifies the maximum allowable skewness value after swapping.

Improve

allows you to improve the wrapper surface based on the **Quality Measure** selected.

Quality Measure

contains the available options for improving the quality of the wrapper surface.

Skewness

allows you to improve the wrapper surface quality based on the skewness.

Size Change

allows you to improve the wrapper surface quality based on size change.

Aspect Ratio

allows you to improve the wrapper surface quality based on the aspect ratio.

Iteration

specifies the number of improving attempts.

Max Angle

specifies the maximum allowable angle between two adjacent face normals.

Quality Limit

specifies the quality limit for the improvement operation. All elements above the specified quality limit will be improved.

Preserve Boundary

allows you to preserve the original locations of the zone boundary nodes.

Zone

contains options for the recovery of zones from the wrapper surface.

Recover Zone

separates the wrapper surface into zones based on the original geometry.

Rezone

smoothes the separated zones by adjusting the node position on the zone boundaries.

Reprojection

enables the projection of nodes onto the geometry after smoothing.

Rename

allows you to rename the boundary zones.

From

specifies the prefix of the existing zone names that you want to rename.

To

specifies the prefix of the new zone names.

Expert

contains advanced improvement options

Auto Post Improve

performs a pre-defined sequence of improvement operations like splitting and merging of nodes to remove small areas, improving the boundary surface quality, collapsing skewed faces, and removing duplicate and intersecting faces.

Min Triangle Area

specifies the minimum face area to be considered while removing small areas.

Max Aspect Ratio

specifies the maximum allowable aspect ratio.

Max Skewness

specifies the maximum allowable skewness.

Max Size Change

specifies the maximum allowable size change.

Recover Single Surface

contains options for recovering thin surfaces and removing any remaining overlapping regions.

Single Surface

allows you to recover a single thin surface.

Automatic

enables the automatic marking and merging of thin regions for recovering the single surface.

First Seed Face, Second Seed Face

specify the seed faces to be used for marking the thin regions.

Critical Thickness

specifies the critical thickness of the surface to be recovered.

Critical Angle

specifies the maximum angle between a pair of opposing faces while checking for overlapping faces. Any pair of faces whose normal vectors form an angle greater than the specified value will be regarded as non-overlapping.

Select

allows you to select the seed faces when the **Automatic** option is disabled.

Mark

marks the faces to be merged.

Merge

merges the thin surfaces into a single surface.

Post Single Surface

allows you to remove unmerged island regions after recovering the thin surface.

Start Seed Face

specifies the seed face to be used for marking the unmerged regions. You need to specify the seed face while marking the unmerged regions manually.

Max Face Count

specifies the maximum allowable face count for the removal of the unmerged regions.

Mark

marks the unmerged regions to be removed.

Delete

deletes the marked faces.

Reset

clears the previously selected seed face.

Auto

automatically marks and deletes the unmerged regions.

Remove Crossover

allows you to remove the crossover configurations the wrapper surface.

Automatic

automatically removes the crossover configurations.

Seed Face

specifies the seed face for determining the crossover configuration.

Max Faces

specifies the maximum number of faces to be collected and deleted after performing this operation.

Iteration

specifies the number of attempts to remove the crossover configuration in case of failure.

Relaxation

specifies the relaxation factor for node smoothing in making different number of attempts.

Modify...

opens the **Modify Boundary** dialog box.

Mark

marks the cells to be considered for the selected post wrapping improvement operation. The **Mark** option is available

Apply

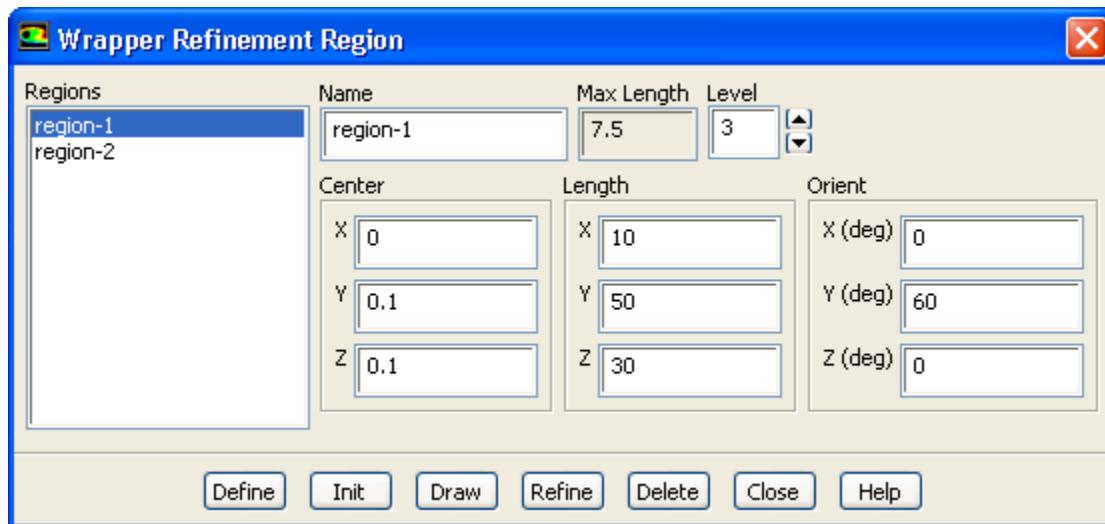
executes the operation selected in the **Options** list.

Draw

displays the zones selected in the **Tri Boundary Zones** list in the graphics window.

21.3.4.2. Wrapper Refinement Region Dialog Box

The **Wrapper Refinement Region** dialog box allows you to define local regions to be refined.



Controls

Regions

contains a list of the defined regions.

Name

specifies the name of the region selected in the **Regions** list, or the region to be created.

Max Length

specifies the maximum cell length in the region. The **Max Length** value is determined by the **Level** specified.

Level

specifies the refinement level for the local region defined.

Center

specifies the coordinates of center of the refinement region.

Length

specifies the extent of the refinement region in the *x*, *y*, *z* directions.

Orient

specifies the orientation of the refinement region.

Note

The region is oriented by rotation first about the *x*-axis, then the *y*-axis, and finally the *z*-axis. You need to take this into account while specifying the orientation of the region as rotation in any other order will produce different results.

Define

creates a new region according to the specified parameters. It also allows you to modify the selected region according to specified parameters.

Init

creates a default region encompassing the entire geometry.

Draw

displays the selected region in the graphics window.

Refine

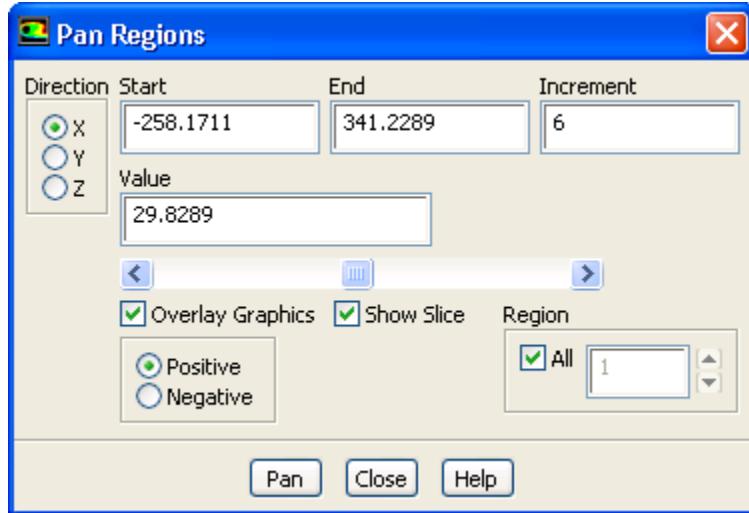
refines the selected region.

Delete

deletes the selected region.

21.3.4.3. Pan Regions Dialog Box

The **Pan Regions** dialog box allows you to observe and analyze the region (you want to wrap) created during the Cartesian grid initialization.

**Controls****Direction**

allows you to select the direction in which you want pan the selected region(s).

Start

specifies the frame number from which you want to start the plane movement.

End

specifies the frame number at which you want to stop the plane movement.

Increment

specifies the interval of the cutting plane coordinate during panning.

Overlay Graphics

allows you to display either the Cartesian grid surface or the geometry along with the pan plane while panning through the selected region(s).

Show Slice

shows the slice through the region.

Positive

displays the surfaces on the positive side of the cutting plane.

Negative

displays the surfaces on the negative side of the cutting plane.

Region

allows you to select the regions you want to pan through.

All

enables panning through all regions. When this option is disabled, you can select a particular region to pan.

Pan

starts the movement of the plane from one end to the other in the specified direction through the Cartesian grid.

21.3.4.4. Trace Path Dialog Box

The **Trace Path** dialog box allows you to locate the position of the hole or leak in the Cartesian grid or input geometry by tracing the path connecting the cells corresponding to specified locations.

**Controls****Locations**

specifies the positions (**X Pos**, **Y Pos**, **Z Pos**) of the **Start** and **End** points between which the path is to be traced. You can either enter the position values or select the points using the mouse.

Select Points...

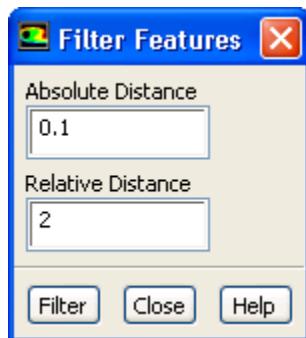
allows you to select the start and end points of the trace using the mouse.

Trace

highlights the shortest possible path between the specified **Start** and **End** points.

21.3.4.5. Filter Features Dialog Box

The **Filter Features** dialog box allows you to retain the feature edges located within a specified distance from the wrapper surface. Feature edges beyond the specified distance will be deleted.



Controls

Absolute Distance

specifies the absolute distance from the wrapper surface, based on which feature edges will be retained.

Relative Distance

specified the relative distance from the wrapper surface, based on which feature edges will be retained.

Filter

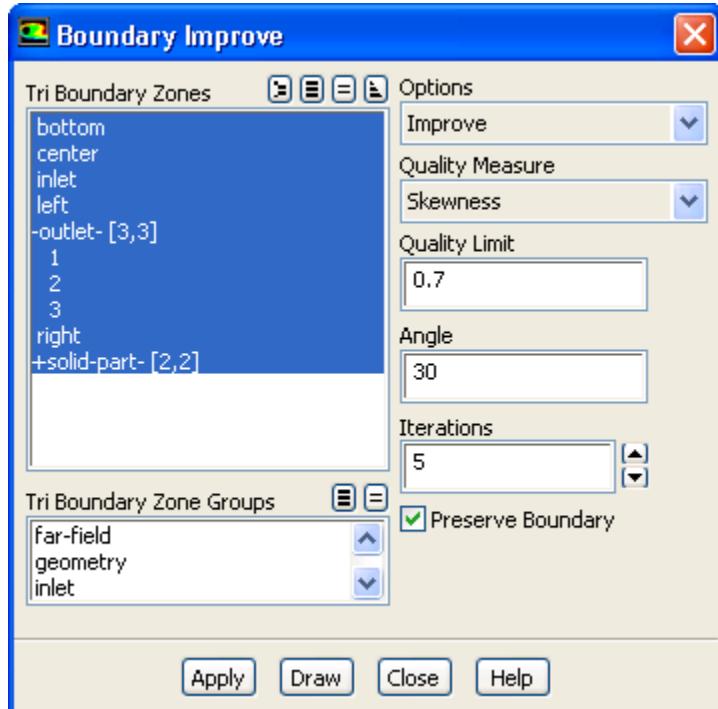
automatically deletes features beyond the calculated distance, thereby retaining only the appropriate feature edges.

21.3.5. Boundary/Mesh/Improve...

The **Boundary/Mesh/Improve...** menu item opens the Boundary Improve Dialog Box (p. 510).

21.3.5.1. Boundary Improve Dialog Box

The **Boundary Improve** dialog box allows you to improve the overall mesh quality.



Controls

Tri Boundary Zones

contains a list from which you can select individual boundary zone(s) to be improved.

Tri Boundary Zone Groups

contains a list of the default boundary zone types and user-defined groups. If you select a zone type/group from this list, all zones of that type/group will be selected in the **Tri Boundary Zones** list. You can also select multiple types to select all the zones of different types (e.g., **inlet** and **outlet**).

Options

contains the options available for improving boundary surfaces.

Improve

allows you to improve the selected zones based on the **Quality Measure** selected.

Smooth

allows you to improve the selected zones by smoothing.

Swap

allows you to improve the selected zones by edge swapping.

Quality Measure

contains the available options for improving the quality of the boundary surfaces. This option is available only when **Improve** is selected in the **Options** drop-down list.

Skewness

allows you to improve the boundary surface quality based on the skewness.

Size Change

allows you to improve the boundary surface quality based on size change.

Aspect Ratio

allows you to improve the boundary surface quality based on the aspect ratio.

Area

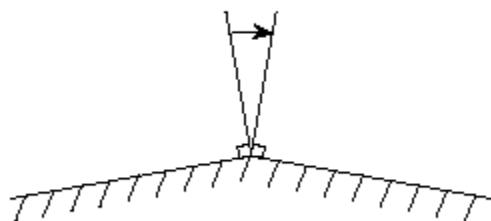
allows you to improve the boundary surface quality based on the area.

Quality Limit

specifies the quality limit for the improvement operation when using the **Improve** option with **Skewness**, **Size Change**, or **Aspect Ratio** selected as the **Quality Measure**. All elements above the specified quality limit will be improved.

Angle

specifies the maximum allowable angle between two adjacent face normals (see [Figure 21.2: Angle Between Adjacent Face Normals \(p. 512\)](#)) when using the **Improve** option with **Skewness**, **Size Change**, or **Aspect Ratio** selected as the **Quality Measure**.

Figure 21.2: Angle Between Adjacent Face Normals**Iteration**

specifies the number of improving attempts when using the **Improve** option with **Skewness, Size Change**, or **Aspect Ratio** selected as the **Quality Measure**.

Preserve Boundary

allows you to preserve the geometry of the surface when using the **Improve** option with **Skewness, Size Change**, or **Aspect Ratio** selected as the **Quality Measure**.

Area Options

contains options for improving the boundary surface based on the area.

Collapse and Swap

allows you to collapse faces having face area smaller than the minimum absolute size specified or relative to the minimum absolute size and then perform edge swapping.

Collapse and Smooth

allows you to collapse faces having face area smaller than the minimum absolute size specified and then perform smoothing.

Min Absolute Size

specifies the minimum absolute size. All faces having area smaller than the specified value will be collapsed.

Min Relative Size

specifies the minimum relative size for the **Collapse and Swap** option only. All faces having area smaller than the value relative to the minimum absolute size will be collapsed.

Max Angle

specifies the maximum allowable angle between two adjacent face normals ([Figure 21.2: Angle Between Adjacent Face Normals \(p. 512\)](#)). The **Max Angle** option is available only when **Smooth** or **Swap** is selected in the **Options** drop-down list.

Relax

specifies the relaxation factor used for smoothing. This option is available only when **Smooth** is selected in the **Options** drop-down list.

Max Skew

specifies the maximum allowable skewness value for the swapping operation. All faces having skewness greater than the specified value will be considered during the swapping operation.

Check

reports the number of unused nodes in the console.

Skew

reports the face with the maximum skewness and the corresponding skewness value in the console.

Limits

reports the minimum and maximum face area for the zone(s) selected in the **Tri Boundary Zones** selection list.

Apply

performs the operation selected in the **Options** drop-down list.

Draw

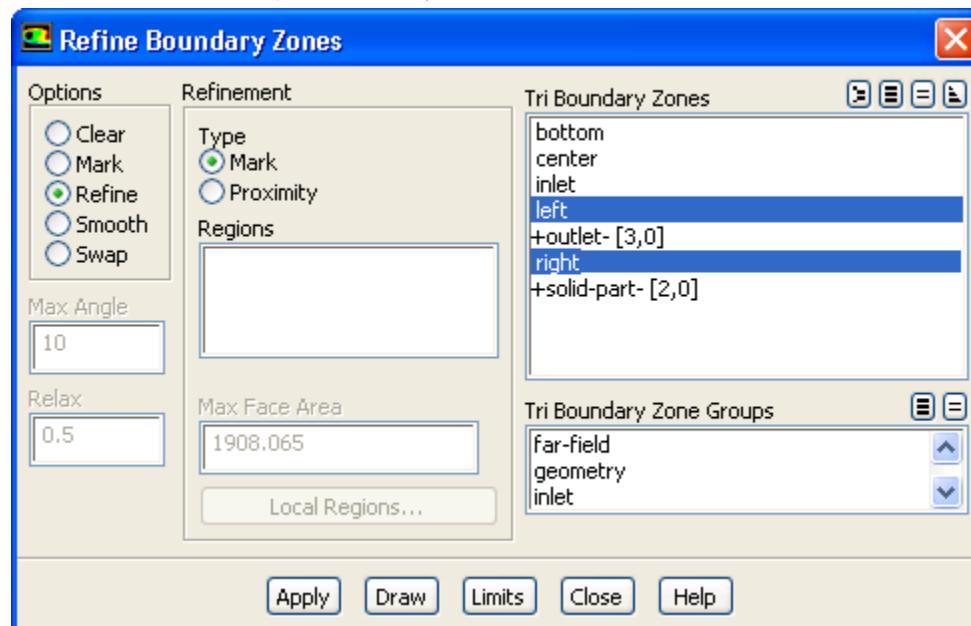
displays the selected zones in the graphics window.

21.3.6. Boundary/Mesh/Refine...

The **Boundary/Mesh/Refine...** menu item opens the [Refine Boundary Zones Dialog Box \(p. 513\)](#).

21.3.6.1. Refine Boundary Zones Dialog Box

The **Refine Boundary Zones** dialog box allows you to refine the triangular boundary zones that touch a local refinement region before you refine the volume mesh in the refinement region.



These commands are described in [Additional Boundary Mesh Text Commands \(p. 172\)](#).

Controls**Options**

contains a number of operations related to the boundary zone refinement. The selected operation will be performed when you click **Apply**.

Clear

clears all refinement marks from all boundary faces.

Mark

marks faces that are larger than the **Max Face Size**. Only faces that border the refinement region will be marked.

Refine

refines the marked faces by dividing them into three faces, as shown in [Figure 9.11: Refining a Triangular Boundary Face \(p. 140\)](#).

Smooth

smooths the nodes of the boundary faces (using Laplacian smoothing, as described in [Laplace Smoothing \(p. 347\)](#)), based on the specified **Max Angle** and **Relax** parameters, to try to lower the maximum skewness.

Swap

swaps the edges of the boundary faces, based on the specified **Max Angle** and **Max Skew** parameters.

Max Angle

specifies the maximum angle between two adjacent face normals. When the **Swap** option is active, only faces with an angle below this value will be swapped. This restriction prevents the loss of sharp edges in the geometry. The valid range of entries is 0 to 180 degrees and the default is 10 degrees. The larger the angle, the greater the chance that a face swap will occur that may have an impact on the flow solution. See [Swapping \(p. 349\)](#) for details about swapping.

When the **Smooth** option is active, the nodes on a face will be smoothed only if one of the angles between the face normals is less than **Max Angle**.

Relax

(used with the **Smooth** option) specifies the relaxation factor by which the computed change in node position should be multiplied before the node is moved. A value of zero results in no node movement, and a value of 1 results in movement equivalent to the entire computed increment.

Max Skew

(used only with the **Swap** option) specifies the maximum allowable face skewness as a result of edge swapping. If swapping causes the skewness of a face to exceed this value, the swap will not be performed. See [Swapping \(p. 349\)](#) for details about swapping.

Refinement

contains controls for defining refinement parameters.

Type

allows you to specify refinement based or marking or proximity when the **Refine** option is selected.

Mark

allows you to refine the marked faces.

Proximity

allows you to refine the face zone based on the proximity with respect to other faces in the current domain. The outer edges of the boundary face zones are also refined to allow better quality meshes after refinement.

Regions

contains a list of the refinement regions that have been defined. Click the **Local Regions...** button to open the **Boundary Refinement Region** dialog box and define the refinement region.

Max Face Area

shows the maximum acceptable face area for the refinement region selected in the **Regions** list; faces on the selected zones that are larger than this will be refined. The **Max Face Area** is defined in the **Boundary Refinement Region** dialog box.

Local Regions...

opens the **Boundary Refinement Region** dialog box, where you can define the refinement region.

Relative Distance

specifies the relative distance for determining the region to be refined based on proximity.

Iterations

specifies the number of face-splitting passes to be performed during the proximity refinement.

Tri Boundary Zones

contains a list from which you can select individual boundary zones to be operated on.

Tri Boundary Zone Groups

contains a list of boundary zone types. If you select a boundary type from this list (e.g., **inlet**), all boundary zones of that type (for this example, all **pressure-inlet** and **velocity-inlet** boundaries) will be selected in the **Boundary Zones** list. This shortcut allows you to easily select all boundary zones of a certain type without having to select each zone individually. You can select multiple boundary types in the **Tri Boundary Zone Groups** list to select all zones of several different types (e.g., **inlet** and **outlet**).

Apply

performs the selected operation.

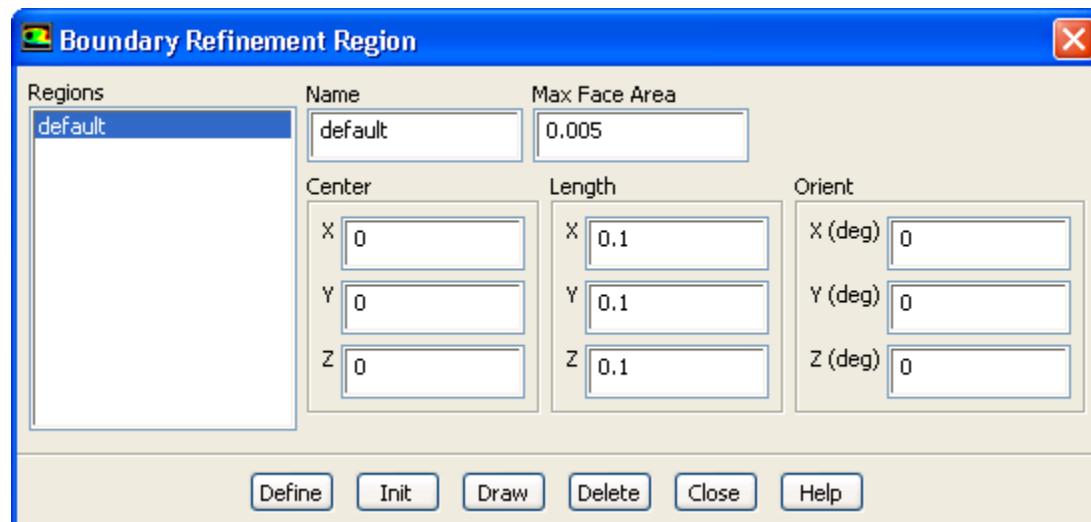
Draw

displays the zones selected in the **Tri Boundary Zones** list.

Limits

prints a report (in the console window) of the minimum and maximum size of each zone that is selected in the **Tri Boundary Zones** list. This report will also tell you how many faces on each selected zone have been marked for refinement.

21.3.6.2. Boundary Refinement Region Dialog Box

**Controls**

Regions

contains a list of the defined regions.

Name

reports the name of the selected region. You can specify a new name by entering it in the text entry box.

Max Face Area

sets the maximum face area for the selected region. You can change the value by entering a new value in this field.

Center

allows you to specify the coordinates of the center of the region you want to create.

Length

allows you to specify the absolute size of the region in the x, y, and z directions.

Orient

allows you to specify the orientation of the region.

Note

The region is oriented by rotation first about the x-axis, then the y-axis, and finally the z-axis. You need to take this into account while specifying the orientation of the region as rotation in any other order will produce different results.

Define

creates a new region according to the specified parameters. It also allows you to modify the selected region according to the changes made.

Init

creates a default region encompassing the entire geometry.

Draw

draws the region in the graphics window. If the grid was displayed (using the **Display Grid** dialog box) before drawing the region, the grid will also be included in the display.

Delete

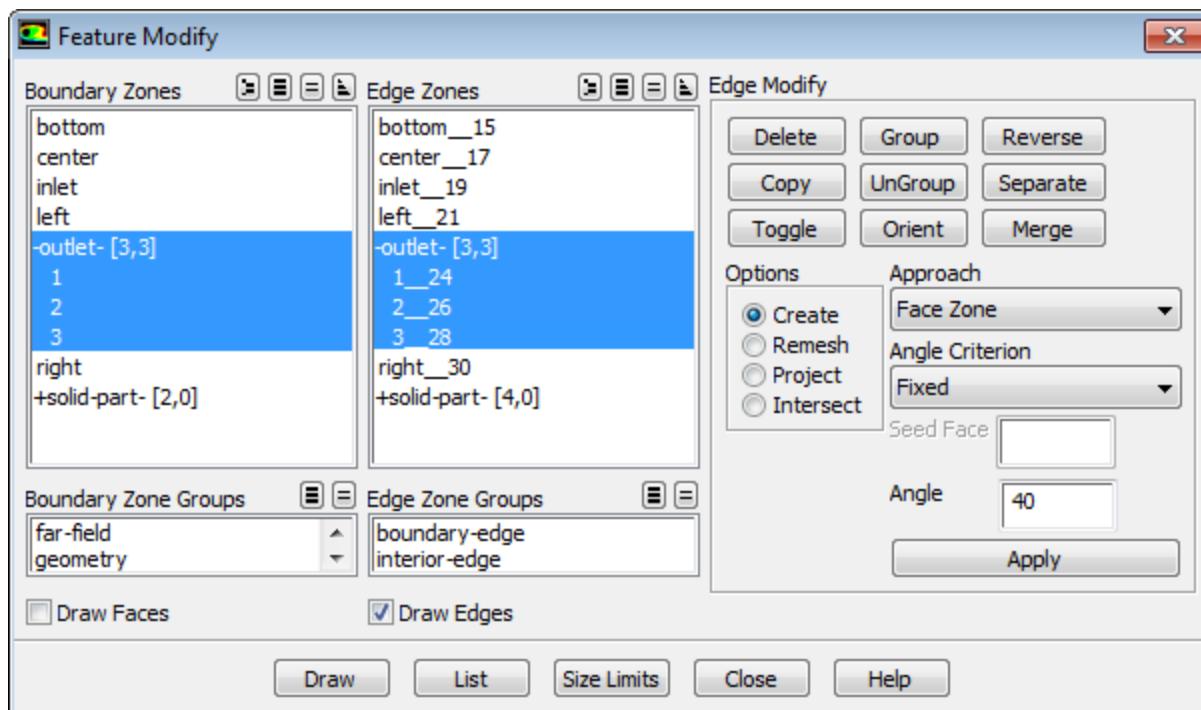
deletes the selected region.

21.3.7. Boundary/Mesh/Feature...

The **Boundary/Mesh/Feature...** menu item opens the [Feature Modify Dialog Box \(p. 516\)](#).

21.3.7.1. Feature Modify Dialog Box

The **Feature Modify** dialog box allows you to create and modify edge loops. See [Creating Edge Loops \(p. 142\)](#) and [Modifying Edge Loops \(p. 145\)](#) for details.



Controls

Boundary Zones

contains a list of boundary face zones from which you can select the boundary zone to be remeshed.

Edge Zones

contains a list of edge loops that have been created for one or more boundary zones, using the **Apply** button.

Edge Zone Groups

contains a list of edge zone types. If you select a zone type from this list (e.g., **boundary-edge**), all edge zones of that type will be selected in the **Edge Zones** list. This shortcut allows you to easily select all zones of a certain type without having to select each zone individually.

Face Zone Groups

contains a list of face zone types. If you select a zone type from this list (e.g., **boundary**), all face zones of that type will be selected in the **Face Zones** list. This shortcut allows you to easily select all zones of a certain type without having to select each zone individually.

Edge Modify

contains buttons for applying the edit operations to the edge loop(s) selected in the **Edge Zones** list.

Delete

deletes the selected edge loops. Note that you should not delete exterior edge loops for non-manifold surfaces (i.e., surfaces with clearly defined borders).

Copy

copies each selected edge loop (including its nodes) to a new edge loop.

Toggle

changes a boundary edge loop to an interior edge loop, or vice-versa.

Group

associates the selected edge loops with the selected face zone (for subsequent remeshing).

UnGroup

ungroups loops that were grouped with the **Group** button.

Orient

orients the edges on the selected edge loops, so that they are all pointing in the same direction.

Reverse

reverses the direction of the edge loop. Note that the direction of a boundary edge loop determines the side from which new faces are formed. The direction should be right-handed with respect to the average normal of the face zone to be remeshed. Faces are always formed on both sides of interior edge loops, so direction is not important for interior loops.

Separate

separates the selected edge loops based on connectivity and the specified **Feature Angle**.

Merge

merges the selected edge loops into a single loop.

Options

contains options and controls for the edge remeshing, projection, and intersection operations that are executed with the combination of the **Remesh**, **Project**, **Create**, and **Intersect** options and the **Apply** button.

Create

contains parameters to create edge loops again.

Approach

contains a drop-down list to choose the approach used for creating edge loops.

Face Zone

specifies the edge loops will be created for the selected face zones in the **Face Zones** list.

Face Seed

specifies the edge loops will be created around a selected seed face.

Angle Criterion

contains a drop-down list to choose the angle criteria used for creating edge loops.

Fixed

specifies the use of fixed angle criteria while creating edge loops. See [Creating Edge Loops \(p. 142\)](#) for details.

Feature Angle

specifies the minimum feature angle that should be considered while creating edge loops.

Adaptive

specifies the use of adaptive angle criteria for creating edge loops. See [Creating Edge Loops \(p. 142\)](#) for details.

Seed Face

specifies the ID of the face that will be used as a seed face.

Remesh

modifies the node distribution on the selected edge loops using the method and spacing defined.

Method

contains the methods that can be used for the node distribution on the edge loops selected in the **Feature Modify** dialog box. You can choose the **Constant**, **Arithmetic**, or **Geometric** spacing method, or **Size Function**.

First Spacing

specifies the node spacing at the beginning of the edge loop or, if the **Constant** method is used, the constant node spacing. If a value of 0 is specified (the default), the spacing will be determined based on the surrounding edges.

Last Spacing

specifies the node spacing at the end of the edge loop. If a value of 0 is specified (the default), the spacing will be determined based on the surrounding edges.

This input is not relevant for the **Constant** method, so it will not be available.

Feature Angle

specifies the minimum feature angle that should be prevented while remeshing.

Quadratic Reconstruction

enables the reconstruction of edges by fitting a quadratic polynomial between the original edge nodes.

Note

The spacing, feature angle, and quadratic reconstruction options are not available when **Size Function** is selected.

Project

projects the edges of the selected edge loop onto the selected face zone, using the specified controls.

Method

contains the methods that can be used for projecting edges.

Closest Point

specifies that each edge should be projected to the closest point on the selected face zone.

Specific Direction

specifies that each edge should be projected in a specified direction onto the selected face zone.

Direction

specifies the **(X, Y, Z)** vector for the direction in which edges should be projected when the **Specified Direction** projection method is used.

Intersect

computes the intersection of the selected edge loops, and creates a new edge loop containing the common edges.

Delete

enables the automatic deletion of all overlapping edges before the intersection is computed. If you want to delete individual overlapping edges, use the **delete-overlapped-edges** command (see [Text Commands for Remeshing \(p. 152\)](#)).

Intersection Tolerance

specifies the tolerance for determining if two edges intersect.

Apply

executes the parameter that you have selected in the **Options** list.

List

reports (in the text window) the zone ID, name, boundary type, and number of faces in each selected edge loop.

Size Limits

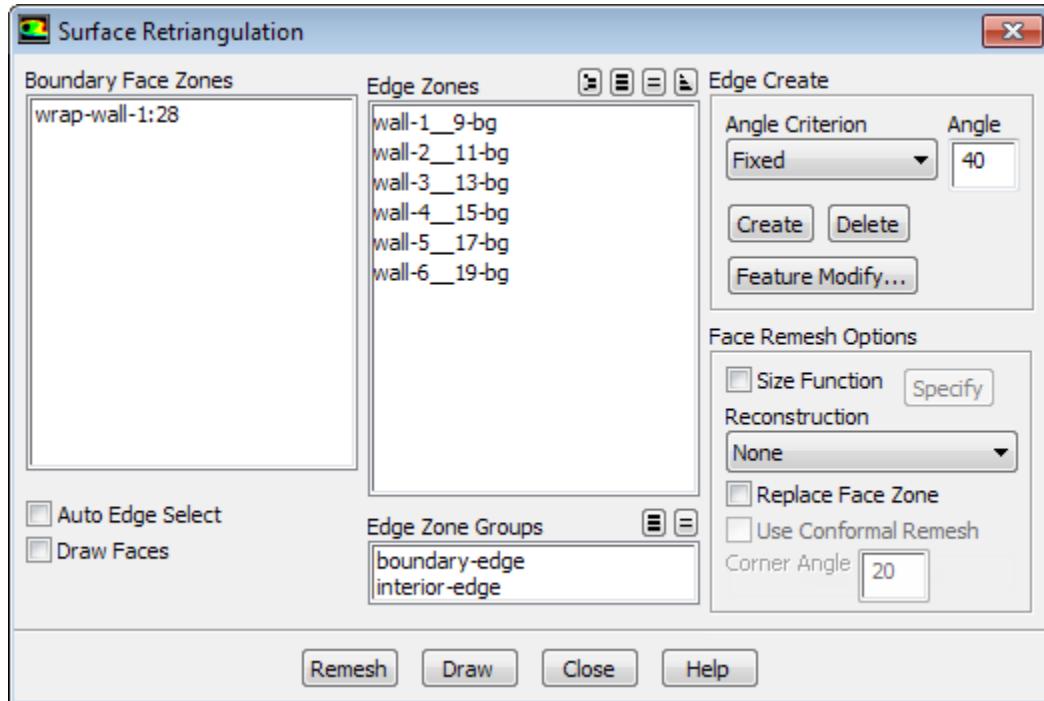
reports (in the **Information** dialog box) the minimum, maximum, and average edge length for each selected edge loop.

21.3.8. Boundary/Mesh/Remesh...

The **Boundary/Mesh/Remesh...** menu item opens the Surface Retriangulation Dialog Box (p. 520).

21.3.8.1. Surface Retriangulation Dialog Box

The **Surface Retriangulation** dialog box allows you to remesh boundary faces. Refer to [Using the Surface Retriangulation Dialog Box \(p. 151\)](#) for details.

**Controls****Boundary Face Zones**

contains a list of boundary face zones from which you can select the boundary zone to be remeshed.

Edge Zones

contains a list of edge loops that have been created for one or more face zones, using the **Create** button.

Edge Zone Groups

contains a list of edge zone types. If you select a zone type from this list (e.g., **boundary-edge**), all edge zones of that type will be selected in the **Edge Zones** list. This shortcut allows you to easily select all zones of a certain type without having to select each zone individually.

Auto Edge Select

selects the edge zones associated with a particular face zones automatically.

Draw Faces

includes the selected face zone in the display when you click the **Draw** button.

Edge Create

contains parameters to create, delete, and modify edge loops.

Angle Criterion

contains a drop-down list to choose the angle criteria used for creating edge loops.

Fixed

specifies the use of fixed angle criteria while creating edge loops. See [Creating Edge Loops \(p. 142\)](#) for details.

(Feature) Angle

specifies the minimum feature angle that will be preserved while creating edge loops.

Adaptive

specifies the use of adaptive angle criteria for creating edge loops. See [Creating Edge Loops \(p. 142\)](#) for details.

Create

creates the edge loops for the face zone selected in the **Face Zones** list. Exterior edge loops are automatically generated on the borders of the face zone, and interior edge loops are generated based on the specified **Feature Angle** or if the edges are multiply-connected.

Delete

deletes the edge loop for the face zone selected in the **Edge Zones** list.

Feature Modify...

opens the **Feature Modify** dialog box using which you can modify the edge loops.

Face Remesh Options

contains parameters for controlling mesh quality of the face zones.

Size Function

allows you to use the advanced size functions for surface remeshing.

Specify

opens the **Size Functions** dialog box where you can define the size functions as appropriate.

Reconstruction (Order)

contains a drop-down list for the reconstruction order that you want to use for the surface you want to remesh.

None

specifies no reconstruction order.

Second Order

specifies the use of second order reconstruction for remeshing. It is recommended to use this option for a coarse surface remeshing.

Third Order

specifies the use of third order reconstruction for remeshing. It is recommended to use this option for a fine surface remeshing.

Replace Face Zone

allows you to replace the original face zone with the remeshed face zone, while retaining the original face zone. When **Replace Face Zone** is enabled, the remeshed face zone will be given the original face zone name while the original face zone will be renamed with **-orig-#** appended to the original name.

Ensure that the new face zone is acceptable before deleting the original zone.

Use Conformal Remesh

allows you to select multiple face zone(s) for conformal remeshing. The feature edge at the shared boundary will be preserved during remeshing.

The shared boundary between different zones will be remeshed only if all the face zones incident to it are selected for conformal remeshing. Also, the remeshed face zone(s) will remain conformal to the rest of the un-remeshed surface.

Note

- **Use Conformal Remesh** is available only when **Size Function** is enabled and **None** is selected in the **Reconstruction** drop-down list. You will be asked to compute the size field or read a size field file.
 - Periodic face zones cannot be remeshed using this option.
-

Corner Angle

specifies the minimum angle between feature edges that will be preserved during remeshing.

Remesh

retriangulates the selected face zone using the edge loops in the **Edge Zones** list. A new face zone will be created upon successful completion of the surface retriangulation.

Important

If you want to exclude an edge loop from the retriangulation, delete it from the **Edge Zones** list before remeshing.

Draw

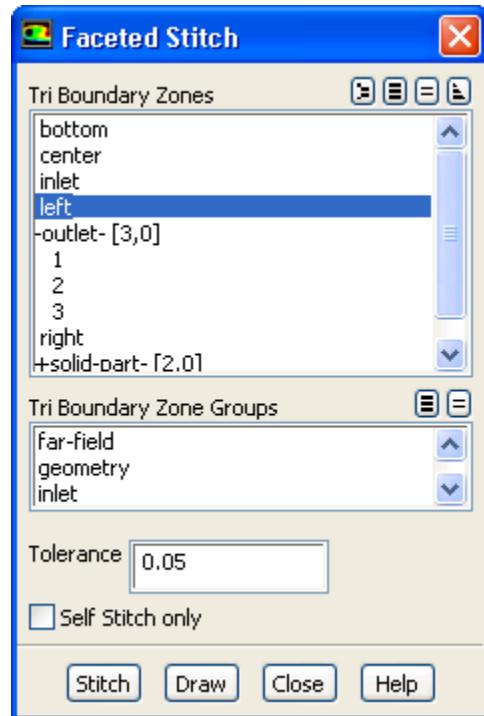
displays the selected **Edge Zones**, using the display settings that are currently defined in the **Display Grid** dialog box. Arrowheads on the edge loops indicate the direction of the loop.

21.3.9. Boundary/Mesh/Faceted Stitch...

The **Boundary/Mesh/Faceted Stitch...** menu item opens the [Faceted Stitch Dialog Box \(p. 523\)](#).

21.3.9.1. Faceted Stitch Dialog Box

The **Faceted Stitch** dialog box contains options for repairing cracks in the surface mesh.



Controls

Tri Boundary Zones

contains a list from which you can select individual boundary zone(s) to be stitched.

Tri Boundary Zone Groups

contains a list of the default boundary zone types and user-defined groups. If you select a zone type/group from this list, all zones of that type/group will be selected in the **Tri Boundary Zones** list. You can also select multiple types to select all the zones of different types (e.g., **inlet** and **outlet**).

Tolerance

specifies the tolerance within which the free edges will be stitched.

Self Stitch only

allows you to stitch the edges within the same boundary zone.

Stitch

stitches the free edges in the zone.

Draw

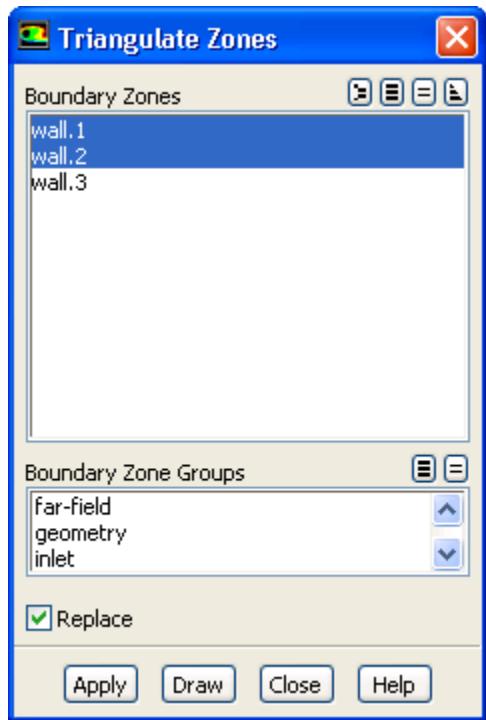
displays the selected zones in the graphics window.

21.3.10. Boundary/Mesh/Triangulate...

The **Boundary/Mesh/Triangulate...** menu item opens the [Triangulate Zones Dialog Box \(p. 524\)](#).

21.3.10.1. Triangulate Zones Dialog Box

The **Triangulate Zones** dialog box contains the options available for triangulating quad zones.



Controls

Boundary Zones

contains a list of the quadrilateral boundary zone(s) available.

Boundary Zone Groups

contains a list of the default boundary zone groups and user-defined groups available. If you select a zone group from this list, all zones of that group will be selected in the **Boundary Zones** list. You can also select multiple types to select all the zones of different types (e.g., **inlet** and **outlet**).

Replace

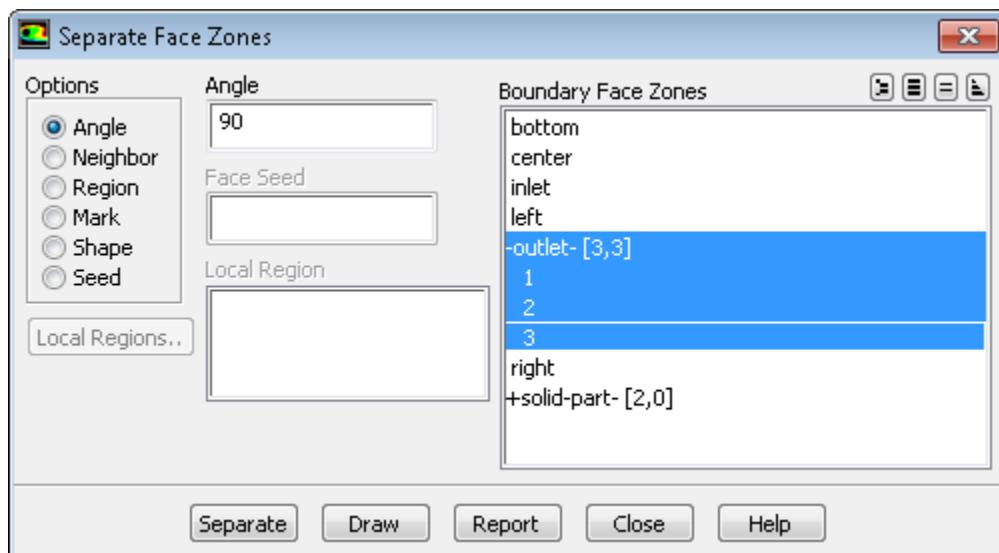
toggles the automatic deleting of the original quadrilateral boundary zone. When this option is enabled, the quadrilateral boundary zone will be replaced by the newly triangulated zone.

21.3.11. Boundary/Zone/Separate...

The **Boundary/Zone/Separate...** menu item opens the Separate Face Zones Dialog Box (p. 524).

21.3.11.1. Separate Face Zones Dialog Box

The **Separate Face Zones** dialog box allows you to separate a single face zone into multiple zones of the same type. See [Methods for Separating Face Zones \(p. 155\)](#) for details.



Controls

Options

specifies the method on which the face separation is to be based.

Angle

indicates that the face zone is to be separated based on significant angle (specified in the **Angle** field).

Neighbor

indicates that the face zone is to be separated based on the cell zones that are adjacent to it.

Region

indicates that the face zone is to be separated based on contiguous regions.

Mark

indicates that the face zone is to be separated based on faces marked.

Shape

indicates that the face zone is to be separated based on the face shape (triangular or quadrilateral).

Seed

indicates that the face zone is to be separated by specifying a seed face element.

Angle

specifies the significant angle to be used when you separate a face zone based on angle. Faces with normal vectors that differ by an angle greater than or equal to the specified significant angle will be placed in different zones when the separation occurs.

Local Region

contains a list of local refinement regions that have been defined using the **Boundary Refinement Region** dialog box. You can select one of these to be used with the **Mark** option, as described in [Methods for Separating Face Zones \(p. 155\)](#).

Face Seed

specifies the label of the face element that you have selected as a seed. The label of the face element is automatically picked up when you select it in the graphics window.

Flood Fill Options

contains options for selecting the method of seed element based separation.

Angle

specifies the significant angle to be used as a feature angle for the face zone separation.

Edge Loop

allows you to separate the face zones based on the edge thread loop associated with the face on which you have defined a seed face element.

Boundary Face Zones

contains a list of face zones from which you can select the zone to be separated.

Local Regions...

opens the **Boundary Refinement Region** dialog box, where you can define a local region to be used in conjunction with the **Mark** option. See [Methods for Separating Face Zones \(p. 155\)](#)

Separate

separates the selected face zone based on the specified parameters.

Draw

displays the zones selected in the **Boundary Face Zones** list.

Report

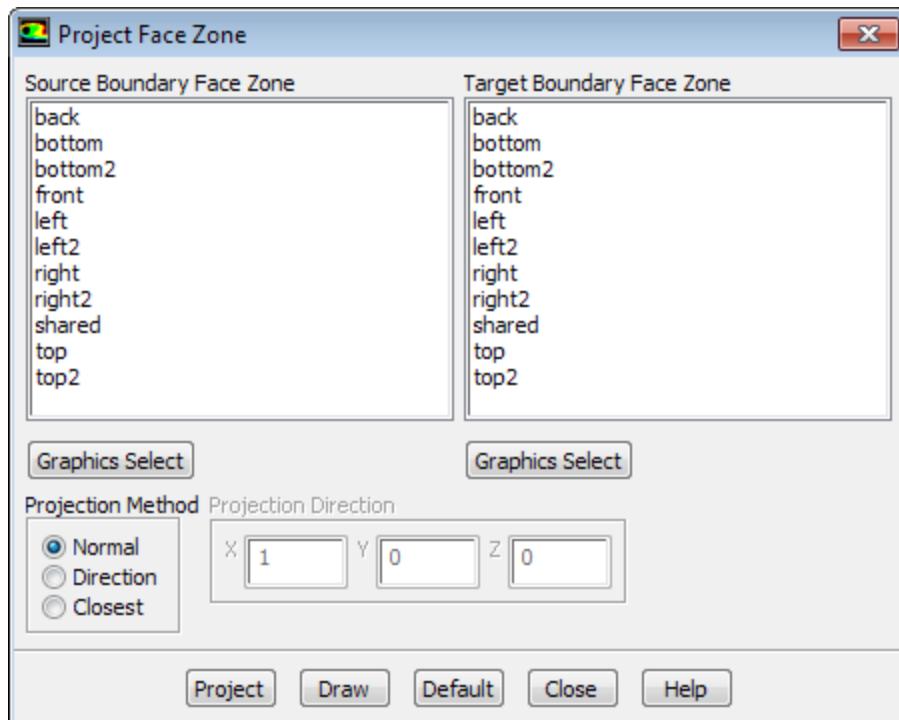
reports the result of the separation without actually separating the face zone.

21.3.12. Boundary/Zone/Project...

The **Boundary/Zone/Project...** menu item opens the [Project Face Zone Dialog Box \(p. 526\)](#).

21.3.12.1. Project Face Zone Dialog Box

The **Project Face Zone** dialog box allows you to project nodes from a selected face zone onto a target face zone.



Controls

Source Boundary Face Zone

contains a list from which you can select a boundary zone to be projected.

Target Boundary Face Zone

contains a list from which you can select a boundary zone to be the target of projection from the source zone.

Graphics Select

allows you to select the zone graphically.

Select a zone in the graphics window, and then click **Graphics Select**.

Projection Method

contains options for defining the method of projection.

Normal

specifies that the projection occurs in the direction normal to the source face zone.

Direction

allows you to specify the direction of projection from the source face zone.

Closest

specifies that, for each node being projected, the projection occurs in the direction of the closest point on the destination face zone.

Projection Direction

specifies the direction of projection (X, Y, Z) when **Direction** is selected in the **Projection Type** list.

Project

completes the zone projection.

Draw

displays the zones selected in the **Source Face Zone** and **Target Face Zone** lists.

Default

resets all controls in the dialog box to their default settings.

21.3.13. Boundary/Zone/Groups...

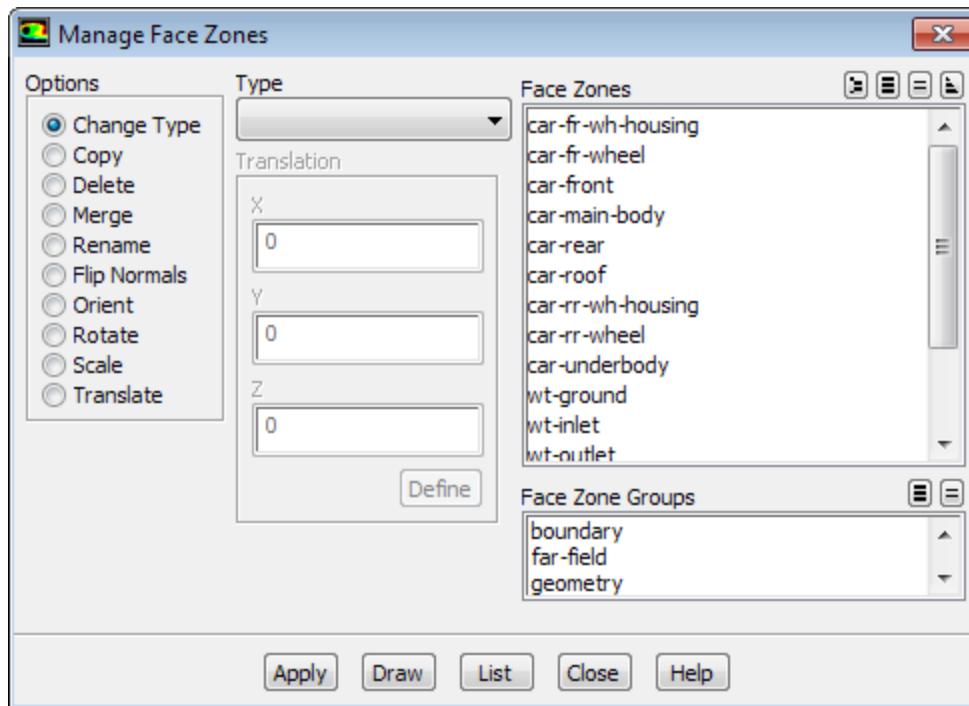
The **Boundary/Zone/Groups...** menu item opens the [User Defined Groups Dialog Box \(p. 428\)](#).

21.3.14. Boundary/Manage...

The **Boundary/Zone/Groups...** menu item opens the [Manage Face Zones Dialog Box \(p. 528\)](#).

21.3.14.1. Manage Face Zones Dialog Box

The **Manage Face Zones** dialog box allows you to manipulate boundary zones. You can use the dialog box to change the boundary type of a zone, delete zones, combine (merge) zones, display (draw) them, print (list) information about them, copy them, rename them, and change their position.

**Controls****Options**

contains a list of the operations to be performed by clicking the **Apply** button.

Change Type

sets the boundary type of the selected zone(s) to be the type selected in the **Type** drop-down list.

Type

contains a drop-down list of boundary types. The boundary types that appear in the list are used only as descriptive names. You can change them in the solution mode in ANSYS Fluent, where

you will also set the related boundary conditions. See the [User's Guide](#) for information about valid boundary types.

Important

The periodic boundary type is not available since it is not possible to change a non-periodic boundary to a periodic boundary using this dialog box. See [Creating Periodic Boundaries \(p. 171\)](#) for information about creating periodic boundaries.

Copy

copies all nodes and faces of the selected zone (or zones), creating a new zone of the same type at the same location. You can then use the **Rotate** and **Translate** options to place the new zone in the appropriate position.

If the copy is placed so that it is connected to an original zone, you will need to merge the duplicate nodes on the original boundary zone and the new boundary zone. This is similar to when you copy a cell zone. See [Figure 18.7: Copying and Translating a Cell Zone \(p. 365\)](#) for details.

Compare free nodes on both boundary zones with all nodes on both boundary zones using the **Merge Boundary Nodes** dialog box.

Delete

deletes all the selected face zone(s). You can also delete the unused nodes associated with the selected face zone(s).

Important

To avoid invalidating the mesh, you cannot delete face zones that are connected to a cell zone.

Delete Nodes

allows you to delete the unused nodes associated with face(s) that are being deleted.

Merge

combines all selected zones into the first zone selected. This operation is useful if you have several different zones that you would like to treat as a single zone.

Merge Options

lists the options available for merging the selected zones.

Alphabetic Order

allows you to retain the name of the zone which comes first in alphabetic order.

Larger Area

allows you to retain the name of the zone having a bigger area.

Warning

The **Delete** and **Merge** operations are irreversible.

Rename

allows you to change the name of the selected zone.

Name

specifies the new name for the zone selected in the **Face Zones** list.

Change Prefix

allows you to change the prefix for the selected face zones.

From

specifies the current prefix for the selected face zones.

To

specifies the required prefix for the selected face zones.

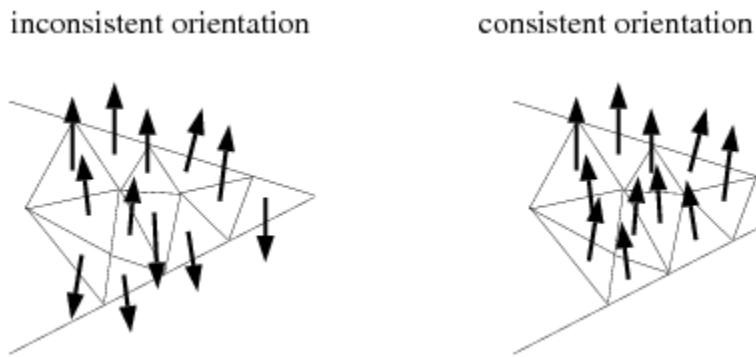
Flip Normals

reverses the normal direction of the selected boundary zone(s).

Orient

orients contiguous faces (i.e., faces that touch each other) in a selected zone so that their normals are all consistently pointing on the same side of the zone (see [Figure 21.3: Inconsistently and Consistently Oriented Contiguous Faces \(p. 530\)](#)). This consistent orientation is especially important if you are going to build prisms from the boundary zone.

Figure 21.3: Inconsistently and Consistently Oriented Contiguous Faces

**Rotate**

rotates the selected zone(s) through the specified angle.

Angle

specifies the angle through which you want to rotate the selected zone(s).

Pivot

specifies the pivot point for the rotate operation.

Axis

specifies the axis of rotation for the rotate operation.

Define

allows you to select 1–6 nodes to define the pivot point and axis of rotation.

Preview

displays a triad at the pivot point, with the local z-axis indicating the axis of rotation.

Clear

clears the preview triad from the graphics window.

Copy Zone(s)

allows you to copy the nodes and faces of the selected zone(s), thereby creating new zone(s) of the same type before positioning the copied zone(s) per the specified **Angle**, **Pivot** and **Axis**.

Scale

scales the selected zone(s) by multiplying each of the node coordinates by the specified **Scale Factors**. The face sizes will increase or decrease accordingly.

Scale Factors

specifies the scale factors applied to the grid in each of the Cartesian coordinate directions (**X**, **Y**, and **Z**).

Copy Zone(s)

allows you to copy the nodes and faces of the selected zone(s), thereby creating new zone(s) of the same type before scaling the copied zone(s) per the specified scale factors.

Translate

translates the selected zone(s) by the specified translation offsets.

Translation

specifies the translation offsets (**X**, **Y**, **Z**) to be added to the Cartesian coordinate of every node in the selected zone(s).

Define

allows you to define the direction vector and total distance based on two nodes or positions selected in the graphics window.

Note

The translation offsets are interpreted as absolute numbers in meshing mode. In solution mode, however, the translation offsets are assumed to be distances in the length unit set. This may lead to differences in domain extents reported after translating the mesh in the respective modes.

Copy Zone(s)

allows you to copy the nodes and faces of the selected zone(s), thereby creating new zone(s) of the same type before positioning the copied zone(s) per the specified translation offsets.

Face Zones

contains a list from which you can select the zone(s) to be modified.

Face Zone Groups

contains a list of the default boundary zone types and the user-defined groups. If you select a boundary type/group from this list (e.g., **inlet**), all boundary zones of that type/group (for this example, all **pressure-inlet** and **velocity-inlet** boundaries) will be selected in the **Face Zones** list. This shortcut allows you to easily select all boundary zones of a certain type without having to select each zone individually. You can select multiple boundary types in the **Face Zone Groups** list to select all zones of several different types (e.g., **inlet** and **outlet**).

Apply

applies the option to the selected zones.

Draw

displays the selected zones in the active graphics window.

List

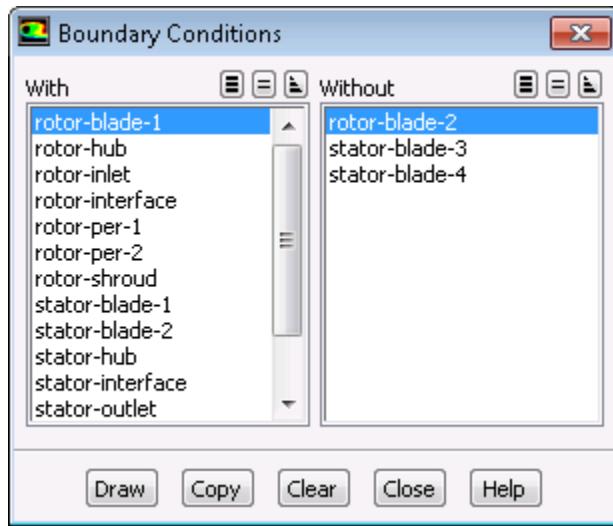
reports (in the console) the zone ID, name, boundary type, the face type, and the number of faces for each selected zone.

21.3.15. Boundary/Boundary Conditions...

The **Boundary/Boundary Conditions...** menu item opens the [Boundary Conditions Dialog Box \(p. 532\)](#).

21.3.15.1. Boundary Conditions Dialog Box

The **Boundary Conditions** dialog box allows you to copy or clear boundary conditions assigned to the boundary zones, when a case file is read.

**Controls****With**

contains a list of boundary zones that have boundary conditions assigned.

Without

contains a list of boundary zones without boundary conditions assigned.

Draw

displays the selected zones in the graphics window.

Copy

allows you to copy the boundary conditions from the zone selected in the **With** list to those selected in the **Without** list.

Clear

allows you to clear the boundary conditions assigned to the zone(s) selected in the **With** selection list. The zone(s) will be available in the **Without** selection list once the assigned boundary conditions are cleared.

21.3.16. Boundary/Create/Bounding Box...

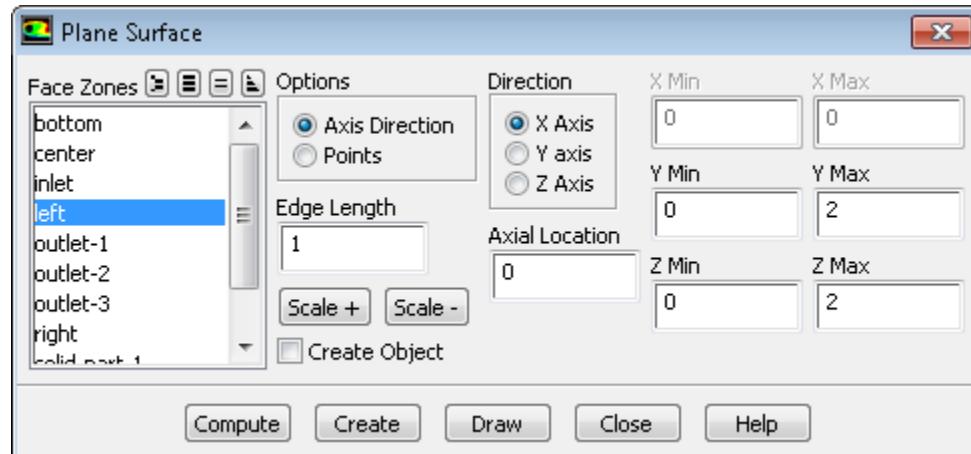
The **Boundary/Create/Bounding Box...** menu item opens the [Bounding Box Dialog Box \(p. 416\)](#).

21.3.17. Boundary/Create/Plane Surface...

The **Boundary/Create/Plane Surface...** menu item opens the [Plane Surface Dialog Box \(p. 533\)](#).

21.3.17.1. Plane Surface Dialog Box

The **Plane Surface** dialog box allows you to create a planar surface mesh.



Controls

Face Zones

contains a list of existing face zones in the geometry.

Options

contains methods available for creating a plane surface.

Axis Direction

allows you to create a planar surface mesh perpendicular to any coordinate axes.

Points

specifies the selection of planar points method to create a plane surface mesh.

Direction

contains check buttons for specifying the axis about which you want to create a perpendicular plane surface.

X Axis, Y Axis, Z Axis

allows you to create the perpendicular plane surface about the selected axis direction. These parameters are available only if you select **Axis Direction** under **Options**.

X Min, X Max, Y Min, Y Max, Z Min, Z Max

allow you to specify the minimum and maximum limits of the surface mesh in the X, Y, and Z directions, respectively.

Axial Location

allows you to specify the location of the surface mesh along the selected axis. This parameter is available only if you select **Axis Direction** under **Options**.

Planar Points

contains options to specify the coordinates of the three points that define the planar surface.

X Pos, Y Pos, Z Pos

specify X, Y, and Z coordinates of the planar points.

P1, P2, P3

denote the three planar points.

Select Points...

allows you to select planar points using the mouse. When you click this button, a **Working** dialog box will appear, prompting you to select three points to define a plane. After you select the three points, this dialog box closes automatically and the **Planar Points** fields will be updated.

Important

Make sure you have displayed the geometry in the graphics window before clicking **Select Points...** button.

Edge Length

allows you to specify the size of the triangular cells that you want to create for the plane surface.

Scale +

displays a rectangular surface, having diagonal size 1.25 times the existing planar surface that you have created in the graphics window. Select the plane surface that you have created in the **Face Zones** list. This will create a temporary surface of larger size and its corresponding coordinates will be updated in the **Planar Points** field.

Scale -

displays a rectangular surface, having diagonal size 0.8 times the existing planar surface that you have created in the graphics window. Select the plane surface that you have created in the **Face Zones** list. This will create a temporary surface of smaller size and its corresponding coordinates will be updated in the **Planar Points** field.

Create Object

allows you to create a wrap object based on the plane surface face zone created. The wrap object created comprises the face zone of the plane surface and the corresponding edges. The object cell zone type will be set to **fluid** and the priority will be set to p-1, where p is the lowest priority specified for the current object(s).

Create

opens the [Zone Type Dialog Box \(p. 417\)](#), where you can specify the zone name and type for the plane surface zone created.

21.3.18. Boundary/Create/Cylinder...

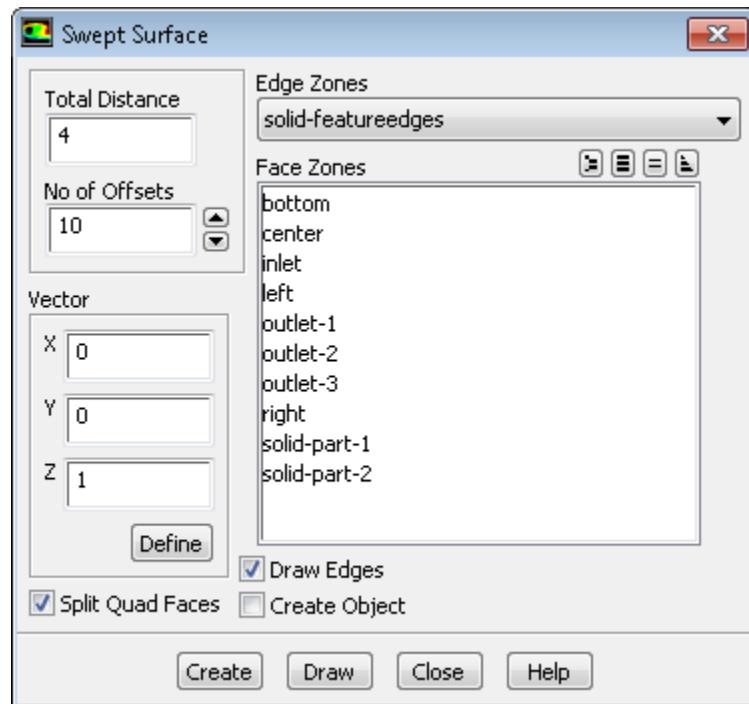
The **Boundary/Create/Cylinder...** menu item opens the [Cylinder Dialog Box \(p. 417\)](#).

21.3.19. Boundary/Create/Swept Surface...

The **Boundary/Create/Swept Surface...** menu item opens the [Swept Surface Dialog Box \(p. 535\)](#).

21.3.19.1. Swept Surface Dialog Box

The **Swept Surface** dialog box allows you to create a swept surface.



Controls

Total Distance

specifies the total distance through which the edge is to be projected.

No. of Offsets

specifies the number of offsets.

Edge Zones

contains a drop-down list of all the existing edge zones.

Face Zones

contains a list of existing face zones in the geometry.

Vector

specifies the direction in which the selected edge is to be swept.

Define

allows you to define the direction vector and total distance based on two nodes or positions selected in the graphics window.

Split Quad Faces

toggles the creation of tri/quad faces for the swept surface.

Draw Edges

displays the selected edge zones, using the display settings that are currently defined in the [Display Grid Dialog Box \(p. 596\)](#). Arrowheads on the edge loops indicate the direction of the loop.

Create Object

allows you to create a wrap object based on the swept surface face zone created. The wrap object created comprises the face zone of the swept surface and the corresponding edges. The object cell zone type will be set to **fluid** and the priority will be set to $p-1$, where p is the lowest priority specified for the current object(s).

Create

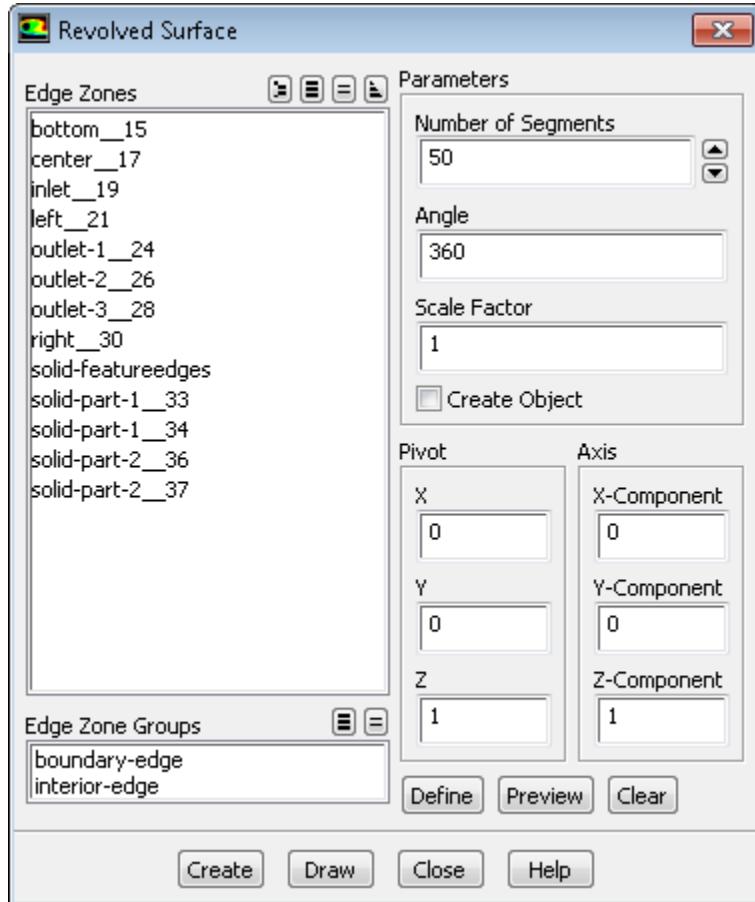
creates the swept surface.

21.3.20. Boundary/Create/Revolved Surface...

The **Boundary/Create/Revolved Surface...** menu item opens the [Revolved Surface Dialog Box \(p. 536\)](#).

21.3.20.1. Revolved Surface Dialog Box

The **Revolved Surface** dialog box allows you to create a revolved surface from the selected edge zone(s).

**Controls****Edge Zones**

contains a list of edge zones that have been created.

Edge Zone Groups

contains a list of edge zone types. If you select a zone type from this list (e.g., **boundary-edge**), all edge zones of that type will be selected in the **Edge Zones** list. This shortcut allows you to easily select all zones of a certain type without having to select each zone individually.

Parameters

contains parameters to be used for creating the revolved surface.

Number of Segments

specifies the number of edge segments to be used for the revolved surface created.

Angle

specifies the angle through which the edge is to be revolved.

Scale Factor

controls the radius of the revolved surface created.

Create Object

allows you to create a wrap object based on the revolved surface zone created. The wrap object created comprises the face zone of the revolved surface and the corresponding edges. The object cell zone type will be set to **fluid** and the priority will be set to $p-1$, where p is the lowest priority specified for the current object(s).

Pivot

specifies the pivot point for revolving the selected edge(s).

Axis

specifies the axis of rotation for revolving the selected edge(s).

Define

allows you to select 1–6 nodes to define the pivot point and axis of rotation.

Preview

displays a triad at the pivot point, with the local z-axis indicating the axis of rotation.

Clear

clears the preview triad from the graphics window.

Create

creates the revolved surface.

21.3.21. Boundary/Create/Periodic...

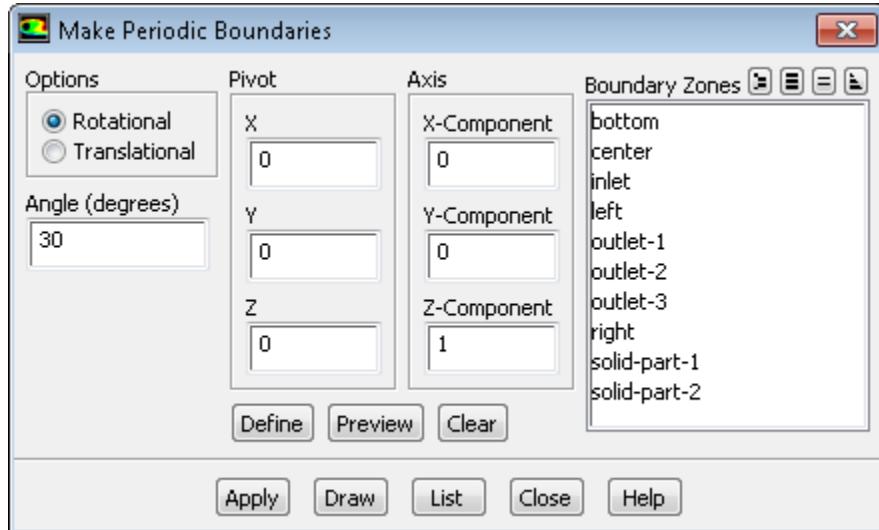
The **Boundary/Create/Periodic...** menu item opens the [Make Periodic Boundaries Dialog Box \(p. 537\)](#).

21.3.21.1. Make Periodic Boundaries Dialog Box

The **Make Periodic Boundaries** dialog box allows you to create a matched pair of periodic/periodic-shadow boundaries from an existing zone. You can select more than one zone for which periodic pairs should be created, if appropriate.

For translational periodicity problems, specify the translational shift, while for rotational periodicity problems specify the angle of rotation and the axis of rotation. After setting the appropriate parameters,

click **Apply**. A periodic-shadow boundary will be created for each zone selected and change each of the selected zones to a periodic boundary zone.



Controls

Options

selects the type of periodicity: rotational or translational.

Rotational

For **Rotational** periodicity, you will specify the **Angle**, **Pivot** and **Axis**.

Angle

specifies the angle of rotation for rotational periodicity.

Pivot

specifies the pivot point for rotational periodicity.

Axis

specifies the axis of rotation for rotational periodicity.

Define

allows you to select 1–6 nodes to define the pivot point and axis of rotation as described in [Creating Periodic Boundaries](#)

Preview

displays a triad at the pivot point, with the local z-axis indicating the axis of rotation.

Clear

clears the preview triad from the graphics window.

Translational

For **Translational** periodicity, you will specify the **Shift**.

Shift

specifies the shift vector (X, Y, and Z components) for translational periodicity.

Define

allows you to define the shift vector based on two nodes or positions selected in the graphics window.

Boundary Zones

contains a list from which you can select the zone(s) to be made periodic.

Apply

creates a periodic-shadow boundary for each zone selected, and changes each of the selected zones to a periodic zone.

Draw

displays the selected zones in the active graphics window.

List

reports (in the text window) the zone ID, name, type, and number of faces in the zone for each selected zone.

21.4. Mesh Menu

21.4.1. Mesh/Auto Mesh...

The **Mesh/Auto Mesh...** menu item opens the [Auto Mesh Dialog Box \(p. 464\)](#).

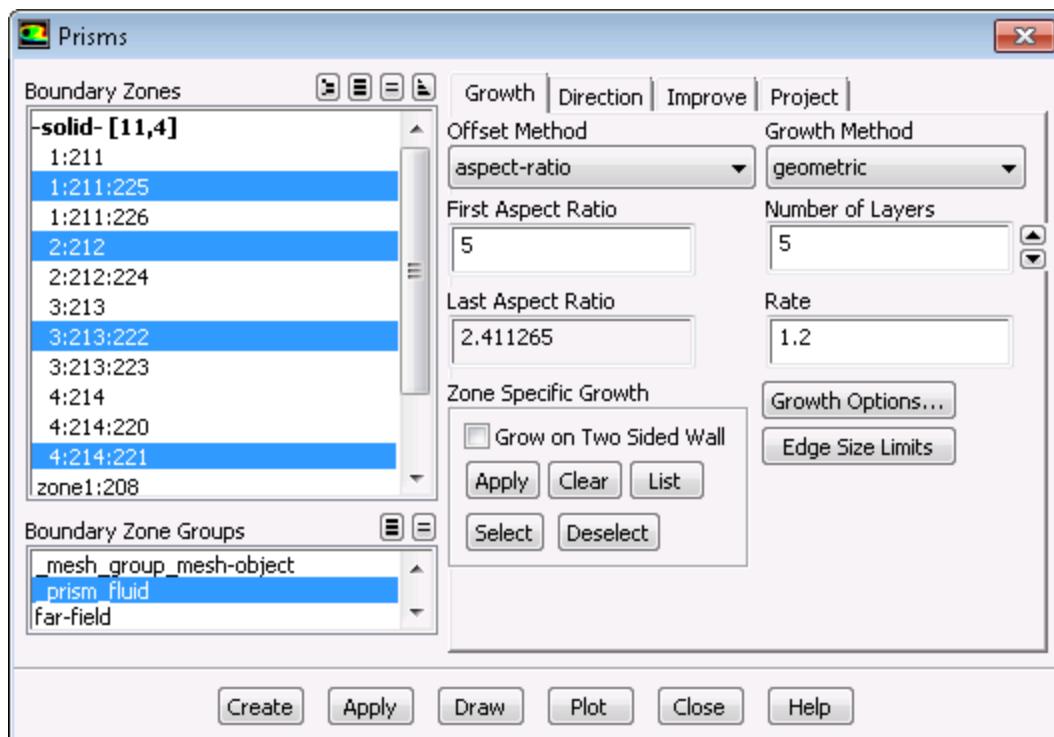
21.4.2. Mesh/Prisms...

The **Mesh/Prisms...** menu item opens the [Prisms Dialog Box \(p. 539\)](#).

21.4.2.1. Prisms Dialog Box

The **Prisms** dialog box allows you to specify parameters needed for prism cell creation, and then create the prisms. When you click any of the tabs along the top of the dialog box, the contents change to show the related controls. The controls for all inactive tabs are hidden.

The **Prisms** dialog box consists of a series of tabbed frames, within the main dialog box, listed horizontally across the top of the dialog box. Each frame has a name and a set of related controls.



Controls

Boundary Zones

contains a selection list of all of the boundary zones in your mesh. You will select the zone(s) from which you want the prisms to be built.

Boundary Zone Groups

contains a list of boundary zone types and user-defined groups. If you select a boundary type (or group) from this list (e.g., **inlet**), all boundary zones of that type (or group) will be selected in the **Boundary Zones** list. This shortcut allows you to easily select all boundary zones of a certain type without having to select each zone individually. You can select multiple boundary types in the **Boundary Zone Groups** list to select all zones of several different types (e.g., **inlet** and **outlet**).

You can also select **tri** or **quad** in this list to select all triangular or quadrilateral boundary zones.

Growth

displays the controls for computing the height of each layer of prisms. This is the default tab of the **Prisms** dialog box.

Offset Method

contains a drop-down list of the methods available for computing the offset distance. See [Offset Distances \(p. 247\)](#) for details.

uniform

specifies the uniform offset method.

aspect-ratio

specifies the aspect-ratio offset method.

last-ratio

specifies the last-ratio offset method.

minimum-height

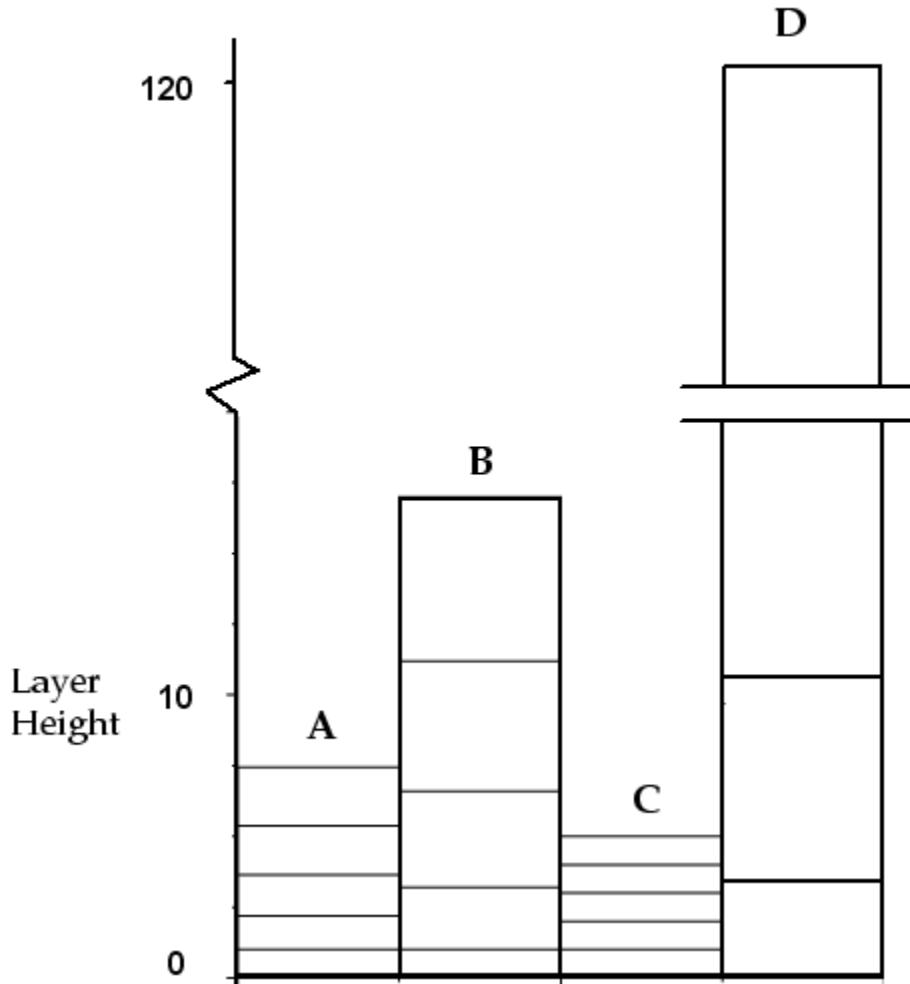
specifies the minimum-height offset method.

Growth Method

contains a drop-down list of the methods available for computing the prism height. [Figure 21.4: Layer Heights Computed Using the Four Growth Methods \(p. 541\)](#) shows layer heights for all four methods, using a first height of 1 and a slope/rate/exponent of 1.2.

Figure 21.4: Layer Heights Computed Using the Four Growth Methods

A = geometric C = constant
 B = linear D = exponential

**constant**

enables the specification of a constant height for all prism layers. When this method is selected, you will need to specify the **First Height** and the **Number of Layers**. The height of each prism layer will be the **First Height (h₁)**.

linear

enables the specification of a variable height that decreases or increases linearly for each layer of prisms:

$$h_n = h_1 + s(n-1)$$

When this method is selected, you will need to specify the **First Height (h_1)**, the **Number of Layers (n)**, and the **Slope (s)**.

geometric

enables the specification of a variable height that decreases or increases geometrically for each layer of prisms:

$$h_n = r^{n-1} h_1$$

When this method is selected, you will need to specify the **First Height (h_1)**, the **Number of Layers (n)**, and the **Rate (r)**.

exponential

enables the specification of a variable height that decreases or increases exponentially for each layer of prisms:

$$h_n = h_1 e^{(n-1) p}$$

When this method is selected, you will need to specify the **First Height (h_1)**, the **Number of Layers (n)**, and the **Exponent (p)**.

First Height

specifies the height of the first prism layer (h_1). The computed heights of subsequent prism layers will be based on this height, depending on the **Growth Method** selected.

Number of Layers

sets the total number of prism layers to be created.

Total Height

reports the total height of the prisms to be grown. You can use this information to check if the specified prism heights will yield a mesh of the required height.

If you want to see the updated **Total Height**, press the **Enter** key after entering a new value for the other growth parameters. If you do not press **Enter** after entering the value, the **Total Height** reported will not reflect the changed value.

First Aspect Ratio

specifies the first aspect ratio to be used when the **aspect-ratio** offset method is selected.

Last Aspect Ratio

reports the last aspect ratio based on the specified first aspect ratio for the **aspect-ratio** offset method.

Last Percent

specifies the percentage to be used along with the **First Height** when the **last-ratio** offset method is selected.

Slope

sets the slope (**s**) of the linear function used to compute the heights of the prism layers when the **linear** growth method is selected.

Rate

sets the rate of growth (**r**) of the geometric function used to compute the heights of the prism layers when the **geometric** growth method is selected.

Exponent

sets the exponent (**p**) for the exponential function used to compute the heights of the prism layers when the **exponential** growth method is selected.

Growth Options...

opens the **Prisms Growth Options** dialog box.

Edge Size Limits

displays (in an **Information** dialog box and in the console) the minimum, maximum, and average edge lengths for all selected **Boundary Zones**. The **First Height** should usually be smaller than the minimum length reported.

Zone Specific Growth

allows you to simultaneously grow prisms from multiple zones using *different* growth parameters.

By default, this option will be enabled if you read in a mesh for which separate growths were specified for different zones and then subsequently saved. It will be disabled if you read in a mesh that does not have saved prism parameters.

You need not use any of these controls if you are building prisms from just one zone, building them from multiple zones using the same growth parameters, or growing layers for multiple zones at different times.

Grow on Two Sided Wall

allows you to grow prisms on both sides of a two sided wall.

Note

The **Grow On Two Sided Wall** option cannot be used when the **Uniform** method is selected.

Apply

applies the specified growth parameters to the selected **Boundary Zones**.

Note

When setting prism parameters for use with the **Auto Mesh** option, it is necessary to apply the prism growth parameters on specific zones to retain the growth parameters in memory. The **Prisms** option in the **Auto Mesh** dialog box will be visible only after applying zone-specific growth (by clicking **Apply** in the **Zone Specific Growth** group box).

Clear

clears the zone-specific growth settings attached to the zones selected in the **Boundary Zones** list.

List

lists the zone-specific growth settings for the zones selected in the **Boundary Zones** list in the console.

Select

selects the zones which have zone-specific growth settings specified in the **Boundary Zones** list, if they are not already selected.

Deselect

deselects the zones which have zone-specific growth settings specified from those selected in the **Boundary Zones** list.

Plot

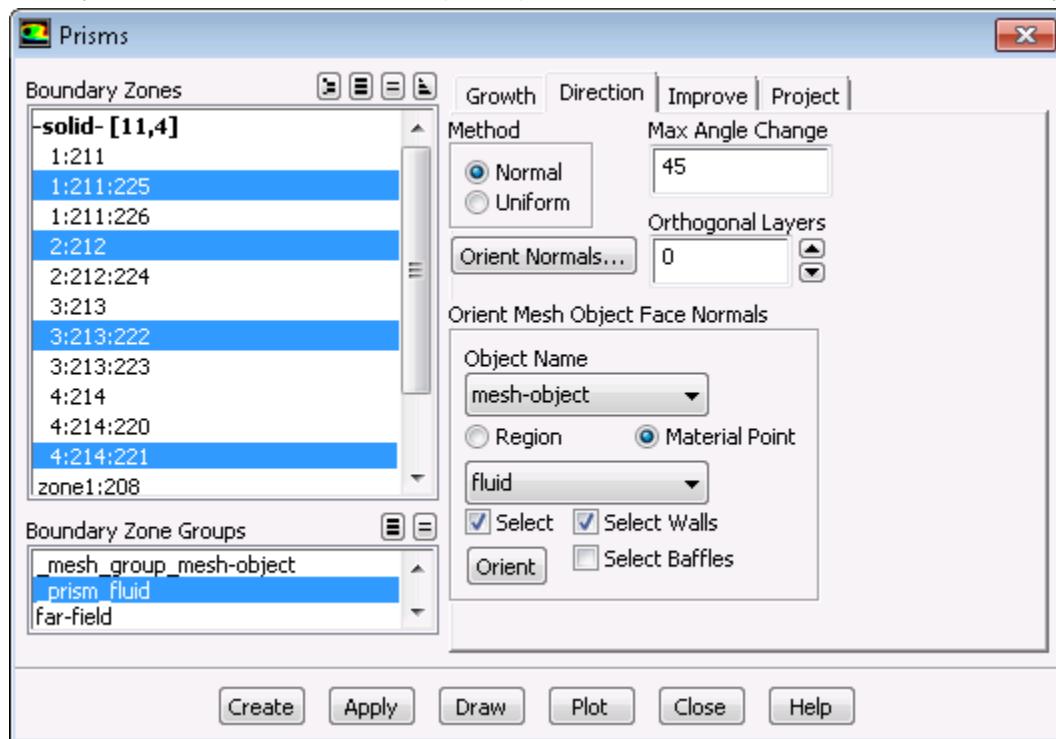
displays (in the active graphics window) an illustration of the prism layers, allowing you to preview the layer heights before actually creating the prisms. The plot will show the layer heights for all selected boundary zones, and (if different) the heights for the current settings in the **Prisms** dialog box.

Print

reports the growth parameters, prism height for each layer, and total height in the console. The report will show the data for all selected zones, and (if different) the data for the current settings in the **Prisms** dialog box.

Direction

displays the controls for computing the growth directions for the nodes on each layer of prisms.

**Method**

lists the available methods for specifying the prism growth direction. See [Direction Vectors \(p. 244\)](#) for details about these methods.

Normal

enables the computation of direction vectors normal to the boundary zones from which you are growing.

Uniform

enables a uniform direction specification for all nodes on all prism layers. When you select this method, you need to specify the direction **Vector**.

This constant vector will be used for all nodes on all prism layers, instead of computing a new direction for each node on each layer. Therefore, this option is most useful for creating straight-sided prism regions without any curvature.

Orient Normals...

allows you to orient faces based on the material point specified. Clicking the **Orient Normals...** button will open the [Orient Normals Dialog Box \(p. 551\)](#).

Max Angle Change

specifies the maximum angle by which the normal direction at a node can change during smoothing. See [Preserving Orthogonality \(p. 239\)](#) and [Normal Smoothing \(p. 245\)](#) for details about this parameter. This option is available only when the **Normal** method is selected.

Vector

specifies the uniform direction for all layers of prisms. You can either enter values in the **X**, **Y**, **Z** fields, or click the **Compute** button to get the average normal direction of the selected zone(s). This option is available only when the **Uniform** method is selected.

Compute

computes the average normal direction to the selected zone or zones (i.e., the average of the unit normals of all the faces in the zones). This direction is usually a good choice for the uniform direction, especially if the zone is planar. This option is available only when the **Uniform** method is selected.

Warning

If you use the **Compute** button to calculate the average normal direction, do not change the sign of the displayed **Vector**. It is important for the uniform direction vector to have the same sign as the normal direction for the face zone.

Orthogonal Layers

specifies the number of layers that should remain orthogonal to the original boundary. For an orthogonal layer, no edge swapping or normal, edge, or node smoothing will be performed. See [Preserving Orthogonality \(p. 239\)](#) for details. This option is available only when the **Normal** method is selected and it is set to zero by default.

Orient Mesh Object Face Normals

contains options for orienting mesh object face normals.

Object Name

contains a list of mesh objects available. You can select the mesh object for which the boundary zones need to be separated.

Region

allows you to select a region into which the prism layers are to be grown. In this case the separation and selection is based on corresponding wrap object face zones.

Material Point

allows you to select a material point inside the fluid zone for orienting the normals.

Select

separates the boundary zones included in the mesh object and selects them in the **Boundary Zones** list.

Select Walls

separates the boundary zones included in the mesh object and selects the wall zones in the **Boundary Zones** list.

Select Baffles

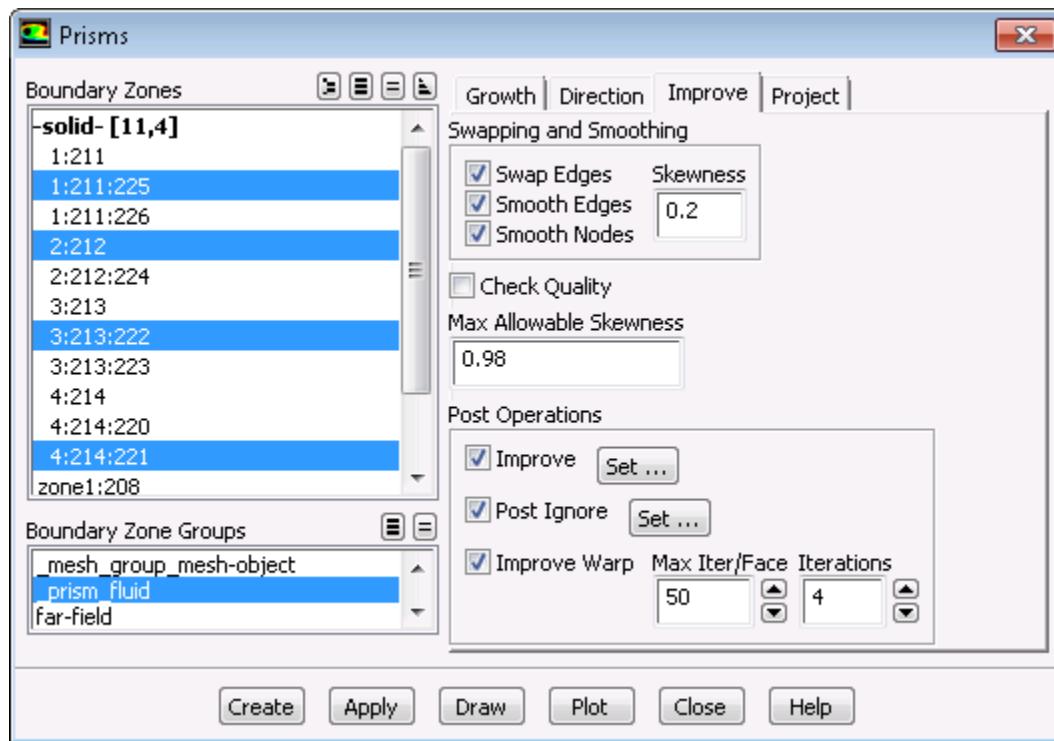
separates the boundary zones included in the mesh object and selects the baffle zones in the **Boundary Zones** list.

Orient

separates the correct boundary zones for the mesh object and orients the face normals for the prism mesh generation.

Improve

contains the controls for various procedures used to improve the quality of the prisms. These controls are not used if the **Uniform** direction method is selected.

**Swapping and Smoothing**

contains controls for the edge swapping and smoothing, and node smoothing.

Swap Edges

enables/disables edge swapping (see [Edge Swapping and Smoothing \(p. 249\)](#)) to decrease the skewness of highly skewed faces. This option is enabled by default.

Smooth Edges

enables/disables edge smoothing (see [Edge Swapping and Smoothing \(p. 249\)](#)) to decrease the skewness of highly skewed faces. This option is enabled by default.

Smooth Nodes

enables/disables node smoothing (see [Node Smoothing \(p. 250\)](#)) to decrease the skewness of highly skewed faces. This option is disabled by default.

Skewness

specifies the face skewness above which edge swapping and smoothing are triggered. Only faces having skewness greater than this value are evaluated for these operations. The default value is 0.7.

Check Quality

enables/disables the checking of face and cell quality after each prism layer is created. When this option is enabled (the default setting), the cell volume of every new cell, the face-handedness of every face on every new cell, and the skewness of all new cap faces will be checked.

If any new cell has zero or negative volume or any left-handed faces, or if the skewness exceeds the specified **Max. Allowable Skewness**, prism layer creation will be stopped. See [Negative Volumes/Left-Handed Faces/High Skewness \(p. 262\)](#) for details.

Max. Allowable Skewness

specifies the maximum skewness allowed for a prism cap face. If the skewness exceeds the specified value, the meshing process will stop with a warning that the skewness for that layer is too high. You can then examine the skewed faces or cells, make appropriate corrections, and recreate the prisms. See [Negative Volumes/Left-Handed Faces/High Skewness \(p. 262\)](#) for details.

Post Operations

contains options for improving the prisms in a postprocessing step which is carried out after all the required prism layers are created

Improve

allows you to improve the prism cell quality based on the quality measure selected (see [Improving the Prism Quality \(p. 250\)](#)).

Set...

opens the [Prism Improve Dialog Box \(p. 581\)](#), where you can select various options for improving the prism cell quality.

Post Ignore

allows you to remove poor quality prism cells based on the options specified in the **Prism Post Ignore** dialog box..

Set...

opens the [Prism Post Ignore Dialog Box \(p. 585\)](#), where you can set options for the prism post-ignore operation.

Improve Warp

allows you to improve face warp (see [Improving Warp \(p. 252\)](#)) after prism generation. This option is disabled by default.

Max Iter/Face

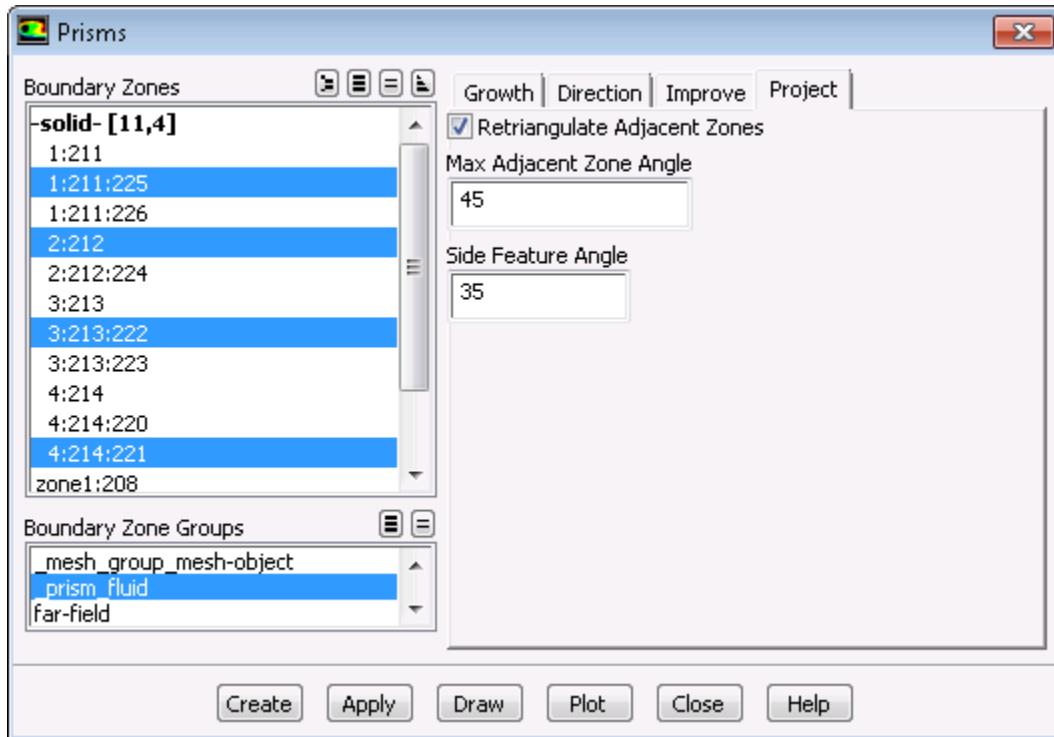
specifies the maximum number of attempts for improving a particular face.

Iterations

specifies the number of iterations through the warp improvement procedure.

Project

contains controls for projection and retriangulation.



Retriangulate Adjacent Zones

specifies whether or not the triangular face zones adjacent to which outer nodes have been projected will be automatically retriangulated. This option is enabled by default. See [Retriangulation \(p. 242\)](#) for details about this procedure.

Max Adjacent Zone Angle

sets the tolerance used to determine whether or not to use an adjacent zone. If a zone shares outer nodes with any of the zones from which the layers are being grown (the "base zones"), its angle with respect to the growth direction is compared with the specified value. If the angle is less than or equal to this value, then the zone will be used. See [Using Adjacent Zones as the Sides of Prisms \(p. 241\)](#) for details. The default is 45 degrees.

Side Feature Angle

specifies the angle used for computing the feature normals.

Create

creates the prisms using the settings specified in the dialog box.

If the volume mesh already exists, a **Question** dialog box will appear, asking if you want to morph the existing volume mesh.

Apply

applies all current settings for prism growth, without creating the prisms. If you wish to set the parameters in advance and return later to generate the prisms, click **Apply** before closing the **Prisms** dialog box.

There is no need to click **Apply** before moving from one tab in the dialog box to another (e.g., from the **Growth** tab to the **Direct** tab).

Note

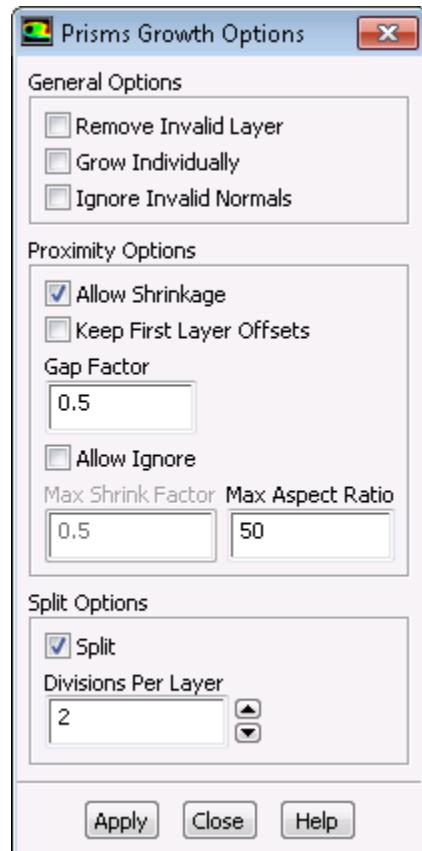
Clicking **Apply** only applies the parameters set in the **Prisms** dialog box. When setting prism parameters for use with the **Auto Mesh** option, click **Apply** in the **Zone Specific Growth** group box to retain the growth parameters on specific zones in memory. The **Prisms** option in the **Auto Mesh** dialog box will be visible only after applying zone-specific growth.

Draw

displays the boundary zone(s) selected in the **Boundary Zones** list. All current settings in the **Display Grid** dialog box will be reflected in the resulting grid display. You may find it useful to enable the display of normal vectors so that you can easily determine the normal direction(s) of the selected boundary zone(s).

21.4.2.2. Prisms Growth Options Dialog Box

The **Prisms Growth Options** dialog box allows you to specify the parameters needed for detecting proximity/collision, retaining the first layer offset, and growing prisms individually from multiple zones.



Controls

General Options

contains additional options for prism growth.

Remove Invalid Layer

allows you to retain the valid prism layers grown before prism layer failure is reported.

Grow Individually

indicates whether individual prism cell zones and prism cap face zones should be retained or merged with like zones. By default this option is disabled, so a single zone of prism cells and a single zone of cap faces will be created for each set of simultaneously grown layers. See [Growing Prisms Simultaneously from Multiple Zones \(p. 232\)](#) for details.

Ignore Invalid Normals

allows you to ignore regions of invalid normals while creating prisms.

Proximity Options

contains options for proximity detection.

Allow Shrinkage

enables the shrinking of prism layers in areas of proximity to avoid prism collision.

Keep First Layer Offsets

allows you to retain the original first offset height while scaling offset heights in case of intersecting prism layers.

Gap Factor

controls the gap between the intersecting prisms layers with respect to the cell size of the prisms.

Allow Ignore

allows you to ignore prism growth parameters in areas of proximity to avoid prism collision.

Max Shrink Factor

determines the shrinkage of prism layers to avoid collision. This option is available only when the **Allow Ignore** option is enabled.

Max Aspect Ratio

specifies the maximum aspect ratio to determine the limit for the shrinkage of prism layers. This option is available only when the **Allow Ignore** option is disabled.

Note

If the cell aspect ratio exceeds the specified maximum aspect ratio, a message will appear during the prism meshing process, indicating that shrinkage was limited by the maximum aspect ratio specified.

Warning

The value specified for **Max Aspect Ratio** is only used as a guideline for limiting the shrinkage of the prism layers. The specified value may not always be adhered to in the final mesh as the value may be relaxed during the prism improvement operations such as smoothing, etc.

Split Options

contains options for splitting the prism layers to generate the total number of layers required.

Split

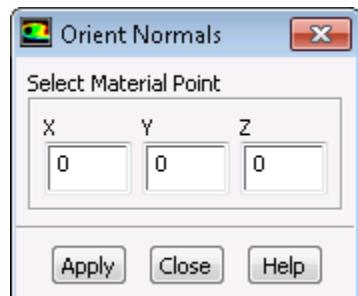
allows you to split the prism layers to generate the total number of layers required.

Divisions Per Layer

specifies the number of divisions to be made per layer

21.4.2.3. Orient Normals Dialog Box

The **Orient Normals** dialog box allows you to specify the material point based on which the normals will be oriented.

**Controls****X Y Z**

specifies the material point location.

Apply

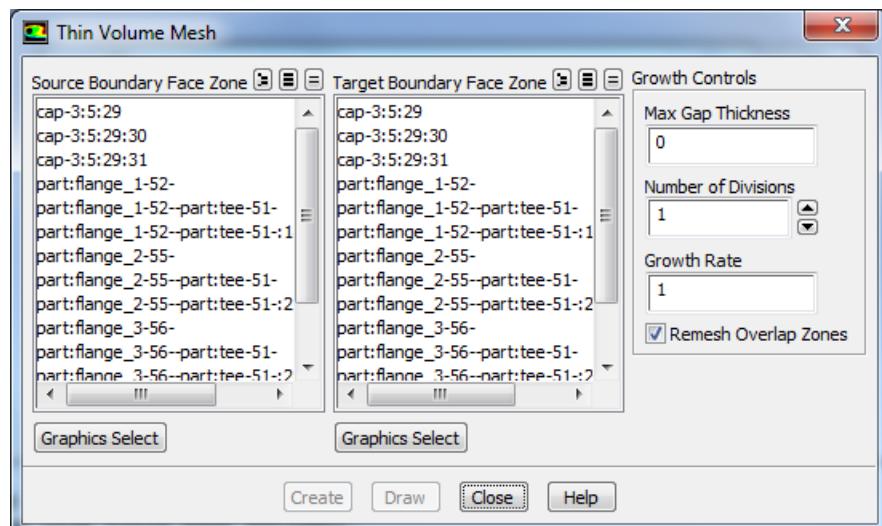
orients the normals based on the specified material point.

21.4.3. Mesh/Thin Volume Mesh...

The **Mesh/Thin Volume Mesh...** menu item opens the **Thin Volume Mesh** dialog box.

21.4.3.1. Thin Volume Mesh Dialog Box

The **Thin Volume Mesh** dialog box allows you to specify all the parameters necessary to create the swept mesh in a thin volume region.



Controls**Source Boundary Face Zone**

is used to choose the source boundary face zones.

Graphics Select

allows you to select zones graphically.

Select a zone, or zones, in the graphics window, and then click **Graphics Select**.

Note

Graphically deselect source zones before selecting target zones to avoid including source zones in the target zone list.

Target Boundary Face Zone

is used to choose target face zone.

Note

Select a single target face zone for the thin volume meshing operation.

Graphics Select

allows you to select zones graphically.

Select a zone in the graphics window, and then click **Graphics Select**.

Gap Thickness

helps determine the mesh region. If set to 0, the mesher will automatically calculate the gap thickness.

If non-zero, the Gap Thickness defines the maximum separation between source and target zones in the swept-mesh region.

Number of Divisions

specifies the number of layers created between source and target face zones.

Growth Rate

specifies the maximum thickness ratio between two adjacent layers.

Remesh Overlap Zones

If checked, any overlapped part of the surface mesh on the target and adjacent face zones will be remeshed. Original meshes are replaced.

Create

creates the thin volume mesh using the source and target zones selected.

Draw

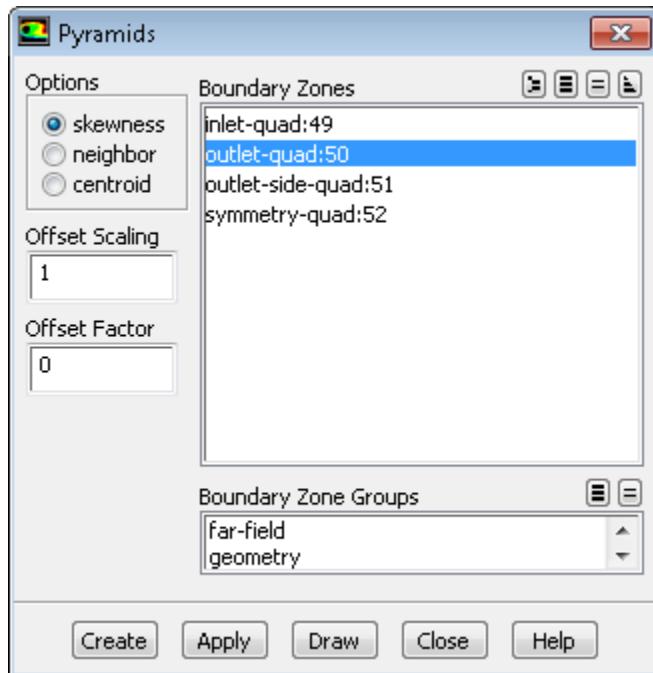
displays the mesh in the graphics window.

21.4.4. Mesh/Pyramids...

The **Mesh/Pyramids...** menu item opens the [Pyramids Dialog Box \(p. 553\)](#).

21.4.4.1. Pyramids Dialog Box

The **Pyramids** dialog box allows you to specify all the parameters required for pyramid creation and to create the pyramids. See [Creating Pyramids \(p. 221\)](#) or details about using the options in the **Pyramids** dialog box.



Controls

Options

contains the options for determining the location of the new vertex of the pyramid.

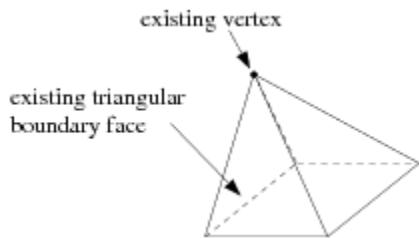
Skewness

specifies that the new vertex of a pyramid should be placed just above the centroid of the quadrilateral base, or the pyramid should share the vertex of a neighboring pyramid or a node on an adjacent boundary face, whichever will yield the best skewness (see [Figure 21.5: Using an Existing Node as the Pyramid Vertex \(p. 554\)](#) and [Figure 21.6: Pyramid Vertex Placed Above the Centroid \(p. 554\)](#)). This is the default option, and is the best choice for most cases.

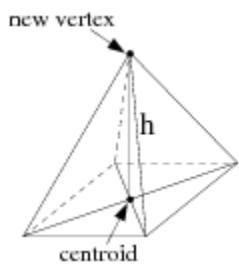
Neighbor

specifies that if the pyramid is adjacent to a triangular boundary face and the angle formed by the quadrilateral base and the triangular boundary face is not greater than 115 degrees, the pyramid should not use a node on that boundary as its new vertex, ([Figure 21.5: Using an Existing Node as the Pyramid Vertex \(p. 554\)](#)).

Otherwise, the new vertex should be placed just above the centroid of the quadrilateral base. If the node of a neighboring boundary face is used as the vertex, the height specification will be ignored for that pyramid.

Figure 21.5: Using an Existing Node as the Pyramid Vertex**Centroid**

specifies that the new vertex of the pyramid should always be placed just above the centroid of the quadrilateral base of the pyramid. See [Figure 21.6: Pyramid Vertex Placed Above the Centroid \(p. 554\)](#).

Figure 21.6: Pyramid Vertex Placed Above the Centroid**Offset Scaling**

specifies the scaling (s) to be used to determine the height of the pyramid. The height (h) is computed from the following equation:

$$h = \frac{sL}{\sqrt{2}} \quad (21.1)$$

where

$$L = \sqrt{A}$$

and A = area of the quadrilateral base of the pyramid.

Offset Factor

specifies the fraction of the computed pyramid height (offset) by which the pyramid heights will be randomly adjusted. The default value is 0, indicating that all pyramids will have the exact height computed. A value of 0.1, for example, will limit each adjustment to $\pm 10\%$ of the computed height.

Boundary Zones

contains a list from which you can select the zone(s) to be operated on.

Boundary Zone Groups

contains a list of boundary zone types. If you select a boundary type from this list (e.g., **inlet**), all boundary zones of that type (for this example, all **pressure-inlet** and **velocity-inlet** boundaries) will be selected in the **Boundary Zones** list. This allows you to easily select all boundary zones of a certain type without having to select each zone individually. You can select multiple boundary types in the **Boundary Zone Groups** list to select all zones of several different types (e.g., **inlet** and **outlet**).

Create

creates the pyramids you have defined.

Draw

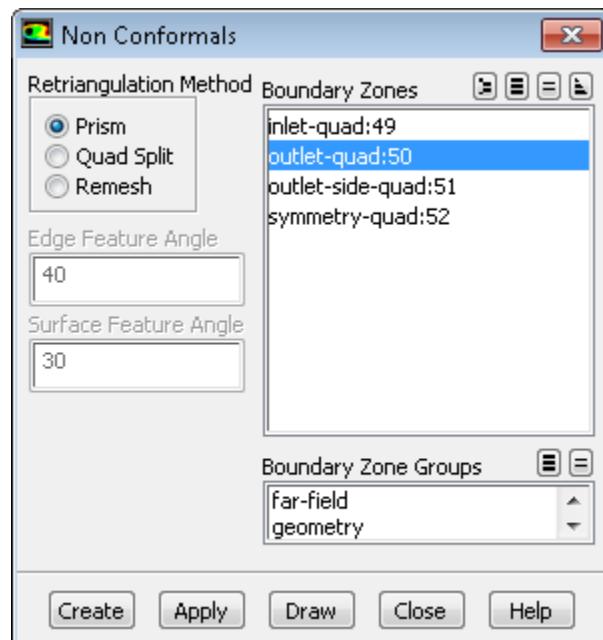
displays the boundary zone(s) selected in the **Boundary Zones** list. All current settings in the **Display Grid** dialog box will be in effect in the resulting grid display.

21.4.5. Mesh/Non Conformals...

The **Mesh/Non Conformals...** menu item opens the [Non Conformals Dialog Box \(p. 555\)](#).

21.4.5.1. Non Conformals Dialog Box

The **Non Conformals** dialog box allows you to specify the method to be used for retriangulating the quad faces on the non-conformal zones.



Controls

Retriangulation Method

contains a list of methods available for retriangulating the quadrilateral faces on the non-conformal zones.

Prism

remeshes the prism-side quad zones named **prism-side*** or ***-quad***.

Quad Split

splits the quad faces diagonally into tri faces.

Remesh

remeshes all the zones comprising quad faces based on the edge and surface feature angle specified.

Boundary Zones

contains a list of the boundary zones.

Boundary Zone Groups

contains a list of the available boundary zone groups, including the user-defined groups.

Edge Feature Angle

specifies the edge feature angle for remeshing when the **Remesh** method is selected.

Surface Feature Angle

specifies the surface feature angle for remeshing when the **Remesh** method is selected.

Create

creates the non-conformal interface according to the specified parameters.

Apply

applies the specified parameters to the zones selected.

Draw

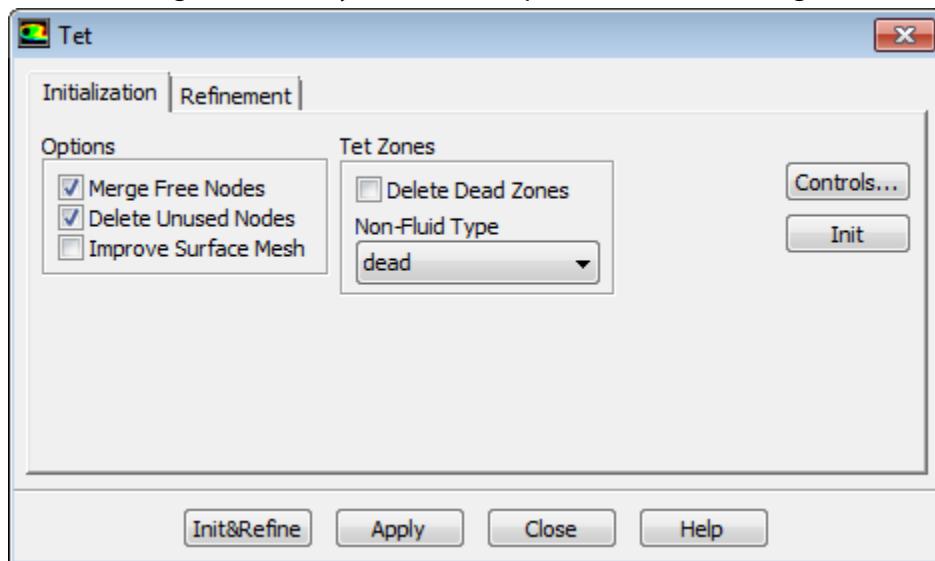
displays the zones selected in the **Boundary Zones** selection list in the graphics window.

21.4.6. Mesh/Tet...

The **Mesh/Tet...** menu item opens the [Tet Dialog Box \(p. 556\)](#).

21.4.6.1. Tet Dialog Box

The **Tet** dialog box allows you to set the parameters controlling tet meshing.

**Controls****Initialization**

contains options for initializing the mesh.

Options

contains options to be included/excluded in the mesh initialization process.

Merge Free Nodes

merges free nodes (see [Free and Isolated Nodes \(p. 119\)](#)).

Delete Unused Nodes

deletes nodes which are not referenced by any faces (see [Free and Isolated Nodes \(p. 119\)](#)).

Improve Surface Mesh

performs edge swapping based on the Delaunay criterion to improve the quality of the surface mesh.

Tet Zones

contains parameters for tetrahedral cell zone initialization.

Delete Dead Zones

toggles the automatic deleting of dead zones.

Non-Fluid Type

allows you to select the non-fluid cell type. The non-fluid zones will be set to the selected type after initializing the mesh.

Note

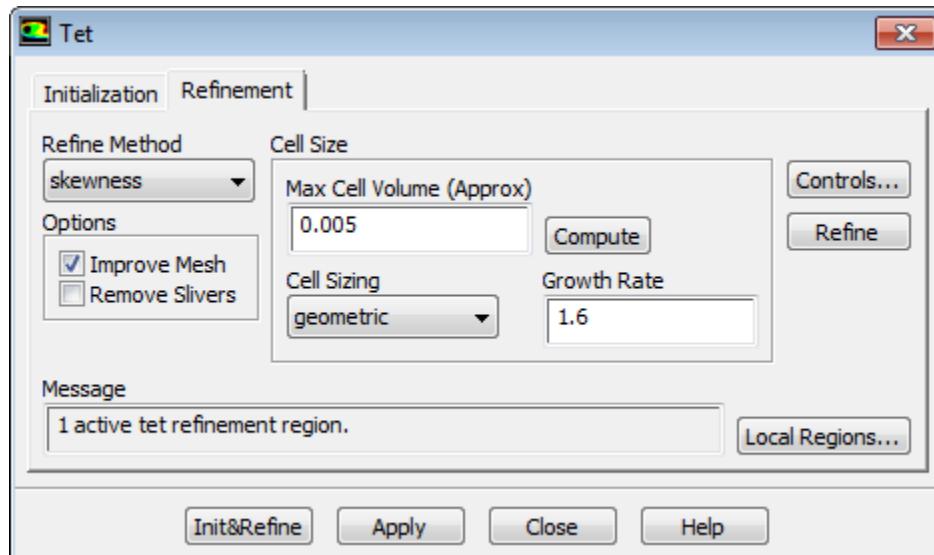
- For mesh objects created using **Sew**, the **Non-Fluid Type** field value will be applied to dead zones which are not claimed by solid/fluid objects and material points.
- For mesh objects created using **Build Topology**, the **Non-Fluid Type** field value will not be respected while using **Auto Mesh**.

Controls...

opens the **Tet Init Controls** dialog box.

Init

initializes the mesh.

**Refinement**

contains options for refining the mesh.

Refine Method

allows you to select the refinement method to be used.

Options

contains options to be included/excluded in the refinement process.

Improve Mesh

improves the mesh during refinement (see [Smoothing Nodes \(p. 347\)](#) and [Swapping \(p. 349\)](#)).

Remove Slivers

removes slivers (see [Removing Slivers from a Tetrahedral Mesh \(p. 350\)](#)).

Cell Size

contains parameters controlling the tetrahedral cell size.

Max Cell Volume (Approx)

specifies the maximum acceptable volume for a cell. Any active cell with a size greater than the specified value will be refined.

Compute

computes the maximum cell volume based on the longest edge in the boundary mesh.

When a mesh object is selected in the **Auto Mesh** dialog box, you can compute the maximum cell volume based on the mesh object selected.

Cell Sizing

allows you to control the cell size distribution in the interior of the mesh.

none

determines the cell size distribution based on the **Max Cell Volume** and skewness. This option is available only for the skewness-based refinement.

linear

determines the cell size distribution by linear interpolation from the boundary cell size.

geometric

determines the cell size distribution by geometric growth from the closest boundary, according to the specified growth factor.

size-function

determines the cell size distribution based on the current size functions or size field.

Growth Rate

specifies the growth factor for determining the cell size distribution when the **geometric** method is selected from the **Cell Size Function** drop-down list.

Controls...

opens the **Tet Refine Controls** dialog box.

Refine

refines the mesh based on the specified parameters and activated regions.

Local Regions...

opens the **Tet Refinement Region** dialog box.

Init&Refine

initializes and refines the mesh.

Apply

applies the parameters set in the **Tet** dialog box.

21.4.6.2. Tet Init Controls Dialog Box

The **Tet Init Controls** dialog box allows you to set additional parameters to control the initial mesh generation.



Controls

Defaults

enables the use of the default parameter values for mesh initialization

Sliver Size

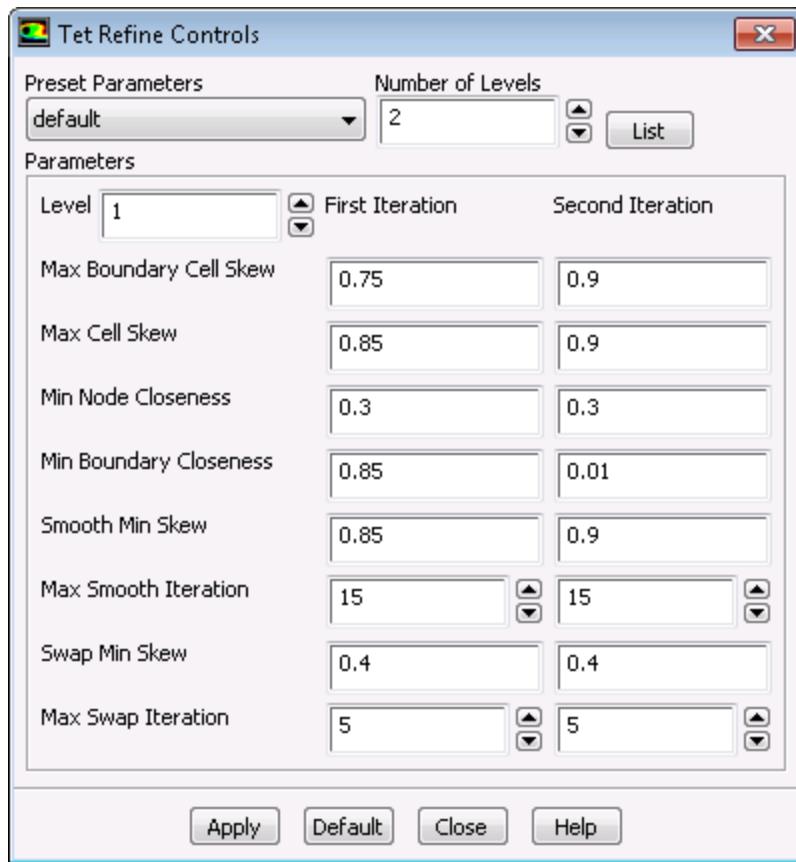
is the smallest cell size which can be determined accurately. Cells having a calculated size less than the specified sliver size will be assigned a size of zero. This option is available only when the **Defaults** option is disabled.

Node Tolerance

is the smallest distance between two distinct nodes. If the distance between two nodes is less than the specified node tolerance, the nodes are considered to be duplicates. This option is available only when the **Defaults** option is disabled.

21.4.6.3. Tet Refine Controls Dialog Box

The **Tet Refine Controls** dialog box allows you to set additional parameters to control the skewness-based refinement of the mesh.



Controls

Preset Parameters

contains a drop-down list of the preset refinement control parameters available.

default

specifies the default refinement parameters.

fast-transition

specifies the refinement parameters which will result in quicker mesh gradation away from the boundaries and will generate fewer cells as compared to the **default** option.

incremental-improve

specifies the refinement parameters which will result in incremental lowering of skewness.

Number of Levels

specifies the number of refinement levels.

List

displays the refinement parameters for the specified number of levels in the console.

Parameters

contains input fields for the refinement parameters.

Level

specifies the level of refinement for which refinement parameters are displayed.

Each refinement level consists of two sweeps through the refinement procedure.

Max Boundary Cell Skew

specifies the maximum acceptable skewness for a boundary cell. All active boundary cells with skewness greater than this value will be refined

Max Cell Skew

specifies the maximum acceptable skewness for a cell. All active cells with skewness greater than this value will be refined.

Min Node Closeness

specifies the distance from an interior node (normalized by the local mesh size) within which it is appropriate to add a node. This restriction prevents a node from being added too close to another node.

Min Boundary Closeness

specifies the distance from a boundary node (normalized by the local mesh size) within which it is appropriate to add a node. Do not add nodes too close to the boundary because it will be nearly impossible to remove a highly skewed cell, if created.

Smooth Min Skew

specifies the minimum skewness required to perform smoothing.

Max Smooth Iteration

specifies the maximum number of smoothing iterations.

Swap Min Skew

specifies the minimum skewness required to perform swapping.

Max Swap Iteration

specifies the maximum number of swapping iterations.

Apply

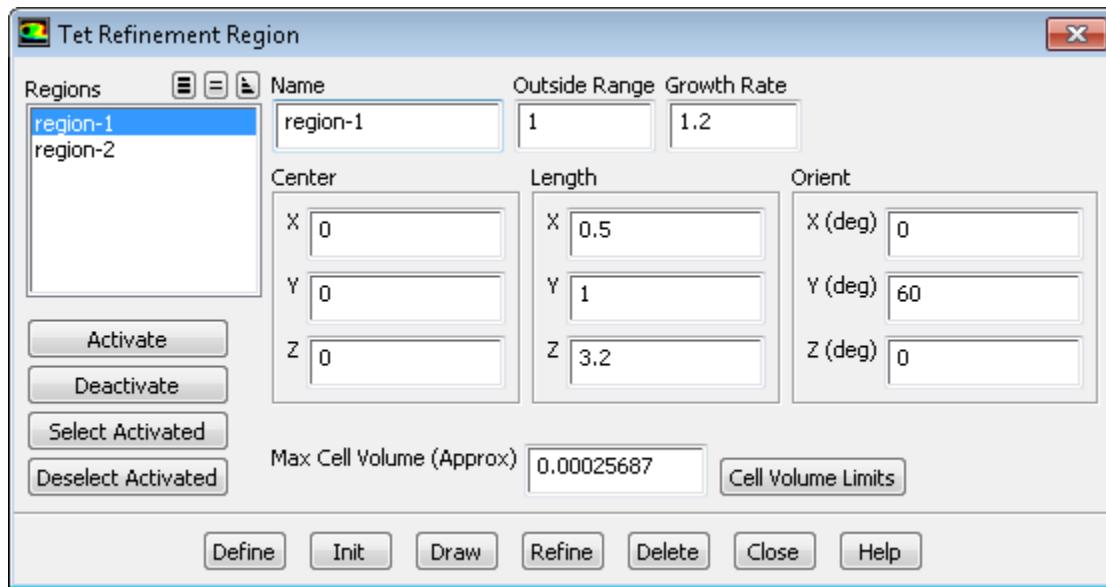
applies the specified refinement parameters.

Default

restores all parameters to their default values.

21.4.6.4. Tet Refinement Region Dialog Box

The **Tet Refinement Region** dialog box provides options for creating, modifying, and activating/deactivating refinement regions, for tetrahedral mesh refinement.



Controls

Regions

contains a list of the defined regions.

Activate

activates the region selected in the **Regions** list.

Deactivate

deactivates the region selected in the **Regions** list.

Select Activated

selects the activated region(s) in the **Regions** list.

Deselect Activated

deselects the activated region(s) in the **Regions** list.

Name

reports the name of the selected region. You can specify a new name by entering it in this text entry box.

Outside Range

defines the size of the outer transition region in which the geometric growth factor is applied to obtain a smooth transition between the original and the refined cells.

Growth Rate

is the geometric growth factor applied in the outer transition region.

Center

allows you to specify the coordinates of the center of the region you want to create. This is useful when you know the region in which refinement is required. Specify the coordinates of a point in that region as a center of the region to be created.

Length

allows you to specify the absolute size of the new region in x, y, and z directions.

Orient

allows you to specify the orientation of the new region.

Note

The region is oriented by rotation first about the x-axis, then the y-axis, and finally the z-axis. You need to take this into account while specifying the orientation of the region as rotation in any other order will produce different results.

Max Cell Volume

sets the maximum cell volume for the selected region. You can change this value by entering a new value in this text entry box.

Cell Volume Limits

reports (in an **Information** dialog box and the console) the number of cells in the selected region, and the minimum, maximum, and average size of these cells.

Define

creates a new region according to the specified parameters. It also allows you to modify the selected region according to the specified changes.

Init

creates a default region encompassing the entire geometry.

Draw

draws the region(s) in the graphics window. The maximum cell size for the refinement and outer regions will also be displayed. If the boundary mesh was displayed (using the **Display Grid** dialog box) before drawing the refinement region, the mesh will also be included in the display.

Refine

refines active cells inside the selected region, based on the specified refinement parameters.

Delete

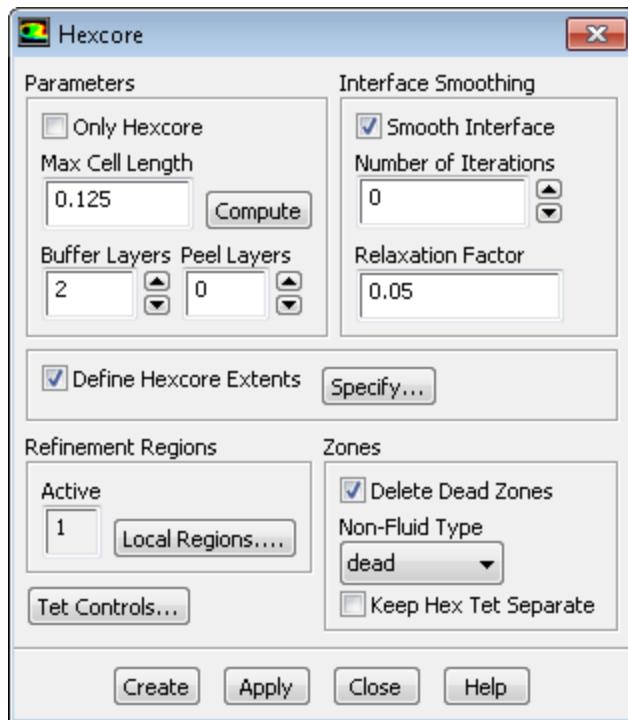
deletes the selected region.

21.4.7. Mesh/Hexcore...

The **Mesh/Hexcore...** menu item opens the [Hexcore Dialog Box \(p. 563\)](#).

21.4.7.1. Hexcore Dialog Box

The **Hexcore** dialog box allows you to control the hexcore meshing procedure.



Controls

Parameters

contains options that control the hexcore meshing in the domain.

Only Hexcore

allows you to prevent the automatic creation of the tetrahedral mesh after hexcore generation. When the **Only Hexcore** option is enabled, the hexcore is created and the tetrahedral domain is activated. You can manually create the tetrahedral mesh separately.

Max Cell Length

sets the maximum length for the hex cells in the domain. The final length may not be the specified value as it is also limited by other parameters.

Compute

computes the maximum cell length based on the surface mesh.

Buffer Layers

specifies the number of additional layer of cells to subdivide. Increasing the buffer layers will significantly increase the number of cells. Refer to [Buffer Layers \(p. 291\)](#) for details.

Peel Layers

controls the gap between the hexahedra core and the geometry. Refer to [Peel Layers \(p. 291\)](#) for details.

Interface Smoothing

specifies the parameters controlling interface smoothing.

Smooth Interface

toggles smoothing at the hexcore interface.

Number of Iterations

sets the number of smoothing iterations on the parent face. Increasing the number of iterations will increase the value of maximum skewness of the hex cells near the boundary. It also helps initializing the tetrahedra and obtaining lower maximum skewness in the tetrahedral region. The default number of iterations is zero. When the number of smoothing iterations is zero, the nodes at the parent face will be perturbed.

Relaxation Factor

sets the value of under relaxation for smoothing the parent face.

Define Hexcore Extents

allows you to extend the hexcore mesh to specified domain extents and/or selected planar boundaries.

Specify...

opens the **Outer Domain Parameters** dialog box. This option is active only when the **Define Hexcore Extents** option is enabled.

Refinement Regions

contains options for defining and manipulating refinement regions.

Active

indicates the number of active refinement regions.

Local Regions...

opens the **Hexcore Refinement Region** dialog box.

Tet Controls...

opens the **Tet** dialog box, where you can set the parameters for tetrahedral meshing.

Zones

contains parameters related to the zones created during hexcore meshing.

Delete Dead Zones

toggles the deleting of the dead zones.

Non-Fluid Type

selects the default non-fluid cell zone type. After the mesh is initialized, any non-fluid zones will be set to the specified type. By default, the non-fluid type is set to **dead**, as the grid is considered to comprise a single fluid region and one or more dead regions. The active zone is considered to be the fluid zone and only this fluid zone will be considered for refinement during the automatic meshing process.

If the mesh includes multiple regions (e.g., the problem for which you are creating a mesh includes a fluid zone and one or more solid zones), and you intend refining all of them using the same refinement parameters, modify the **Non-Fluid Type** before generating the hexcore mesh. When the **Non-Fluid Type** is set to a type other than **dead** (e.g., **solid**), all the zones will be active after the initialization is complete. Hence, all zones will be refined automatically during the refinement process.

Note

- For sewn mesh objects, the **Non-Fluid Type** field value will be applied to dead zones which are not claimed by solid/fluid objects and material points.

- For mesh objects created using **Build Topology**, the **Non-Fluid Type** field value will not be respected while using **Auto Mesh**.

Keep Hex Tet Separate

toggles the merging of the Cartesian cells with the tetrahedral (and wedge) cells at the end of the mesh generation.

Note

This option will be overridden by the **Merge Cell Zones** option in the **Auto Mesh** dialog box when the Auto Mesh procedure is used.

Create

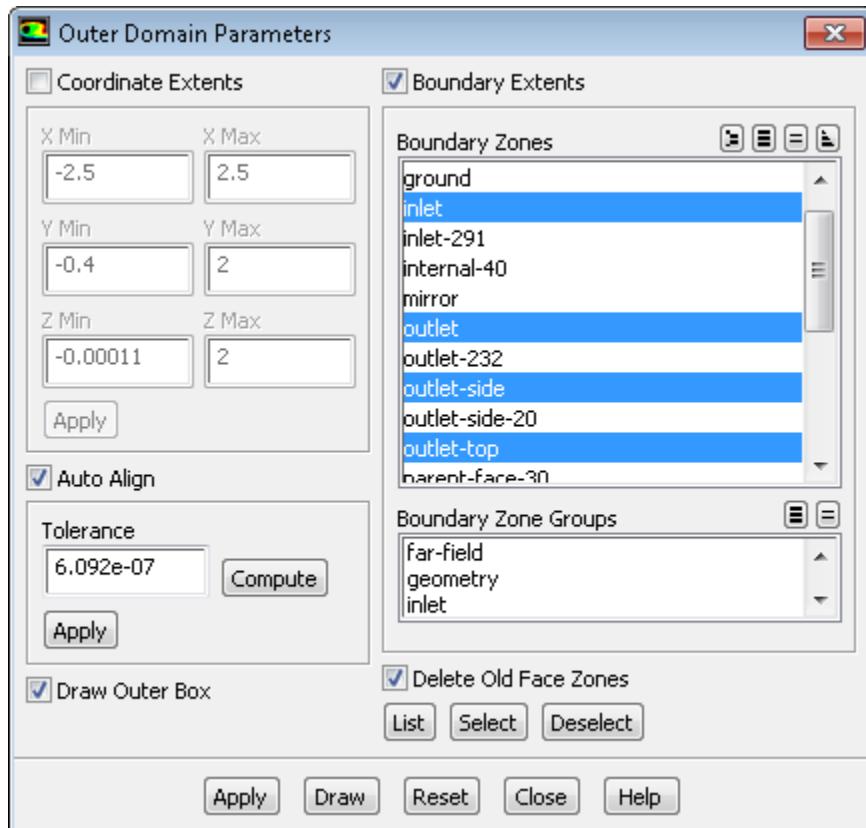
creates the hexcore mesh according to the parameters specified in the **Hexcore** dialog box.

Apply

applies the parameters specified in the **Hexcore** dialog box.

21.4.7.2. Outer Domain Parameters Dialog Box

The **Outer Domain Parameters** dialog box allows you to specify the outer domain parameters.



Controls

Coordinate Extents

allows you to specify the outer domain extents.

X Min, Y Min, Z Min, X Max, Y Max, Z Max

specify the extents of the domain.

Apply

sets the specified domain extents.

Boundary Extents

allows you to specify selected boundaries to which the hexcore mesh is to be generated.

Boundary Zones

contains a list of the boundary zone(s) available.

Boundary Zone Groups

contains a list of the default boundary zone groups and user-defined groups available. If you select a zone group from this list, all zones of that group will be selected in the **Boundary Zones** list. You can also select multiple types to select all the zones of different types (e.g., **inlet** and **outlet**).

Delete Old Face Zones

allows you to delete the original tri boundary face zones which have been replaced by a mix of quad and tri face zones during the hexcore meshing process. If this option is disabled, the original tri face zones will be retained with "**:old**" appended to the name.

List

lists the selected boundaries to which the hexcore mesh is to be generated.

Select

selects the boundaries to which the hexcore mesh is to be generated in the **Boundary Zones** list.

Deselect

deselects the boundaries to which the hexcore mesh is to be generated in the **Boundary Zones** list.

Note

You need to select the boundary zones to which the hexcore mesh is to be generated and click **Apply** in the **Outer Domain Parameters** dialog box before clicking the **List**, **Select**, and **Deselect** buttons.

Auto Align

allows you to axis-align slightly misaligned boundary zones to which hexcore mesh is to be generated.

Tolerance

specifies the tolerance for aligning the boundary zones.

Compute

computes the recommended tolerance for aligning the boundary zones.

Apply

aligns the selected zones based on the tolerance specified.

Draw Outer Box

allows you to draw the defined outer box.

Apply

applies the parameters specified in the **Outer Domain Parameters** dialog box.

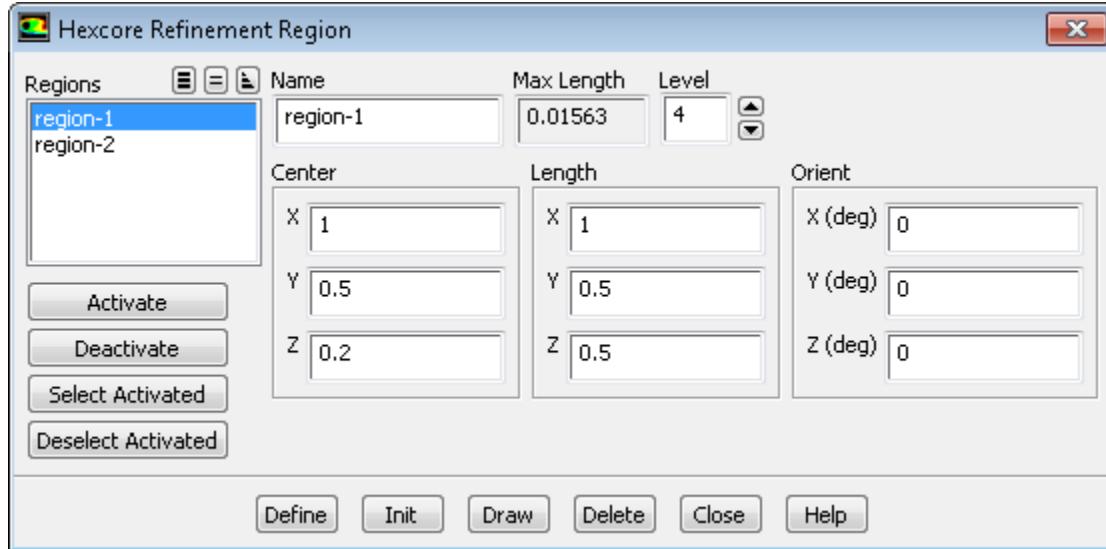
Draw

draws the selected boundary zones, and the outer box if the **Draw Outer Box** option is enabled.

Reset

resets the specified extents and selected boundaries to the default.

21.4.7.3. Hexcore Refinement Region Dialog Box



The **Hexcore Refinement Region** dialog box provides options for creating, modifying, and activating/deactivating a local region for hexcore mesh refinement.

Controls**Regions**

contains a list of the defined regions.

Name

specifies the name of the region selected in the **Regions** list, or the name of the region to be created.

Max Length

specifies the maximum cell length in the region. The **Max Length** value is determined by the specified refinement level.

Level

specifies the refinement level for the local region defined.

Center

allows you to specify the coordinates of the center of the region you want to create.

Length

allows you to specify the absolute size of the new region in the x, y, and z directions.

Orient

allows you to specify the orientation of the new region.

Note

The region is oriented by rotation first about the x-axis, then the y-axis, and finally the z-axis. You need to take this into account while specifying the orientation of the region as rotation in any other order will produce different results.

Activate

activates the region(s) selected in the **Regions** list.

Deactivate

deactivates the region(s) selected in the **Regions** list.

Select Activated

selects the activated region(s) in the **Regions** list.

Deselect Activated

deselects the activated region(s) in the **Regions** list.

Define

creates a new region according to the specified parameters. It also allows you to modify the selected region according to the specified changes.

Init

creates a default region encompassing the entire geometry.

Draw

draws the region(s) in the graphics window. The maximum cell size for the refinement regions will also be displayed. If the boundary grid was displayed (using the **Display Grid** dialog box) before drawing the refinement region, the grid will also be included in the display.

Delete

deletes the selected region.

21.4.8. Mesh/CutCell...

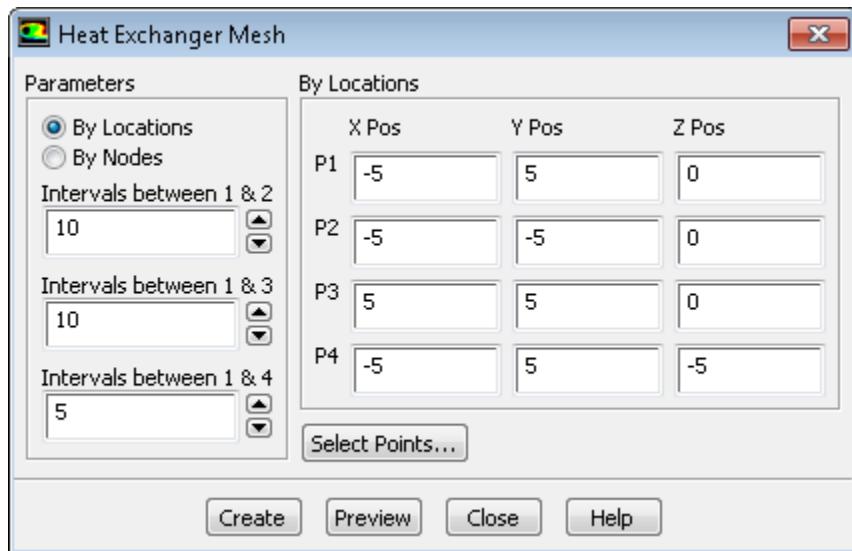
The **Mesh/CutCell...** menu item opens the [CutCell Dialog Box \(p. 466\)](#).

21.4.9. Mesh/Create/Heat Exchanger...

The **Mesh/Create/Heat Exchanger...** menu item opens the [Heat Exchanger Mesh Dialog Box \(p. 569\)](#).

21.4.9.1. Heat Exchanger Mesh Dialog Box

The **Heat Exchanger Mesh** dialog box allows you to specify the parameters required to create the heat exchanger mesh.



Controls

Parameters

contains inputs to be specified for heat exchanger mesh creation.

By Locations

allows you to specify the locations (**X Pos**, **Y Pos**, **Z Pos**) of the four points **P1**, **P2**, **P3**, and **P4**. You can either enter the position values or select the points using the mouse.

By Nodes

allows you to specify the nodes to be used. You can either enter the **Node ID** or select the points using the mouse.

Intervals between 1 & 2

specifies the number of intervals between points 1 and 2.

Intervals between 1 & 3

specifies the number of intervals between points 1 and 3.

Intervals between 1 & 4

specifies the number of intervals between points 1 and 4.

Select Points...

allows you to select the points (by selecting the locations or nodes) using the mouse. When you click this button, a **Working** dialog box will open, prompting you to select 3 points to define the plane, and the 4th point to grow prisms.

Create

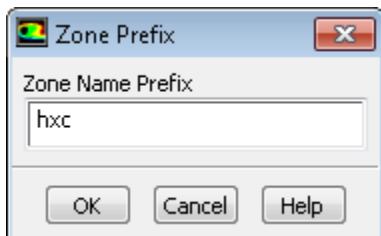
opens the **Zone Prefix** dialog box, where you can specify the prefix for the heat exchanger zones to be created.

Preview

allows you to preview the heat exchanger to be created.

21.4.9.2. Zone Prefix Dialog Box

The **Zone Prefix** dialog box allows you to specify the prefix for the zones created (e.g., **hxc**) using the Heat Exchanger Mesh Dialog Box (p. 569).



Controls

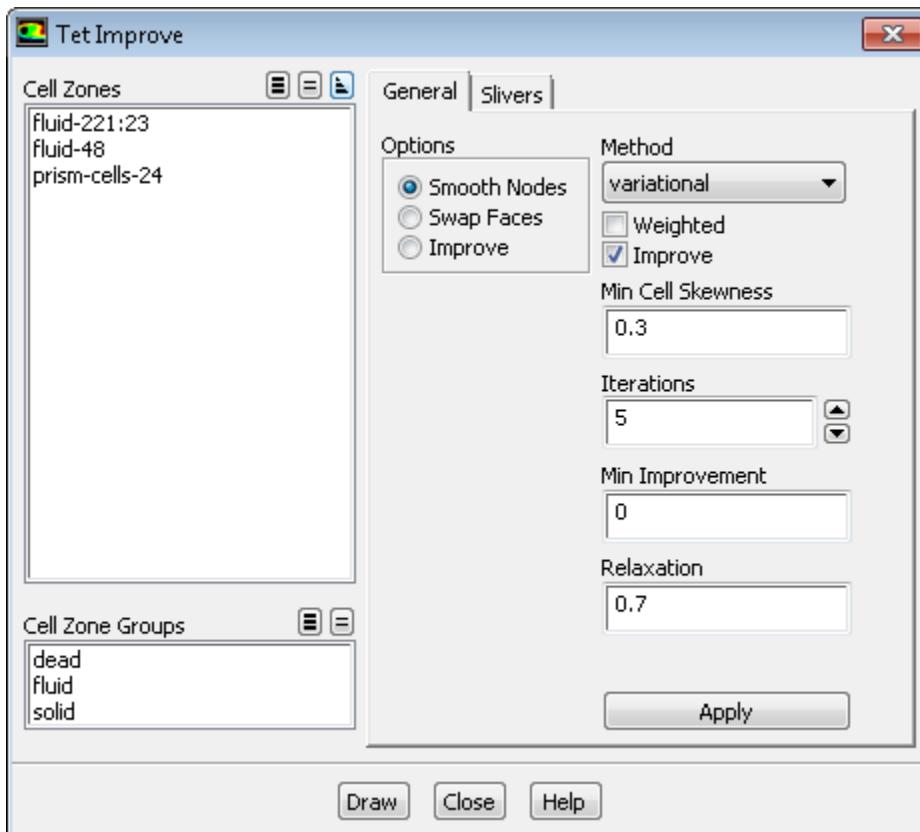
Zone Name Prefix

specifies the prefix to be used for the zones being created.

21.4.10. Mesh/Tools/Tet Improve...

The **Mesh/Tools/Tet Improve...** menu item opens the **Tet Improve Dialog Box** (p. 571).

21.4.10.1. Tet Improve Dialog Box



Controls

Cell Zones

contains a list from which you can select the cell zones to be improved.

Cell Zone Groups

contains a list of the cell zone types. If you select a cell type from the list, (e.g., **fluid**), all cell zones of that type will be selected in the **Cell Zones** list.

General

contains options for smoothing nodes, swapping faces, and improving cells.

Options

contains a list of the options available for improving the mesh.

Smooth Nodes

smooths the nodes to improve the mesh quality.

Method

contains a list of the smoothing methods available.

Weighted

implies that the cell size (as given by the cell size function) at the neighboring nodes (for the Laplace method) or the circumcenter (for the variational method) is used to weight the influence of each node (or circumcenter) when computing the average node location. This option is available only when **variational** or **laplace** is selected in the **Method** drop-down list.

Improve

implies that the node will be repositioned only when the maximum skewness of the cells connected to the node is reduced. This option is available only when **variational** or **laplace** is selected in the **Method** drop-down list.

Min Cell Skewness

specifies the skewness above which the nodes will be smoothed.

Iterations

specifies the number of smoothing iterations.

Min Improvement

specifies the convergence criterion for smoothing. For skewness based smoothing, smoothing will be stopped when the largest change in skewness is below the limit specified. For Laplace and variational smoothing, the convergence criterion is the root mean square of the distance the nodes were moved during smoothing.

Relaxation

specifies the factor used for computing the smoothing distance. The value varies between 0 and 1. A value of 0 results in no movement, while a value of 1 results in movement equivalent to the computed distance. A smaller value will result in a more gradual change in the mesh. A reduced value may also result in a higher quality mesh, but at the expense of more smoothing iterations. This option is available only when **variational** or **laplace** is selected in the **Method** drop-down list.

Swap Faces

swaps the faces of the cells to improve the mesh quality.

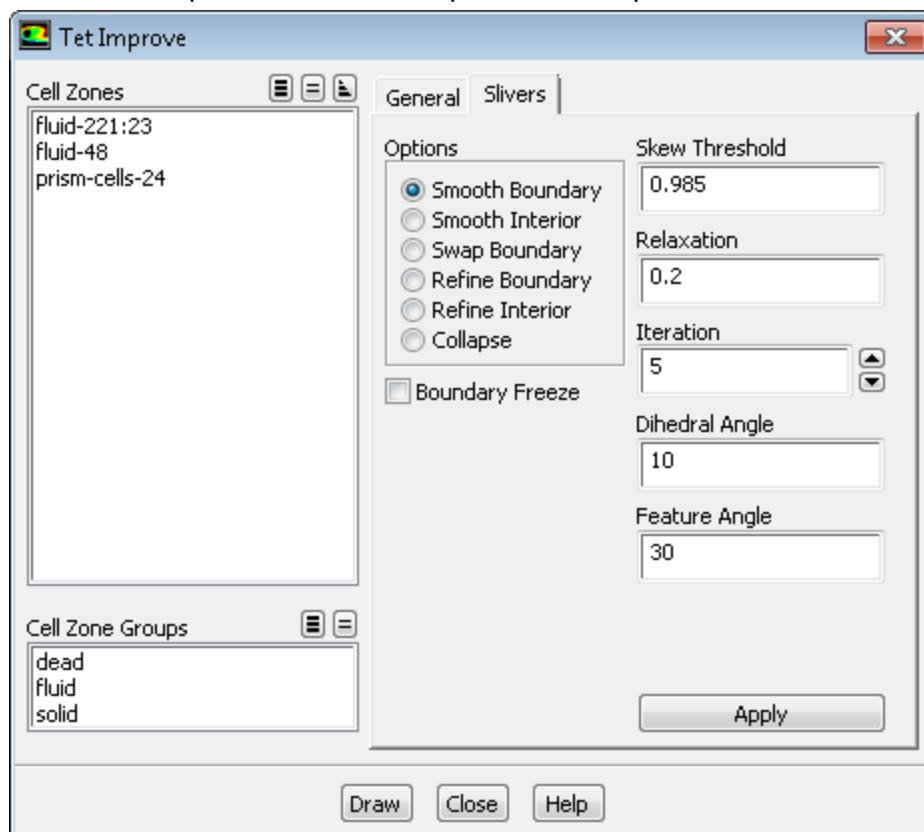
Swap Configuration

contains a list of the possible configurations for swapping.

- Skewness**
specifies the skewness above which cells will be considered for face swapping.
- Iterations**
specifies the number of swapping iterations.
- Improve**
improves the mesh quality based on the specified parameters.
- Skew Threshold**
specifies the skewness threshold for improving the mesh quality.
- Dihedral Angle**
specifies the threshold dihedral angle. If the dihedral angle between two boundary faces is below the specified value, the cell is considered to be a valid boundary sliver (which cannot be improved).
- Max Attempts**
specifies the number of improvement attempts.
- Apply**
performs the selected operation.

Slivers

contains the options available and parameters required for sliver removal.



Options

contains a list of the operations available for sliver removal.

Smooth Boundary

smooths nodes on sliver cells having any node on the boundary such that skewness of the surrounding cells is improved.

Smooth Interior

smooths non-boundary nodes on sliver cells having skewness greater than the specified threshold value.

Swap Boundary

removes boundary slivers by moving the boundary to exclude the cells from the zone.

Refine Boundary

refines the boundary slivers by edge splitting.

Refine Interior

removes the sliver by placing a node at or near the centroid of the sliver cell.

Collapse

attempts to collapse the nodes of a skewed sliver cell on any one of its neighbors.

Skew Threshold

specifies the skewness threshold above which sliver cells will be removed.

Relaxation

specifies the relaxation factor for smoothing boundary and interior slivers.

Iteration

specifies the number of iterations for the smoothing of boundary and interior slivers.

Min Skewness

specifies the skewness above which the boundary slivers will be swapped.

Dihedral Angle

specifies the threshold dihedral angle. If the dihedral angle between two boundary faces is below the specified value, the cell is considered to be a valid boundary sliver (which cannot be improved).

Feature Angle

specifies the angle defining the features for smoothing of boundary slivers.

Boundary Freeze

allows you to fix the boundary during the sliver removal operations.

Apply

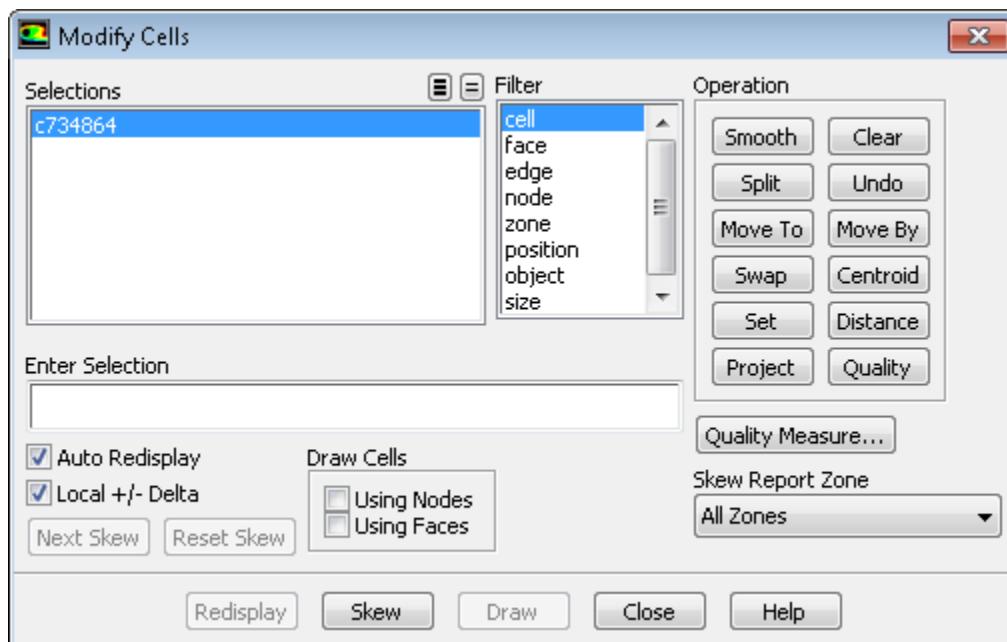
performs the selected operation.

21.4.11. Mesh/Tools/Cell Modify...

The **Mesh/Tools/Cell Modify...** menu item opens the [Modify Cells Dialog Box \(p. 574\)](#).

21.4.11.1. Modify Cells Dialog Box

The **Modify Cells** dialog box allows you to perform operation such as smoothing nodes, moving nodes, swapping cells, etc.



Controls

Selections

contains a list of the selected entities. You need to ensure the appropriate entities are selected in the **Selections** list before clicking the required button in the **Operation** group box.

Filter

lists the types of entities that can be selected. Only entities of the type selected in the **Filter** list can be selected. To select entities of more than one type, select the appropriate option in the **Filter** list and select the required entities of the same type. Then, change the selection in the **Filter** list and continue to select the entities as required.

Operation

contains buttons for performing cell modification operations.

Smooth

repositions the selected node based on the average of the surrounding nodes.

Clear

removes the selected entities from the **Selections** list.

Split

refines the selected cell by adding a node at the centroid and splitting the cell into four cells.

Undo

undoes the previous operation.

Move To

moves the selected node to the specified position.

Move By

moves the selected node by the specified increment.

Swap

swaps the cell configuration (3-2 or 2-3) according to the entities selected.

Centroid

reports the centroid for the entity selected in the **Selections** list.

Set

defines the projection line or plane determined by the entities selected in the **Selections** list.

Distance

reports the distance between the two entities selected in the **Selections** list.

Project

projects the selected node(s) onto to the defined projection line or plane.

Quality

reports the quality of the selected entity according to the **Measure** selected in the **Quality Measure** dialog box.

Enter Selection

allows you to enter the name of an entity or the coordinates of a particular position as an alternative to selecting the entity from the graphics window.

Auto Redisplay

toggles the automatic update of display after a cell modification operation has been performed. When **Auto Redisplay** is enabled, the display will be automatically updated, allowing you to see the change effected by the operation.

Local +/- Delta

toggles the limiting of the grid display to a neighborhood around the selected entity. When **Local +/- Delta** is enabled, the selected entity and some faces around it will be displayed.

Draw Cells

contains options for drawing cells.

Using Nodes

enables the display of cells comprising the selected node(s).

Using Faces

enables the display of cells comprising the selected face(s).

Quality Measure...

opens the **Quality Measure** dialog box (see [Quality Measure Dialog Box \(p. 638\)](#)).

Skew Report Zone

contains a drop-down list of the cell zones for which you can locate skewed cells. This allows you to select the cell zone of interest instead of locating the skewed cells for all zones.

Next Skew

displays the cell having the next highest skewness value after the currently displayed cell. Clicking the **Next Skew** button repeatedly will display the skewed cells in the descending order of skewness. The **Next Skew** button will be activated only after the **Skew** button has been clicked in the **Modify Cells** dialog box.

Reset Skew

resets the skewness values. When you click **Reset Skew**, both the **Next Skew** and **Reset Skew** buttons will be deactivated. Click **Skew** and then **Next Skew** to display the skewed cells again.

Redisplay

updates the display in the display window. This button is activated only when the **Auto Redisplay** option is disabled.

Important

You can disable **Auto Redisplay** for large meshes or when the graphics performance of your computer is slow. You can then use the **Redisplay** button to view the updated display.

Skew

displays the cell with the worst (highest) skewness. The cell will also be added to the **Selections** list.

Draw

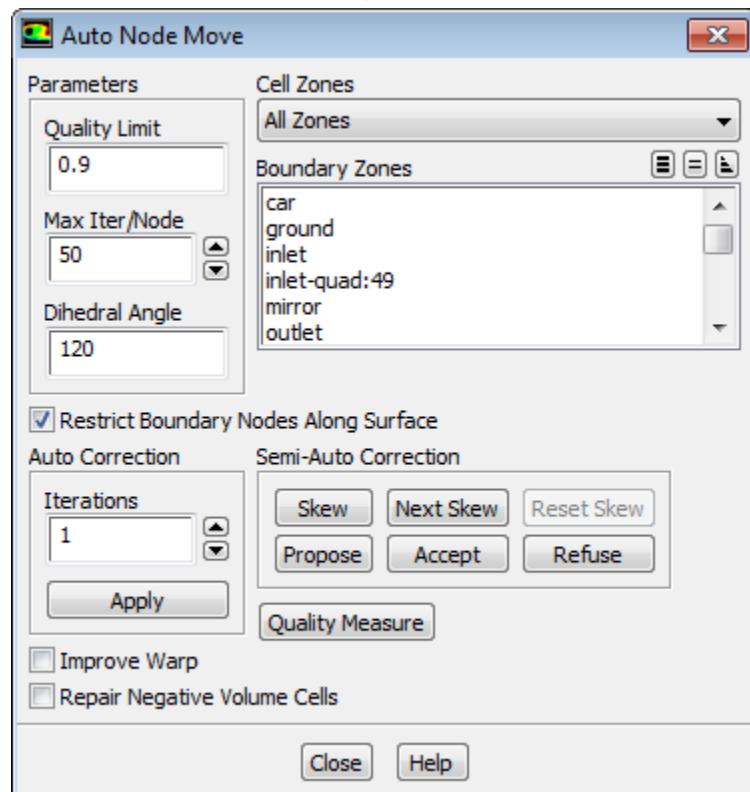
displays the cells connected to the selected node(s) or face(s), according to the options enabled in the **Draw Cells** group box.

21.4.12. Mesh/Tools/Auto Node Move...

The **Mesh/Tools/Auto Node Move...** menu item opens the [Auto Node Move Dialog Box \(p. 577\)](#).

21.4.12.1. Auto Node Move Dialog Box

The **Auto Node Move** dialog box contains options for moving nodes to improve the mesh quality.

**Controls****Parameters**

contains the parameters for node movement.

Quality Limit

specifies the quality limit for the cells. All cells above the specified limit will be considered for improvement.

Max Warp

specifies the maximum allowable warp for quadrilateral faces. All quad cells having faces with warp greater than this value will be considered for improvement. This option is available only when the **Improve Warp** option is enabled.

Max Iter/Node

specifies the number of attempts to improve the skewness by moving a particular node.

Max Iter/Face

specifies the number of attempts to improve the warp of a face. This option is available only when the **Improve Warp** option is enabled.

Dihedral Angle

specifies the feature angle to be considered when moving boundary nodes and allows you to maintain features of the geometry. A boundary node will not be moved if the angle between the faces sharing the boundary node is less than the specified value.

Cell Zones

contains a drop-down list of the cell zones from which you can select those to be considered for improvement.

Boundary Zones

contains a list of the boundary zones from which you can select those to be considered for improvement.

Restrict Boundary Nodes Along Surface

allows you to restrict the movement of boundary nodes within the plane containing the faces sharing the node. This option is available only for the quality-based improvement.

Auto Correction

contains options for using the automatic correction procedure for improvement.

Iterations

specifies the number of repetitions through the automatic correction procedure.

Apply

performs the automatic correction procedure according to the number of iterations specified.

Semi-Auto Correction

contains options for using the semiautomatic correction procedure for quality-based improvement.

Skew

displays the cell with the worst quality and cells/faces within a pre-defined range of the cell.

Next Skew

displays the cell with the next worst quality.

Reset Skew

allows you to reset the quality values.

Propose

highlights the node to be moved and the alternative node position. The improvement in the quality will also be reported in the console.

Accept

allows you to accept the proposed node movement.

Refuse

allows you to reject the proposed node movement.

Quality Measure

opens the **Quality Measure** dialog box where you can select the quality measure for the quality-based improvement.

Improve Warp

allows you to use the automatic correction procedure for warp-based improvement.

Repair Negative Volume Cells

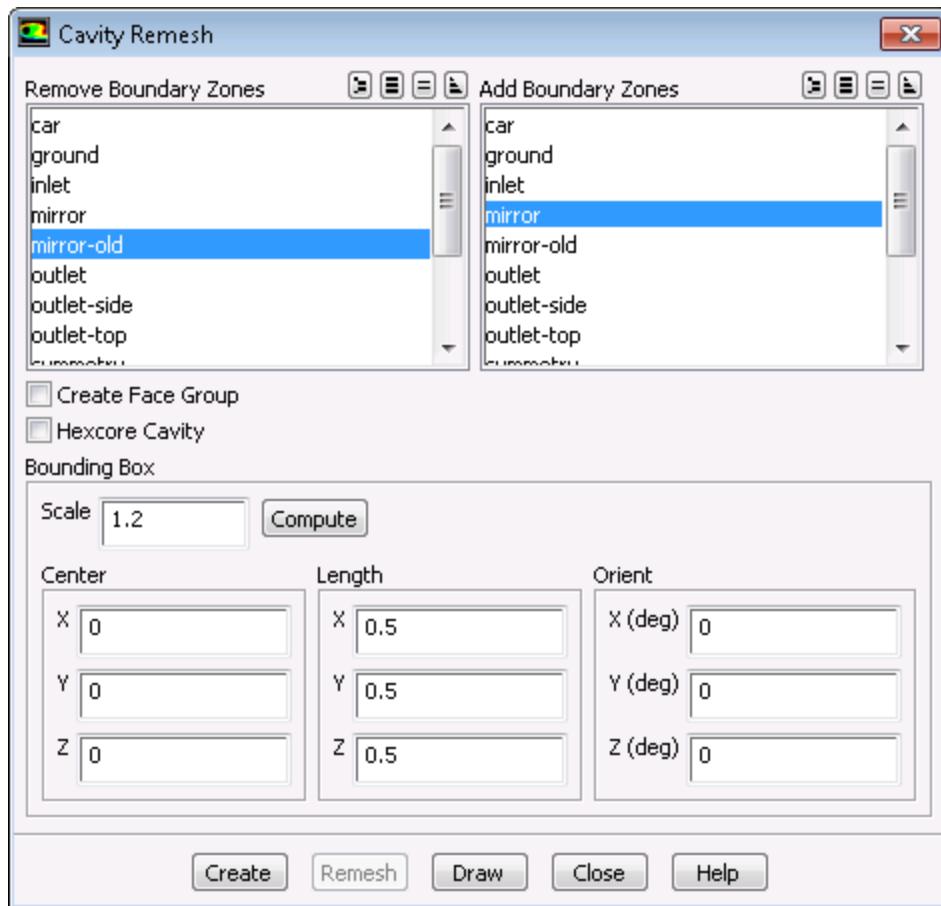
allows you to repair negative volume cells when using the automatic correction procedure for quality-based improvement.

21.4.13. Mesh/Tools/Cavity Remesh...

The **Mesh/Tools/Cavity Remesh...** menu item opens the [Cavity Remesh Dialog Box \(p. 579\)](#).

21.4.13.1. Cavity Remesh Dialog Box

The **Cavity Remesh** dialog box contains options for creating a cavity in an existing volume mesh.



Controls

Remove Boundary Zones

contains a selection list of available boundary zones from which you can select the zones to be removed from the volume mesh.

Add Boundary Zones

contains a selection list of available boundary zones from which you can select the zones to be added to the volume mesh.

Create Face Group

allows you to create a UDG comprising the zone(s) defining the cavity domain. This option is disabled when the **Hexcore Cavity** option is enabled.

Hexcore Cavity

allows you to create the cavity and remesh it using hexcore mesh.

Bounding Box

defines the extents of the cavity to be created.

Scale

specifies the scale factor to be applied while calculating the bounding box for the zones selected in the **Remove Boundary Zones** and **Add Boundary Zones** selection lists.

Compute

computes the bounding box extents according to the zones selected in the **Remove Boundary Zones** and **Add Boundary Zones** selection lists, and the scale factor specified.

Center

specifies the coordinates of the center of the cavity.

Length

specifies the length of the cavity bounding box along the **X**, **Y**, and **Z** axes.

Orient

specifies the orientation of the cavity bounding box about the center of the cavity. This option is disabled when the **Hexcore Cavity** option is enabled.

Create

creates a cavity according to the bounding box defined.

Remesh

remeshes the cavity with a hexcore mesh. This option is available only when the **Hexcore Cavity** option is enabled and is activated only after the cavity has been created.

Note

If required, you need to manually create the prism layers before remeshing the cavity.

Note

If any active hexcore refinement regions are present, they will also be considered during the cavity remeshing process.

Draw

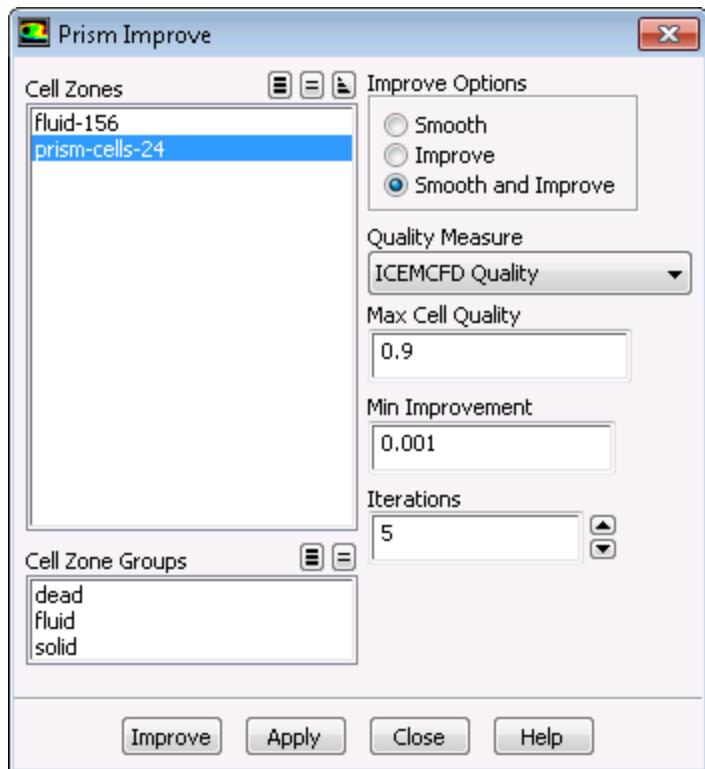
allows you to preview the cavity to be created.

21.4.14. Mesh/Tools/Prism/Improve...

The **Mesh/Tools/Prism/Improve...** menu item opens the Prism Improve Dialog Box (p. 581).

21.4.14.1. Prism Improve Dialog Box

The **Prism Improve** dialog box contains options that allow you to improve the prism cell quality.



Controls

Cell Zones

contains a list of cell zones from which you can select the prism cell zone(s) to be improved.

Cell Zone Groups

contains a list of cell zone types. If you select a cell type from this list (e.g., **fluid**), all cell zones of that type will be selected in the **Cell Zones** list. This method allows you to easily select all cell zones of a certain type without having to select each zone individually. You can select multiple cell types in the **Cell Zone Groups** list to select all zones of several different types (e.g., **solid** and **fluid**).

Improve Options

contains a list of improve operations available for improving the prism mesh quality.

Smooth

allows optimization based smoothing of prism cells. Poor quality cells can be identified based on the ICEM CFD quality, ortho skew, skewness, or squish measures. The nodes of cells with quality worse than the specified **Max Cell Quality** value will be moved to improve quality. The cell aspect ratio will also be maintained based on the value specified for `max-aspect-ratio`.

Improve

collects and smooths cells in layers around poor quality cells. Poor quality cells can be identified based on the ICEM CFD quality, ortho skew, skewness, or squish measures. Cells with quality worse than the specified **Max Cell Quality** value will be identified, and the nodes of the cells surrounding the poor quality cells will be moved to improve quality. The cell aspect ratio will also be maintained based on the value specified for `max-aspect-ratio`.

Smooth and Improve

uses a combination of node movement and optimized smoothing to improve the quality. This option is a combination of the **Smooth** and **Improve** options. The cell aspect ratio will also be maintained based on the value specified for `max-aspect-ratio`.

Quality Measure

specifies the quality measure based on which the prism cells will be improved. The quality measures available include ICEM CFD quality, ortho skew, skewness, and squish.

Max Cell Quality

specifies the quality threshold for improving the prism cell quality. Cells with quality worse than the specified **Max Cell Quality** value will be identified and improved.

Min Improvement

specifies the minimum improvement required for the prism cell quality improvement operation.

Iterations

specifies the number of improvement iterations to be performed.

Improve

improves the prism mesh quality based on the parameters specified.

Apply

sets the parameters for improving the prism mesh quality.

21.4.15. Mesh/Tools/Prism/Split...

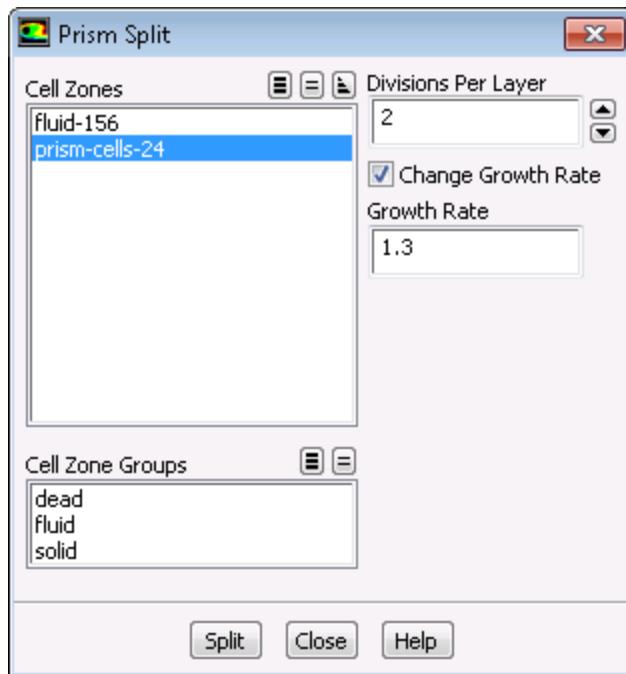
The **Mesh/Tools/Prism/ Split...** menu item opens the [Prism Split Dialog Box \(p. 583\)](#).

21.4.15.1. Prism Split Dialog Box

The **Prism Split** dialog box contains options that allow you to split the prism cells layers based on the number of layer splits specified.

Note

Prism Split will recompute all existing prism layer heights, including the first layer height, based on the **Divisions Per Layer** and **Change Growth Rate** specified.



Controls

Cell Zones

contains a list of cell zones from which you can select the prism cell zone(s) for the split operation.

Cell Zone Groups

contains a list of cell zone types. If you select a cell type from this list (e.g., **fluid**), all cell zones of that type will be selected in the **Cell Zones** list. This method allows you to easily select all cell zones of a certain type without having to select each zone individually. You can select multiple cell types in the **Cell Zone Groups** list to select all zones of several different types (e.g., **solid** and **fluid**).

Divisions Per Layer

specifies the number of divisions for each prism layer.

Change Growth Rate

allows you to change the prism growth rate for the prism layer split operation. This option is disabled by default and the existing layer growth rate will be used. When enabled, you can specify the growth rate to be used.

Growth Rate

specifies the growth rate to be used for the prism layer split operation when **Change Growth Rate** is enabled.

Split

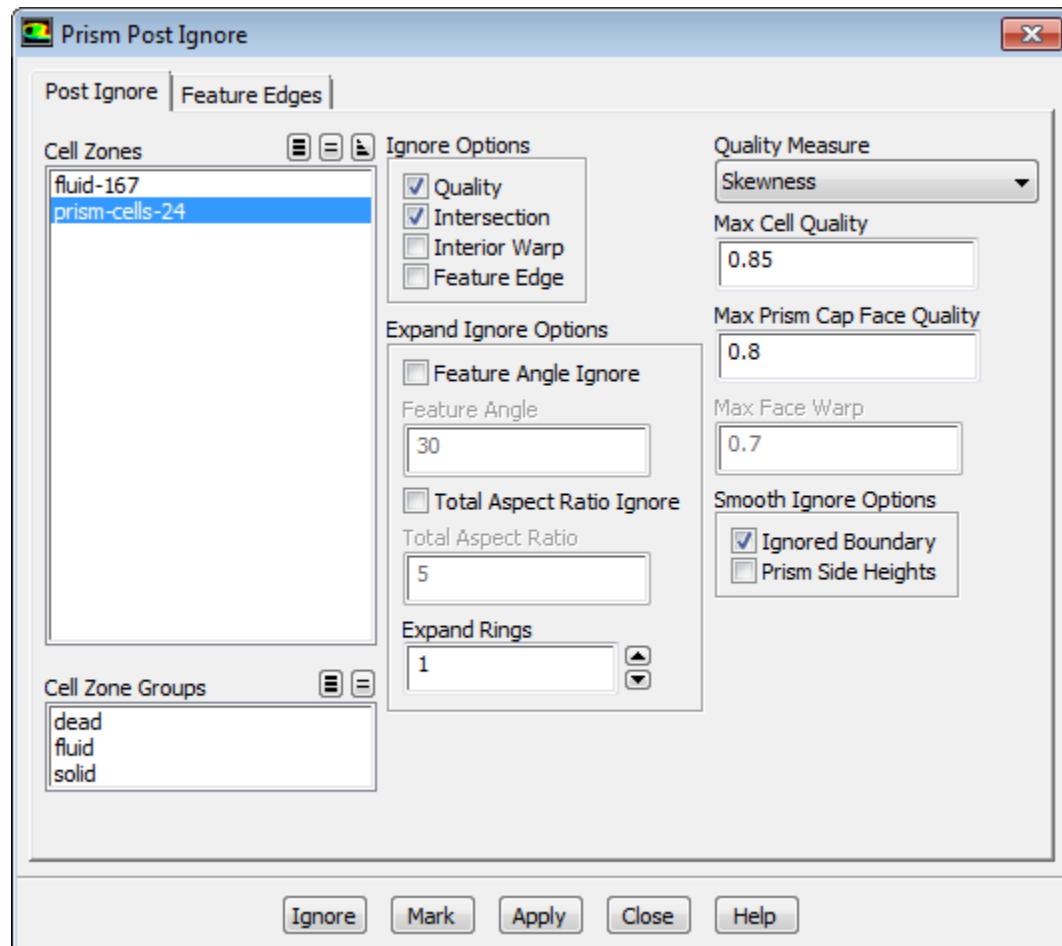
splits the selected prism cell layers based on the parameters specified.

21.4.16. Mesh/Tools/Prism/Post Ignore...

The **Mesh/Tools/Prism/Post Ignore...** menu item opens the [Prism Post Ignore Dialog Box \(p. 585\)](#).

21.4.16.1. Prism Post Ignore Dialog Box

The **Prism Post Ignore** dialog box contains options that allow you to remove prism cells in layers around poor quality cells and sharp corners.



Controls

Post Ignore

contains options controlling the removal of prism cells.

Cell Zones

contains a list of cell zones from which you can select the prism cell zone(s) for the post-ignore operation.

Cell Zone Groups

contains a list of cell zone types. If you select a cell type from this list (e.g., **fluid**), all cell zones of that type will be selected in the **Cell Zones** list. This method allows you to easily select all cell zones of a certain type without having to select each zone individually. You can select multiple cell types in the **Cell Zone Groups** list to select all zones of several different types (e.g., **solid** and **fluid**).

Ignore Options

contains options based on which prism cells will be removed.

Quality

allows you to ignore prism cells based on quality. This option is enabled by default.

Intersection

allows you to ignore prism cells based on intersection. This option is enabled by default.

Interior Warp

allows you to ignore prism cells based on cell warp. This option is disabled by default.

Feature Edge

allows you to ignore prism cells based on feature edges. This option is disabled by default.

Note

This option requires a prism-cap zone to be present so it cannot be used as a **Post Operation** control during prism generation.

Expand Ignore Options

contains options for removing rings of additional cells in regions based on feature angle or high aspect ratio.

Feature Angle Ignore

allows you to ignore additional prism cells for removal in regions of sharp angles around the exposed prism side.

Feature Angle

specifies the feature angle for ignoring additional prism cells.

Total Aspect Ratio Ignore

allows you to ignore additional prism cells for removal in regions of high aspect ratio around the exposed prism side.

Total Aspect Ratio

specifies the maximum total aspect ratio for ignoring additional prism cells.

Expand Rings

specifies the number of additional cell rings to be removed around the originally marked prism cells.

Quality Measure

allows you to select the quality measure based on which prism cells will be ignored. The quality measures available are skewness, ICEM CFD quality, ortho skew, and squish.

Max Cell Quality

specifies the maximum allowable cell quality based on the quality measure selected.

Max Prism Cap Face Quality

specifies the maximum allowable prism cap face quality.

Max Face Warp

specifies the maximum allowable warp when ignoring prism cells based on warp.

Smooth Ignore Options

contains options for smoothing the boundary zone(s) after the prism cells have been removed.

Ignored Boundary

allows you to smooth the boundary at feature corners after the prism cells have been removed

Prism Side Heights

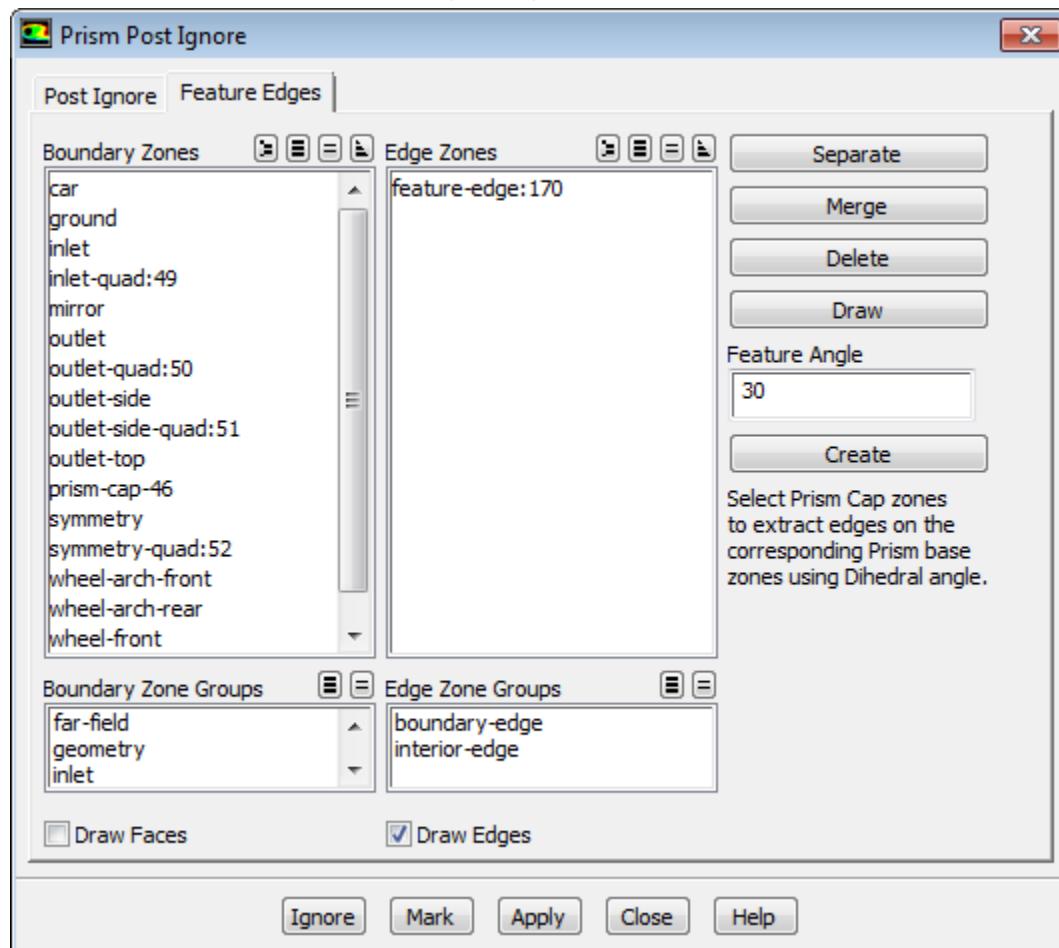
allows you to smooth the prism side nodes from the base node to the cap node to create better triangles for the non-conformal interface.

Ignore

removes the prism cells based on the options selected.

Mark

marks the prism cap faces for ignoring prism cells based on the options selected.



Feature Edges

contains options for manipulating feature edges to be used for the post-ignore operation.

Boundary Zones

contains a list of boundary zones from which you can select the prism cap zone to extract edge zones on the corresponding prism base zones based on the feature angle specified.

Boundary Zone Groups

contains a list of boundary zone types. If you select a zone type from this list (e.g., **boundary**), all face zones of that type will be selected in the **Face Zones** list. This shortcut allows you to easily select all zones of a certain type without having to select each zone individually.

Edge Zones

contains a list of edge zones that have been created for one or more boundary zones. Select the edge zone(s) extracted from the prism base zones based on the feature angle when using the **Feature Edge** option in the **Post-Ignore** tab.

Edge Zone Groups

contains a list of edge zone types. If you select a zone type from this list (e.g., **boundary-edge**), all edge zones of that type will be selected in the **Edge Zones** list. This shortcut allows you to easily select all zones of a certain type without having to select each zone individually.

Separate

separates the selected edge zones based on connectivity and the specified **Feature Angle**.

Merge

merges the selected edge zones into a single edge zone.

Delete

deletes the selected edge zones.

Draw

displays the selected face and/or edge zones based on the **Draw Faces** and **Draw Edges** options.

Feature Angle

specifies the angle based on which edge zones are extracted from the prism base zones, when the prism cap zone is selected in the **Boundary Zones** list. You can also specify the feature angle for separating the edge zones using the **Separate** option.

Create

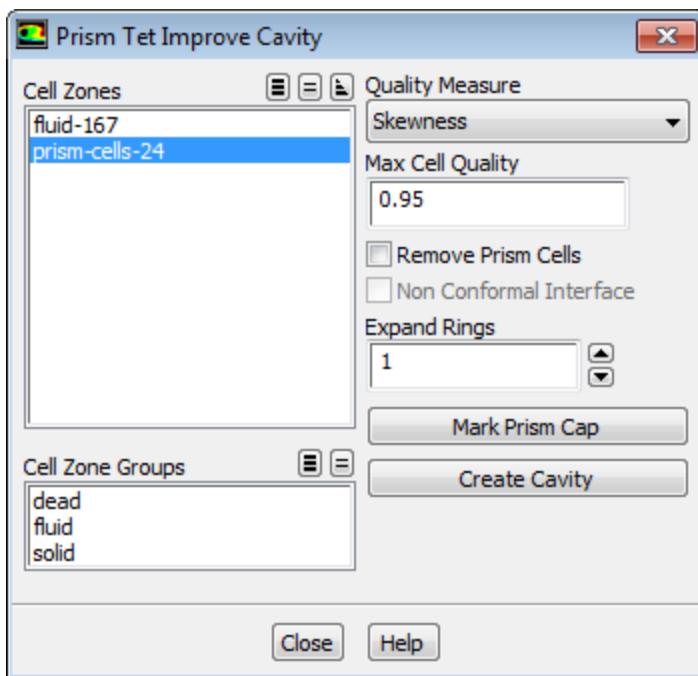
extracts edge zones from the prism base zones when the prism cap zone is selected in the **Boundary Zones** list. The edge zones are extracted based on the **Feature Angle** specified.

21.4.17. Mesh/Tools/Prism/Tet Improve Cavity

The **Mesh/Tools/Prism/Tet Improve Cavity...** menu item opens the [Prism Tet Improve Cavity Dialog Box](#) (p. 588).

21.4.17.1. Prism Tet Improve Cavity Dialog Box

The **Prism Tet Improve Cavity** dialog box contains options that allow you to creates a cavity in regions where prism quality is adequate, but the quality of adjacent tetrahedra is poor.



Controls

Cell Zones

contains a list of cell zones from which you can select the tetrahedral cell zone(s) for creating the cavity.

Cell Zone Groups

contains a list of cell zone types. If you select a cell type from this list (e.g., **fluid**), all cell zones of that type will be selected in the **Cell Zones** list. This method allows you to easily select all cell zones of a certain type without having to select each zone individually. You can select multiple cell types in the **Cell Zone Groups** list to select all zones of several different types (e.g., **solid** and **fluid**).

Quality Measure

allows you to select the quality measure based on which tetrahedral cells will be removed. The methods available are skewness, ICEM CFD quality, ortho skew, and squish.

Max Cell Quality

specifies the maximum allowable cell quality based on the quality measure selected.

Remove Prism Cells

allows you to include prism cells along with tetrahedral cells in the cavity created.

Non-Conformal Interface

allows you to specify whether the non-conformal interface is to be created when prism cells are also included in the cavity.

Expand Rings

specifies the number of additional cell rings to be removed around the originally marked prism cells.

Mark Prism Cap

marks the prism cap faces under the tetrahedral cells bounding the cavity to be created based on the options selected.

Create Cavity

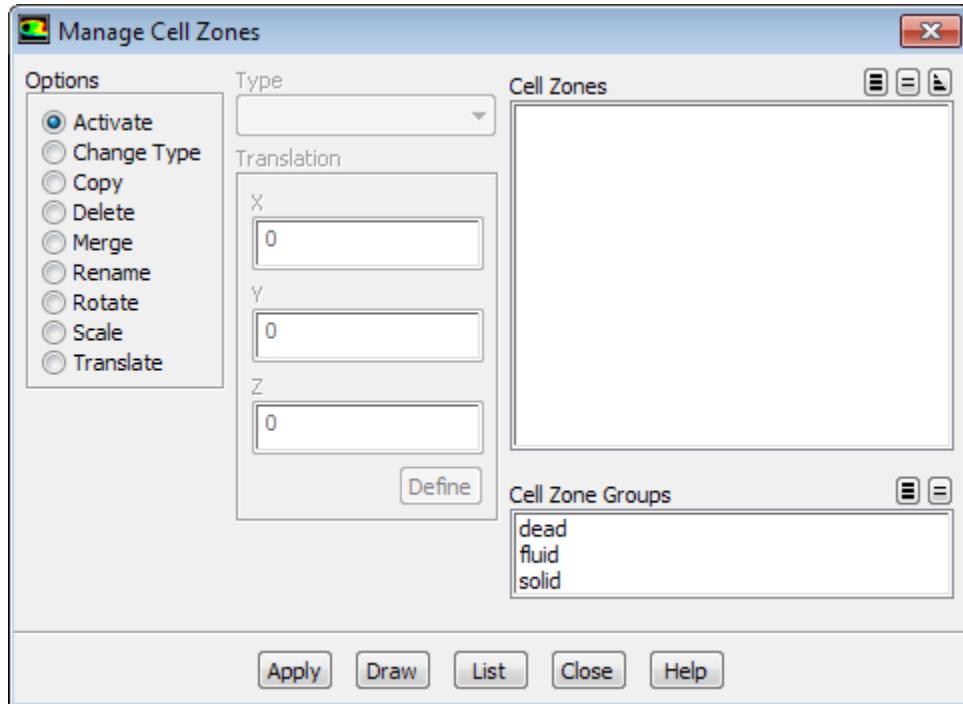
creates a cavity comprising only tetrahedral cells or comprising tetrahedral and prism cells based on the options selected.

21.4.18. Mesh/Manage...

The **Mesh/Manage...** menu item opens the [Manage Cell Zones Dialog Box \(p. 590\)](#).

21.4.18.1. Manage Cell Zones Dialog Box

The **Manage Cell Zones** dialog box allows you to change the cell type of a zone, set active cell zones, delete zones, and merge two or more zones into one. It also allows you to rename, rotate, scale, or translate one or more cell zones. The default zone name consists of a cell type (dead, fluid, or solid) and a zone ID number.

**Controls****Options**

contains the following zone manipulation options:

Activate

activates the selected zone(s).

- When you click **Apply**, the selected zone(s) will be made active, and the zone(s) that are not selected will be made inactive.
- When you refine the mesh only cells in active zones will be refined.
- When you display the grid, the cells (including boundary cells), interior nodes, and interior faces that are displayed are those in the active zones. See [Active Zones and Cell Types \(p. 364\)](#) for details.

Change Type

sets the type of the selected cell zone to the type selected in the **Type** drop-down list.

Type

contains a drop-down list of available cell zone types: **dead**, **fluid**, and **solid**.

ANSYS Fluent will read in the two live cell zone types (**fluid** and **solid**) and assign default conditions as described in the ANSYS Fluent User's Guide. Dead cell zones are regions that are not part of the computational domain. They will not be read by ANSYS Fluent.

Copy

copies all nodes, faces, and cells of the selected zone (or zones), creating a new zone of the same type at the same location.

Delete

deletes the selected cell zone (s), along with the associated faces and nodes. You may want to delete dead zones to free up some memory in your computer. Commands for checking and deleting these faces are described in [Additional Boundary Mesh Text Commands \(p. 172\)](#).

This operation is irreversible. After deleting cell zones, check to see if there are any boundary faces located between two deleted zones. Such faces must be removed if you are generating a mesh for ANSYS Fluent.

Merge

merges two or more selected zones into one. The resulting cell zone will have the name, type, and ID of the first selected zone that appears in the **Cell Zones** list. This operation is irreversible.

Merge Sub Zones

controls whether or not same-type face zones and same-type node zones associated with the cell zones being merged should also be merged.

For example, if you merge two cell zones that are bounded by wall zones, the wall zone bounding one cell zone will be merged with the adjacent wall zone bounding the other cell zone.

Rename

allows you to change the name of the selected zone.

Name

specifies the new name for the zone selected in the **Cell Zones** list.

Rotate

rotates the selected zone(s) through the specified angle. See [Copying and Moving Cell Zones \(p. 364\)](#) for details.

Angle

specifies the angle of rotation through which you want to rotate the selected zone(s).

axis

allows you to specify the axis about which you want to rotate the selected zone(s).

pivot

allows you to specify a pivot point about which you want to rotate the selected zone(s).

Copy Zone(s)

allows you to copy the nodes of the selected zone(s), thereby creating new zone(s) of the same type before positioning the copied zone(s) per the specified **Angle** and **axis** (or **pivot**).

Scale

scales the selected zone(s) by multiplying each of the node coordinates by the specified **Scale Factors**. The cell sizes will increase or decrease accordingly.

Scale Factors

specifies the scale factors applied to the grid in each of the Cartesian coordinate directions (**X**, **Y**, and **Z**).

Copy Zone(s)

allows you to copy the nodes of the selected zone(s), thereby creating new zone(s) of the same type before scaling the copied zone(s) per the specified scale factors.

Translate

translates the selected zone(s) by the specified translation offsets. See [Copying and Moving Cell Zones \(p. 364\)](#) for details.

Translation

specifies the translation offsets (**X**, **Y**, **Z**) to be added to the Cartesian coordinate of every node in the selected zone(s).

Define

allows you to define the direction vector and total distance based on two nodes or positions selected in the graphics window.

Note

The translation offsets are interpreted as absolute numbers in meshing mode. In solution mode, however, the translation offsets are assumed to be distances in the length unit set. This may lead to differences in domain extents reported after translating the mesh in the respective modes.

Copy Zone(s)

allows you to copy the nodes of the selected zone(s), thereby creating new zone(s) of the same type before positioning the copied zone(s) per the specified translation offsets.

Cell Zones

contains a list from which you can select the zone(s) to be modified.

Cell Zone Groups

contains a list of cell zone types. If you select a cell type from this list (e.g., **fluid**), all cell zones of that type will be selected in the **Cell Zones** list. This method allows you to easily select all cell zones of a certain type without having to select each zone individually. You can select multiple cell types in the **Cell Zone Groups** list to select all zones of several different types (e.g., **solid** and **fluid**).

Apply

performs the selected option on the selected zones.

Draw

displays the boundaries that define the selected cell zones. If any faces of a boundary are used in defining the zone, then the entire boundary is displayed.

List

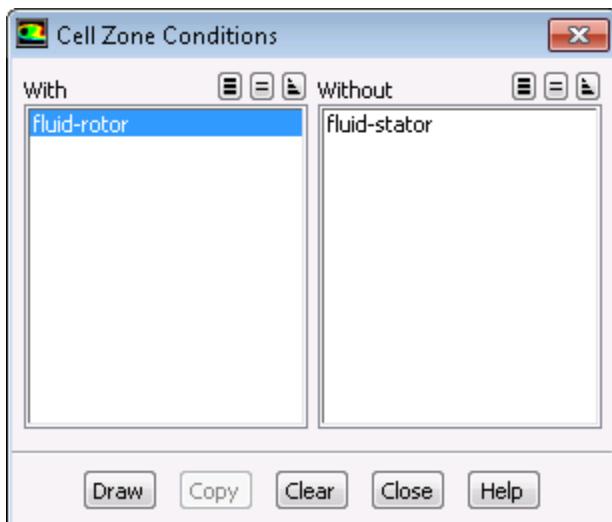
reports (in the console) the ID, name, type, and number of cells for each selected zone.

21.4.19. Mesh/Cell Zone Conditions...

The **Mesh/Cell Zone Conditions...** menu item opens the [Cell Zone Conditions Dialog Box \(p. 593\)](#).

21.4.19.1. Cell Zone Conditions Dialog Box

The **Cell Zone Conditions** dialog box allows you to copy or clear cell zone conditions assigned to the cell zones, when a case file is read.



Controls

With

contains a list of cell zones that have cell zone conditions assigned.

Without

contains a list of cell zones without cell zone conditions assigned.

Draw

displays the selected zones in the graphics window.

Copy

allows you to copy the cell zone conditions from the zone selected in the **With** list to those selected in the **Without** list.

Clear

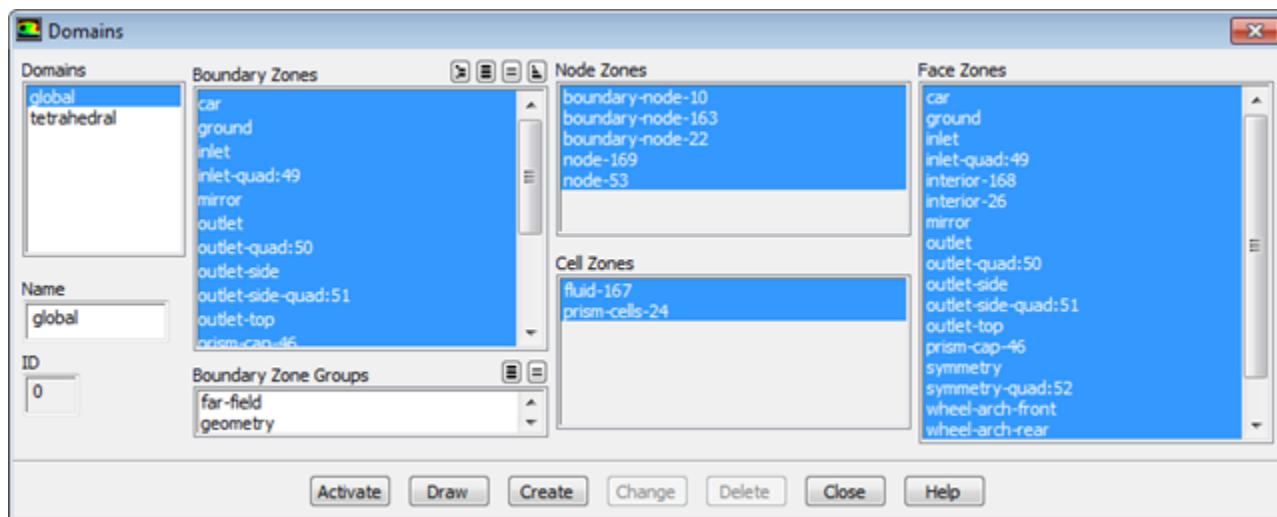
allows you to clear the cell zone conditions assigned to the zone(s) selected in the **With** selection list. The zone(s) will be available in the **Without** selection list once the assigned cell zone conditions are cleared.

21.4.20. Mesh/Domains...

The **Mesh/Domains...** menu item opens the [Domains Dialog Box \(p. 593\)](#).

21.4.20.1. Domains Dialog Box

The **Domains** dialog box allows you to create domains encompassing the region(s) of a hybrid mesh that are to be meshed with tetrahedra or domains containing boundary zones that you wish to group for other reasons. See [Using Domains \(p. 368\)](#) and [Defining Domains \(p. 369\)](#) for details.



Controls

Domains

contains a list of all domains that have been defined, including the **global** domain (containing all zones in the mesh).

Name

sets the name of the new domain you are about to create by clicking on the **New** button, or the name of the selected domain that you are about to change by clicking on the **Change** button.

ID

shows the number of the selected domain. The **global** domain has an ID of 0, the next domain to be created has an ID of 1, and so on.

Boundary Zones

contains a list from which you can select the zone(s) to be included in the domain.

Boundary Zone Groups

contains a list of boundary zone types. If you select a boundary type from this list (e.g., **inlet**), all boundary zones of that type (in this case, all **pressure-inlet** and **velocity-inlet** boundaries) will be selected in the **Boundary Zones** list. This allows you to easily select all boundary zones of a certain type without having to select each zone individually. It is also possible to select all triangular or quadrilateral face zones, by selecting **tri** or **quad** in the **Boundary Zone Groups** list. You can select multiple boundary types in the **Boundary Zone Groups** list to select all zones of several different types (e.g., **inlet** and **outlet**).

Node Zones

shows the names of all node zones in the domain. Those that are included in the currently selected domain (based on your selections in the **Boundary Zones** dialog box) will be highlighted. The association will be updated when you click the **Change** button after modifying your selections in the **Boundary Zones** list.

Cell Zones

shows the names of all cell zones in the global domain. Those that are included in the currently selected domain (based on your selections in the **Boundary Zones** dialog box) will be highlighted. The association will be updated when you click the **Change** button after modifying your selections in the **Boundary Zones** list.

Face Zones

shows the names of all face zones in the domain. Those that are included in the currently selected domain (based on your selections in the **Boundary Zones** dialog box) will be highlighted. The association will be updated when you click the **Change** button after modifying your selections in the **Boundary Zones** list.

Important

For **Node Zones**, **Cell Zones**, and **Face Zones** you cannot modify the face zones in a given domain by changing this list. This list shows (by highlighting) the node zone, cell zones, and face zones respectively, that are contained in the domain as you have defined it.

Activate

sets the domain selected in the **Domains** list to be the “active” domain. When you create the volume mesh, only those zones that are in the active domain will be meshed. When you display the grid, the zones available for display will be only those that are included in the active domain. Similarly, diagnostic reports will report information about only those zones. If you want to check a subset of the global domain, you can create and activate a domain that includes the desired zones, and then proceed with the display or report. If you want your grid display or report to include all zones in the domain, make sure you activate the **global** domain.

Note

The **Activate** button will be disabled as soon as a domain is activated. It will become available again if you select a different domain. When a new domain is created, it will automatically be identified as the active domain and the **Activate** button will be disabled.

Draw

displays the grid for the selected **Boundary Zones**, based on the current settings in the **Display Grid** dialog box.

Create

creates a new domain.

Change

changes the definition of the selected domain to the definition currently shown in the dialog box.

Delete

deletes the selected domain definition. The individual zones that were in the domain are not affected. You can only delete the *grouping* of these zones that was defined by the domain.

21.4.21. Mesh/Clear

The **Mesh/Clear** menu item is used to clear the volume mesh. See [Clearing the Mesh \(p. 371\)](#) for details.

21.4.22. Mesh/Check

The **Mesh/Check** menu item is used to verify the validity of the mesh. See [Checking the Mesh Quality \(p. 371\)](#) for details.

21.4.23. Mesh/Check Quality

The **Mesh/Check Quality** menu item is used to verify the quality of the mesh before transferring the mesh data to solution mode or writing out the mesh/case file. See [Checking the Mesh \(p. 370\)](#) for details.

21.5. Display Menu

The Display Menu includes the following options:

- 21.5.1. [Display/Grid...](#)
- 21.5.2. [Display/Plot/Face Distribution...](#)
- 21.5.3. [Display/Plot/Cell Distribution...](#)
- 21.5.4. [Display/Options...](#)
- 21.5.5. [Display/Views...](#)
- 21.5.6. [Display/Lights...](#)
- 21.5.7. [Display/Scene...](#)
- 21.5.8. [Display/Mouse Buttons...](#)
- 21.5.9. [Display/Mouse Probe...](#)
- 21.5.10. [Display/Annotate...](#)
- 21.5.11. [Display/Selection Helper...](#)
- 21.5.12. [Display/Controls...](#)

21.5.1. Display/Grid...

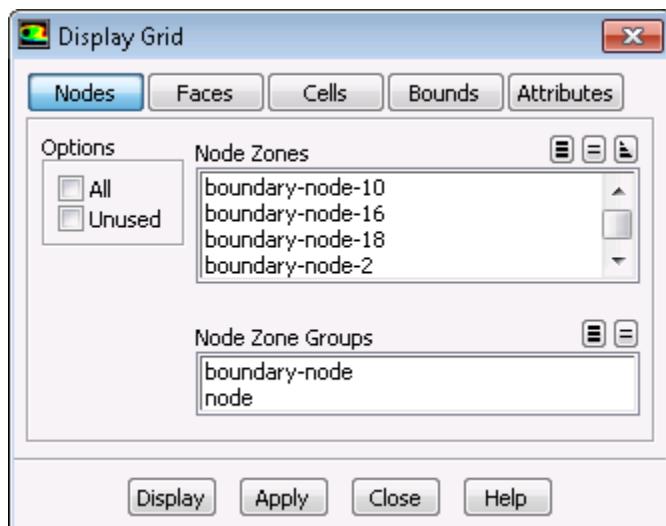
The **Display/Grid...** menu item opens the [Display Grid Dialog Box \(p. 596\)](#).

21.5.1.1. Display Grid Dialog Box

The **Display Grid** dialog box allows you to select the grid zones to be drawn. You can also display unmeshed, free, or multiply-connected faces, or unused nodes, and you can limit the display within certain quality and/or size ranges. These controls may help you in determining the causes of boundary mesh problems.

Cells (including boundary cells), interior nodes, and interior faces will be displayed only if they are in the active cell zones. In addition, if you have used domains to generate the mesh or grouped zones for reporting (as described in [Using Domains to Group and Mesh Boundary Faces \(p. 368\)](#)), only the cells, faces, etc. in the active domain can be displayed.

Controls



Nodes

contains options for displaying nodes.

Options

allows you to select one or more of the following options:

All

draws all nodes in the node zones selected from the **Node Zones** list, colored by their zone type.
Select one or more zones for any to be displayed.

Unused

draws nodes that are not currently used by any faces or cells. The unused nodes are usually displayed with a different color and marker symbol to distinguish them from nodes in use.

Node Zones

contains a list of node zones from which you can select those to be drawn.

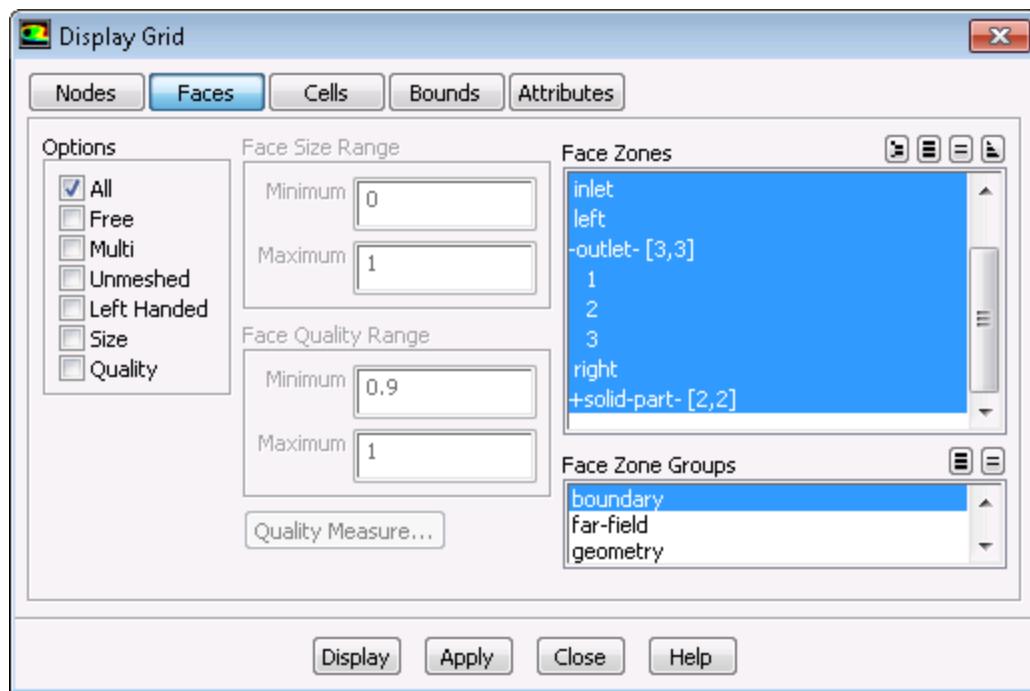
Node Zone Groups

contains a list of node zone types. Selecting a zone type from this list (e.g., **boundary-node**), will select all node zones of that type (**boundary-node-1**, **boundary-node-2**, etc.) in the **Node Zones** list.

This shortcut allows you to select all zones of a certain type without having to select each zone individually.

Faces

contains options for displaying faces.



Options

allows you to select one or more of the following options:

All

draws all faces in the selected face zones, colored by their zone type. Select one or more zones from the **Face Zones** list for any to be displayed.

Free

draws free faces (faces with no neighboring face on at least one edge) on the selected zones.

Multi

draws multiply-connected faces on the selected zones, along with their nodes. A multiply-connected face is a boundary face that shares an edge with more than one other face, while a multiply-connected node is a node that is on a multiply-connected edge (i.e., an edge that is shared by more than two boundary faces).

Multiply-connected faces or nodes are usually an indication that something is wrong with the surface mesh, and it needs to be regenerated. Such a surface mesh is often referred to as "non-manifold". However, if you are creating a mesh with more than one fluid, solid, or porous zone, you can expect multiply-connected faces or nodes to occur at the edges of adjacent zones. This does not indicate any problem.

Unmeshed

draws unmeshed faces on the selected zones, along with their nodes. Unmeshed nodes and faces are those that were not inserted into the initial mesh. If any unmeshed nodes or faces are displayed, you need to regenerate the initial mesh.

Left Handed

draws left-handed faces on the selected zones. If the normal on a face (defined using the right-hand rule) does not point into the cell, the face is left-handed. Since the solver assumes a right-handed relationship between faces and cells, left-handed faces introduce errors into the solution. Left-handed faces may be created if there are problems with the generation of prismatic cells. See [Negative Volumes/Left-Handed Faces/High Skewness \(p. 262\)](#) for details.

Size

draws faces within the size range indicated by the **Face Size Range** limits.

Quality

draws faces within the quality range indicated by the **Face Quality Range** limits. The default measure of quality is skewness, but you can use the **Quality Measure** dialog box to specify aspect ratio or change in size instead.

Face Size Range

limits the faces displayed to those with a size within the specified range. To specify a face size range, enable **Size** in the **Options** group box, set the appropriate **Minimum** and **Maximum** values, and click **Display** or **Apply**.

Minimum

sets the minimum value for the size range, in the physical dimensions of the mesh. This value should be less than the **Maximum** value.

Maximum

sets the maximum value for the skewness range, in the physical dimensions of the mesh. This value should be greater than the **Minimum** value.

Face Quality Range

limits the faces displayed to those with a quality measure value (skewness, by default) within the specified range. To specify a face quality range, enable **Quality** in the **Options** group box, set the appropriate **Minimum** and **Maximum** values, and click **Display** or **Apply**. Use the **Report Face Limits** dialog box to determine an appropriate range.

Minimum

sets the minimum value for the quality range. This value should be greater than or equal to zero, and less than the **Maximum** value.

Maximum

sets the maximum value for the quality range. This value should be greater than the **Minimum** value. If the quality measure being used is skewness, it should also be less than or equal to one.

Quality Measure...

opens the **Quality Measure** dialog box, in which you can select the measure of quality to be used (skewness, aspect ratio, change in size, etc.).

Face Zones

contains a list of face zones from which you can select those to be drawn.

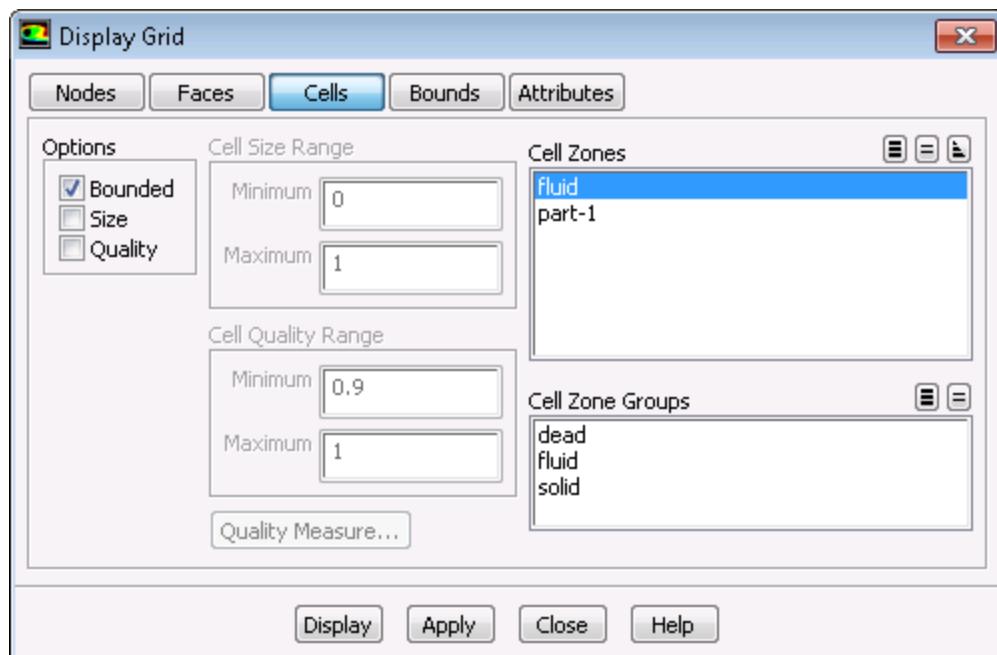
Face Zone Groups

contains a list of face zone types. Selecting a zone type from this list (e.g., **inlet**), will select all face zones of that type (**pressure-inlet-3**, **velocity-inlet-5**, etc.) in the **Face Zones** list. This shortcut allows you to easily select all zones of a certain type without having to select each zone individually.

You can also select multiple zone types in the **Zone Groups** list to select all zones of different types (e.g., **wall** and **symmetry**).

Cells

contains options for displaying cells.



Options

allows you to select one or more of the following options.

Bounded

will be activated after some bounds are set in the **Bounds** tab. If enabled, draws all cells within the applied bounds, in the selected cell zones, colored by their zone type. Select one or more zones from the **Cell Zones** list for any to be displayed.

Size

draws cells within the size range indicated by the **Cell Size Range** limits.

Quality

draws cells within the quality range indicated by the **Cell Quality Range** limits. The default measure of quality is skewness, but you can use the **Quality Measure** dialog box to specify aspect ratio or change in size instead.

Cell Size Range

limits the cells displayed to those having a size within the specified range. To specify a cell size range, enable **Size** in the **Options** group box, set the appropriate **Minimum** and **Maximum** values, and click **Display** or **Apply**.

Minimum

sets the minimum value for the size range, in the physical dimensions of the mesh. This value should be less than the **Maximum** value.

Maximum

sets the maximum value for the skewness range, in the physical dimensions of the mesh. This value should be greater than the **Minimum** value.

Cell Quality Range

limits the cells displayed to those with a quality measure value (skewness, by default) within the specified range. To specify the cell quality range, enable **Quality** in the **Options** group box, set the appropriate **Minimum** and **Maximum** values, and click **Display** or **Apply**. Use the **Report Cell Limits** dialog box to determine an appropriate range.

Minimum

sets the minimum value for the quality range. This value should be greater than or equal to zero, and less than the **Maximum** value.

Maximum

sets the maximum value for the quality range. This value should be greater than the **Minimum** value. If the quality measure being used is skewness, it should also be less than or equal to one.

Quality Measure...

opens the **Quality Measure** dialog box , in which you can select the measure of quality to be used (skewness, aspect ratio, change in size, etc.).

Cell Zones

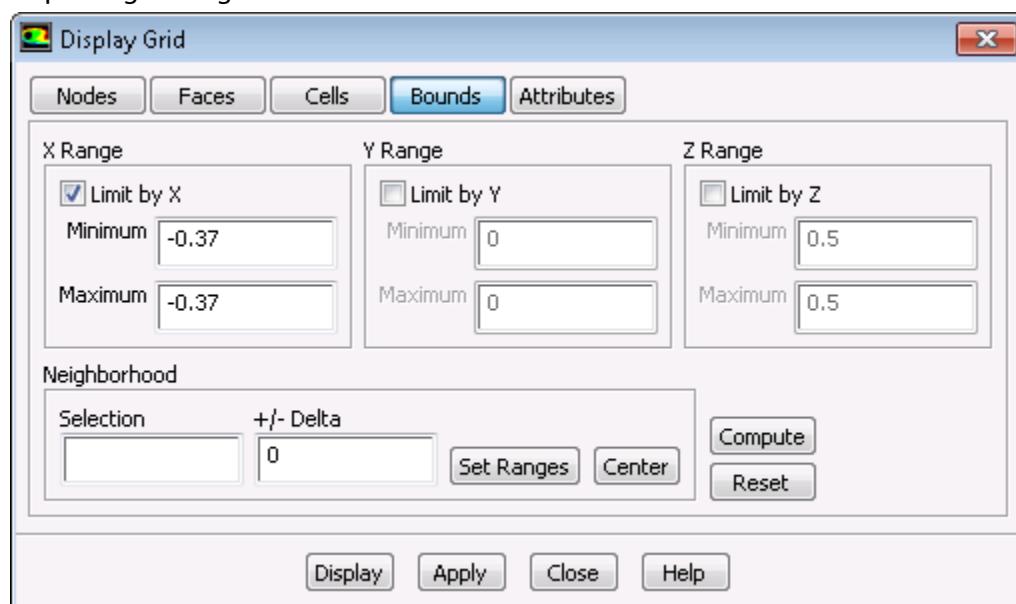
contains a list of cell zones from which you can select those to be drawn.

Cell Zone Groups

contains a list of cell zone types. Selecting a zone type from this list (e.g., **solid**), will select all cell zones of that type (**solid-4**, **solid-5**, etc.) in the **Cell Zones** list. You can select multiple zone types in the **Zone Groups** list to select all zones of several different types (e.g., **fluid** and **solid**).

Bounds

contains options allowing you to limit the items drawn in the display to those inside specified bounds or passing through them.

**X Range**

allows you to limit the display to the specified X coordinate range. To use the range, enable **Limit by X**, then enter the required **Minimum** and **Maximum** values, in the physical dimensions of the mesh.

Y Range

allows you to limit the display to the specified Y coordinate range. To use the range, enable **Limit by Y**, then enter the required **Minimum** and **Maximum** values, in the physical dimensions of the mesh.

Z Range

allows you to limit the display to the specified Z coordinate range. To use the range, enable **Limit by Z**, then enter the required **Minimum** and **Maximum** values, in the physical dimensions of the mesh.

Neighborhood

allows you to set the range to be within a specified neighborhood of a selected entity. Specify the **Selection** and **+/- Delta**, and click **Set Ranges**.

Selection

indicates the name of the selected entity on which the specified range is based. You can select entities with the mouse in the graphics window if the mouse probe function (see [Controlling the Mouse Probe Functions \(p. 393\)](#)) is currently **select** and the **filter** is set to **node, edge, face, cell, zone, position, or object**.

+/- Delta

specifies the neighborhood range about the selected entity.

Set Ranges

sets the **X**, **Y**, and **Z Range** to the values determined by the entities selected.

Center

changes the display to place the selected entity in the center of the view.

Compute

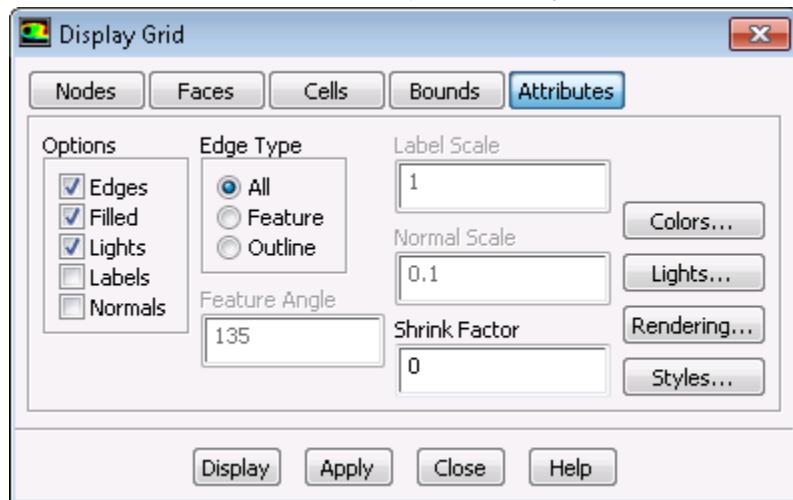
sets the range to include the entire geometry.

Reset

restores the controls to their default settings (i.e., no limits on x, y, or z range). The previously set maximum and minimum values for each range will remain visible, but they will be grayed out, indicating that they are not in effect.

Attributes

contains options for customizing the display.

**Options**

allows you to select one or more of the following options:

Edges

enables display of face edges.

Filled

enables faces and cells to be drawn with filled polygons. This is most useful when the **Lights** option is enabled.

Lights

enables display lighting to make the display realistic. This is only useful if **Filled** is enabled.

Labels

allows you to display labels. You can control the size of the labels using an appropriate **Label Scale**.

Normals

enables the display of face normals. You can control the size of the normal vectors using an appropriate **Normal Scale**.

Edge Type

contains options controlling the display of edges. These options are available only when the **Edges** option is enabled. See [Adding Features to an Outline Display \(p. 375\)](#) for details.

All

enables the display of all grid edges.

Feature

enables feature lines in an outline display.

Outline

enables the display of the grid outline.

Feature Angle

controls the amount of detail added to a feature outline display. This item is available only when **Feature** is selected in the **Edge Type** list. See [Adding Features to an Outline Display \(p. 375\)](#) for details.

Label Scale

controls the size of the labels displayed.

Normal Scale

controls the size of the face normals displayed.

Shrink Factor

specifies the amount by which to shrink faces and cells.

Colors...

opens the **Grid Colors** dialog box, using which you can specify colors for different grid zones.

Lights...

opens the **Lights** dialog box, using which you can control the lighting used in the display.

Rendering...

opens the **Display Options** dialog box, using which you can control various rendering options.

Styles...

opens the **Style Attributes** dialog box, where you can change how various types of grid entities are displayed.

Display

updates the display according to the options selected in all frames in the **Display Grid** dialog box. All options are saved before the display is updated, as if you had clicked the **Apply** button.

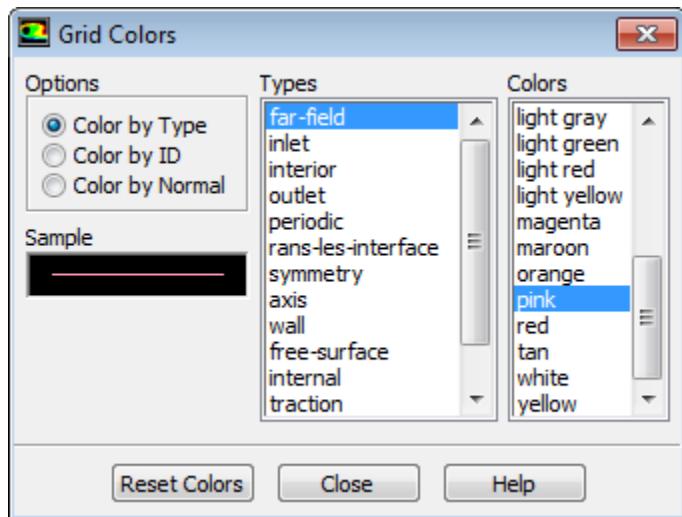
Apply

saves the options selected in all frames of the **Display Grid** dialog box, but does not update the display.

The **Display** and **Apply** buttons (along the bottom of the dialog box) apply to *all* controls in *all* frames of the dialog box (i.e., for all tabs). For example, you can change some controls in the **Faces** tab, then click the **Bounds** tab and change some controls there, and then click **Display** to draw the grid with the new settings. Even though a tab's controls are not visible, its current settings affect the display.

21.5.1.2. Grid Colors Dialog Box

Click the **Colors...** button in the **Attributes** tab of the **Display Grid** dialog box to open the **Grid Colors** dialog box. It allows you to control the color of the zones that are displayed using the **Display Grid** dialog box. You can color zones by their type, by their ID number, or by the normals. You can see the effect of these changes the next time you display the grid.

**Options**

contains options for setting the colors.

Color by Type

allows you to set the color for each type of zone (outlet, wall, etc.).

Color by ID

modifies the grid colors so that each zone (**wall-1**, **wall-2**, **outlet-3**, etc.) will be assigned one of the colors from the **Colors** list. If there are more zones than colors, some zones will be the same color.

Color by Normal

colors one side of the faces in grey and the other side in yellow, depending on the normal direction. The side colored grey indicates normal direction (normals pointing toward the user), while the side colored yellow has normals pointing away.

Sample

displays a sample of the currently selected color.

Types

contains a selectable list of zone types. You can select the zone type for which you want to set the color. Note that you can set colors separately for each zone grid, using the **Scene Description** dialog box.

Colors

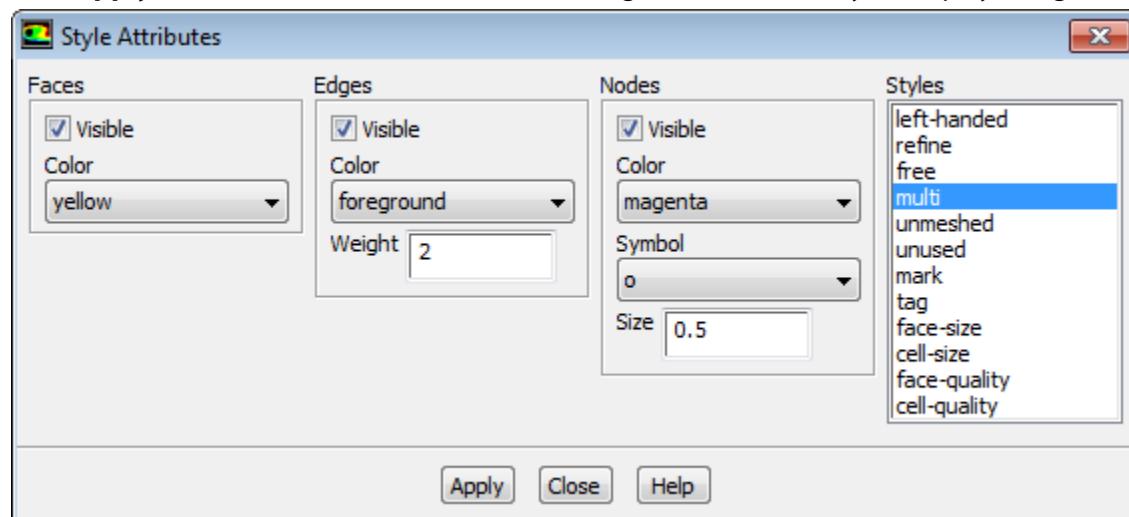
contains a list from which you can select a color for the selected type.

Reset Colors

clears any colors you have set in the **Scene Description** dialog box so that all colors will revert to their definition by type or by ID, as indicated under **Options**.

21.5.1.3. Style Attributes Dialog Box

The **Style Attributes** dialog box is opened by clicking on the **Styles...** button in the **Attributes** tab frame of the **Display Grid** dialog box. It controls the appearance of faces, edges, and nodes that are displayed using the **Display Grid** dialog box. To modify the attributes of a certain type of face, edge, or node (e.g., unmeshed), select the appropriate item in the **Styles** list, change the parameters, and click **Apply**. You can see the effect of these changes the next time you display the grid.

**Faces**

contains parameters that control the display of faces.

Visible

indicates whether or not the selected type of face should be visible in the graphics display. If a face is visible, its interior will be filled. To see the outline of a face, you can make **Edges** visible.

Color

specifies the color to be used for drawing the selected type of face. Choose the color you want from the drop-down list.

Edges

contains parameters that control the display of edges.

Visible

indicates whether or not the edges of the selected type of face should be visible in the graphics display.

Color

specifies the color to be used for drawing the edges of the selected type of face. Choose the color you want from the drop-down list.

Weight

specifies the thickness of edge lines, as a multiple of the default thickness.

Nodes

contains parameters that control the display of nodes.

Visible

indicates whether or not the selected type of node should be visible in the graphics display.

Color

specifies the color to be used for drawing the selected type of node. Choose the color you want from the drop-down list.

Symbol

specifies the symbol to be used for drawing the selected type of node.

Size

specifies the size of the symbol.

Styles

contains a list of all the different types of nodes and faces that can be displayed. The settings under **Faces**, **Edges**, and **Nodes** will apply to the node or face type selected in the **Styles** list. These types correspond to the options in the **Display Grid** dialog box. The default style attributes are listed in [Table 21.1: Default Style Attributes \(p. 607\)](#).

refine

indicates boundary faces to be refined.

free

indicates free nodes or faces.

left-handed

indicates faces that do not follow the right-hand rule with respect to their cell neighbors.

multi

indicates multiply-connected nodes or faces.

unmeshed

indicates unmeshed nodes or faces.

unused

indicates unused nodes or faces.

mark

indicates marked entities (for expert users).

tag

indicates tagged entities (for expert users).

face-size

indicates faces within the specified face size range.

cell-size

indicates cells within the specified cell size range.

face-quality

indicates faces within the specified face quality range.

cell-quality

indicates cells within the specified cell quality range.

Table 21.1: Default Style Attributes

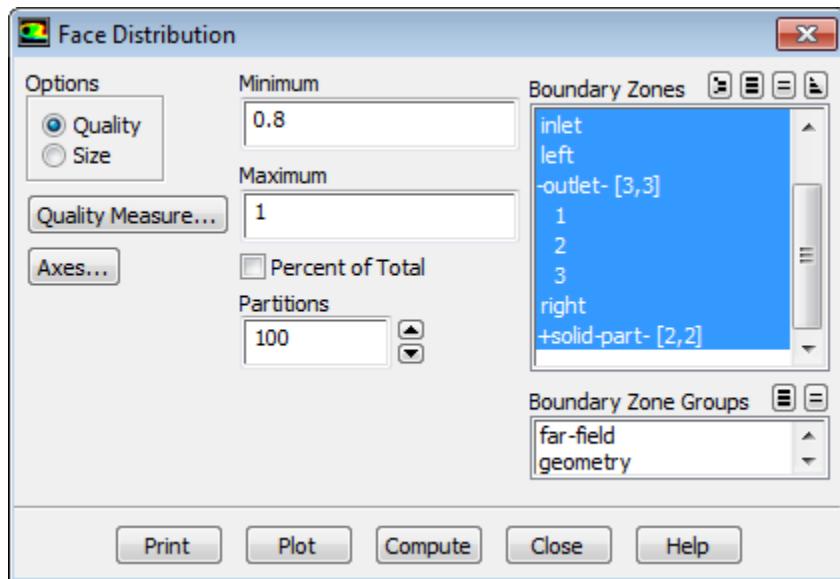
Styles	Faces Vis- ible	Edges Col- or	Nodes Vis- ible	Col- or	Weight	Vis- ible	Col- or	Sym- bol	Size
left-handed	yes	red	yes	red	2	yes	red	o	0.5
re-fine	yes	magenta	yes	yellow	2	yes	magenta	o	0.5
free	yes	cyan	yes	foreground	2	yes	orange	o	0.5
multi	yes	yellow	yes	foreground	2	yes	magenta	o	0.5
un-meshed	no	–	yes	orange	2	yes	red	o	0.5
un-used	no	–	yes	green	2	yes	green	x	0.5
mark	yes	magenta	yes	yellow	2	yes	magenta	o	0.5
tag	yes	magenta	yes	yellow	2	yes	magenta	o	0.5
face-size	yes	blue	yes	foreground	1	no	–	–	–
cell-size	no	–	yes	red	1	no	–	–	–
face-quality	yes	red	yes	foreground	1	no	–	–	–
cell-quality	no	–	yes	red	1	no	–	–	–

21.5.2. Display/Plot/Face Distribution...

The **Display/Plot/Face Distribution...** menu item opens the [Face Distribution Dialog Box \(p. 607\)](#).

21.5.2.1. Face Distribution Dialog Box

The **Face Distribution** dialog box allows you to generate a histogram of boundary face size or quality.



The number of faces on the selected zones with a size or quality measure value (e.g., skewness) in the specified range in each of the regularly spaced partitions is displayed in a histogram format. The x axis shows the size or quality and the y axis gives the number of faces or the percentage of the total number of faces.

Note

If you have used domains to generate the mesh or group zones for reporting (as described in [Using Domains to Group and Mesh Boundary Faces \(p. 368\)](#)), you can report the face distribution only for those face zones that are in the active domain.

Options

specifies a histogram of face **Quality** or **Size**. The default measure of quality is skewness, but you can use the **Quality Measure** dialog box to specify aspect ratio or change in size instead.

Quality Measure...

opens the **Quality Measure** dialog box, in which you can select the measure of quality to be reported (skewness, aspect ratio, or change in size).

Axes...

opens the **Axes** dialog box, in which you can set parameter ranges, number formats, rules, etc.

Minimum

specifies that only faces with a quality measure value (skewness, by default) greater than this value will be plotted.

Maximum

specifies that only faces with a quality measure value (skewness, by default) less than this value will be plotted.

Percent of Total

specifies that the number of faces as a percentage of the total number of faces counted will be plotted. By default, the actual number of faces is plotted.

Partitions

specifies the number of partitions in the histogram. Increasing the number of partitions will improve the resolution of the plot (see [Figure 21.7: Histogram with 50 Partitions \(p. 609\)](#) and [Figure 21.8: Histogram with 100 Partitions \(p. 609\)](#)).

Figure 21.7: Histogram with 50 Partitions

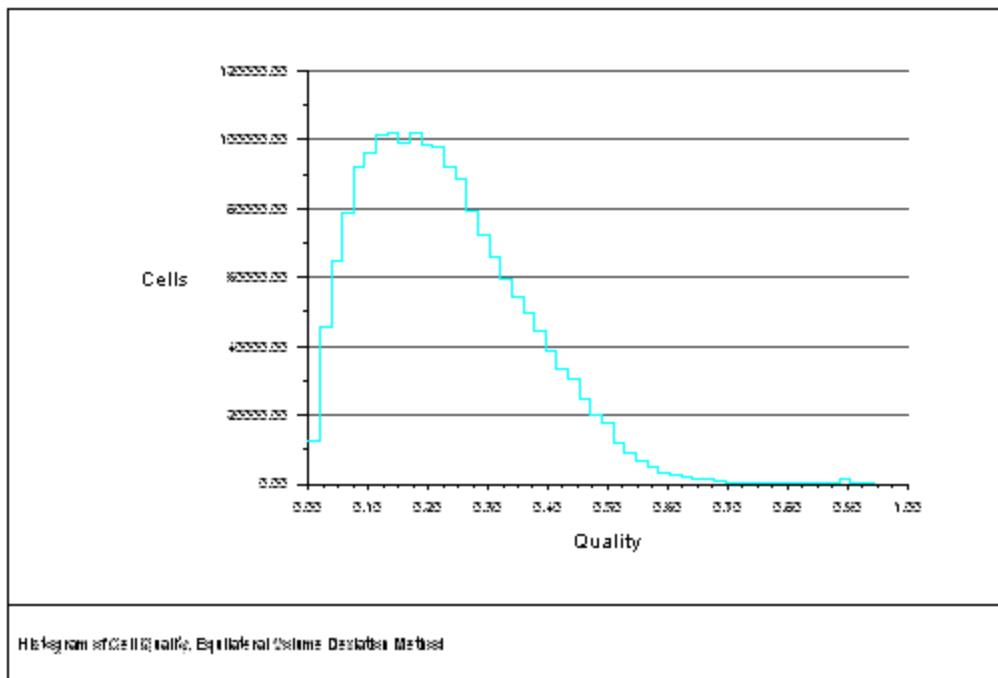
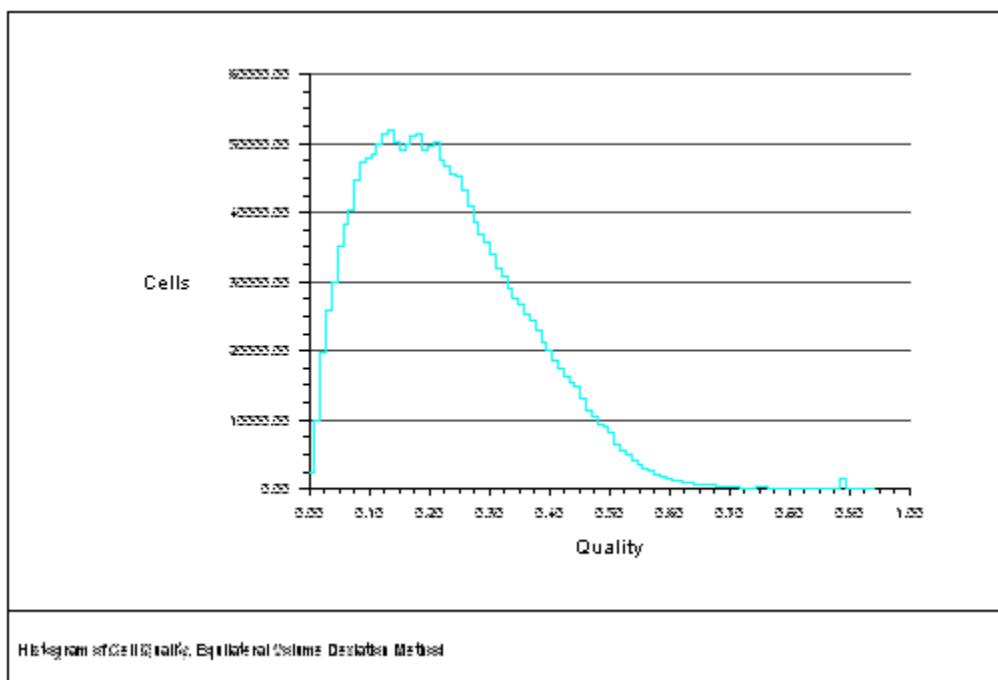


Figure 21.8: Histogram with 100 Partitions



Boundary Zones

contains a list from which you can select one or more zones to be used to generate the histogram.

Boundary Zone Groups

contains a list of boundary zone types. If you select a boundary type from this list (e.g., **inlet**), all boundary zones of that type (for this example, all **pressure-inlet** and **velocity-inlet** boundaries) will be selected in the **Boundary Zones** list. This shortcut allows you to easily select all boundary zones of a certain type without having to select each zone individually. Note that you can select multiple boundary types in the **Boundary Zone Groups** list to select all zones of several different types (e.g., **inlet** and **outlet**).

Print

reports the distribution of face size or quality in the text window.

Plot

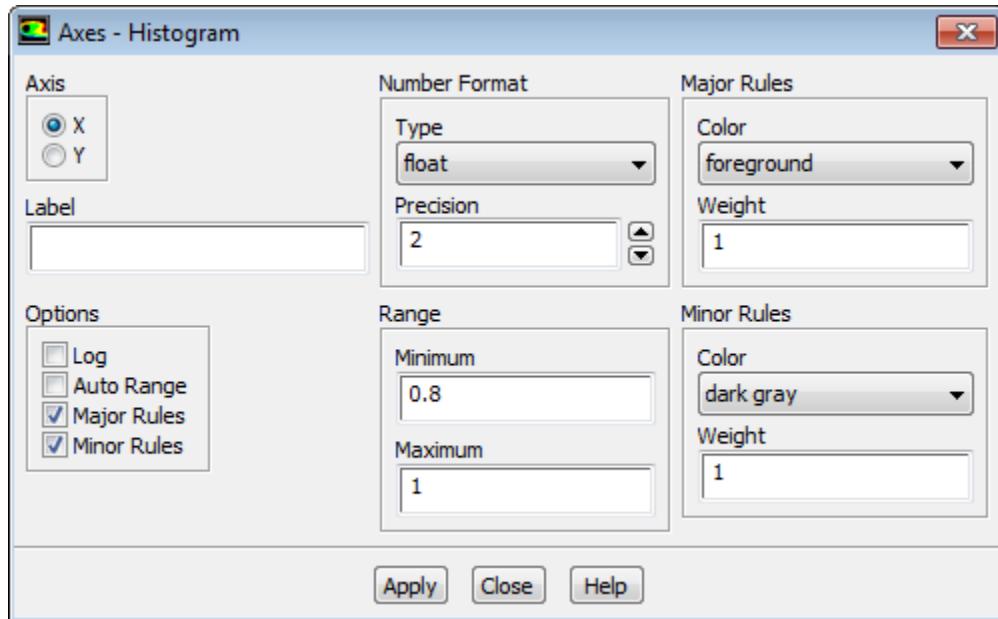
plots a histogram of boundary face size or quality.

Compute

determines the minimum and maximum face size or quality.

21.5.2.2. Axes - Histogram Dialog Box

The **Axes** dialog box allows you to independently control the characteristics of the ordinate and abscissa on an XY plot or histogram. You can change the labels, scale, range, number format, and major and minor rules visibility and appearance.

**Axis**

contains check buttons that allow you to set abscissa (x-axis) or ordinate (y-axis) characteristics.

X

allows you to specify the abscissa characteristics.

Y

allows you to specify the ordinate characteristics.

Label

defines the character string that will label the active axis (the one selected in **Axis**) in the display.

Options

contains check buttons for scale, range, major rules, and minor rules.

Log

toggles logarithmic scaling of the active axis. By default, decimal scaling is used.

Auto Range

toggles automatic computation of the range of the active axis. If you deactivate this option, you can enter the **Minimum** and **Maximum** values in the **Range** box.

Major Rules

toggles the display of major rules on the active axis. Major rules are the horizontal or vertical lines that mark the primary data divisions and span the whole plot window to produce a "grid".

Minor Rules

toggles the display of minor rules on the active axis. Minor rules are the horizontal or vertical lines that mark the secondary data divisions and span the whole plot window to produce a "grid".

Number Format

contains commands for changing the format of the data labels on the active axis. Data labels are the character strings used to define the primary data divisions on the axes.

Type

sets the form of the data labels. You may select from a drop-down list of options, including the following:

general

displays the real value with either float or exponential form based on the size of the number and the defined **Precision**.

float

displays the real value with an integral and fractional part (e.g., 1.0000), where the number of digits in the fractional part is determined by **Precision**.

exponential

displays the real value with a mantissa and exponent (e.g., 1.0e-02), where the number of digits in the fractional part of the mantissa is determined by **Precision**.

Precision

defines the number of fractional digits displayed in the data labels.

Range

contains the range or extents of the active axis. To set the range manually, you must turn off **Auto Range**. Otherwise the extents are computed automatically.

Minimum

sets the minimum data value for the active axis.

Maximum

sets the maximum data value for the active axis.

Major Rules

contains commands for modifying the appearance of the major rules. To use these commands you must activate **Major Rules** in the **Options** list.

Color

sets the color of the major rules from a drop-down list with numerous color selections.

Weight

sets the line thickness of the major rule. A line of weight 1.0 is normally 1 pixel wide. A weight of 2.0 would make the line twice as thick (i.e., 2 pixels wide).

Minor Rules

contains commands for modifying the appearance of the minor rules. To use these commands you must activate **Minor Rules** in the **Options** list.

Color

sets the color of the minor rules from a drop-down list with numerous color selections.

Weight

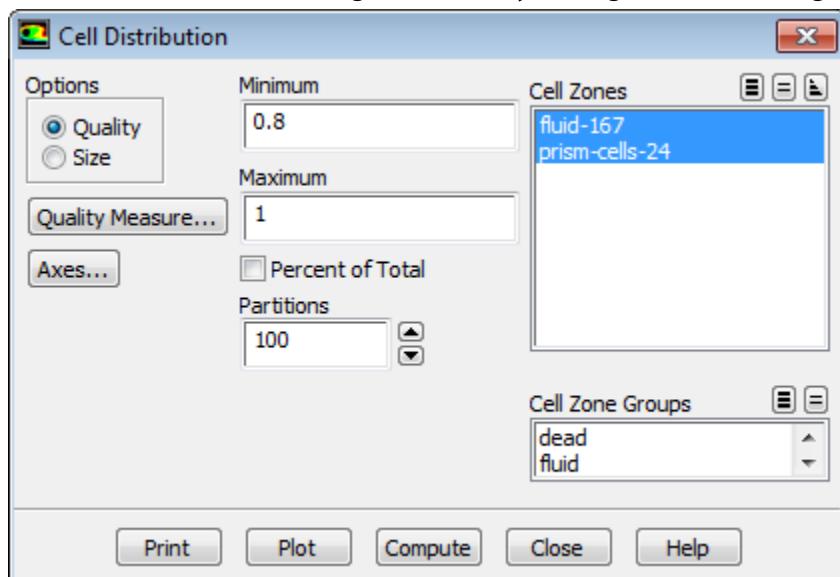
sets the line thickness of the minor rule. A line of weight 1.0 is 1 pixel wide. A weight of 2.0 would make the line twice as thick (i.e., 2 pixels wide).

21.5.3. Display/Plot/Cell Distribution...

The **Display/Plot/Cell Distribution...** menu item opens the [Cell Distribution Dialog Box \(p. 612\)](#).

21.5.3.1. Cell Distribution Dialog Box

The **Cell Distribution** dialog box allows you to generate a histogram of cell size or quality.



The number of cells in the specified range in each of the regularly spaced partitions is displayed in a histogram format. The x axis shows either the size or the quality and the y axis gives the number of cells or the percentage of the total.

Note

If you have used domains to generate the mesh or group zones for reporting (as described in [Using Domains to Group and Mesh Boundary Faces \(p. 368\)](#)), you can report the cell distribution only for those cell zones that are in the active domain.

Controls**Options**

specifies a histogram of cell **Quality** or **Size**. The default measure of quality is skewness, but you can use the **Quality Measure** dialog box to specify aspect ratio or change in size instead.

Quality Measure...

opens the **Quality Measure** dialog box, in which you can select the measure of quality to be reported (skewness, aspect ratio, or change in size).

Axes...

opens the **Axes** dialog box, in which you can set parameter ranges, number formats, rules, etc.

Minimum

specifies that only cells with size or quality measure value (skewness, by default) greater than this value will be plotted.

Maximum

specifies that only cells with size or quality measure value (skewness, by default) less than this value will be plotted.

Percent of Total

specifies that the number of cells as a percentage of the total number of cells counted will be plotted. By default, the actual number of cells is plotted.

Partitions

specifies the number of partitions in the histogram. Increasing the number of partitions will improve the resolution of the plot (see [Figure 21.7: Histogram with 50 Partitions \(p. 609\)](#) and [Figure 21.8: Histogram with 100 Partitions \(p. 609\)](#)).

Cell Zones

contains a list from which you can select one or more zones to be used to generate the histogram.

Cell Zone Groups

contains a list of cell zone types. If you select a zone type from this list (e.g., **fluid**), all cell zones of that type will be selected in the **Cell Zones** list. This shortcut allows you to easily select all zones of a certain type without having to select each zone individually. You can select multiple types in the **Cell Zone Groups** list to select all zones of several different types (e.g., **fluid** and **solid**).

Print

reports the distribution of cell quality or size in the text window.

Plot

plots a histogram of cell distribution.

Compute

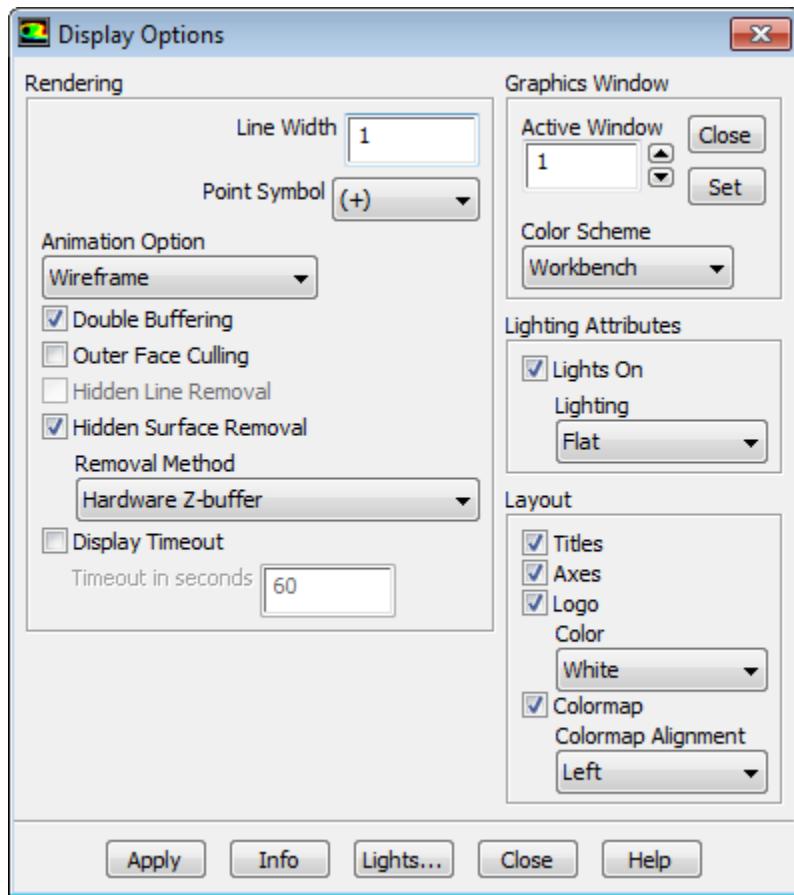
determines the minimum and maximum cell size or quality.

21.5.4. Display/Options...

The **Display/Options...** menu item opens the [Display Options Dialog Box \(p. 614\)](#).

21.5.4.1. Display Options Dialog Box

The **Display Options** dialog box provides an interactive mechanism for setting attributes or options that control how and where a scene is rendered.



Controls

Rendering

allows you to modify characteristics of the display that are related to the way in which scenes are rendered.

Line Width

controls the thickness of lines. The default is 1.

Point Symbol

sets the symbol used for nodes and data points.

Animation Options

contains a list of the animation options available.

All

uses a solid-tone shading representation of all geometry during mouse manipulation.

Wireframe

uses a wireframe representation of all geometry during mouse manipulation. This option is selected by default. Select an alternative option only if your computer has a graphics accelerator; otherwise the mouse manipulation may be very slow.

Double Buffering

allows you to enable/disable double buffering, if it is supported by the driver. Double buffering dramatically reduces screen flicker during graphics updates. If your display hardware does not support double buffering and you enable this option, double buffering will be done in the software. Software double buffering uses extra memory.

Outer Face Culling

allows you to disable/enable the display of outer faces in wall zones. Outer face culling is useful for displaying both sides of a slit wall. By default, when you display a slit wall, one side will "bleed" through to the other.

When you enable the **Outer Face Culling** option, the display of a slit wall will show each side distinctly as you rotate the display. This option can also be useful for displaying two-sided walls (i.e., walls with fluid or solid cells on both sides).

Hidden Line Removal

allows you to enable/disable hidden line removal. If you do not use hidden line removal, ANSYS Fluent will not try to determine which lines in the display are behind others. It will display all of them resulting a cluttered display.

Note

This option is not available when the **Workbench** color scheme is used.

Hidden Surface Removal

allows you to enable/disable hidden surface removal. If you do not use hidden surface removal, ANSYS Fluent will not try to determine which surfaces in the display are behind others. It will display all of them resulting a cluttered display.

Removal Method

lists the methods available for hidden surface removal. These options vary in speed and quality, depending on the device you are using. The choices available are as follows:

Hardware Z-buffer

is the fastest method if supported by your hardware. The accuracy and speed of this method is hardware dependent.

Painters

will show fewer edge-aliasing effects than hardware-z-buffer. This method is often used instead of software-z-buffer when memory is limited.

Software Z-buffer

is the fastest of the accurate software methods available (especially for complex scenes), but it is memory intensive.

Z-sort only

is a fast software method, but it is not as accurate as software-z-buffer.

Display Timeout

enables the setting of a timeout for displaying the cell zones (in meshing mode only).

Timeout in seconds

specifies the value for the timeout.

Important

When the geometry fails to display within the time specified, an error message will be displayed in the console, indicating that the model could not be displayed within the specified time.

Graphics Window

allows you to open and close graphics windows.

Active Window

indicates the graphics window to be opened, closed, or set active. Its title displays the graphics window ID.

Open/Close

opens or closes the window with the ID shown in the **Active Window** box. If the indicated window is open, the **Close** button will appear. If the indicated window is not open, the **Open** button will appear.

Set

sets the window with the ID shown in the **Active Window** box to be the active graphics window.

Color Scheme

contains a drop-down list of available graphics window color schemes to choose from. You can select either the **Workbench** (blue background) scheme or the **Classic** (black background) scheme.

Lighting Attributes

controls lighting attributes for all lights in the active graphics window.

Lights On

allows you to enable/disable *all* lights in the active graphics window.

Lighting

specifies the method to be used in lighting interpolation. **Flat**, **Gouraud**, or **Phong**. **Flat** is the most basic method. There is no interpolation within the individual polygonal facets. **Gouraud** and **Phong** have smoother gradations of color because they interpolate on each facet.

Layout

controls the display of captions , axes, and the colormap in the graphics display window.

Titles

enables/disables the display of captions in the graphics display window.

Axes

enables/disables the display of the axis triad.

Logo

allows you to hide or display the ANSYS logo.

Color

contains a drop-down list of colors that can be used for the logo. The choices are **White** (default) or **Black**.

Colormap

enables/disables the display of the colormap.

Colormap Alignment

allows you to adjust the alignment of the colormap in the graphics window. Select the side (i.e., **Left**, **Top**, **Right**, **Bottom**) where you want to align the colormap from the drop-down list.

Apply

applies the specified attributes and re-renders the scene in the active graphics window with the new attributes. To see the effect of the new attributes on other graphics windows, redisplay them.

Info

prints out information about the graphics driver in the console.

Lights...

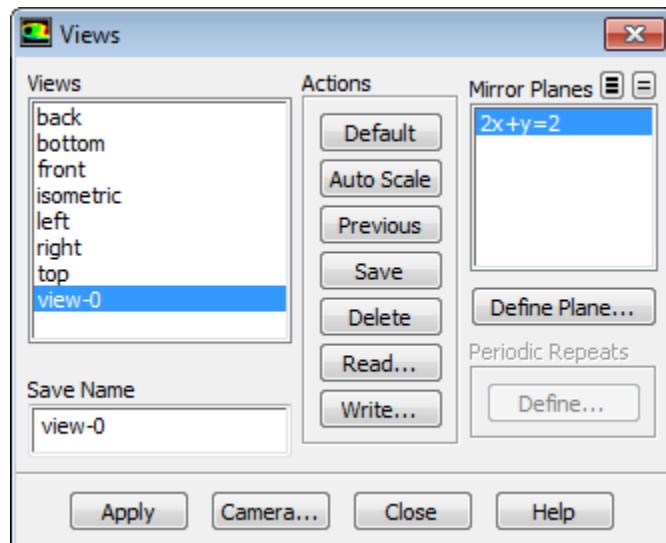
opens the **Lights** dialog box, which allows you to create, delete, and modify directional light sources.

21.5.5. Display/Views...

The **Display/Views...** menu item opens the [Views Dialog Box \(p. 617\)](#).

21.5.5.1. Views Dialog Box

With the **Views** dialog box, you can make various modifications to the view displayed in the active graphics window.

**Controls****Views**

lists the currently defined views. Clicking on a view name highlights that name and enters it into the **Name** field. Double-clicking on a view name restores that view in the active graphics window.

Save Name

specifies the name to use when saving a view.

Actions

contains buttons for performing various actions related to the **Views** list and the **Save Name**.

Default

restores the “front” view in the active graphics window.

Auto Scale

modifies the view in the active graphics window by scaling and centering the current scene without changing its orientation.

Save

stores the view in the active graphics window with the name in the **Save Name** box.

Restore

sets the view of the active graphics window to be the view selected in the **Views** list.

Delete

removes the selected view name from the **Views** list. Be careful not to delete any of the predefined views.

Read...

opens the **Select File** dialog box, in which you can specify the name of a view file to be read.

Write...

opens the **Write Views** dialog box, in which you can select the views to be saved to a view file.

Mirror Planes

displays a list of all symmetry planes in the domain. Mirror images are drawn for all selected symmetry planes.

Define Plane...

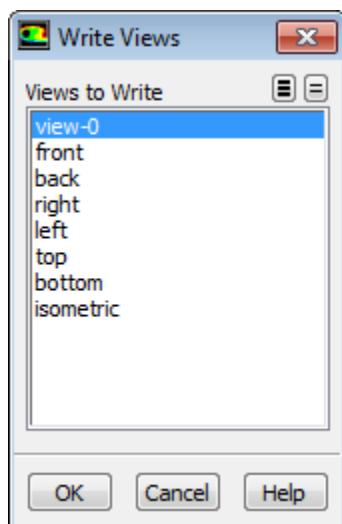
opens the **Mirror Planes** dialog box, in which you can define a mirror plane for a non-symmetric domain.

Camera...

opens the **Camera Parameters** dialog box.

21.5.5.2. Write Views Dialog Box

The **Write Views** dialog box allows you to save selected views to a view file. This feature allows you to transfer views between mesh files.

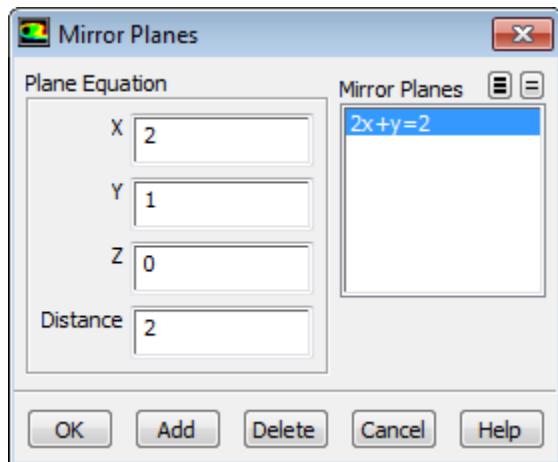
**Controls**

Views To Write

is a selectable list of the defined views. The selected views will be saved to a view file when you click **OK**.

21.5.5.3. Mirror Planes Dialog Box

The **Mirror Planes** dialog box allows you to define a symmetry plane for a non-symmetric domain for use with graphics.

**Controls****Plane Equation**

contains inputs for specifying the equation for the mirror plane:

$$AX + BY + CZ = Distance$$

Mirror Planes

contains a list of all mirror planes you have defined using this dialog box. Mirror planes that exist in the domain due to symmetry will not appear in this list, since they cannot be modified.

Add

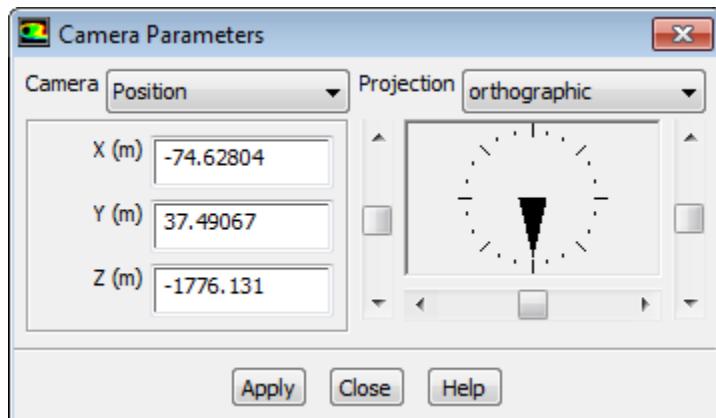
adds the plane defined by the **Plane Equation** to the **Mirror Planes** list.

Delete

deletes the plane(s) selected in the **Mirror Planes** list.

21.5.5.4. Camera Parameters Dialog Box

The **Camera Parameters** dialog box allows you to modify the "camera" through which you are viewing the graphics display.

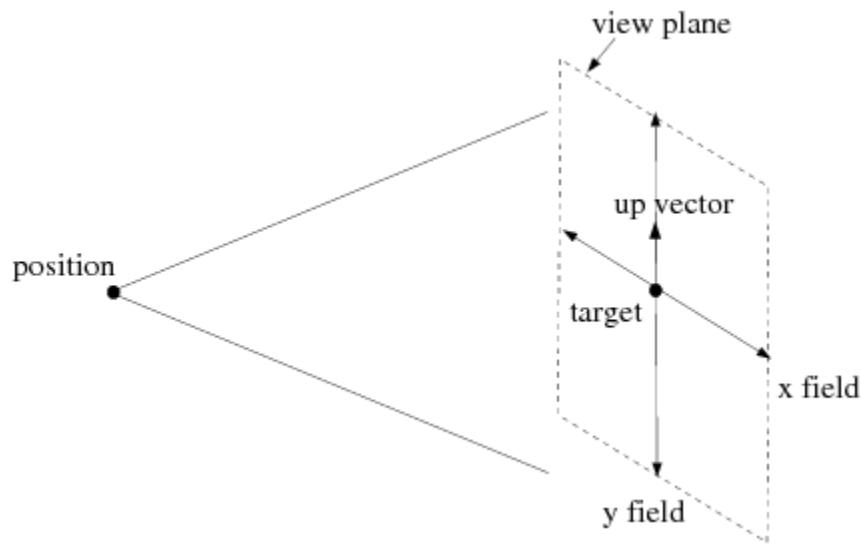


Controls

Camera

contains a drop-down list of the parameters that define the camera (**Position**, **Target**, **Up Vector**, and **Field**) and **X**, **Y**, and **Z** fields in which you can define the coordinates or field distances for the parameter selected in the drop-down list. [Figure 21.9: Camera Definition \(p. 620\)](#) illustrates the definition of the camera by these parameters.

Figure 21.9: Camera Definition



Projection

contains a drop-down list that allows you to select a **Perspective** or **Orthographic** view.

Dial

allows you to rotate the graphics display respectively.

- The dial controls rotation about the axis at the center of and perpendicular to the screen.

Sliders

allows you to scale the graphics display respectively.

- The slider on the scale to the left of the dial rotates the display about the horizontal axis at the center of the screen.

- The slider on the scale below the dial rotates the display about the vertical axis at the center of the screen.
- The slider on the scale to the right of the dial zooms in or out in the display.

Important

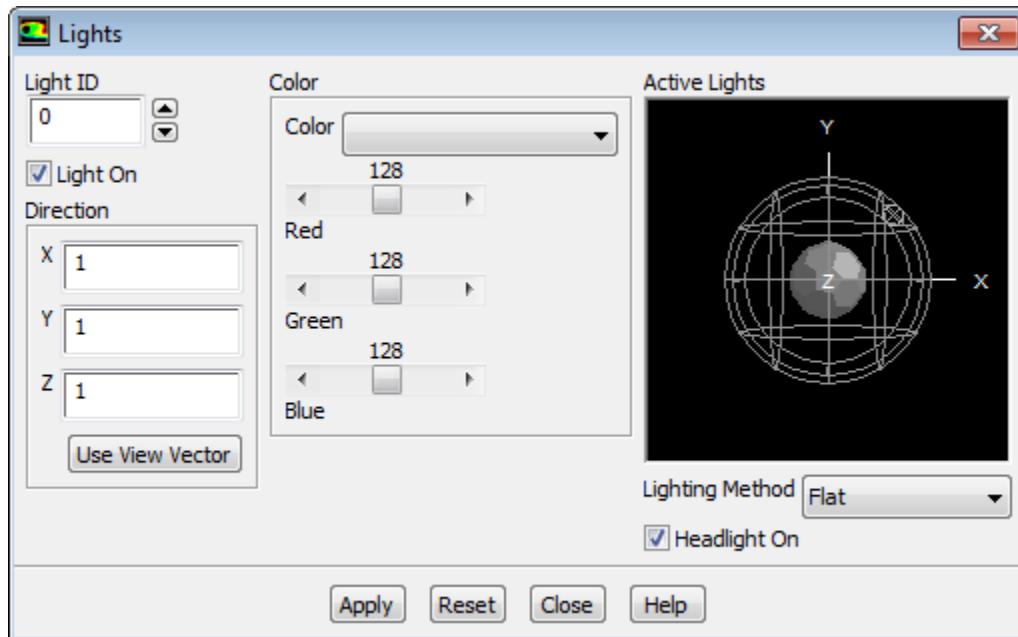
When using the sliders and dial to manipulate the view, select **All** in the **Display Options** dialog box, so that you can watch the display move interactively while you move the slider or the dial indicator.

21.5.6. Display/Lights...

The **Display/Lights...** menu item opens the [Lights Dialog Box \(p. 621\)](#).

21.5.6.1. Lights Dialog Box

The **Lights** dialog box provides an interactive mechanism for placing colored, directional lights in a scene.



Controls

Light ID

indicates the light that is being added, deleted, or modified. By default, light 1 is defined to be dark gray with a direction of (1,1,1).

Light On

indicates whether or not the light specified in **Light ID** is on or off. By turning off the **Light On** option for a particular light, you can remove this light from the display, while still retaining its definition. To add it to the display again, simply turn on the **Light On** button.

Direction

allows you to specify the direction of the light (the position on the unit sphere from which the light emanates) by entering the **X**, **Y**, and **Z** coordinates or by computing the coordinates based on the current view in the graphics window.

X,Y,Z

specify the direction of the light. For example, the direction (1,1,1) means that the rays from the light will be parallel to the vector from (1,1,1) to the origin.

Use View Vector

updates the **X,Y,Z** fields with the appropriate values for the current view in the active graphics window, and shines a light in that direction.

Instead of entering the **X,Y,Z** values for a light's direction vector, use the mouse to change the view in the graphics window so that the position in reference to the geometry is the position from which you would like a light to shine. Then click on the **Use View Vector** button to update the **X,Y,Z** fields with the appropriate values for the current position and update the graphics display with the new light direction.

This method is convenient if you know where you want a light to be, but you are not sure of the exact direction vector.

Color

allows you to specify the light color with sliders. You can create your desired color by increasing and decreasing the slider values for the colors **Red**, **Green**, and **Blue**. You can also enter a descriptive string (e.g., lavender) in the **Color** field.

Active Lights

shows the position and color of all defined directional lights, and allows you to change the position of a light.

All directional lights are assumed to be at infinity and pass through the unit sphere at the position shown. All light rays arriving at the scene from one light are parallel.

The colored markers on the surface of the sphere represent the color and direction of these *distant* lights. These lights point towards the center of the sphere (the origin, which is usually where the geometry is).

Lighting Method

specifies the method to be used in lighting interpolation: **Off**, **Flat**, **Gouraud**, or **Phong**.

Flat is the most basic method. There is no interpolation within the individual polygonal facets.

Gouraud and **Phong** have smoother gradations of color because they interpolate on each facet.

When **Off** is selected, lighting effects are disabled.

Headlight On

turns on a light that moves with the camera. This option is not especially relevant in the meshing mode of ANSYS Fluent.

Reset

resets the light definitions to their last saved state (i.e., the lighting that was in effect the last time you opened the dialog box or clicked on **Apply**).

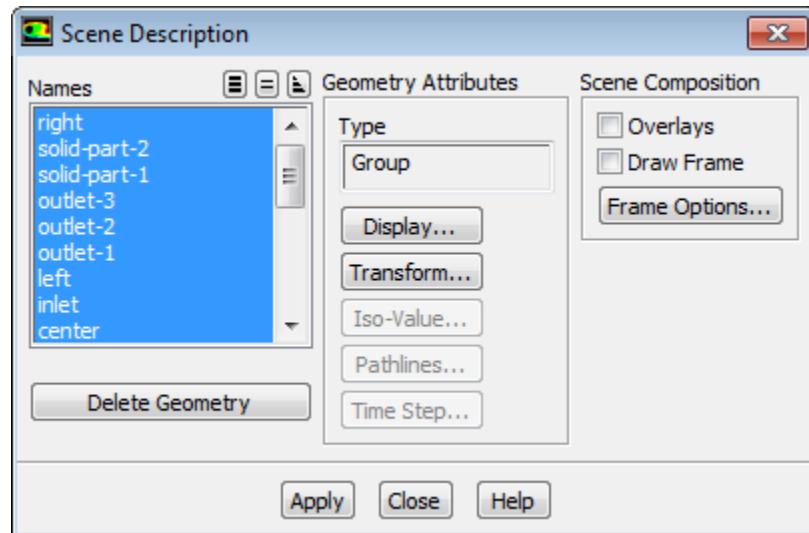
21.5.7. Display/Scene...

The **Display/Scene...** menu item opens the [Scene Description Dialog Box \(p. 623\)](#).

21.5.7.1. Scene Description Dialog Box

The **Scene Description** dialog box allows you to turn overlays on and off, select geometric entities in the display for modification or deleting, and open dialog boxes that control various characteristics of the selected entities.

This menu item becomes available after you display a grid in the graphics window. You cannot use the **Scene Description** dialog box to control XY plot and histogram displays.



Controls

Names

contains a list of the geometric entities that currently exist in the scene (including those that are presently invisible).

You can specify the entities to be manipulated by selecting names in this list. If you select more than one entity at a time, any operation (transformation, color specification, etc.) will apply to all the selected entities.

You can also select entities by clicking on them in the graphics display using the mouse probe button. The default is the right mouse button. For information about mouse button functions, see [Controlling the Mouse Buttons \(p. 391\)](#). To deselect a selected entity, click on its name in the **Names** list.

Geometry Attributes

contains information about the type of the selected geometric entity and push buttons which open dialog boxes for modifying the entity.

Type

reports the type of the selected entity. Possible types include grid and Group. This information is especially helpful when you need to distinguish two or more entities with the same name. When more than one entity is selected, the type displayed is Group.

Display...

opens the **Display Properties** dialog box, which allows you to change the color, visibility, and other properties for the selected entity.

Transform...

opens the **Transformations** dialog box, which allows you to translate, rotate, and scale the selected entity.

IsoValue...

is not available in the meshing mode in ANSYS Fluent.

Pathlines...

is not available in the meshing mode in ANSYS Fluent.

Time Step...

is not available in the meshing mode in ANSYS Fluent.

Scene Composition

contains controls for enabling overlays and bounding frames.

Overlays

activates the superimposition of a new geometry onto a currently displayed geometry.

Draw Frame

activates the display of a bounding frame in the graphics display.

Frame Options...

opens the **Bounding Frame** dialog box, in which you can define properties of the bounding frame.

Delete Geometry

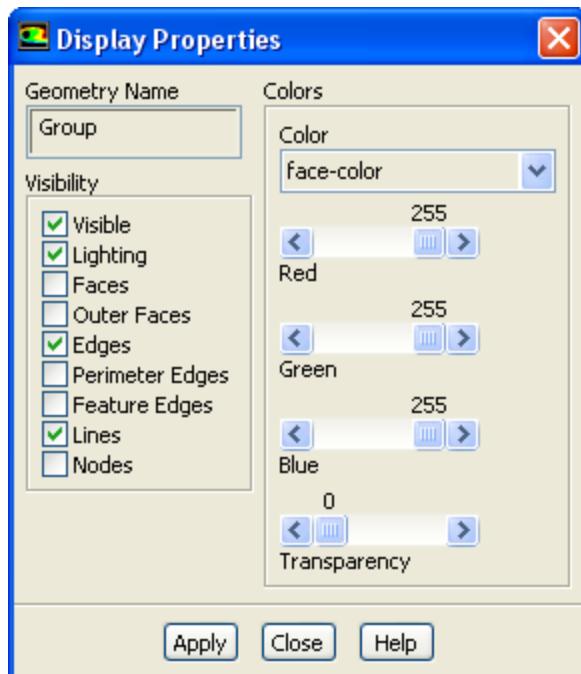
deletes the geometric entity that is currently selected in the **Names** list. The ability to delete individual entities is especially useful if you have overlays on and you generate an unwanted entity (e.g., if you generate contours of the wrong variable). You can simply delete the unwanted entity and continue your scene composition, instead of starting over from the beginning.

Apply

saves the status of **Overlays** and **Draw Frame**. When you turn **Overlays** or **Draw Frame** on or off, click **Apply** to see the effect of the change on subsequent display operations.

21.5.7.2. *Display Properties Dialog Box*

To modify the color, visibility, and other display properties for individual geometric entities in the graphics display, use the **Display Properties** dialog box. Click **Display...** in the **Scene Description** dialog box to open this dialog box.



Controls

Geometry Name

displays the name of the entity you selected for modification in the **Scene Description** dialog box.

Visibility

contains check buttons that control options related to the visibility of the selected entity.

Visible

toggles the visibility of the selected entity.

- If the option is enabled, the entity will be visible in the display.
- If the option is disabled, the entity will be invisible.

Lighting

turns the effect of lighting for the selected entity on or off. You can choose to have lighting affect only certain entities instead of all of them.

Faces

toggles the filled display of faces for the selected grid or surface entities. Enabling **Faces** on has the same effect as enabling the **Filled** option in the **Display Grid** dialog box.

Outer Faces

toggles the display of outer faces.

Edges

toggles the display of interior and exterior edges of the geometric entity.

Perimeter Edges

toggles the display of the outline of the geometric entity. This option has no effect on the display of grids.

Feature Edges

(not applicable for the meshing mode in ANSYS Fluent)

Lines

toggles the display of the lines (if any) in the geometric entity.

Nodes

toggles the display of the nodes (if any) in the geometric entity.

Colors

contains controls for setting face, edge, line, and node colors, and transparency for faces.

Color

specifies the face, edge, line, or node color for modification. When you turn on this button, the color scales below will show the current color specification, which you can modify by moving the sliders on the color scales.

Red, Green, Blue

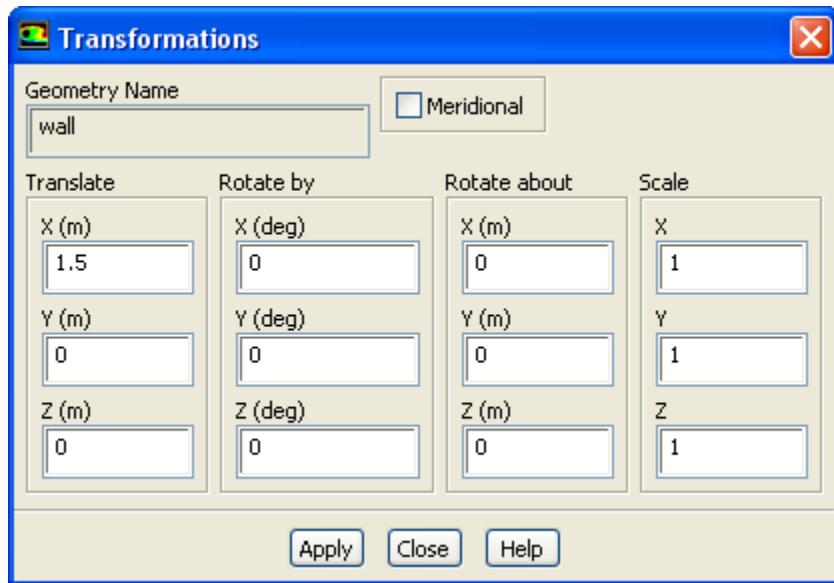
are color scales with which you can specify the RGB components of the face, edge or line color.

Transparency

sets the relative transparency of the selected entity. An entity with a transparency of 0 is opaque. An entity with a transparency of 100 is transparent.

21.5.7.3. Transformations Dialog Box

You can use the **Transformations** dialog box to translate, rotate, or scale individual entities in the graphics display. To open this dialog box, click the **Transform...** button in the **Scene Description** dialog box.

**Controls****Geometry Name**

displays the name of the entities you selected for modification in the **Scene Description** dialog box.

Meridional

enables the display of the meridional view. This option is especially useful in turbomachinery applications.

Translate

contains **X**, **Y**, and **Z** real number fields in which you can enter the distance by which to translate the selected entities in each direction. Translations are not cumulative, so you can easily return to a known state. To return to the original position, enter 0 in all three real number fields.

Rotate By

contains **X**, **Y**, and **Z** integer number fields in which you can enter the number of degrees by which to rotate the selected entity about each axis. You can enter any value between -360 and 360.

Rotations are not cumulative, so you can easily return to a known state. To return to the original position, enter 0 in all three integer number fields.

Rotate About

specifies the point about which to rotate the entity.

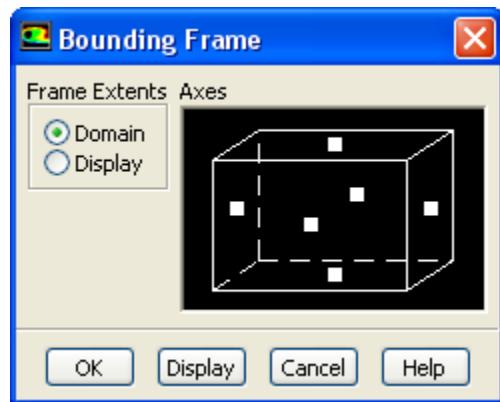
Scale

contains **X**, **Y**, and **Z** real number fields in which you can enter the amount by which to scale the selected entity in each direction.

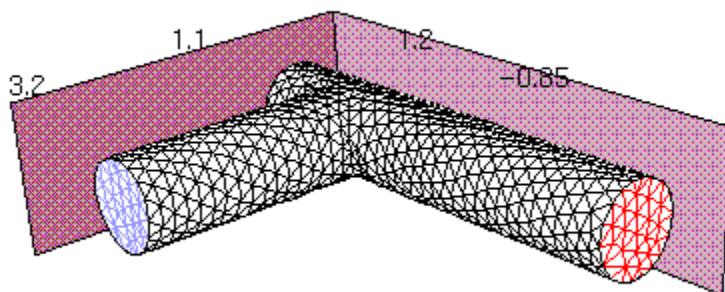
- To avoid distortion of the shape of the entity, be sure to specify the same value for all three entries. Scaling is not cumulative, so you can easily return to a known state.
- To return the entity to its original size, enter 1 in all three real number fields.

21.5.7.4. Bounding Frame Dialog Box

To open this dialog box, click on the **Frame Options...** push button in the **Scene Description** dialog box.



The **Bounding Frame** dialog box allows you to add a bounding frame with optional measure markings to the display of the domain (e.g., [Figure 21.10: Graphics Display with Bounding Frame \(p. 628\)](#)).

Figure 21.10: Graphics Display with Bounding Frame

Controls

Frame Extents

indicates the extents of the bounding frame.

Domain

specifies that the frame should encompass the domain extents.

Display

specifies that the frame should encompass the portion of the domain that is shown in the display.

Axes

contains controls for specifying the frame boundaries and measurements.

Display

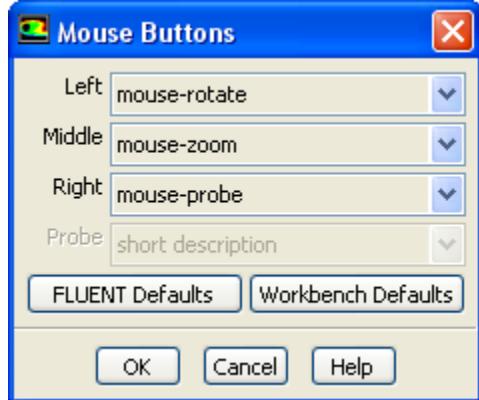
updates the active graphics window with the current frame settings.

21.5.8. Display/Mouse Buttons...

The **Display/Mouse Buttons...** menu item opens the [Mouse Buttons Dialog Box \(p. 628\)](#).

21.5.8.1. Mouse Buttons Dialog Box

The mouse button is used to set the actions taken when you click the mouse button in a graphics window. Clicking any mouse button in a graphics window will make that window the active window.



Each mouse button can be associated with a different function. The predefined functions include **mouse-rotate**, **mouse-dolly**, **mouse-zoom**, **mouse-roll-zoom**, **mouse-probe**, and **mouse-annotate**. These functions apply only to the graphics windows; they are not active when an XY plot or histogram is displayed.

Clicking any mouse button in a graphics window will make that window the active window.

Controls

Left

sets the function associated with the left button. The default setting is **mouse-rotate**.

Middle

sets the function associated with the middle button. The default setting is **mouse-zoom**.

Right

sets the function associated with the right button. This is not used on a two button mouse. The default setting is **mouse-probe**.

Fluent Defaults

sets the mouse button functions according to ANSYS Fluent defaults.

Workbench Defaults

sets the mouse button functions according to Workbench defaults.

OK

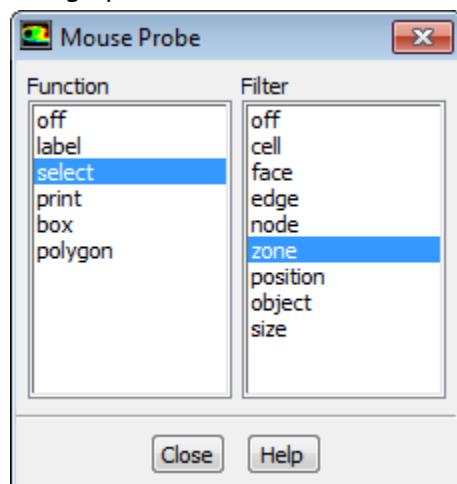
saves the current settings and closes the dialog box. The appropriate function will be executed when a mouse button is subsequently clicked in a graphics window. You do not have to redraw the graphics window for the new button functions to take effect.

21.5.9. Display/Mouse Probe...

The **Display/Mouse Probe...** menu item opens the [Mouse Probe Dialog Box \(p. 629\)](#).

21.5.9.1. Mouse Probe Dialog Box

The **Mouse Probe** dialog box is used to set the actions taken when the mouse probe button is clicked in a graphics window.



Controls

Function

selects the function to be performed when the mouse probe is clicked in the graphics window. The available functions are:

off

disables the mouse probe.

label

displays the label for the selected entity in the graphics window.

select

saves the selected entity in a list that can be used in conjunction with the boundary modification functions (see [Modifying the Boundary Mesh \(p. 128\)](#)).

print

prints information about the selected entity in the text window (see [Printing Grid Information \(p. 406\)](#)).

box

enables the selection of a group of entities within a box, to be used in conjunction with the boundary modification functions (see [Modifying the Boundary Mesh \(p. 128\)](#)). To define the selection box, click the mouse probe button at one corner of the region to be selected, drag the mouse to the opposite corner, and release the mouse probe button.

polygon

enables the selection of a group of entities within a polygonal region, to be used in conjunction with the boundary modification functions (see [Modifying the Boundary Mesh \(p. 128\)](#)). To define the selection polygon, click the mouse probe button at one vertex of the polygonal region to be selected, and use the left mouse button to successively select each of the remaining vertices.

Click the mouse probe button again (anywhere in the graphics window) to complete the polygon definition.

Filter

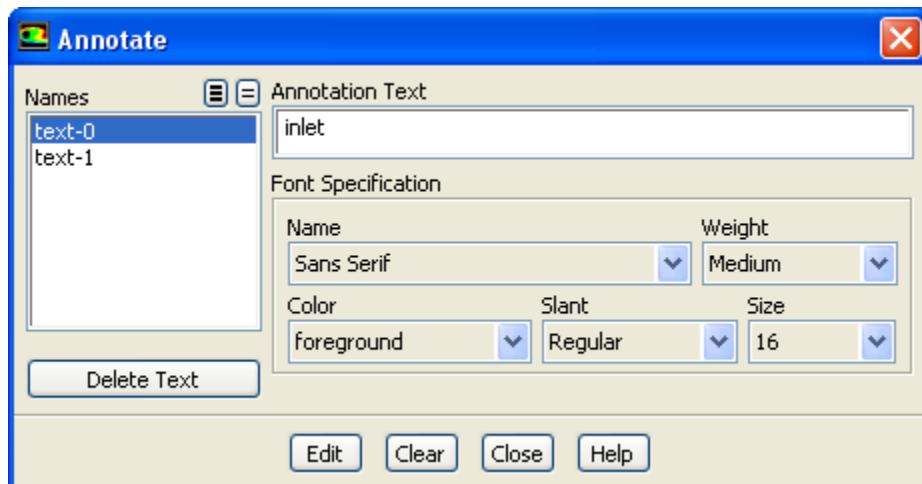
indicates what the mouse probe button is to select. The available filters are **off**, **cell**, **face**, **edge**, **node**, **zone**, **position**, **object**, and **size**. If **off** is chosen, when a selection is made it is first checked to see if it is a cell, then a face, an edge, and so on. When the **node** filter is used, if a cell or face is selected the node closest to the selection point is picked. Thus nodes do not have to be displayed to be picked.

21.5.10. Display/Annotate...

The **Display/Annotate...** menu item opens the [Annotate Dialog Box \(p. 630\)](#).

21.5.10.1. Annotate Dialog Box

You can use the **Annotate** dialog box to add text with optional attachment lines to the graphics windows. It can also be used to remove all annotations from the active graphics window.

**Names**

contains a selectable list of all annotation text strings that have been defined. You can choose a text string to be deleted or edited.

Delete Text

deletes the text strings selected in the **Names** list from the display.

Annotation Text

contains the annotation text string you wish to add, or the annotation text string for the item selected in the **Names** list.

Font Specification

contains controls for defining or modifying the font in the annotation text string.

Name

contains a drop-down list of various font styles.

Weight

contains a drop-down list from which you can select **Medium** or **Bold**.

Size

contains a drop-down list of font sizes (in points).

Color

contains a drop-down list of colors that can be used for the text.

Slant

contains a drop-down list from which you can select **Regular**, **Italic**, or **Oblique** as the slant type.

Add

adds the current **Annotation Text** to the active graphics window. A dialog will prompt you to select a screen location using the mouse-probe button. For more information on setting the mouse buttons see [Controlling the Mouse Buttons \(p. 391\)](#).

Edit

updates the edited text in the active graphics window. This button will replace the **Add** button when you are editing an existing text string from the **Names** list.

Clear

removes all annotation text and attachment lines from the active graphics window.

21.5.11. Display/Selection Helper...

The **Display/Selection Helper...** menu item opens the Zone Selection Helper Dialog Box (p. 632).

21.5.11.1. Zone Selection Helper Dialog Box

The **Zone Selection Helper** dialog box allows you to find and select zones based on a certain pattern in their names, or by the number of zone entities present in them, or using the minimum or maximum face zone area.

**Controls****Options**

contains options to choose zone selection criteria.

By Zone Name

specifies the use of zone selection criteria based on zone name pattern.

By Zone Entities

specifies the use of zone selection criteria based on the number of selected zone entities present in them.

By Zone Area

specifies the use of zone selection criteria based on the face zone area.

Max Zone Area

indicates that zone selection will be based on the maximum face zone area.

Min Zone Area

indicates that zone selection will be based on the minimum face zone area.

Zone Type

contains a drop-down list of types of zones (**face**, **node**, **edge**, **cell**). This is available for the selection by zone name or by zone entities.

Zone Name Pattern

specifies the pattern to look for in the names of zones of the selected **Zone Type** when the selection is by zone name. Press RETURN after you enter the pattern.

Max. Zone Entities

specifies the maximum number of zone entities to look for in the zones of the selected **Zone Type** when the selection is by zone entities. Press RETURN after you enter the value.

Face Zone Area

specifies the face zone area when the selection is by zone area. The value specified is indicative of the maximum or minimum face zone area as selected. Press RETURN after you enter the value.

Matching Zones

contains a list of the zones of the specified **Zone Type** that match the specified **Zone Name Pattern** or **Max. Zone Entities**.

Select

selects the zones in the **Matching Zones** list (in all open dialog boxes).

Deselect

deselects the zones in the **Matching Zones** list (in all open dialog boxes).

21.5.12. Display/Controls...

The **Display/Controls...** menu item opens the **Controls** dialog box to set file specific variables called tgvars.

21.5.12.1. Controls Dialog Box



The **Controls** dialog box is used to set file specific variables called tgvars, in your mesh data file.

The dialog box has two components: the **Categories** drop-down list and the actual variables associated with that category.

Select the desired Category, and then enter the desired default values. Click **Apply** to update your file with the new values.

21.6. Report Menu

21.6.1. Report/Mesh Size...

The **Report/Mesh Size...** menu item opens the Report Mesh Size Dialog Box (p. 634).

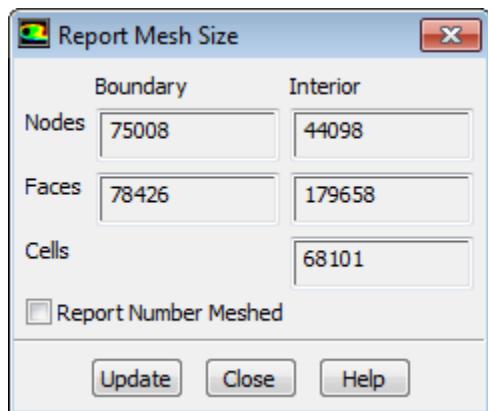
21.6.1.1. Report Mesh Size Dialog Box

The **Report Mesh Size** dialog box reports the number of nodes, faces, and cells. Nodes and faces are grouped into those defining the boundaries and those used inside cell zones.

If the generation of the initial mesh fails, you can determine how many boundary nodes and faces were not meshed by enabling **Report Number Meshed** in the **Report Mesh Size** dialog box. The total number of boundary nodes and faces will be reported along with the number that were meshed. The headings **Boundary** and **Interior** will be replaced by **Total** and **Meshed**, respectively when you use this option and the reported information will appear in the appropriate boxes.

Important

If you used domains to generate the mesh or group zones for reporting (as described in [Using Domains to Group and Mesh Boundary Faces \(p. 368\)](#)), the report will apply only to the active domain.



Controls

Boundary

indicates the number of boundary nodes and boundary faces.

Interior

indicates the number of interior nodes, faces, and cells.

Total

indicates the total number of nodes and faces when the **Report Number Meshed** option is enabled.

Meshed

indicates the number of nodes and faces that have been meshed when the **Report Number Meshed** option is enabled.

Report Number Meshed

toggles the reporting of the number of nodes and faces that have been meshed.

Update

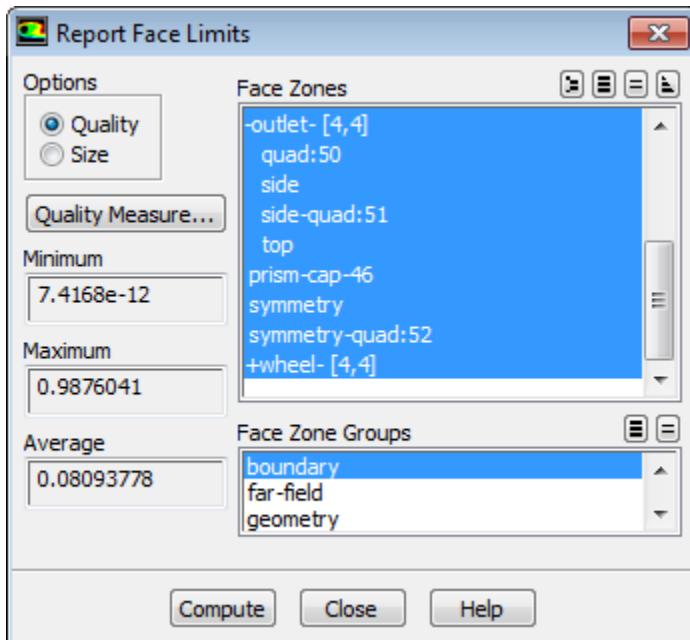
updates the reported values.

21.6.2. Report/Face Limits...

The **Report/Face Limits...** menu item opens the [Report Face Limits Dialog Box \(p. 635\)](#).

21.6.2.1. Report Face Limits Dialog Box

The **Report Face Limits** dialog box reports on the size or quality limits of selected face zones. The minimum, maximum, and average values are reported. If you used domains to generate the mesh or group zones for reporting (as described in [Using Domains to Group and Mesh Boundary Faces \(p. 368\)](#)), you can report face limits only for the face zones that are in the active domain.



Controls

Options

specifies either **Quality** or **Size** to be computed. Skewness is the default measure of quality.

Quality Measure...

opens the [Quality Measure](#) dialog box, in which you can select the measure of quality to be reported (skewness, aspect ratio, etc.).

Minimum/Maximum/Average

report the computed values for size or quality.

Face Zones

contains a list from which you can select the zones for the report.

Face Zone Groups

contains a list of face zone types. If you select a face type from this list (e.g., **inlet**), all face zones of that type (for this example, all **pressure-inlet** and **velocity-inlet** boundaries) will be selected in the **Face Zones** list. This allows you to easily select all face zones of a certain type without selecting each zone individually. Select multiple types in the **Face Zone Groups** list to select all zones of several different types (e.g., **inlet** and **outlet**).

Compute

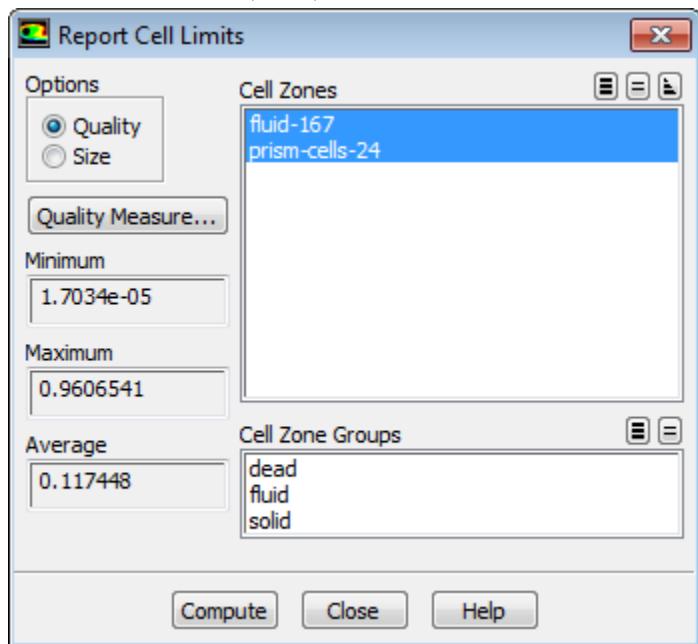
calculates the **Minimum**, **Maximum**, and **Average** values for the selected option (size or quality) on the selected face zones.

21.6.3. Report/Cell Limits...

The **Report/Cell Limits...** menu item opens the **Report Cell Limits Dialog Box** (p. 636).

21.6.3.1. Report Cell Limits Dialog Box

The **Report Cell Limits** dialog box reports the size or quality limits of the selected cell zones. The minimum, maximum, and average values are reported. If you used domains to generate the mesh or group zones for reporting (as described in [Using Domains to Group and Mesh Boundary Faces \(p. 368\)](#)), the report will apply only to the active domain.

**Controls****Options**

specifies either **Quality** or **Size** to be computed. Skewness is the default measure of quality.

Quality Measure...

opens the **Quality Measure** dialog box, in which you can select the measure of quality to be reported (skewness, aspect ratio, change in size, etc.).

Minimum/Maximum/Average

report the computed values for size or quality.

Cell Zones

contains a list from which you can select the zones of interest.

Cell Zone Groups

contains a list of cell zone types. If you select a zone type from this list (e.g., **fluid**), all cell zones of that type will be selected in the **Cell Zones** list. This shortcut allows you to easily select all zones of a certain type without having to select each zone individually. You can select multiple types in the **Cell Zone Groups** list to select all zones of several different types (e.g., **fluid** and **solid**).

Compute

calculates the **Minimum**, **Maximum**, and **Average** values for the selected option (size or quality).

21.6.4. Report/Boundary Cell Limits...

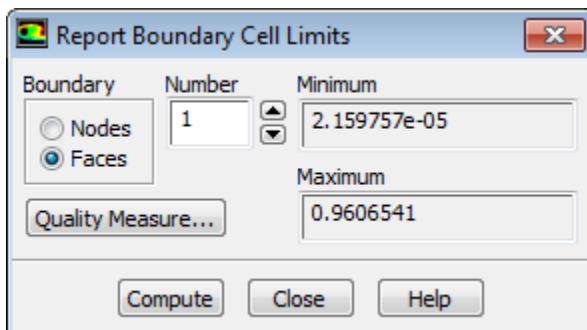
The **Report/Boundary Cell Limits...** menu item opens the **Report Boundary Cell Limits Dialog Box** (p. 637).

21.6.4.1. Report Boundary Cell Limits Dialog Box

The **Report Boundary Cell Limits** dialog box reports the quality of cells containing a specified number of boundary faces. The minimum and maximum skewness, aspect ratio, or size change values are reported. Though this dialog box can be used to check all quality measures and all boundary cell shapes, it is used primarily to check the skewness of tetrahedral boundary cells.

Important

If you have used domains to generate the mesh or group zones for reporting (as described in [Using Domains to Group and Mesh Boundary Faces \(p. 368\)](#)), the report will apply only to the active domain.



Controls

Boundary

allows you to specify whether you want to set the number of boundary nodes or faces. Select **Nodes** or **Faces**, and specify the number of nodes/faces in the **Number** field. To obtain information about cells that touch the boundary (but do not actually have any boundary faces), specify the number of boundary nodes instead of the number of boundary faces.

Number

sets the number of boundary nodes or faces (depending on the selection in the **Boundary** list). The quality of cells containing this number of boundary nodes or faces will be reported when you click on **Compute**.

Quality Measure...

opens the **Quality Measure** dialog box, in which you can select the measure of quality to be reported (skewness, aspect ratio, etc.).

Minimum

shows the reported minimum quality measurement (skewness, by default) of the cells with the specified number of boundary nodes or faces.

Maximum

shows the reported maximum quality measurement (skewness, by default) of the cells with the specified number of boundary nodes or faces.

Compute

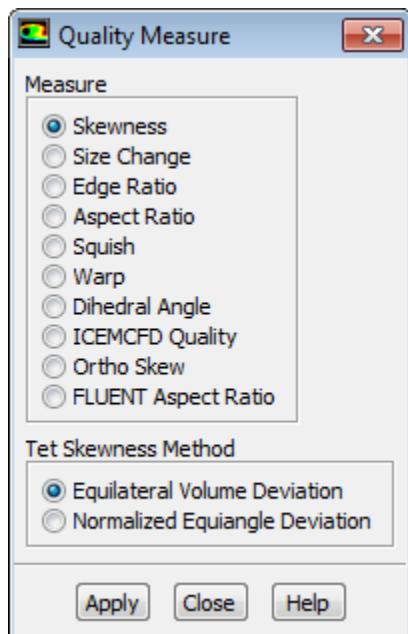
calculates the minimum and maximum quality measurement for boundary cells with the specified number of boundary nodes or faces.

21.6.5. Report/Quality Measure...

The **Report/Quality Measure...** menu item opens the [Quality Measure Dialog Box \(p. 638\)](#).

21.6.5.1. Quality Measure Dialog Box

The **Quality Measure** dialog box allows you to specify the measure used to evaluate the quality of the mesh (skewness, change in size, edge ratio, aspect ratio, squish, warp, dihedral angle, ICEM CFD quality, or orthoskew). For tetrahedral cells, you can also select the method to be used to report and display skewness.

**Controls****Measure**

contains the available measures that you can use to check the quality of your mesh. See [Mesh Quality \(p. 399\)](#) for details.

Skewness

specifies that skewness is the quality measure to be reported or displayed.

Size Change

specifies that change in face or cell size is the quality measure to be reported or displayed.

Edge Ratio

specifies that edge ratio is the quality measure to be reported or displayed.

Aspect Ratio

specifies that aspect ratio is the quality measure to be reported or displayed.

Squish

specifies that squish is the quality measure to be reported or displayed.

Warp

specifies that warp is the quality measure to be reported or displayed.

Dihedral Angle

highlights the faces having the dihedral angle between them within the range specified in the **Cell Quality Range** box of the **Display Grid** dialog box.

ICEM CFD quality

specifies that the ICEM CFD quality measure is to be reported or displayed.

Ortho Skew

specifies that the ortho skew measure is to be reported or displayed.

Fluent Aspect Ratio

specifies that the Fluent aspect ratio measure is to be reported or displayed.

Tet Skewness Method

indicates which method you want to use to report and display the skewness of tetrahedra.

Important

The skewness method you choose will only affect reports and displays based on skewness. The tetrahedral meshing procedure will always be based on the equilateral volume skewness method.

Equilateral Volume Deviation

selects the skewness method that is based on the volume of an ideal (equilateral) face or cell.

Normalized Equiangle Deviation

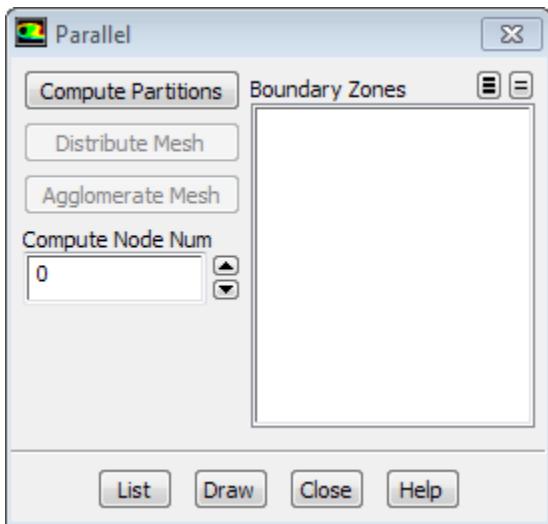
selects the skewness method that is based on the deviation from a normalized equiangular angle. This method is always used for pyramids and prisms.

21.7. Parallel Menu

The options in the **Parallel** menu are used to manage parallel processes. See [Parallel Processing in Meshing Mode](#) for more information.

21.7.1. Parallel Dialog Box

Click **Parallel > Partitions** to open the **Parallel** dialog box.



Controls

Compute Partitions

All the **Boundary Zones** are first assigned to closed regions, and then these regions are assigned to the available partitions based on location with some consideration for load balancing. The number of available partitions is the smaller of the number of regions and the number of meshing processes specified in the Fluent Launcher.

Distribute Mesh

Mesh data is distributed to the compute nodes based on the computed partitions.

Agglomerate Mesh

Recombine all mesh data into a single partition on compute node 0. This is useful in case some operation not supported in parallel is required.

Compute Node Num

Only the **Boundary Zones** assigned to the indicated compute node are displayed.

List

Select from the available **Boundary Zones** and click to display statistics in the console.

Draw

Select from the available **Boundary Zones** and click to display them in the graphics area.

21.8. View Menu

21.8.1. View/Toolbars

The **View/Toolbars** menu item allows you to toggle the visibility of the toolbars in the application window.

21.8.2. View/Navigation Pane

The **View/Navigation Pane** menu item allows you to toggle the visibility of the navigation pane in the application window.

21.8.3. View/Task Page

The **View/Task Page** menu item allows you to toggle the visibility of the task pages in the application window.

21.8.4. View/Graphics Window

The **View/Graphics Window** menu item allows you to toggle the visibility of the graphics window in the application window. This option is only visible when the graphics window is embedded in the application.

21.8.5. View/Embed Graphics Window

The **View/Embed Graphics Window** menu item allows you to toggle between attaching and detaching the graphics window within the application.

21.8.6. View>Show All

The **View>Show All** menu item allows you to display the toolbars, the navigation pane, the task pages, and the graphics window, if one or more of them is currently not visible.

21.8.7. View>Show Only Console

The **View>Show Only Console** menu item allows you to display only the console window in the application.

21.8.8. View/Graphics Window Layout

The **View/Graphics Window Layout** menu item displays several layout options for use with multiple graphics windows. This option is only visible when the graphics window is embedded in the application.

Multiple graphics windows should not be used in meshing mode.

21.8.9. View/Save Layout

The **View/Save Layout** menu item is used to save the current arrangement of dialog boxes and graphics windows. The positions of these items on your screen will be written to a `.cxl` layout file in your home folder.

21.9. Help Menu

21.9.1. Help/User's Guide Contents...

The **Help/User's Guide Contents...** menu item opens the help viewer to the **Table of Contents** page of the Meshing User's Guide. See [Using the GUI Help System \(p. 41\)](#) for details.

21.9.2. Help/PDF

The **Help/PDF** menu item displays a submenu that allows you access to the Meshing User's Guide and other manuals in printable format (PDF).

21.9.3. Help/Context Sensitive Help...

The **Help/Context-Sensitive** Help menu item allows you to get help for a specific topic represented by a pull-down menu item, a dialog box, or another part of the graphical user interface. When you select **Context-Sensitive Help**, the screen cursor will change to the shape of a question mark. Now you can select an item from a pull-down menu or click another component of the graphical user interface. The web browser will open directly to the corresponding section of the User's Guide.

Important

Context-sensitive help is not available on Windows systems.

21.9.4. Help/Using Help...

The **Help/Using Help...** menu item opens the help viewer directly to the section on using the on-line help facility. See [Using the GUI Help System \(p. 41\)](#) for details. For additional information on using the help system, see [Using Help](#).

21.9.5. Help/Online Technical Resources...

The **Help/Online Technical Resources...** menu item opens the web browser to the ANSYS Customer Portal web site where you can search for solutions and log requests. See [Using the GUI Help System \(p. 41\)](#) for details.

21.9.6. Help/License Usage...

The **Help/License Usage...** menu item provides license information. See [Using the GUI Help System \(p. 41\)](#) for details.

21.9.7. Help/Version...

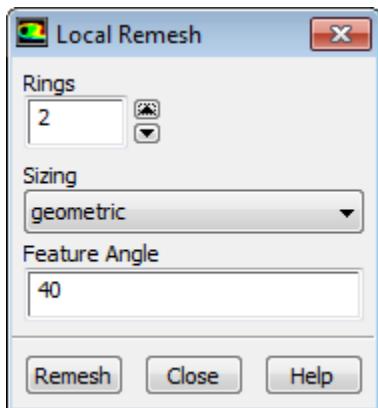
The **Help/Version...** menu item shows information about the version and release you are running.

21.10. Hot Key Activated Dialog Boxes

Several Hot Keys will open dialog boxes for additional input.

21.10.1. Local Remesh Dialog Box

The **Local Remesh** dialog box contains options for remeshing faces based on selections in the graphics window. Select the faces to be remeshed and use the hot-key **Ctrl+Shift+r** to open the **Local Remesh** dialog box.



Controls

Rings

specifies the number of radial layers of faces to be remeshed.

Sizing

allows you to select the size source for remeshing.

size-function

remeshes the faces based on the current size functions/size field.

geometric

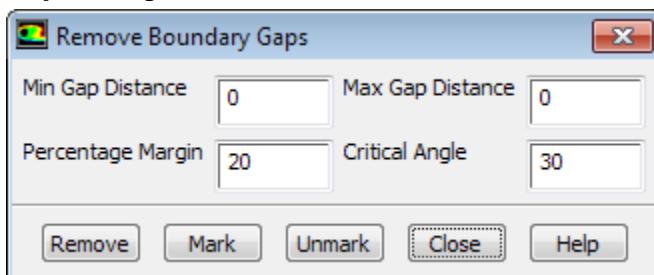
remeshes the faces based on the face boundary.

Feature Angle

specifies the feature angle to be preserved while remeshing the selected faces.

21.10.2. Remove Boundary Gaps Dialog Box

The **Remove Boundary Gaps** dialog box contains options for removing gaps between the source and target zones selected in the graphics window. Select the source zone and use the hot-key **Ctrl+s** to set the source zone. Select the target zone and use the hot-key **Ctrl+k** to open the **Remove Boundary Gaps** dialog box.



Controls

Min. Gap Distance

specifies the minimum distance across the gap which will be considered for gap removal.

Max. Gap Distance

specifies the maximum distance across the gap which will be considered for gap removal.

Percentage Margin

specifies a tolerance margin for determining the gap distance.

Critical Angle

specifies the angle between the opposite faces constituting the gap to be removed.

Remove

removes the gap between the source and target zones selected.

Mark

marks the faces that will be used for projection for the gap removal operation.

Unmark

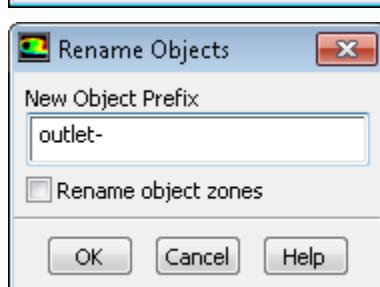
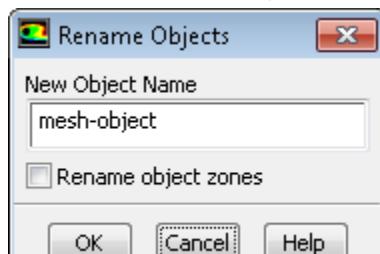
unmarks the faces marked for projection for the gap removal operation.

21.10.3. Rename Objects/Zones Dialog Box

Rename Objects Dialog Box

The **Rename Objects** dialog box is used to change the name of objects based on selections in the graphics window. Select one or more objects and then use the hot-key **Ctrl+Shift+n** to open the **Rename Objects** dialog box.

If a single object is selected, you can specify a new object name. If multiple objects are selected, you can specify a common prefix for the objects selected. This will allow you to easily view the objects with the same prefix using the tree view button for selection lists.

**Controls****New Object Name**

specifies the name for the object selected.

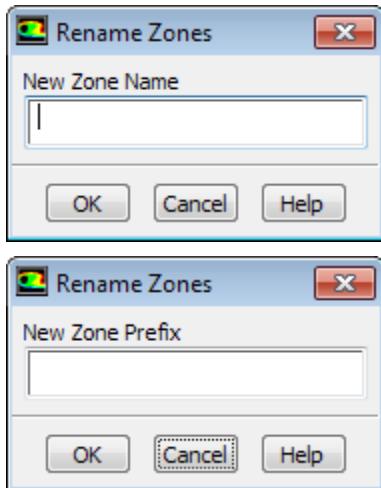
New Object Prefix

specifies the common prefix for the objects selected.

Rename object zones

allows you to optionally rename the face and edge zones comprising the object, based on the object name.

Rename Zones Dialog Box



The **Rename Zones** dialog box is used to change the name of one or more zones based on selections in the graphics window. This is helpful when working with hierarchical lists.

Select one or more zones, then press **Ctrl+Shift+n** to activate the **Rename Zones** dialog box.

Note

If you select one zone/object, this will simply change its name.

If you select multiple zones/objects, this will allow you to add a prefix to all their names to arrange them in a hierarchical list.

Appendix A. Importing Boundary and Volume Meshes

The volume mesh generation scheme requires sets of triangular and/or quadrilateral elements defining the boundaries of the computational domain. In addition to the basic capability provided in GAMBIT, data interfaces have been written to popular CAD/CAE software packages for use by ANSYS Fluent.

A.1. GAMBIT Meshes

A.2. TetraMesher Volume Mesh

A.3. Meshes from Third-Party CAD Packages

A.1. GAMBIT Meshes

GAMBIT can create both surface and volume meshes. See the GAMBIT Modeling Guide for details.

A.2. TetraMesher Volume Mesh

ICEM CFD Engineering writes a RAMPANT file from TetraMesher. This file can be read using the **File/Read/Mesh...** menu item or the `file/read-mesh` text command. TetraMesher generates tetrahedral volume grids using a recursive subdivision octree scheme that interfaces with ICEM CFD.

A.3. Meshes from Third-Party CAD Packages

A single filter allows you to convert files created by several finite-element packages to the grid file format used by ANSYS Fluent. This filter, `fe2ram`, allows you to convert surface or volume meshes from ANSYS, I-deas, NASTRAN, PATRAN, or other packages. ARIES files can be converted if they are first saved as ANSYS Prep7 files, as described in [ARIES Files \(p. 651\)](#).

A.3.1. Using the fe2ram Filter to Convert Files

A.3.2. I-deas Universal Files

A.3.3. PATRAN Neutral Files

A.3.4. ANSYS Files

A.3.5. ARIES Files

A.3.6. NASTRAN Files

A.3.1. Using the fe2ram Filter to Convert Files

When you import one of these types of grid files using the menu items in the **File/Import** menu, the changes are transparent and you can follow the instructions provided in [Importing Files \(p. 69\)](#). If you choose to convert the file manually before reading it, enter the following command:

```
utility fe2ram [dimension] read-format [merge] [zoning] [write-format] input-file output-file
```

The items enclosed in square brackets are optional.

`dimension` indicates the dimension of the dataset. Replace `dimension` by `-d2` to indicate that the grid is two-dimensional. For a 3D grid, you need not enter anything for `dimension`, because 3D is the default. For a surface mesh, replace `dimension` by `-surface`.

`read-format` indicates the format of the file you wish to convert. Replace `read-format` by `-tANSYS` for an ANSYS file, `-tIDEAS` for an I-deas file, `-tNASTRAN` for a NASTRAN file, or `-tPATRAN` for a PATRAN file. For a list of conversion capabilities from other CAD packages, type utility `fe2ram -cl -help`.

`merge` indicates the grid tolerance. The default is 10^{-6} ($1.0e-06$), and you can reset the tolerance to this value by replacing `merge` by `-m`. To set another tolerance value, replace `merge` by `-mTOLERANCE`, where `TOLERANCE` is an appropriate real number value.

`zoning` indicates how zones were identified in the CAD package. Replace `zoning` by `-zID` for a grid that was zoned by property ID's, or `-zNONE` to ignore all zone groupings. For a grid zoned by group, do not enter anything for `zoning`, because zoning by groups is the default.

`write-format` indicates the output format for the file you wish to convert. Replace `write-format` by `-oFIDAP7` to write the grid in **FIDAP** format. To write the grid for use in ANSYS Fluent, do not enter anything for `write-format`, because it is the default selection.

input-file and *output-file* are the names of the original file and the file to which you want to write the converted grid information, respectively.

For example, to convert the 2D I-deas volume mesh file `sample.unv` to an output file called `sample.msh`, enter the following command:

```
utility fe2ram -d2 -tIDEAS sample.unv sample.msh
```

After the output file has been written, you can read it using the **File/Read/Mesh...** menu item in the meshing mode of ANSYS Fluent. For volume meshes, the resulting output file can also be read into the solution mode of ANSYS Fluent.

Important

All boundary types are considered as wall zones. You can set the appropriate boundary types in the meshing mode or in the solution mode.

Note

The `fe2ram` utility supports VRML files from VRML version 1.0.

A.3.2. I-deas Universal Files

For I-deas surface meshes, the filter reads triangular and quadrilateral elements that define the boundaries of the domain and have been grouped within I-deas to create zones. For volume meshes, the filter reads a 2D or 3D mesh that has its boundary nodes or 2D boundary elements appropriately grouped to create boundary zones. Do not include nodes and boundary elements in the same group. All boundary zones will be considered wall zones; you can set the appropriate boundary types in the meshing mode or in the solution mode. See [Using the fe2ram Filter to Convert Files \(p. 647\)](#) for further details.

A.3.2.1. Recognized I-deas Datasets

The following Universal file datasets are recognized by the grid filter:

Node Coordinates

dataset number 15, 781, 2411

Elements

dataset number 780 or 2412

Permanent Groups

dataset number 752, 2417, 2429, 2430, 2432, 2435

Since ANSYS Fluent uses linear elements, you should use linear elements to generate the grid inside the mesh areas. If parabolic elements exist in the dataset, the filter ignores the mid side nodes. This assumption is valid if the edges of the element are near linear. However, if this is not the case, an incorrect topology may result from this assumption. For example, in regions of high curvature the parabolic element may look much different than the linear element.

For volume meshes, note that mesh area/mesh volume datasets are *not* recognized. This implies that writing multiple mesh areas/mesh volumes to a single Universal file may confuse `fe2ram` or ANSYS Fluent.

A.3.2.2. Grouping Elements to Create Zones for a Surface Mesh

The Group command in I-deas is used to create the boundary zones needed by ANSYS Fluent. All faces grouped together are listed together in the output as a single zone. In ANSYS Fluent, boundary conditions are set on a per-zone basis.

One technique is to generate groups automatically based on mesh areas—i.e., every mesh area will be a different zone. Although this method may generate a large number of zones, the zones can be merged in the meshing mode or in the solution mode of ANSYS Fluent. Another technique is to create a group of elements related to a given mesh area manually. This allows you to select multiple mesh areas for one group.

A.3.2.3. Grouping Nodes to Create Zones for a Volume Mesh

The Group command is used in I-deas to create the boundary zones needed by ANSYS Fluent. All nodes grouped together are listed together in the output as a single zone. It is important not to group nodes of internal faces with nodes of boundary faces.

One technique is to generate groups automatically based on curves or mesh areas—i.e., every curve or mesh area will be a different zone in ANSYS Fluent. Another technique is to create the groups manually, generating groups consisting of all nodes related to a given mesh area (3D).

A.3.2.4. Periodic Boundaries

In general, it is difficult to generate a valid grid with periodic boundaries in I-deas. However, a special feature exists in the meshing mode in ANSYS Fluent that allows you to generate a grid in a domain with periodic boundaries. See [Creating Periodic Boundaries \(p. 171\)](#) for further details.

A.3.2.5. Deleting Duplicate Nodes

I-deas often generates duplicate nodes in the process of creating triangular elements. These must be removed by using either the remove coincident node command in I-deas or the **Merge** button in the **Merge Boundary Nodes** dialog box (or the `/boundary/merge-duplicates` text command). This node merging process is usually faster in the meshing mode in ANSYS Fluent but more visual in I-deas.

A.3.3. PATRAN Neutral Files

For PATRAN surface meshes, the filter reads triangular and quadrilateral linear elements that define the boundaries of the domain and have been grouped by named component or identified by property ID's within PATRAN to create zones. For volume meshes, the filter reads a 2D or 3D mesh that has its boundary nodes grouped by named component to create boundary zones. All boundary zones will be considered wall zones; you can set the appropriate boundary types in the meshing mode or in the solution mode of ANSYS Fluent. See [Using the fe2ram Filter to Convert Files \(p. 647\)](#) for details.

A.3.3.1. Recognized PATRAN Datasets

The following Neutral file datasets are recognized by the grid filter:

Node Data

Packet Type 01

Element Data

Packet Type 02

Distributed Load Data

Packet Type 06

Node Temperature Data

Packet Type 10

Name Components

Packet Type 21

File Header

Packet Type 25

A.3.3.2. Grouping Elements to Create Zones

In PATRAN, named components are applied to the nodes to create groups of faces called zones. In ANSYS Fluent, boundary conditions are applied to each zone. For instance, all nodes on a curve or patch may be put in a Name Component.

For 2D volume meshes, an additional constraint is placed on the elements: existence in the Z=0 plane.

A.3.3.3. Periodic Boundaries

In general, it is difficult to generate a valid grid with periodic boundaries in PATRAN. However, a special feature exists in the meshing mode in ANSYS Fluent that allows you to generate a grid in a domain with periodic boundaries. See [Creating Periodic Boundaries \(p. 171\)](#) for further details.

A.3.4. ANSYS Files

For ANSYS surface meshes, the filter reads triangular and quadrilateral linear elements that define the boundaries of the domain and have been grouped within ANSYS using node and element selection. For volume meshes, the filter reads a 2D or 3D mesh that has its boundary nodes grouped within ANSYS using node and element selection. All boundary zones will be considered wall zones; you can set the appropriate boundary types in the meshing mode or in the solution mode in ANSYS Fluent. See [Using the fe2ram Filter to Convert Files \(p. 647\)](#) for details.

A.3.4.1. Recognized Datasets

The following datasets are recognized by the grid filter:

NBLOCK

node block data

EBLOCK

element block data

CMBLOCK

element/node grouping

The elements must be STIF63 linear shell elements. In addition, if element data without an explicit element ID is used, the filter assumes sequential numbering of the elements when creating the zones.

A.3.4.2. Periodic Boundaries

In general, it is difficult to generate a valid grid with periodic boundaries in ANSYS. However, a special feature exists in the meshing mode in ANSYS Fluent that allows you to generate a grid in a domain with periodic boundaries. See [Creating Periodic Boundaries \(p. 171\)](#) for further details.

A.3.5. ARIES Files

ARIES provides a filter or you may write a Prep7 file from ARIES and use the `fe2ram` filter with arguments for an ANSYS file. For more information on importing ANSYS files, see [ANSYS Files \(p. 650\)](#).

In general, to write a Prep7 file within ARIES the following criteria must be met:

- Name the part in the Geom module.
- Create a material or read one from the `mat_lib` in the Material module. To create a material, you must supply density, Poisson's ratio, and elastic modulus.
- Generate face pressures for the surface in the Environment module. Later, when you write the Prep7 file, these will be transferred to the individual elements.
- Generate at least one restraint in the Environment module.
- Set the element type to be STIF63 (triangular shell elements) and specify some finite thickness.
- Write the Prep7 file, making sure you let it automatically assign the pressure to the elements.

The Prep7 file can be filtered using the ARIES or ANSYS Fluent filter, whichever you find most convenient.

A.3.6. NASTRAN Files

For NASTRAN surface meshes, the filter reads triangular and quadrilateral linear elements that define the boundaries of the solution domain. For volume meshes, the filter reads a 2D or 3D mesh. All boundary zones will be considered wall zones; you can set the appropriate boundary types in the meshing mode or in the solution mode in ANSYS Fluent. See [Using the fe2ram Filter to Convert Files \(p. 647\)](#) for details.

A.3.6.1. Recognized NASTRAN Bulk Data Entries

The following NASTRAN bulk entries are recognized by the grid filter:

GRID

single-precision node coordinates

GRID*

double-precision node coordinates

CBAR

line elements

CTETRA, CTRIA3

tetrahedral and triangular elements

CHEXA, CQUAD4, CPENTA

hexahedral, quadrilateral, and wedge elements

Since ANSYS Fluent uses linear elements, you should use linear elements in the mesh generation process. If parabolic elements exist in the dataset, the filter ignores the mid side nodes. This assumption is valid if the edges of the element are near linear. However, if this is not the case, an incorrect topology may result from this assumption. For example, in regions of high curvature the parabolic element may look much different than the linear element.

A.3.6.2. Periodic Boundaries

In general, it is difficult to generate a valid grid with periodic boundaries in NASTRAN. However, a special feature exists in the meshing mode in ANSYS Fluent that allows you to generate a grid in a domain with periodic boundaries. See [Creating Periodic Boundaries \(p. 171\)](#) for further details.

A.3.6.3. Deleting Duplicate Nodes

NASTRAN often generates duplicate nodes in the process of creating triangular elements. These must be removed by using the **Merge** button in the **Merge Boundary Nodes** dialog box (or the /boundary/merge-duplicates text command).

Appendix B. Mesh File Format

The content and format of ANSYS Fluent mesh files is described in the following sections:

- [B.1. Guidelines](#)
 - [B.2. Formatting Conventions in Binary Files and Formatted Files](#)
 - [B.3. Grid Sections](#)
 - [B.4. Non-Grid Sections](#)
 - [B.5. Example Files](#)
-

Note

A *mesh* file is a subset of a case file, containing only those sections pertaining to the mesh. The currently defined sections relevant for the mesh file are explained in the following sections. For information about sections in the case file, refer to [Grid Sections](#) and [Other \(Non-Grid\) Case Sections](#) in the [Fluent User's Guide](#).

B.1. Guidelines

The mesh files are broken into several sections according to the following guidelines:

- Each section is enclosed in parentheses and begins with a decimal integer indicating its type. This integer is different for formatted and binary files (see [Formatting Conventions in Binary Files and Formatted Files \(p. 653\)](#)).
- All groups of items are enclosed in parentheses. This makes skipping to ends of (sub)sections and parsing them very easy. It also allows for easy and compatible addition of new items in future releases.
- Header information for lists of items is enclosed in separate sets of parentheses, preceding the items, which are in their own parentheses.

B.2. Formatting Conventions in Binary Files and Formatted Files

For formatted files, examples of file sections are given in [Grid Sections \(p. 654\)](#) and [Non-Grid Sections \(p. 663\)](#). For binary files, the header indices described in subsequent sections (e.g., 10 for the node section) are preceded by 20 for single-precision binary data, or by 30 for double-precision binary data (i.e., 2010 or 3010 instead of 10). The end of the binary data is indicated by `End of Binary Section 2010` or `End of Binary Section 3010` before the closing parameters of the section.

An example with the binary data represented by periods is as follows:

```
(3010 (2 1 2aad 2 3)(  
.  
.  
.)  
End of Binary Section 3010)
```

B.3. Grid Sections

Grid sections are stored in the case file. A *mesh* file is a subset of a case file, containing only those sections pertaining to the mesh. The currently defined grid sections are explained in the following sections.

- [B.3.1. Comment](#)
- [B.3.2. Header](#)
- [B.3.3. Dimensions](#)
- [B.3.4. Nodes](#)
- [B.3.5. Periodic Shadow Faces](#)
- [B.3.6. Cells](#)
- [B.3.7. Faces](#)
- [B.3.8. Edges](#)
- [B.3.9. Face Tree](#)
- [B.3.10. Cell Tree](#)
- [B.3.11. Interface Face Parents](#)

The section ID numbers are indicated in both symbolic and numeric forms. The symbolic representations are available as symbols in a Scheme source file (`xfile.scm`), which is available from ANSYS, Inc., or as macros in a C header file (`xfile.h`), which is located in your installation area.

B.3.1. Comment

Index:	0
Scheme symbol:	<code>xf-comment</code>
C macro:	<code>XF_COMMENT</code>
Status:	optional

Comment sections can appear anywhere in the file (except within other sections) as:

```
(0 "comment text")
```

You should precede each long section or group of related sections, by a comment section explaining what is to follow.

For example,

```
(0 "Variables:")
(60 (
(max-skew-limit 1.)
(max-cell-skew 0.85)
(skewness-method 0)
))
```

B.3.2. Header

Index:	1
Scheme symbol:	<code>xf-header</code>
C macro:	<code>XF_HEADER</code>
Status:	optional

Header sections can appear anywhere in the file (except within other sections). The following is an example:

```
(1 "ANSYS(R) TGrid(TM) 3D, serial 15.0.0")
```

The purpose of this section is to identify the program that wrote the file. Although this section can appear anywhere, it is typically one of the first sections in the file. Additional header sections indicate other programs that may have been used in generating the file. This provides a history mechanism showing where the file came from and how it was processed.

B.3.3. Dimensions

Index:	2
Scheme symbol:	xf-di-mension
C macro:	XF_DI-MENSION
Status:	optional

The dimensions of the grid appear as:

```
(2 ND)
```

where ND is 2 or 3. This section is supported as a check that the grid has the appropriate dimensions.

B.3.4. Nodes

Index:	10
Scheme symbol:	xf-node
C macro:	XF_NODE
Status:	required

The nodes section appears as:

```
(10 (zone-id first-index last-index type ND)
  x1 y1 z1
  x2 y2 z2
  .
  .
  .)
)
```

- If zone-id is zero, this provides the total number of nodes in the mesh. In this case, first-index will be one, last-index will be the total number of nodes *in hexadecimal*, type is zero, ND is omitted, and there are no coordinates following (the parentheses for the coordinates are omitted as well).

For example,

```
(10 (0 1 2d5 0))
```

- If zone-id is greater than zero, it indicates the zone to which the nodes belong. In this case, first-index and last-index are the indices of the nodes in the zone *in hexadecimal*. The values of last-

index in each zone must be less than or equal to the value in the declaration section. `type` indicates the type of nodes in the zone. The following values are used to indicate the node type:

- Zero for “virtual” nodes.
- One for no (any) type.
- Two for boundary nodes.

Nodes of type zero are ignored but types one and two are read and written.

`ND` is an optional argument that indicates the dimensionality of the node data, where `ND` is 2 or 3. If the number of dimensions in the grid is two, as specified in the [Dimensions \(p. 655\)](#) or in the node header, then only `x` and `y` coordinates are present on each line.

The following is an example of a two-dimensional grid:

```
(10 (1 1 2d5 1 2)(  
 1.500000e-01 2.500000e-02  
 1.625000e-01 1.250000e-02  
 .  
 .  
 1.750000e-01 0.000000e+00  
 2.000000e-01 2.500000e-02  
 1.875000e-01 1.250000e-02  
 ))
```

As the grid connectivity is composed of integers representing pointers (see [Cells \(p. 657\)](#) and [Faces \(p. 658\)](#)), using hexadecimal conserves space in the file and provides for faster file input and output. The header indices are also in hexadecimal so that they match the indices in the bodies of the grid connectivity sections. The `zone-id` and `type` are also in hexadecimal for consistency.

B.3.5. Periodic Shadow Faces

Index:	18
Scheme symbol:	xf-periodic-face
C macro:	XF_PERIODIC_FACE
Status:	required only for grids with periodic boundaries

This section indicates the pairings of periodic faces on periodic boundaries. Grids without periodic boundaries do not have sections of this type.

The format of the section is as follows:

```
(18 (first-index last-index periodic-zone shadow-zone)(  
 f00 f01  
 f10 f11  
 f20 f21  
 .  
 .  
 ))
```

where,

`first-index` is the index of the first periodic face pair in the list.

`last-index` is the index of the last periodic face pair in the list.

`periodic-zone` is the zone ID of the periodic face zone.

`shadow-zone` is the zone ID of the corresponding shadow face zone.

These are in hexadecimal format.

The indices in the section body (`f*`) refer to the faces on each of the periodic boundaries (in hexadecimal), the indices being offsets into the list of faces for the grid.

Note

The `first-index` and `last-index` do *not* refer to face indices. They refer to indices in the list of periodic pairs.

An example of such a section is as follows:

```
(18 (1 2b a c) (
 12 1f
 13 21
 ad 1c2
 .
 .
 .
 ))
```

B.3.6. Cells

Index:	12
Scheme symbol:	xf-cell
C macro:	XF_CELL
Status:	required

The declaration section for cells is similar to that for nodes.

```
(12 (zone-id first-index last-index type element-type))
```

When `zone-id` is zero, it indicates that it is a declaration of the total number of cells. If `last-index` is zero, then there are no cells in the grid. This is useful when the file contains only a surface mesh as it serves to alert ANSYS Fluent that it cannot be used in the solver.

In a declaration section, the `type` has a value of zero, while the `element-type` is not present.

For example,

```
(12 (0 1 3e3 0))
```

states that there are $3e3$ (hexadecimal) = 995 cells in the grid. This declaration section is required and must precede the regular cell sections.

The `element-type` in a regular cell section header indicates the type of cells in the section, as follows:

ele- ment- type	de- scrip- tion	nodes/cell	faces/cell
0	mixed		
1	triangular	3	3
2	tetra- hedral	4	4
3	quadri- lateral	4	4
4	hexa- hedral	8	6
5	pyram- id	5	5
6	wedge	6	5
7	poly- hedral	NN	NF

where NN and NF will vary, depending on the specific polyhedral cell.

Regular cell sections have no body, but they have a header of the same format where `first-index` and `last-index` indicate the range for the particular zone, `type` indicates whether the cell zone is fluid (`type = 1`) or solid (`type = 17`).

A `type` of zero indicates a dead zone and will be skipped when the file is read in solution mode in ANSYS Fluent. If a zone is of mixed type (`element-type=0`), it will have a body that lists the `element-type` of each cell.

In the following example, there are 3d (hexadecimal) = 61 cells in cell zone 9, of which the first 3 are triangles, the next 2 are quadrilaterals, and so on.

```
(12 (9 1 3d 0 0)
 1 1 1 3 3 1 1 3 1
 .
 .
 .
 ))
```

Note

The cell section is not required in meshing mode when the file contains only a surface mesh.

B.3.7. Faces

Index:	13
Scheme symbol:	xf-face
C macro:	XF_FACE
Status:	required

The face section has a header with the same format as that for cells, but with a section index of 13. The format is as follows:

```
(13 (zone-id first-index last-index bc-type face-type))
```

where:

`zone-id` = zone ID of the face section

`first-index` = index of the first face in the list

`last-index` = index of the last face in the list

`bc-type` = ID of the boundary condition represented by the face section

`face-type` = ID of the type(s) of face(s) in the section

A `zone-id` of zero indicates a declaration section, which provides a count of the total number of faces in the mesh. Such a section omits the `bc-type` and is not followed by a body with further information.

A non-zero `zone-id` indicates a regular face section, and will be followed by a body containing information about the grid connectivity. Each line describes one face and appears as follows:

```
n0 n1 n2 c0 c1
```

where `n*` are the defining nodes (vertices) of the face, and `c*` are the adjacent cells.

This is an example of the format for a 3D grid with a triangular face format. The actual number of nodes depends on the `face-type`. The order of the cell indices is important and is determined by the right-hand rule: if you curl the fingers of your right hand in the order of the nodes, your thumb will point toward `c0`.

If the face zone is of mixed type (`face-type= 0`) or of polygonal type (`face-type= 5`), each line of the section body will begin with a reference to the number of nodes that make up that particular face, and has the following format:

```
x n0 n1 ... nf c0 c1
```

where `x` = number of nodes (vertices) of the face and `nf` is the final node of the face.

All cells, faces, and nodes have positive indices. If a face has a cell only on one side, then either `c0` or `c1` is zero. For files containing only a boundary mesh, both these values are zero.

`bc-type` indicates the ID of the boundary condition represented by the face section. The current valid boundary condition types are defined in the following table:

bc-type	Description
2	interior
3	wall
4	pressure-inlet, inlet-vent, intake-fan
5	pressure-outlet, exhaust-fan, outlet-vent
7	symmetry

bc-type	Description
8	periodic-shadow
9	pressure-far-field
10	velocity-inlet
12	periodic
14	fan, porous-jump, radiator
20	mass-flow-inlet
24	interface
31	parent (hanging node)
36	outflow
37	axis

The faces resulting from the intersection of the non-conformal grids are placed in a separate face zone. A factor of 1000 is added to the **bc-type** of these sections, e.g., 1003 is a wall zone.

face-type indicates the type of faces in the zone as defined in the following table:

face-type	description	nodes/face
0	mixed	
2	linear	2
3	triangular	3
4	quadrilateral	4
5	polygonal	NN

where NN will vary, depending on the specific polygonal face.

B.3.8. Edges

Index:	11
Scheme symbol:	xf-edge
C macro:	XF_EDGE
Status:	optional

The edge section has a header of the following format:

```
(11 (zone-id first-index last-index type element-type))
```

where,

zone-id is the zone ID for the edge section

first-index and **last index** are the index of the first and last edge in the list, respectively

type indicates the edge type

The **element-type** is ignored completely.

A zone-id of zero indicates a declaration section with no body, which provides a count of the total number of edges in the mesh.

A non-zero zone-id indicates a regular edge section, and will be followed by a body containing information about the grid connectivity. Each line describes one edge and appears as follows:

v0 v1

where v0, v1 are the vertices defining the edge.

type denotes the edge type as defined in the following table:

Description	type
boundary edge	5
interior edge	6

Note

The Edges section will be omitted in the case file written for ANSYS Fluent.

B.3.9. Face Tree

Index:	59
Scheme symbol:	xf-face-tree
C macro:	XF_FACE_TREE
Status:	only for grids with hanging-node adaption

This section indicates the face hierarchy of the grid containing hanging nodes. The format of the section is as follows:

```
(59 (face-id0 face-id1 parent-zone-id child-zone-id)
  (
    number-of-kids kid-id-0 kid-id-1 ... kid-id-n
    .
    .
  ))
```

where

face-id0 is the index of the first parent face in the section.

face-id1 is the index of the last parent face in the section.

parent-zone-id is the ID of the zone containing the parent faces

child-zone-id is the ID of the zone containing the children faces.

number-of-kids is the number of children of the parent face.

kid-id-n are the face IDs of the children.

These are in hexadecimal format. You can read files that contain this section in the meshing mode in ANSYS Fluent.

B.3.10. Cell Tree

Index:	58
Scheme symbol:	xf-cell-tree
C macro:	XF_CELL_TREE
Status:	only for grids with hanging-node adaption

This section indicates the cell hierarchy of the grid containing hanging nodes.

The format of the section is as follows:

```
(58 (cell-id0 cell-id1 parent-zone-id child-zone-id)
  (
    number-of-kids kid-id-0 kid-id-1 ... kid-id-n
    .
    .
    .
  ))
```

where,

cell-id0 is the index of the first parent cell in the section.

cell-id1 is the index of the last parent cell in the section.

parent-zone-id is the ID of the zone containing the parent cells.

child-zone-id is the ID of the zone containing the children cells.

number-of-kids is the number of children of the parent cell.

kid-id-n are the cell IDs of the children.

These are in hexadecimal format. You cannot read files that contain this section in the meshing mode in ANSYS Fluent.

B.3.11. Interface Face Parents

Index:	61
Scheme symbol:	xf-face-parents
C macro:	XF_FACE_PARENTS
Status:	only for grids with non-conformal interfaces

This section indicates the relationship between the intersection faces and original faces. The intersection faces (children) are produced from intersecting two non-conformal surfaces (parents) and are some fraction of the original face. Each child will refer to at least one parent.

The format of the section is as follows:

```
(61 (face-id0 face-id1)
(
  parent-id-0 parent-id-1
  .
  .
))
)
```

where,

face-id0 is the index of the first child face in the section.

face-id1 is the index of the last child face in the section.

parent-id-* is the index of the parent faces.

These are in hexadecimal format.

If you set up and save a non-conformal mesh in the solution mode and then read it into the meshing mode of ANSYS Fluent, this section will be skipped. Hence, all the information necessary to preserve the non-conformal interface will not be maintained. When you switch to the solution mode or read the mesh back into the solution mode, you will need to recreate the interface.

B.4. Non-Grid Sections

The non-grid sections contain the boundary conditions, material properties, and solver control settings.

B.4.1. Zone

Index:	39 or 45
Scheme symbol:	xf-rp-tv
C macro:	XF_RP_TV
Status:	required

Typically, there is one zone section for each zone referenced by the grid. Although some grid zones may not have corresponding sections, there cannot be more than one zone section for each zone.

The zone section has the following form:

```
(39 (zone-id zone-type zone-name domain-id)(
  (condition1 . value1)
  (condition2 . value2)
  (condition3 . value3)
  .
  .
))
)
```

Grid generators and preprocessors need only provide the section header and leave the list of conditions empty, as in:

```
(39 (zone-id zone-type zone-name domain-id)())
```

The empty parentheses at the end are required. The solver adds conditions as appropriate, depending on the zone type.

When only zone-id, zone-type, zone-name, and domain-id are specified, the index 45 may be used for a zone section. However, the index 39 must be used if boundary conditions are present, because any and all remaining information in a section of index 45 after zone-id, zone-type, zone-name, and domain-id will be ignored.

In meshing mode, the zone name and type can be extracted from the boundary condition section 39 (refer to the ANSYS Fluent User's Guide for details) or 45, but only section 39 can be written.

The zone-id is in decimal format. This is in contrast to the use of hexadecimal in the grid sections.

The zone-type is one of the following:

```
degassing
exhaust-fan
fan
fluid
geometry
inlet-vent
intake-fan
interface
interior
internal
mass-flow-inlet
outflow
outlet-vent
parent-face
porous-jump
pressure-far-field
pressure-inlet
pressure-outlet
radiator
solid
symmetry
velocity-inlet
wall
wrapper
```

The interior, fan, porous-jump, and radiator types can be assigned only to zones of faces inside the domain. The interior type is used for the faces within a cell zone; the others are for interior faces that form infinitely thin surfaces within the domain. ANSYS Fluent allows the wall type to be assigned to face zones both on the inside and on the boundaries of the domain.

Some zone types are valid only for certain types of grid components (e.g., cell zones can be assigned only either fluid or solid type). All other types listed can be used for only boundary (face) zones.

The zone-name is a label for the zone. It must be a valid Scheme symbol and is written without quotes. The rules for a valid zone-name are as follows:

- The first character must be a lowercase letter or a special-initial.
- Each subsequent character must be a lowercase letter, a special-initial, a digit, or a special-subsequent.

A special-initial character is one of the following:

! \$ % & * / : < = > ? ~ _ ^

and a special subsequent is one of the following:

. + -

Some examples of zone sections produced in the meshing mode are as follows:

```
(39 (1 fluid fuel 1)())
(39 (8 pressure-inlet pressure-inlet-8 2)())
(39 (2 wall wing-skin 3)())
(39 (3 symmetry mid-plane 1)())
```

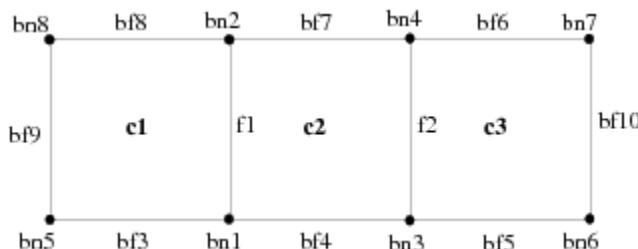
B.5. Example Files

The examples show 2D quadrilateral meshes for easier illustration. The same concepts are applied to 3D meshes.

Example 1

[Figure 1: Quadrilateral Mesh \(p. 665\)](#) illustrates a simple quadrilateral mesh with no periodic boundaries or hanging nodes.

Figure 1: Quadrilateral Mesh



The following describes this mesh:

```
(0 "Grid:")
(0 "Dimensions:")
(2 2)

(12 (0 1 3 0))
(13 (0 1 a 0))
(10 (0 1 8 0 2))

(12 (7 1 3 1 3))

(13 (2 1 2 2 2)(
1 2 1 2
3 4 2 3))

(13 (3 3 5 3 2)(
5 1 1 0
1 3 2 0
3 6 3 0))

(13 (4 6 8 3 2)(
7 4 3 0
4 2 2 0
2 8 1 0))

(13 (5 9 9 a 2)(
8 5 1 0))

(13 (6 a a 24 2)(
6 7 3 0))

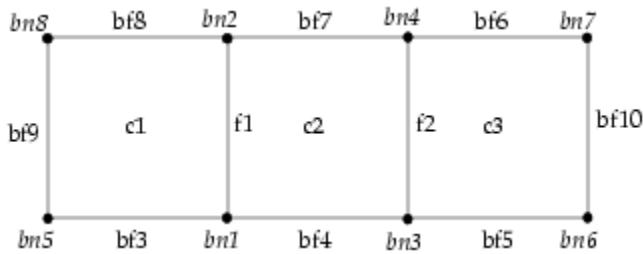
(10 (1 1 8 1 2)
(
1.00000000e+00 0.00000000e+00
1.00000000e+00 1.00000000e+00
2.00000000e+00 0.00000000e+00
2.00000000e+00 1.00000000e+00
0.00000000e+00 0.00000000e+00
```

```
3.00000000e+00 0.00000000e+00
3.00000000e+00 1.00000000e+00
0.00000000e+00 1.00000000e+00))
```

Example 2

Figure 2: Quadrilateral Mesh with Periodic Boundaries (p. 666) illustrates a simple quadrilateral mesh with periodic boundaries but no hanging nodes. In this example, bf9 and bf10 are faces on the periodic zones.

Figure 2: Quadrilateral Mesh with Periodic Boundaries



The following describes this mesh:

```
(0 "Dimensions:")
(2 2)

(0 "Grid:")

(12 (0 1 3 0))
(13 (0 1 a 0))
(10 (0 1 8 0 2))

(12 (7 1 3 1 3))

(13 (2 1 2 2 2)(
1 2 1 2
3 4 2 3))

(13 (3 3 5 3 2)(
5 1 1 0
1 3 2 0
3 6 3 0))

(13 (4 6 8 3 2)(
7 4 3 0
4 2 2 0
2 8 1 0))

(13 (5 9 9 c 2)(
8 5 1 0))

(13 (1 a a 8 2)(
6 7 3 0))

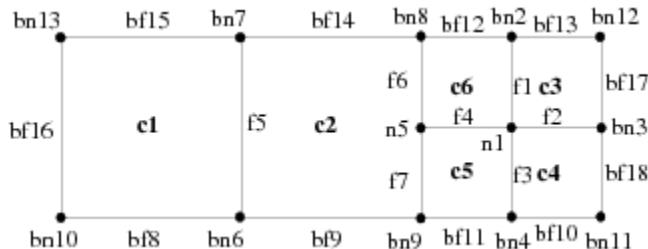
(18 (1 1 5 1)(
9 a))

(10 (1 1 8 1 2)(
1.00000000e+00 0.00000000e+00
1.00000000e+00 1.00000000e+00
2.00000000e+00 0.00000000e+00
2.00000000e+00 1.00000000e+00
0.00000000e+00 0.00000000e+00
3.00000000e+00 0.00000000e+00
3.00000000e+00 1.00000000e+00
0.00000000e+00 1.00000000e+00))
```

Example 3

Figure 3: Quadrilateral Mesh with Hanging Nodes (p. 667) illustrates a simple quadrilateral mesh with hanging nodes.

Figure 3: Quadrilateral Mesh with Hanging Nodes



The following describes this mesh:

```
(0 "Grid:")
(0 "Dimensions:")
(2 2)

(12 (0 1 7 0))
(13 (0 1 16 0))
(10 (0 1 d 0 2))

(12 (7 1 6 1 3))
(12 (1 7 7 20 3))

(58 (7 7 1 7) (
4 6 5 4 3))

(13 (2 1 7 2 2) (
1 2 6 3
1 3 3 4
1 4 4 5
1 5 5 6
6 7 1 2
5 8 2 6
9 5 2 5))

(13 (3 8 b 3 2) (
a 6 1 0
6 9 2 0
4 b 4 0
9 4 5 0))

(13 (4 c f 3 2) (
2 8 6 0
c 2 3 0
8 7 2 0
7 d 1 0))

(13 (5 10 10 a 2) (
d a 1 0))

(13 (6 11 12 24 2) (
3 c 3 0
b 3 4 0))

(13 (b 13 13 1f 2) (
c 8 7 0))

(13 (a 14 14 1f 2) (
b c 7 0))

(13 (9 15 15 1f 2) (
```

```
9 b 7 0))  
(13 (8 16 16 1f 2)(  
9 8 2 7))  
(59 (13 13 b 4)(  
2 d c))  
(59 (14 14 a 6)(  
2 12 11))  
(59 (15 15 9 3)(  
2 b a))  
(59 (16 16 8 2)(  
2 7 6))  
(10 (1 1 d 1 2)  
(  
2.5000000e+00 5.0000000e-01  
2.5000000e+00 1.0000000e+00  
3.0000000e+00 5.0000000e-01  
2.5000000e+00 0.0000000e+00  
2.0000000e+00 5.0000000e-01  
1.0000000e+00 0.0000000e+00  
1.0000000e+00 1.0000000e+00  
2.0000000e+00 1.0000000e+00  
2.0000000e+00 0.0000000e+00  
0.0000000e+00 0.0000000e+00  
3.0000000e+00 0.0000000e+00  
3.0000000e+00 1.0000000e+00  
0.0000000e+00 1.0000000e+00))
```

Appendix C. Shortcut Keys

Creating a mesh and modifying and/or repairing it involves number of operations. Various tools are available for making boundary repairs, enabling you to perform primitive operations on the boundary mesh, such as creating and deleting nodes/faces, moving nodes, swapping edges, merging/smoothing nodes, collapsing nodes, edge(s), and face(s), etc.

These operations can be performed using the appropriate dialog boxes. However, if you are handling a large and complicated mesh, you may find it difficult to perform these operations repetitively using the dialog boxes. To make your task easy, several keyboard shortcuts are available.

C.1. Arrow Keys

C.2. Help Keys

C.3. Hot Keys

Important

To make use of the shortcut keys, display the geometry in the graphics window. Click in the graphics window and then press the key you want to use.

C.1. Arrow Keys

The arrow keys used are described in the following table:

Arrow Keys	Description
Up arrow	Increases the display bounds.
Down arrow	Decreases display bounds.
Right arrow	Allows you to view the skewed faces in the decreasing order of their skewness by finding the triangular face of next highest skewness value after the worst skewed face in the mesh/active group or the marked faces. When you press this button, the triangular face of the next highest skewness value will be displayed in the graphics window.
Left arrow	Clears the selection of skewed faces.

C.2. Help Keys

The various help keys are described in the following table:

Help Keys	Description
F2	Clears the previously selected entities in the Selections list of the Modify Boundary dialog box. Clears existing Material Points .
F3	Toggles between the mouse-dolly and the option currently selected for right-mouse button in the Mouse Buttons dialog box.
F4	Toggles the selection of the mouse probe function between polygon and the option selected in the Mouse Probe dialog box.
F5	Creates the entity selected in the Filter list in the Modify Boundary dialog box.
F6	Smooths the selected entity in the Filter list.
F7	Performs the split operation on the selected entity (only face and edge) in the Filter list.
F8	Performs the swap operation on the selected entity (only face and edge) in the Filter list.
F9	Allows you to merge the selected pair of nodes.
F11	Allows you to locate the worst skewed face in the mesh/active group. When you press this hot key, the graphics window will be updated to show only the worst skewed face and its ID. The skewness value is printed in the console.
F12	Undoes the latest operation performed on the mesh.

When modifying the boundary mesh, it is not always possible to undo an operation.

C.3. Hot Keys

Hot Keys	Description
Ctrl+a	Adjusts the mesh position so that the mesh fits in the graphics window.
Ctrl+b	Selects object as the Filter for the Mouse Probe function.
Ctrl+c	Selects cell as the Filter for the Mouse Probe function. This option does not work in the Windows version.
Ctrl+d	Calculates the distance between the two selected entities and reports it in the message window.
Ctrl+e	Selects edge as the Filter for the Mouse Probe function.
Ctrl+f	Selects face as the Filter for the Mouse Probe function.
Ctrl+g	Displays the neighboring face zones of the selected entity.
Ctrl+h	Prints the available hot keys in the message and graphics window.
Ctrl+i	Prints detailed information about the selected entities in the message window.
Ctrl+j	Performs the collapse operation on the selected nodes, edges, or faces.
Ctrl+k	Opens the Remove Boundary Gaps Dialog Box to remove gaps between the selected objects.
Ctrl+l	Finds the centroid of the selected face.
Ctrl+n	Selects node as the Filter for the Mouse Probe function.
Ctrl+o	Rezones the faces/cells selected to the zone selected.

Hot Keys	Description
Ctrl+p	Projects selected nodes/node loops/zones onto what you have set using Ctrl+s .
Ctrl+r	Repairs the selected zone(s) by filling all holes associated with free edges.
Ctrl+s	Allows you to set a plane, or line, or zone, on which you can project the nodes. Set the plane or line either by selecting nodes or by selecting positions in the mesh.
Ctrl+t	Trace path between selected faces.
Ctrl+u	Clears the marked faces selected.
Ctrl+v	Allows you to move nodes from their original location to a new position.
Ctrl+w	Deletes the selected entities (nodes/faces/zones/objects) without confirming.
Ctrl+x	Selects position as the Filter for the Mouse Probe function.
Ctrl+y	Selects computed Size Function size as the Filter for the Mouse Probe function.
Ctrl+z	Selects zone as the Filter for the Mouse Probe function.
Ctrl+^	Performs the collapse operation on the selected nodes, edges, or faces. Ctrl+j does the same operation.
Ctrl+_	Displays the previous view in the graphics window.
Ctrl+[Deselect the last selected entity. Escape key performs the same operation.
Ctrl+Shift+a	Shows all zones/objects in the graphics window, unhides previously hidden zones/objects.
Ctrl+Shift+b	<p>Enters the list selection mode (updates the dialog box selection list after select/deselect objects or zones graphically). The following sub-options are available:</p> <ul style="list-style-type: none"> • Zone selections: <ul style="list-style-type: none"> – Ctrl+q: Adds the selected zones to the current selections in the dialog box list. – Ctrl+s: Removes the selected zones from the current selections in the dialog box list. – Ctrl+d: Sets the selected zones as the current selections in the dialog box list. • Object selections: <ul style="list-style-type: none"> – Ctrl+j: Adds the selected objects to the current selections in the dialog box list. – Ctrl+k: Removes the selected objects from the current selections in the dialog box list. – Ctrl+l: Sets the selected objects as the current selections in the dialog box list.

Hot Keys	Description
Ctrl+Shift+c	<p>Enters the color selection mode. The following sub-options are available:</p> <ul style="list-style-type: none"> • Ctrl+o: Colors the items by object. • Ctrl+z: Colors the items by zone. • Ctrl+n: Colors the items based on normals. • Ctrl+p: Toggles the color palette. • Ctrl+r: Randomizes the colors used in the display.
Ctrl+Shift+d	<p>Enters the conditional display options mode. The following sub-options are available:</p> <ul style="list-style-type: none"> • Ctrl+s: isolate zones/objects of similar surface area. • Ctrl+c: isolate zones/objects of similar surface curvature. • Ctrl+n: isolate zones/objects with neighborhood. • Ctrl+Shift+a: show all zones/objects in the graphics window, unhides previously hidden zones/objects.
Ctrl+Shift+e	<p>Shows/hides edges on selected zones and objects independent of the mode of selection. If no object/zone is selected then the edges on the entire geometry are shown/hidden.</p>
Ctrl+Shift+f	<p>Allows you to improve feature capture and/or rezone wrap face zones.</p> <ul style="list-style-type: none"> • When wrap face zones are selected, they will be rezoned. • When wrap face zones and edge zones are selected, feature capture on the wrap face zones will be improved and the face zones will be rezoned. • When wrap face zones and edge entities are selected, feature capture on the wrap face zones will be improved.
Ctrl+Shift+g	<p>Align objects.</p>
Ctrl+Shift+h	<p>Shows/hides the selected objects or zones in the display depending on the mode of selection set. The following sub-options are available:</p> <ul style="list-style-type: none"> • Ctrl+Shift+u: Reverses the last show/hide operation. • Ctrl+Shift+a: Shows all zones/objects in the graphics window.
Ctrl+Shift+i	<p>Isolate selected zones/objects. The following sub-options are available:</p> <ul style="list-style-type: none"> • Ctrl+Shift+u: Undo the last show/hide operation. • Ctrl+Shift+a: Shows all zones/objects in the graphics window.

Hot Keys	Description
Ctrl+Shift+l	<p>Allows you to select a loop of nodes for probe selection or for creating an edge loop or capping surface. The following sub-options are available:</p> <ul style="list-style-type: none"> • Selection options: <ul style="list-style-type: none"> – Ctrl+o: Toggle between an open or closed loop. – Ctrl+d: Toggle between using the direct or node loop closing path between the first and last nodes when the closed node loop option is selected. – Ctrl+p: Allows you to toggle between using the node path or the direct path between the last two nodes selected. • Create options: <ul style="list-style-type: none"> – Ctrl+j: Selects all nodes on the loop path. – Ctrl+k: Creates a capping surface based on the nodes selected. – Ctrl+l: Creates an edge loop based on the nodes selected.
Ctrl+Shift+m	<p>Merges all the selected objects/zones into one object/zone depending on which mode of selection is set. The resulting object/zone inherits the name of the object/zone selected first. If the selected zones are from different objects, then the resulting merged zone will be in the object from which a zone was selected first. The other objects will retain a copy of the zone(s) which went into the merging.</p>
Ctrl+Shift+n	<p>Opens the Rename Objects or Rename Zones dialog box, depending on your selection. See Rename Objects/Zones Dialog Box.</p>
Ctrl+Shift+o	<p>Randomly allocates colors to selected objects/zones, or to entire geometry if no object/zone is selected.</p>
Ctrl+Shift+r	<p>If faces are graphically selected, the Local Remesh Dialog Box will be available for local remesh. If zones are graphically selected, they will be remeshed using size field, if available.</p>
Ctrl+Shift+s	<p>Separates the face or edge zones based on the selected entity. If a face is selected, the face zone is separated by seed. If a zone is selected and more than one region is present, then the face zones are separated based on regions. If the selected zone consists of only one region, then the face zones will be separated based on angle. Edge zones will be separated by seed.</p>
Ctrl+Shift+t	<p>Toggles the transparency of the selected objects/zones depending on the mode of selection set. If no object/zone is selected then the entire geometry is made transparent so that internal objects/zones are visible.</p>
Ctrl+Shift+u	<p>Reverses the previously performed hide/isolate operations one step at a time. Reversing of operation will depend on which mode of selection is set.</p>
Ctrl+Shift+v	<p>Selects all the visible objects/zones in the graphics window.</p>

Hot Keys	Description
Ctrl+Shift+x	Toggles between a normal view and an exploded view of the objects in the geometry.
Ctrl+Shift+y	Yanks the zones to the objects.
Ctrl+Shift+z	<p>Toggles the selection mode to either face or edge zone components. Default is face zone selection.</p> <p>To restrict selection to edge zones and edge zone entities, use this hot-key followed by the appropriate hot-key (e.g., Ctrl+Shift+z followed by Ctrl+e for selecting edges comprising the edge zone).</p> <hr/> <p>Note</p> <p>Ensure that display of free faces is disabled when selecting edge zones as this may incorrectly select the face zone instead.</p>

Use the **Esc** key to deselect the previously selected entity. You can also use the **Ctrl+[** shortcut key.

Note

The information displayed for the selected entities using the hot key **Ctrl+i** is as follows:

Entity selected	Information displayed
Zone	Zone name Zone ID Location Zone type Object(s) that the zone is included in
Face	Face name Zone ID Nodes Adjoining cell(s) Face skewness (if applicable) Periodic twins (if applicable) Child faces (if applicable)
Edge	Edge name Zone ID

	Adjoining faces
Cell	Cell name Zone ID Nodes Faces Cell center Cell skewness Cell simplex radius Cell size
Node	Node name Zone ID Location Adjoining faces (if applicable) Adjoining cells (if applicable)

Appendix D. Query Functions

When performing a series of operations, you may want to list all the zones or objects of a particular type or containing a particular text string, or all the zones of a group or object. You can query the zones or objects through the TUI based on a regular expression and a specified variable. You can:

- Identify face/cell zones closest to a specified point using the query functions.
- Identify zones/objects of a specific type or containing a particular text string. You can also identify zones belonging to a group or object.
- Use the query functions to determine the zones created during a particular operation by performing Boolean operations on lists returned by specific query functions.
- Use the eval-expr function to evaluate the lists returned by the query functions in order that they can be used as input in the text user interface commands.

Refer to [Examples \(p. 679\)](#) for examples of query functions.

These are the query functions:

(get-face-zone-at-location '(x y z))
Returns the face zone at or closest to the specified location

(get-cell-zone-at-location '(x y z))
Returns the cell zone at or closest to the specified location

(get-zones-of-type 'type)
Returns a list of zones of the specified default zone type (*type*)

Face zone example:

```
(get-zones-of-type 'symmetry)
```

Edge zone example:

```
(get-zones-of-type 'boundary-edge)
```

Cell zone example:

```
(get-zones-of-type 'fluid)
```

Node zone example:

```
(get-zones-of-type 'boundary-node)
```

(get-zones-of-group 'group)
Returns a list of the face zones in the specified group (*group*)

Default face zone group example:

```
(get-zones-of-group 'geometry)
```

Default edge zone group example:

```
(get-zones-of-group 'boundary-edge)
```

Default cell zone group example:

```
(get-zones-of-group 'fluid)
```

Default node zone group example:

```
(get-zones-of-group 'boundary-node)
```

User-defined group example:

(get-zones-of-group 'inlets) returns a list of all inlets in the geometry, where *inlets* is the user-defined group comprising all inlets

(get-face-zones-of-filter 'filter)

Returns a list of the face zones whose names contain the specified filter string

Example:

```
(get-face-zones-of-filter 'prism-cap*)
```

(get-cell-zones-of-filter 'filter)

Returns a list of the cell zones whose names contain the specified filter string

Example:

```
(get-cell-zones-of-filter 'prism-cells*)
```

(get-edge-zones-of-filter 'filter)

Returns a list of the edge zones whose names contain the specified filter string

Example:

```
(get-edge-zones-of-filter 'prism-cap*)
```

(get-node-zones-of-filter 'filter)

Returns a list of the node zones whose names contain the specified filter string

Example:

```
(get-face-zones-of-filter 'boundary-node*)
```

(get-wrapped-zones)

Returns a list of the wrapped face zones

(get-objects-of-type 'type)

Returns a list of objects of the specified type (*type*)

Example:

```
(get-objects-of-type 'wrap)
```

(get-objects-of-filter 'filter)

Returns a list of objects whose names contain the specified filter string (*filter*)

Example:

```
(get-objects-of-filter 'cap*)
```

(get-face-zones-of-objects '(objects))

Returns a list of face zones in the specified objects (*objects*)

Example:

```
(get-face-zones-of-objects '(inlet outlet))
```

(get-edge-zones-of-objects '(objects))

Returns a list of edge zones in the specified objects (*objects*)

Example:

```
(get-edge-zones-of-objects '(inlet outlet))
```

D.1. Using Boolean Operations with Query Functions

You can perform Boolean operations on lists returned by the query functions:

- Union of lists: (list-union list-1 list-2 ...)
- Intersection of lists: (list-intersection list-1 list-2 ...)
- Subtraction of lists: (list-subtract list-1 list-2)

Note

Only two lists can be used as arguments for a subtraction operation.

For example,

```
list-1 = '(1 2 3 4)
list-2 = '(1 5 6)
list-3 = '(1 4 7)
```

then:

```
(list-union list-1 list-2 list-3) = '(1 2 3 4 5 6 7)
(list-intersection list-1 list-2 list-3) = '(1)
(list-subtract list-1 list-2) = '(2 3 4)
```

D.2. Examples

Some examples of using query functions:

- Deleting the geometry while retaining the wrapper surface after wrapping operations

1. Obtain a list of the boundary zones:

```
(define initial-zones(get-zones-of-group 'boundary))
```

2. Perform the wrapping operations as required.
3. Obtain a list of the boundary zones after the wrapping operations:

```
(define final-zones(get-zones-of-group 'boundary))
```

4. Delete the geometry:

```
/boundary/manage/delete (eval-expr '(list-intersection initial-zones  
final-zones))
```

- Smoothing the **prism-cap** zone created during the prism creation operation

1. Obtain a list of the zones named **prism-cap***

```
(define initial-zones(get-face-zones-of-filter 'prism-cap*))
```

2. Apply appropriate prism parameters and create prisms.

3. Obtain a list of the zones named **prism-cap*** after the prism creation operation

```
(define final-zones(get-face-zones-of-filter 'prism-cap*))
```

4. Smooth the recently created **prism-cap** zone

```
/boundary/improve/smooth(eval-expr '(list-subtract final-zones ini-  
tial-zones))
```

Bibliography

- [1] TJ Baker. *Three Dimensional Mesh Generation By Triangulation of Arbitrary Point Sets*. Technical Report AIAA-87-1124. American Institute of Aeronautics and Astronautics. 1987.
- [2] TJ Barth, Wiltberger NL, and AS Gandhi. *Three-Dimensional Unstructured Grid Generation via Incremental Insertion and Local Optimization In Software Systems for Surface Modeling and Grid Generation*. Langley Research Center, Hampton, Virginia, NASA Conference Publication. 3143. April 1992.
- [3] KR Blake and GS Spragle. *A 3D Delaunay Unstructured Grid Generator Applied to Trains, Planes, and Automobiles*. Technical Report AIAA93-0673. American Institute of Aeronautics and Astronautics. 1993.
- [4] A Bowyer. *Computing Dirichlet Tessellations*. *The Computer Journal*. 24(2). 162–166. May 1981.
- [5] M Garland and P S Heckbert. *Surface Simplification Using Quadric Error Metrics*. *24th Annual Conference on Computer Graphics and Interactive Techniques*. 209–216. 1997.
- [6] DG Holmes and DD Snyder. *The Generation of Unstructured Triangular Meshes Using Delaunay Triangulation*. DG Holmes and DD Snyder In S O'Sengupta, J Hauser, PR Eiseman, and JF Thompson, editors *Numerical Grid Generation in Computational Fluid Mechanics 88*. 643–652. Pineridge Press Limited . 1988.
- [7] Y Kallinderis, A Khawaja, and H McMorris. *Hybrid Prismatic/Tetrahedral Grid Generation for Complex Geometries* Technical Report AIAA-95-0211. AIAA 33rd Aerospace Sciences Meeting and Exhibit Reno, Nevada. 1995.
- [8] Y Kallinderis and S Ward. *Prismatic Grid Generation for Three-Dimensional Complex Geometries*. *AIAA Journal*,. 31(10). 1850–1856. October 1993.
- [9] L Kobbelt, J Vorsatz, U Labsik, and H P Seidel. *A Shrink Wrapping Approach to Remeshing Polygonal Surfaces*. *Computer Graphics Forum*. 18(3). 119–130. 1999.
- [10] CL Lawson. *Properties of n-dimensional Triangulations*. *Computer-Aided Geometric Design*. 3. 231–246. December 1986.
- [11] R Löhner and P Parikh. *Generation of Three-Dimensional Unstructured Grids by the Advancing Front Method* Technical Report, AIAA-88-0515. American Institute of Aeronautics and Astronautics. 1988.
- [12] DJ Mavriplis. *Adaptive Mesh Generation for Viscous Flows Using Delaunay Triangulation*. *Journal of Computational Physics*. 90(12). 271–291. October 1990.
- [13] JD Müller, PL Roe, and Deconinck H. *A Frontal Approach for Node Generation in Delaunay Triangulations*. *International Journal For Numerical Methods in Fluids*. 17(3). 241–255. August 1993.
- [14] S Pirzadeh. *Unstructured Viscous Grid Generation by the Advancing-Layers Method*. *AIAA Journal*. 32(8). 1735–1737. August 1994.
- [15] S Pirzadeh. *Three-Dimensional Unstructured Viscous Grids by the Advancing-Layers Method*. *AIAA Journal*. 34(1). 43–49. January 1996.
- [16] W R Quadros, K Shimada, S J Owen, and M Brewer. *Finite Element Mesh Sizing for Surfaces Using Skeleton*. In Proc 13th Int Meshing Roundtable. 2004.

- [17] J Z Wang and K Srinivasan.*An Adaptive Cartesian Grid Generation Method for 'Dirty' Geometry.* *International Journal For Numerical Methods in Fluids.* 39. 703–717. 2002.
- [18] DF Watson. *Computing the n-dimensional Delaunay tessellation with application to Voronoi polytopes.* *The Computer Journal.,* 24(2). 167–172. May 1981.
- [19] NP Weatherill. *A Method For Generating Irregular Computational Grids in Multiply Connected Planar Domains.* *International Journal For Numerical Methods in Fluids.* 8(2). 181–197 . February 1988.
- [20] MA Yerry and MS Shephard. *Automatic Three-Dimensional Mesh Generation by the Modified-Octree Technique.* *International Journal for Numerical Methods in Engineering.* 20. 1965–1990. 1984.
- [21] J Zhu, T Blacker, and R Smith. *Background Overlay Grid Size Functions.* In Proc 11th Int Meshing Roundtable. 2002.