® 2025 ANSYS, Inc. or its affiliated companies
Unauthorized use, distribution, or duplication is prohibited.

ANSYS Meshing User's Guide

Ansys

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

Release 2025 R1
January 2025

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

Copyright and Trademark Information

© 2025 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

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

Disclaimer Notice

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

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

U.S. Government Rights

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

Third-Party Software

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

Published in the U.S.A.

Table of Contents

Capabilities in Ansys Workbench 19
Meshing Overview 19
Meshing Implementation in Ansys Workbench 19
Types of Meshing 20
Part/Body Level Meshing 20
Mesh Methods 20
Conformal and Non-Conformal Meshing 21
Conformal Meshing Within a Part 21
Conformal Meshing Across Parts 24
Connections/Interface Handling 24
Usage in Workbench 27
Basic Meshing Application Workflows 27
Overview of the Meshing Process in Ansys Workbench 27
Overview of the Meshing Process for CFD/Fluids Analyses 28
Overview of the Meshing Process for Hydrodynamics Analysis 30
Combining CFD/Fluids Meshing and Structural Meshing 31
Strategies for CFD/Fluids Meshing in Ansys Workbench 33
Accessing Meshing Functionality 35
Overview of the Meshing Application Interface 36
Determination of Physics, Analysis, and Solver Settings 39
Working with Legacy Mesh Data 40
Exporting Meshes or Faceted Geometry 42
Mesh Application File Export 43
Fluent Mesh Export 43
Classes of Zone Types in Ansys Fluent 46
Standard Naming Conventions for Naming Named Selections 48
Zone Type Assignment 49
Examples of Fluent Mesh Export: An Alternative to Using a Fluid Flow (Fluent) Analysis System 55
Polyflow Classic Export 63
CGNS Export 64
Ansys ICEM CFD Export 65
Exporting Faceted Geometry to Ansys Fluent Meshing 74
Named Selections and Regions for Ansys CFX 77
Passing Named Selections to the Solver 78
Repairing Geometry in Overlapping Named Selections 79
Resolving Overlapping Contact Regions 79
Extended Ansys ICEM CFD Meshing 83
Writing Ansys ICEM CFD Files 84
Rules for Interactive Editing 86
Limitations of Ansys ICEM CFD Interactive 87
Working with Meshing Application Parameters 87
Ansys Workbench and Mechanical APDL Application Meshing Differences 87
Mesh Controls Overview 89
Global and Local Mesh Controls 89
Understanding the Influence of the Sizing Options 89
Adaptive Sizing 90
Uniform, Curvature, Proximity, or Proximity and Curvature Sizing 90
Overriding Sizing Minimum and Maximum Sizes 91

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Global Mesh Controls 93
Display Group 93
Display Style 93
Defaults Group 96
Physics Preference 96
Solver Preference 99
Export Format 100
Export Unit 100
Export Preview Surface Mesh 100
Element Order 101
Element Size 102
Sizing Group 103
Mesh Sizing Defaults 104
Using Dynamic Mesh Sizing Defaults 104
Sizing Options 105
Curvature-Based Sizing 107
Proximity-Based Sizing 107
Uniform Sizing 108
Setting Sizing Options 109
Resolution 109
Growth Rate 110
Max Size 110
Mesh Defeaturing 110
Transition 112
Span Angle Center 112
Initial Size Seed 113
Curvature Min Size 113
Curvature Normal Angle 114
Proximity Min Size 115
Proximity Gap Factor 115
Proximity Size Function Sources 115
Use Uniform Size Function for Sheets 116
Enable Washers 117
Height of Washer 117
Allow Nodes to be Moved off Boundary 117
Limitations for Washers 119
Bounding Box Diagonal 119
Average Surface Area 120
Minimum Edge Length 120
Quality Group 120
The Mesh Quality Workflow 120
Check Mesh Quality 121
Error and Warning Limits 127
Target Element Quality 130
Target Skewness 131
Target Jacobian Ratio (Corner Nodes) 131
Target Characteristic Length (LS-DYNA) 132
Target Aspect Ratio (Height) 132
Smoothing 133
Mesh Metric 133
Element Quality 139

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Aspect Ratio Calculation for Triangles 140
Aspect Ratio Calculation for Quadrilaterals 141
Aspect Ratio Calculation for Tetrahedrons 142
Jacobian Ratio 142
Warping Factor 147
Parallel Deviation 149
Maximum Corner Angle 150
Skewness 151
Orthogonal Quality 153
Characteristic Length (Autodyn) 155
Minimum Tri Angle 156
Maximum Tri Angle 156
Minimum Quad Angle 157
Maximum Quad Angle 157
Warping Angle 157
Tet Collapse 158
Aspect Ratio (Height) 159
Minimum Element Edge Length 159
Maximum Element Edge Length 160
Characteristic Length (LS-DYNA) 160
Inflation Group 161
Use Automatic Inflation 163
None 164
Program Controlled 164
All Faces in Chosen Named Selection 165
Inflation Option 166
Transition Ratio 168
Maximum Layers 169
Growth Rate 169
Number of Layers 169
Maximum Thickness 169
First Layer Height 169
First Aspect Ratio 170
Aspect Ratio (Base/Height) 170
Inflation Algorithm 170
Inflation Element Type 174
View Advanced Options 174
Collision Avoidance 174
Fix First Layer 176
Gap Factor 177
Maximum Height over Base 177
Growth Rate Type 177
Maximum Angle 178
Fillet Ratio 178
Use Post Smoothing 179
Smoothing Iterations 180
Advanced Group 180
Number of CPUs for Parallel Part Meshing 180
Straight Sided Elements 180
Rigid Body Behavior 182
Triangle Surface Mesher 182

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Topology Checking ... 183
Protecting Topology Defined Prior to Meshing ... 184
Protecting Topology Post Meshing ... 186
Pinch ... 187
Pinch Control Automation Overview ... 190
How to Define Pinch Control Automation ... 193
How to Define or Change Pinch Controls Manually ... 194
Usage Information for Pinch Controls ... 194
Loop Removal ... 196
Automatic Methods ... 197
Sheet Body Method ... 197
Sweepable Body Method ... 198
Statistics Group ... 198
Nodes ... 198
Elements ... 199
Show Detailed Statistics ... 199
Model Assembly Group ... 199
Local Mesh Controls ... 201
Method Control ... 202
Method Controls and Element Order Settings ... 202
Setting the Method Control for Solid Bodies ... 206
Automatic Method Control ... 206
Tetrahedrons Method Control ... 207
Patch Conforming Algorithm for Tetrahedrons Method Control ... 208
Patch Independent Algorithm for Tetrahedrons Method Control ... 210
Hex Dominant Method Control ... 233
Sweep Method Control ... 235
MultiZone Method Control ... 242
Cartesian Method Control ... 261
Layered Tetrahedrons Method Control ... 266
Particle Method ... 272
Automatic (PrimeMesh) ... 273
Setting the Method Control for Sheet Bodies ... 282
Quadrilateral Dominant Method Control ... 282
Triangles Method Control ... 284
MultiZone Quad/Tri Method Control ... 284
Automatic (PrimeMesh) ... 287
Sizing Control ... 296
Notes on Element Sizing ... 297
Applying a Local Sizing Control ... 300
Descriptions of Local Sizing Control Options ... 302
Contact Sizing Control ... 311
Refinement Control ... 313
Face Meshing Control ... 314
Setting Basic Face Meshing Controls for Mapped Meshing ... 315
Understanding Advanced Mapped Face Meshing Controls ... 320
Restrictions Related to Vertex Types ... 321
Restrictions Related to Edge Mesh Intervals ... 322
Selecting Faces and Vertices ... 322
Effect of Vertex Type on Face Meshes ... 324
Setting Advanced Face Meshing Controls for Mapped Meshing ... 325

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes on Face Meshing Controls for Mapped Meshing 327
Example Usage 331
Mesh Copy Control 331
Match Control 334
Cyclic Match Control 336
Arbitrary Match Control 338
Pinch Control 340
Defining Pinch Controls Locally 340
Changing Pinch Controls Locally 343
Inflation Control 345
Gasket Control 349
Feature Suppress 350
Repair Topology 355
Connect 359
Weld 362
Quad Layer 380
Deviation 385
Geometry Fidelity 386
Options 389
Accessing the Options Dialog Box 389
Meshing Options on the Options Dialog Box 389
Licensing Option 394
Mesh Quality Option 395
Specialized Meshing 399
Mesh Sweeping 399
Thin Model Sweeping 406
MultiZone Meshing 419
MultiZone Algorithms 420
MultiZone for Sweepable Bodies 422
Using MultiZone 423
MultiZone Source Face Selection Tips 426
MultiZone Source Face Imprinting Guidelines 427
Internal Loops 427
Boundary Loops 428
Multiple Internal Loops 428
Multiple Connected Internal Loops 429
Parallel Loops 430
Intersecting Loops 431
MultiZone Face Mappability Guidelines 432
Side Face Handling of Imprinted Regions 432
Using Virtual Topology to Handle Fillets in MultiZone Problems 439
MultiZone Support for Inflation 440
MultiZone Limitations and Hints 442
Selective Meshing 444
Inflation Controls 453
Mesh Refinement 461
Mixed Order Meshing 461
Contact Meshing 462
Winding Body Meshing 462
Wire Body Meshing 462
Pyramid Transitions 462

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Match Meshing and Symmetry 462
Rigid Body Meshing 463
Thin Solid Meshing 466
CAD Instance Meshing 466
Meshing and Hard Entities 468
Baffle Meshing 470
Parallel Part Meshing 472
Mesh Control Interaction Tables 475
Interactions Between Mesh Methods 475
Interactions Between Mesh Methods and Mesh Controls 478
Miscellaneous Tools 483
Generation of Contact Elements 483
Renaming Mesh Control Tools 484
Mesh Numbering 484
Mesh Editing 484
Inserting a Mesh Edit Object 485
Mesh Connections 485
Contact Matches 496
Considerations for Contact Matches 497
How Mesh Size Affects Contact Matches 497
How Tolerances Affect Contact Matches 498
Applying Contact Matches 500
Displaying Multiple Views of Contact Matches 504
Troubleshooting Failed Contact Matches 505
Node Merge 508
Node Move 512
Pull 516
Common Display Features 524
Hiding or Suppressing Bodies 524
Hiding or Showing Faces 525
Creating Section Planes 525
Ease of Use Features 527
Updating the Mesh Cell State 527
Generating Mesh 528
Previewing Surface Mesh 531
Exporting a Previewed Surface Mesh in Fluent Format 533
Previewing Source and Target Mesh 533
Previewing Inflation 534
Exporting a Previewed Inflation Mesh in Fluent Format 535
Previewing Weld Geometry 535
Previewing Weld Mesh 536
Previewing Quad Layer 536
Showing Program Controlled Inflation Surfaces 536
Showing Sweepable Bodies 537
Showing Problematic Geometry 537
Showing Problematic Location 537
Showing Problematic Elements 538
Showing Elements that Do Not Meet the Target Metric 538
Showing Removable Loops 538
Showing Suppressible Features 539
Displaying Element Normals on Mesh 539

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inspecting Large Meshes Using Named Selections 540
Generating Multiple Mesh Controls from a Template 540
Clearing Generated Data 541
Showing Missing Tessellations 542
Showing Mappable Faces 542
Grouping Mesh Objects By Type 543
Virtual Topology 545
Introduction 545
Creating and Managing Virtual Cells 546
Creating and Managing Virtual Split Edges 561
Creating and Managing Virtual Split Faces 565
Creating and Managing Virtual Hard Vertices 568
Common Virtual Topology Operations 568
Common Virtual Topology Features 572
Troubleshooting 579
Mesh Workflows 601
Mesh Workflow Concepts 601
Mesh Workflows Introduction 607
Mesh Workflows Context Tab 612
Mesh Workflow 613
Input 615
Steps 615
Output 617
Mesh Workflow Types 618
External FEM Acoustics 619
Internal FEM Acoustics 621
BEM Acoustics 623
Mesh Workflow Steps 624
Fill Holes 625
Wrap 626
Mesh Surface 627
Create Enclosure 627
Mesh Volume 628
Improve Volume Mesh 628
Extrude 629
Create Topology 629
Merge Volumes 629
Manage Zone Properties 630
Patch Holes 630
Read Mesh 631
Write Mesh 631
Create Size Field 631
Mesh Workflow Controls 632
Hole Filling 633
Constant Size Wrapper 635
Custom Names Wrapper 637
Material Point 637
Leak Detection 638
Constant Size Surface Mesher 638
Wrapper Specific Surface Mesher 640
External Part Enclosure 641

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Spherical Enclosure 643
Hemispherical Enclosure 645
Convex Irregular Shape Enclosure 647
HemiConvex Irregular Shape Enclosure 649
Custom Names 652
Constant Size Volume Mesher 652
Volume Mesh Improvement 654
Second Order Conversion 655
Extrusion 655
Topology Creation 657
Volumes Merging 657
Zone Material Assignment 658
Zone Thickness Assignment 658
Checkpoint 659
Hole Patching 659
Mesh Reading 660
Mesh Writing 660
Custom Names(Sizing) 660
Constant Sizing 661
Curvature Sizing 662
Proximity Sizing 664
Body of Influence Sizing 665
Size Field Surface Mesher 667
Size Field Volume Mesher 668
Size Field Wrapper 670
Mesh Workflow Outcomes 672
Scope 673
Failure Info 673
Face Zone Scope 674
Volume Zone Scope 674
Internal Enclosure Scope 674
External Enclosure Scope 675
Extrusion Start Scope 675
Extrusion End Scope 675
Internal Volume Zone Scope 676
Size Field Name 676
Property Worksheet 676
Mesh Workflow States 678
Understanding the States of Mesh Workflows 679
Understanding the States of Mesh Workflow Steps 681
Mesh Workflow Domain Browser 682
Show Wizards 684
Apply Scoping 685
Create Sizing 686
Index 689

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

List of Figures

1. Baffle Model Constructed in Discovery Modeling 23
2. Solid/Skin Model With Coincident Faces 24
3. Meshing Application Interface 37
4. Boundary Zone Type and Continuum Zone Type Specifications in Ansys Fluent 47
5. Multibody Part Containing All Fluid Bodies in the DesignModeler Application 56
6. Named Selections Defined in Meshing Application 57
7. Boundary Zone Names and Types Transferred to Ansys Fluent 58
8. Continuum Zone Names and Types Transferred to Ansys Fluent 59
9. Multibody Part Containing Mix of Solid and Fluid Bodies in the DesignModeler Application 60
10. Multibody Part Being Edited in the Meshing Application 61
11. Changing the Fluid/Solid Material Property of a Body 62
12. Continuum Zone Names and Types Transferred to Ansys Fluent 63
13. Meshed Model (Four Separate Workbench Parts) Ready for Export to Ansys ICEM CFD 67
14. Opening the .prj File (Four Separate Workbench Parts) in Ansys ICEM CFD 68
15. Meshed Model (One Multibody Workbench Part) Ready for Export to Ansys ICEM CFD 69
16. Opening the .prj File (One Multibody Workbench Part) in Ansys ICEM CFD 70
17. Meshed Model (with Named Selections) Ready for Export to Ansys ICEM CFD 71
18. Fluid1_Fluid2 Named Selection 71
19. InterfaceSolidFluid2 Named Selection 72
20. SharedEdge Named Selection 72
21. SharedVertices Named Selection 73
22. Opening the .prj File (with Named Selections) in Ansys ICEM CFD 74
23. Part, Body, and Named Selection Names in the Meshing Application 76
24. Objects/Zone Names Transferred to Ansys Fluent Meshing 76
25. First Contact Region: One Contact and One Target 80
26. Second Contact Region: One Contact, Two Targets 80
27. Third Contact Region: One Contact, Two Targets 81
28. Fourth Contact Region: One Contact, Two Targets 81
29. Geometry with Cyclic Redundancies 83
30. Proximity Sizing Limitation 108
31. Washers Generated Around Two Holes 117
32. Washer Element Nodes Not Moved 118
33. Washer Element Nodes Moved 119
34. Mesh Metrics Bar Graph 134
35. Geometry View After Selecting an Individual Bar 135
36. Clicking and Holding on an Individual Bar 136
37. Bar Graph Controls Page 137
38. Quadrilateral Aspect Ratio Calculation 141
39. Aspect Ratios for Quadrilaterals 141
40. Jacobian Ratio (MAPDL) 145
41. Jacobian Ratio (Corner Nodes) 146
42. Jacobian Ratio (Gauss Points) 147
43. Shell Average Normal Calculation 148
44. Shell Element Projected onto a Plane 148
45. Quadrilateral Shell Having Warping Factor 149
46. Warping Factor for Bricks 149
47. Parallel Deviation Unit Vectors 150
48. Parallel Deviations for Quadrilaterals 150
49. Maximum Corner Angles for Triangles 150

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

50. Maximum Corner Angles for Quadrilaterals 151
51. Ideal and Skewed Triangles and Quadrilaterals 151
52. Vectors Used to Compute Orthogonal Quality for a Cell 153
53. Vectors Used to Compute Orthogonal Quality for a Face 155
54. Inflation into Volume Mesh Methods 162
55. Last Aspect Ratio Option 167
56. Different Numbers of Layers Are Respected 172
57. Portion of Project Tree 173
58. Different Numbers of Layers Are Not Respected 173
59. Layer Compression vs. Stair Stepping Option (Full Mesh View) 176
60. Layer Compression vs. Stair Stepping Option (Detail View) 176
61. Maximum Angle = 140 178
62. Maximum Angle = 180 178
63. Fillet Ratio = 0.0 179
64. Fillet Ratio = 0.5 179
65. Fillet Ratio = 1.0 179
66. Triangle Surface Mesher = Program Controlled 183
67. Triangle Surface Mesher = Advancing Front 183
68. Protecting Topology 186
69. Locations of Pinch Controls 188
70. Mesh Generated Without Pinch Controls 189
71. Mesh Generated With Pinch Controls 189
72. Automatic Pinch Control for Edges on Left; Manual Pinch Control Required for Edges on Right 191
73. Mesh Generated with Automatic Pinch Control and Manual Pinch Control on Similar Geometry 191
74. Pinch Not Recommended for Models with Multiple Complications 196
75. Mixed Order Meshing of a Multibody Part 205
76. Mixed Order Elements 206
77. Geometry Input to Patch Independent Tetra Mesher 211
78. Full Tetrahedron Enclosing the Geometry 211
79. Full Tetrahedron Enclosing the Geometry in Wire Frame Mode 212
80. Cross-Section of the Tetrahedron 212
81. Mesh After Capture of Surfaces and Separation of Useful Volume 213
82. Final Mesh Before Smoothing 213
83. Final Mesh After Smoothing 214
84. Example (a) Showing Base Geometry 223
85. Example (b) Min Size Limit (Described Below) Set to 1 223
86. Example (c) Min Size Limit (Described Below) Set to 0.5 224
87. Example (d) Defeature Size Set to 1 224
88. Example (e) Defeature Size Set to 1 and Element Order Set to Linear 225
89. Example (f) Defeature Size Set to 1 and Min Size Limit Set to 0.5 225
90. Example (a) Showing Base Geometry 229
91. Example (b) Default Patch Independent Tetrahedron Mesher 229
92. Example (c) Patch Independent Tetrahedron Mesher with Min Size Limit Set to Capture Curvature 230
93. Effect of Smooth Transition Setting 230
94. Element Edge Lengths Smaller Than Specified Element Size 232
95. Sweep Method Would Require Slicing to Obtain Pure Hex Mesh 243
96. MultiZone Generates Pure Hex Mesh without Slicing 243
97. Free Mesh Type = Tetra 251
98. Free Mesh Type = Tetra/Pyramid 252
99. Free Mesh Type = Hexa Dominant 253
100. Free Mesh Type = Hexa Core 254

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

101. Source / Target Face Selection for MultiZone 255
102. Layered Tetrahedrons Mesh 268
103. Sweeping a Closed Torus 298
104. Resulting Mesh for Closed Torus 299
105. Inside Corner Vertex 321
106. Face Vertex Types 323
107. Seven-sided Planar Face 324
108. Example Face Mesh—Side Inside Corner Vertex 325
109. Example Face Mesh—Corner Inside Corner Vertex 325
110. Mesh Copy Scope 333
111. Generated Mesh 333
112. Match Controls Used with Thin Sweeping 336
113. Coordinate Systems for Arbitrary Mesh Matching 339
114. Matched Mesh 340
115. Snap to Boundary Set to Yes 343
116. Snap to Boundary Set to No 343
117. Axis Sweep Representation 401
118. Edge Only Sweep Path 401
119. Edge Plus Closed Surface Sweep Path 402
120. Example (a) Showing Invalid Closed Cylindrical Face as Source Face 403
121. Example (b) Valid Open Cylindrical Face as Source Face 403
122. Example (c) Multiple Connected Side Faces 404
123. Axial Sweep Model 404
124. Axial Sweep Model: Face Meshing Control 405
125. Axial Sweep Model: Hard Edge Sizing Control 406
126. Axial Sweep Model: Meshed 406
127. Example (a) N Source to 1 Target or 1 Target to N Source Topology 408
128. Example (b) N Source to N Target Topology 409
129. Example (c) 1 Source to N Target Mesh 409
130. Example (d) N Source to 1 Target Mesh 410
131. Example (e) N Source to N Target Mesh 410
132. Using Virtual Topology to Create Single Edge Between Source/Target Faces 411
133. Example (a) Mapped Face Control Applied to Target Is Ignored 411
134. Example (b) Mapped Face Control Applied to Source Is Respected 412
135. Thin Solid Sweeper Used to Mesh a Single Body Part 413
136. Thin Solid Sweeper Used to Mesh a Single Body Part: Detail 413
137. Thin Solid Sweeper Used to Mesh a Multibody Part 414
138. Thin Solid Sweeper and Laminated Composite Models 415
139. Ambiguous Source Face Definition for Laminated Composite Model 416
140. Recommended Source Face Definition for Laminated Composite Model 417
141. Thin Solid Sweeper Limitation 418
142. Adding Face Projections (Splits) in the DesignModeler Application 418
143. Defining Source Faces when Face Splits Are Present 419
144. Three Plates Model Meshed with Thin Solid Sweeper 419
145. Blocking Algorithm—Sample Geometry 420
146. Blocking Algorithm—Step 1: 2D Blocking 421
147. Blocking Algorithm—Step 2: 3D Blocking 421
148. Blocking Algorithm—Step 3: Inflation 422
149. Classifying the Problem: Sources 423
150. Classifying the Problem: Handling of Paths and Imprints 424
151. Collective Source Faces 424

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

152. Classifying the Problem: Sweep Path 425
153. Valve Body: Traditional Approach 425
154. Valve Body: Automatic Source Faces with MultiZone 426
155. Source Imprinting Classifications: Internal Loops 428
156. Source Imprinting Classifications: Boundary Loops 428
157. Source Imprinting Classifications: Multiple Internal Loops 429
158. Source Imprinting Classifications: Multiple Connected Internal Loops View 1 429
159. Source Imprinting Classifications: Multiple Connected Internal Loops View 2 430
160. Source Imprinting Classifications: Parallel Loops 431
161. Source Imprinting Classifications: Intersecting Loops View 1 431
162. Source Imprinting Classifications: Intersecting Loops View 2 432
163. Simple Cutout Case 433
164. Cutouts at Multiple Levels 433
165. Intersections Between Levels and Sides 434
166. Meshed Model 434
167. Vertices in a Split Circle 435
168. 360 ° Cutout 435
169. Internal Loops along Side Faces of the Sweep Path 436
170. Map Face Control Assigned to Side Faces 437
171. Connecting Faces Assigned as Source Faces 437
172. Using Inflation on Cylindrical Side Faces 438
173. Using Inflation on Cylindrical Side Faces 439
174. Fillets and MultiZone 439
175. Fillets and Inflation 440
176. Fillets as Side Faces 440
177. Sphere of Influence on Face that Doesn't Intersect Edges 443
178. Two Boxes with Sizing on One Face 446
179. Mesh Generated for Entire Part 446
180. Selective Meshing: Left Body First 447
181. Selective Meshing: Right Body First 447
182. Mesh Worksheet 449
183. Mesh Worksheet Step Deactivation 453
184. Sweep Method With Inflation: Hex Fill 455
185. Sweep Method With Inflation: Wedge Fill 456
186. Swept Body Shares Source/Target Face With Tet Body 459
187. Defining Inflation for a Swept Body Sharing Source/Target Face With Tet Body 459
188. Detail of Proper Connections on the Common Interface 460
189. Tet Body Surrounds Swept Body 460
190. Detail of Well-aligned Layers Between the Swept and Tet Regions 461
191. 2D Rigid Body Contact Meshing 465
192. Error Handling for Instances 468
193. Cylinder Containing Baffles 471
194. Section Cut Showing Baffle Meshing 471
195. Detail of Inflation on Baffles 472
196. Setting the Contact Match Tolerance 499
197. Contact Match with Gaps Between "Primary" and "Secondary" Bodies 499
198. Viewing the "Primary" and "Secondary" Bodies in Auxiliary Windows 505
199. Previewed Inflation Mesh 535
200. Section Plane View of Previewed Inflation Mesh 535
201. Mesh Objects Grouped By Type 543
202. Merge Face Edges Off 548

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

203. Merge Face Edges On 548
204. Single Face Virtual Cell Limitations 550
205. Formation of Virtual Faces 551
206. Virtual Faces After Operation 551
207. Formation of Virtual Edges 552
208. Gauss Curvature Angle 553
209. Curvature Angle at 25, 60, and 120 degrees 553
210. Feature Angle 554
211. Feature Angle at 20, 40, and 80 Degrees 554
212. Aspect Ratio at 0.2, 0.5, and 0.9 554
213. Contact Angle at 270, 330, and 355 degrees 555
214. Edge Angle at 80, 100, and 150 degrees 555
215. Shared Boundary Ratio at 0.2, 0.3, and 0.4 555
216. Small Edges Between Faces 556
217. Small Edges Removed 557
218. Small Edges Attached to the Same Faces 557
219. Small Edge Repair with Edge Merge 558
220. Sliver Face 558
221. Sliver Repair 559
222. Small Face 559
223. Small Face Repair 560
224. Original Virtual Split Edge with Dependent Virtual Split Edge 563
225. Unlocked Dependent Splits 564
226. Locked Dependent Splits 564
227. Overridden Locked Dependent Splits 564
228. Types of Faces Requiring Two Virtual Split Face Operations 566
229. Splits Requiring a Series of Virtual Split Face Operations 567
230. Virtual Topology Properties Dialog: Example 1 574
231. Virtual Topology Properties Dialog: Example 2 574
232. Virtual Topology Properties Dialog: Example 3 575
233. Virtual Topology Properties Dialog: Example 4 576
234. Virtual Topology Properties Dialog: Example 5 576
235. Obsolete Mesh 581
236. Failed Mesh 582
237. Example with Missing Face 586
238. Problematic Topology Highlighted During Meshing 587
239. Failed Surface Mesh Due to Protected Topology 587
240. Mesh Respecting Protected Topology 588
241. Patch Independent Tet Mesh Failure Due to Geometry Gap 588
242. Patch Independent Tet Mesh Failure Corrected with Larger Mesh 589
243. Edge Biasing Not Respected by MultiZone 594
244. Edge Biasing Respected by MultiZone 595

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

List of Tables

1. Washer Limitations 119
2. Mesh Matching for Symmetrical Parts 462
3. Rigid Body Meshing: Default Behaviors for Rigid Dynamics, Transient Structural, and Explicit Dynamics Analyses 464
4. Mesh Matching for Gaps 499

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Meshing: Capabilities in Ansys Workbench

The following topics are discussed in this section.

Meshing Overview
Meshing Implementation in Ansys Workbench
Types of Meshing
Conformal and Non-Conformal Meshing

Meshing Overview

Philosophy

The goal of meshing in Ansys Workbench is to provide robust, easy to use meshing tools that will simplify the mesh generation process. These tools have the benefit of being highly automated along with having a moderate to high degree of user control.

Physics Based Meshing

When the Ansys Meshing application is launched (that is, edited) from the Ansys Workbench Project Schematic, the physics preference will be set based on the type of system being edited. For analysis systems, the appropriate physics is used (p. 39). For a Mechanical Model system, the **Mechanical** physics preference is used. For a Mesh system, the physics preference defined in **Tools > Options > Meshing > Default Physics Preference** (p. 389) is used.

Meshing Implementation in Ansys Workbench

The meshing capabilities are available within the following Ansys Workbench applications. Access to a particular application is determined by your license level.

- The Ansys Mechanical application - Recommended if you plan to stay within the Ansys Mechanical application to continue your work (preparing and solving a simulation). Also, if you are planning to perform a Fluid-Structure Interaction problem, and desire to use a single project to manage your Ansys Workbench data, you can use the Mechanical application to perform your fluid meshing. This is most conveniently done in a separate model branch from the structural meshing and structural simulation.
- The Ansys Meshing application (p. 27) - Recommended if you plan to use the mesh to perform physics simulations in Ansys CFX or Ansys Fluent. If you wish to use a mesh created in the Meshing

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

application for a solver supported in the Mechanical application, you can replace the Mesh system with a Mechanical Model system. See Replacing a Mesh System with a Mechanical Model System (p. 36).

Note:

In the 2025 R1 release, Ansys Autodyn runs inside the Mechanical application. The recommendation is to use an Explicit Dynamics analysis system, in which meshing comes as part of that system. As an alternative, you can also use this system to prepare a model for the traditional Ansys Autodyn application (AUTODYN component system). For simple Ansys Autodyn models, you can use the meshing tools within the traditional Ansys Autodyn application (AUTODYN component system).

Types of Meshing

The following types of meshing are discussed in this section.

Part/Body Level Meshing

Mesh Methods

Part/Body Level Meshing

In part-based meshing, parts are meshed individually and have no connections (other than mesh connections (p. 485) or node merge (p. 508)).

Assemblies can also be meshed using part-based meshing methods, but in such cases the mesher operates one part at a time, and therefore cannot mesh virtual bodies or evaluate parts that occupy the same space.

Mesh Methods

This section describes types of meshing in terms of element shapes. Applicable mesh control options are presented for each element shape shown below, and operate at the part level unless otherwise noted. See the Method Control (p. 202) section for further details.

Tet Meshing

- Patch Conforming Tetrahedron Mesher (p. 208)
- Patch Independent Tetrahedron Mesher (p. 210)
- Layered Tetrahedrons Mesher (p. 266)
- Automatic (PrimeMesh) (p. 287)

Hex Meshing

- Swept Mesher (p. 235)
- Hex Dominant Mesher (p. 233)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Thin Solid Mesher (p. 237)
- MultiZone Mesher (p. 242)

Cartesian Meshing

- Body-Fitted Cartesian (p. 261) (part level)

Quad Meshing

- Quad Dominant (p. 282)
- MultiZone Quad/Tri (p. 284)

Triangle Meshing

- All Triangles (p. 284)

Conformal and Non-Conformal Meshing

This section describes meshing options for Conformal and Non-conformal meshing.

Conformal Meshing Within a Part

Conformal Meshing Across Parts

Connections/Interface Handling

Conformal Meshing Within a Part

A conformal mesh can be generated in a variety of ways when meshing in Ansys Workbench. In general, a part can be used to prescribe a region where you want a conformal mesh. When a part is meshed, only the geometry in that part is sent to the mesher. So, geometry in other parts will generally not influence the mesh of a given part (there are some exceptions to this, for example, contact sizing controls). To create a conformal mesh consisting of multiple different regions or bodies, a multibody part should be used.

The following notes apply to multibody parts:

- Multibody parts need to be formed in DesignModeler or Discovery Modeling.
- There are different types of multibody parts. The types of multibody parts prescribe how the interface between the different bodies should be handled. Options include:
  - **Shared Topology**: Merges the pair(s) of common faces between bodies into a single set of common faces (2 faces become 1 face). The shared faces belong to both of the neighboring bodies. When meshing these types of multibody parts, the mesh at the interface will be conformal.
  - No Shared Topology: Leaves the pair(s) of common faces between bodies separate. The separate faces are meshed separately so the mesh would be non-conformal. Contact detection (p. 24) should pick up these duplicate faces so that you can easily see the non-conformal interfaces.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ Often it is helpful to imprint the bodies with each other so that there are pairs of common faces with the face boundaries being the same. This could help in creating more similar mesh at the non-conformal interface between the bodies. DesignModeler has **Imprint** as a method for **Shared Topology**. It is similar to **None**, but it will imprint all bodies in the multibody part with each other.

→ In some cases this option is used to help organize bodies into parts, but it is important to note that this has ramifications. A multibody part will have all bodies meshed using only 1 meshing process. If instead the bodies are in separate parts, parallel part by part meshing will be utilized and could significantly reduce the meshing time.

→ **Patch Independent Tetrahedrons** has an option **Match Mesh Where Possible** that will try to make the mesh conformal across bodies even without shared topology.

Note:

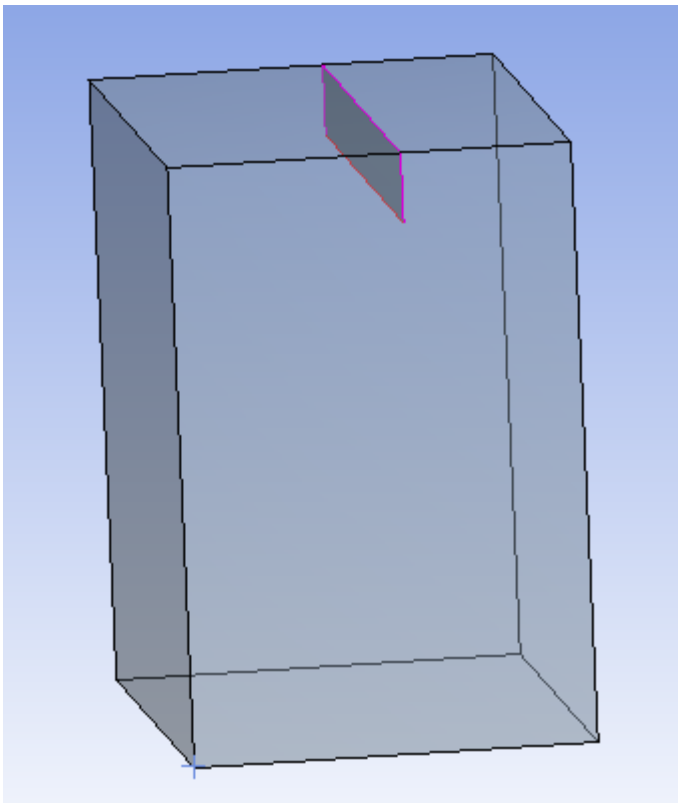
The **Patch Independent Tetrahedrons** method is being deprecated and will be removed in future releases.

- It is common to use multibody parts in hex meshing. The approach is to slice the model into sweepable bodies and use **Shared Topology** to get conformal mesh between those sweepable bodies.
- Generally multibody parts are formed from all solid bodies or all sheet bodies or all line bodies.
  - Multibody parts of solids, sheets, and line bodies are not allowed.
  - Multibody parts of solids and sheets are allowed, but note the following:
    → Solids are meshed first by default. You can use selective meshing (p. 444) to obtain a different behavior.
    → In Discovery Modeling, the **Shared Topology** options: **Share** and **Merge**, both create shared topology. Use **Merge** when trying to embed sheets within a solid to construct a zero-thickness or baffle model. Otherwise, use the **Share** option.

    Figure 1: Baffle Model Constructed in Discovery Modeling (p. 23) shows a model with one solid body with a baffle face embedded in it. This model was constructed from one solid body and one sheet body, using the **Merge** option in Discovery Modeling.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1: Baffle Model Constructed in Discovery Modeling



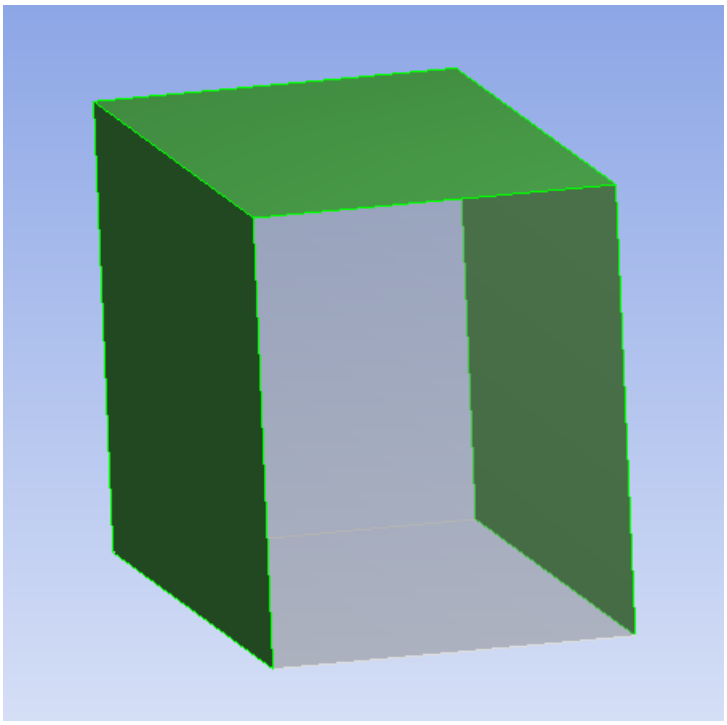
To construct the same model in Ansys DesignModeler, use the **Automatic** option for **Shared Topology**.

→ Solid/skin models can be constructed using the **Share** option in Discovery Modeling. Like with other multibody parts, the common faces will become shared.

Solid/skin models where the sheets are coincident to the solid's faces are supported (Figure 2: Solid/Skin Model With Coincident Faces (p. 24)).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 2: Solid/Skin Model With Coincident Faces



  - Multibody parts of sheets and beams (line bodies) (p. 462) are supported.

- For related information, refer to Geometry Introduction in the Mechanical help.

Conformal Meshing Across Parts

In some cases, you may want to get conformal mesh without having to use **Shared Topology**.

For example, maybe instances are being used to create repetitive copies of the mesh, and you want to get conformal mesh between the copies. **Shared Topology** could be used within the part that is being copied, but each copy would have to be a unique part. Or, maybe meshing speed is paramount so you want to use parallel part by part meshing and later connect the meshes. There are several mesh-based ways to make the mesh conformal across parts. These include:

- Mesh connections (p. 485): This approach starts from an existing shell mesh and imprints and connects the mesh to make it conformal.

- Node merge (p. 508): After meshing, you can insert a node merge operation to make coincident mesh (nodes close together within a tolerance) conformal. Merge nodes can be used in conjunction with match mesh (p. 334) or contact match mesh (p. 496) controls to make the nodes coincident prior to merging.

Connections/Interface Handling

When coming into the Meshing/Mechanical application, by default, connections are found between parts that have faces in proximity of each other. This automatic detection of connections or interfaces can be disabled if it is not desired. The **Auto Detect Contact On Attach** can be changed by selecting

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tools> Options from the Ansys Workbench main menu, and then selecting either the Mechanical or Meshing category as appropriate. The option is enabled by default in both applications.

These contact regions can be used for mesh sizing (p. 311), and/or are used to define the behavior on how the contact/interface is treated when running the simulation.

For structural solvers see the description of connections in the Ansys Mechanical help.

For CFD solvers, contact regions are used differently by the Ansys CFX and Ansys Fluent solvers:

- Contact regions are used in Ansys CFX as General Grid Interface (GGI) definitions. For details, refer to the documentation available under the Help menu within CFX.
- Contact regions are used in Ansys Fluent as mesh interfaces. See Special Cases (p. 54) for details.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Meshing: Usage in Workbench

The Ansys Meshing application is a separate Ansys Workbench application. The Meshing application is data-integrated with Ansys Workbench, meaning that although the interface remains separate, the data from the application communicates with the native Ansys Workbench data. The following topics are addressed in this section:

Basic Meshing Application Workflows
Strategies for CFD/Fluids Meshing in Ansys Workbench
Accessing Meshing Functionality
Overview of the Meshing Application Interface
Determination of Physics, Analysis, and Solver Settings
Working with Legacy Mesh Data
Exporting Meshes or Faceted Geometry
Extended Ansys ICEM CFD Meshing
Working with Meshing Application Parameters
Ansys Workbench and Mechanical APDL Application Meshing Differences

Basic Meshing Application Workflows

The following sections describe several basic workflows for using the Meshing application in Ansys Workbench:

Overview of the Meshing Process in Ansys Workbench
Overview of the Meshing Process for CFD/Fluids Analyses
Overview of the Meshing Process for Hydrodynamics Analysis
Combining CFD/Fluids Meshing and Structural Meshing

Overview of the Meshing Process in Ansys Workbench

The following steps provide the basic workflow for using the Meshing application as part of an Ansys Workbench analysis (non-Fluid Flow). Refer to the Ansys Workbench help for detailed information about working in Ansys Workbench.

1. Select the appropriate template in the Toolbox, such as *Static Structural*. Double-click the template in the Toolbox, or drag it onto the Project Schematic.

2. If necessary, define appropriate engineering data for your analysis. Right-click the Engineering Data cell, and select **Edit**, or double-click the Engineering Data cell. The Engineering Data workspace appears, where you can add or edit material data as necessary.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Attach geometry to your system or build new geometry. Right-click the Geometry cell and select **Import Geometry...** to attach an existing model or select **New SpaceClaim Geometry...** or **New DesignModeler Geometry...** to launch the Discovery Modeling or Ansys DesignModeler application, respectively.

4. Access the Meshing application functionality. Right-click the Model cell and choose **Edit**. This step will launch the Ansys Mechanical application.

5. Once you are in the Mechanical application, you can move between its components by highlighting the corresponding object in the Tree as needed. Select the **Mesh** object in the Tree to access Meshing application functionality and apply mesh controls.

6. Define loads and boundary conditions. Right-click the Setup cell and select **Edit**. The appropriate application for your selected analysis type will open (such as the Mechanical application). Set up your analysis using that application's tools and features.

7. You can solve your analysis by issuing an Update, either from the data-integrated application you're using to set up your analysis, or from the Ansys Workbench GUI.

8. Review your analysis results.

Note:

You should save your data periodically (**File> Save Project**). The data will be saved as a `.wbpj` file. Refer to the Ansys Workbench help for more information about project file management in Workbench.

For more information:

- For information on identifying and correcting mesh failures, refer to Meshing: Troubleshooting (p. 579).
- For information about using the Ansys Meshing application to import or export mesh files, refer to Working with Legacy Mesh Data (p. 40) and Exporting Meshes or Faceted Geometry (p. 42).
- Fluids users of the Ansys DesignModeler, Ansys Meshing, and Ansys CFX applications should refer to Named Selections and Regions for Ansys CFX (p. 77) for important information about region definitions.
- Fluids users of the Ansys DesignModeler, Ansys Meshing, and Ansys Fluent applications should refer to Fluent Mesh Export (p. 43) for important information about Named Selection support.

Overview of the Meshing Process for CFD/Fluids Analyses

This section describes the basic process for using the Ansys Meshing application to create a mesh as part of an Ansys Workbench CFD/fluids analysis. Refer to Strategies for CFD/Fluids Meshing in Ansys Workbench (p. 33) for information about different CFD/Fluids meshing strategies. Refer to the Ansys Workbench help for detailed information about working in Ansys Workbench. There are four basic steps to creating a mesh:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create Geometry

You can create geometry for the Meshing application in the Discovery Modeling or Ansys DesignModeler application. You can also import the geometry from an external CAD file.

The Meshing application requires you to construct solid bodies (not surface bodies) to define the region for the 3D mesh (for 2D simulations a sheet body can be used). A separate body must be created for each region of interest in the fluids simulation. For example, a region in which you want the fluids solver to solve only heat transfer must be created as a separate body. Multiple bodies are created in the DesignModeler application by using the **Freeze** command, see **Freeze** in the DesignModeler help for details.

It is best practice to explicitly identify any fluid regions in the model as fluids rather than solids.

For new users or new models it is often useful to first generate a default mesh, evaluate it, and then apply the controls described in Define Mesh Attributes (p. 29) as appropriate to improve various mesh characteristics.

Define Named Selections

During the fluids simulation setup, you will need to define boundary conditions where you can apply specific physics. For example, you may need to define where the fluid enters the geometry or where it exits. Although it may be possible to select the faces that correspond to a particular boundary condition inside the solver application, it is rather easier to make this selection ahead of time in either the CAD connection, the Ansys DesignModeler application, or the Meshing application. In addition, it is much better to define the location of periodic boundaries before the mesh is generated to allow the nodes of the surface mesh to match on the two sides of the periodic boundary, which in turn allows for a more accurate fluids solution.

Creating a Named Selection will affect how the mesher treats that topology. For details, see Protecting Topology Defined Prior to Meshing (p. 184).

You can define the locations of boundaries by defining Named Selections, which can assist you in the following ways:

- You can use Named Selections to easily hide the outside boundary in an external flow problem.
- You can assign Named Selections to all faces in a model except walls, and **Program Controlled** (p. 164) inflation will automatically select all walls in the model to be inflation boundaries.

For more information:

- Fluids users of the Ansys DesignModeler, Ansys Meshing, and Ansys CFX applications should refer to Named Selections and Regions for Ansys CFX (p. 77).
- Fluids users of the Ansys DesignModeler, Ansys Meshing, and Ansys Fluent applications should refer to Fluent Mesh Export (p. 43).

Define Mesh Attributes

The mesh generation process in the Meshing application is fully automatic. However, you have considerable control over how the mesh elements are distributed. To ensure that you get the best fluids

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

solution possible with your available computing resources, you can dictate the background element size, type of mesh to generate, and where and how the mesh should be refined. In general, setting up the length scale field for your mesh is a three-step process, as outlined below:

- Assign a suitable set of global mesh controls (p. 93).
- Override the default mesh type by inserting a different mesh method (p. 202).
- Override the global sizing or other controls locally on bodies, faces, edges, or vertices and the regions close to them by scoping local mesh controls (p. 201).

Generate Mesh

When you are ready to compute the mesh, you can do so by using either the **Update** feature or the **Generate Mesh** feature. Either feature computes the entire mesh. The surface mesh and the volume mesh are generated at one time. The mesh for all parts/bodies is also generated at one time. For help in understanding the difference between the **Update** and **Generate Mesh** features, see Updating the Mesh Cell State (p. 527).

For information on how to generate the mesh for selected parts/bodies only, refer to Generating Mesh (p. 528). The Previewing Surface Mesh (p. 531) and Previewing Inflation (p. 534) features are also available if you do not want to generate the entire mesh at one time.

Once the mesh is generated, you can view it by selecting the **Mesh** object in the Tree Outline. You can define Section Planes to visualize the mesh characteristics, and you can use the Mesh Metric (p. 133) feature to view the worst quality element based on the quality criterion for a selected mesh metric.

Note:

Fluids users should refer to Generation of Contact Elements (p. 483) for recommendations for defining contact for fluids analyses.

Overview of the Meshing Process for Hydrodynamics Analysis

This section describes the basic process for using the ANSYS Meshing application to create a mesh as part of a hydrodynamics analysis. Refer to the *Aqwa User's Manual* for more information about hydrodynamics analysis. There are three basic steps to creating a mesh:

Create Geometry

You can create geometry for the Meshing application in the Discovery Modeling or Ansys Design-Modeler applications, or import a geometry from an external file. The model should consist of surface and line bodies only (no solid bodies). See General Modeling Requirements in the *Aqwa User's Manual* for full details of what is required for creating geometry for a hydrodynamics analysis.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select Physics Preference

Before performing meshing, select **Hydrodynamics** as your Physics Preference (p. 96).

Note:

Hydrodynamics can be set as the default preference by selecting **Tools** > **Options**, clicking the **Meshing** heading, and changing the **Default Physics Preference** to **Hydrodynamics**. For more information, see Meshing: Options (p. 389).

Once defined, you can define the following mesh controls:

- Element Size (p. 102)
- Defeature Size (p. 111)

Define Mesh Attributes

The mesh generation process in the Meshing application is fully automatic. The Hydrodynamics preference corresponds to the uniform sizing with quad dominant method. However, you have considerable control over how the mesh elements are distributed. To ensure that you get the best solution possible with your available computing resources, you can dictate the background element size, type of mesh to generate, and where and how the mesh should be refined. In general, setting up your mesh is a two-step process, as outlined below:

- Assign suitable global mesh sizes within the Hydrodynamics Physics Preference
- Override the global sizing locally on bodies, faces, edges, or vertices and the regions close to them by scoping local mesh controls (p. 201)

Generate Mesh

When you are ready to compute the mesh, you can do so by using either the **Update** feature or the **Generate Mesh** feature. The mesh for all parts/bodies is also generated at one time. For help in understanding the difference between the **Update** and **Generate Mesh** features, see Updating the Mesh Cell State (p. 527).

Once the mesh is generated, you can view it by selecting the **Mesh** object in the Tree Outline. You can use the Mesh Metric (p. 133) feature to view the worst quality element based on the quality criterion for a selected mesh metric.

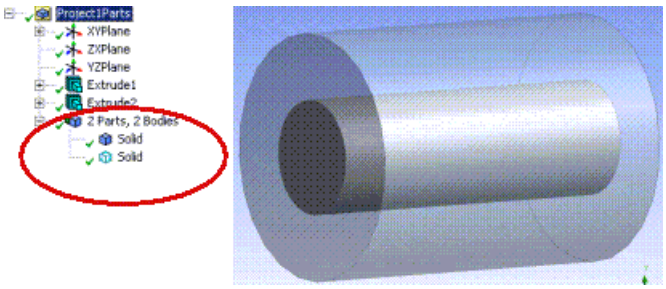
Combining CFD/Fluids Meshing and Structural Meshing

In some applications, a CFD/fluids mesh *and* a structural mesh are required within the same workflow. For these one-way coupling applications, the loading, solving, and postprocessing of the fluids meshed part(s) later occur in a Fluid Flow analysis system, while the loading, solving, and postprocessing of the structurally meshed part(s) later occur in a Structural analysis system. The best approach for this kind of application is to break the model into separate parts rather than use a continuous multibody part.

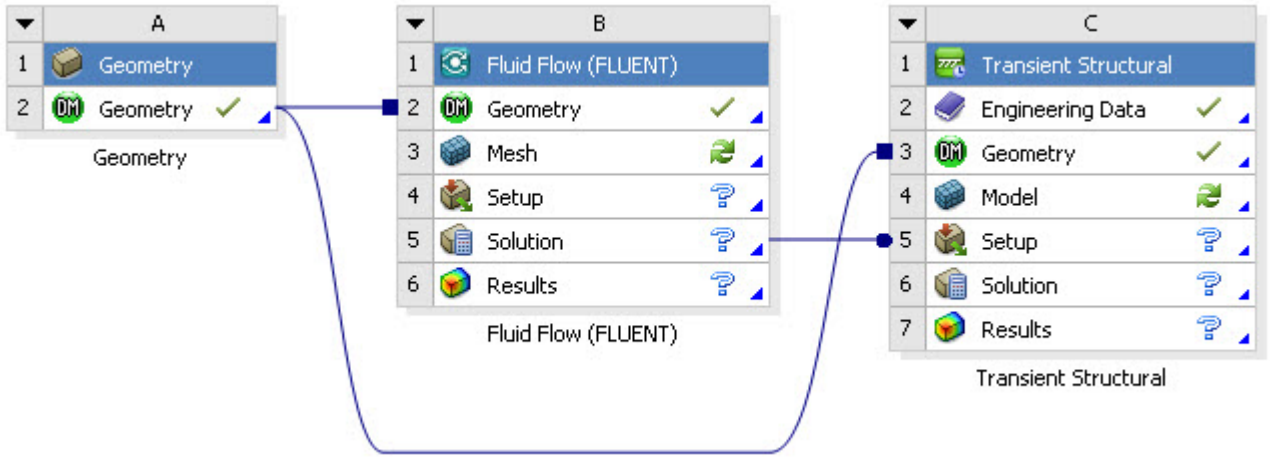
The following approach is recommended for these applications:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Attach the model to a Geometry (DesignModeler) system and use the Explode Part feature to create independent parts within the model.

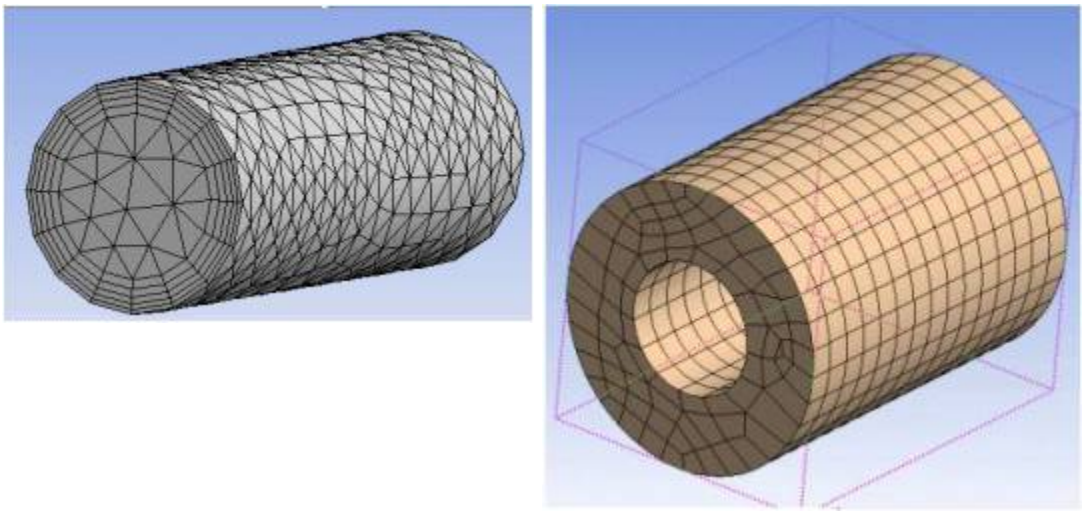


2. Link a Fluid Flow analysis system and a Structural analysis system to the Geometry system. The geometries may be shared or not, depending on whether defeaturing needs to be done to one or the other system. Dedicate the Fluid Flow analysis system to meshing the appropriate fluid domain for the fluids application. Suppress the structural part(s) in the model. Dedicate the Structural analysis system to meshing the appropriate structural part(s). Suppress the fluid part(s) in this model.



In this case, only the respective parts are meshed. The mesh of the Fluid Flow analysis system is shown below on the left, and the mesh of the Structural analysis system is shown on the right.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

- You can set up the workflow schematic in different ways depending on various factors, including variations in the fluid/structural model, persistence, the desired multiphysics simulation, and so on.
- The coupling of the solvers is also handled from the Project Schematic. For details, refer to the discussion about creating and linking a second system in the Ansys Workbench help.
- For geometry persistence, both models will require updating when changing CAD parameters.

Strategies for CFD/Fluids Meshing in Ansys Workbench

Ansys Workbench offers various strategies for CFD/Fluids meshing. For each strategy, certain defaults are in place to target the particular needs of an analysis. The strategies and circumstances in which each of them are appropriate are described below. In all cases, your first decision is to determine whether you want to use assembly meshing (p. 20) or part-based meshing.

Tetra Dominant Meshing - Patch Conforming Tetra/Prism Meshing

The first strategy is to use conformal tetra/prism meshing plus the default Sweep (p. 235) method. This strategy is recommended for models involving moderately clean CAD (for example, native CAD, Parasolid, ACIS, and so on) for which you desire a tetra/hybrid dominant mesh.

Although the Patch Conforming Tetra (p. 208) mesh method is fully automated, it interacts with additional mesh controls and capabilities as necessary, including:

- Advanced tetra and inflation layer technology
- Pinch (p. 187) controls for removing small features at the mesh level (offered as an alternative to Virtual Topologies (p. 545), which work at the geometry level)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Sizing (p. 105) controls for providing greater control over mesh distribution
- Conformal swept regions (p. 458)
- Body of influence (p. 305) (BOI) for setting one body as a size source for another body

Tetra Dominant Meshing - Patch Independent Tetra/Prism Meshing

An alternative for those desiring a tetra dominant mesh is Patch Independent Tetra (p. 210)/Prism meshing. This approach is best for "dirty CAD"—CAD models with many surface patches (for example, IGES, CATIA V4, and so on) and in cases with large numbers of slivers/small edges/sharp corners. It includes support for post inflation (p. 173), as well as CAD simplification built-in to the tetra mesher.

Note:

The **Patch Independent Tetrahedrons** method is being deprecated and will be removed in future releases.

Mapped Hex Meshing - All Hex Swept Meshing

This mapped hex approach (which includes both general Sweep (p. 399) and thin Sweep (p. 406)) is recommended for clean CAD. It supports single source to single target volumes, and may require you to perform manual geometry decomposition.

Benefits of this approach include:

- Support for Sizing (p. 105) controls
- Compatibility with Patch Conforming Tetra (p. 208) meshing
- Support for swept inflation (p. 454)

Mapped and Free Meshing - MultiZone Meshing

Best for moderately clean CAD, the MultiZone (p. 242) strategy for meshing provides multi-level sweep with automatic decomposition of geometry into mapped (structured) and free (unstructured) regions. When defining the MultiZone mesh method, you can specify a **Mapped Mesh Type** and a **Free Mesh Type** that will be used to fill structured and unstructured regions respectively. Depending on your settings and specific model, the mesh may contain a mixture of hex/prism/tetra elements.

The MultiZone mesh method and the Sweep (p. 235) mesh method described above operate similarly. However, MultiZone has capabilities that make it more suitable for a class of problems for which the Sweep method would not work without extensive geometry decomposition.

Additional benefits of this approach include:

- Support for 3D inflation
- Ability to selectively ignore small features

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Accessing Meshing Functionality

You can access Meshing application functionality from the Model/Mesh cell in an analysis system, or from the Mesh cell in a Mesh component system. Before using the steps provided in this section, you should be familiar with the concepts of analysis systems and component systems in Ansys Workbench.

Accessing Meshing Functionality from an Analysis System

The Model cell (Mesh cell in Fluid Flow analysis systems) allows you to access a meshing application or share a mesh with another system. Model corresponds to the contents of the Model branch within the Ansys Mechanical application and allows you to perform physics-based meshing capabilities, such as spot welds, contact, etc. Mesh contains just node coordinates and mesh connectivity.

To launch the Meshing application from a Model cell in an analysis system (non-Fluid Flow):

1. From the Analysis Systems group of the Ansys Workbench Toolbox, either double-click or drag an analysis system onto the Project Schematic. As a result, a template for that type of analysis system appears in the Project Schematic.

2. In the analysis system, right-click the Geometry cell and choose **New SpaceClaim Geometry...** or **New DesignModeler Geometry...** to create geometry within the Discovery Modeling or Ansys DesignModeler application respectively or choose **Import Geometry** to attach existing geometry.

3. Right-click the Model cell and choose **Edit**. This step will launch the Mechanical application. From the Mechanical application, you can access the Meshing application controls by clicking on the **Mesh** object in the Tree Outline.

To access meshing from a Mesh cell in a Fluid Flow analysis system:

1. From the Analysis Systems group of the Ansys Workbench Toolbox, either double-click or drag a Fluid Flow analysis system onto the Project Schematic. As a result, a template for that type of analysis system appears in the Project Schematic.

2. In the analysis system, right-click the Geometry cell and choose **New SpaceClaim Geometry...** or **New DesignModeler Geometry...** to create geometry within the Discovery Modeling or Ansys DesignModeler application respectively or choose **Import Geometry** to attach existing geometry.

3. Right-click the Mesh cell and choose **Edit**. This step will launch the appropriate mesh application (for example, the Meshing application, etc.).

Accessing Meshing Functionality from a Mesh Component System

To launch the Meshing application from a Mesh component system:

1. From the Component Systems group of the Ansys Workbench Toolbox, either double-click or drag a Mesh component system onto the Project Schematic. As a result, a template of a Mesh system appears in the Project Schematic.

2. In the Mesh system, right-click the Geometry cell and choose **New SpaceClaim Geometry...** or **New DesignModeler Geometry...** to create geometry within the Discovery Modeling or Ansys DesignModeler application respectively or choose **Import Geometry** to attach existing geometry.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Right-click the Mesh cell and choose **Edit**. This step will launch the appropriate mesh application (for example, the Meshing application, etc.).

Replacing a Mesh System with a Mechanical Model System

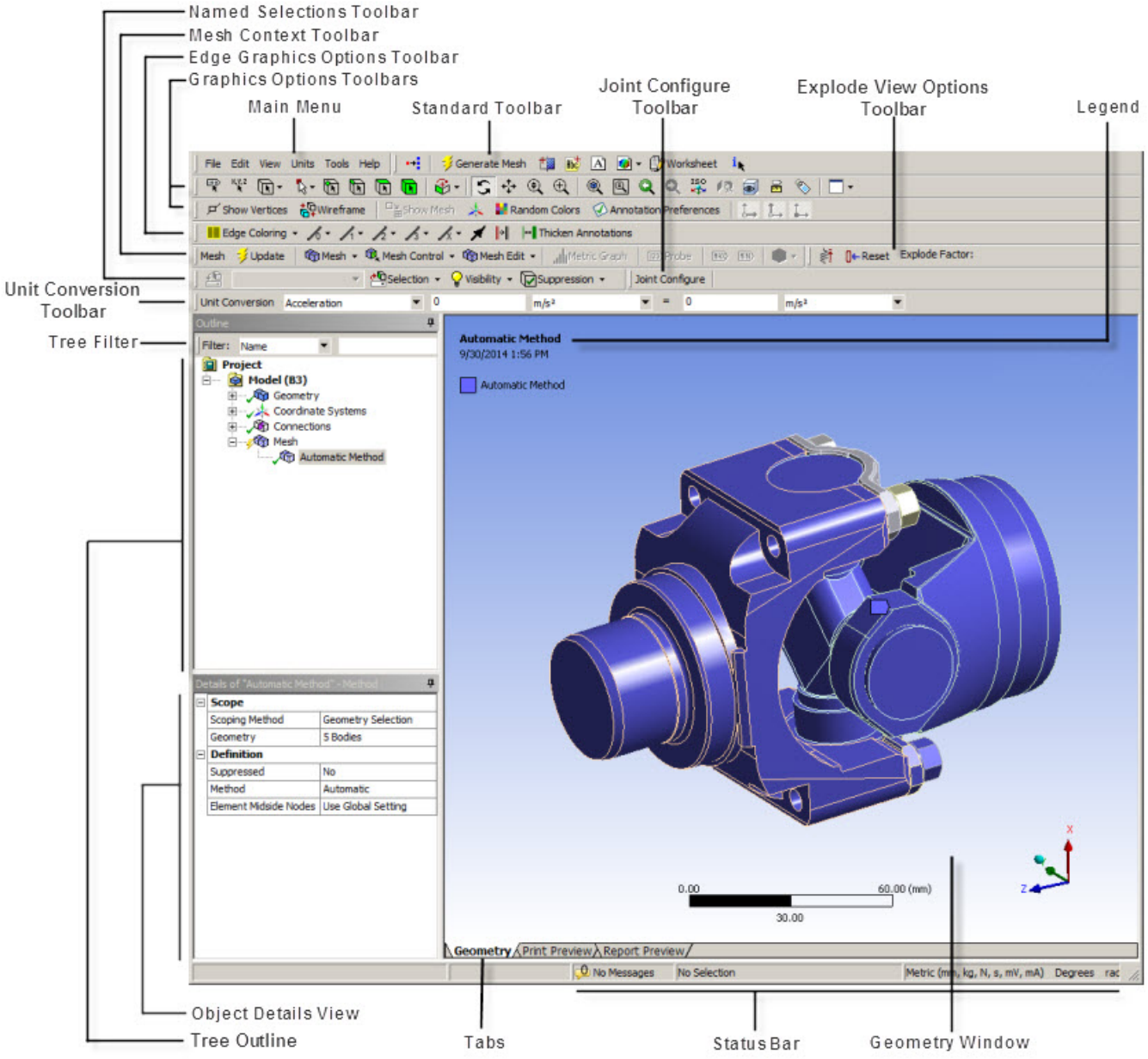
You can replace a Mesh component system with a Mechanical Model component system. This system can then be shared with any analysis system. For details, refer to the description of Mechanical Model systems in the Ansys Workbench help.

Overview of the Meshing Application Interface

The intuitive Meshing application interface, which is shown in Figure 3: Meshing Application Interface (p. 37), facilitates your use of all meshing controls and settings (p. 89).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3: Meshing Application Interface



The functional elements of the interface are described in the following table.

Note:

The links in the table redirect you to topics in the Ansys Mechanical help that contain supplemental information. Not all of the items described in the Mechanical help are available in the Meshing application interface.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Window Component	Description
Main Menu	This menu includes the basic menus such as **File** and **Edit**.
Standard Toolbar	This toolbar contains commonly used application commands.
Graphics Toolbar	This toolbar contains commands that control pointer mode or cause an action in the graphics browser.
Context Options Toolbar	This toolbar contains task-specific commands that change depending on where you are in the Tree Outline.
Unit Conversion Toolbar	Not visible by default. This toolbar allows you to convert units for various properties.
Named Selections Toolbar	Not visible by default. This toolbar contains options to manage Named Selections similar to how they are managed in the Mechanical application.
Edge Graphics Options Toolbar	This toolbar provides quick access to features that are intended to improve your ability to distinguish edge and mesh connectivity in a surface body model.
Explode View Options Toolbar	This toolbar is a graphical display feature used to create imaginary distance between geometry bodies (only) of your model for viewing purposes.
Tree Outline	Outline view of the project. Always visible. Location in the outline sets the context for other controls. Provides access to object's context menus. Allows renaming of objects. Establishes what details display in the Details View.
Details View	The Details View corresponds to the Outline selection. Displays a details window on the lower left panel which contains details about each object in the Outline.
Geometry Window (also sometimes called the Graphics window)	Displays and manipulates the visual representation of the object selected in the Outline. This window may display: • 3D Geometry • 2D/3D Graph • Spreadsheet • HTML Pages • Scale ruler

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Window Component	Description
	• Triad control • Legend **Note:** • Hover your mouse over the time stamp in the legend and RMB click to toggle the time stamp off and on. • The Geometry window may include splitter bars for dividing views. • See ANSYS.com > Support > Platform Support for a complete list of 3Dconnexion products certified with the current release of ANSYS applications.
Reference Help	Opens an objects reference help page for the highlighted object. These are the same pages that are available in the Mechanical application.
Tabs	The document tabs that are visible on the lower right portion of the window.
Status Bar	Brief in-context tip. Selection feedback.
Splitter Bar	Application window has up to 3 splitter bars.

Determination of Physics, Analysis, and Solver Settings

Most systems in Ansys Workbench are defined by three primary attributes: physics type, analysis type, and solver type. The method you use to launch the Meshing application functionality determines how default physics, analysis, and/or solver settings are defined:

- Mesh systems, which are a type of component system, are unfiltered (physics, analysis, and solver). If you launch the Meshing application from a Mesh component system (p. 35), your preferences will be set to the defaults you previously defined within the Meshing application. See Meshing Overview (p. 19) for more information.
- If you launch the Meshing application from an analysis system (p. 35) (whether it be via the Model cell in a non-Fluid Flow analysis system or the Mesh cell in a Fluid Flow analysis system),

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

your Physics, Analysis, and Solver settings will be set according to the selected type of analysis system. To change the **Physics Preference**, you must use the Details View of the **Mesh** folder.

Note:

- To view the physics, analysis, and solver types that are defined for an analysis system, right-click the Model cell (non-Fluid Flow analyses) or Mesh cell (Fluid Flow analyses) and select Properties. This step will open the Properties window, where you can view the attributes. For example, for an *Electric* system, the Properties window will show that Physics is Electric, Analysis is Steady-State Electric Conduction, and Solver is Mechanical APDL.
- Mechanical Model systems, which are a type of component system, are unfiltered (physics and solver). For details, refer to the discussion of Mechanical Model systems in the Ansys Workbench help.

For more information:

- For a list of analysis systems available in Ansys Workbench and basic steps for building each type of system, refer to the discussion of analysis systems in the Ansys Workbench help.
- For details about the various types of non-Fluid Flow analyses and how to perform them, refer to the discussion of analysis types in the Ansys Mechanical help.
- For details about Fluid Flow analyses and how to perform them, refer to the documentation available under the Help menu within Ansys CFX or Ansys Fluent.

Working with Legacy Mesh Data

You can import legacy mesh files using the following methods. The method that is best for you depends on the type of file that you want to import and how you intend to use it:

- Choose **File> Import** from the Ansys Workbench Menu bar or click the **Import** button on the Ansys Workbench Toolbar to read legacy Ansys Workbench mesh data
- Right-click the Mesh cell and choose **Import Mesh File** to import a read-only mesh for downstream use
- Use the **External Model** system to read third-party mesh formats for other Ansys Workbench systems. You can use the **External Model** system for importing the following file formats:
  - NASTRAN Bulk Data (.bdf, .dat, .nas)
  - Abaqus Input (.inp)
  - Fluent Input (.msh, .cas)
  - ICEM CFD Input (.uns)
  - LS-DYNA Input (.k)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Importing Using File> Import or the Import Button

If you choose **File> Import** or click the **Import** button from Ansys Workbench, you can import mesh files that have an extension of .cmdb or .meshdat. Doing so creates a Mesh system in the Ansys Workbench Project Schematic. When the Mesh cell is edited, the mesh will open in the Meshing application where you can edit it.

For more information about reading a simulation/mesh database (.dsdb/.cmdb) from previous ANSYS versions, refer to the discussion of importing legacy databases in the Ansys Workbench help.

Importing Read-only Meshes for Downstream Application Use

You can right-click a Mesh cell and choose **Import Mesh File** to browse to a mesh file that you want to import, provided the file is of one of the following types:

- Ansys CFX mesh file with extension .gtm or .cfx
- Ansys ICEM CFD mesh file with extension .cfx, cfx5, or .msh
- Ansys Fluent mesh file with extension .cas, .msh, or .gz
- Ansys Polyflow Classic mesh file with extension .poly, .neu, or .msh

---

Note:

When you use this method, in the strictest sense you are not "importing" the mesh file. That is, you will not be able to edit the file in the Meshing application using this method. Rather, you are making the mesh available for downstream systems.

To be able to edit these types of files in the Meshing application, you must import the mesh into the **External Model** application, and then into another system.

The **Import Mesh File** method is enabled when:

- No geometry is associated with the Geometry cell.
- No generated mesh is associated with the Mesh cell. (Imported meshes do not disable the **Import Mesh File** menu item.)
- No incoming connections are associated with the Geometry cell or Mesh cell.
- No outgoing connections are associated with the Geometry cell.
- No outgoing connections from the Mesh cell are connected to the Mechanical APDL or Ansys Autodyn applications.

When you import the mesh to the Mesh cell:

- The Geometry cell is deleted.
- The title of the cell changes from "Mesh" to "Imported Mesh."
- The state of the Mesh cell is "Up to Date."

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- No incoming connections are allowed.
- Outgoing connections can be established with a Mechanical APDL, Ansys Autodyn, Ansys CFX, Ansys Fluent, or Ansys Polyflow Classic system.
- The Geometry cell in the target system is deleted.
- Using the reset command (right-clicking on the Imported Mesh cell and choosing **Reset**) deletes the imported mesh.

Exporting Meshes or Faceted Geometry

A mesh generated by the Meshing application can be exported into the following file formats:

- Meshing File format (*.meshdat), suitable for import into Ansys Workbench (p. 43)
- Ansys Fluent mesh format (*.msh), suitable for import into Ansys Fluent
- Ansys Polyflow Classic format (*.poly), suitable for import into Ansys Polyflow Classic
- CGNS format (*.cgns), suitable for import into a CGNS-compatible application
- Ansys ICEM CFD format (*.prj), suitable for import into Ansys ICEM CFD
- TGrid Faceted Geometry format (*.tgf), suitable for import into Fluent Meshing (formerly TGrid)

To export a mesh:

1. Generate the mesh.
2. If necessary, select an **Export Unit** (p. 100) or **Export Format** (p. 100) from the **Defaults** group in the Details View.
3. Select **File> Export** from the main menu.
4. In the **Save As** dialog box, choose a directory and specify a file name for the file. Then choose a file type from the **Save as type** drop-down menu and click **Save**.

Note:

- You can also use the Meshing application to export faceted geometry for use in Ansys Fluent Meshing. In such cases you can skip step 1 above. A file with the extension .tgf is created, suitable for import into Ansys Fluent Meshing.
- When the same entity is a member of more than one Named Selection, those Named Selections are said to be "overlapping". If you are exporting a mesh into the Ansys Fluent, Ansys Polyflow Classic, CGNS, or Ansys ICEM CFD format (or faceted geometry into the Ansys Fluent Meshing format), and overlapping Named Selections are detected, the export will fail and you must resolve the overlapping Named Selections before proceeding. Any Named Selection whose **Send to Solver** (p. 78) option is set to **No** is skipped during the check for overlapping entities in Named Selections. For this reason, an easy way to avoid

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

overlapping Named Selections is to set all values of **Send to Solver** to **No**. For details, see Repairing Geometry in Overlapping Named Selections (p. 79).

For details, refer to:

Mesh Application File Export

Fluent Mesh Export

Polyflow Classic Export

CGNS Export

Ansys ICEM CFD Export

Exporting Faceted Geometry to Ansys Fluent Meshing

Named Selections and Regions for Ansys CFX

Passing Named Selections to the Solver

Repairing Geometry in Overlapping Named Selections

Resolving Overlapping Contact Regions

Mesh Application File Export

When you export a mesh file to Meshing File format (**File> Export** from the Meshing application main menu, then **Save as type Meshing File**), a file with the extension .meshdat is created. The exported file can be imported as a legacy file into Ansys Workbench by either selecting **File >Import** (p. 40) from the Menu bar or clicking the **Import** button on the Toolbar, and then selecting an **Importable Mesh File**. This will create a Mesh System in the Ansys Workbench Project Schematic.

Fluent Mesh Export

When you export a mesh file to Ansys Fluent mesh format (**File> Export** from the Meshing application main menu, then **Save as type Fluent Input Files**), a mesh file with the extension .msh is created. The exported mesh file is suitable for import into another product such as Ansys Fluent, or into Ansys Fluent Meshing outside of Ansys Workbench. For more control over the input file, refer to Meshing Options on the Options Dialog Box (p. 389).

If the mesh file you export contains quadratic elements, all midside nodes will be dropped during export. That is, all element types will be exported as linear element types for Ansys Fluent.

An orientation check/correction will be performed for 3D geometry models exported as 2D mesh such that all 2D cells will have the same orientation. You do not need to manually correct the orientation of the geometry face(s).

When the mesh file is exported to Ansys Fluent mesh format, the material properties of the bodies/parts in the model must be translated to proper continuum zone types (p. 46) for use in Ansys Fluent. To provide this information to Ansys Fluent, the following logic is used:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. If Physics Preference (p. 96) is set to **CFD** and you do not use either of the methods described in steps 2 or 3 below to explicitly assign a body/part to be either solid or fluid, all zones are exported to Ansys Fluent mesh format as *FLUID* zones by default.

   Note:

   An exception to the above involves models that include an enclosure. If you used the Enclosure feature in the Ansys DesignModeler application, the enclosure body will be assigned a continuum zone type of *FLUID* by default.

2. For models created/edited in the DesignModeler application, a **Fluid/Solid** material property can be assigned to a solid body or a multibody part (if the multibody part contains at least one solid body). This material assignment appears under **Details of Body** in the Details View of the DesignModeler application.

   For multibody parts, you can change the material property for all bodies in one operation in the DesignModeler application by modifying the **Fluid/Solid** property for the multibody part and the modification will propagate to any solid bodies in the part. Similarly, you can use the DesignModeler application to modify the **Fluid/Solid** property for a solid body that belongs to a multibody part, and the **Fluid/Solid** property for the multibody part will be modified accordingly.

   When exported to Fluent Meshing format, a body/part with a material property of **Solid** will be assigned a continuum zone type of *SOLID* and a body/part with a material property of **Fluid** will be assigned a continuum zone type of *FLUID*. This setting in the DesignModeler application overrides the default behavior described in step 1.

   Note:

   Refer to Figure 5: Multibody Part Containing All Fluid Bodies in the DesignModeler Application (p. 56) for an example that illustrates where to set the **Fluid/Solid** material property in the DesignModeler application.

3. Finally, the **Fluid/Solid** material property setting in the Meshing application is considered. This material assignment appears in the Details View of the Meshing application when a prototype (Body object) is selected in the Tree Outline. Similar to the DesignModeler application feature described in step 2, the Meshing application lets you change the **Fluid/Solid** material property for a body.

   When exported to Ansys Fluent mesh format, a body with a material property of **Solid** will be assigned a continuum zone type of *SOLID* and a body with a material property of **Fluid** will be assigned a continuum zone type of *FLUID*. This setting in the Meshing application

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

overrides any assignments that were made based on the default behavior described in step 1 or the **Fluid/Solid** setting described in step 2.

Note:

If there are multiple continuum zones or boundary zones of the same type in the DesignModeler application or the Meshing application, each zone name in the exported Ansys Fluent mesh file will contain the necessary prefix and an arbitrary number will be appended to the name to make it unique. Refer to Examples of Fluent Mesh Export: An Alternative to Using a Fluid Flow (Fluent) Analysis System (p. 55) for an example that illustrates multiple zones of the same type.

For related information, refer to:

Classes of Zone Types in Ansys Fluent

Standard Naming Conventions for Naming Named Selections

Zone Type Assignment

Examples of Fluent Mesh Export: An Alternative to Using a Fluid Flow (Fluent) Analysis System

---

Note:

- Contact regions are said to be "overlapping" when the same entity (face or edge) is a member of more than one contact region or when multiple contact regions share the same geometry. Ansys Fluent does not support overlapping contact regions. For part-based meshing only, if you are exporting a mesh into the Ansys Fluent format and overlapping contact regions are detected, the software will attempt to combine the regions. If it is unable to combine them, the export will fail and you must resolve the overlapping contact regions manually before proceeding. You can use the **Check Overlapping Contact Regions** option to identify the problematic contact regions. For details, see Resolving Overlapping Contact Regions (p. 79).

- If you are performing a 2D analysis and intend to export to Ansys Fluent, you should disable the **Auto Detect Contact On Attach** option to avoid problems that may otherwise occur upon export. You can access this option by selecting **Tools> Options** from the Ansys Workbench main menu, and then selecting either the Mechanical or Meshing category as appropriate. The option is enabled by default in both applications.

- Fluent mesh export may fail if you are using shared licensing, no licenses are available, and Ansys Fluent is running already. In such cases, the error is due to shared licensing restrictions, but the error message that is issued does not identify licensing as the cause.

- For additional information about importing files into Ansys Fluent or Ansys Fluent Meshing, refer to the documentation available under the Help menu within the respective product.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Classes of Zone Types in Ansys Fluent

Zone type specifications in Ansys Fluent define the physical and operational characteristics of the model at its boundaries and within specific regions of its domain. There are two classes of zone type specifications:

- Boundary zone types (sometimes called "face zones")
- Continuum zone types (sometimes called "cell zones")

Boundary zone type specifications, such as *WALL* or *INLET_VENT*, define the characteristics of the model at its external or internal boundaries. Boundary zones are collections of faces in 3D, and collections of edges in 2D. By default, a boundary zone of type *WALL* is created for the boundary of each body in the geometry during export to Ansys Fluent mesh format. Continuum zone type specifications, such as *FLUID* or *SOLID*, define the characteristics of the model within specified regions of its domain. By default, a continuum zone is created for each body in the geometry during export to Ansys Fluent mesh format. (By default, continuum zone types will be assigned as described in Fluent Mesh Export (p. 43).)

If you do not want the default zone type to be assigned to a boundary zone, you can set the Auto Zone Type Assignment (p. 389) option to **Off**. You can then override default type of *WALL* by defining Named Selections and naming them according to the conventions (p. 48) provided. All faces belonging to the same body in a Named Selection are placed into a single boundary zone.

Note:

For 2D models, you can group sheet surface bodies into a Named Selection, and the underlying faces contained in the sheet surface bodies will be placed into a single continuum zone (as long as the faces themselves are not contained in a Named Selection).

The following sections further describe boundary zone type and continuum zone type specifications and illustrate their purposes in the definition of an example computational model involving simple geometry.

For details on how zones are named and how zone types are assigned during export, refer to Zone Type Assignment (p. 49).

For an example that illustrates the basic workflow for using Ansys Workbench to create a model in the Ansys DesignModeler application, mesh it in the Meshing application, and export the mesh to Ansys Fluent, refer to Examples of Fluent Mesh Export: An Alternative to Using a Fluid Flow (Fluent) Analysis System (p. 55). In the example, Named Selections are defined in the Meshing application and the correct Ansys Fluent zone names/types are assigned in the exported Fluent mesh file based on those definitions.

Comparing and Contrasting Boundary Zone Types and Continuum Zone Types

Boundary zone type specifications define the physical and operational characteristics of the model at those topological entities that represent model boundaries. For example, if an *INFLOW* boundary zone type is assigned to a face entity that is part of three-dimensional model, the model is defined such that material flows into the model domain through the specified face. Likewise, if a *SYMMETRY* boundary zone type is assigned to an edge entity that is part of a two-dimensional model, the model is defined such that flow, temperature, and pressure gradients are identically zero along the

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

specified edge. As a result, physical conditions in the regions immediately adjacent to either side of the edge are identical to each other.

Continuum zone type specifications define the physical characteristics of the model within specified regions of its domain. For example, if a *FLUID* continuum zone type is assigned to a body entity, the model is defined such that equations of momentum, continuity, and species transport apply at mesh nodes or cells that exist within the body. Conversely, if a *SOLID* continuum zone type is assigned to a body entity, only the energy and species transport equations (without convection) apply at the mesh nodes or cells that exist within the body.

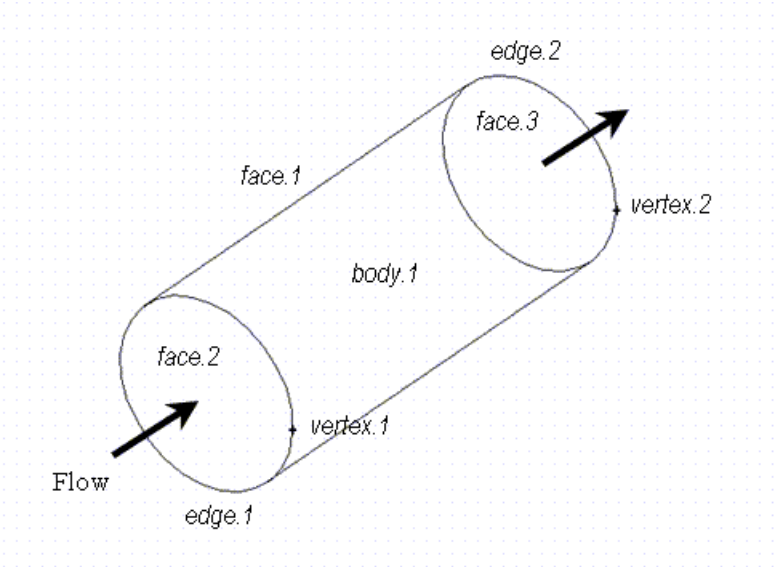
Generally speaking, entities are assigned to zone type classes in Ansys Fluent as shown in the following table:

Model Dimension	Entity	Zone Type Class
3D	Body	Continuum
	Face	Boundary
2D	Face	Continuum
	Edge	Boundary

The Effect of Zone Type Specifications

As an example of the effect of zone type specifications on the specification of a computational model, consider the geometry shown in Figure 4: Boundary Zone Type and Continuum Zone Type Specifications in Ansys Fluent (p. 47), which consists of a single volume in the shape of a straight, elliptical cylinder. The geometry includes one body, three faces, two edges, and two vertices.

Figure 4: Boundary Zone Type and Continuum Zone Type Specifications in Ansys Fluent



The geometry shown in Figure 4: Boundary Zone Type and Continuum Zone Type Specifications in Ansys Fluent (p. 47) can be used to model many different types of transport problems, including fluid flow through a straight, elliptical pipe and heat conduction through a solid, elliptical rod. The following table shows the zone type specifications associated with the fluid flow problem using the geometry shown in Figure 4: Boundary Zone Type and Continuum Zone Type Specifications in Ansys Fluent (p. 47).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity	Zone Class	Zone Type
face.1	Boundary	*WALL*
face.2	Boundary	*INFLOW*
face.3	Boundary	*OUTFLOW*
body.1	Continuum	*FLUID*

The following table shows the zone type specifications associated with the heat conduction problem using the geometry shown in Figure 4: Boundary Zone Type and Continuum Zone Type Specifications in Ansys Fluent (p. 47).

Entity	Zone Class	Zone Type
face.1	Boundary	*WALL*
body.1	Continuum	*SOLID*

Note:

For additional information about boundary (face) zones and continuum (cell) zones in Ansys Fluent, refer to the documentation available under the Help menu within Ansys Fluent.

Standard Naming Conventions for Naming Named Selections

This section is applicable to boundary zones only.

If you want to override the default boundary zone type assignments by using Named Selections, naming conventions have been provided for you to follow. Use these conventions when defining Named Selections in either the Ansys DesignModeler application or the Meshing application. By following these naming conventions consistently, you can ensure that Ansys Fluent boundary zone types will be assigned correctly and predictably in the exported Ansys Fluent file.

When naming Named Selections, it is best practice to specify the appropriate name from the list below exactly as shown (case-insensitive). In cases where the name shown below contains an underscore character, a hyphen is also acceptable (for example, both *EXHAUST_FAN* and *EXHAUST-FAN* will work). (Note, however, that although the Meshing application allows both hyphens and underscore characters to be used when defining Named Selections, the DesignModeler application allows only underscore characters.)

The name of each Named Selection is filtered upon export such that only allowable characters remain. Allowable characters include all alphanumeric characters as well as the following special characters:

```
` ! $ % ^ & * _ + - = : < > . ? /
```

All other characters, including spaces, are invalid. If an invalid character is used, it is replaced by an underscore (_) upon export.

In addition, the export process does allow for partial matches and special abbreviations, which are described in Zone Type Assignment (p. 49).

AXIS

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EXHAUST_FAN
FAN
INLET_VENT
INTAKE_FAN
INTERFACE
INTERIOR
MASS_FLOW_INLET
OUTFLOW
OUTLET_VENT
PERIODIC
POROUS_JUMP
PRESSURE_FAR_FIELD
PRESSURE_INLET
PRESSURE_OUTLET
RADIATOR
RECIRCULATION_INLET
RECIRCULATION_OUTLET
SYMMETRY
THIN
VELOCITY_INLET
WALL

Note:

For details about the boundary (face) zone and continuum (cell) zone types in Ansys Fluent, refer to the documentation available under the Help menu within Ansys Fluent.

Zone Type Assignment

This section describes zone naming and the zone type assignment process.

Zone Naming

By default, zones are named after the part and body from which they are derived. For example, part "part" and body "solid body" will result in a zone name of "part-solid_body." When the zone name is created:

- Any invalid characters (such as the space in "solid body") are replaced by an underscore character ("solid_body").
- Names that begin with a digit are prefixed by "zone."

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If the part name and the body name are identical, only the body name is used to create the zone name. The same rule applies to single body parts.

Note:

If you set the **Auto Zone Type Assignment** option to **Off** (as described in Meshing Options on the Options Dialog Box (p. 389)), all boundary zones are set to the default zone type *WALL*.

If a boundary zone was created for a Named Selection (as described in Classes of Zone Types in Ansys Fluent (p. 46)), the name of the boundary zone is set to the name of the Named Selection.

In cases where the zone naming process could lead to conflicting zone names (for example, in a situation where the potential exists for a zone name that is already in use to be used to name a new zone), one of the following approaches is used:

- If the zone type *is not similar to* the zone name in question, the zone type will be prefixed to the zone name to make it unique. For example, an existing continuum zone named "fluid" and a new boundary zone named "fluid" (with zone type *WALL*) will result in the boundary zone being renamed "wall-fluid."

  A unique integer will be suffixed to the zone name, preceded by a dot character (.). For example, an existing continuum zone named "solid" (with zone type *SOLID*) and another continuum zone named "solid" (with zone type *FLUID*) will result in the continuum zone being renamed "solid.1."

- If the zone type *is similar to* the zone name in question, the approach is dependent on whether the zone is a continuum zone or a boundary zone:

  – If multiple continuum zones have the same type and same zone name, they are merged into a single continuum zone.

  – If multiple boundary zones have the same type and same zone name, a unique integer will be suffixed to the zone name, preceded by a dot character (.). For example, an existing boundary zone named "wall" and a second boundary zone named "wall" will result in the second boundary zone being renamed "wall.1." Subsequent boundary zones named "wall" will be renamed "wall.2," "wall.3," and so on.

Because part and body names influence the creation of continuum zones in Ansys Fluent, it is best practice to assign distinct part and body names that follow these rules:

- Names should include only allowable characters. Allowable characters include all alphanumeric characters as well as the following special characters:

  ` ! $ % ^ & * _ + - = : < > . ? /

  All other characters, including spaces, are invalid. If an invalid character is used, it is replaced by an underscore (_) upon export.

- It is not sufficient for part and body names to differ by case only. This approach may lead to name conflicts for zones in Ansys Fluent, because names are converted to all lowercase characters upon export.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Using "INTER" as zone name or part of a zone name should be avoided. Because the mesher does not allow named selections with identical names and interface definitions require two named selections, using "INTER" can cause zone type conflicts.

Zone Type Assignment Process

The section is applicable to boundary zones only.

The zone type is derived from the zone name. To assign zone types, the string comparison operations detailed below are performed during the export process. These string comparison operations, which correspond to the naming conventions described in Standard Naming Conventions for Naming Named Selections (p. 48), are applied in the order in which they are listed below (that is, at first an exact match is tested, after that a partial match is tested, etc.) and are always *case-insensitive*. For example, fan, Fan, FAn, and faN are all exact matches for the 'FAN' string comparison and result in a zone type of *FAN* being assigned.

When the search operation begins, it will start by searching the first portion (or sub-string) of the string and if no match is found, it will search for a match anywhere in the string. For example, if a Named Selection with the name wall_inlet_flange is defined, it will be exported as zone type *WALL*. The 'inlet' portion of the name will have no effect on zone type assignment.

In some cases a name may match multiple patterns or rules, and the expected zone type may not be assigned. For this reason it is strongly advisable to use only unambiguous names that exactly match one of the rules presented below.

Once they are exported, names are *all* lowercase. The single quotation marks that are shown enclosing the strings below are not considered during the string comparison operations.

1. Exact matches are checked:

```
'AXIS'
'EXHAUST_FAN'
'FAN'
'INLET_VENT'
'INTAKE_FAN'
'INTERFACE'
'INTERIOR'
'INTERNAL'
'MASS_FLOW_INLET'
'OUTFLOW'
'OUTLET_VENT'
'POROUS_JUMP'
'PRESSURE_FAR_FIELD'
'PRESSURE_INLET'
'PRESSURE_OUTLET'
'RADIATOR'
'RECIRCULATION_INLET'
'RECIRCULATION_OUTLET'
'SYMMETRY'
'THIN'
'VELOCITY_INLET'
'WALL'
```

2. Partial matches are considered only if an exact match was not found in step 1:

```
'AXIS'
{'EXHAUST' && 'FAN'}
'FAN'
{'INLET' && 'VENT'}
{'INTAKE' && 'FAN'}
```

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
'INTERFACE'
'INTERIOR'
'INTERNAL'
{'MASS' && 'FLOW' && 'INLET'}
'OUTFLOW'
{'OUTLET' && 'VENT'}
{'POROUS' && 'JUMP'}
{'PRESSURE' && 'FAR' && 'FIELD'}
{'PRESSURE' && 'INLET'}
{'PRESSURE' && 'OUTLET'}
'RADIATOR'
{'RECIRCULATION' && 'INLET'}
{'RECIRCULATION' && 'OUTLET'}
'SYMMETRY'
{'VELOCITY' && 'INLET'}
```

3. String comparisons to the special abbreviations listed in the table below are performed if no match was found in step 1 or step 2. If an exact match to one of the strings listed in the table is found, the corresponding zone type is assigned:

When a match for this string is found...	This zone type is assigned...
`'CNDBY'`	*INTERFACE*
`'EXFAN'`	*EXHAUST FAN*
`'IFACE'`	*INTERFACE*
`'IN'`	*PRESSURE INLET*
`'INFAN'`	*INTAKE FAN*
`'INTERFACE'`	*INTERFACE*
`'IVENT'`	*INLET VENT*
`'MASFI'`	*MASS FLOW INLET*
`'OUT'`	*PRESSURE OUTLET*
`'OVENT'`	*OUTLET VENT*
`'PFAR'`	*PRESSURE FAR FIELD*
`'PORJ'`	*POROUS JUMP*
`'PRESF'`	*PRESSURE FAR FIELD*
`'PRESI'`	*PRESSURE INLET*
`'PRESO'`	*PRESSURE OUTLET*
`'PRESS'`	*PRESSURE FAR FIELD*
`'RAD'`	*RADIATOR*
`'RINLT'`	*RECIRCULATION INLET*
`'ROUT'`	*RECIRCULATION OUTLET*
`'SLIDE'`	*INTERFACE*

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When a match for this string is found...	This zone type is assigned...
'SYM'	*SYMMETRY*
'SYMET'	*SYMMETRY*
'SYMM'	*SYMMETRY*
'VELF'	*VELOCITY INLET*
'VELI'	*VELOCITY INLET*

4. Partial matches are considered if no match was found in steps 1, 2, or 3. If a partial match to one of the strings listed in the following table is found, the corresponding zone type is assigned:

When a match for this string is found...	This zone type is assigned...
'EXHAUST'	*EXHAUST FAN*
'INLET'	*VELOCITY INLET*
'OUTLET'	*PRESSURE OUTLET*
'SYM'	*SYMMETRY*
'THIN'	*THIN*
'WALL'	*WALL*

5. If none of the string comparisons described in steps 1, 2, 3, or 4 result in a match, the boundary zone is assigned zone type *WALL* by default.

Note:

- When Named Selections defined in the Ansys DesignModeler application are required in the Meshing application, you must set the appropriate geometry import options to ensure the Named Selections will be transferred properly. Refer to Importing DesignModeler Named Selections into the Meshing Application (p. 77) for details.

Handling Periodic Regions

To export matching face meshes as *PERIODIC* boundary zone types, you must perform the following steps:

1. Define matching face meshes (for 3D) or edge meshes (for 2D only) as **Periodic Regions** or **Cyclic Regions** in a **Symmetry** folder (p. 462) or as **Match Controls** or **Arbitrary Match Controls**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Define two **Named Selections** for each periodic boundary condition, one Named Selection per side, ensuring that "`periodic`" is used in the name (for example, `periodic1A`, `periodic1B`...). The same Named Selections could be used to define the periodic or cyclic region.

Note:

- Periodic regions must belong to the same Fluent mesh cell zone to ensure a successful export.
- When match controls on faces are used, only one periodic or cyclic transformation is supported (for example, you can export multiple match controls as long as they use the same coordinate system and have the same angle/translation).

Special Cases

Be aware of the following special cases related to boundary zone type assignment:

- If Physics Preference (p. 96) is set to **CFD** and no other zone assignment has been explicitly defined, all zones are exported as *FLUID* zones. See Fluent Mesh Export (p. 43) for more information.
- A boundary zone type of *INTERFACE* is assigned automatically to the contact source and contact target entities that compose contact regions at the time of mesh export. For this reason, you are not required to explicitly define *INTERFACE* zones to resolve contact regions. The zone type of *INTERFACE*, along with the contact information, is written to the mesh file.

  When reading the mesh file, Ansys Fluent creates a mesh interface for each contact region automatically. Each interface can consist of any number of *INTERFACE* zones for the source and any number of *INTERFACE* zones for the target.

  For 3D meshes, only face zones referenced in the contact region are considered during export. For 2D meshes, only face zones consisting of edges are considered. Each contact region can contain one or more face zones as sources and one or more face zones as targets. If any entity referenced in a contact region (either as source or target) is also contained in a Named Selection, that contact region is not considered during export.

  For more accurate results in cases in which one face zone in a pair has a higher element count than the other, the zone with the lower element count should be defined as **Interface Zone 1** of the mesh interface and the zone with the higher element count should be defined as **Interface Zone 2**. In the transfer of the contact region source and target regions it is ensured that the source regions are defined as interface zone 1 and the target regions as interface zone 2.
- A boundary zone type of *INTERNAL* is assigned automatically between two *FLUID* zones (sharing a common boundary) at the time of mesh export. For this reason, you are not required to explicitly define an *INTERNAL* zone in such cases if the name of this zone does not contain any special name derived from the part/body or Named Selection name.
- A boundary zone type of *WALL* is assigned automatically to a baffle, unless the baffle is part of a Named Selection that was defined in the Ansys DesignModeler application or the

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Meshing application, and the name of the Named Selection results in a different zone type assignment.

- A boundary zone type of *WALL* is assigned automatically between a *FLUID* zone and a *SOLID* zone at the time of mesh export. For this reason, you are not required to explicitly define a *WALL* zone in such cases. When reading the mesh file, Ansys Fluent will generate an additional *WALL SHADOW* zone automatically.

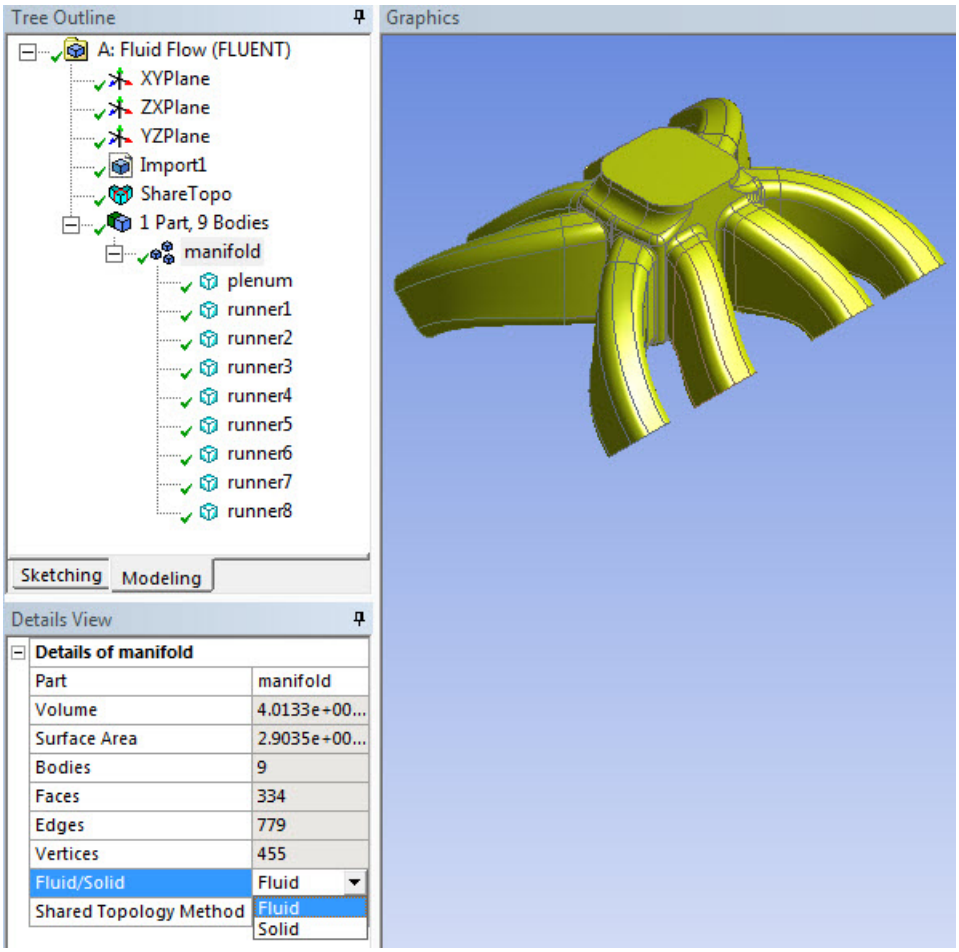
Examples of Fluent Mesh Export: An Alternative to Using a Fluid Flow (Fluent) Analysis System

The first example illustrates the basic workflow you can follow to create a multibody part in the Ansys DesignModeler application, mesh the model in the Meshing application, and export the mesh to Ansys Fluent. In the example, the bodies are renamed in the DesignModeler application, and Named Selections are defined in the Meshing application. Based on these definitions, Ansys Fluent zone names/types are assigned correctly and predictably (for both continuum and boundary zones) in the exported Fluent mesh file.

First, the model is imported into the DesignModeler application. The model consists of nine solid bodies after import. In the DesignModeler application, a multibody part is formed, the bodies are renamed, and all bodies are assigned a material property of fluid. (See Fluent Mesh Export (p. 43) for more information about the **Fluid/Solid** material property in the DesignModeler application.) Shared Topology is also used in this example. Refer to Figure 5: Multibody Part Containing All Fluid Bodies in the DesignModeler Application (p. 56).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

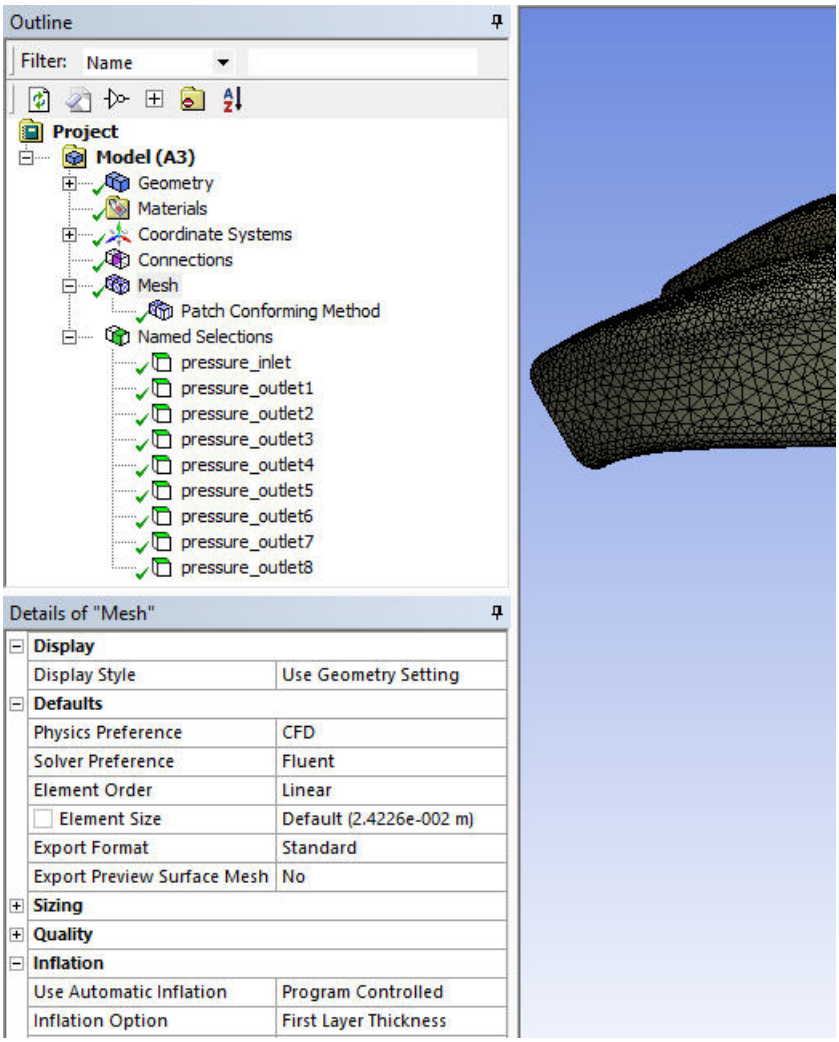
Figure 5: Multibody Part Containing All Fluid Bodies in the DesignModeler Application



Next, the model is edited in the Meshing application. The patch conforming mesh method is applied with inflation, and Named Selections are defined for boundary zones. Virtual Topology is also used in this example to provide geometry cleanup. Refer to Figure 6: Named Selections Defined in Meshing Application (p. 57).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

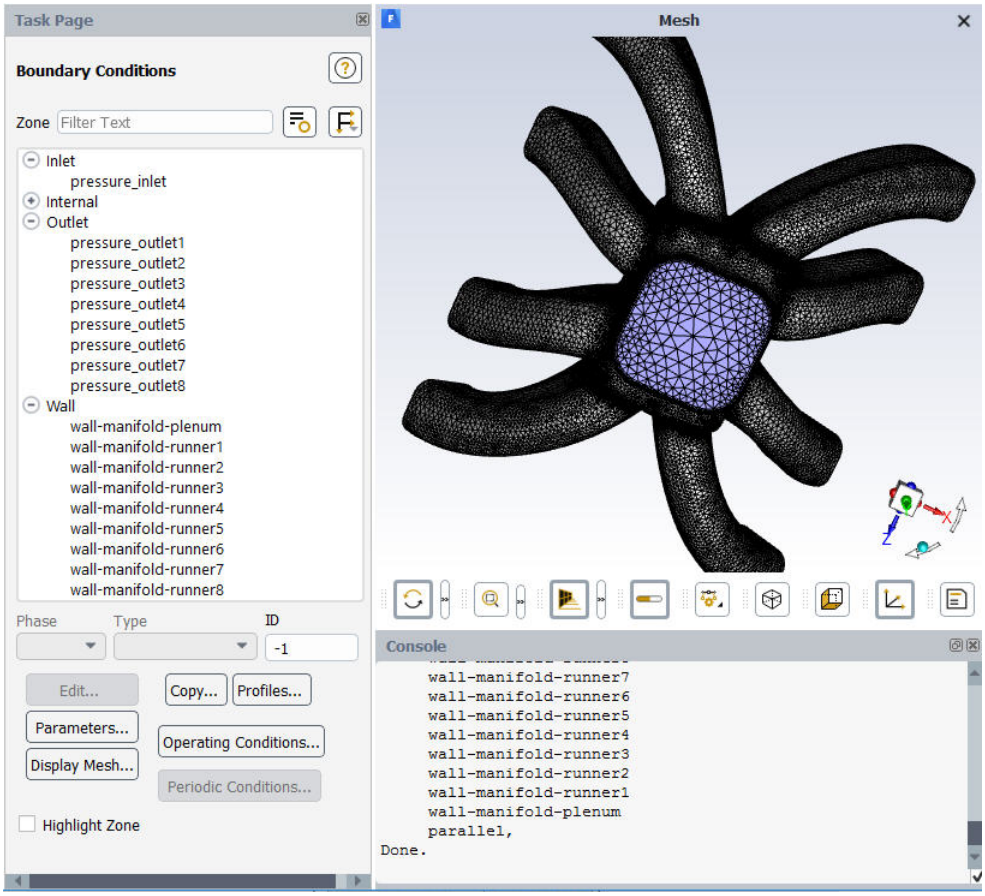
Figure 6: Named Selections Defined in Meshing Application



After meshing, the mesh is exported to Ansys Fluent format and read into Ansys Fluent. As shown in Figure 7: Boundary Zone Names and Types Transferred to Ansys Fluent (p. 58), the boundary zone names and types are transferred as expected.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

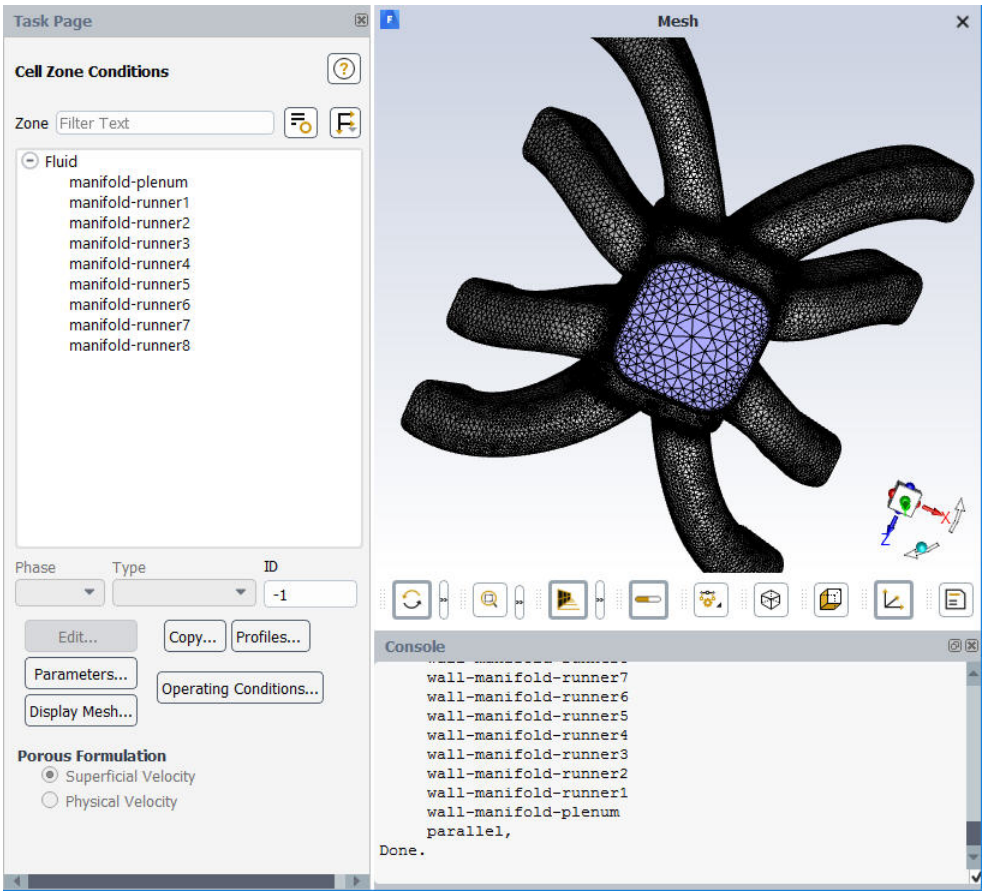
Figure 7: Boundary Zone Names and Types Transferred to Ansys Fluent



Similarly, continuum (or cell) zone names and types (in this case, all fluid) are transferred as expected. Refer to Figure 8: Continuum Zone Names and Types Transferred to Ansys Fluent (p. 59).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8: Continuum Zone Names and Types Transferred to Ansys Fluent



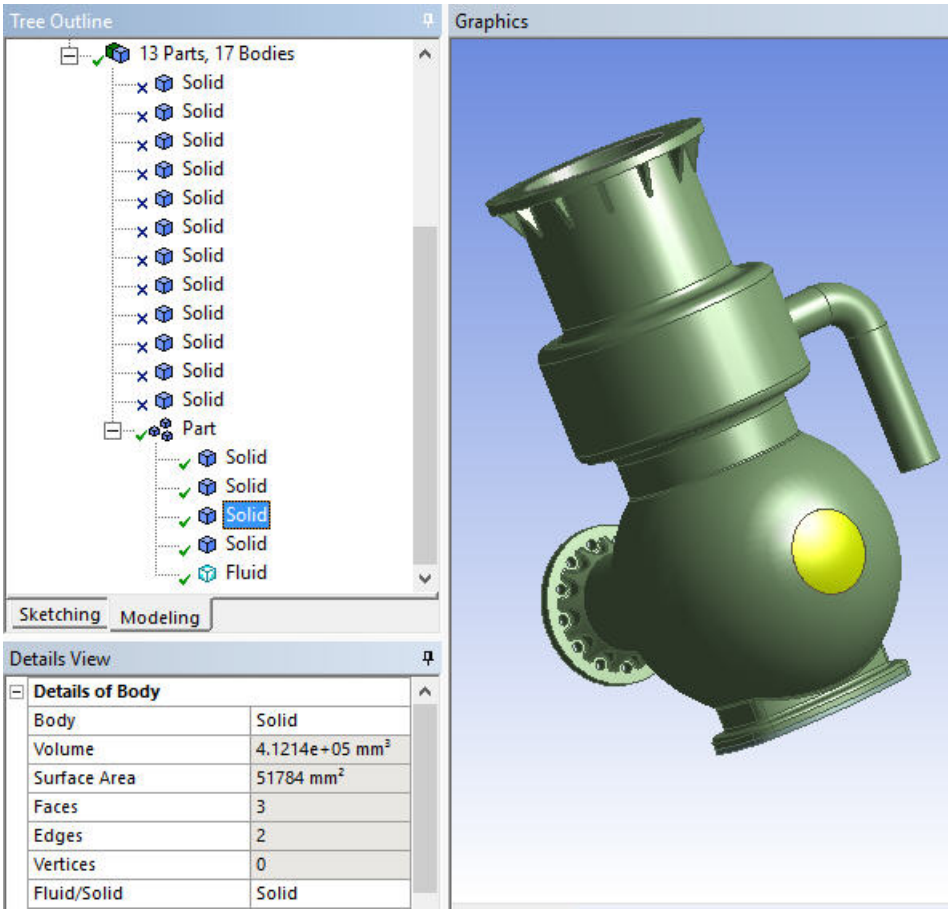
The second example also illustrates a workflow involving the DesignModeler application, the Meshing application, and Ansys Fluent. However, in this example the **Fluid/Solid** material property of a body is changed while the model is being edited in the Meshing application.

First, the model is imported into the DesignModeler application. The model consists of 13 parts and 17 bodies, but 12 of the bodies are not needed for this example and are suppressed. This leaves a multibody part consisting of five bodies. The **Fluid/Solid** material property is set to **Solid** for four of the bodies and to **Fluid** for the remaining body. Notice that one body is selected in the Tree Outline and is highlighted in the **Geometry** window. Refer to Figure 9: Multibody Part Containing Mix of Solid and Fluid Bodies in the DesignModeler Application (p. 60).

See Fluent Mesh Export (p. 43) for more information about the **Fluid/Solid** material property in the DesignModeler application.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

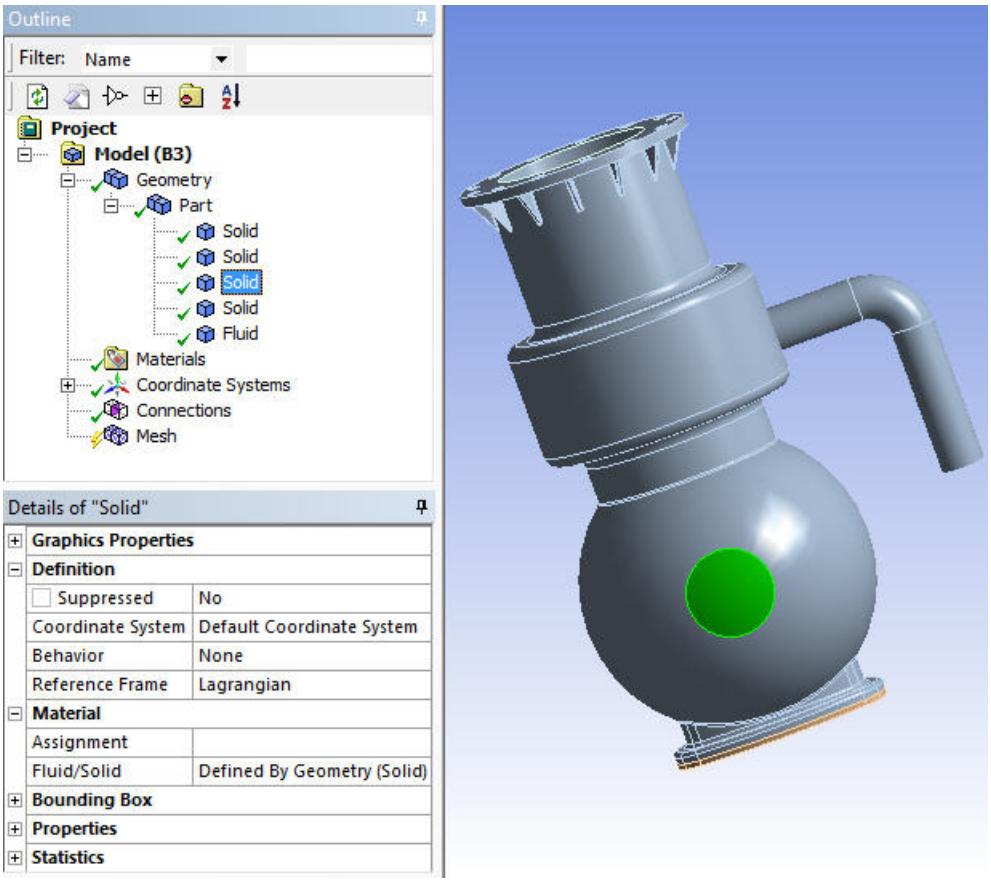
Figure 9: Multibody Part Containing Mix of Solid and Fluid Bodies in the DesignModeler Application



Next, the model is opened in the Meshing application for editing. Notice that the same body has been selected in the Tree Outline and is highlighted in the **Geometry** window. Also notice that the **Fluid/Solid** setting for the body is set to **Defined By Geometry (Solid)** in the Details View. When set to **Defined By Geometry**, the value is based on the **Fluid/Solid** material property that was assigned to the body in the DesignModeler application. Refer to Figure 10: Multibody Part Being Edited in the Meshing Application (p. 61).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

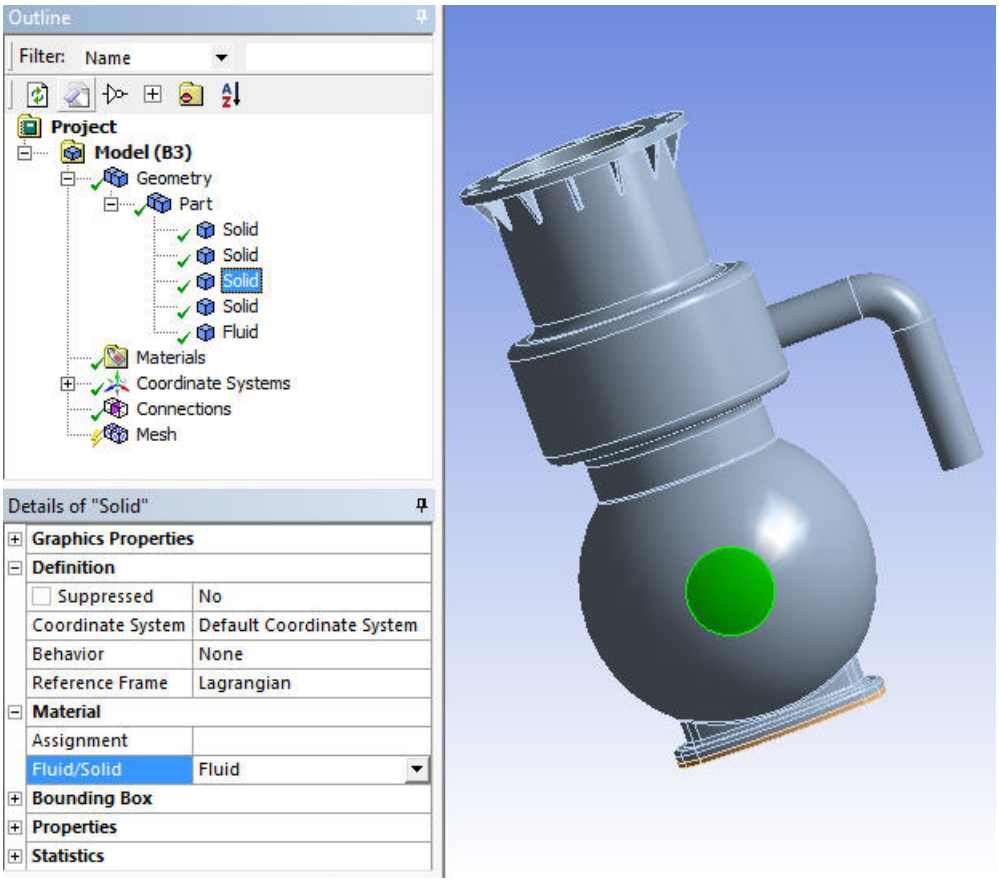
Figure 10: Multibody Part Being Edited in the Meshing Application



Next, the **Fluid/Solid** material property of the highlighted body is changed to **Fluid**. For all other bodies, the **Fluid/Solid** material property that was assigned in the DesignModeler application will be retained for this example. Refer to Figure 11: Changing the Fluid/Solid Material Property of a Body (p. 62).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

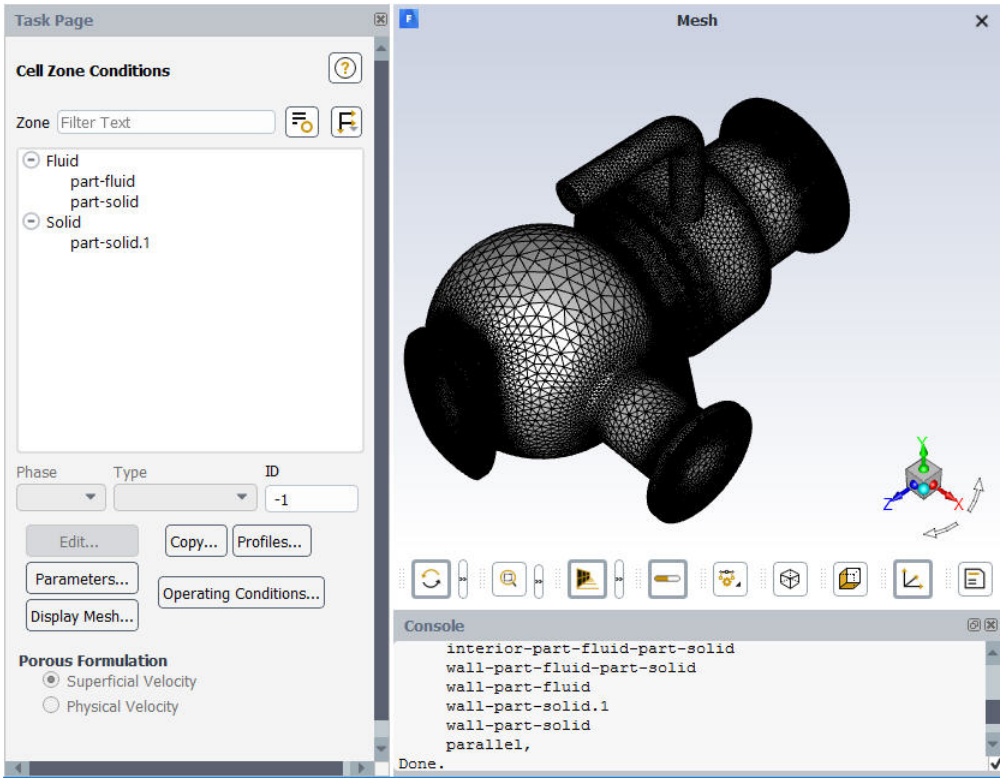
Figure 11: Changing the Fluid/Solid Material Property of a Body



After meshing, the mesh is exported to Ansys Fluent format and read into Ansys Fluent. As shown in Figure 12: Continuum Zone Names and Types Transferred to Ansys Fluent (p. 63), the continuum (or cell) zone names and types are transferred as expected. Notice that the zone named "part-solid," which is highlighted in the panel on the left, has a zone type of *FLUID*.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12: Continuum Zone Names and Types Transferred to Ansys Fluent



Polyflow Classic Export

When you export a mesh file to Polyflow Classic format (**File> Export** from the Meshing application main menu, then **Save as type Polyflow Classic Input Files**), a mesh file with the extension .poly is created. The exported mesh file is suitable for import into Ansys Polyflow Classic and supports the following features:

- Named Selections - Named Selections that are present in the Meshing application will appear in the exported mesh file.
- PMeshes - You can create Named Selections to specify specialized modeling conditions on edges for 2-D or shell geometry; and edges and faces for 3-D geometry. The exported mesh file will contain the mesh nodes and elements associated with those Named Selections in PMesh format. For more information, refer to Generating Meshes in Ansys Meshing for Polyflow Classic.
- Element types - Those that are supported in the exported mesh file are listed in the table below. Only linear meshes are supported for Polyflow Classic export.

Dimension	Supported Element Type
3D	8–node hexahedral
	4–node tetrahedral
	5–node pyramid
	6–node wedge
2D	3–node triangle

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dimension	Supported Element Type
	4–node quadrilateral

Note:

- As an alternative to the export process described here, you can transfer a mesh from a Mesh system into a downstream Polyflow Classic system.
- If you change your mesh settings after generating the mesh and do not perform an **Update** (p. 527), the mesh that is exported is the currently existing mesh (that is, a mesh that does not reflect your new settings). In such cases, a warning message is issued that indicates that the mesh you are exporting to Polyflow Classic format has not been updated. If you want the latest settings to affect the mesh, perform an **Update** and export the mesh again.

CGNS Export

When you export a mesh file to CGNS format (**File> Export** from the Meshing application main menu, then **Save as type CGNS Input Files**), a CGNS mesh file with the extension .cgns is created. The exported mesh file is suitable for import into a CGNS-compatible application. For more control over the input file, refer to Meshing Options on the Options Dialog Box (p. 389).

Named Selections are supported in the CGNS file.

Element types that are supported in the exported CGNS mesh are listed in the table below. Only linear meshes are supported for CGNS export.

Dimension	Supported Element Type
3D	8–node hexahedral
	4–node tetrahedral
	5–node pyramid
	6–node wedge
2D	3–node triangle
	4–node quadrilateral

The maximum number of elements in a part multiplied by the number of nodes per element cannot exceed 2 billion. For linear HEXA meshes, the maximum number of elements in a part (or body) should not exceed 250 million elements. For linear TETRA meshes, the maximum number of elements in a part (or body) should not exceed 500 million elements.

To export larger meshes, split them into several smaller parts (or Named Selections).

If you exceed the CGNS export limits, you will get the error message:

```
Error: CGNS section CGNS_section_name exceeds maximum mesh size limit, please split the mesh into smaller sectio
```


Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys ICEM CFD Export

When you export from the Meshing application to Ansys ICEM CFD format, an Ansys ICEM CFD project file with the extension .prj, along with a geometry file (*.tin) and/or mesh file (*.uns) are written. The files that are created are the same as those that are created if you begin in Ansys ICEM CFD and use its Import Model option to import a file from the Meshing application.

Anytime you plan to export from the Meshing application to Ansys ICEM CFD format, it is best practice to define the desired part and body names for your model in the DesignModeler or Discovery Modeling application prior to meshing the model in the Meshing application. This is recommended because the Ansys ICEM CFD part names will be derived from the part and body names that are defined for the model when you *initially* open the model in the Meshing application. The export process will ignore any renaming or Named Selections created in the Meshing application.

There are three ways to export meshing data to ICEM CFD format:

1. Choose **File> Export** from the Meshing application main menu, then **Save as type ICEM CFD Input Files.**

   Ansys ICEM CFD part names that appear in the exported files are derived from the Ansys Workbench geometry part and body names. In the case of a single body part, only the body name is used.

   Note:

   The concept of a *part* in Ansys Workbench and a *part* in Ansys ICEM CFD is not the same. For information about parts in Ansys Workbench, refer to Conformal and Non-Conformal Meshing (p. 21) in the Meshing application help and Geometry Introduction in the Mechanical help. For information about parts in Ansys ICEM CFD, refer to the documentation available under the Help menu within Ansys ICEM CFD.

2. Save your Ansys Workbench files (*.mechdat or *.meshdat) and use the Ansys ICEM CFD **File > Import Model** option to import the files into Ansys ICEM CFD (as long as Ansys Workbench and Ansys ICEM CFD are installed on the same machine). Legacy formats such as *.dsdb and *.cmdb are also supported.

   Any defined Named Selections will be imported into Ansys ICEM CFD as subsets because subsets support non-exclusive sets (overlapping Named Selections). However, each entity can only be in a single part (exclusive sets). In cases where you want overlapping Named Selections to be converted to Ansys ICEM CFD parts, the overlapping subsets can be cleaned up in Ansys ICEM CFD and then converted into parts. For details about handling imported Ansys Workbench files in Ansys ICEM CFD, refer to the documentation available under the Help menu within Ansys ICEM CFD.

3. Use the Ansys ICEM CFD Workbench Add-In. In Ansys Workbench, right-click a **Geometry** or **Mesh** cell and choose **Transfer Data Into New > ICEM CFD**. The advantage of the Add-In connection is that it maintains the connectivity once the geometry is modified. So, unlike the previous two methods, you can easily refresh the geometry in ICEM CFD and then update the mesh and the solver input. For more information, see Component Systems in the [Ansys Workbench User's Guide].

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rules Followed By the Export Process

When exporting to Ansys ICEM CFD format, these rules are followed:

Note:

The series of examples that follows this list illustrates many of the rules listed here.

- To achieve unique Ansys ICEM CFD part names in the Ansys ICEM CFD format files, a unique integer is suffixed to all Ansys Workbench part/body names.
- A single body part in Ansys Workbench will appear as <*part_name*>_<*part_index*> in the Ansys ICEM CFD format files.
- A multibody part in Ansys Workbench will appear as <*part_name*>_<*part_index*>/<*body_name*>_<*body_index*> in the Ansys ICEM CFD format files. The / character denotes hierarchy.
- Bodies that are in a multibody part in Ansys Workbench are put into an Ansys ICEM CFD assembly. The structuring in the Ansys ICEM CFD format files reflects the part/body structure present in Ansys Workbench.
- As long as they are *not* contained in Named Selections, faces that are shared between bodies in the same multibody part in Ansys Workbench are put into separate Ansys ICEM CFD parts. This type of shared face is named according to the bodies having the face in common, with the body names separated by the # character.
- Entities that are contained in a Named Selection are put into a separate Ansys ICEM CFD part.
- For each body, an Ansys ICEM CFD Material Point is created and put into the corresponding Ansys ICEM CFD part. The names of Material Points have the suffix _MATPOINT.
- If a mesh has been generated, it is exported along with the geometry. In such cases, these additional rules are followed:
  - As long as they are *not* contained in a Named Selection, node/line/surface mesh cells are associated with the corresponding geometry part/body in Ansys ICEM CFD.
  - As long as they are *not* contained in a Named Selection, volume mesh cells are associated with the Material Point part.
  - Mesh cells that are associated with geometry entities that are contained in a Named Selection are associated with the Ansys ICEM CFD part that corresponds to that Named Selection.

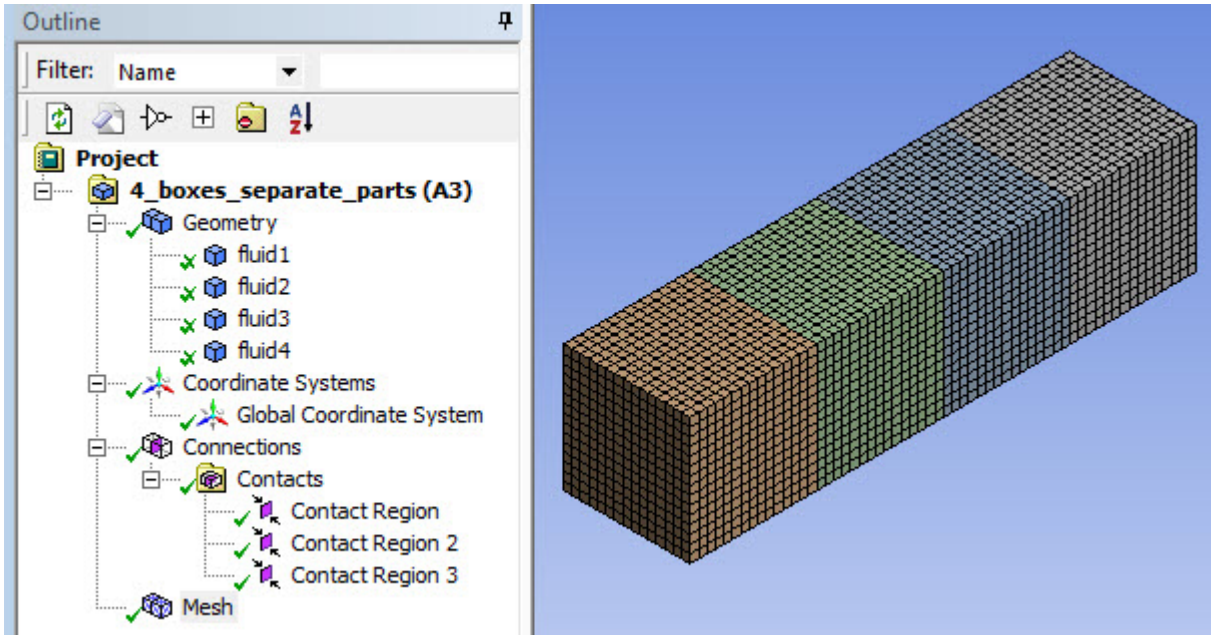
The first example is a model consisting of four separate single body parts in Ansys Workbench. The single body parts are named fluid1, fluid2, fluid3, and fluid4. The table below shows the geometry in Ansys Workbench and the corresponding part names that will appear in Ansys ICEM CFD:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This geometry in Ansys Workbench...	Results in these part names in Ansys ICEM CFD...
A model consisting of four separate single body parts named:	
fluid1	FLUID1_1
fluid2	FLUID2_2
fluid3	FLUID3_3
fluid4	FLUID4_4

The figure below shows the model after it was meshed in the Meshing application:

Figure 13: Meshed Model (Four Separate Workbench Parts) Ready for Export to Ansys ICEM CFD

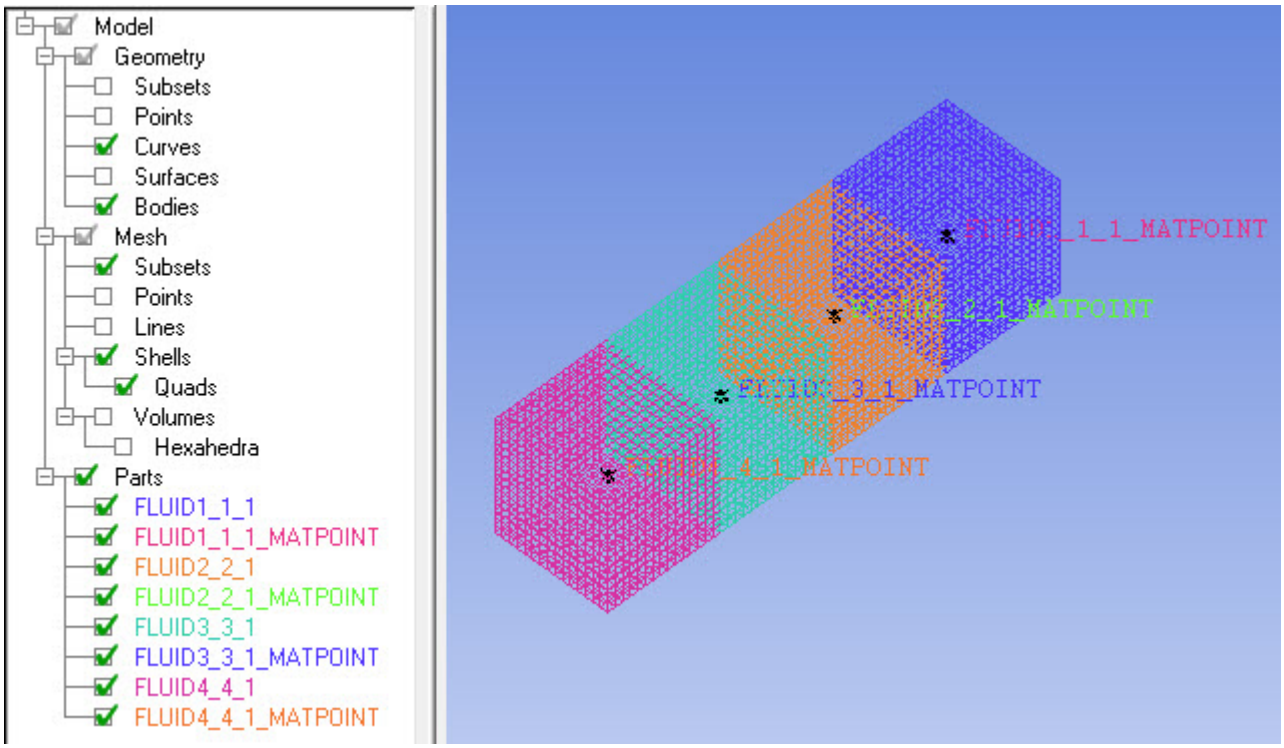


Next, the model was exported from the Meshing application to Ansys ICEM CFD format. In the figure below, the corresponding .prj file has been opened in Ansys ICEM CFD. Notice the names that are assigned to the various entities in the Ansys ICEM CFD format file:

- Each body/part name has been suffixed with a unique integer to distinguish it from similarly named bodies/parts. (Note that in this example, *part_name* is equal to *body_name*.)
- Each single body part in Ansys Workbench appears as <*part_name*>_<*part_index*> in the Ansys ICEM CFD format files. For example, the part named fluid1 in Ansys Workbench has a part name of FLUID1_1 in Ansys ICEM CFD, which appears as FLUID1_1_1 in the Ansys ICEM CFD format files after the *part_index* is added.
- For each body in the Ansys Workbench file (fluid1, fluid2, fluid3, fluid4), a Material Point has been assigned (FLUID1_1_1_MATPOINT, FLUID2_2_1_MATPOINT, FLUID3_3_1_MATPOINT, FLUID4_4_1_MATPOINT).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14: Opening the .prj File (Four Separate Workbench Parts) in Ansys ICEM CFD

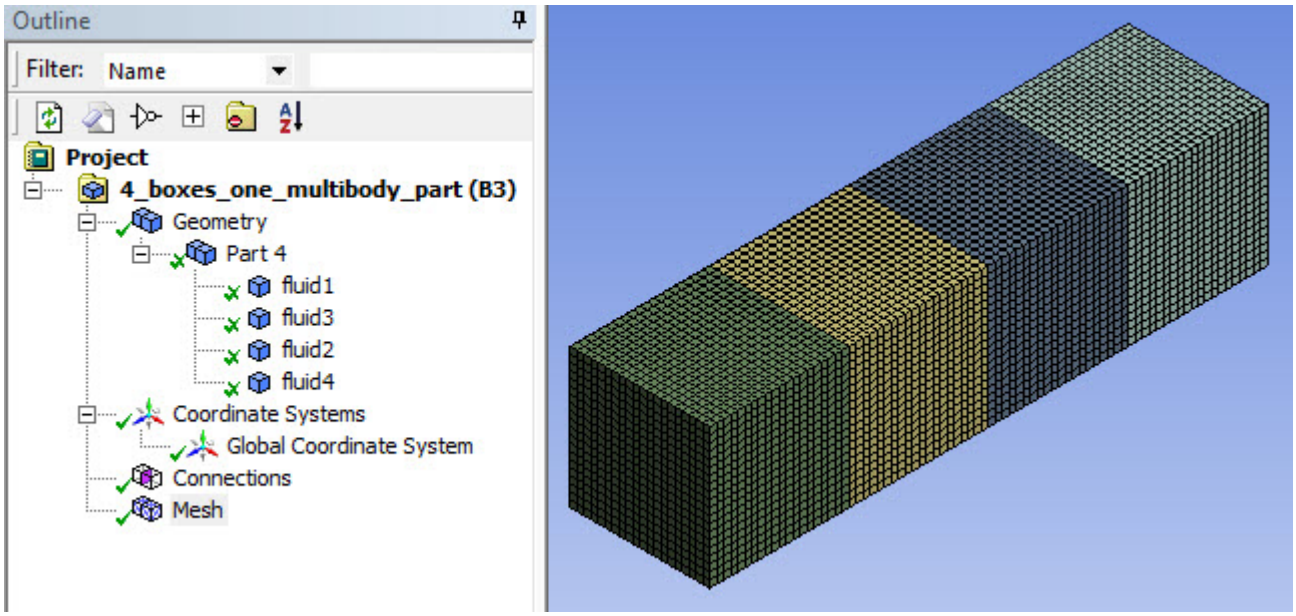


The second example is a model consisting of one multibody part in Ansys Workbench. The multibody part, which is named Part 4, contains four bodies named fluid1, fluid2, fluid3, and fluid4. The table below shows the geometry in Ansys Workbench and the corresponding part names that will appear in Ansys ICEM CFD:

This geometry in Ansys Workbench...	**Results in these part names in Ansys ICEM CFD (the / character denotes hierarchy)...**
A model consisting of one multibody part named Part 4, containing four bodies named:	
fluid1	PART_4_1/FLUID1_3
fluid2	PART_4_1/FLUID2_2
fluid3	PART_4_1/FLUID3_1
fluid4	PART_4_1/FLUID4_4

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15: Meshed Model (One Multibody Workbench Part) Ready for Export to Ansys ICEM CFD

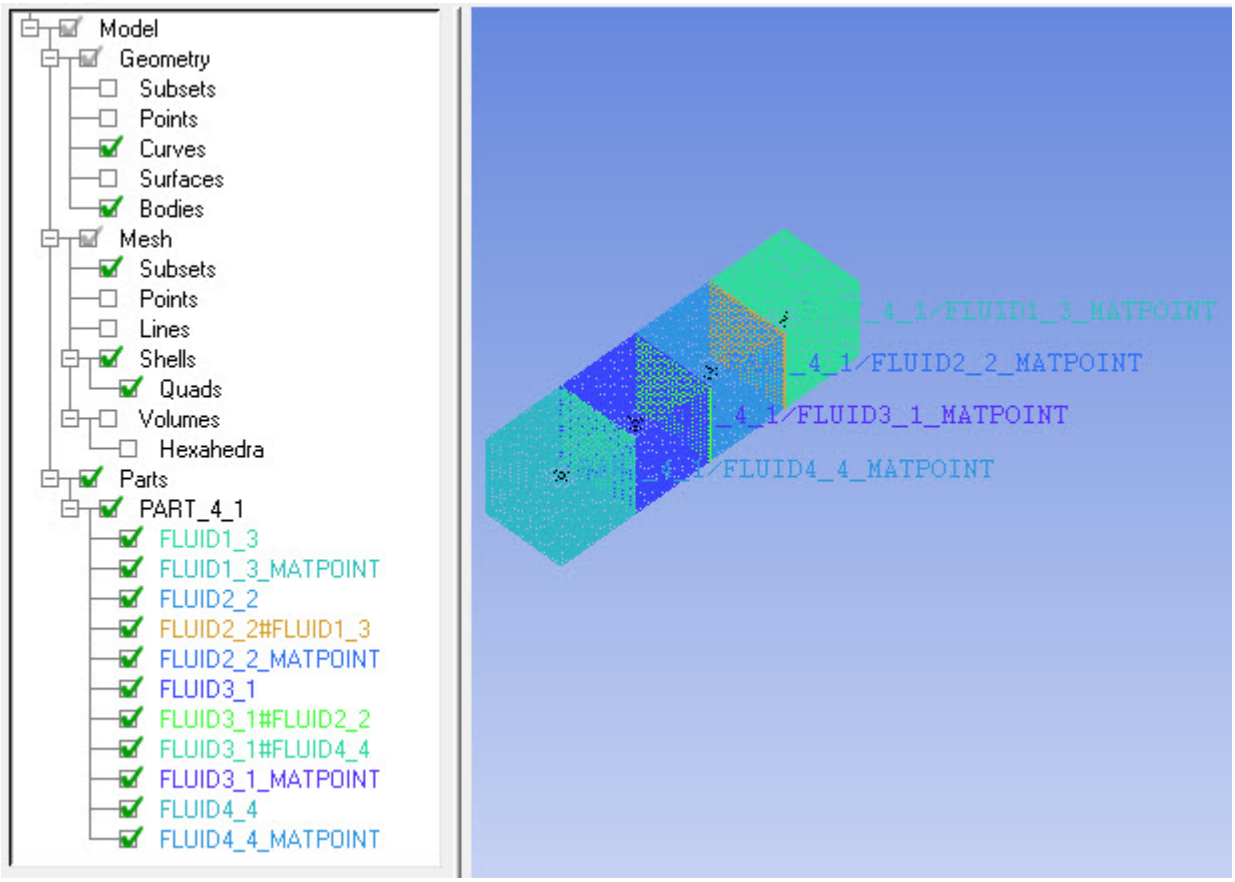


Next, the model was exported from the Meshing application to Ansys ICEM CFD format. In the figure below, the corresponding .prj file has been opened in Ansys ICEM CFD. Notice the names that are assigned to the various entities in the Ansys ICEM CFD format file:

- Each body/part name has been suffixed with a unique integer to distinguish it from similarly named bodies/parts.
- Each multibody part in Ansys Workbench appears as <*part_name*>_<*part_index*>/<*body_name*>_<*body_index*> in the Ansys ICEM CFD format files. For example, the fluid1 body in Part 4 in Ansys Workbench has a part name of PART_4_1/FLUID1_3 in the Ansys ICEM CFD format files.
- The bodies that are in the multibody part in the Ansys Workbench file (fluid1, fluid2, fluid3, and fluid4) have been put into an Ansys ICEM CFD assembly named Part_4.
- The faces that are shared between the various pairs of bodies have been named FLUID2_2#FLUID1_3, FLUID3_1#FLUID2_2, and FLUID3_1#FLUID4_4.
- For each body in the Ansys Workbench file (fluid1, fluid2, fluid3, fluid4), a Material Point has been assigned (FLUID1_3_MATPOINT, FLUID2_2_MATPOINT, FLUID3_1_MATPOINT, FLUID4_4_MATPOINT).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16: Opening the .prj File (One Multibody Workbench Part) in Ansys ICEM CFD

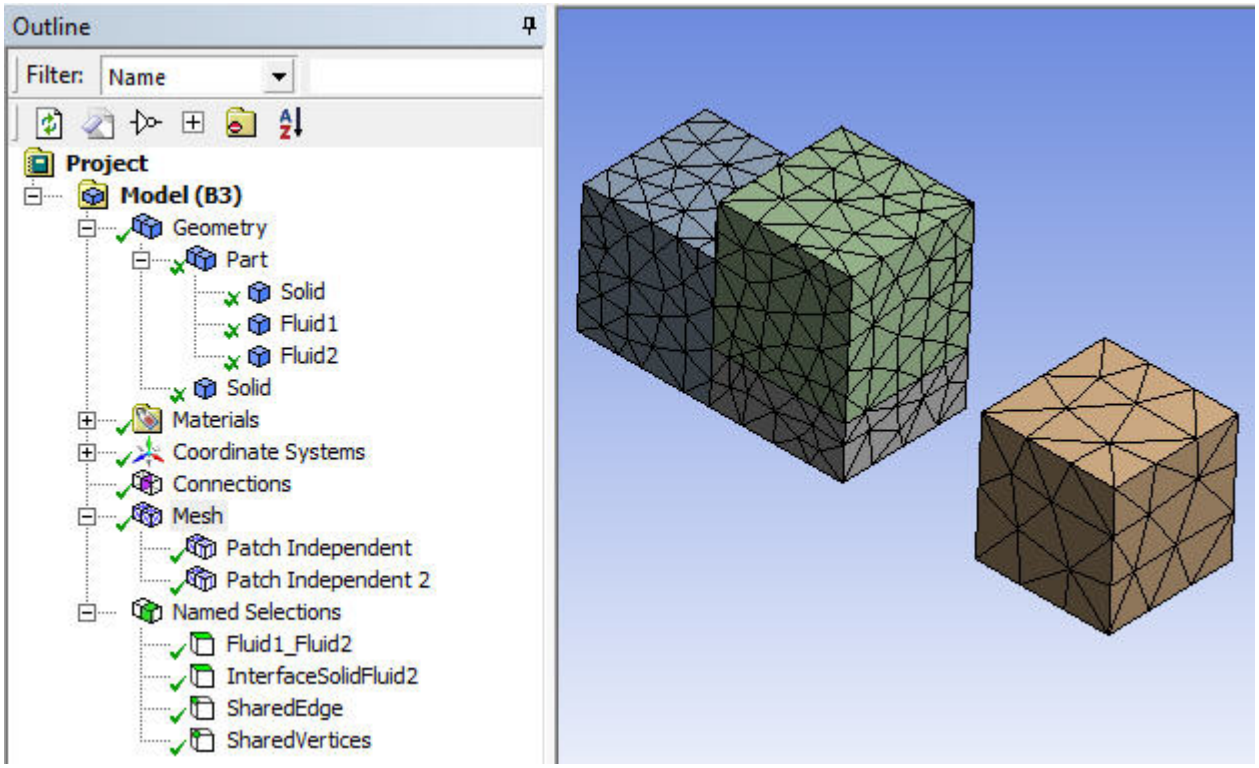


The third (and final) example involves a model for which four Named Selections are defined in the DesignModeler application. The model is meshed in the Meshing application, exported to Ansys ICEM CFD format, and opened in Ansys ICEM CFD.

The first figure shows the model after it was meshed in the Meshing application.

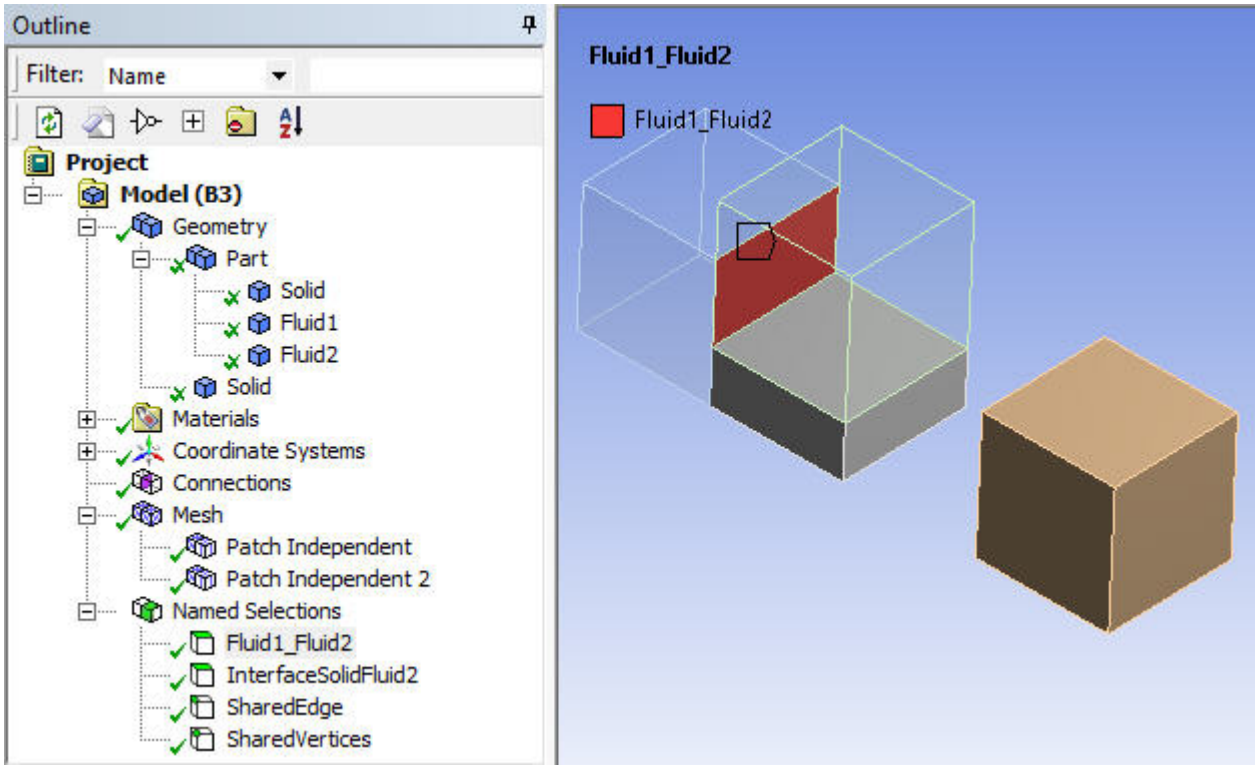
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 17: Meshed Model (with Named Selections) Ready for Export to Ansys ICEM CFD



The next four figures show the entit(ies) in the model that are contained in each of the four Named Selections. In the figure below, the Fluid1_Fluid2 Named Selection is highlighted.

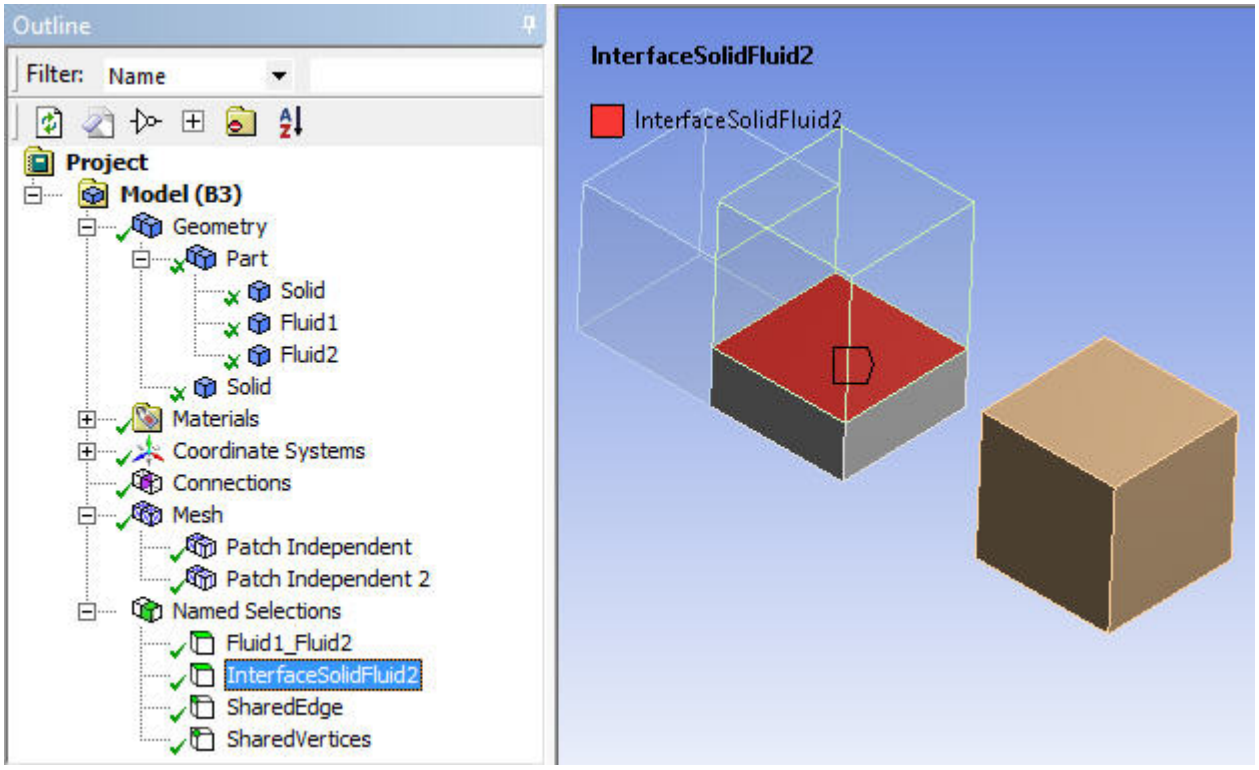
Figure 18: Fluid1_Fluid2 Named Selection



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

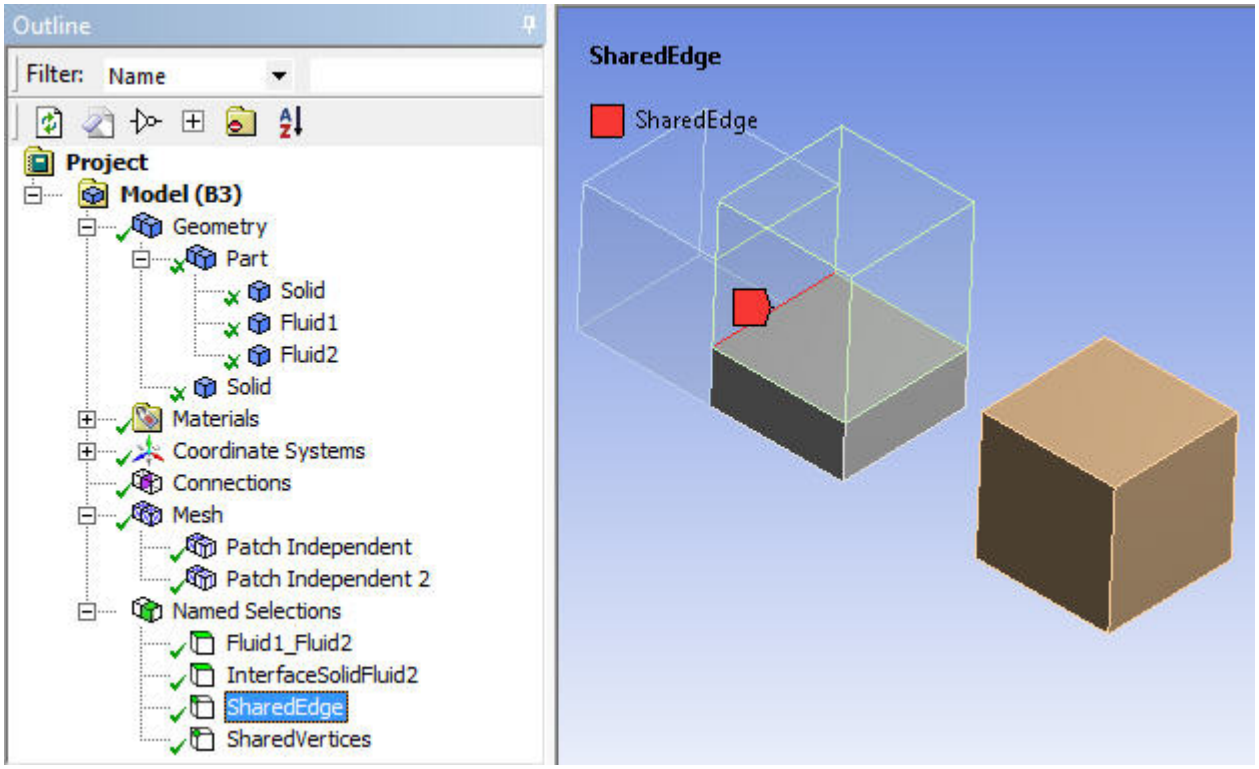
In the figure below, the InterfaceSolidFluid2 Named Selection is highlighted.

Figure 19: InterfaceSolidFluid2 Named Selection



In the figure below, the SharedEdge Named Selection is highlighted.

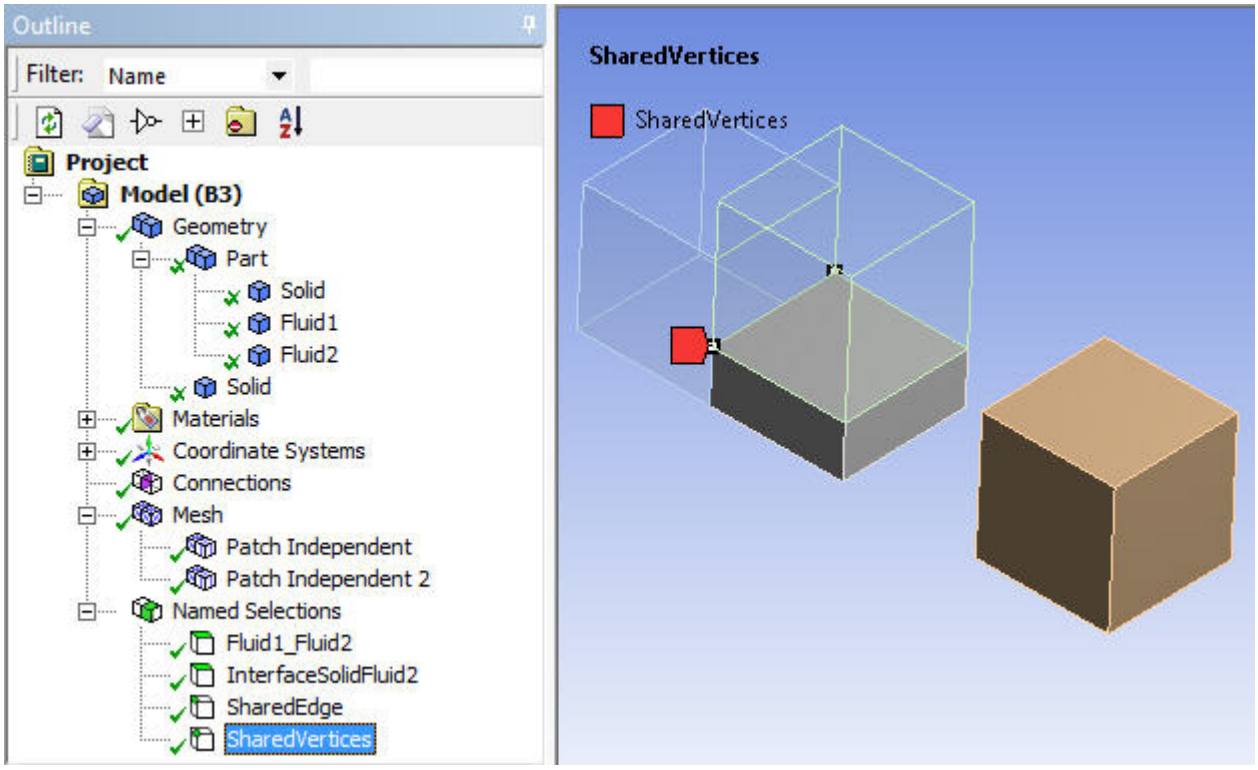
Figure 20: SharedEdge Named Selection



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the figure below, the SharedVertices Named Selection is highlighted.

Figure 21: SharedVertices Named Selection

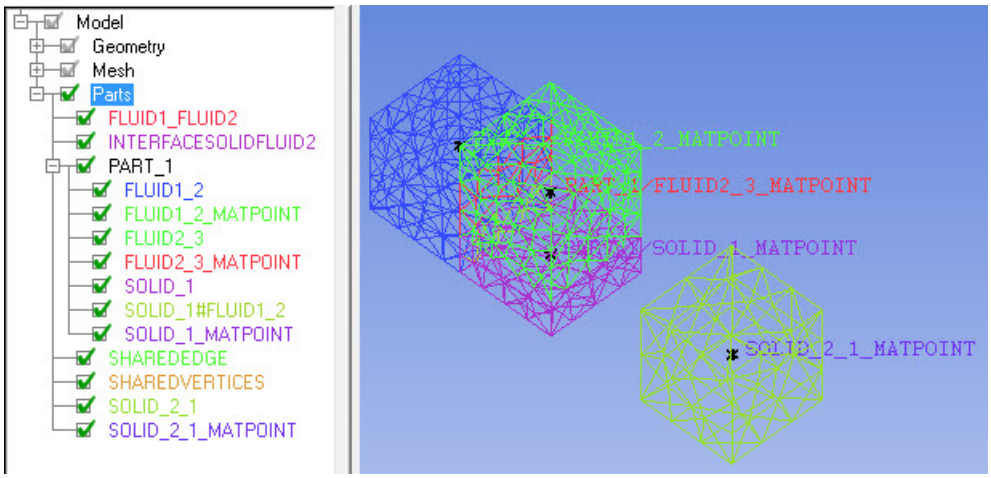


Next, the model was exported from the Meshing application to Ansys ICEM CFD format. In the figure below, the corresponding .prj file has been opened in Ansys ICEM CFD. Notice the names that are assigned to the various entities in the Ansys ICEM CFD format file:

- Each body/part name has been suffixed with a unique integer to distinguish it from similarly named bodies/parts.
- The bodies that are in the multibody part in the Ansys Workbench file (Solid, Fluid1, and Fluid2) have been put into an Ansys ICEM CFD assembly named Part_1.
- The face that is shared between SOLID_1 and FLUID1_2 has been named SOLID_1#FLUID1_2.
- Because Fluid1_Fluid2, InterfaceSolidFluid2, SharedEdge, and SharedVertices are all Named Selections in the Ansys Workbench file, each of them has been put into a separate Ansys ICEM CFD part.
- For each body in the Ansys Workbench file (Solid, Fluid1, Fluid2, Solid), a Material Point has been assigned (SOLID_1_MATPOINT, FLUID1_2_MATPOINT, FLUID2_3_MATPOINT, and SOLID_2_1_MATPOINT).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 22: Opening the .prj File (with Named Selections) in Ansys ICEM CFD



Note:

For additional information, refer to the documentation available under the Help menu within Ansys ICEM CFD.

Exporting Faceted Geometry to Ansys Fluent Meshing

You can use the Meshing application to export faceted geometry for use in Ansys Fluent Meshing (formerly TGrid):

1. Select **File> Export** from the main menu.
2. In the **Save As** dialog box, choose a directory and specify a file name for the file. Then choose **TGrid Faceted Geometry File** from the **Save as type** drop-down menu and click **Save**.

As a result, a file with the extension .tgf is created. The exported file can be imported into Ansys Fluent Meshing, where you can use such features as the Ansys Fluent Meshing wrapper utility.

Note:

The .tgf file has the same format as a .msh file and will be recognized as a "Mesh File" when read into Ansys Fluent Meshing (File/Read/Mesh... menu item).

Upon export, Ansys Fluent Meshing objects and zones are created according to geometry bodies and Named Selections. Part and body names are used in the Ansys Fluent Meshing object/zone names to identify the parts and bodies from which they originated.

Remember the following information when exporting to Ansys Fluent Meshing:

- The quality of the exported facets is derived from the CAD system. You can use the **Facet Quality** option (**Tools > Options > DesignModeler > Graphics > Facet Quality**) to control the quality of the exported facets (the higher the setting, the higher the quality). However, be aware that higher settings create large numbers of facets, which can slow down the Meshing application.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The part, body, and Named Selection names that were present in the Meshing application are exported in all lowercase characters for use in the corresponding Ansys Fluent Meshing zone names.
- Only part and body names that were imported into the Meshing application are used in the exported zone name. For example, names that were initially defined in the Ansys DesignModeler application or initially appeared in the Tree Outline when a CAD file was loaded directly into the Meshing application will be used. Any subsequent renaming of parts and bodies that occurs in the Meshing application will not be taken into account in the exported zone names.
- Vertices (regardless of whether they are contained in a Named Selection) are ignored.
- The name of each Named Selection is filtered upon export such that only allowable characters remain in the name of the Ansys Fluent Meshing zone. Allowable characters include all alpha-numeric characters as well as the following special characters:

  _ + - : .

  All other characters, including spaces, are invalid. If an invalid character is used, it is replaced by a hyphen (-) upon export.
- When the same entity is a member of more than one Named Selection, those Named Selections are said to be "overlapping." If you are exporting faceted geometry into the Ansys Fluent Meshing format (or a mesh into the Ansys Polyflow Classic, CGNS, or Ansys ICEM CFD format), and overlapping Named Selections are detected, the export will fail and you must resolve the overlapping Named Selections before proceeding. For details, see Repairing Geometry in Overlapping Named Selections (p. 79).

The figures below illustrate the process of exporting geometry in faceted representation from the Meshing application to Ansys Fluent Meshing. Figure 23: Part, Body, and Named Selection Names in the Meshing Application (p. 76) shows the model in the Meshing application. The geometry consists of a multibody part named AeroValve, and the three bodies that AeroValve contains are named Outletbody, Valve, and Inletbody. Notice that three Named Selections have been defined and are highlighted in the **Geometry** window: Inlet, Outlet, and Valve_opening.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23: Part, Body, and Named Selection Names in the Meshing Application

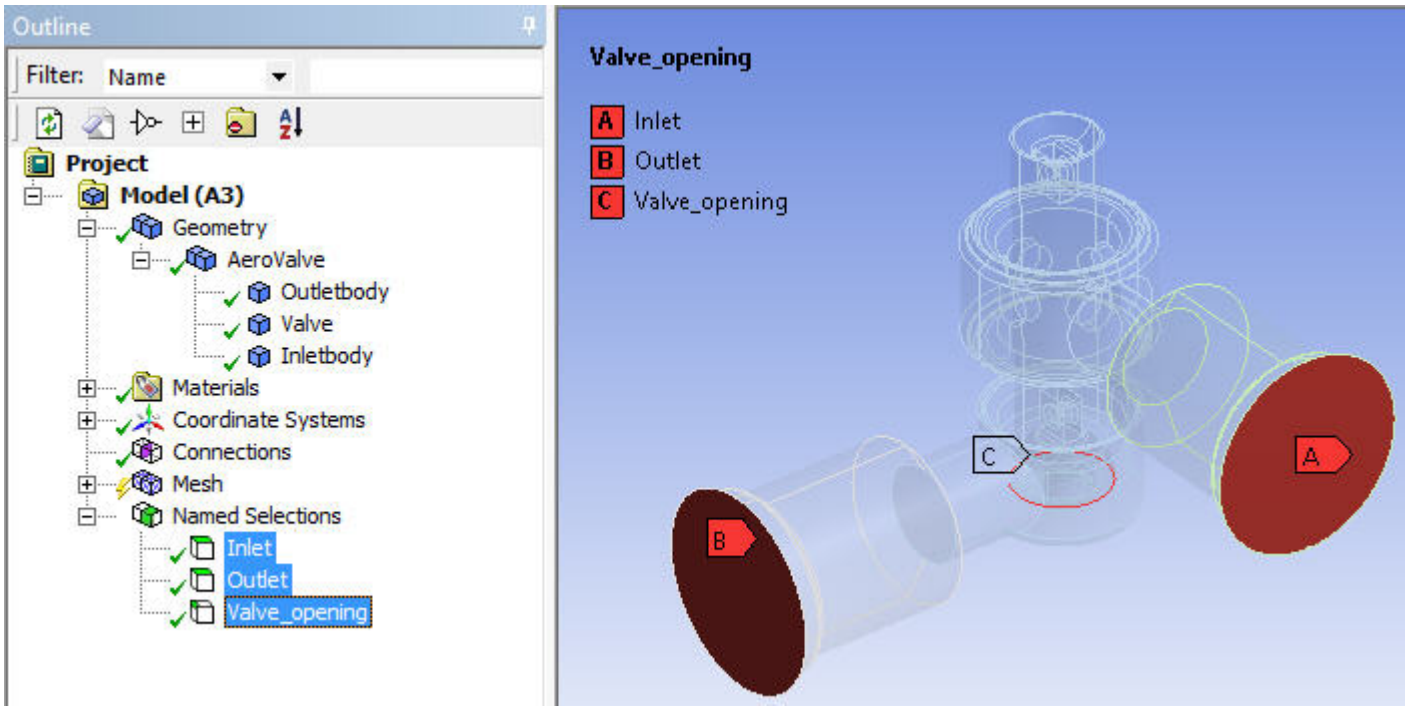
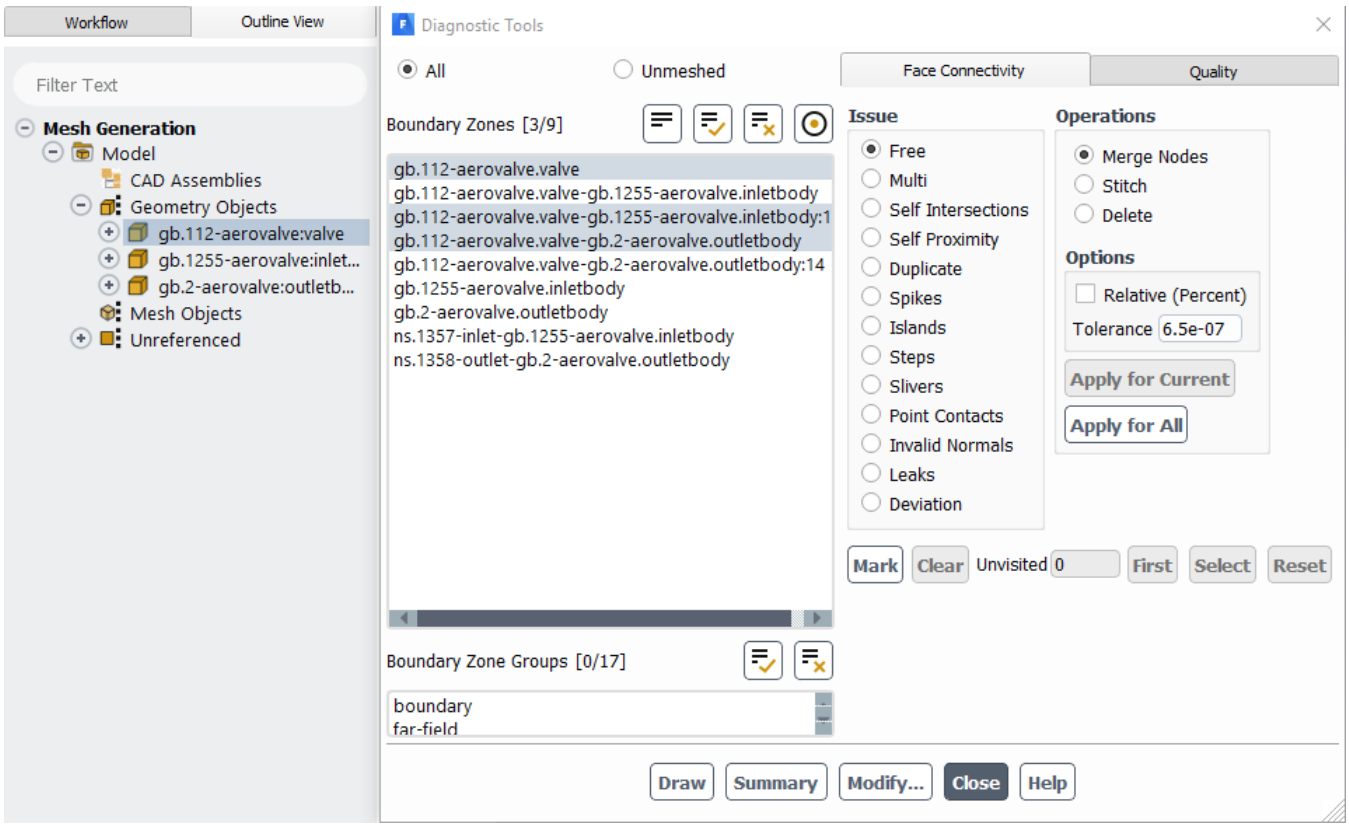


Figure 24: Objects/Zone Names Transferred to Ansys Fluent Meshing (p. 76) shows the Surface Retriangulation panel after the exported .tgf file is imported into Ansys Fluent Meshing.

Figure 24: Objects/Zone Names Transferred to Ansys Fluent Meshing



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Named Selections and Regions for Ansys CFX

There are a number of places in the Ansys Workbench and CFX applications where geometric faces and regions of mesh can be grouped together and assigned names. While this provides a large amount of flexibility, it can lead to confusion. To have more control over which groups of names are loaded into CFX to help simplify this issue, this section describes the 'best practices' for region definitions and is targeted mainly at Ansys CFD users of the DesignModeler, Meshing, and CFX applications.

Note:

Refer to Passing Named Selections to the Solver (p. 78) for related information.

Defining Names in the DesignModeler Application or the Meshing Application

When creating or importing geometry in the DesignModeler application or editing a mesh in the Meshing application, Named Selections can be defined in terms of one or more CAD faces. If it is desirable for these Named Selections to be available in CFX-Pre, then they must adhere to these simple rules: Named Selections should not overlap or be defined as multi-dimensional and all Named Selections must have unique names.

Importing DesignModeler Named Selections into the Meshing Application

If the Named Selections defined in the DesignModeler application are required in the Meshing application, you must set the appropriate geometry import options to ensure the Named Selections will be transferred properly:

1. From the Ansys Workbench Menu Bar, select **Tools> Options**.
2. In the left pane of the **Options** dialog box, select **Geometry Import**.
3. In the right pane of the **Options** dialog box, select the **Named Selections** check box and either clear the **Filtering Prefixes** field (to import all names) or set it to the appropriate filter if required.

The next time that you attach geometry containing Named Selections and launch the Meshing application, the application will start, load the geometry, and load any Named Selections previously defined for that geometry. The preferences you set on the **Options** dialog box are local settings and affect only you.

If a mesh is generated, the Named Selections defined in the DesignModeler application will be available when the mesh is imported into CFX-Pre.

Note:

For detailed descriptions of the geometry import options, see the CAD Integration section in the ANSYS Workbench help.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using Multiple Mesh Methods

Named Selections can be defined in the Meshing application, the DesignModeler application, or supported CAD systems. When the resulting mesh is loaded into CFX-Pre, all of these Named Selections are available. Control over which names are imported with the mesh is available in the Mesh Import options.

It is possible to define Named Selections that conflict with some virtual topology, or do not conform to the CFX Topology Model in some other way. Such Named Selections will not be imported into CFX-Pre.

If a Named Selection is created that contains characters not supported in CFX-Pre names, these names will be modified on import to remove the illegal characters. Note that non-ASCII characters are illegal in CFX-Pre names.

Invalid Named Selections

The use of Virtual Topology within the Meshing application can invalidate Named Selection definitions. Consider the case where a Named Selection 'Region1' is defined in terms of 4 CAD faces (1, 2, 3 and 4). If this geometry is then loaded into the Meshing application and Virtual Topology is used to merge 2 of these faces (3 and 4), the original Named Selection can no longer be resolved. In CFX-Pre, the Named Selection will be modified so that it only refers to the 'resolvable' faces. Hence, in CFX-Pre, the Named Selection 'Region1' would be defined in terms of faces 1 and 2 only.

Named Selections Involving Interior Faces

If an interior face (for example, a face that is shared between two bodies in the same part in DesignModeler) is included in a Named Selection in the Meshing application, then the corresponding location in CFX-Pre will be a Composite region that includes two Primitive regions: one for each side of the interior face. Care should be taken when using the resulting Composite region for evaluating expressions and performing calculations: for example, areas evaluated on such a Composite region would be likely to give double the expected result (since the area is evaluated over both of the primitive regions that form the two sides of the interior face) and mass flows will be approximately zero (since the mass flow is evaluated on both of the primitive regions and will have the opposite sign on each). You can optionally edit the Composite region in CFX-Pre to include only one of the primitive regions.

Passing Named Selections to the Solver

When defining Named Selections, you can use the Send to Solver option to control whether a Named Selection is passed to the solver. **Send to Solver** is available in both the Mechanical and Meshing applications.

The default is **Yes** for Named Selections that you create, and **No** for Named Selections that are generated automatically by the **Mesh** worksheet (p. 448). When set to **No**, the selected Named Selection is not passed to Ansys Fluent, Ansys CFX, Polyflow Classic, CGNS, Ansys ICEM CFD, Ansys Fluent Meshing, Mechanical APDL, Ansys Autodyn, or any other downstream system that may be connected to the Mechanical or Meshing application.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Repairing Geometry in Overlapping Named Selections

When you export (p. 42) a mesh into the Ansys Fluent, Polyflow Classic, CGNS, or Ansys ICEM CFD format, or as a faceted geometry (p. 74) for use in Ansys Fluent Meshing, each Named Selection must be unique. That is, the entities (bodies, faces, or edges) in one Named Selection cannot also exist in a second Named Selection. Any entity that is in two or more Named Selections is considered overlapping. The overlapping entities need to be resolved before the mesh can be exported.

Note:

Any Named Selection whose **Send to Solver** (p. 78) option is set to **No** is skipped during the processing described above. For this reason, an easy way to avoid overlapping Named Selections is to set all values of **Send to Solver** to **No**.

If a mesh fails to export and an error about overlapping Named Selections occurs, you can display the overlapping entities by right-clicking on the **Mesh** object in the Tree Outline and choosing **Show**>**Geometry in Overlapping Named Selections**. You can then inspect the Named Selections to remove the duplicate (overlapping) entities and proceed with exporting the mesh.

Alternatively, you can repair the overlapping Named Selections automatically by right-clicking on the **Named Selections** folder and choosing **Repair Overlapping Named Selections**. This option checks all Named Selections (for which **Send to Solver** is set to **Yes**) in the order in which they appear in the Tree (top to bottom). When it encounters an overlapping Named Selection, it moves that Named Selection to a new folder called **Overlapping Named Selections** and sets its **Send to Solver** option to **No**. You can delete the **Overlapping Named Selections** folder.

The repaired Named Selections are placed at the bottom of the Tree and their names are prefixed with **Repaired**. To repair a Named Selection, the Meshing application removes any duplicate (overlapping) entities from the Named Selection and sets its **Send to Solver** option to **Yes**.

Note:

- In cases where you want overlapping Named Selections to be converted to Ansys ICEM CFD parts, the overlapping subsets can be cleaned in Ansys ICEM CFD and then converted into parts.
- If you are using Named Selections for mesh settings (such as sizing), set the **Send to Solver** (p. 78) option for these Named Selections to **No** so that they are skipped when the check for overlapping Named Selections is performed. Otherwise, the check may cause the mesh state to go out-of-date.

Resolving Overlapping Contact Regions

Contact regions are said to be "overlapping" when the same entity (face or edge) is a member of more than one contact region or when multiple contact regions share the same geometry. When you export a mesh to Ansys Fluent or Ansys CFX, contact regions become mesh interfaces and interfaces must be unique. That is, if there are overlapping contact regions, the overlaps need to be resolved into exclusive groups for the export to be successful.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Overlapping entities can be from a variety of sources. In the examples shown in the following series of images, the first contact region has one face as a contact and one as a target, this is a good connection for Ansys Fluent or Ansys CFX. The other three contact regions have two faces as targets and only one face as a contact, the extra target face in each of these contact regions overlaps. Hence, results in overlapping contact regions that may need to be addressed for the export to succeed. In this particular case, the default tolerance that was used to detect contacts was too large. This problem could be fixed by deleting all the contacts and using a smaller Face Overlap Tolerance to recreate them.

Figure 25: First Contact Region: One Contact and One Target

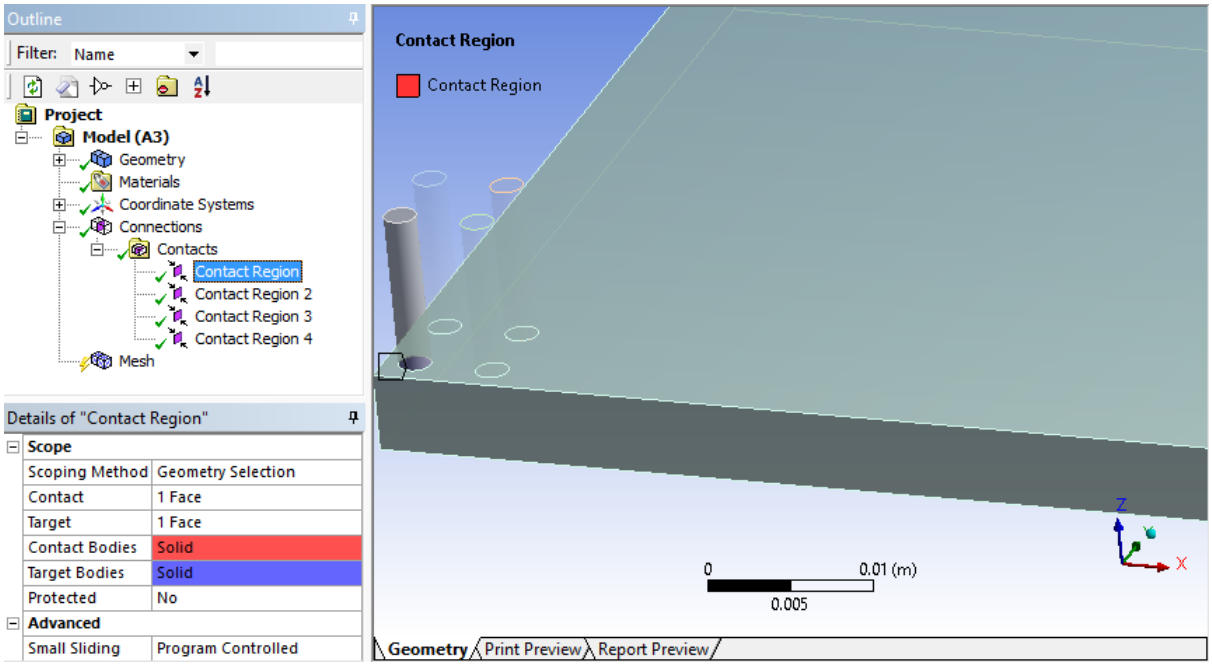
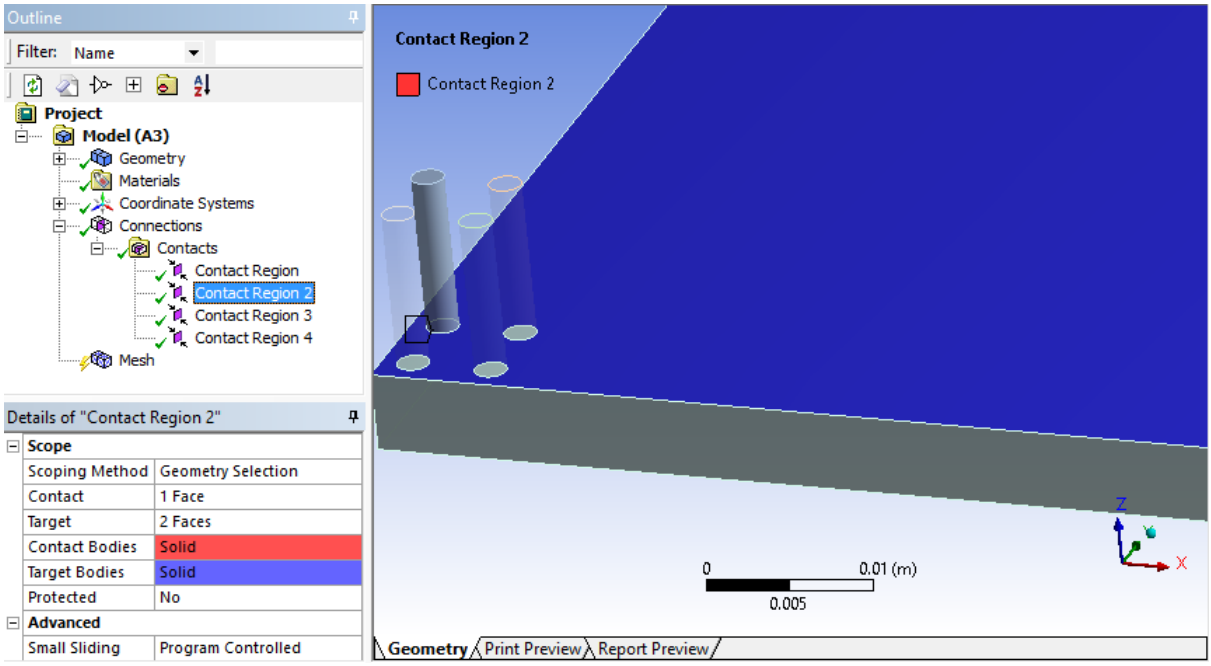


Figure 26: Second Contact Region: One Contact, Two Targets



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 27: Third Contact Region: One Contact, Two Targets

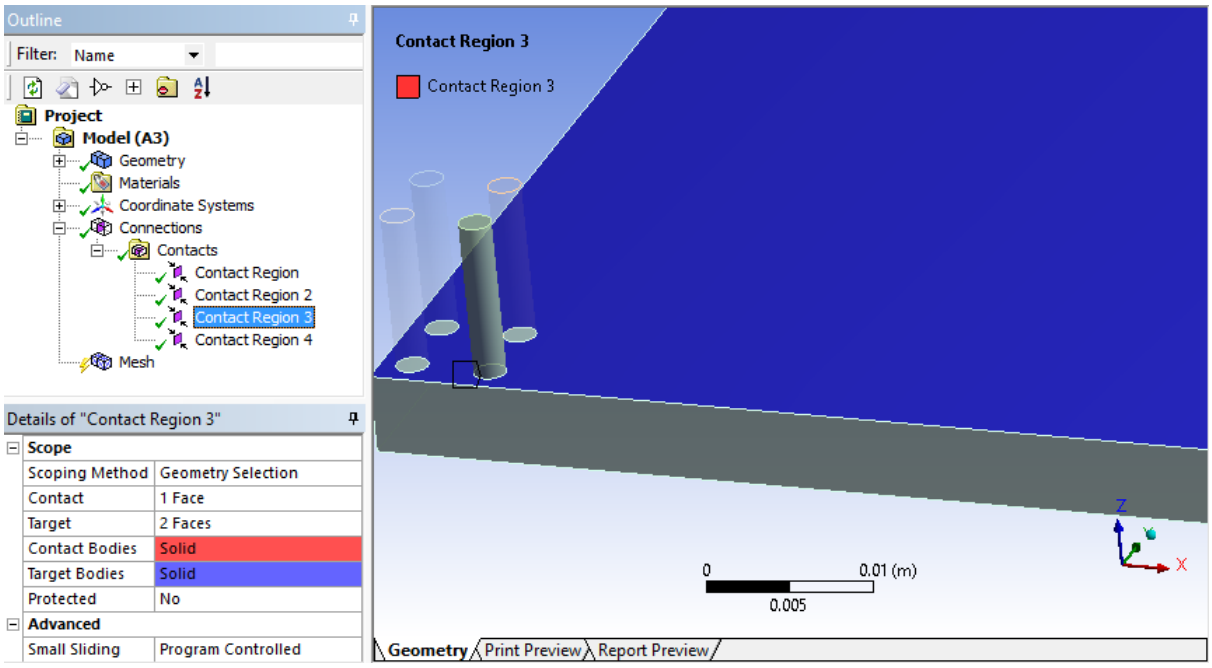
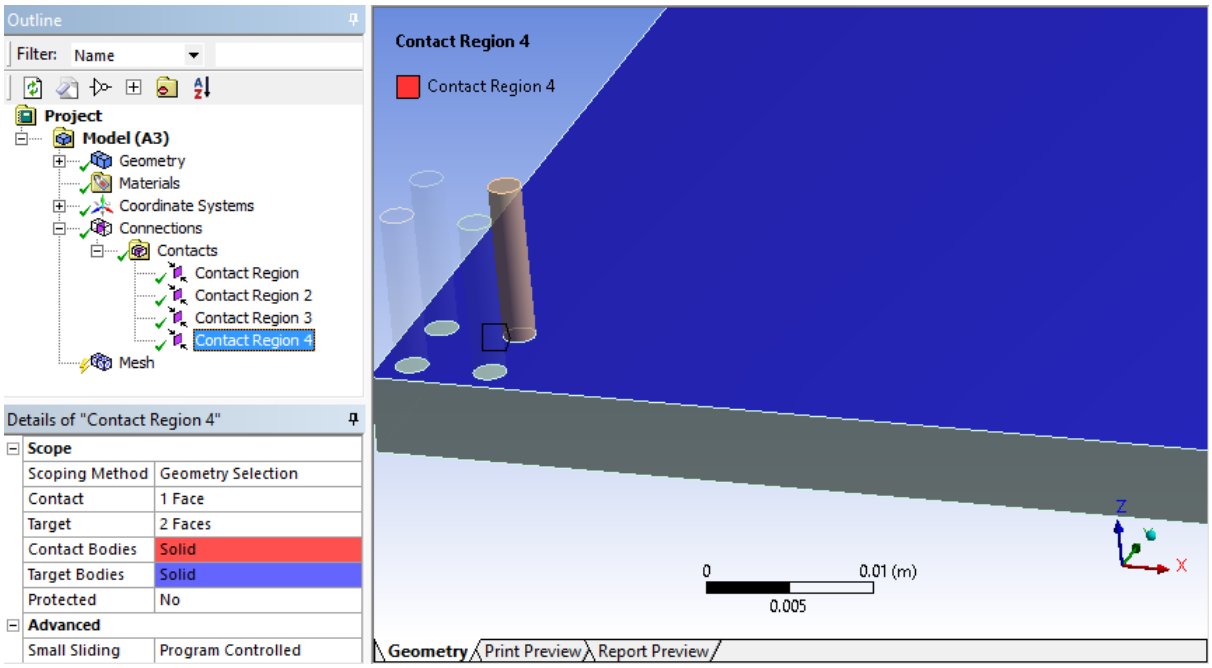


Figure 28: Fourth Contact Region: One Contact, Two Targets



Other cases of overlap may have different causes and solutions. To identify overlapping contact regions so that you can resolve them and export the mesh successfully, follow these steps:

1. Right-click the **Connections**, **Connection Group**, or **Contacts** folder in the Tree and choose **Check Overlapping Contact Regions**.

   If any overlapping contact regions are found, an informational message appears in the **Messages** window.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Right-click the message and select **Go To Object**.

   The overlapping contact regions that are responsible for the message are highlighted in the Tree.

3. With all of the overlapping contact regions still highlighted in the Tree, right-click one of them and select **Group**.

   All of the overlapping contact regions are grouped into a new folder to isolate them for diagnosis.

To resolve the overlapping contact regions, select each contact region that was placed into the new folder one at a time and examine its properties. Verify that all associated properties are defined appropriately. The following approaches are recommended:

- Often tolerances are set too large, as described in the example shown above. Try deleting all the contacts and using smaller tolerance values to recreate them. It may help to set **Tolerance Type** to **Value** rather than **Slider**, so that you can enter an exact **Tolerance Value**. Make sure the Tolerance Value and Face Overlap Tolerance are set to reasonable values before regenerating the contacts.

- Depending on the case, setting Group By to **Faces** may help to pinpoint problematic faces. In other situations, setting **Group By** to **Bodies** and promoting the overlapping contact regions to Named Selections may be a better approach.

- If the tolerances are different for different bodies, run contact on selected bodies with different tolerances.

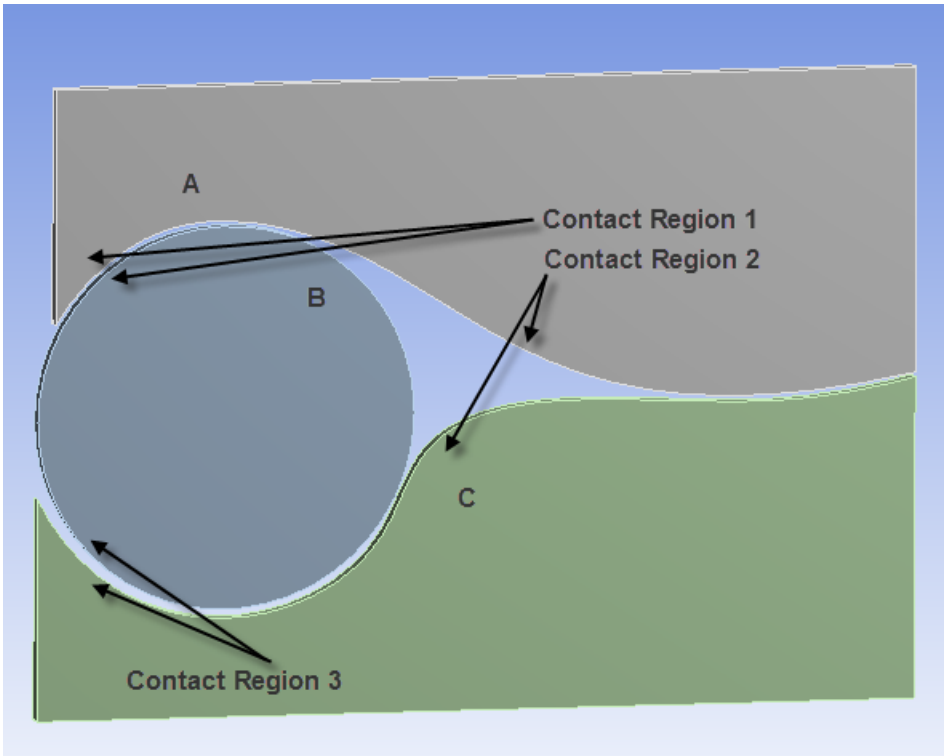
- If the faceting of the geometry is too coarse, increase the facet tolerance in DesignModeler.

- If some of the overlapping contact regions are not needed in Ansys Fluent or Ansys CFX, you can suppress them so they will not affect the success of the export.

- If the case involves cyclic constraints, use imprinting in DesignModeler or Discovery Modeling. Imprinting results in exact pairs and ensures faces have common boundaries between parts. Refer to Figure 29: Geometry with Cyclic Redundancies (p. 83), which illustrates a complex case of cyclic redundancy. Here the overlap is A->B->C->A, where A is the face contact of the top body, B is the cylindrical face of the middle body, and C is the face in contact of the bottom body.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 29: Geometry with Cyclic Redundancies



After you have resolved all of the overlapping contact regions, you can retry the export.

Note:

This feature is applicable to part-based meshing only.

Extended Ansys ICEM CFD Meshing

The features described here extend Meshing application functionality through integration of the Meshing application with Ansys ICEM CFD, and enable you to use Ansys Workbench to drive automation of Ansys ICEM CFD.

With this approach you can leverage advantages of Ansys Workbench such as:

- Its capabilities for running design optimization and what-if scenarios using parameters
- The strength of its core product solvers

The following topics are discussed in this section:

Writing Ansys ICEM CFD Files
Rules for Interactive Editing
Limitations of Ansys ICEM CFD Interactive

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Writing Ansys ICEM CFD Files

The **Write ICEM CFD Files** control determines whether Ansys ICEM CFD files are written, and includes options for running Ansys ICEM CFD interactively or in batch mode from an Ansys ICEM CFD Replay file.

Note:

The **Write ICEM CFD Files** control is available when you use any Ansys ICEM CFD mesh method that is available in the Meshing application (Patch Independent Tetra (p. 210), MultiZone (p. 242), or MultiZone Quad/Tri (p. 284)). However, the **Interactive** and **Batch** options are available only if Ansys ICEM CFD is installed. A warning is issued if you try to use one of these options and Ansys ICEM CFD cannot be found.

Note:

Another method for writing meshing data to ICEM CFD files is to select the Mesh cell in Workbench and select **Transfer Data to New>ICEM CFD**. While the **Write ICEM CFD** control writes the geometry of a part, using the Ansys ICEM CFD Add-in component links to the entire Geometry assembly. For more information, see Component Systems in the Ansys Workbench User's Guide.

Options for **Write ICEM CFD Files** include:

No

No files are written. This is the default.

Yes

Writes Ansys ICEM CFD files. Useful when you are working in Ansys Workbench but you want to export your project files for further mesh editing in Ansys ICEM CFD. If this control is set to **Yes**, your Ansys ICEM CFD project (.prj), geometry (.tin), unstructured domain (.uns), and blocking (.blk) files will be saved during mesh generation.

- *If your Ansys Workbench project file has been saved,* your Ansys ICEM CFD files will be written to the Ansys Workbench project directory. Ansys Workbench creates a project folder as the top level folder for each project, at the same level as the project file. The project folder will be named `<filename>_files`, where `<filename>` is a name you provide. The project file will be named `<filename>.wbpj`. Under the project folder is a design point subdirectory for each design point in the project. The active design point is always design point 0 (dp0) and corresponds to the design point labeled **Current** in the Ansys Workbench GUI. Under each design point folder are system folders for each system type in the project (Mechanical, Fluent, etc.). Each system folder contains solver-specific files and folders, such as input files, model directories, engineering data, resources, etc. Following this structure, the Ansys ICEM CFD files will be written to `<filename>_files\dp0\global\MECH\SYS`.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- *If your Ansys Workbench project file has not been saved,* your Ansys ICEM CFD files will be written to the temporary Ansys Workbench folder, as follows: `%TEMP%\WB_<computer>\unsaved_project_files\dp0\global\MECH\SYS`.

Note:

- Only appropriate files are written, based on the selected mesh method.
- Refer to the Ansys Workbench help for more information about project file management in Ansys Workbench.

Interactive

Applicable only when Ansys ICEM CFD is installed.

When the **Interactive** option is set and you select **Generate Mesh**, the Meshing application launches Ansys ICEM CFD in interactive mode. When you specify the **Interactive** option, you must also select an option for **ICEM CFD Behavior** (p. 86) to determine whether the geometry and/or mesh is transferred to Ansys ICEM CFD.

Typically, you run in interactive mode to set up an Ansys ICEM CFD Replay script file (*.rpl) that can be run later in either batch or interactive mode. You can begin by loading a generic Replay file. It is important to use this default Replay file because the batch process requires the pre and post steps that are defined within it. To load the default Replay file:

1. From within Ansys ICEM CFD, select **File** > **Replay Scripts** > **Replay Control** and the **Replay Control** window appears.
2. Click **Load** on the **Replay Control** window and the **Open Script File** window appears.
3. Click **Open** on the **Open Script File** window to load the default Replay file.

You can incorporate your custom commands into the Replay file by using the Replay Control feature or a text editor. The Replay file will be associated with your Ansys ICEM CFD project when you save the project and exit Ansys ICEM CFD.

After the mesh is returned to the Meshing application and the Ansys Workbench project is saved, the Replay file will be written to the Ansys Workbench project directory. Later you can set **Write ICEM CFD Files** to **Batch** and the mesh will be updated automatically in batch. You can change parameters on the Ansys Workbench project page and **Update** the mesh in batch from either the project page or from within the Meshing application.

Refer to Rules for Interactive Editing (p. 86) and Limitations of Ansys ICEM CFD Interactive (p. 87) for related information.

For more information about Replay Control, refer to the documentation available under the Help menu within Ansys ICEM CFD.

Batch

Applicable only when Ansys ICEM CFD is installed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Runs Ansys ICEM CFD in batch mode from an existing Replay file. If you specify the **Batch** option, you must also select an option for **ICEM CFD Behavior** (p. 86) to determine whether the geometry and/or mesh is transferred to Ansys ICEM CFD.

ICEM CFD Behavior

Determines Ansys ICEM CFD behavior when running Ansys ICEM CFD in **Interactive** or **Batch** mode.

For Patch Independent Tetra, the available options under **Advanced** are:

- **Generate Mesh** - After the meshing operation completes, transfers both the geometry and mesh to Ansys ICEM CFD for editing.
- **Skip Meshing** - Bypasses the meshing operation and transfers only the geometry to Ansys ICEM CFD for meshing and editing.

For MultiZone and MultiZone 2D, the available options under **Advanced** are:

- **Generate Blocking and Mesh** - After the meshing operation completes, transfers the geometry, blocking, and mesh to Ansys ICEM CFD for editing.
- **Generate Blocking** - Bypasses the meshing operation and transfers only the geometry and blocking to Ansys ICEM CFD for meshing and editing.
- **Update pre-existing Blocking** - Bypasses the blocking operation, updates pre-existing blocking and meshes, and transfers the geometry and mesh to ANSYS ICEM CFD for meshing and editing. When you select **Update pre-existing Blocking**, the **Update Sizes** is set to **Yes** by default and updates the bunching with the given Workbench parameters while meshing. When the **Update Sizes** is set to **No**, protects the bunching in ICEM CFD files.
- **Named Selection Treatment**: Provides named selection transfer for Interactive and Batch. The available options are Parts and Subsets.

Rules for Interactive Editing

The final mesh must pass Ansys Workbench shape and topology checks in order for the mesh to be returned to Ansys Workbench. This requirement imposes the following rules and guidelines for editing in Ansys ICEM CFD:

- Do not modify the geometry.
- Pay attention to face and volume part naming. Use the same naming conventions and do not adjust the part naming.
- Pay attention to mesh quality. If the mesh quality becomes degraded, the mesh will not be returned to Ansys Workbench unless Check Mesh Quality (p. 121) is set to **No**.
- The final mesh must pass Ansys Workbench shape and topology checks in order for the mesh to be returned to Ansys Workbench without a warning or error message.
- Follow file naming conventions for proper archiving and batch interaction. Retain the default names. Changing the names will break the association between the Replay file and the Ansys Workbench project.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Limitations of Ansys ICEM CFD Interactive

Be aware of the following limitations when using Ansys ICEM CFD Interactive:

- Ansys ICEM CFD Interactive is designed to work at the part level. If you have assigned multiple Ansys ICEM CFD mesh methods (Patch Independent Tetra (p. 210), MultiZone (p. 242), or MultiZone Quad/Tri (p. 284)) to different bodies of a multibody part, the Ansys ICEM CFD Interactive options specified for the Ansys ICEM CFD method control that appears lowest in the Tree will be honored. That is, Ansys ICEM CFD Interactive options will affect all bodies being meshed with Ansys ICEM CFD methods regardless of whether a particular option is turned on for a particular body.

Working with Meshing Application Parameters

The term *parameters* in the Meshing application includes mesh input parameters (such as element size, height of the initial inflation layer, and growth rate) as well as mesh output parameters (number of elements, number of nodes, and mesh metric values).

A check box appears to the left of each field in the Details View that can be treated as a parameter. Clicking the check box causes a **P** to appear in the box, which indicates that the field has been exposed as a parameter for use in the Parameter Workspace. Fields that cannot be parameterized do not contain a check box and are left-aligned to save space. The Parameter Workspace collects all specified parameters and lists them in the Parameter Workspace grids for later use and/or modification.

Note:

- If an object has a parameterized field, and that object definition is changed in a way that makes that parameterization non-meaningful, the non-meaningful parameterization will *not* be removed by the program. For example, if there is a parameterized **Number of Divisions** sizing control defined in the Meshing application and you switch to the **Element Size** sizing control and parameterize it, the **Number of Divisions** parameterization will not be removed and will continue to appear in the Parameter Workspace grids. The presence of a non-meaningful parameter in the grids has no harmful effect, but you can delete the parameter manually if you do not want it to appear. To do so, return to the Meshing application and clear the corresponding check box in the Details View.
- If a mesh control is suppressed, the parameter associated with the control will be deleted.
- If you are using a parameterized field in a Design Point study, you must specify a real value for the field rather than using its default value.
- Refer to the Ansys Workbench help for detailed information about the Parameter Workspace and Design Points.

Ansys Workbench and Mechanical APDL Application Meshing Differences

While the meshing algorithms used by Ansys Workbench Mechanical originated from the meshing capabilities present in the Mechanical APDL (MAPDL) application, over time these algorithms have diverged. The divergence was due to the different focus of early versions of the Workbench Mechanical

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

application (then known as DesignSpace). As Workbench Mechanical and its meshing and solving capabilities evolved, new technology was added and existing technology was enhanced, making Workbench Mechanical a full, general-purpose, finite element code that supports all levels of multiphysics disciplines.

To accommodate ease of use in Workbench Mechanical, as well as the need for different default meshes based on simulation type, physics preferences (p. 96) are available. These physics preferences automate default mesh settings related to element size, element quality, and so on. The **Mechanical** physics preference has two choices for **Error Limits** (p. 127), or shape check values:

- **Standard Mechanical**, which uses quality error limits that are less strict than those used by MAPDL.
- **Aggressive Mechanical**, which uses quality error limits that are similar to those used by MAPDL.

In addition to letting you set a value for **Error Limits**, Workbench Mechanical lets you set values for **Target Quality** and **Check Mesh Quality** (p. 121). The mesher uses the **Target Quality** as a goal for mesh quality. You can think of the **Target Quality** as a warning limit (in MAPDL terminology), but you can set the **Target Quality** however you deem appropriate. **Check Mesh Quality** is very similar to MAPDL's **Level of shape checking**. Depending on the setting of **Check Mesh Quality**, you can check for errors, errors and warnings, or neither (turn off checks altogether).

One of the major differences between **Standard Mechanical** and **Aggressive Mechanical** error limits is in the computation of the Jacobian ratio. The Jacobian Ratio (p. 142) is a metric that compares a given element's shape to that of an ideal element. Jacobian ratio can be computed at the corner nodes (**Aggressive Mechanical**) or at Gauss points (**Standard Mechanical**). Depending on the type of simulation, the calculation at the Gauss points may be sufficient and therefore **Standard Mechanical** makes meshing more robust. If you are interested in higher accuracy or your problem is nonlinear, you can set **Error Limits** to **Aggressive Mechanical** or set **Physics Preference** to **Nonlinear Mechanical**. In both cases, the Jacobian ratio that is used is computed at the corner nodes (as it is in MAPDL).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Meshing: Mesh Controls Overview

When in the Mechanical application, your part or multibody part is automatically meshed at solve time. The default element size is determined by the software (for details, see default sizing (p. 104)).

You can also create the mesh before solving. To do this, select **Mesh** from the Tree Outline and right-click your mouse. Mesh controls are available to assist you in fine tuning the mesh to your analysis. These controls are described in the sections that follow.

Mesh control overview topics include:

Global and Local Mesh Controls

Understanding the Influence of the Sizing Options

Global and Local Mesh Controls

If you want more control over the mesh, you can get it by using mesh control tools. Both global mesh controls (p. 93) and local mesh controls (p. 201) are available. Global mesh controls are located in the Details View when the **Mesh** object is selected in the Tree Outline. To access the local mesh control tools, highlight the **Mesh** object, then either click the right mouse button and choose **Insert> {<choose a mesh control tool>}**, or choose a mesh control from the **Mesh Control** context toolbar.

In general, in cases where a numeric value can be specified for an option but a value of 0 would not make sense within the context of the option, specifying 0 resets the option to its default. For example, the **Curvature Normal Angle** (p. 114) option accepts a value from 0 to 180, but since 0 does not make sense within its context, specifying 0 resets **Curvature Normal Angle** to its default.

Understanding the Influence of the Sizing Options

An important aspect of meshing in Workbench is the size function. Depending on the physics preference, the sizing options allow you to control whether you want a uniform mesh or whether the mesh should be influenced by the model's curvature and/or proximity of faces next to each other (thickness of model).

The following chart recommends how to get the appropriate mesh:

	Uniform mesh	Mesh with Curvature Refinement	Mesh with Proximity Refinement	Mesh with 2D Curvature and Proximity Refinement
Physics Preference = **Nonlinear Mechanical**	Set **Capture Curvature** and **Capture Proximity** to **No**	Set **Capture Curvature** to **Yes**	Set **Capture Proximity** to **Yes**	Not recommended

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If using any other **Physics Preference**	Set **Use Adaptive Sizing**, **Capture Curvature**, and **Capture Proximity** to **No**	Set **Capture Curvature** to **Yes**	Set **Capture Proximity** to **Yes**	Set both **Capture Curvature** and **Capture Proximity** to **Yes**

Note:

When **Physics Preference** is **Hydrodynamics**, the only properties you can set are Element Size (p. 102) and Defeature Size (p. 111).

Adaptive Sizing

Adaptive Sizing can be thought of as a 2D curvature and proximity-based refinement approach which refines edges based on curvature and/or proximity but does not propagate the refined mesh along the face. It works with **Resolution**, **Span Angle Center** and **Transition**, and has historically been used as a way to reduce the total element count while capturing every edge of the model. This approach can significantly stretch the mesh leading to poor mesh quality, so it is generally recommended to use one of the other sizing options and use defeaturing to reduce the total element count when needed.

When **Use Adaptive Sizing** is set to **Yes**, the mesher uses the value of the element size property to determine a starting point for the mesh size. The value of the element size property can be set by the user or automatically computed using Defaults (p. 104). When meshing begins, edges are meshed with this size initially, and then they are refined for curvature and 2D proximity. Next, mesh based defeaturing and pinch control execution occurs. The final edge mesh is then passed into a least-squares fit size function, which guides face and volume meshing.

Uniform, Curvature, Proximity, or Proximity and Curvature Sizing

A size function is computed when meshing begins. The mesher examines the global and local sizes and uses the smallest, most local size to seed the mesh. It uses the growth rate to transition the mesh from a small size to a larger size.

The following factors contribute to the final mesh distribution obtained by the mesher:

- The value of the Element Size (p. 102) option
- The value of the Curvature Min Size (p. 113)/Proximity Min Size (p. 115) option
- The value of the Max Size (p. 110) option
- The value of the Growth Rate (p. 110) option
- Features of the geometry, which can be any of the following:
  - Edge and face curvature (p. 107), based on the normal angle variation between adjacent mesh elements in 2D (edge) or 3D (face)
  - Edge and face proximity (p. 107), based on the number of element layers created between a gap between edges in 2D or between faces in 3D

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Local element sizing (p. 296) on selected edges, faces, or bodies
- Sphere of influence (p. 304) scoped to a selected body, face, edge, or vertex
- Body of influence (p. 305) scoped to a selected body
- Influence of an existing mesh or a swept (p. 399) body

Note:

- The mesher uses the sizes defined by the user or computes its own Defaults (p. 104).
- The size function works within parts, but not across parts.
- When **Capture Curvature** and/or **Capture Proximity** are set to **Yes**, the default pinch tolerance (p. 192) is 90% of the value of **Curvature Min Size**/**Proximity Min Size** (whichever is smaller). This differs from the tolerance used by the default mesh based defeaturing. Refer to Mesh Defeaturing (p. 110) for details.
- The size function computes a background grid prior to meshing and re-uses that background grid during meshing. This background grid could differ for solid parts and sheet parts.
- The size function may over-refine bodies being meshed with sweep or MultiZone as side faces can influence source faces and source faces can influence side faces to try to make the mesh more uniform. Use hard edge sizing controls or set the number of divisions along the sweep path to override such behavior.
- In some situations, the mesh will exceed the applied **Element Size** setting to create a better quality mesh. To set a strict upper limit for each element edge, set the **Element Size** to a size slightly smaller than the desired size.

Overriding Sizing Minimum and Maximum Sizes

One of the most important values related to sizing is minimum size (Curvature Min Size (p. 113) and Proximity Min Size (p. 115) controls). Setting a value for minimum size that is too small may lead to an unnecessarily fine mesh and longer meshing time or robustness issues due to the mesh tolerances being smaller than the geometry tolerances (mesh falling into holes/gaps in the model). On the other hand, setting a value that is too large may mean that important features are not captured by the mesh.

The following notes may help:

- Use local sizing controls to override global controls to either increase or decrease the mesh size as necessary.
- Use the **Mechanical Min Size Factor** and/or **CFD Min Size Factor** to change the relationship between **Element Size** and the default min size. See Sizing (p. 392) for details.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Meshing: Global Mesh Controls

Global mesh controls are located in the Details View when the **Mesh** object is selected on the Tree Outline.

Global settings are categorized into the following groups and are available through drop-down menus:

Display Group

Defaults Group

Sizing Group

Quality Group

Inflation Group

Advanced Group

Automatic Methods

Statistics Group

Model Assembly Group

Display Group

Display group allows you to display mesh on parts or bodies of the model based on the available styles. **Display** group have the following option:

Display Style

Display Style

Display Style allows you to change the display style of the mesh in the Geometry window based on **Display Style** options. The default value is **Use Geometry Setting**. **Display Style** has the following options:

- **Use Geometry Setting**: The display of the mesh in the **Geometry** window is based on the setting of the **Display Style** property of the **Geometry**.
- **Shell Thickness** (visible for surface/shell bodies only)
- Element Quality (p. 139)
- **Aspect Ratio**, based on either

  Aspect Ratio Calculation for Triangles (p. 140)

  Aspect Ratio Calculation for Quadrilaterals (p. 141)
- Jacobian Ratio (p. 142) (MAPDL, Corner Nodes, or Gauss Points)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

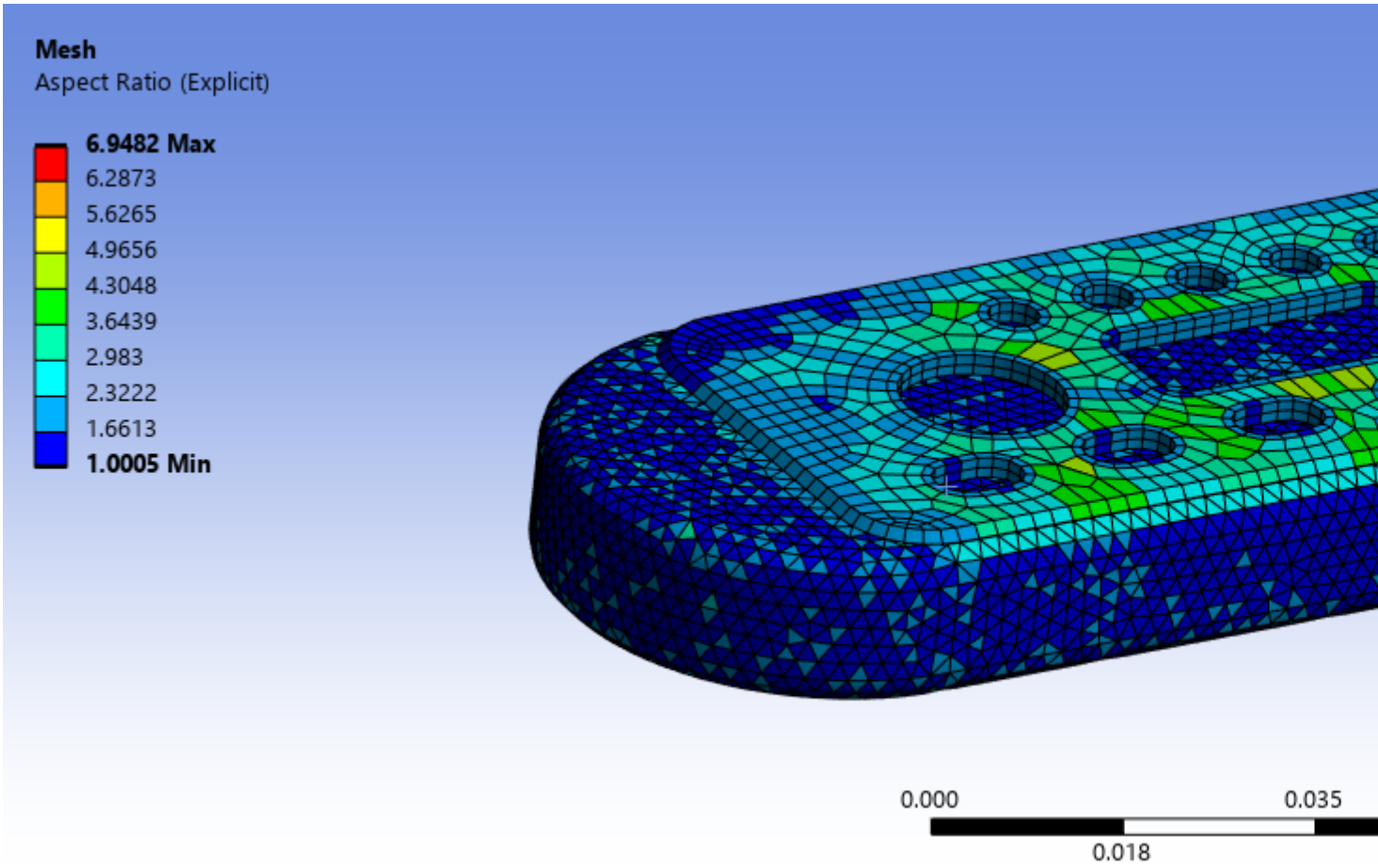
- Warping Factor (p. 147)
- Parallel Deviation (p. 149)
- Maximum Corner Angle (p. 150)
- Skewness (p. 151)
- Orthogonal Quality (p. 153)
- Characteristic Length (Autodyn) (p. 155)
- Minimum Tri Angle (p. 156)
- Maximum Tri Angle (p. 156)
- Minimum Quad Angle (p. 157)
- Maximum Quad Angle (p. 157)
- Warping Angle (p. 157)
- Tet Collapse (p. 158)
- Aspect Ratio (Height) (p. 159)
- Minimum Element Edge Length (p. 159)
- Maximum Element Edge Length (p. 160)
- Characteristic Length (LS-DYNA) (p. 160)
- **Deviation from Geometry**: Display the deviation of the mesh nodes from the geometry.

Note:

The use of **Display Style** to color mesh according to quality and the use of **Mesh Metrics** (p. 133) to view quality statistics are mutually exclusive. For example, when you are viewing **Mesh Metrics**, you cannot also view the mesh quality by color.

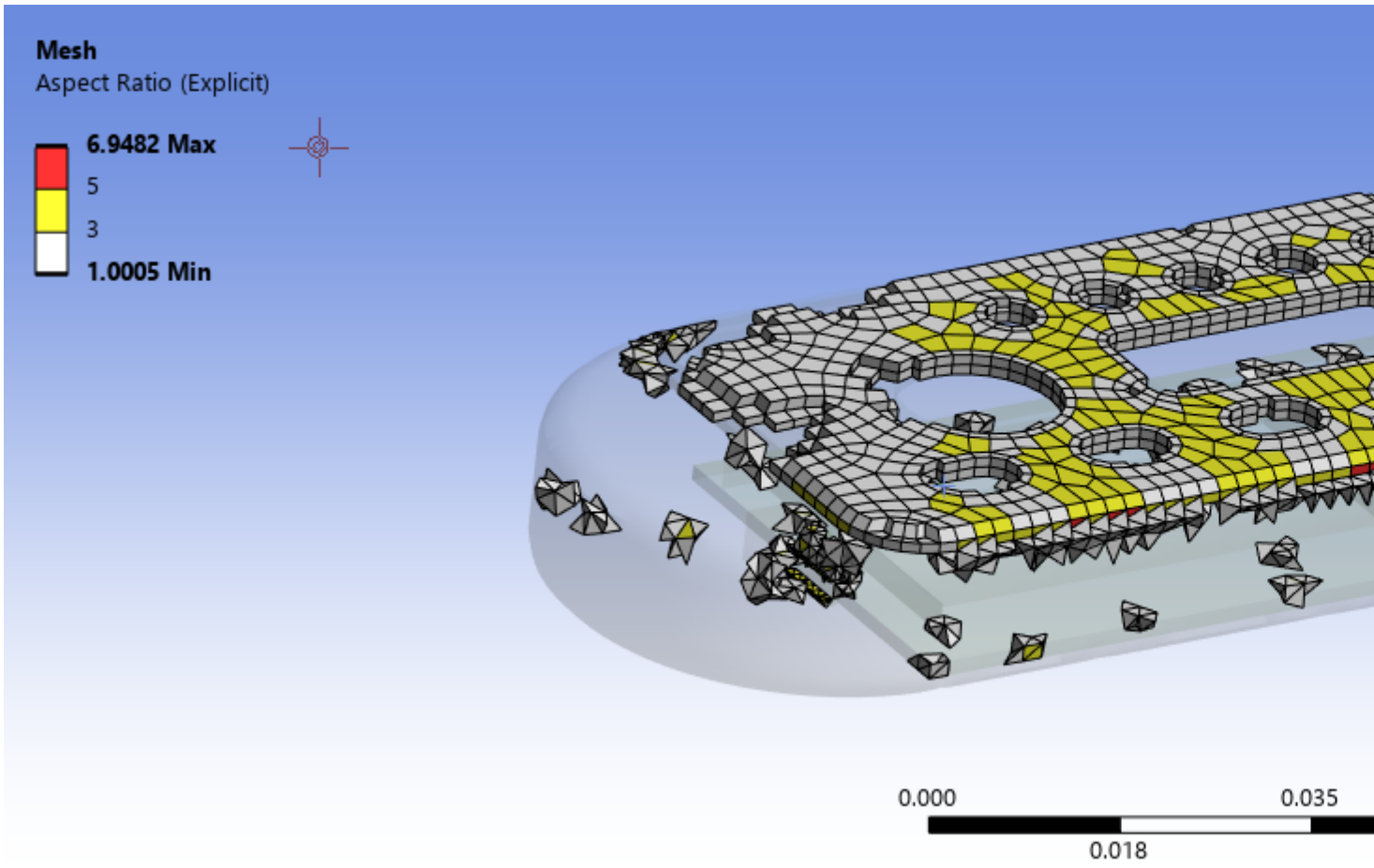
When **Check Mesh Quality** is **No**, **Yes Errors and Warnings** and **Yes, Errors** the **Show Contours** in **Mesh Quality Worksheet** set the **Display Style** to selected quality criterion.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When **Check Mesh Quality** is **Mesh Quality Worksheet** the **Show Contours** in **Mesh Quality Worksheet** set the **Display Style** to selected quality criterion and displays the warning (yellow) and failed (red) elements for the selected quality criterion.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Defaults Group

The **Defaults** group allows you to control these options:

Physics Preference

Solver Preference

Export Format

Export Unit

Export Preview Surface Mesh

Element Order

Element Size

Physics Preference

Physics Preference option allows you to establish how Workbench performs meshing based on the physics of the analysis type that you specify. The available options are:

- **Mechanical**
- **Nonlinear Mechanical**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Electromagnetics**
- **CFD**
- **Explicit**
- **Hydrodynamics**

The value of the **Physics Preference** option sets the default for various meshing controls, as detailed in the following table.

Note:

- The solver options for the **Mechanical** physics preference appear within the Mechanical application only, and only under certain circumstances. See Solver Preference (p. 99) for more information.
- When **Physics Preference** is **Hydrodynamics**, the only properties you can set are Element Size (p. 102) and Defeature Size (p. 111). The meshing controls that appear in the table below are not applicable to the **Hydrodynamics** physics preference. Therefore, **Hydrodynamics** is not included in the table. Refer to Overview of the Meshing Process for Hydrodynamics Analysis (p. 30) for related information.
- When the **Physics Preference** is **Explicit**, the quality based meshing considers **Aspect Ratio (Height)** for **Patch Conforming Tetrahedron Method**. Also, in the **Mesh Details** view under **Advanced** options, the **Triangle Surface Mesher** is set to **Advancing Front** by default. This provides more uniform triangle meshing. If the **Multizone Mesh Method** is selected, the **Use Split Angle** in **Advanced** option is enabled to give automatically more orthogonal meshing for many cases.
- Refer to the Meshing Overview (p. 19) section for further details about physics based meshing.

Meshing Control	Physics Preference							
	Mechanical		Nonlinear Mechanical	Electromagnetics	CFD			Explicit
	Mechanical APDL Solver (p. 99)	Ansys Rigid Dynamics Solver (p. 99)			CFX-Solver (p. 99)	Fluent Solver (p. 99)	Polyflow Classic Solver (p. 99)	
Element Order (p. 101)	Program Controlled	Linear (Read-only)	Program Controlled	Quadratic	N/A	N/A	N/A	Linear
Straight Sided Elements (p. 180)	No	N/A	No	Yes	N/A	N/A	N/A	N/A
Element Size (p. 102)	Assembly Diagonal * Bound-	Assembly Diagonal *	Assembly Diagonal * Bound-	0.5 * Assembly Diagon-	Assembly Diagonal *	Assembly Diagonal *	Assembly Diagonal *	0.5 * Assembly Diagon-

Meshing Control	Physics Preference							
	Mechanical		Nonlinear Mechanical	Electromagnetics	CFD			Explicit
	Mechanical APDL Solver (p. 99)	Ansys Rigid Dynamics Solver (p. 99)			CFX-Solver (p. 99)	Fluent Solver (p. 99)	Polyflow Classic Solver (p. 99)	
	ing Box Factor	**Bounding Box Factor**	**ing Box Factor**	**al * Bounding Box Factor**	**Bounding Box Factor**	**Bounding Box Factor**	**Bounding Box Factor**	**al * Bounding Box Factor**
Sizing Options (p. 105)	**Capture Curvature** set to **No** for sheet bodies; otherwise **Use Adaptive Sizing** set to **Yes**	**Capture Curvature** set to **Yes** for shell models; otherwise **Use Adaptive Sizing** set to **Yes**	**Capture Curvature** set to **Yes**	**Use Adaptive Sizing** set to **Yes**	**Capture Curvature** set to **Yes**	**Capture Curvature** set to **Yes**	**Capture Curvature** set to **Yes**	**Capture Curvature** set to **Yes** for solid bodies and **No** for sheet bodies
Transition (p. 112) / Growth Rate (p. 110)	**Fast / 1.85** for solid bodies and **1.2** for sheet bodies	**Fast** / **1.85** for solid bodies and **1.2** for sheet bodies	**N/A**/ **1.5**	**Fast** /**1.85** for solid bodies and **1.2** for sheet bodies	**Slow**/ **1.2**	**Slow** / **1.2**	**Slow** / **1.2**	**Slow** / **1.5**
Span Angle Center (p. 112) / Curvature Normal Angle (p. 114)	**Coarse** / 70.395°	**Coarse** / 70.395°	**N/A** / 60°	**Coarse** / 70.395°	**Fine** / 18°	**Fine** / 18°	**Fine** / 18°	**Coarse** / 72°
Error Limits (p. 127)	**Aggressive Mechanical**	**Aggressive Mechanical**	**Nonlinear Mechanical**	**Electromagnetics**	**CFD**	**CFD**	**CFD**	**Explicit**
Smoothing (p. 143)	**Medium**	**Medium**	**N/A**	**Medium**	**Medium**	**Medium**	**Medium**	**High**
Inflation Algorithm (p. 170)	**Pre**	**Pre**	**Pre**	**Pre**	**Pre**	**Pre**	**Pre**	**Pre**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Meshing Control	Physics Preference							
	Mechanical		Nonlinear Mechanical	Electromagnetics	CFD			Explicit
	Mechanical APDL Solver (p. 99)	Ansys Rigid Dynamics Solver (p. 99)			CFX-Solver (p. 99)	Fluent (p. 99)	Polyflow Classic Solver (p. 99)	
Collision Avoidance (p. 174)	Stair Stepping	Stair Stepping	Stair Stepping	Stair Stepping	Stair Stepping	Layer Compression	Stair Stepping	Stair Stepping
Transition Ratio (p. 168)	0.272	0.272	0.272	0.272	0.77	0.272	0.272	0.272
Rigid Body Behavior (p. 182)	Dimensionally Reduced	Dimensionally Reduced	Dimensionally Reduced	Dimensionally Reduced	Dimensionally Reduced	Dimensionally Reduced	Dimensionally Reduced	Full Mesh
Mesh Quality Workflow Targets	Element Quality	Element Quality	Skewness, Jacobian Ratio (Corner Nodes)	Jacobian Ratio (Corner Nodes)	Element Quality	Skewness	Skewness	Element Quality, Aspect Ratio (Height), Characteristic Length (LS-Dyna)
Defeature Size (p. 111)	0.005	0.005	0.005	0.005	0.005	0.005	0.005	0.1*Element Size

Solver Preference

Solver Preference allows you to select the solver preference for your **Physics Preference**. **Solver Preference** is available only when **Physics Preference** is **CFD**. The available options are **Fluent**, **CFX**, and **Polyflow**. Based on your selection, the Meshing application sets certain defaults that result in a mesh that is more favorable to the **CFX**, **Fluent**, or **Polyflow solver,** respectively. Inflation controls with defaults that are affected by the Solver Preference setting include:

- **Aspect Ratio (Base/Height)** (p. 170)
- **Collision Avoidance** (p. 174)
- **Transition Ratio** (p. 168)

Within the Mechanical application only, choosing **Mechanical** as your **Physics Preference** also causes a **Solver Preference** option to appear, provided the analysis system is a Transient Structural or Rigid Dynamics system during the initial geometry attach. In such cases, you can set **Solver Preference** to

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

either **Mechanical APDL** or **Ansys Rigid Dynamics**. **Solver Preference** setting affects controls with defaults, including **Element Order** (p. 101).

Other differences (such as what happens when you insert a **Method** control) are dependent on the behavior of the geometry bodies and the **Rigid Body Behavior** setting. See Rigid Body Meshing (p. 463) for more information.

Refer to Physics Preference (p. 96) for more information about defaults that are set based on **Physics Preference** and **Solver Preference**.

Export Format

Export Format defines the format for the mesh when exported to Ansys Fluent. **Export Format** is available only when **Physics Preference** is **CFD** and the **Solver Preference** is **Fluent**. The available options are:

- **Standard**: Export the mesh as face based fluent mesh. The default value is **Standard**.
- **Large Model Support**: Export the mesh as a cell based fluent mesh.

Note:

The **Large Model Support** option is not available for the following types of meshes:

- 2D

For more information about exporting meshes to Ansys Fluent, see Fluent Mesh Export (p. 43).

Export Unit

Export Unit defines the unit of measurement for the mesh when exported to Ansys Polyflow. **Export Unit** is available only when **Physics Preference** is **CFD** and the **Solver Preference** is **Polyflow**. The default value is **Use Project Unit**, which means the mesh is not scaled. The available options are **Meters**, **Centimetres**, **Millimetres**, **Micrometers**, **Inches**, or **Feet**. Mesh scales according to your selected export unit.

Export Preview Surface Mesh

Export Preview Surface Mesh option controls the export of the preview surface mesh elements. **Export Preview Surface Mesh** is available only when **Physics Preference** is **CFD**, the Solver Preference is **Fluent**.

You can use **Export Preview Surface Mesh** when the bodies are only partially meshed. That is, not all volumes have been filled with elements and only surface meshes are previewed. In such cases, you can choose to export the previewed surface meshes and continue meshing in Fluent Meshing. The available options are:

- **No**: Exports only volume mesh element to the Fluent mesh file. The default value is **No**.
- **Yes**: Exports both volume mesh and preview surface mesh to the Fluent mesh file.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element Order

The global **Element Order** option allows you to control whether meshes are to be created with midside nodes (quadratic elements) or without midside nodes (linear elements). Reducing the number of midside nodes reduces the number of degrees of freedom.

Element Order has the following options:

- **Program Controlled**
- **Linear**
- **Quadratic**

Refer **Physics Preference** for default values.

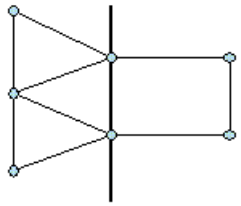
Examples are presented below for linear and quadratic elements. The heavy vertical line in each graphic represents the body boundary.

- **Program Controlled**: Automatically sets the element order for the geometry body based on the **Physics Preference**. For sheet bodies and beam models, **Program Controlled** is identical to the **Linear** option described below. For solid bodies and 2-D models, **Program Controlled** is identical to the **Quadratic** option.

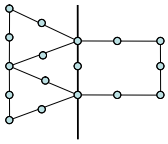
  The selected **Physics Preference** option sets the default for **Program Controlled**, as detailed in the following table:

Physics Preference	Mechanical	Nonlinear Mechanical	Electromagnetics	CFD	Explicit
Program Controlled	Linear for shell Bodies and Quadratic for Solid Bodies	Linear for shell Bodies and Quadratic for Solid Bodies	Linear for shell Bodies and Quadratic for Solid Bodies	N/A	Linear for shell Bodies and Quadratic for Solid Bodies

- **Linear**: Removes the midside nodes on all elements. Example shown below are for a solid body:



- **Quadratic**: Retains midside nodes on elements created in the part or body. All elements have midside nodes.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If **Element Order** is **Quadratic**, and if **Straight Sided Elements** is set to **No**, the mesher places the midside nodes on the geometry so that the mesh elements properly capture the shape of the geometry. However, if the location of a midside node might affect the mesh quality, the midside node may be relaxed to improve the element shape. Therefore, some midside nodes might not follow the shape of the geometry precisely.

For information on how the **Straight Sided Elements** control affects midside nodes, see Straight Sided Elements (p. 180).

Note:

- When **Physics Preference** is **CFD**, **Element Order** is not available.
- **Element Order** supports both linear and quadratic elements for 2D axisymmetric models. This includes support for 2D linear and quadratic inflation layers on axisymmetric parts. You must set **Element Order** to **Linear** to obtain linear elements in such cases.
- **Element Order** supports both linear and quadratic elements for 2D shell and line models. This includes support for 2D linear and quadratic inflation layers on shell models.
- When setting the Method control (p. 202) to a scoped body, you can change the setting of the **Element Order** option to control whether meshes are to be created on the scoped body with midside nodes or without midside nodes. You do this by using the **Element Order** setting under **Definition** in the Method Control Details View. This setting overrides the setting of the global **Element Order** option. For Information refer to Method Controls and Element Order Settings (p. 202).
- **Automatic (PrimeMesh)** uses global Element Order for meshing.

Element Size

Element Size allows you to specify the maximum surface element size used for the entire model. This size will be used for all edge, face, and body meshing. You can parametrize **Element Size**. When you set **Adaptive Sizing** to **Yes**, mesher do not follow the **Element Size**.

- For solids, the **Element Size** is calculated as Bounding Box Diagonal (p. 119) * Bounding Box Factor (p. 392).
- For sheets or solids/sheets, the **Element Size** is calculated as sqrt(Average Surface Area (p. 120)) * Surface Area Factor (p. 392).

Element Size is calculated based on the **Physics Preference** you select. The **Element Size** calculation for each physics preference is as follows:

- When **Physics Preference** is **Mechanical**, **Nonlinear Mechanical**, **CFD** or **Hydrodynamics**,

  Element Size = Assembly Diagonal * Bounding Box Factor

- When **Physics Preference** is **Explicit** or **Electromagnetics**,

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element Size = 0.5* Assembly Diagonal * Bounding Box Factor

Note:

When you set **Physics Preference** as **Explicit**, the default **Max Size** is equal to **Element Size**. That is, the volume max size is automatically set equal to the surface max size to create a uniform mesh.

Sizing Group

Mesh Sizing Defaults (p. 104) describes how default sizes are computed and how the values respond when you modify other sizing values.

The **Sizing** group contents depend on the **Physics Preference** being used.

- When the **Physics Preference** is **Mechanical**, **Electromagnetics**, or **Explicit**, **Use Adaptive Sizing** is set to **Yes** by default.
- When the **Physics Preference** is **Nonlinear Mechanical**, or **CFD**, **Capture Curvature** is set to **Yes** by default.
- When the **Physics Preference** is **Hydrodynamics**, the only properties you can set are Element Size (p. 102) and Defeature Size (p. 111).

With **Use Adaptive Sizing** set to **Yes**, it includes:

- Resolution (p. 109)
- Mesh Defeaturing (p. 110) (Defeature size)
- Transition (p. 112)
- Span Angle Center (p. 112)
- Initial Size Seed (p. 113)
- Enable Washers (p. 117)
- Bounding Box Diagonal (p. 119)
- Average Surface Area (p. 120)
- Minimum Edge Length (p. 120)

With **Use Adaptive Sizing** set to **No**, it includes:

- Growth Rate (p. 110)
- Max Size (p. 110)
- Mesh Defeaturing (p. 110) (Defeature size)
- Capture Curvature (Curvature Min Size (p. 113) and Curvature Normal Angle (p. 114))

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Capture Proximity (Proximity Min Size (p. 115), Proximity Gap Factor (p. 115), and Proximity Size Function Sources (p. 115))
- Enable Washers (p. 117)
- Bounding Box Diagonal (p. 119)
- Average Surface Area (p. 120)
- Minimum Edge Length (p. 120)

Mesh Sizing Defaults

When a model is loaded, the default **Element Size** is automatically set by the software using the physics preference and characteristics about the model.

- If the model only has solid parts, the **Element Size** is set as a factor of the Bounding Box Diagonal (p. 119). When **Use Adaptive Sizing** is set to **Yes**, the factor is determined by using a combination of the Physics Preference (p. 96) and Initial Size Seed (p. 113). For other sizing options, the default **Bounding Box Factor** is set to 0.05, but you can modify this in the **Options** dialog box (**Tools > Options > Meshing > Bounding Box Factor** (p. 392)).
- If the model has sheet parts, the **Element Size** is set as a factor of the average surface area. When **Use Adaptive Sizing** is set to **Yes**, the factor is determined by using a combination of the Physics Preference (p. 96), and Initial Size Seed (p. 113). For other sizing options, the default **Surface Area Factor** is set to 0.125, but you can modify this in the **Options** dialog box (**Tools > Options > Meshing > Surface Area Factor** (p. 392)).

Other default mesh size settings, such as defeature size, curvature minimum size, and proximity minimum size, are set in relation to the **Element Size**. If **Use Adaptive Sizing** is set to **No**, a simple factor is used to scale the defeature size, curvature minimum size, and proximity minimum size.

Beginning in Release 18.2, you can rely on dynamic defaults to scale other sizes off of the element size. When you modify the element size, the other default sizes update dynamically in response, thereby providing a more direct scaling of sizing values.

Dynamic defaults are controlled by the **Mechanical Min Size Factor**, **CFD Min Size Factor**, **Explicit Min Size Factor**, **Mechanical Defeature Size Factor, Explicit Defeature Size Factor** and **CFD Defeature Size Factor** options. These options are available in the Options (p. 389) dialog box. You use these options to set your preferences for the scale factors (p. 392) that will be used to calculate the corresponding default sizes. Essentially, the scale factors control the default values for global minimum size and global defeature size, as well as the default sizes used by local mesh sizing controls when **Type** is set to Factor of Global Size (p. 306).

Using Dynamic Mesh Sizing Defaults

To use dynamic defaults:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Retain the default settings of **Defeature Size**, **Curvature Min Size**, **Proximity Min Size**, and/or **Max Size**.

   Note:

   If you modify any of these default sizes, their values will not update dynamically. This is because user-defined sizes are always retained.

2. Adjust the **Element Size**.

   In response, the other default global mesh sizes will update dynamically.

3. (Optional) For more control over global size scaling, modify the values of the **Mechanical Min Size Factor**, **CFD Min Size Factor**, **Mechanical Defeature Size Factor** and/or **CFD Defeature Size Factor** options (p. 392) to obtain the desired scaling.

4. (Optional) To control scaling locally, choose Factor of Global Size (p. 306) as the **Type** when you define a local sizing control. Then, depending on the entities and other options you select for the control, use the **Defeature Size Scale**, **Curvature Min Size Scale**, and **Proximity Min Size Scale** local sizing control options (p. 302) to obtain local sizing based on the global sizing.

   Note:

   By setting **Type** to **Factor of Global Size**, you ensure the local sizes will update dynamically if you change the global element size.

Sizing Options

The **Sizing** options provide greater control over the following properties:

- Mesh growth (transition) between small and large sizes based on a specified growth rate
- Curvature based refinement and angles between normals for adjacent mesh elements (curvature (p. 107)-based sizing)
- Number of mesh elements employed in the gaps between two geometric entities (proximity (p. 107)-based sizing)

By default, **Use Adaptive Sizing** is set to **Yes**, unless Physics Preference (p. 96) is set to **CFD**, **Explicit** or **Nonlinear Mechanical** (in which case the default is **Capture Curvature** set to **Yes**), or **Hydrodynamics** (in which case the sizing is **Uniform**). The option you choose determines which refinement mechanisms are activated, as described in the following sections.

Controls for sizing include **Element Size** (p. 102), **Growth Rate** (p. 110), **Max Size** (p. 110), and **Curvature Min Size** (p. 113)/ **Proximity Min Size** (p. 115). The **Curvature Min Size**/**Proximity Min Size** and **Max Size** specifications represent, respectively, the global minimum and global maximum allowable element size. The **Element Size** specification represents the global maximum allowable size of the elements created by the free surface meshers of the supported methods. The **Growth**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rate represents the increase in element edge length with each succeeding layer of elements from the edge or face. For example, a growth rate of 1.2 results in a 20% increase in element edge length with each succeeding layer of elements.

Sizing options are applicable to the following mesh methods:

Volume Meshing:

- General Sweep (p. 399) (Surface mesh only)
- Thin Sweep (p. 406) (Surface mesh only)
- Hex Dominant (p. 233) (Surface mesh only)
- Patch Conforming Tetrahedron (p. 208)
- Patch Independent Tetrahedron (p. 210)
- MultiZone (p. 242)

Note:

- The sizing controls are passed to the Patch Independent Tetrahedron method. That is, the Patch Independent Tetrahedron method does not *use* the same sizing algorithms; rather, this method *interprets* the controls and uses internal algorithms to respect the user-defined settings.
- The General Sweep, Thin Sweep, and Hex Dominant mesh methods use the sizing controls in the creation of the surface mesh, but the volume mesh does not use them.

Surface Meshing:

- Quad Dominant (p. 282)
- All Triangles (p. 284)
- MultiZone Quad/Tri (p. 284)
- Automatic (PrimeMesh) (p. 287)

Note:

For an overview of how using the sizing influences mesh distribution, refer to Understanding the Influence of the Sizing Options (p. 89).

Sizing topics include:

- Curvature-Based Sizing (p. 107)
- Proximity-Based Sizing (p. 107)
- Uniform Sizing (p. 108)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Curvature-Based Sizing

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

Curvature-based sizing is defined by the following properties:

- **Curvature Normal Angle** (p. 114)
- **Curvature Min Size** (p. 113)
- **Element Size** (p. 102)
- **Max Size** (p. 110)
- **Growth Rate** (p. 110)

The **Curvature Normal Angle** (p. 114) is the maximum allowable angle that one element edge is allowed to span given a certain geometry curvature.

Proximity-Based Sizing

You can specify the minimum number of element layers created in regions that constitute "gaps" in the model for proximity-based sizing. For the purposes of specifying proximity-based sizing, a "gap" is defined in one of two ways:

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

Proximity-based sizing is defined by the following properties:

- **Proximity Min Size** (p. 115)
- **Proximity Gap Factor** (p. 115)
- **Proximity Size Function Sources** (p. 115)
- **Element Size** (p. 102)
- **Max Size** (p. 110)
- **Growth Rate** (p. 110)

The **Proximity Gap Factor** (p. 115) is the number of layers of elements to be generated in the gaps. **Proximity Size Function Sources** (p. 115) determines which regions of proximity are considered in calculations.

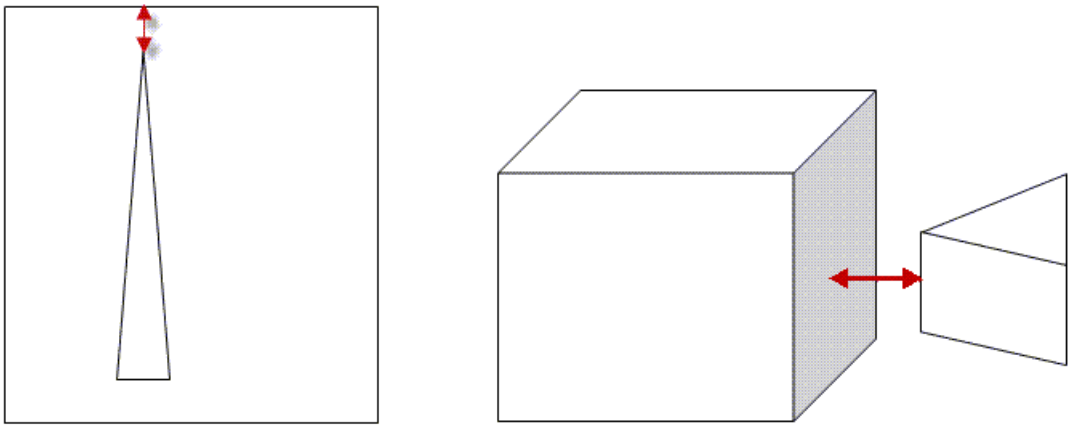
Note:

The proximity sizing does not recognize the proximity between certain entities in cases involving voids in a model. For example, the proximity between a vertex and an edge on

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a face in 2D (below left) is ignored if the triangle is a void in the rectangle. Similarly, the proximity between a vertex or edge and a face in 3D (below right) is ignored if the prism and the block are voids in a larger domain. The two-headed arrows in the figure indicate the areas where proximity is ignored.

Figure 30: Proximity Sizing Limitation



Uniform Sizing

Uniform sizing does not refine the mesh based on curvature or proximity. Rather, you specify defeature and maximum sizes and gradation is provided between sizes based on a specified growth rate.

Uniform sizing is defined by the following properties:

- **Defeature Size** (p. 111)
- **Element Size** (p. 102)
- **Max Size** (p. 110)
- **Growth Rate** (p. 110)

With uniform sizing, you must use size controls to control mesh sizes locally, as opposed to the curvature and proximity sizing, which refine the sizes locally based on curvature and proximity of features in the geometry. Even if the specified local sizes are **Soft** (p. 310) sizes, they may override the global sizes when the uniform sizing is being used.

Note:

The **Use Uniform Size Function for Sheets** (p. 116) enables you to use uniform sizing while you are using other sizing options to refine the mesh for the rest of the model. This setting is available only when the model contains mixed body types (for example, sheet and solid bodies), and when **Capture Curvature** and/or **Capture Proximity** is set to **Yes**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Sizing Options

By default, **Use Adaptive Sizing** is set to **Yes**, unless Physics Preference (p. 96) is set to **CFD** or **Nonlinear Mechanical** (in which case the default is **Capture Curvature** set to **Yes**), or **Hydrodynamics** (in which case the sizing is uniform). The option you choose determines which refinement mechanisms are activated.

The following options are available:

Resolution
Growth Rate
Max Size
Mesh Defeaturing
Transition
Span Angle Center
Initial Size Seed
Curvature Min Size
Curvature Normal Angle
Proximity Min Size
Proximity Gap Factor
Proximity Size Function Sources
Use Uniform Size Function for Sheets
Enable Washers
Bounding Box Diagonal
Average Surface Area
Minimum Edge Length

Resolution

Available when **Use Adaptive Sizing** is set to **Yes**, the **Resolution** option controls the mesh distribution. The default setting is **Program Controlled** (see **Adaptive Resolution** in the **Options** dialog box (p. 392)). The range of values that can be set is 0 to 7, with the mesh resolution changing from coarse (**0**) to fine (**7**).

A value of **-1** will set the **Resolution** to the default value set for **Adaptive Resolution** in the **Options** dialog box (p. 392).

- If **Adaptive Resolution** is set to **Program Controlled**, the default value is **4** when the Physics Preference (p. 96) is **Electromagnetics** or **Explicit**. For all other Physics Preference (p. 96), the default value is **2**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If **Adaptive Resolution** is explicitly set in the **Options** dialog box (p. 392) (**0**-**7**), the value set will be used as default.

Note:

- If you set the **Resolution** explicitly, the set value will be retained.
- For databases from previous releases (prior to Release 2025 R1), the **Resolution** will be explicitly set to a value which is **not** the default.
- For new databases, if **Resolution** is set to **Default** and the **Adaptive Resolution** in the **Options** dialog box (p. 392) is different than it was when the database was saved, the new value will be used instead.

Growth Rate

Growth Rate represents the increase in element edge length 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. This option is available when **Use Adaptive Sizing** is set to **No**. Specify a value from 1 to 5 or accept the default.

The default is calculated based on the Physics Preference (p. 96) option and the presence of sheet bodies.

Note:

For sheet models, the Details View does not display the default value of **Growth Rate**.

Max Size

Max Size is the maximum size that the sizing controls will return to the mesher. This option is available when **Use Adaptive Sizing** is set to **No**. Specify a value greater than 0 or accept the default.

Note:

- The **Max Size** option is hidden if there are no solids in your model.
- For information about overriding sizes, refer to Overriding Sizing Minimum and Maximum Sizes (p. 91).
- For information about the relationship between the **Element Size** control and the **Min Size**, and **Max Size** controls, refer to Understanding the Influence of the Sizing Options (p. 89).

Mesh Defeaturing

The Meshing application automatically defeatures small features and dirty geometry according to the **Defeature Size** you specify here.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh defeaturing is supported for the following mesh methods:

Solid Meshing:

- Patch Conforming Tetrahedron (p. 208)
- Patch Independent Tetrahedron (p. 210)
- MultiZone (p. 242)
- Thin Sweep (p. 406)
- Hex Dominant (p. 233)

Surface Meshing:

- Quad Dominant (p. 282)
- All Triangles (p. 284)
- MultiZone Quad/Tri (p. 284)
- Automatic (PrimeMesh) (p. 287)

For the Patch Independent Tetrahedron (p. 210), MultiZone (p. 242), and MultiZone Quad/Tri (p. 284) methods, the **Defeature Size** you set here will be populated to the local (scoped) method controls. If you subsequently make changes to the local settings, the local settings will override the global **Defeature Size** set here. See the descriptions of the individual methods for more information.

See Protecting Topology Defined Prior to Meshing (p. 184) for details on protecting defeatured topology.

The options for defining mesh defeaturing are described below.

Mesh Defeaturing

Use the **Mesh Defeaturing** control to enable and disable defeaturing. When **Mesh Defeaturing** is **Yes** (default), features smaller than or equal to the value of **Defeature Size** are removed automatically.

Defeature Size

Defeature Size is available only when **Mesh Defeaturing** is set to **Yes**. Specify a positive value to set the global tolerance for defeaturing, or accept the default. Specifying a value of **0.0** resets the **Defeature Size** to its default, which is determined by the **Mechanical Defeature Size Factor** or **CFD Defeature Size Factor** preference (p. 392) specified in the Options dialog box. If you retain the

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

default **Defeature Size** and subsequently modify the **Element Size** (p. 102), the default **Defeature Size** is recalculated to reflect the size factor preferences.

Note:

- If a user-defined value has been specified for **Defeature Size**, that value will be used for everything—sheets and solids. In addition, the value will not update dynamically if you modify the **Element Size** because user-defined values are always retained.
- If you allow the **Defeature Size** to default and you modify the mesh size by applying local sizing controls (for example, local face or edge sizing), the tolerance may be modified automatically. In such cases, a warning message is issued to notify you that the global defeature size has been modified. To prevent this behavior, manually specify a value for **Defeature Size**.
- Some bodies or parts might be ignored if the **Defeature Size** is too high. If this happens, the body will be considered to be meshed but will have no elements. These bodies or parts can be suppressed if desired. To capture these bodies or parts in the mesh, regenerate the mesh with a reduced **Defeature Size**.
- If the geometry contains close vertices, setting the **Defeature Size** can resolve them.

  On the **Graphics Options** toolbar, you can click the **Close Vertices** button to determine if there are any close vertices in the model, and then set the **Defeature Size** to a value that will sufficiently defeature any extra vertices.
- You can also set the **Defeature Size** on a body, face, or edge by inserting a local size control (p. 300).

Transition

When **Use Adaptive Sizing** is set to **Yes**, **Transition** affects the rate at which adjacent elements will grow. **Slow** produces smooth transitions while **Fast** produces more abrupt transitions.

Span Angle Center

When **Use Adaptive Sizing** is set to **Yes**, **Span Angle Center** sets the goal for curvature-based refinement. For a curved region, the mesh will subdivide along the curvature until the individual elements span this angle.

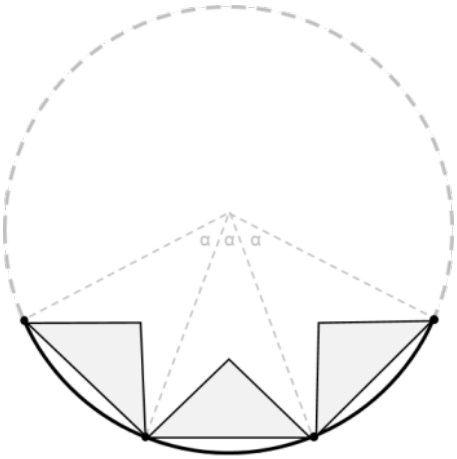
You can set the span angle by choosing one of the following options:

- **Coarse** - 91^{o} to 60^{o}
- **Medium** - 75^{o} to 24^{o}
- **Fine** - 36^{o} to 12^{o}

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example 1: How the Span Angle Center Controls Element Placement

The following figure shows a curved region. If the curvature were extended to 360° to create a circle, the Curvature Normal Angle is the arc span from the center of the circle to the curved region. The Curvature Normal Angle ($\alpha = 45^{o}$) is computed based on the Span Angle Center setting.



Initial Size Seed

When **Use Adaptive Sizing** is set to **Yes**, **Initial Size Seed** allows you to control the initial seeding of the mesh size for each part.

- **Assembly** (default) bases the initial seeding on the diagonal of the bounding box that encloses all assembly parts regardless of the number of suppressed parts. As a result, the mesh never changes due to part suppression.
- **Part** bases the initial seeding on the diagonal of the bounding box that encloses each particular individual part as it is meshed. The mesh never changes due to part suppression. This option typically leads to a finer mesh and is recommended for situations where the fineness of an individual part mesh is important relative to the overall size of the part in the assembly.

Note:

When **Use Adaptive Sizing** is set to **No**, **Assembly** is set internally and cannot be modified.

Curvature Min Size

Available when **Capture Curvature** (p. 107) is set to **Yes**, **Curvature Min Size** is the minimum size returned to the mesher. Some element sizes may be smaller than this based on local feature sizes or other geometric anomalies.

Specify a positive value or accept the default, which is determined by the size factor preferences (p. 392) (**Mechanical Min Size Factor**, **Explicit Min Size Factor** and/or **CFD Min Size Factor**) specified in the Options dialog box. If you retain the default **Curvature Min Size** and subsequently

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

modify the **Element Size** (p. 102), the default **Curvature Min Size** is recalculated to reflect the size factor preferences.

Note:

- The appropriate size factor is based on the physics preference. The **Mechanical Min Size Factor** applies when physics preference is **Mechanical**, **Electromagnetics**. The **CFD Min Size Factor** applies when physics preference is **CFD**. The **Explicit Min Size Factor** applies when the physics preference is **Explicit**.
- When **Capture Proximity** (p. 107) is also set to **Yes**, you can specify a global **Proximity Min Size** to be used in proximity calculations in addition to the global **Curvature Min Size**. When only **Capture Proximity** is selected, only **Proximity Min Size** is available. Refer to **Proximity Min Size** (p. 115) for details.
- For information about overriding sizes, refer to Overriding Sizing Minimum and Maximum Sizes (p. 91).
- For information about the relationship between the **Element Size** control and the **Max Size** controls, refer to Understanding the Influence of the Sizing Options (p. 89).

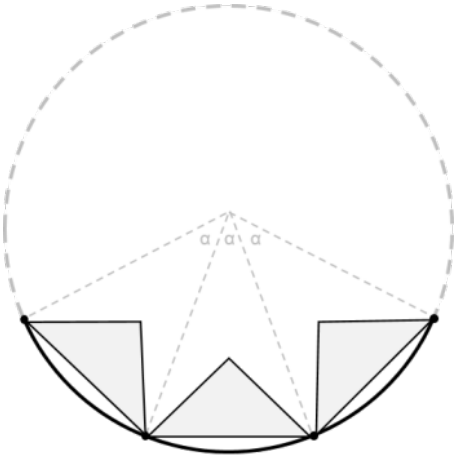
Curvature Normal Angle

Curvature Normal Angle is the maximum allowable angle that one element edge is allowed to span given a particular geometry curvature. This option is available only when **Capture Curvature** is set to **Yes**. You can specify a value from 0 to 180 degrees or accept the default. (A value of 0 resets the option to its default.) The default value depends on the Physics Preference (p. 96) selected.

You can use the **Curvature Normal Angle** to limit the number of elements that are generated along a curve or surface if the Curvature Min Size (p. 113) is too small for that particular curve.

Example 2: How the Curvature Normal Angle Controls Element Placement

The following figure shows a curved region with a Curvature Normal Angle of $\alpha = 45^{o}$. If the curvature were extended to 360^{o} to create a circle, the Curvature Normal Angle is the arc span from the center of the circle to the curved region. Three arc spans are needed to span the entire curvature, which results in three elements placed along the curvature: one in each arc span.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Proximity Min Size

Available when **Capture Proximity** (p. 107) is set to **Yes**, **Proximity Min Size** enables you to specify a global minimum size to be used in proximity sizing calculations. Specify a positive value or accept the default, which is set equal to the default of **Curvature Min Size**.

When only **Capture Proximity** is selected, only **Proximity Min Size** is available.

Some element sizes may be smaller than this based on local feature sizes or other geometric anomalies.

Proximity Gap Factor

Proximity Gap Factor is the minimum number of layers of elements to be generated in the gaps. This option is available only when **Capture Proximity** is set to **Yes**. You can specify a value from 0 to 100, or accept the default (3).

Remember the following information:

- The value of **Proximity Gap Factor** is an estimate, it may not be exactly satisfied in every gap. When using mapped Face Meshing controls (p. 314) or sweeping (p. 399), interval assignment may change the number of divisions (elements or cells) in a gap.
- For thin regions, to control the aspect ratio of the cells in the gap, you can use **Proximity Gap Factor** less than 1. For example, if you want to refine the surface mesh to allow elements with an aspect ratio of 4 in the gap, you can use a **Proximity Gap Factor** of 0.25
- In cases involving Patch Conforming tetra (p. 208) meshing and Swept (p. 399) meshing, the proximity sizing drives the surface mesh size distribution as follows. The value of **Proximity Gap Factor** is applicable to both 3D proximity (the number of 3D elements/cells between two faces in a body) and 2D proximity (the number of 2D elements/cells between two edges on a face), and the global **Growth Rate** (p. 110) value is automatically taken into account in the gap. However, the 3D proximity sizing affects only the surface mesh in the gap, and assumes the volume mesh will use the global settings. Hence, if you define local mesh sizing on a body and specify local **Element Size** (p. 303) or local **Growth Rate** (p. 309) settings that differ drastically from the global sizing settings (or if inflation is specified), the final number of cells across a 3D gap may deviate from the specified **Proximity Gap Factor** value.
- When **Method** (p. 202) is set to Automatic (p. 206), the proximity calculation in swept bodies may result in an under-refined mesh. You can use inflation (p. 345) layers or a scoped fixed size at such locations to produce a sufficient number of elements/cells.

Proximity Size Function Sources

Proximity Size Function Sources determines whether regions of proximity between faces and/or edges are considered when proximity size function calculations are performed. This option is available only when **Capture Proximity** is set to **Yes**. You can specify **Edges**, **Faces**, or **Faces and Edges**:

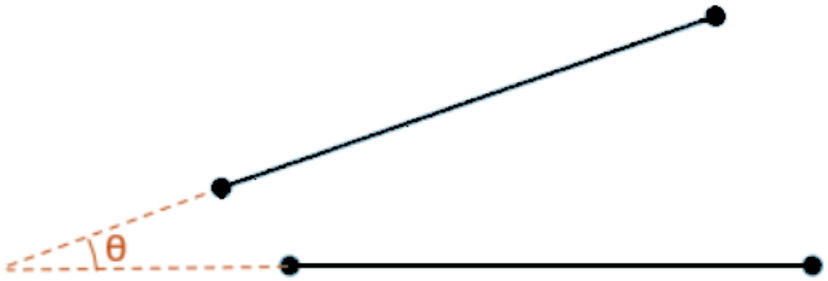
- **Faces** - Considers face-face proximity between faces. Face-edge and edge-edge proximity are not considered (that is, the trailing edge of fluid around wings will not be captured with this setting).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Faces and Edges** - This is the default for part/body-based meshing methods. Considers face-face and edge-edge proximity. Face-edge proximity is not considered.

Note:

- In cases involving face-face proximity, the face normal orientation is ignored during the proximity calculation.
- It is important to resolve all edges as much as possible for better feature capturing and for minimizing the occurrence of non-manifold nodes. For this reason, you should specify a setting of either **Edges** or **Faces and Edges**. For many models, the **Edges** setting may be sufficient to resolve all proximity situations. For large complex models, using either the **Faces and Edges** or **Faces** setting may result in longer computation time.
- When **Capture Proximity** is **Yes**, a critical angle of 30 degrees is considered by default for proximity calculation. Hence, the mesher ignores any face pair having a critical angle above 30 degrees for proximity calculation.



Here, the angle between the faces should be less than 30 degrees for computing the proximity of the faces.

Use Uniform Size Function for Sheets

Use Uniform Size Function for Sheets enables you to use the **Uniform** sizing while you are using other options to refine the mesh for the rest of the model. This setting is available only when the model contains mixed body types (for example, sheet and solid bodies), and when **Capture Curvature** and/or **Capture Proximity** is set to **Yes**. The default for this setting is **No**.

Note:

When the **Physics Preference** is **Explicit**, **Use Uniform Size Function for Sheets** is set to **Yes** by default when both solid and sheet bodies are present.

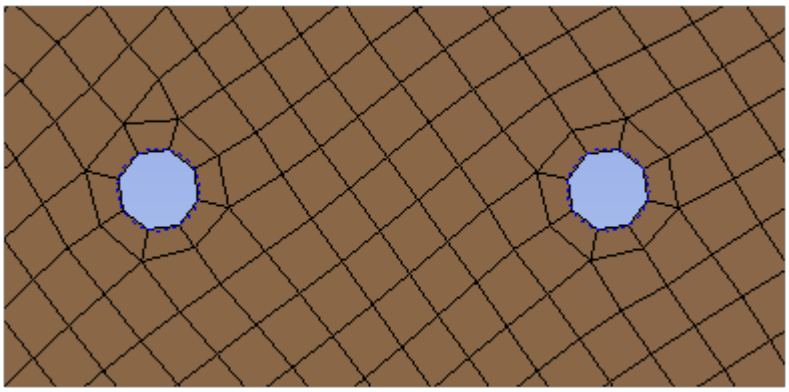
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enable Washers

Enable Washers controls the pattern of the mesh that is generated around any holes in a sheet body. If you set this option to **Yes**, the mesh will be generated with a layer of equally-spaced quadratic elements—called washers—around each hole.

The **Enable Washers** option is only available for sheet bodies when **Use Adaptive Sizing**, **Capture Curvature**, and **Capture Proximity** are all set to **No**. If the model contains both solid and sheet bodies, **Enable Washers** is available, but washers will only be generated for the sheet bodies.

Figure 31: Washers Generated Around Two Holes



If you enable washers, you can specify the height of the washers and whether the washer element nodes should be moved for holes that are close to a boundary. You should be aware of the scenarios in which a washer might not be generated for a hole.

Height of Washer

Allow Nodes to be Moved off Boundary

Limitations for Washers

Height of Washer

Height of Washer controls the height of the washer elements to be generated around each hole. By default, the height is set to the **Defeature Size** (p. 111).

You should ensure that the height of the washer is set to a value between the target minimum element size in the model and the **Element Size** (p. 102) (the nominal uniform element size expected in the overall model).

Note:

The mesher attempts to respect the minimum size and **Element Size** when the mesh is generated. However, you may notice variations in the final mesh due to local feature sizes, transition areas, or other geometric anomalies.

Allow Nodes to be Moved off Boundary

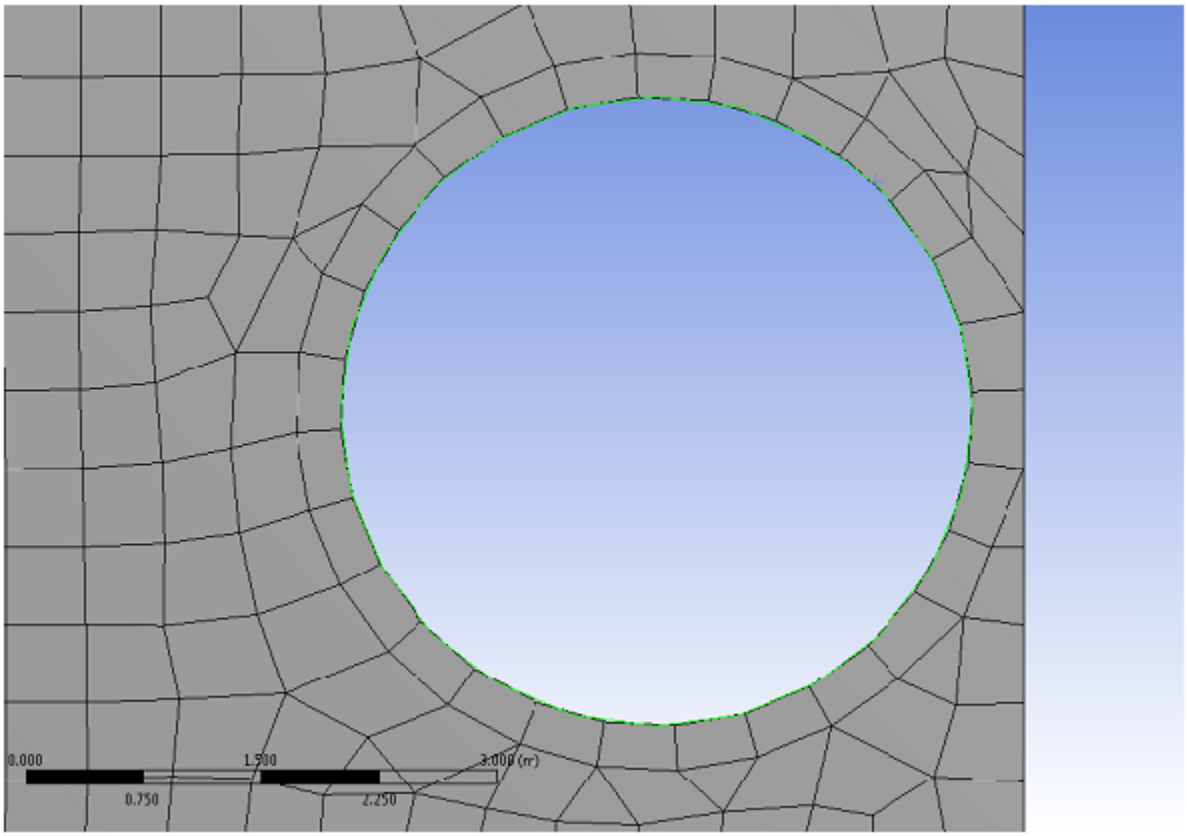
Allow Nodes to be Moved off Boundary controls the placement of the nodes for holes that are too close to a boundary to place the washer elements. If you select **Yes**, the nodes will be moved

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

such that the washer elements can be placed without compromising any boundaries. The default value for **Tolerance for Moving Nodes** is 10% of the value specified for **Height of Washer**. If you select **No**, the mesher will not generate any washer elements in any locations in which the elements do not fit.

The following figure shows a hole that is too close to an edge for the washer elements to be generated between the hole and the edge.

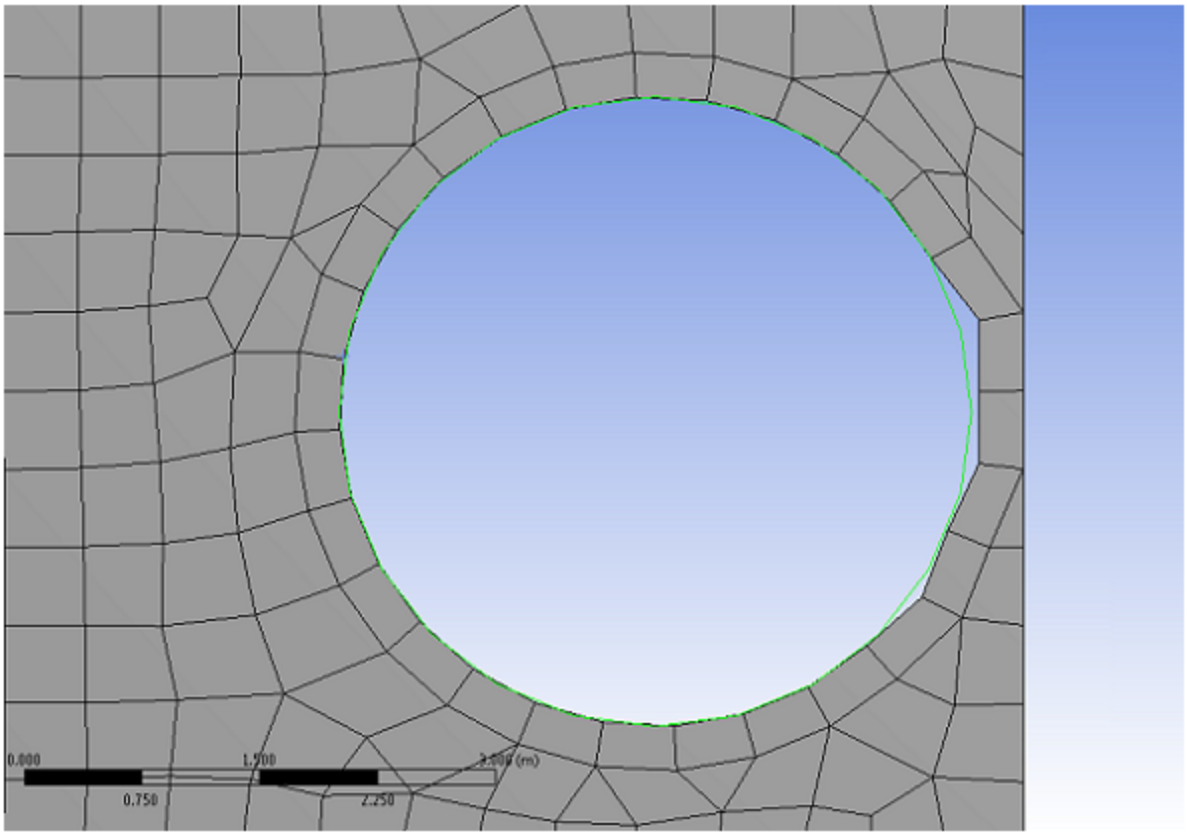
Figure 32: Washer Element Nodes Not Moved



After setting **Allow Nodes to be Moved off Boundary** to **Yes**, the washer elements are now generated by slightly moving some of the nodes away from the edge of the hole.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 33: Washer Element Nodes Moved



Limitations for Washers

The following table describes the scenarios in which a washer might not be generated for a hole:

Table 1: Washer Limitations

Scenario	Description
A hole is shared by two faces	The hole is not detected, and a washer is not generated around the hole. To ensure that a washer is generated, you should merge the faces (p. 546).
A local mesh control (p. 201) is set	Washers are not generated on any face to which a local mesh control has been applied.
The **Defeature Size** (p. 111) is greater than the **Minimum Edge Length**	Washers might not be generated.
A hole has a diameter less than the **Minimum Edge Length**	A washer will not be generated for the hole.
The **Min Size** and **Element Size** (p. 102) are set to the same value	Washers might not be generated.

Bounding Box Diagonal

Bounding Box Diagonal provides a read-only indication of the length of the assembly diagonal.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Average Surface Area

Average Surface Area provides a read-only indication of the average surface area of the model.

Minimum Edge Length

Minimum Edge Length provides a read-only indication of the smallest edge length in the model.

When you click , **Show Minimum Edge Lengths** option is displayed. **Show Minimum Edge Length** helps you to find and display the smallest edges in the model. The smallest edges may help you determine the defeature size when defeaturing is required to refine the mesh to resolve the feature. For example, **Curvature Min Size** (or Local Size Control).

Quality Group

The Mesh Quality Workflow (p. 120) describes the steps for configuring mesh quality. The **Quality** group includes:

- Check Mesh Quality (p. 121)
- Error and Warning Limits (p. 127)
- Target Element Quality (p. 130)
- Target Skewness (p. 131)
- Target Jacobian Ratio (Corner Nodes) (p. 131)
- Smoothing (p. 133)
- Mesh Metric (p. 133)

The Mesh Quality Workflow

The general workflow for configuring mesh quality is as follows:

1. Set Physics Preference (p. 96).
2. (Optional) If **Physics Preference** is **Mechanical**, set Error Limits (p. 127).
3. Set Check Mesh Quality (p. 121) depending on how you want the mesher to respond when elements reach error or warning limits.
4. Set appropriate quality targets based on your physics preference:
   - Target Quality (p. 130)
   - Target Skewness (p. 131)
   - Target Jacobian Ratio (Corner Nodes) (p. 131)
5. Generate the mesh. (p. 528)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Review warning or error messages. Use Show Elements (p. 538) if needed.
7. Review mesh metrics (p. 133).
8. Perform troubleshooting (p. 579) if needed.

Check Mesh Quality

Check Mesh Quality determines how the software behaves with respect to error and warning limits (p. 127):

- **Yes, Errors** (default) - If the meshing algorithm cannot generate a mesh that passes all error limits, an error message is printed and meshing fails.
- **Yes, Errors and Warnings** - If the meshing algorithm cannot generate a mesh that passes all error limits, an error message is printed and meshing fails. In addition, if the meshing algorithm cannot generate a mesh that passes all warning (target) limits, a warning message (p. 538) is printed.
- **No** - Mesh quality checks are done at various stages of the meshing process (for example, after surface meshing prior to volume meshing). The **No** setting turns off most quality checks, but some minimal checking is still done. In addition, even with the **No** setting, the target quality metrics are still used to improve the mesh. The **No** setting is intended for troubleshooting (p. 579) and should be used with caution as it could lead to solver failures or incorrect solution results.
- **Mesh Quality Worksheet**: Measures the mesh quality and allows you to set warning and error limits for each mesh quality criterion. When you select **Mesh Quality Worksheet**, the **Display Style** contours are modified to display the elements that exceed the warning and error limits.

Note:

Changing the **Check Mesh Quality** setting after you have meshed affects the mesh status as follows:

– If, after meshing, you change the **Check Mesh Quality** setting from **No** to **Yes, Errors** or **Yes, Errors and Warnings**, or you change the setting from **Yes, Errors** to **Yes, Errors and Warnings**, the mesh becomes out-of-date and requires action.

– If, after meshing, you change the **Check Mesh Quality** setting from **Yes, Errors and Warnings** to another setting, or you change the setting from **Yes, Errors** to **No**, the mesh will not go out-of-date. However, some old messages may not directly apply anymore.

– If, after meshing, you change the **Check Mesh Quality** setting from **No** or **Yes, Errors** or **Yes, Errors and Warnings** to **Mesh Quality Worksheet**, it validates the generated mesh.

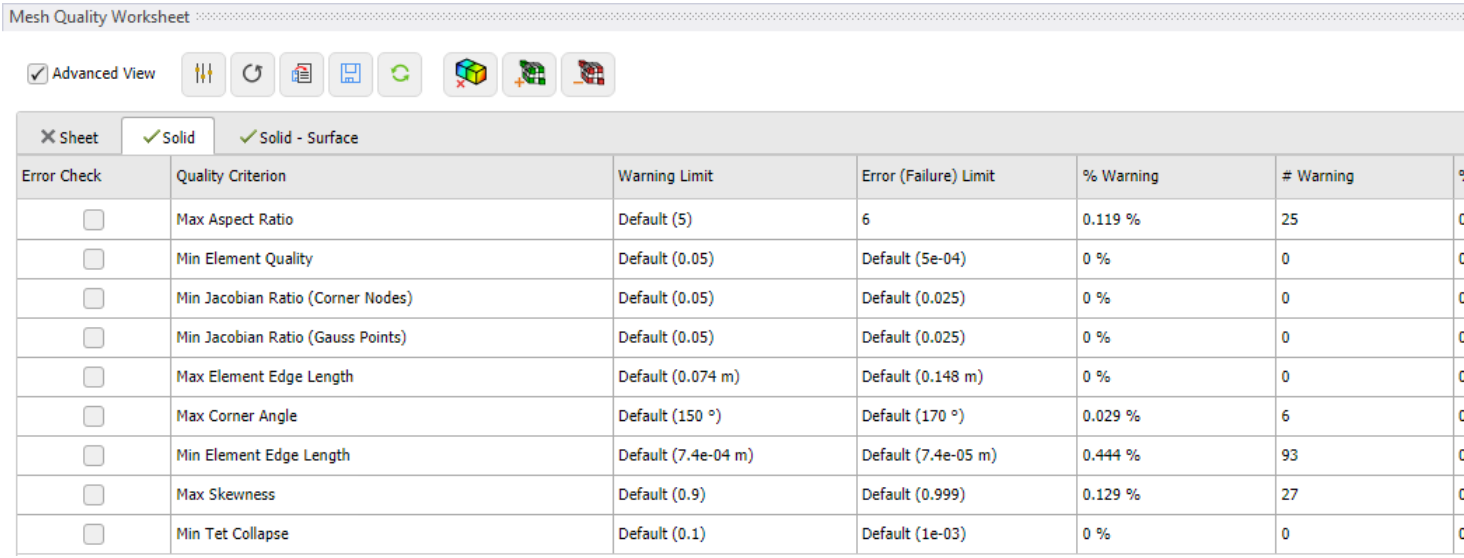
Mesh Quality Worksheet

Mesh Quality Worksheet helps you to measure the quality of mesh, provide visualization tools for finding areas of low mesh quality and improve the mesh. The **Mesh Quality Worksheet** displays the quality criteria and its default warning and error limit based on the selected **Physics Preference**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you change the **Physics Preference**, the worksheet metrics column gets reset to the default values. To access **Mesh Quality Worksheet**, You can click **Mesh** on the Tree Outline and click **Quality Worksheet** in the **Mesh Context** tab on the ribbon. **Mesh Quality Worksheet** is available for **No** or **Yes, Errors** or **Yes, Errors and Warnings** and **Mesh Quality Worksheet** in **Check Mesh Quality**.

Mesh Quality Worksheet options are as follows:



Error Check	Quality Criterion	Warning Limit	Error (Failure) Limit	% Warning	# Warning
☐	Max Aspect Ratio	Default (5)	6	0.119 %	25
☐	Min Element Quality	Default (0.05)	Default (5e-04)	0 %	0
☐	Min Jacobian Ratio (Corner Nodes)	Default (0.05)	Default (0.025)	0 %	0
☐	Min Jacobian Ratio (Gauss Points)	Default (0.05)	Default (0.025)	0 %	0
☐	Max Element Edge Length	Default (0.074 m)	Default (0.148 m)	0 %	0
☐	Max Corner Angle	Default (150 °)	Default (170 °)	0.029 %	6
☐	Min Element Edge Length	Default (7.4e-04 m)	Default (7.4e-05 m)	0.444 %	93
☐	Max Skewness	Default (0.9)	Default (0.999)	0.129 %	27
☐	Min Tet Collapse	Default (0.1)	Default (1e-03)	0 %	0

Advanced View: Allows you to have a detailed view of the quality metrics. It displays the **%Warning**, **#Warning**, **%Failed**, **#Failed** and **Average** columns when enabled.

: Open the **Options** window to set the mesh metric visibility for the selected quality criterion in the **Mesh Quality Worksheet** . you can use right-click options to select single or multiple quality criterion. The available right-click options are:

- **Physics Preference**: Sets the default value for quality criterion based on the **Physics Preference** in the Details view.
- **On**: Enables the quality criterion visibility in the **Mesh Quality Worksheet** for the selected mesh metric.
- **Off**: Disables the quality criterion visibility in the **Mesh Quality Worksheet** for the selected mesh metric.

: Allows you to reset the worksheet warning and error limit values to default ones.

: Allows you to load or resume the saved mesh quality worksheet by reading a *.csv file.

: Allows you to save the mesh quality worksheet in *.csv format.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

: Allow you to change the errors or warning limits and recompute the failed or warning elements without invalidating the mesh.

When you click **Show Contours**, the available options are:

: Allows you to switch off the metrics contour and reset the **Display Style** to **Use Geometry Setting**.

: Allows you to increase the number of mesh layers for the selected quality criterion. It is applicable only when **Check Mesh Quality** is set to **Mesh Quality Worksheet**.

: Allows you to decrease the number of mesh layers for the selected quality criterion. It is applicable only when **Check Mesh Quality** is set to **Mesh Quality Worksheet**.

: Allows you to switch the contour from **Classic Contours** to **Warning/Error Contours**.

: Allows you to switch the contour from **Warning/Error Contours** to **Classic Contours**.

Sheet: Displays the mesh quality metrics worksheet for sheet bodies.

Solid: Displays the mesh quality metrics worksheet for solid bodies.

Solid - Surface: Displays the mesh quality metrics worksheet for exterior face elements of solid bodies. You can use **Preview Surface Mesh** and view the surface mesh quality in the worksheet for solid bodies before proceeding to volume meshing if required.

Note:

In **Mesh Quality Worksheet,** when you use **Preview Surface Mesh**, you can view the **Mesh Metric** only from the **Solid - Surface** tab.

Sheet, **Solid**, **Solid-Surface** tab have the following states:

A cross mark × denotes that you cannot compute the mesh metrics. For example, if there is cross mark on the sheet tab, it defines that there is no sheet bodies in the geometry.

A refresh mark denotes you need to compute the mesh metrics.

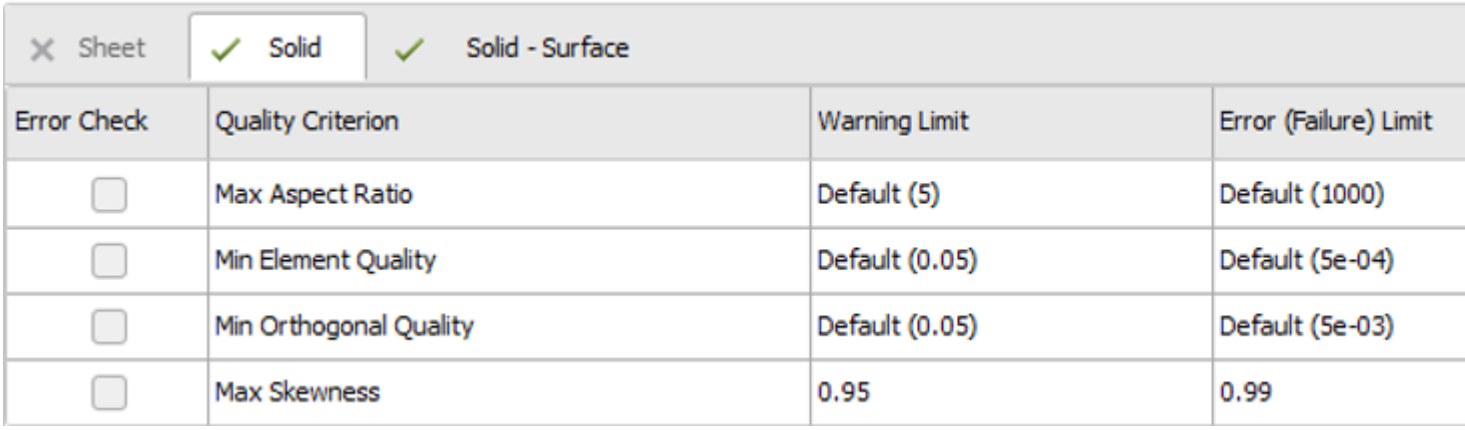
A tick mark ✓ denotes the mesh metrics of a mesh are up-to-date.

The columns in **Mesh Quality Worksheet** are as follows:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Error Check**: Allows you to activate or deactivate error check for a given quality criterion. When **Error Check** is enabled, the mesh becomes invalidated if the error limits for the quantity are exceeded upon mesh completion. If you have strict quality requirements, **Error Check** can be used to avoid mesh continuation to solver. For example, automated design of experiment studies in the Workbench. **Error Check** is only available when **Check Mesh Quality** is set to **Mesh Quality Worksheet**.
- **Quality Criterion**: Displays quality mesh metric for each row. This column is read-only.
- **Warning (Target) Limit**: Allows you to specify the warning limit constraints for all elements of the quality metric.
- **Error (Failure) Limit**: Allows you to specify the error limit constraints for all elements of the quality metric.
- **% Warning**: The percentage of elements that exceed the specified warning limit.
- **# Warning**: The number of elements that exceed the specified warning limit.
- **% Failed**: The percentage of elements that exceed the error limit.
- **# Failed**: The number of elements that exceed the error limit.
- **Average**: Displays average of quality mesh metric for each row.
- **Worst**: Displays the metric value for the worst quality element for each metric and provides color coding for the quality metric displayed. The green color indicates that no elements exceed the **Warning Limit**. Yellow color highlights indicates that bad elements exceed **Warning Limit** but not **Error Limit** and red color indicates that the worst element(s) exceed the **Error limit**. For some metrics, a high value is considered as bad. For example, Skewness. For other metrics, a low value is considered bad. For example, tet collapse. See Mesh Metrics (p. 133) for more details on quality metrics. In the below image, you can see that **Min Element Quality** is green (good), **Max Aspect Ratio** and **Min Orthogonal Quality** are yellow (warning) and **Max Skewness** is red (Error).

If **Error Check** was selected for the **Max Skewness**, after completion the mesh becomes invalid.



You can customise the quality criteria as per your preference under **Mesh Quality** in **File > Options > Meshing**. The default quality criteria displayed in the worksheet is based on **Physics Preference** and differs for **Sheet**, **Solid** and **Solid - Surface**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you right-click a **Quality Criterion** in the **Mesh Quality Worksheet**, you have the following options:

- **Create NS Group: Failed Elements**: Generates element based named selection groups to view the failed elements in the mesh when you select the named selection group in the tree. **Create NS Group: Failed Elements** is available in **Solid** and **Sheet** tab.
- **Create NS Group: Warning Elements**: Generates element based named selection groups to view the warning elements in the mesh when you select the named selection group in the tree. **Create NS Group: Warning Elements** is available in **Solid** and **Sheet** tab.
- **Show Contours**: Sets the **Display style** to selected quality criterion when **Check Mesh Quality** is **No**, or **Yes, Errors and Warnings**, or **Yes, Errors**.

  Show Contours sets the **Display Style** to selected quality criterion and displays the warning (yellow) and failed (red) elements for the selected quality criterion when **Check Mesh Quality** is set to **Mesh Quality Worksheet**. **Show Contours** is available in **Sheet**, **Solid**, **Solid-Surface** tabs. There are two contours available in **Mesh Quality Worksheet**:

  – **Classic Contours**: Displays the contours banded across a range from red to blue in the geometry window. Red indicates poor quality elements and blue indicates good quality elements.

  – **Warning/Error Contours**: Displays the contours banded with three colors in the geometry window. **Warning/Error Contours** is based on the warning and error limits defined in the **Mesh Quality Worksheet**. Red indicates elements that exceed the **Error Limit**, yellow indicates elements between **Error and Warning** Limits and grey indicates elements that are below the **Warning Limit**.
- **Show Histogram**: Displays the mesh metrics histogram for the selected quality criterion. **Show Histogram** is available in **Solid** and **Sheet** tab.
- **Isolate Failed Element Bodies**: Isolate the failed element bodies for the selected quality criterion. **Isolate Failed Element Bodies** is available in **Solid** and **Sheet** tab.
- **Open Troubleshooting Documentation**: Opens the troubleshooting documentation page from the browser. **Open Troubleshooting Documentation** is available in all tabs.
- **Visibility Off**: Allows you to disable the selected quality criterion from the **Mesh Quality Worksheet**. **Visibility Off** is available in all tabs.

If **Create NS Group: Failed Elements** or **Create NS Group: Warning Elements** is used, you can right-click the created named selection group in the tree and click **Select Mesh Clusters in Group** to select all the elements clusters in the named selection. You can then use the **Walk** under the **Selection** tab to navigate through the clusters of problem elements.

You can click **File > Options > Meshing** and select **Check Mesh Quality** as **Mesh Quality Worksheet** under **Quality** in **Meshing** to use the worksheet as the default for mesh validation method.

You can customize the quality criteria as per your preference under **Mesh Quality** in **File > Options > Meshing**. The default quality criteria displayed in the worksheet differs for **Sheet**, **Solid** and **Solid - Surface**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can select **On** or **Off** to hide or display the quality criteria in the **Mesh Quality Worksheet** from **File > Options > Meshing > Mesh Quality**. **Mesh Quality Worksheet** calculates quality for available criteria only when you mesh or refresh the worksheet. You should hide quality criteria that are not used, to reduce time taken for quality calculation in larger meshes.

When **Check Mesh Quality** is set to **Quality Worksheet** and you can view contours of mesh quality using **Display Style**, the properties of the contours are modified from the default display. For a given criterion, the **Geometry** window displays elements between the **Warning Limit** and **Error Limit** as yellow and ones exceeding the **Error Limit** as red.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Some additional element layers in grey color around the red color and yellow color elements help you understand the cause for mesh issues. To increase the number of mesh layers for required quality elements press **Shift+W** or , for decreasing number of mesh layers for required quality elements press **Shift+S** or . If you press **Shift+W** or several times, all mesh elements are shown. To go back to the cluster view, you must use **Shift+S** or to remove element layers.

Limitation

- When you enable **Error Check** for any mesh metrics in the **Mesh Quality Worksheet**, **Mesh Quality Worksheet** does not invalidate the mesh generated by the Mesh Workflow (p. 601).

Error and Warning Limits

During mesh generation, an element's quality (which is sometimes called the element shape) is calculated. The meshing algorithm uses error limits to obtain a valid mesh. It performs extra mesh cleanup to ensure the error limits are met, such that a valid mesh is one that satisfies the necessary (minimum) conditions and can be consumed by the solver. The meshing algorithm then attempts to improve quality based on warning (target) limits. Error and warning (target) limits can be defined further as:

- An error limit is the value at which an element's quality is not suitable for the solver being used, and by default the mesh would fail (see Check Mesh Quality (p. 121)). Priority is given to ensure there are no elements below the error limits. Error limits are determined by the physics preference as shown in the table below. You cannot change error limits. However, if you are using the **Mechanical** physics preference, you can choose from two sets of error limits: **Standard Mechanical** or **Aggressive Mechanical**. You can think of the error limits as the minimum quality criteria of the meshing.
- A warning (target) limit has two purposes:

  1. It is used as a warning limit. If the mesh has elements that are questionable for the solver being used, these elements can be flagged for you via the warning limit. To configure this warning behavior, set Check Mesh Quality (p. 121) to **Yes, Errors and Warnings**.
  2. It is used as a target limit. The mesh methods will first try to improve the mesh to ensure there are no elements that do not pass the error limits. If successful, the mesh methods do further improvements to try to meet the target limits. You can think of the target limits as the quality goals for the meshing. If the mesher cannot meet the goals, a warning can be issued. To configure this warning behavior, set Check Mesh Quality (p. 121) to **Yes, Errors and Warnings**.

Note:

All mesh methods use the warning (target) limits to flag warning elements if Check Mesh Quality (p. 121) is set to **Yes, Errors and Warnings**. However, not all mesh methods use

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the target limits to improve the mesh. Currently, only the Patch Conforming Tetra (p. 208) mesh method uses the target limits to improve the mesh.

You can control how the mesher responds when it reaches error and warning limits by setting the **Check Mesh Quality** (p. 121) option.

The error limits for each physics preference are:

- **Mechanical** - Uses either of these sets of error limits, depending on the setting of the **Error Limits** option:
  - **Standard Mechanical** - These error limits have proven to be effective for linear, modal, stress, and thermal problems.
  - **Aggressive Mechanical** - These error limits are more restrictive than the error limits for **Standard Mechanical**. **Aggressive Mechanical** may produce more elements, fail more often, and take longer to mesh. As an alternative, you can set **Physics Preference** to **Nonlinear Mechanical**. However, doing so changes other defaults and may significantly change the mesh size and/or which features the mesh is capturing, and therefore may have a big impact on mesh quality. This is the default when **Physics Preference** is set to **Mechanical**, but you can use the **Error Limits** option to change it.
- **Nonlinear Mechanical** - Uses error limits as shown in the table below to produce a high quality mesh that meets the shape checking requirements of tetrahedral elements for nonlinear analysis. If the element quality cannot meet the error limits, the mesh is not desirable for nonlinear analysis. These error limits are used whenever **Physics Preference** is set to **Nonlinear Mechanical**, you cannot change them.

  Note:

  Using the **Nonlinear Mechanical** option typically produces more elements and longer meshing times. If the element size is too coarse, meshing robustness may be problematic because it is sometimes difficult to get a good quality mesh that not only meets the coarse element size but also captures the features of the model. In such cases, you should reduce the element size, simplify the model, or set Check Mesh Quality (p. 121) to **No** to turn off the error checks.

- **Electromagnetics** - Uses error limits based on element volume, face warping, and face angle. These error limits are used whenever **Physics Preference** is set to **Electromagnetics**, you cannot change them.
- **CFD** - For non-assembly meshing algorithms, uses error limits based on element volume.
- **Explicit** - Uses error limits based on Jacobian ratio and element volume. These error limits are used whenever **Physics Preference** is set to **Explicit**, you cannot change them.

The following table presents the error and warning (target) limits for different values of Physics Preference (p. 96). The **Hydrodynamics** physics preference does not have error and warning limits, so it is not included in the table.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Physics Preference	Mechanical			Nonlinear Mechanical		Electromagnetics		CFD		Explicit	
Criterion	**Standard Mechanical Error Limit**	**Aggressive Mechanical Error Limit**	**Warning (Target) Limit**	**Error Limit**	**Warning (Target) Limit**	**Error Limit**	**Warning (Target) Limit**	**Error Limit**	**Warning (Target) Limit**	**Error Limit**	**Warning (Target) Limit**
Element Quality (p. 139)	< 5 x 10^{-4} for 3D < 0.01 for 2D < 0.75 for 1D	< 5 x 10^{-4} for 3D < 0.02 for 2D < 0.85 for 1D	< 0.05 (default)	< 5 x 10^{-4} for 3D < 0.02 for 2D < 0.85 for 1D	N/A	N/A	< 0.05 (default)	N/A	N/A	N/A	< 0.2 (default)
Jacobian Ratio (p. 142) (Gauss Points)	< 0.025	N/A	N/A	< 0.025	N/A	N/A	N/A	N/A	N/A	N/A	N/A
Jacobian Ratio (p. 142) (Corner Nodes)	N/A	< 0.025	N/A	< 0.001	< 0.04 (default)	N/A	N/A	N/A	N/A	< 0.001	N/A
Skewness (p. 151)	N/A	N/A	N/A	N/A	> 0.9 (default)	N/A	N/A	N/A	> 0.9 (default)	N/A	N/A
Orthogonal Quality (p. 153)	N/A	N/A	N/A	N/A	N/A	N/A	N/A	<= 0 for meshing, not used for other methods	<= 0.05 for assembly meshing, not used for other methods	N/A	N/A
Element Volume	< 0	< 0	N/A	< 0	N/A	< 10^{-30}	N/A	< 10^{-32},	N/A	< 0	N/A
Aspect Ratio (for triangles (p. 140)	N/A	N/A	N/A	N/A	N/A	N/A	N/A	N/A	N/A	N/A	N/A

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Physics Preference	Mechanical			Nonlinear Mechanical		Electromagnetics		CFD		Explicit	
and quadrilaterals (p. 141))											
Face Angle	N/A	N/A	N/A	N/A	N/A	> 150	N/A	N/A	N/A	N/A	N/A
Face Warping (p. 147)	N/A	N/A	N/A	N/A	N/A	> 0.4	N/A	N/A	N/A	N/A	N/A
Aspect Ratio (Height)	N/A	N/A	N/A	N/A	N/A	N/A	N/A	N/A	N/A	N/A	>5 (default)
Characteristic Length (LS-DYNA)	N/A	N/A	N/A	N/A	N/A	N/A	N/A	N/A	N/A	N/A	<10% Element Size (default)

Target Element Quality

The **Target Element Quality** global option allows you to set a target Element Quality (p. 139) that you would like the mesh to satisfy.

The **Target Element Quality** value drives improvements to tetrahedral elements. If you set the target quality and the mesh contains tetrahedral elements, the mesher will attempt to improve the tetrahedral elements to meet the target quality that you specified. If the target quality cannot be met, a valid mesh may still be generated. In addition, if Check Mesh Quality (p. 121) is set to **Yes, Errors and Warnings**, a warning message is displayed to help you address the issues preventing the mesh from satisfying the target quality. You can right-click the Message field and select the Show Elements (p. 538) option from the context menu to create Named Selections for the elements that don't meet the target.

You should set the target quality if you intend to run a simulation that is sensitive to mesh quality. However, because setting the target quality increases memory usage and the time required to generate the mesh, you should not set the quality any higher than necessary.

To set the **Target Element Quality**, enter a value between **0** (lower quality) and **1** (higher quality). The default is 0.05.

Note:

- **Target Element Quality** is supported for the Patch Conforming Tetra (p. 208) mesh method only.
- The Adaptive sizing can result in coarse mesh sizes with stretched elements that cannot be improved with a higher target quality value. Therefore, if you are using the Adaptive sizing, you should set the **Target Element Quality** to a value < 0.1. Alternatively, you could use a different sizing option (such as Curvature (p. 107)).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Target Skewness

The **Target Skewness** global option allows you to set a target Skewness (p. 151) that you would like the mesh to satisfy.

The target skewness value drives improvements to tetrahedral elements. If you set the target skewness and the mesh contains tetrahedral elements, the mesher will attempt to improve the tetrahedral elements to meet the target skewness that you specified. If the target skewness cannot be met, a valid mesh may still be generated. In addition, if Check Mesh Quality (p. 121) is set to **Yes, Errors and Warnings**, a warning message is displayed to help you address the issues preventing the mesh from satisfying the target skewness. You can right-click the Message field and select the Show Elements (p. 538) option from the context menu to create Named Selections for the elements that don't meet the target.

You should set the target skewness if you intend to run a simulation that is sensitive to mesh quality. However, because setting the target skewness increases memory usage and the time required to generate the mesh, you should not set the skewness any lower than necessary.

To set the **Target Skewness**, enter a value between **0** (higher quality) and **1** (lower quality). The default is 0.9. For a tetrahedral mesh, you should not set **Target Skewness** to a value < 0.8.

Note:

Target Skewness is supported for the Patch Conforming Tetra (p. 208) mesh method only.

Target Jacobian Ratio (Corner Nodes)

The **Target Jacobian Ratio (Corner Nodes)** global option allows you to set a target Jacobian Ratio (p. 142) that you would like the mesh to satisfy.

The target Jacobian ratio value drives improvements to tetrahedral elements. If you set the target Jacobian ratio and the mesh contains tetrahedral elements, the mesher will attempt to improve the tetrahedral elements to meet the target Jacobian ratio that you specified. If the target Jacobian ratio cannot be met, a valid mesh may still be generated. In addition, if Check Mesh Quality (p. 121) is set to **Yes, Errors and Warnings**, a warning message is displayed to help you address the issues preventing the mesh from satisfying the target Jacobian ratio. You can right-click the Message field and select the Show Elements (p. 538) option from the context menu to create Named Selections for the elements that don't meet the target.

You should set the target Jacobian ratio if you intend to run a simulation that is sensitive to mesh quality. However, because setting the target Jacobian ratio increases memory usage and the time required to generate the mesh, you should not set the Jacobian ratio any higher than necessary.

To set the **Target Jacobian Ratio (Corner Nodes)**, enter a value between **0** (lower quality) and **1** (higher quality). The default is 0.04.

Note:

Target Jacobian Ratio (Corner Nodes) is supported for the Patch Conforming Tetra (p. 208) mesh method only.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Target Characteristic Length (LS-DYNA)

The **Target Characteristic Length (LS-DYNA)** global option allows you to set a target Characteristic Length (LS-DYNA) (p. 160) that you would like the mesh to satisfy to avoid small elements and resulting small critical time-step for Explicit simulations in the Mesh Details view.

If Check Mesh Quality (p. 121) is set to **Yes, Errors and Warnings**, a warning message is displayed to help you address the issues preventing the mesh from satisfying the target Characteristic Length (LS-DYNA). You can right-click the Message field and select the Show Elements (p. 538) option from the context menu to create Named Selections for the elements that do not meet the target.

By default, **Target Characteristic Length (LS-DYNA)** is available only when the **Physics Preference** is **Explicit**. When the **Target Characteristic Length (LS-DYNA)** is larger than the global **Element Size**, the **Target Characteristic Length (LS-DYNA)** is internally reset to half of the global **Element Size**.

Note:

- **Target Characteristic Length (LS-DYNA)** is supported for the Patch Conforming Tetra (p. 208) mesh method only.
- When the **Target Characteristic Length (LS-DYNA)** is not met, an error message is displayed to provide the percentage volume of elements that failed to meet the **Target Characteristic Length (LS- DYNA)** out of the total volume of elements on the flexible parts.

Target Aspect Ratio (Height)

The **Target Aspect Ratio (Height)** global option allows you to set a target Aspect Ratio (Height) (p. 159) that you would like the mesh to satisfy in the Mesh Details view.

The **Target Aspect Ratio (Height)** value drives improvements to tetrahedral elements through improvements to the surface mesh aspect ratio. If you set the **Target Aspect Ratio (Height)** and the mesh contains tetrahedral elements, the mesher will attempt to improve the tetrahedral elements to meet the **Target Aspect Ratio (Height)** that you specified. If the **Target Aspect Ratio (Height)** cannot be met, a valid mesh may still be generated. In addition, if Check Mesh Quality (p. 121) is set to **Yes, Errors and Warnings**, a warning message is displayed to help you address the issues preventing the mesh from satisfying the **Target Aspect Ratio (Height)**. You can right-click the Message field and select the Show Elements (p. 538) option from the context menu to create Named Selections for the elements that do not meet the target.

To set the **Target Aspect Ratio (Height)**, enter a value between **2** (higher quality) and **100** (lower quality). The default is **5**.

Note:

Target Aspect Ratio (Height) improvement is supported for the Patch Conforming Tetra (p. 208) mesh method only.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Target Aspect Ratio (Height) is available only when the **Physics Preference** is **Explicit**.

Smoothing

Smoothing attempts to improve element quality by moving locations of nodes with respect to surrounding nodes and elements. The **Low**, **Medium**, or **High** option controls the number of smoothing iterations along with the threshold metric where the mesher will start smoothing.

Note:

- When **Smoothing** is set to **High**, additional smoothing of inflation layers occurs. This may slow down the prism generation process.
- For more controls on smoothing to improve quality, you can use **Automatic Node Movement** in Patch Conforming Tetrahedrons Method (p. 208).

Mesh Metric

The **Mesh Metric** option allows you to view mesh metric information and thereby evaluate the mesh quality. Once you have generated a mesh, you can choose to view information about any of the following mesh metrics: Element Quality (p. 139), **Aspect Ratio** for triangles (p. 140) or quadrilaterals (p. 141), Jacobian Ratio (p. 142) (MAPDL, corner nodes, or Gauss points), Warping Factor (p. 147), Parallel Deviation (p. 149), Maximum Corner Angle (p. 150), Skewness (p. 151), Orthogonal Quality (p. 153), Characteristic Length (Autodyn) (p. 155), Minimum Tri Angle (p. 156), Maximum Tri Angle (p. 156), Minimum Quad Angle (p. 157), Maximum Quad Angle (p. 157), Warping Angle (p. 157), Tet Collapse (p. 158), Aspect Ratio (Height) (p. 159), Minimum Element Edge Length (p. 159), Maximum Element Edge Length (p. 160) and Characteristic Length (LS-DYNA) (p. 160). Selecting **None** turns off mesh metric viewing.

When you select a mesh metric, its **Min**, **Max**, **Average**, and **Standard Deviation** values are reported in the Details View, and a bar graph is displayed under the **Geometry** window. The graph is labeled with color-coded bars for each element shape represented in the model's mesh, and can be manipulated to view specific mesh statistics of interest (p. 134).

Note:

If the model contains multiple parts or bodies, you can view the mesh metric information for an individual part or body. To do so, return to the Tree Outline. Under the **Geometry** object, click the specific part or body of interest. The Details view displays the **Nodes**, **Elements**, **Min**, **Max**, **Average**, and **Standard Deviation** values for the selected metric and part/body under **Statistics**. (The graph is not available at the part/body level.)

Accessing Mesh Metric Information

To access mesh metric information:

1. Generate the mesh. You can view mesh metric information for any mesh that was successfully generated using the Generate Mesh (p. 528), Preview Surface Mesh (p. 531), or Preview Inflation (p. 534) feature.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click the **Mesh** object in the Tree Outline.
3. In the Details View, expand the **Quality** folder.
4. For the **Mesh Metric** control, select the metric of interest from the drop-down menu.

   By default, the **Min**, **Max**, **Average**, and **Standard Deviation** values for the selected metric are reported in the Details View. In addition, a bar graph (p. 134) is displayed under the **Geometry** window.

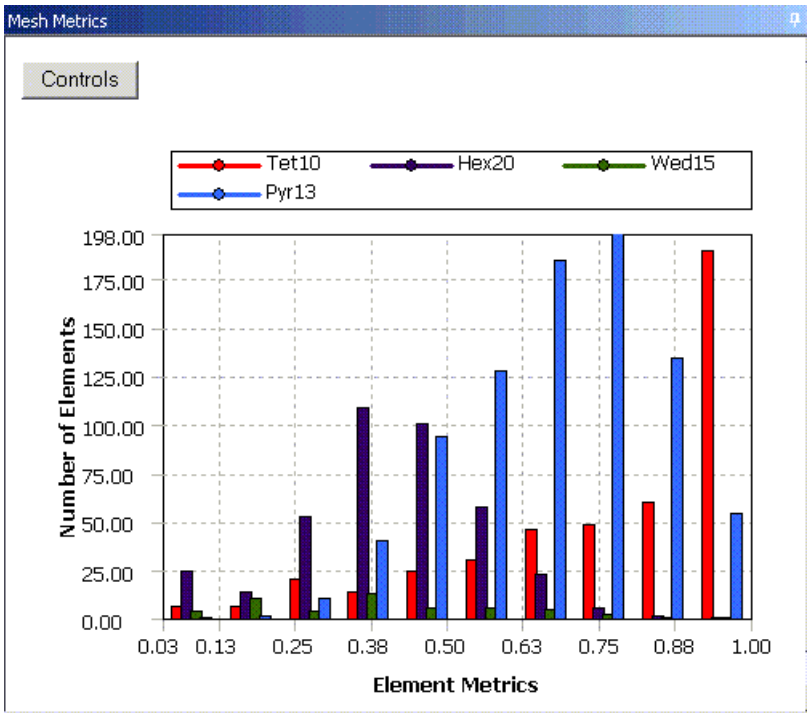
Note:

To view the numbers of **Nodes** and **Elements** in the meshed model, expand the Statistics (p. 198) folder in the Details View.

Viewing Advanced Mesh Statistics

When you select a mesh metric, a bar graph is displayed as shown in Figure 34: Mesh Metrics Bar Graph (p. 134). For this illustration, the **Element Quality** mesh metric was selected in the Details View, so the bar graph displays the minimum to maximum **Element Quality** values over the entire mesh.

Figure 34: Mesh Metrics Bar Graph



In Figure 34: Mesh Metrics Bar Graph (p. 134), the X-axis represents the value of the selected mesh metric. Using the **Y-Axis Option** setting described in Using the Bar Graph Controls (p. 137), you control whether the Y-axis represents the number of elements within a particular quality factor range (the default), or the percentage of the total volume represented by the elements within a particular quality factor range. In Figure 34: Mesh Metrics Bar Graph (p. 134), the Y-axis represents the number of elements. The alternative would be for the Y-axis to represent the percentage of the total volume. Remember that a model could have a large number of poorly shaped elements that are confined to

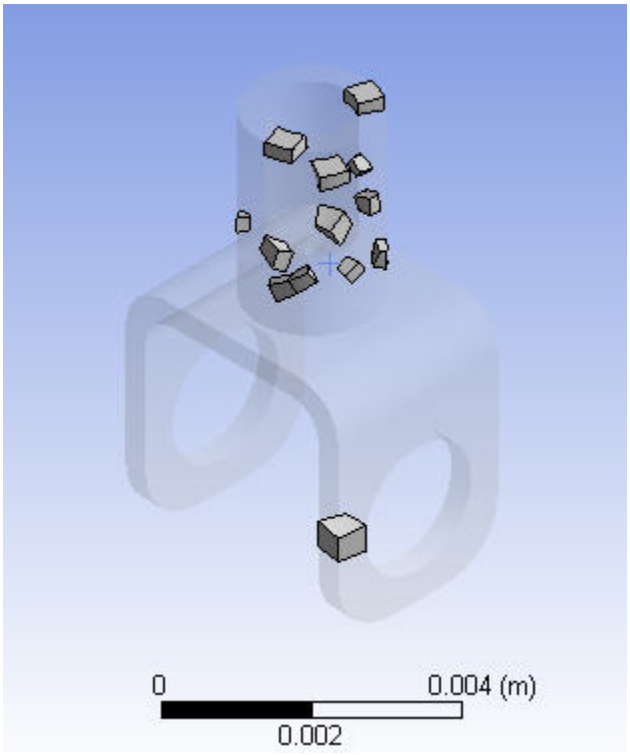
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a small local area. The total volume of these elements might not be significant compared to the volume of the entire model. As a result, the bar corresponding to this low quality factor may not be significant. The **Mesh Metric** option displays the selected mesh metric without qualifying the elements for acceptability.

Additional, characteristics of the bar graph include:

- The graph is displayed only when a mesh metric is selected. If you set **Mesh Metric** to **None**, the graph is not displayed. Alternatively, you can click the **Metric Graph** button on the toolbar to hide/show the graph.
- Resuming a model retains the last-saved state of the graph.
- Clicking the **Controls** button accesses the graph controls described in Using the Bar Graph Controls (p. 137).
- The location of an individual bar along the X-axis is the mid-point of the range of metric values covered by that bar.
- Clicking an individual bar on the graph (or in the column of white space above the bar) changes the view in the **Geometry** window. The geometry becomes transparent and only those elements meeting the criteria values corresponding to the selected bar are displayed, as shown in Figure 35: Geometry View After Selecting an Individual Bar (p. 135). (The option to click in the column above the bar is helpful if the graph contains very short bars that are difficult to click.)

Figure 35: Geometry View After Selecting an Individual Bar

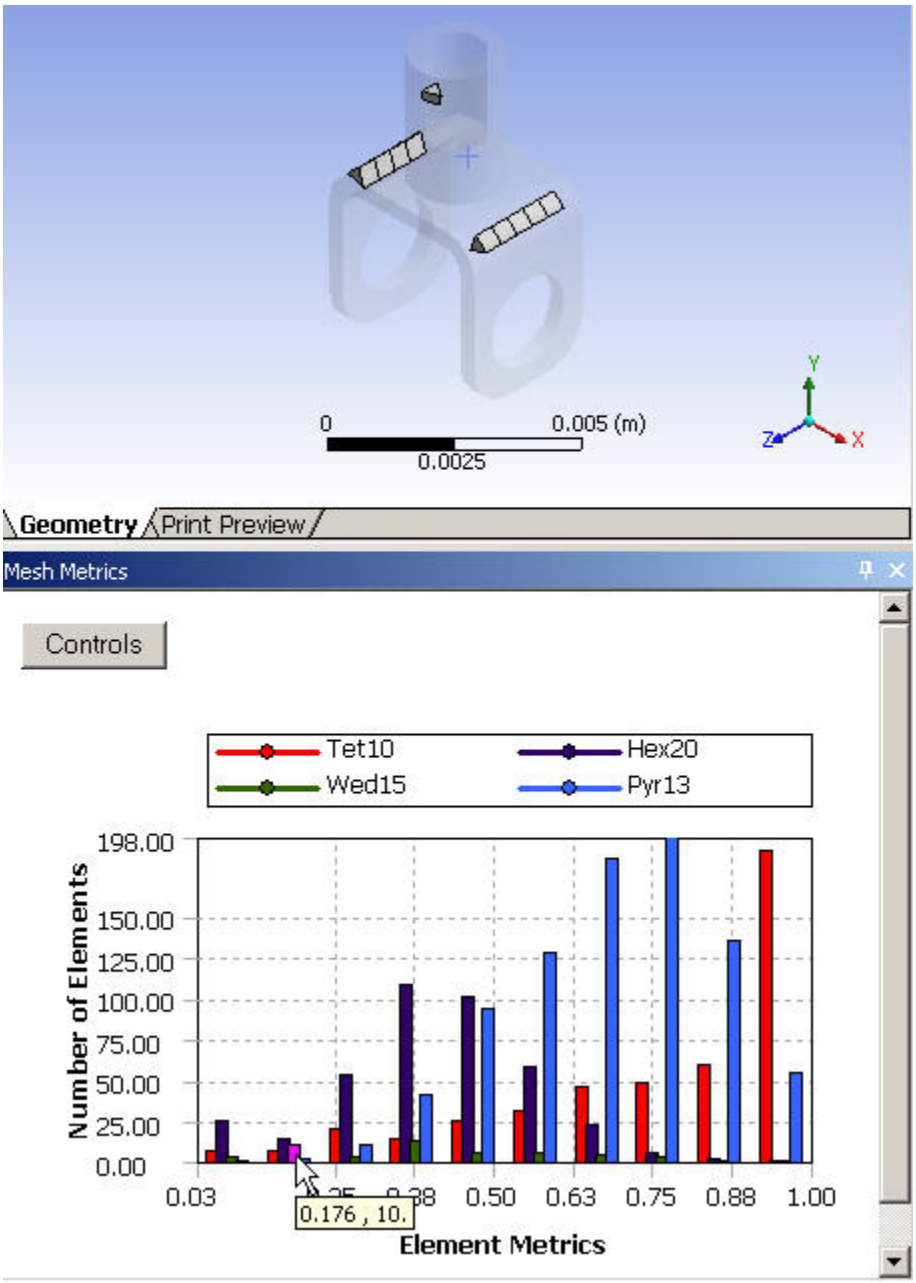


- If you click and hold the cursor on an individual bar or column, you see a tooltip showing the metric value associated with the bar, along with either a number of elements or the percent of total volume represented by the elements (depending on the **Y-Axis Option** setting). For

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

example, in Figure 36: Clicking and Holding on an Individual Bar (p. 136), 0.176 is the mid-point of the range of metric values covered by the selected bar, and there are 10 elements with values that fall within that range. The 10 elements are displayed in the **Geometry** window.

Figure 36: Clicking and Holding on an Individual Bar



- To select multiple bars, hold the **Ctrl** key and click all desired bars. All elements corresponding to all selected bars are displayed in the **Geometry** window.
- To return the **Geometry** window to the full mesh view (no transparency, all elements are displayed), click on *empty white space* on the graph. Empty white space does not include the column of white space above a bar, as clicking in this area selects the bar and displays only those elements associated with it.

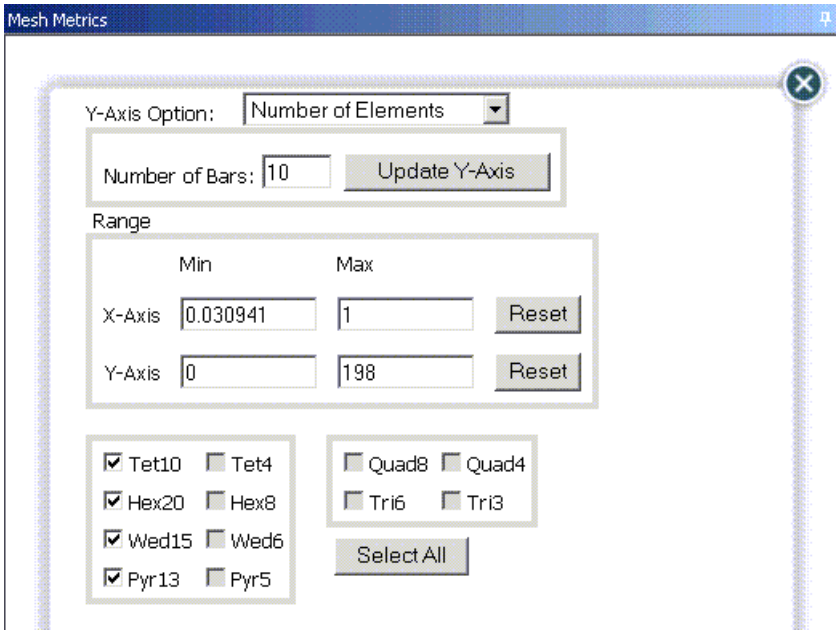
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you click in a column for which there are 0 elements, all that is displayed in the **Geometry** window is the transparent geometry.
- The graph can be filtered based on element types. See Using the Bar Graph Controls (p. 137) for more information.
- The graph respects section planes and hiding of bodies in the **Geometry** window. For example, if you hide a body and then click an individual bar to view the elements corresponding to the selected bar, elements in the hidden body are not displayed in the **Geometry** window, even if they meet the criteria that the bar represents.
- To zoom the graph, hold the **ALT** key and use your mouse to define a selection box on the graph (click the graph and drag the mouse downward and to the right to define the area to zoom, then release the mouse button). To reset the graph to its initial view, hold the **ALT** key, click the graph and drag the mouse downward and to the left, then release the mouse button.
- The values of the X-axis and Y-axis labels on the graph correspond to the *visible* ranges, rather than to global values. For example, the value 198 in Figure 36: Clicking and Holding on an Individual Bar (p. 136) is the maximum end of the range for the Y-axis, based on the current content of the graph. If you zoom the graph or define a new range of values to display as described in Using the Bar Graph Controls (p. 137), the values of the X-axis and Y-axis labels change accordingly along with the content of the graph.

Using the Bar Graph Controls

When you click the **Controls** button on the graph, the graph is replaced by the controls page as shown in Figure 37: Bar Graph Controls Page (p. 137). Clicking the **X** button applies any changes on the controls page and returns you to the graph.

Figure 37: Bar Graph Controls Page



From the controls page shown in Figure 37: Bar Graph Controls Page (p. 137), you can set the following values:

- **Y-Axis Option** - Determines what the heights of the bars represent. Options include **Number of Elements** and **Percent of Volume/Area**. The default is **Number of Elements**.
- **Number of Bars** - Determines the number of bars to include in the graph. You can enter any whole number greater than or equal to 0. The default is 10. When you click **Update Y-Axis**, the **Min** and/or **Max** values for the **Y-Axis** are recomputed so that the graph and the **Y-Axis** values on the controls page reflect the new number of bars.
- **Range** - Defines a range for the selected metric to display only those elements that fall within the specified range.
  - **X-Axis** - Specify a **Min** and/or **Max** value. To locate and estimate the number of worst elements in the mesh, adjust the **Min** and **Max** values to the lower or upper end of the quality criterion (depending on metric) and click **Update Y-Axis**. (Determining the distribution and location of all the bad elements at one time is helpful in cases where you may need to re-import your model into the DesignModeler application to remove the corresponding problematic geometry.) Click **Reset** to return to the **X-Axis** defaults. (Note: Negative values are acceptable.)
  - **Y-Axis** - Specify a **Min** and/or **Max** value. By lowering the **Max** value, you can clip the Y-axis for easier visualization of small bars, especially as they relate to different element types. Click **Reset** to return to the **Y-Axis** defaults.
- List of element types - Determines which element types to include in the graph. Element types that do not appear in the mesh are read-only on the controls page. Select the element types that you want to include in the graph, or click **Select All** to include all available element types in the graph. By default, all available element types are selected.

Note:

Because the bars approximate the metrics across the range of the **X-Axis**, choosing a very small number of bars over a large **X-Axis** range may move the bars away from the actual average metric of the elements represented by each bar.

Calculation Details

For information about the calculations that are performed for each metric, refer to:

Element Quality

Aspect Ratio Calculation for Triangles

Aspect Ratio Calculation for Quadrilaterals

Aspect Ratio Calculation for Tetrahedrons

Jacobian Ratio

Warping Factor

Parallel Deviation

Maximum Corner Angle

Skewness

Orthogonal Quality

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Characteristic Length (Autodyn)

Minimum Tri Angle

Maximum Tri Angle

Minimum Quad Angle

Maximum Quad Angle

Warping Angle

Tet Collapse

Aspect Ratio (Height)

Minimum Element Edge Length

Maximum Element Edge Length

Characteristic Length (LS-DYNA)

Element Quality

The **Element Quality** option provides a composite quality metric that ranges between 0 and 1. This metric is based on the ratio of the volume to the sum of the square of the edge lengths for 2D quad/tri elements, or the square root of the cube of the sum of the square of the edge lengths for 3D elements. A value of 1 indicates a perfect cube or square while a value of 0 indicates that the element has a zero or negative volume.

This can also be expressed as follows:

- For two-dimensional quad/tri elements:

$$Quality = C\left(area \Big/ \sum (EdgeLength)^2 \right)$$

Note:

For the **Mechanical** (p. 96) physics preference, if **Error Limit** is set to **Aggressive Mechanical** (p. 127) and the Jacobian Ratio (Corner Nodes) (p. 142) is less than zero, an error occurs.

- For three-dimensional brick elements:

$$Quality = C\left[volume \Big/ \sqrt{\left[\Sigma\left(Edge\ length\right)^2\right]^3} \right]$$

The following table lists the value of C for each type of element:

Element	Value of C
Triangle	6.92820323
Quadrangle	4.0
Tetrahedron	124.70765802
Hexahedron	41.56921938

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

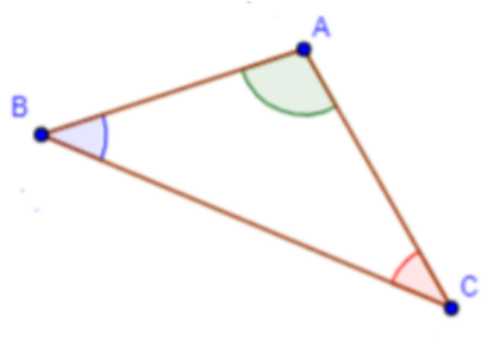
Wedge	62.35382905
Pyramid	96

Aspect Ratio Calculation for Triangles

The aspect ratio for triangles is calculated using the triangle interior angles. For each triangle node (corner node, not mid node), a nodal aspect ratio, r, is defined as:

$$r = \begin{cases} 0.5/(1-\cos\alpha), & \cos\alpha \geq 0.5 \\ 1.5/(1+\cos\alpha), & \cos\alpha < 0.5 \end{cases}$$

Where, α is the interior angle at the node.



$$\angle A + \angle B + \angle C = 180°$$

The mean nodal aspect ratio of a triangle is defined as:

$$\overline{r} = \frac{1}{3}\sum_{i=1}^{3} r_i$$

Where, r_i is the nodal aspect ratio of the i^{th}

The aspect ratio of the triangle is defined as:

$$R_t = 2\left(\sqrt{\overline{r}} - 1\right) + 1$$

For Explicit Physics Preference, **Aspect Ratio (Height)** is calculated as:

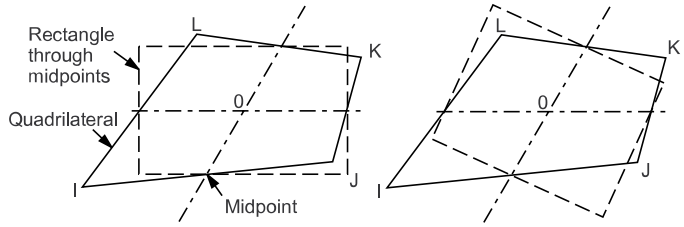
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$h^* = \left(\frac{2}{sqrt(3)}\right) * h$$

Aspect Ratio Calculation for Quadrilaterals

The aspect ratio for a quadrilateral is computed by the following steps, using only the corner nodes of the element (Figure 38: Quadrilateral Aspect Ratio Calculation (p. 141)):

Figure 38: Quadrilateral Aspect Ratio Calculation



1. If the element is not flat, the nodes are projected onto a plane passing through the average of the corner locations and perpendicular to the average of the corner normals. The remaining steps are performed on these projected locations.

2. Two lines are constructed that bisect the opposing pairs of element edges and which meet at the element center. In general, these lines are not perpendicular to each other or to any of the element edges.

3. Rectangles are constructed centered about each of the 2 lines, with edges passing through the element edge midpoints. The aspect ratio of the quadrilateral is the ratio of a longer side to a shorter side of whichever rectangle is most stretched.

4. The best possible quadrilateral aspect ratio, for a square, is one. A quadrilateral having an aspect ratio of 20 is shown in Figure 39: Aspect Ratios for Quadrilaterals (p. 141).

Figure 39: Aspect Ratios for Quadrilaterals

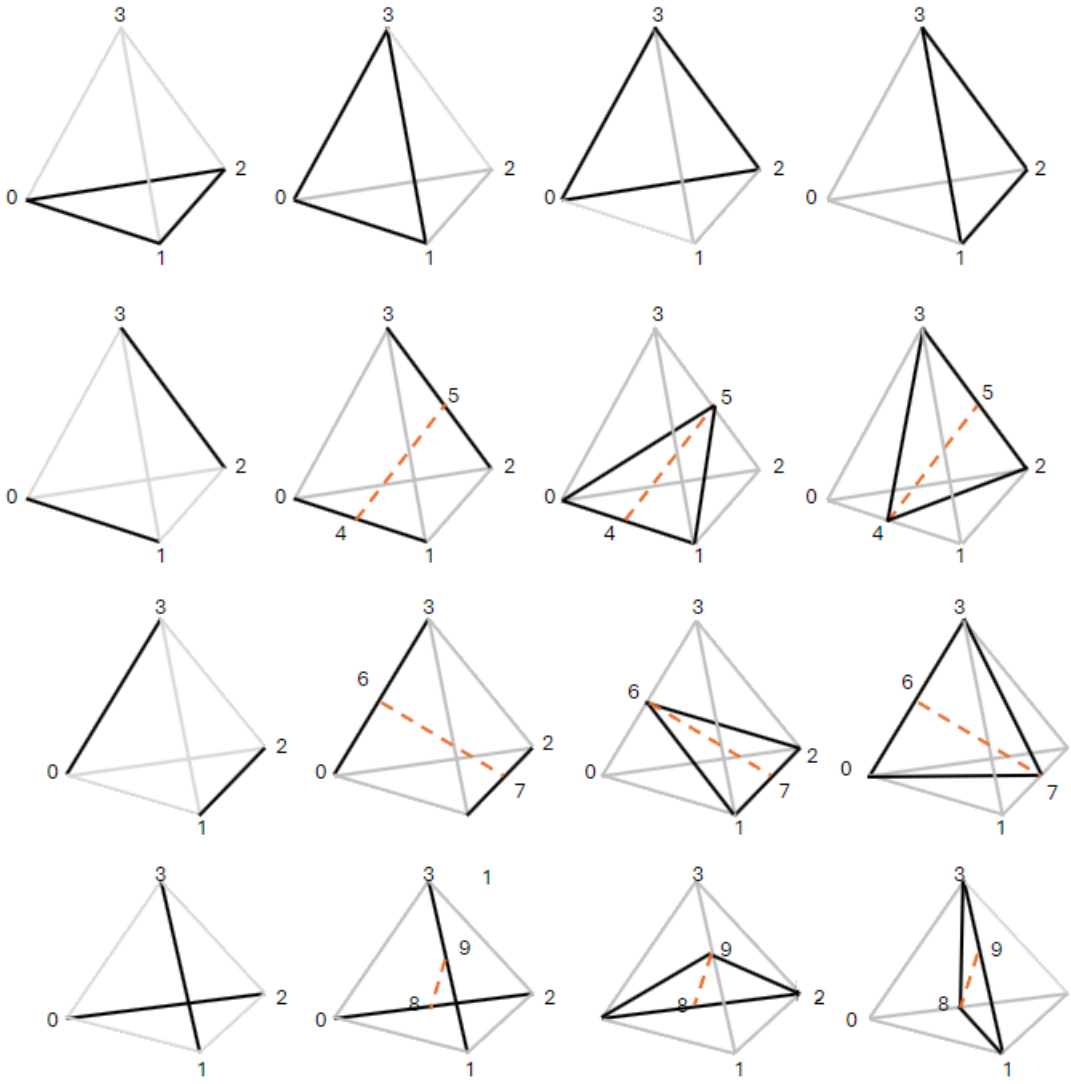
For Explicit Physics Preference, **Aspect Ratio (Height)** is calculated as:

$$h^* = h$$

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Aspect Ratio Calculation for Tetrahedrons

The aspect ratio for tetrahedrons is defined using four boundary triangles and six mid triangles. The mid triangles are formed connecting the corner nodes to the shortest distance point of the adjoining edges.



Tetrahedron Aspect Ratio is defined as the maximum aspect ratio of ten triangles, four boundary triangles and six projected triangles. Here the boundary triangles are (0,1,2), (0,1,3),(0,2,3), (1,2,3). The projected triangles formed by connecting the corner angles to the shortest distance point of the adjoining edges are (0,1,5) (1,2,3), (2,5,1) (0,1,2)(0,2,1) (1,5,2).

$$R = \max(R_t)$$

Jacobian Ratio

The Jacobian ratio is a measurement of the shape of a given element compared to that of an ideal element. If an element has a bad quality Jacobian ratio, the element may not map well from element space to real space, thereby making computations based on the element shape less reliable. The ideal shape of an element depends on element type.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Jacobian ratio is not calculated for triangles and tetrahedra that are linear (no midside nodes) or have straight-sided midside nodes. This is because the shape function for these types of elements is linear, and the partial derivatives of linear functions are constant. Therefore, the determinant of the Jacobian ratio for these types of elements is constant over the entire element, which means $R_J(\min) = R_J(\max)$. For such elements, the Jacobian ratio is always 1.

Jacobian Ratio Calculation

There are two ways to calculate the Jacobian ratio: either based on corner nodes (nodal points) or based on Gauss points (integration points).

When the Jacobian ratio calculation is based on...	Be aware that...
A sampling of element corner nodes	• The calculation is more restrictive. • To view mesh metric information based on this calculation, you must set Mesh Metric (p. 133) to one of the following: – **Jacobian Ratio (Corner Nodes)**, which is bounded by -1 (worst) and 1 (best) on the mesh metrics bar graph (p. 134). An element with a Jacobian ratio <= 0 should be avoided. – **Jacobian Ratio (MAPDL)**, which is used by the MAPDL solver, is the inverse of **Jacobian Ratio (Corner Nodes)**. **Jacobian Ratio (MAPDL)** is bounded by negative infinity and positive infinity, but all negative-value elements are collected and arbitrarily assigned the value -100 for purposes of the mesh metrics bar graph (p. 134). An element with a Jacobian ratio <= 0 should be avoided. A Jacobian ratio whose value is close to 1 is best. **Note:** When **Physics Preference** is set to **Mechanical**, the Error Limits (p. 127) option for shape checking is set to **Standard Mechanical**. The Jacobian ratio calculation used for **Standard Mechanical** checks Gauss points but does not check corner nodes. This may lead to a situation where you have an element that has a **Jacobian Ratio (Gauss Points)** that is > 0 and a **Jacobian Ratio (Corner Nodes)** that is <= 0.
A sampling of element Gauss points	• The calculation is less restrictive. • To view mesh metric information based on this calculation, you must set Mesh Metric (p. 133) to **Jacobian Ratio (Gauss Points)**, which is bounded by -1 (worst) and 1 (best) on the mesh metrics bar graph (p. 134). An element with a Jacobian ratio <= 0 should be avoided.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

An element's Jacobian ratio is computed by the following steps, using the full set of nodes for the element:

1. Sampling locations are based on the selected **Mesh Metric** option [**Jacobian Ratio (MAPDL)**, **Jacobian Ratio (Corner Nodes)**, or **Jacobian Ratio (Gauss Points)**]. At each sampling location listed in the table below, the determinant of the Jacobian matrix is computed and called R_J. R_J at a given point represents the magnitude of the mapping function between element natural coordinates and real space. In an ideally-shaped element, R_J is relatively constant over the element, and does not change sign.

Element Shape	**R_J Sampling Locations for Jacobian Ratio (MAPDL) and Jacobian Ratio (Corner Nodes)**	**R_J Sampling Locations for Jacobian Ratio (Gauss Points)**
10-node tetrahedra	Corner nodes	Four Gauss quadrature points
5-node or 13-node pyramids	Base corner nodes and near apex node (apex R_J factored so that a pyramid having all edges the same length will produce a Jacobian ratio of 1)	5-node pyramids use 1 Gauss quadrature point 13-node pyramids use 8 Gauss quadrature points
8-node quadrilaterals	Corner nodes and centroid	Four Gauss quadrature points
20-node bricks	All nodes and centroid	Eight Gauss quadrature points
all other elements	Corner nodes	Choose the optimal number of Gauss quadrature points for integration

2. For **Jacobian Ratio (MAPDL)**, the Jacobian ratio of the element is the ratio of the *maximum* to the *minimum* sampled value of R_J, while for **Jacobian Ratio (Corner Nodes)** and **Jacobian Ratio (Gauss Points)**, it is the ratio of the *minimum* to the *maximum*. For **Jacobian Ratio (MAPDL)**, if the maximum and minimum have opposite signs, the Jacobian ratio is arbitrarily assigned to be -100 (and the element is clearly unacceptable).

3. If the element is a midside-node tetrahedron, an additional R_J is computed for a fictitious straight-sided tetrahedron connected to the 4 corner nodes. For **Jacobian Ratio (MAPDL)**, if that R_J differs in sign from any nodal R_J (an extremely rare occurrence), the Jacobian ratio is arbitrarily assigned to be -100. For **Jacobian Ratio (Corner Nodes)** and **Jacobian Ratio (Gauss Points)**, the Jacobian ratio is assigned to be -1.

4. If the element is a line element having a midside node, the Jacobian matrix is not square (because the mapping is from one natural coordinate to 2-D or 3-D space) and has no determinant. For this case, a vector calculation is used to compute a number which behaves like a Jacobian ratio. This calculation has the effect of limiting the arc spanned by a single element to about 106°.

The Jacobian ratio of various element shapes may worsen as follows:

- A triangle or tetrahedron has a Jacobian ratio of 1 if each midside node, if any, is positioned at the average of the corresponding corner node locations. This is true no matter how oth-

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

erwise distorted the element may be. Hence, this calculation is skipped entirely for such elements. Moving a midside node away from the edge midpoint position will worsen the Jacobian ratio. If the node is moved significantly, the Jacobian ratio will become negative and the element is invalid.

- Any rectangle or rectangular parallelepiped having no midside nodes, or having midside nodes at the midpoints of its edges, has a Jacobian ratio of 1. Moving midside nodes toward or away from each other can worsen the Jacobian ratio. If the node is moved significantly, the Jacobian ratio will become negative and the element is invalid.
- A quadrilateral or brick has a Jacobian ratio of 1 if (a) its opposing faces are all parallel to each other, and (b) each midside node, if any, is positioned at the average of the corresponding corner node locations. As a corner node moves near the center, the Jacobian ratio worsens. If the node is moved significantly, the Jacobian ratio will become negative and the element is invalid.

The figures below illustrate mesh quality by color for **Jacobian Ratio (MAPDL)** and **Jacobian Ratio (Corner Nodes)**.

Note:

To view mesh quality by color, in the Details view, set Display Style to the desired metric. The use of **Display Style** to color mesh according to quality and the use of mesh metrics (p. 133) to view quality statistics are mutually exclusive. For example, when you are viewing mesh metrics, you cannot also view the mesh quality by color.

Figure 40: Jacobian Ratio (MAPDL)

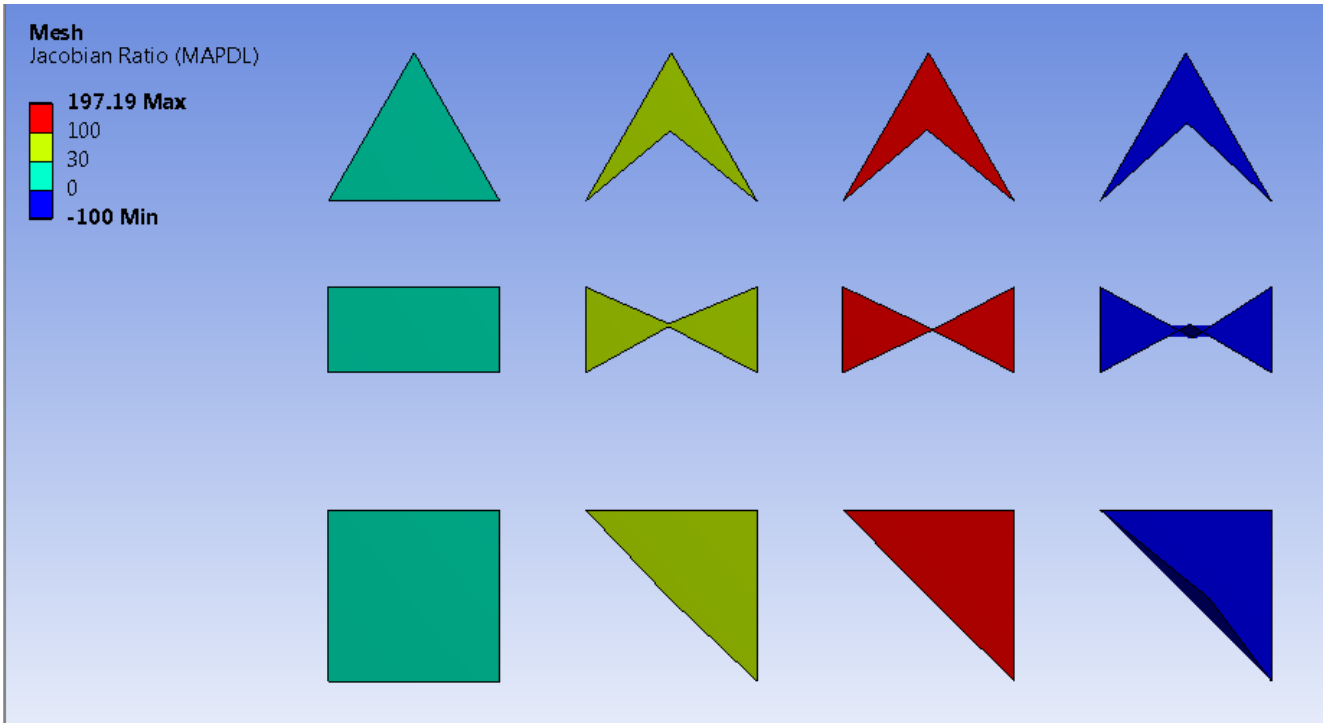
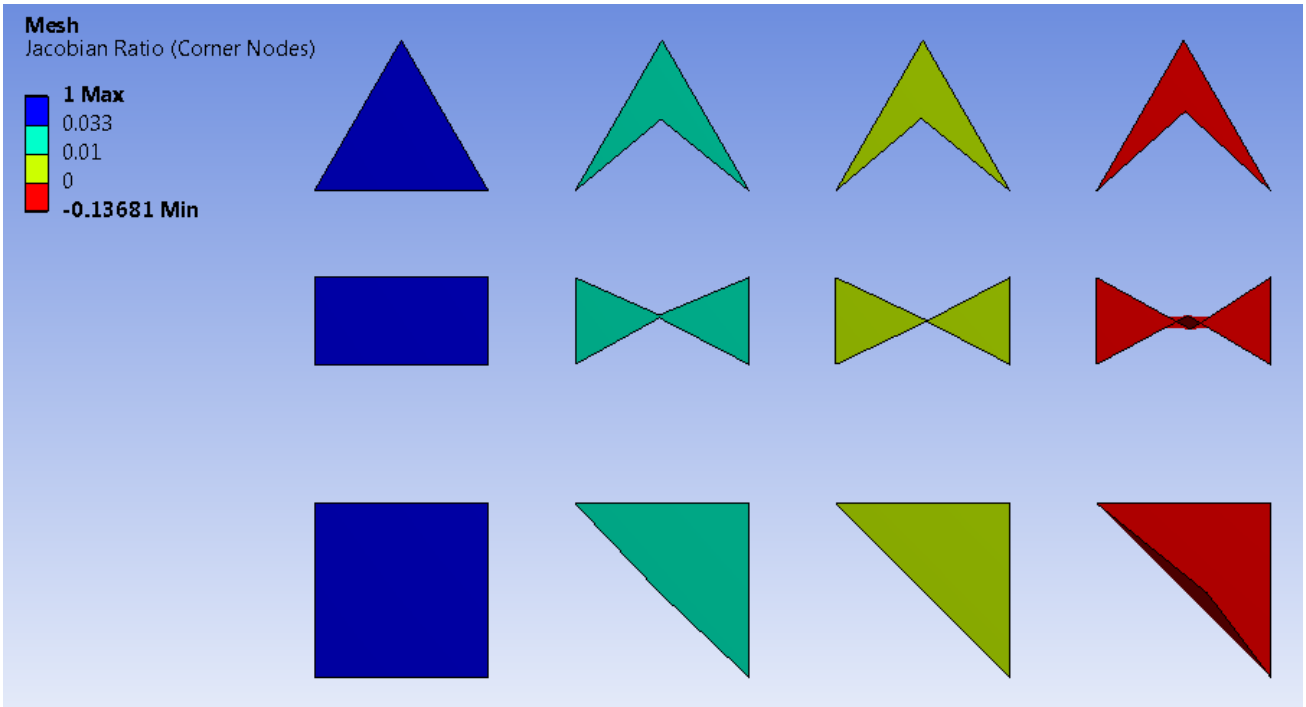


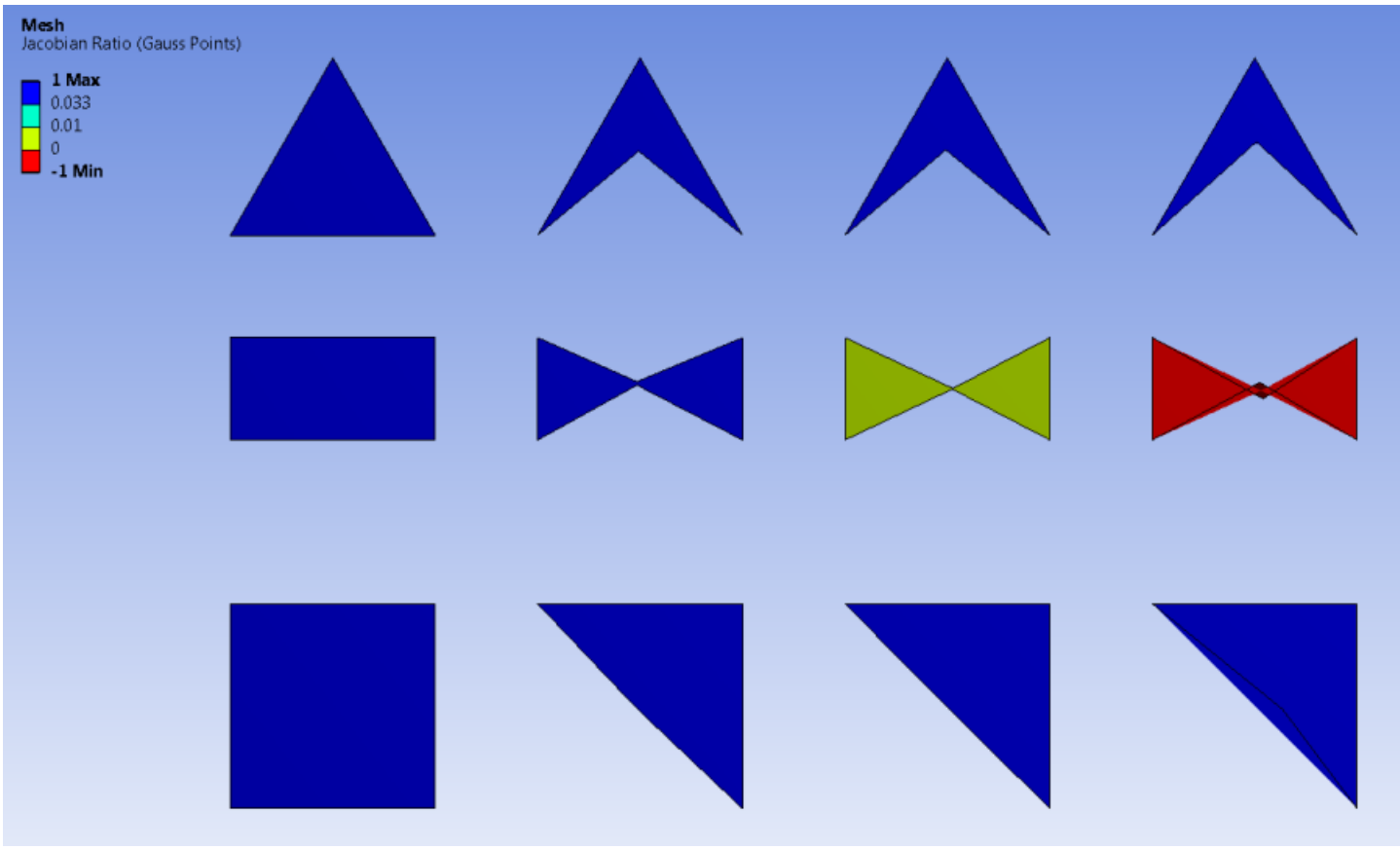
Figure 41: Jacobian Ratio (Corner Nodes)



Jacobian Ratio (Gauss Points) is a good indicator of quality for quadratic tetrahedrons because its formulation is similar to that used by the solver. However, it is not very helpful for shell meshes. For shell meshes, Element Quality (p. 139) is a better indicator of mesh quality. For example, the 2D elements that register as bad in the preceding figures will not necessarily register as bad elements for the **Jacobian Ratio (Gauss Points)** mesh metric, as shown below:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 42: Jacobian Ratio (Gauss Points)



Warping Factor

Warping factor is computed and tested for some quadrilateral shell elements, and the quadrilateral faces of bricks, wedges, and pyramids. A high factor may indicate a condition the underlying element formulation cannot handle well, or may simply hint at a mesh generation flaw.

Warping Factor Calculation for Quadrilateral Shell Elements

A quadrilateral element's warping factor is computed from its corner node positions and other available data by the following steps:

Note:

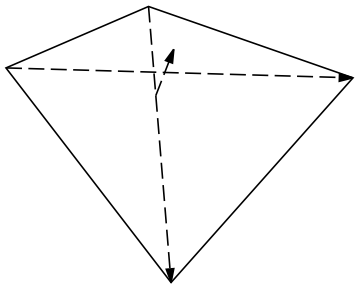
When computing the warping factor for a quadrilateral shell element, the Meshing application assumes 0 thickness for the shell.

---

1. An average element normal is computed as the vector (cross) product of the 2 diagonals (Figure 43: Shell Average Normal Calculation (p. 148)).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

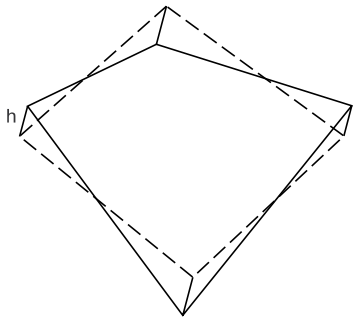
Figure 43: Shell Average Normal Calculation



2. The projected area of the element is computed on a plane through the average normal (the dotted outline on Figure 44: Shell Element Projected onto a Plane (p. 148)).

3. The difference in height of the ends of an element edge is computed, parallel to the average normal. In Figure 44: Shell Element Projected onto a Plane (p. 148), this distance is 2h. Because of the way the average normal is constructed, h is the same at all four corners. For a flat quadrilateral, the distance is zero.

Figure 44: Shell Element Projected onto a Plane



4. The "area warping factor" (F_a^w) for the element is computed as the edge height difference divided by the square root of the projected area.

5. For all shells except those in the "membrane stiffness only" group, if the thickness is available, the "thickness warping factor" is computed as the edge height difference divided by the average element thickness. This could be substantially higher than the area warping factor computed in 4 (above).

6. The warping factor tested against warning and error limits (and reported in warning and error messages) is the larger of the area factor and, if available, the thickness factor.

7. The best possible quadrilateral warping factor, for a flat quadrilateral, is zero.

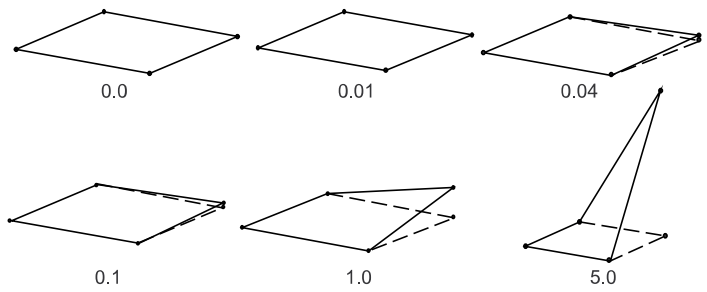
Figure 45: Quadrilateral Shell Having Warping Factor (p. 149) shows a "warped" element plotted on top of a flat one. Only the right-hand node of the upper element is moved. The element is a unit square, with a real constant thickness of 0.1.

When the upper element is warped by a factor of 0.01, it cannot be visibly distinguished from the underlying flat one.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the upper element is warped by a factor of 0.04, it just begins to visibly separate from the flat one.

Figure 45: Quadrilateral Shell Having Warping Factor



Warping of 0.1 is visible given the flat reference, but seems trivial; however, it is well beyond the error limit for a membrane shell. Warping of 1.0 is visually unappealing. This is the error limit for most shells.

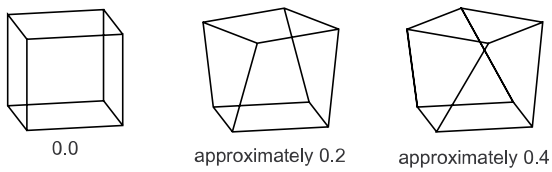
Warping beyond 1.0 would appear to be obviously unacceptable. However, SHELL181 permits even this much distortion. Furthermore, the warping factor calculation seems to peak at about 7.0. Moving the node further off the original plane, even by much larger distances than shown here, does not further increase the warping factor for this geometry. Users are cautioned that manually increasing the error limit beyond its default of 5.0 for these elements could mean no real limit on element distortion.

Warping Factor Calculation for 3-D Solid Elements

The warping factor for a 3-D solid element face is computed as though the 4 nodes make up a quadrilateral shell element with no real constant thickness available, using the square root of the projected area of the face as described in 4 (above).

The warping factor for the element is the largest of the warping factors computed for the 6 quadrilateral faces of a brick, 3 quadrilateral faces of a wedge, or 1 quadrilateral face of a pyramid. Any brick element having all flat faces has a warping factor of zero (Figure 46: Warping Factor for Bricks (p. 149)).

Figure 46: Warping Factor for Bricks



Twisting the top face of a unit cube by 22.5° and 45° relative to the base produces warping factors of about 0.2 and 0.4, respectively.

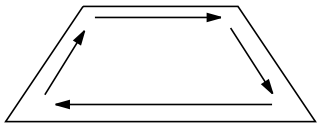
Parallel Deviation

Parallel deviation is computed using the following steps:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Ignoring midside nodes, unit vectors are constructed in 3-D space along each element edge, adjusted for consistent direction, as demonstrated in Figure 47: Parallel Deviation Unit Vectors (p. 150).

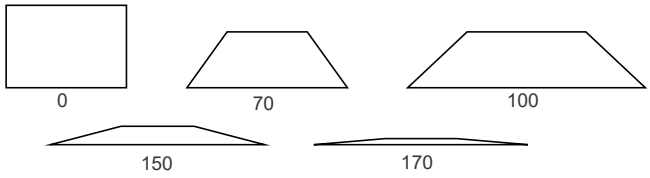
   Figure 47: Parallel Deviation Unit Vectors

2. For each pair of opposite edges, the dot product of the unit vectors is computed, then the angle (in degrees) whose cosine is that dot product. The parallel deviation is the larger of these 2 angles. (In the illustration above, the dot product of the 2 horizontal unit vectors is 1, and acos (1) = 0°. The dot product of the 2 vertical vectors is 0.342, and acos (0.342) = 70°. Therefore, this element's parallel deviation is 70°.)

3. The best possible deviation, for a flat rectangle, is 0°. Figure 48: Parallel Deviations for Quadrilaterals (p. 150) shows quadrilaterals having deviations of 0°, 70°, 100°, 150°, and 170°.

   Figure 48: Parallel Deviations for Quadrilaterals

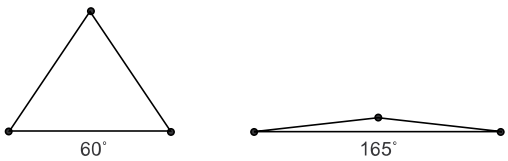
Maximum Corner Angle

Maximum corner angle is computed and tested for all except Emag elements. Some in the finite element community have reported that large angles (approaching 180°) degrade element performance, while small angles don't.

Maximum Corner Angle Calculation

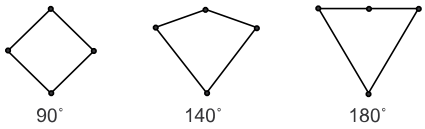
The maximum angle between adjacent edges is computed using corner node positions in 3-D space. (Midside nodes, if any, are ignored.) The best possible triangle maximum angle, for an equilateral triangle, is 60°. Figure 49: Maximum Corner Angles for Triangles (p. 150) shows a triangle having a maximum corner angle of 165°. The best possible quadrilateral maximum angle, for a flat rectangle, is 90°. Figure 50: Maximum Corner Angles for Quadrilaterals (p. 151) shows quadrilaterals having maximum corner angles of 90°, 140° and 180°.

Figure 49: Maximum Corner Angles for Triangles



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

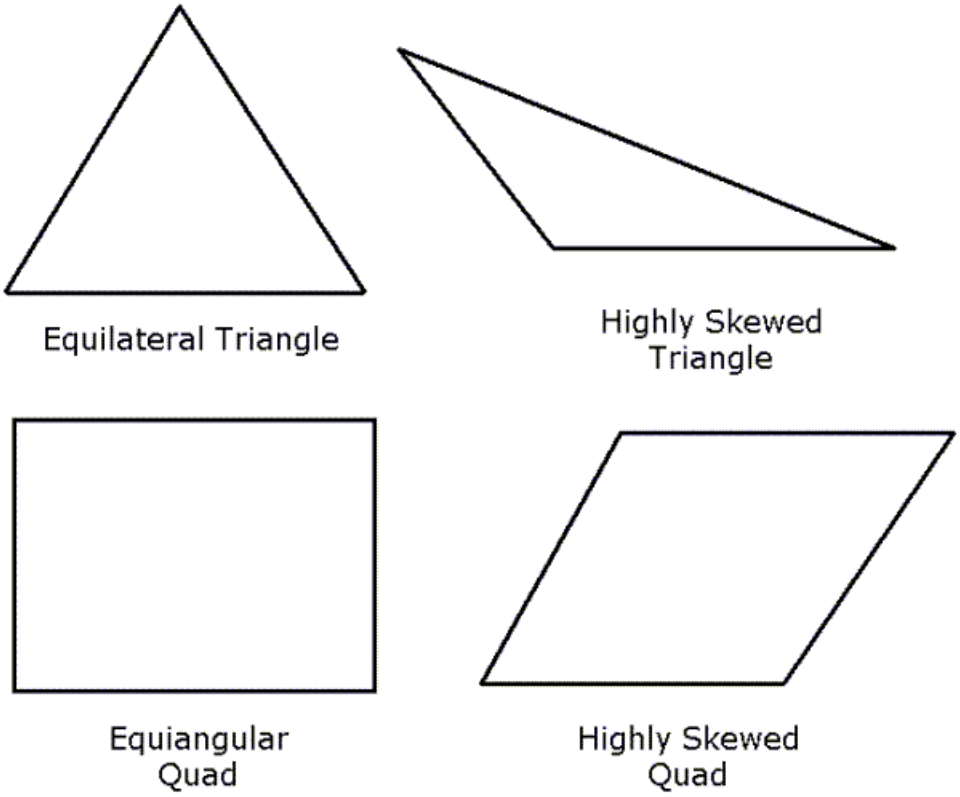
Figure 50: Maximum Corner Angles for Quadrilaterals



Skewness

Skewness is one of the primary quality measures for a mesh. Skewness determines how close to ideal (equilateral or equiangular) a face or cell is (see Figure 51: Ideal and Skewed Triangles and Quadrilaterals (p. 151)).

Figure 51: Ideal and Skewed Triangles and Quadrilaterals



The following table lists the range of skewness values and the corresponding cell quality.

Value of Skewness	Cell Quality
1	degenerate
0.9 — <1	bad (sliver)
0.75 — 0.9	poor
0.5 — 0.75	fair
0.25 — 0.5	good
>0 — 0.25	excellent
0	equilateral

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 (worst). Degenerate cells (slivers) are characterized by nodes that are nearly coplanar (colinear in 2D).

Highly skewed faces and cells are unacceptable because the equations being solved assume that the cells are relatively equilateral/equiangular.

Two methods for measuring skewness are:

- Based on the equilateral volume (applies only to triangles and tetrahedra).
- Based on the deviation from a normalized equilateral angle. This method applies to all cell and face shapes, including pyramids and prisms.

Equilateral-Volume-Based Skewness

In the equilateral volume deviation method, skewness is defined as

$$\text{Skewness} = \frac{\text{Optimal Cell Size - Cell Size}}{\text{Optimal Cell Size}}$$

where, the optimal cell size is the size of an equilateral cell with the same circumradius.

Quality meshes have a skewness value of approximately 0.1 for 2D and 0.4 for 3D. The table above provides a general guide to the relationship between cell skewness and quality.

In 2D, all cells should be good or better. The presence of cells that are fair or worse indicates 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.

In 3D, 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.

Note:

The Equilateral-Volume-Based Skewness quality metric applies to any mesh element that includes a triangular face. For triangular and tetrahedral elements, all faces of which are strictly triangular, the Equilateral-Volume-Based Skewness metric applies directly. For wedge or pyramidal elements, which include combinations of triangular and quadrilateral faces, the Meshing application computes both Equilateral-Volume-Based Skewness metrics (for the triangular faces) *and* Normalized Equiangular Skewness metrics (for the quadrilateral faces and 3-D element, itself) and reports the maximum computed metric as the element skewness. As a result, Equilateral-Volume-Based Skewness metrics reported for meshes that contain wedge and/or pyramidal elements may include skewness values attributable to Normalized Equiangular Skewness computations.

Normalized Equiangular Skewness

In the normalized angle deviation method, skewness is defined (in general) as

$$\max\left[\frac{\theta\text{max}-\theta_e}{180-\theta_e}, \frac{\theta_e-\theta_{\text{min}}}{\theta_e}\right]$$

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 (60 for a triangle, 90 for a square)

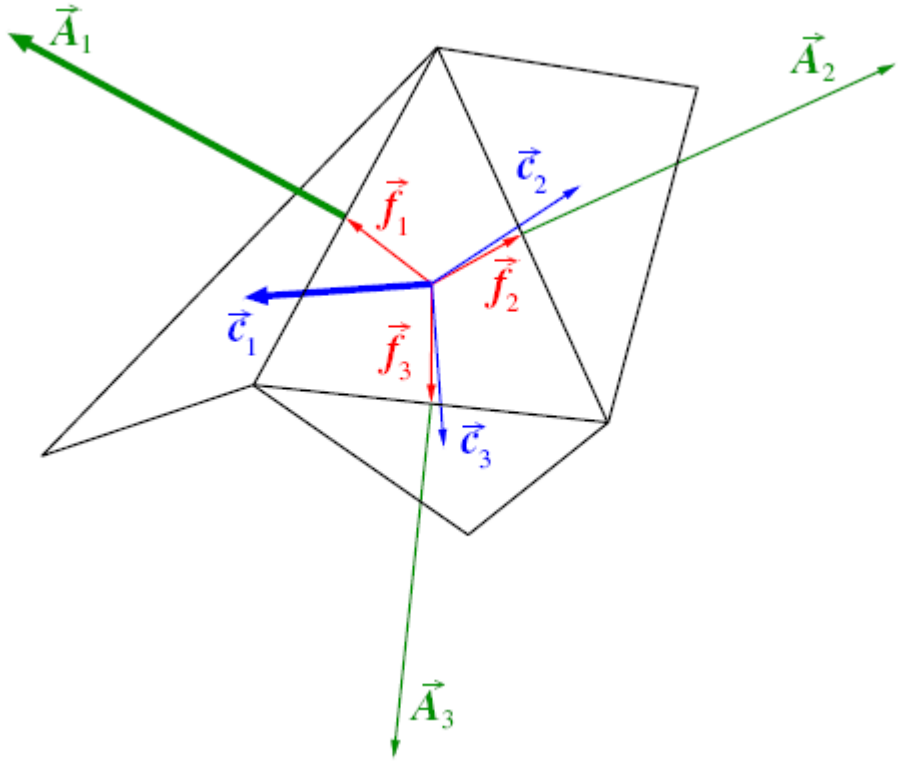
For a pyramid, the cell skewness will be the maximum skewness computed for any face. An ideal pyramid (skewness = 0) is one in which the 4 triangular faces are equilateral (and equiangular) and the quadrilateral base face is a square. The guidelines in the table above apply to the normalized equiangular skewness as well.

Orthogonal Quality

The range for orthogonal quality is 0-1, where a value of 0 is worst and a value of 1 is best.

The orthogonal quality for cells is computed using the face normal vector, $\vec{A}_i$ for each face, the vector from the cell centroid to the centroid of each of the adjacent cells, $\vec{c}_i$; and the vector from the cell centroid to each of the faces, $\vec{f}_i$. Figure 52: Vectors Used to Compute Orthogonal Quality for a Cell (p. 153) illustrates the vectors used to determine the orthogonal quality for a cell.

Figure 52: Vectors Used to Compute Orthogonal Quality for a Cell



For each face, the cosines of the angle between $\vec{A}_i$ and $\vec{c}_i$, and between $\vec{A}_i$ and $\vec{f}_i$, are calculated. The smallest calculated cosine value is the orthogonality of the cell. Finally, **Orthogonal Quality** depends on cell type:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For tetrahedral, prism, and pyramid cells, the Orthogonal Quality is the minimum of the orthogonality and (1 - cell skewness).
- For hexahedral and polyhedral cells, the Orthogonal Quality is the same as the orthogonality.

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 (a baffle), 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.

Orthogonal quality in the Meshing application is equivalent to Inverse Orthogonal Quality in Ansys Fluent Meshing, except that the scale is reversed:

Inverse Orthogonal Quality = 1 – Orthogonal Quality

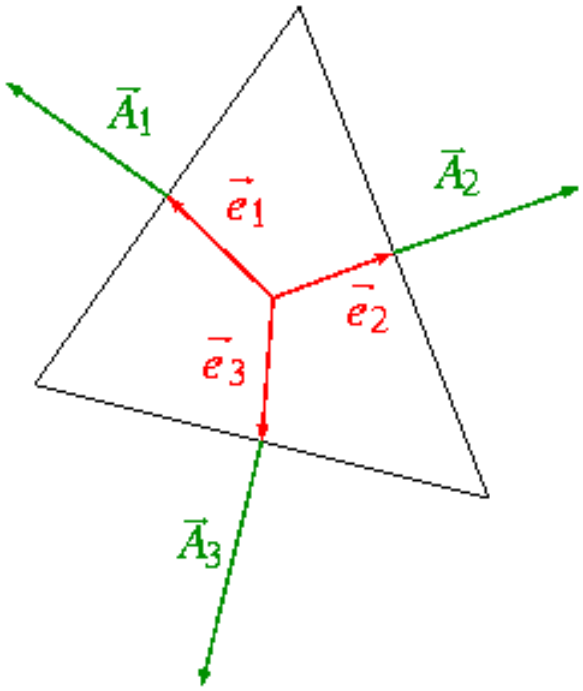
The orthogonal quality values may not correspond exactly with the inverse orthogonal quality values in Ansys Fluent because the computation depends on boundary conditions on internal surfaces (*WALL* vs. *INTERIOR*/*INTERNAL*/*FAN*/*RADIATOR*/*POROUS-JUMP*). Ansys Fluent may return different results which reflect the modified mesh topology on which CFD simulations are performed. The **Tetrahedrons** assembly algorithm uses only traditional element types.

For more information about Inverse Orthogonal Quality, see Quality Measure.

In a similar way, orthogonal quality for faces is computed as the smallest cosine of the angle between the edge normal vector, $\vec{A}_i$ for each edge and the vector from the face centroid to the centroid of each edge, $\vec{e}_i$. Figure 53: Vectors Used to Compute Orthogonal Quality for a Face (p. 155) illustrates the vectors used to determine the orthogonal quality for a face.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 53: Vectors Used to Compute Orthogonal Quality for a Face



Characteristic Length (Autodyn)

Characteristic length (also sometimes called characteristic dimension) is used to compute the time step that satisfies the Courant-Friedrichs-Lewy (CFL) condition for a given analysis setup.

The CFL condition is of interest mostly in explicit dynamics and computational fluid dynamics analyses. It governs the maximum time step for which a solution will be stable, and it must be met for the solution to converge. The CFL condition can be expressed as follows:

$$\Delta t \leq f * \left[\frac{h}{c}\right]_{min}$$

where,

f = time step safety factor (commonly/default 0.9)

h = characteristic length

c = material sound speed

such that if you know the characteristic length and material sound speed, you can determine the time step safety factor.

As h decreases, so does the time step. The definition of h varies based on element type:

Element Type	Definition of Characteristic Length (h)
Hexahedral or wedge	The volume of the element divided by the square of the longest diagonal and scaled by $\frac{3}{\sqrt{2}}$

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element Type	Definition of Characteristic Length (*h*)
Tetrahedral	The minimum distance of any element node to its opposing element face

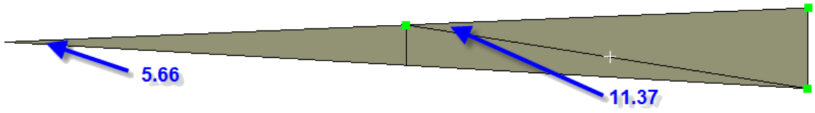
Characteristic Length (Autodyn) calculated by the Mesh Metrics and the User Defined Function are the same.

Note:

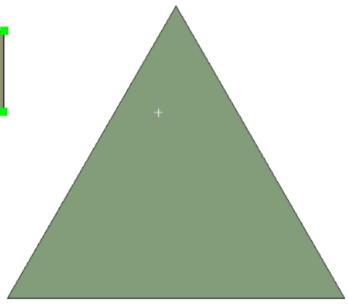
Characteristic Length (Autodyn) is available in the Mesh Details view only when the **Characteristic Length (Autodyn)** option is set to **On** in **File > Options > Meshing > Mesh Quality**.

Minimum Tri Angle

Minimum Tri Angle is the smallest angle between the two edge elements of the triangular element. The allowed range of value for **Minimum Tri Angle** is from 0 to 60 degrees.



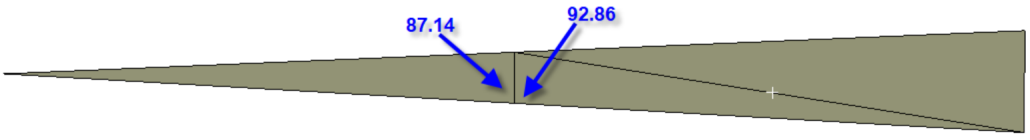
Mesh Metric	Minimum Tri Angle
Min	5.6614
Max	11.379
Average	7.5861
Standard Deviation	3.285



Mesh Metric	Minimum Tri Angle
Min	60.
Max	60.
Average	60.
Standard Deviation	0.

Maximum Tri Angle

Maximum Tri Angle is the largest angle between the two edge elements of the triangular element. The allowed range of value for **Maximum Tri Angle** is from 60 to 180 degrees.



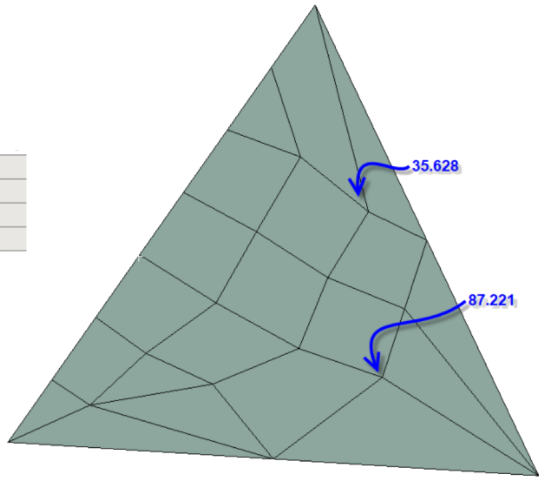
Mesh Metric	Maximum Tri Angle
Min	87.141
Max	92.859
Average	89.047
Standard Deviation	3.3011

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Minimum Quad Angle

Minimum Quad Angle is the smallest angle between the two edge elements of the quadrilateral mesh. The allowed range of value for **Minimum Quad Angle** is from 0 to 90 degrees.

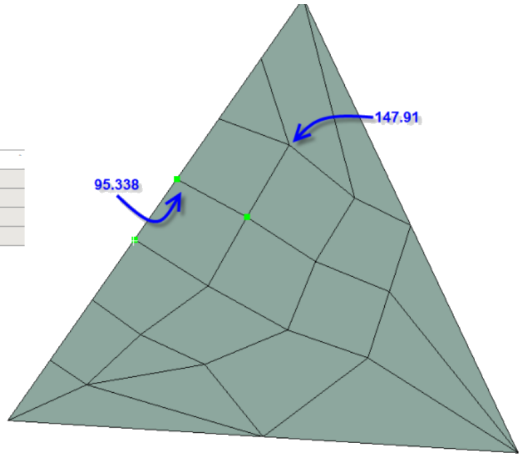
Mesh Metric	Minimum Quad Angle
Min	35.628
Max	87.221
Average	71.921
Standard Deviation	16.751



Maximum Quad Angle

Maximum Quad Angle is the largest angle between the two edge elements of the quadrilateral mesh. The allowed range of value for **Maximum Quad Angle** is from 90 to 180 degrees.

Mesh Metric	Maximum Quad Angle
Min	95.338
Max	147.91
Average	111.19
Standard Deviation	18.762

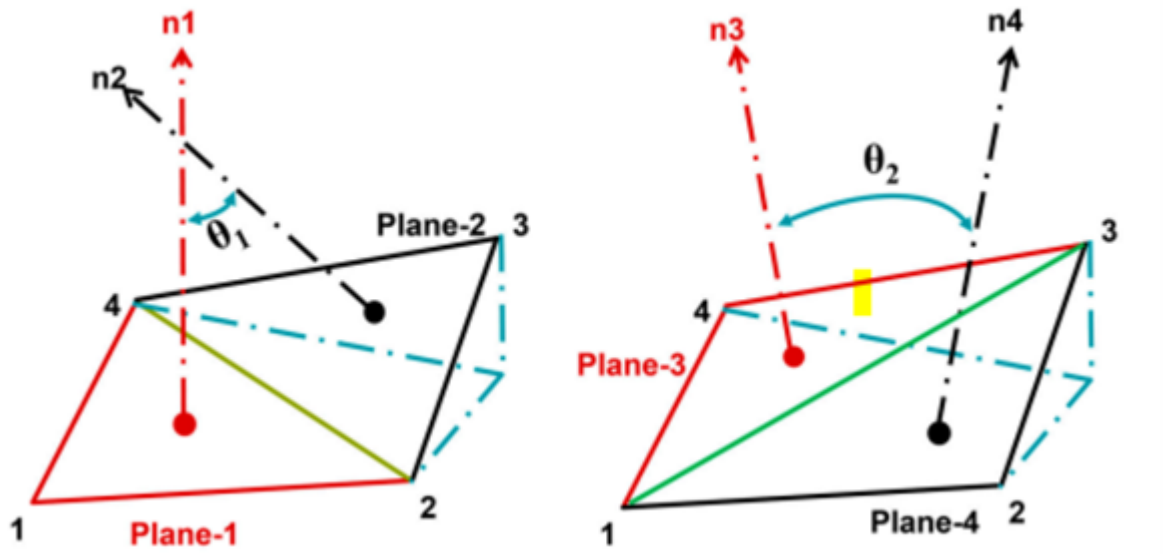


Warping Angle

Warping Angle is the measure of angular deviation of a quad element from the plane. The quad is divided into two triangles along the diagonal and the angle between the normals of the triangles are measured. The allowed range of value for **Warping Angle** is from 0 to 90 degrees.

You can calculate **Warping Angle** from the faces of volume elements. Warping angle is applicable to quad faces only. Hence, you cannot compute warping angle for tet elements. You can compute warping angle for quad elements in sheet and any elements containing quad faces for solid bodies. A pyramid element has only one quad face for computing the warping angle. The worst warping angle of all the quad faces determines the warping angle for hex elements.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Here,

n1 is the normal for Plane-1(1,2,4)

n2 is the normal for Plane-2 (2,3,4)

n3 is the normal for Plane-3 (1,4,3)

n4 is the normal for Plane-4 (1,2,3)

θ_1 is the angle between **n1** and **n2** for Plane-1 and Plane-2.

θ_2 is the angle between **n3** and **n4** for Plane-3 and Plane-4.

Warping Angle is calculated as the **max(θ_1,θ_2)**. **Warping Angle** is measure of how far the two tri-angles deviate from being coplanar. A warpage angle of 0 denotes the quad is perfectly planar whereas larger angles denote more warping.

Tet Collapse

Tet Collapse is defined as the :

min{h/[1.24*sqrt(A)]}

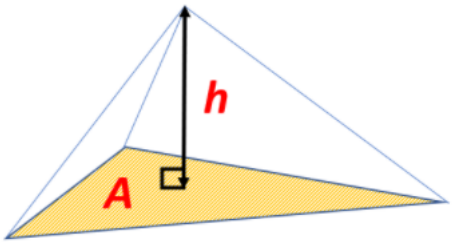
where,

h is the height of a corner node to opposite tet face

A is the area of the tet face

The allowed range of value for **Tet Collapse** is from 0 to 1.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Aspect Ratio (Height)

Aspect Ratio (Height) is the ratio of the longest edge of an element to its shortest edge or it is the shortest distance from the corner node to its opposite edge. For calculating the aspect ratio of 3D elements, each face of the element is treated as 2D element to determine the aspect ratio and the largest aspect ratio among the faces is considered as the aspect ratio for the 3D element.

The Aspect Ratio is calculated as follows:

AR= lmax/min(l,h*)

where,

lmax is the maximum edge length.

lmin is the minimum edge length.

l is the linear edge length

h is the height of the node to the non-connected face edges

hmin is the minimum face height

hmax is the maximum face height

h* is the normalized height

- For triangular elements, the normalized height is calculated as

  h*=2/sqrt(3)*h

- For quadrilateral elements, the normalized height is

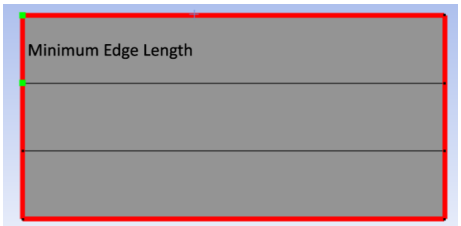
  h*=h

For tetrahedron elements, **Aspect Ratio (Height)** is the ratio of longest edge of an element to its shortest height measured from a node to its opposing face.

Minimum Element Edge Length

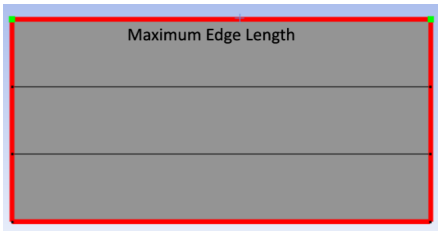
Minimum Element Edge Length is the shortest edge length of an element. For quadratic mesh, the length is the total of two segments from the start node to mid node and the end node to mid node.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Maximum Element Edge Length

Maximum Element Edge Length is the longest edge length of an element. For quadratic mesh, the length is the total of two segments from the start node to mid node and the end node to mid node.



Characteristic Length (LS-DYNA)

Characteristic Length (LS-DYNA) (also called characteristic dimension) is used to compute the time step that satisfies the Courant-Friedrichs-Lewy (CFL) condition for a given analysis setup. The CFL condition is of interest mostly in explicit dynamics and computational fluid dynamics analyses. It governs the maximum time step for which a solution will be stable.

Characteristic Length (LS-DYNA) has different definition for different element type.

$$L_c = \frac{V}{A_{max}}$$

For Tetrahedron elements,

$$L_c = \min(h)$$

where,

h is the height of a tetrahedron element from a node to its opposite face.

For Hex elements,

$$L_c = \frac{V}{\max(A)}$$

where,

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

V is the volume of the hex element

A is area of the element face

For Pyramid (5-node solid element), Explicit analysis does not use pyramid element and therefore no definition for this element type. To avoid graphic display issues for element summary, pyramid height is used for this element type.

$$L_c = \min(h)$$

where, h is the height of the apex node to the base of the pyramid. Here, the base quad face can be divided into 4 different triangles, the minimum height to these 4 triangles is used.

For Shell elements and Quad elements,

$$L_c = \frac{A}{\min(\max(L), \max(D))}$$

where,

A is element area

L is element edge length

D is the diagonal of quad element.

For Tri element,

$$L_c = \frac{2A}{\max(L)}$$

where,

A is element area

L is element edge length

For Beam element, the characteristic length is the linear length to the beam element.

Inflation Group

Inflation is useful for CFD boundary layer resolution, electromagnetic air gap resolution or resolving high stress concentrations for structures. Inflation is supported for the following mesh methods:

Volume Meshing:

- Patch Conforming (p. 208)
- Patch Independent (p. 210)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Sweep (p. 235)

> **Note:**
>
> Inflation is supported for the sweep mesh method only when the **Src/Trg Selection** option is set to either **Manual Source** or **Manual Source and Target**.

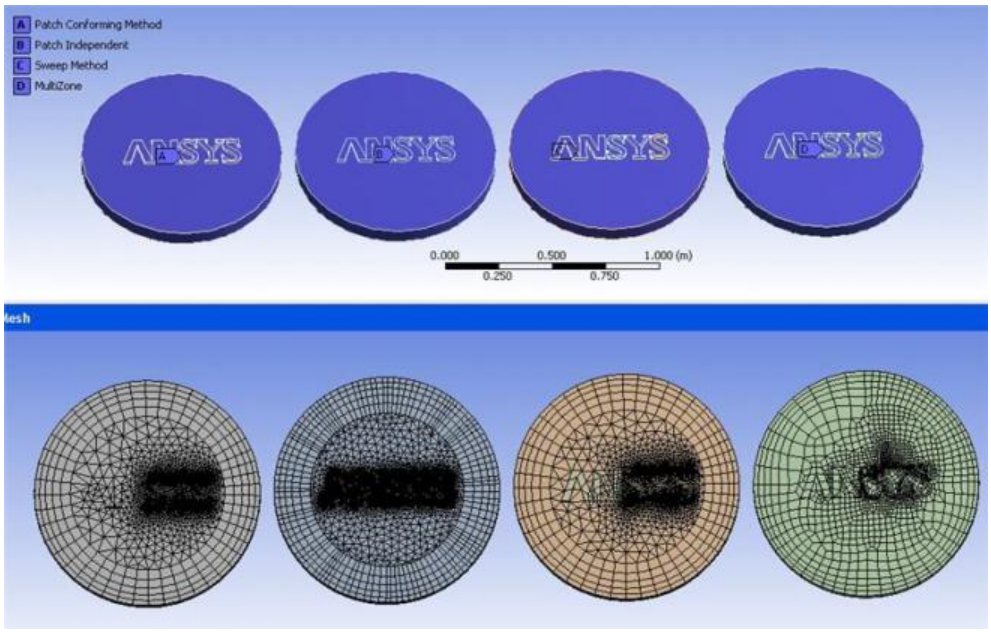
- MultiZone (p. 242)

Surface Meshing:

- Quad Dominant (p. 282)
- All Triangles (p. 284)
- MultiZone Quad/Tri (p. 284)

Figure 54: Inflation into Volume Mesh Methods (p. 162) illustrates inflation into the patch conforming, patch independent, sweep, and MultiZone mesh methods respectively.

Figure 54: Inflation into Volume Mesh Methods



Defining Global Inflation Controls

The **Inflation** group of global mesh controls appears in the Details View when the **Mesh** object is selected in the Tree Outline. The options in the **Inflation** group provide global control over all inflation boundaries.

In most cases, the controls in the **Inflation** group apply to both 3D and 2D inflation. Additional information that is specific to 2D inflation is noted where applicable.

Basic options include:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use Automatic Inflation
Inflation Option
Transition Ratio
Maximum Layers
Growth Rate
Number of Layers
Maximum Thickness
First Layer Height
First Aspect Ratio
Aspect Ratio (Base/Height)
Inflation Algorithm
Inflation Element Type
View Advanced Options

Defining Local Inflation Controls

In addition to setting global inflation controls, you can use local (scoped) inflation controls to apply inflation to specific boundaries. In most cases, the values that you set globally will be populated to the local inflation controls. If you subsequently make changes to the local inflation settings, the local settings will override the global settings. For details, refer to Inflation Control (p. 345).

Inflation and Mesh Method Controls

For steps to follow to assign inflation depending on the selected mesh method, refer to Inflation Controls (p. 453). For general information on applying inflation controls in combination with the various mesh method controls, refer to Meshing: Mesh Control Interaction Tables (p. 475).

Use Automatic Inflation

You can set the **Use Automatic Inflation** control so that inflation boundaries are selected automatically depending on whether or not they are members of Named Selections groups. The following options are available:

None
Program Controlled
All Faces in Chosen Named Selection

Remember the following information when using inflation:

- Automatic inflation is supported only for 3D inflation on volume models. It is not supported for 2D inflation on shell models. You cannot select Program Controlled (p. 164) or All Faces in Chosen Named Selection (p. 165) for the **Use Automatic Inflation** option to mesh a 2D model. To apply 2D inflation on a shell model, use local inflation mesh controls (p. 345) instead.
- In the following scenarios, using inflation results in automatic suppression of the refinement (p. 313) control:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - When automatic inflation (either Program Controlled (p. 164) or All Faces in Chosen Named Selection (p. 165)) is used with refinement in the same model
  - When local inflation (p. 345) is used with refinement in the same body or in the same part

- In general, using a mixture of local inflation and automatic inflation within the same part is *not recommended*. When you mix local and automatic inflation, the local inflation settings will be used for the bodies to which they are applied. The automatic inflation settings will create inflation *only* on those bodies that *do not* have local inflation applied to them. The automatic inflation settings will ignore all bodies and all attached faces to which local inflation settings are applied.

None

If you select **None**, inflation boundaries are not selected globally. Instead, the inflation boundaries that you identify using the local inflation mesh controls (p. 345) are used. This is the default.

Program Controlled

Program Controlled inflation operates differently depending on whether meshing occurs at the part/body level or assembly level.

Note:

Program Controlled inflation is only supported for 3D models.

Program Controlled Inflation for Part/Body Level Meshing

If you are using a mesh method that operates at the part/body level and you select **Program Controlled** inflation, all faces in the model are selected to be inflation boundaries, except for the following:

- Faces in Named Selection(s)

  Note:

  - By default, faces in Named Selections are not selected to be inflation boundaries when **Use Automatic Inflation** is set to **Program Controlled**. However, you can select specific Named Selections to be included in **Program Controlled** inflation. For details, see the discussion of Named Selections and **Program Controlled** inflation in the Mechanical help.
  - When you select **Use Automatic Inflation** as **Program Controlled** and add or remove a named selection, the application assumes the scoped faces have changed. Hence, the mesh becomes obsolete and must be regenerated.

- Faces in Contact region(s)
- Faces in Symmetry definition

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Faces that belong to a part/body that has a mesh method defined on it that does not support 3D inflation definitions (mesh method is set to Sweep (p. 235) or Hex Dominant (p. 233))
- Faces in sheet bodies
- Faces on bodies that have manual inflation controls

When **Program Controlled** automatic inflation is used for part or body level meshing, the mesher inflates in the following manner:

- For single body parts, the face always inflates into the body.
- For multibody parts with a mix of fluid and solid parts:
  - Faces on fluid region inflate into fluid region, solid region is not inflated.
  - Faces on parts of the same material are not inflated.
- For parts of the same material, shared faces are not inflated.

The manner in which inflation is applied depends on values that you enter for the following options:

- Inflation Option (p. 166)
- Inflation Algorithm (p. 170)
- Inflation Element Type (p. 174)
- View Advanced Options (p. 174)

Note:

When **Program Controlled** automatic inflation is being used for part/body level meshing, you can view the surfaces that have been selected for inflation by using the **Show Program Controlled Inflation Surfaces** (p. 536) feature.

All Faces in Chosen Named Selection

If you select **All Faces in Chosen Named Selection**, a **Named Selection** field is displayed to let you scope inflation to the Named Selection. The manner in which inflation is applied to the Named Selections group depends on values that you enter for the following options:

- Inflation Option (p. 166)
- Inflation Algorithm (p. 170)
- Inflation Element Type (p. 174)
- View Advanced Options (p. 174)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inflation Option

The **Inflation Option** settings determine the heights of the inflation layers. The following options are available:

- **Smooth Transition** - This is the default. The **Smooth Transition** option uses the local tetrahedral element size to compute each local initial height and total height so that the rate of volume change is smooth. Each triangle that is being inflated will have an initial height that is computed with respect to its area, averaged at the nodes. This means that for a uniform mesh, the initial heights will be roughly the same, while for a varying mesh, the initial heights will vary.

  The computations used for prism layer growth are as follows:

  – The following value is computed at each node on the prism base:

    Height of last prism (H) = Transition Ratio (p. 168) * average edge length

  – The height of the first layer (h) is computed using the following formula, where g = Growth Rate (p. 169), n = Number of Layers (p. 169) and H = Height of last prism:

    H = h * (g ^ (n-1))

  Increasing the value of the **Growth Rate** control reduces the total height of the inflation layer. The total height approaches an asymptotic value with respect to the number of inflation layers.

  For details about the additional controls that appear when **Smooth Transition** is selected, refer to the descriptions of the Transition Ratio (p. 168), Maximum Layers (p. 169), and Growth Rate (p. 169) controls.

  Note:

  The **Smooth Transition** option works differently for the **MultiZone** mesh method. See MultiZone Support for Inflation (p. 440) for details.

- **Total Thickness** - The **Total Thickness** option creates constant inflation layers using the values of the **Number of Layers** and **Growth Rate** controls to obtain a total thickness as defined by the value of the **Maximum Thickness** control. Unlike inflation with the **Smooth Transition** option, with the **Total Thickness** option the thickness of the first inflation layer and each following layer is constant.

  For details about the additional controls that appear when **Total Thickness** is selected, refer to the descriptions of the Number of Layers (p. 169), Growth Rate (p. 169), and Maximum Thickness (p. 169) controls.

- **First Layer Thickness** - The **First Layer Thickness** option creates constant inflation layers using the values of the **First Layer Height**, **Maximum Layers**, and **Growth Rate** controls to generate the inflation mesh. Unlike inflation with the **Smooth Transition** option, with the **First Layer Thickness** option the thickness of the first inflation layer and each following layer is constant.

  For details about the additional controls that appear when **First Layer Thickness** is selected, refer to the descriptions of the First Layer Height (p. 169), Maximum Layers (p. 169), and Growth Rate (p. 169) controls.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **First Aspect Ratio** - The **First Aspect Ratio** option creates inflation layers using the values of the **First Aspect Ratio**, **Maximum Layers**, and **Growth Rate** controls to generate the inflation mesh.

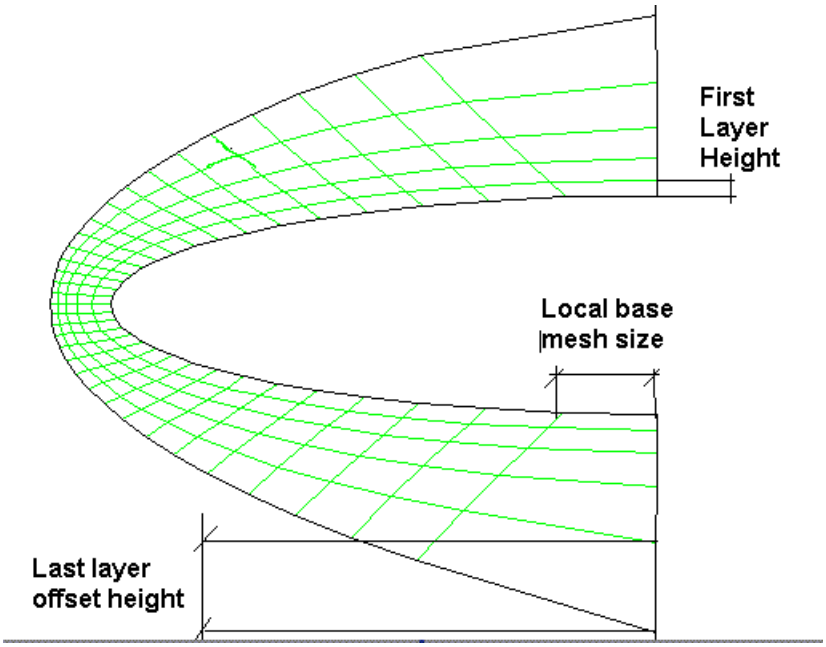
  Post inflation (p. 173) is not supported when **Inflation Option** is **First Aspect Ratio**.

  For details about the additional controls that appear when **First Aspect Ratio** is selected, refer to the descriptions of the First Aspect Ratio (p. 170), Maximum Layers (p. 169), and Growth Rate (p. 169) controls.

- **Last Aspect Ratio** - The **Last Aspect Ratio** option creates inflation layers using the values of the **First Layer Height**, **Maximum Layers**, and **Aspect Ratio (Base/Height)** controls to generate the inflation mesh.

  Figure 55: Last Aspect Ratio Option (p. 167) illustrates this option. With the **Last Aspect Ratio** method, the **First Layer Height** is specified. The offset height for the last layer is calculated from the local base mesh size and specified **Aspect Ratio (base/height)**. For example, if you specify a value of 3 for **Aspect Ratio (base/height)**, the offset height of the last layer will be the local base mesh size divided by 3. The local growth rate is calculated using **Maximum layers** to create exponential growth through the intermediate layers.

  Figure 55: Last Aspect Ratio Option



  Post inflation (p. 173) is not supported when **Inflation Option** is **Last Aspect Ratio**.

  For details about the additional controls that appear when **Last Aspect Ratio** is selected, refer to the descriptions of the First Layer Height (p. 169), Maximum Layers (p. 169), and Aspect Ratio (Base/Height) (p. 170) controls.

Note:

For bodies meshed with the **Body Fitted Cartesian** method, the Inflation option cannot be changed. If **Physics Preference** is set to **CFD**, then three inflation layers are created

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

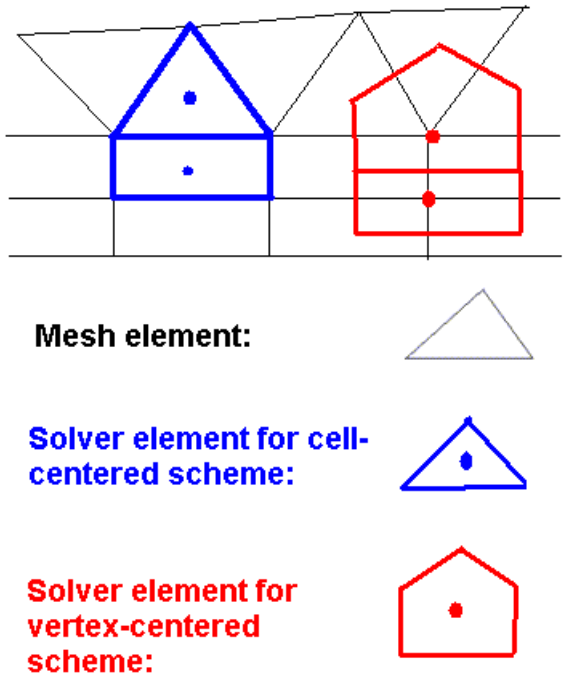
with total thickness proportional to **Element Size**. For any other physics preference, a single inflation layer is created with thickness proportional to element size.

Transition Ratio

In basic terms, the **Transition Ratio** control determines the rate at which adjacent elements grow. It is the volume-based size change between the last layer of elements in the inflation layer and the first elements in the tetrahedron region. The value of **Transition Ratio** is an ideal value and should produce accurate size change for inflation from a planar boundary. However, be aware that areas of strong curvature will introduce an inaccuracy into the size change. If proximity (p. 105) detection is activated, elements in proximity and elements with prism aspect ratios meeting the value defined by the Maximum Height over Base (p. 177) control will ignore this transition ratio.

The **Transition Ratio** control is applicable only when Inflation Option (p. 166) is **Smooth Transition**. Valid values for **Transition Ratio** are from **0** to **1**. When Physics Preference (p. 96) is set to **CFD** and Solver Preference (p. 99) is set to **CFX**, the default for **Transition Ratio** is **0.77**. For all other physics preferences, including **CFD** when **Solver Preference** is set to either **Fluent** or **Polyflow Classic**, the default is **0.272**.

The reason for this difference is because the **Fluent** and **Polyflow Classic** solvers use a cell-centered scheme for transition ratio. In a cell-centered scheme, the fluid flow variables are allocated at the center of the computational cell, and the mesh-element is the same as the solver-element. In contrast, the **CFX** solver uses a vertex-centered scheme. In a vertex-centered scheme, the fluid flow variables are stored at the cell vertex, and the solver-element is a "dual" of the mesh-element. This means that the vertex of the mesh-element *is* the center of the solver-element. The difference between the two schemes is illustrated in the figure below.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maximum Layers

The **Maximum Layers** control determines the maximum number of inflation layers to be created in the mesh. Valid values are from **1** to **1000**. The default is **5** for solid bodies and **2** for surface bodies.

If you set a different number of maximum layers on adjacent regions, stair stepping will be used between the regions.

The **Maximum Layers** control is applicable only when Inflation Option (p. 166) is **Smooth Transition**, **First Layer Thickness**, **First Aspect Ratio**, or **Last Aspect Ratio**.

Note:

For **Explicit** physics preference, the default value is 1.

Growth Rate

The **Growth Rate** control determines the relative thickness of adjacent inflation layers. As you move away from the face to which the inflation control is applied, each successive layer is approximately one growth rate factor thicker than the previous one. Valid values are from **0.1** to **5.0**. The default is **1.2**.

The **Growth Rate** control is applicable only when Inflation Option (p. 166) is **Smooth Transition**, **First Layer Thickness**, **Total Thickness**, or **First Aspect Ratio**.

Number of Layers

The **Number of Layers** control determines the actual number of inflation layers in the mesh, except in places where layers are removed locally for reasons of improving mesh quality (for example, in areas where inflation layers would otherwise collide with each other). Valid values are from **1** to **1000**. The default is **5** for solid bodies and **2** for surface bodies.

If you set a different number of layers on adjacent regions, stair stepping will be used between the regions.

The **Number of Layers** control is applicable only when Inflation Option (p. 166) is **Total Thickness**.

Maximum Thickness

The **Maximum Thickness** control determines the desired thickness of the inflation layer. You must enter a value for this control, and it must be greater than 0.

The **Maximum Thickness** control is applicable only when Inflation Option (p. 166) is **Total Thickness**.

First Layer Height

The **First Layer Height** control determines the height of the first inflation layer. This first inflation layer consists of a single layer of prism elements that is formed against the faces of the inflation boundary. You must enter a value for this control, and it must be greater than 0.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **First Layer Height** control is applicable only when Inflation Option (p. 166) is **First Layer Thickness** or **Last Aspect Ratio**.

First Aspect Ratio

By choosing the **First Aspect Ratio** option for the **Inflation Option** control, you can control the heights of the inflation layers by defining the aspect ratio of the inflations that are extruded from the inflation base. The *aspect ratio* is defined as the ratio of the local inflation base size to the inflation layer height. Use the **First Aspect Ratio** control to specify the first aspect ratio to be used. Enter a value greater than **0**. The default is **5**.

The **First Aspect Ratio** control is applicable only when Inflation Option (p. 166) is **First Aspect Ratio**.

Aspect Ratio (Base/Height)

By choosing the **Last Aspect Ratio** option for the **Inflation Option** control, you can control the heights of the inflation layers by defining the aspect ratio of the inflations that are extruded from the inflation base. The *aspect ratio* is defined as the ratio of the local inflation base size to the inflation layer height. Use the **Aspect Ratio (Base/Height)** control to specify the aspect ratio to be used. Enter a value between **0.5** and **20**. The default is **1.5** when Solver Preference (p. 99) is **CFX**, and **3** when Solver Preference (p. 99) is **Fluent** or **Polyflow**.

The **Aspect Ratio (Base/Height)** control is applicable only when Inflation Option (p. 166) is **Last Aspect Ratio**.

Inflation Algorithm

The **Inflation Algorithm** control determines which inflation algorithm will be used. Options for **Inflation Algorithm** are **Pre** and **Post** and are dependent upon the selected mesh method.

Note:

Post Inflation is being deprecated and will be removed in future releases.

The following table shows which inflation algorithms are applicable to each mesh method. For information on how the inflation algorithm is handled when a combination of mesh methods is being used, see Interactions Between Mesh Methods (p. 475).

Mesh Method	Inflation Algorithm	
	Pre	Post
Patch Conforming Tetrahedrons (p. 208)	Yes, 3D	Yes, 3D
Patch Independent Tetrahedrons (p. 210)	N/A	Yes, 3D
Hex Dominant (p. 233)	N/A	N/A

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh Method	Inflation Algorithm	
	Pre	**Post**
Sweep (p. 235)	Yes, 2D. Occurs in the following manner: 1) source face is meshed with triangles, 2) inflation occurs on tri surface mesh, and 3) source is swept. Intervals on source and target are fixed.	N/A
MultiZone (p. 242)	The **Inflation Algorithm** displays as **Pre** but an O-grid-based algorithm specific to **MultiZone** is used. As with the **Pre** inflation algorithm, the mesh is inflated during the meshing process.	N/A
MultiZone Quad/Tri (p. 284)	The **Inflation Algorithm** displays as **Pre** but an O-grid-based algorithm specific to **MultiZone Quad/Tri** is used. As with the **Pre** inflation algorithm, the mesh is inflated during the meshing process.	Yes, 2D
Quad Dominant (p. 282)	Yes, 2D	N/A
All Triangles (p. 284)	Yes, 2D	N/A
Cartesian (p. 261)	The **Inflation Algorithm** displays as **Pre** but algorithm specific to **Body Fitted Cartesian** is used. As with the **Pre** inflation algorithm, the mesh is inflated during the meshing process. **Note:** • 3D only • If **Physics Preference** is **CFD**, then 3 layers are created. For other physics preferences, only one layer is created.	N/A

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pre

When **Pre** is selected, the surface mesh will be inflated first, and then the rest of the volume mesh will be generated. This is the default for all physics types.

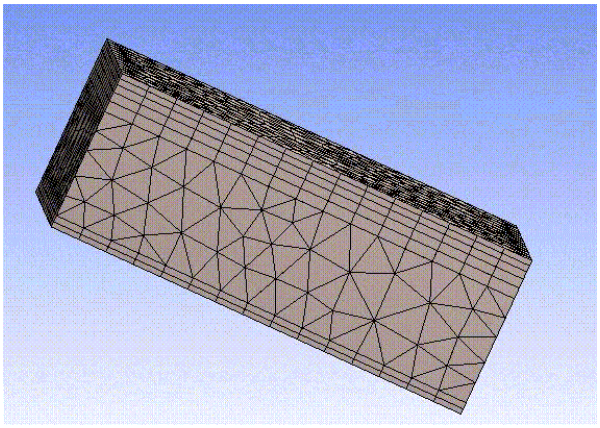
Remember the following information:

- **Inflation Algorithm** must be set to **Pre** for the Previewing Inflation (p. 534) feature to work.
- When Inflation Option (p. 166) is either **First Aspect Ratio** or **Last Aspect Ratio**, **Inflation Algorithm** is set to **Pre** and is read-only.
- Hard points are not supported and may be ignored for 3D Pre inflation.
- Hard edges are not supported and may be ignored for Pre inflation.
- In some cases an additional smoothing attempt will be performed automatically. Refer to Smoothing (p. 133) for details.
- Match controls (p. 334) on faces are supported with Pre inflation, regardless of whether inflation is set to Program Controlled (p. 164) or has been set through any global or local inflation definition. In contrast, match controls on edges are not supported with Pre inflation. Match controls (both faces and edges) are not supported with Post inflation (p. 173). For all these non-supported cases, Ansys Workbench automatically suppresses/disables the Match Control feature.
- For swept meshes with inflation (p. 454) and match control (p. 334), inflation is performed ahead of the match mesh and sweeping. This can affect the sizings on the match controls, which can in turn lead to meshing failure. Therefore, when using both match controls and inflation with sweeping, it might improve meshing robustness if you assign hard edge sizings (p. 300) to the high and low edges of the source face for the sweep.

Pre Inflation and Different Numbers of Layers on Adjacent Faces

The **Pre Inflation Algorithm** does not support the definition of different numbers of inflation layers on adjacent faces. For example, Figure 56: Different Numbers of Layers Are Respected (p. 172) shows a case in which different numbers of inflation layers have been specified on two faces. Since the faces are not connected, the different numbers of layers are respected.

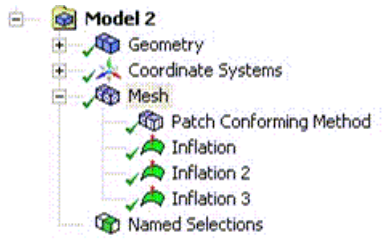
Figure 56: Different Numbers of Layers Are Respected



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

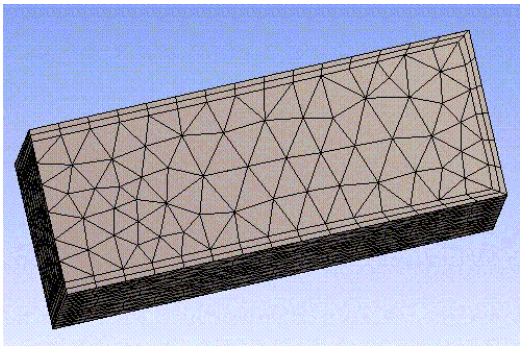
Now assume that the **Inflation** controls shown in Figure 57: Portion of Project Tree (p. 173) define four inflation layers for the **Inflation** control, five inflation layers for the **Inflation 2** control, and two inflation layers for the **Inflation 3** control, and that the controls are defined on adjacent faces.

Figure 57: Portion of Project Tree



In this case, although different numbers of inflation layers were defined, a two-layer (fewest number of layers defined) inflation mesh is generated as shown in Figure 58: Different Numbers of Layers Are Not Respected (p. 173).

Figure 58: Different Numbers of Layers Are Not Respected



Post

When **Post** is selected, a postprocessing technique that works after the tetrahedral mesh is generated is used. A benefit of this option is that the tetrahedral mesh does not have to be generated each time the inflation options are changed.

Remember the following information:

- Match controls (p. 334) on faces are supported with Pre inflation (p. 172), regardless of whether inflation is set to Program Controlled (p. 164) or has been set through any global or local inflation definition. In contrast, match controls on edges are not supported with Pre inflation. Match controls (both faces and edges) are not supported with Post inflation. For all these non-supported cases, Ansys Workbench automatically suppresses/disables the Match Control feature.
- Post inflation is not supported when there is a mixture of tetrahedron and non-tetrahedron mesh methods applied to the bodies in a multibody part. If you want to apply Post inflation to a multibody part, all bodies in the part must have a tetrahedron mesh method applied to them.
- Post inflation is not supported when Inflation Option (p. 166) is either **First Aspect Ratio** or **Last Aspect Ratio**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Refer to Using the Mesh Worksheet to Create a Selective Meshing History (p. 448) for information about how Post inflation operations are processed by the **Mesh** worksheet.

Inflation Element Type

Inflation Element Type determines the element type for creating inflation layer. The default value is **Wedges**. The available options are:

- **Wedges**: Creates inflation layers containing wedge elements.
- **Tetrahedrons**: Creates inflation layers containing tetrahedron elements.

View Advanced Options

The **View Advanced Options** control determines whether advanced inflation options appear in the Details View. Choices are **No** (default) and **Yes**. When this control is set to **Yes**, the following options are available:

Collision Avoidance

Maximum Height over Base

Growth Rate Type

Maximum Angle

Fillet Ratio

Use Post Smoothing

Smoothing Iterations

Note:

In addition to viewing and/or modifying the advanced inflation options in the Details View, you can view and/or modify them by using the **Options** dialog box (p. 389).

Collision Avoidance

The **Collision Avoidance** control determines the approach that is to be taken in areas of proximity to avoid collisions that may occur from marching inflated surface meshes from opposite sides into each other.

For 2D inflation, the **Collision Avoidance** control detects geometry limitations in faces that would otherwise cause inflation mesh to overlap or cross face boundaries, or result in a space that is so small that it leads to bad quality mesh for the remaining region of inflation mesh. When **Collision Avoidance** is set to **Layer Compression** or **Stair Stepping**, the value of the Gap Factor (p. 177) control, along with the local mesh size, will determine how much space will be adequate for the remaining region of inflation mesh.

Note:

- The option that you choose for **Collision Avoidance** is used *only in areas of proximity*. In areas of proximity, if the option is set to **Layer Compression**, layer compression is

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

performed. If it is set to **Stair Stepping**, stair stepping is performed. If it is set to **None**, no collision checking is performed. However, in all other problematic scenarios (for example, invalid normals, quality failure, bad surface mesh and so on), local stair stepping is performed regardless of which option you choose.

- When **Collision Avoidance** is set to **Layer Compression** and local stair stepping occurs after compression, poor quality pyramids may be introduced into the mesh. Because of this possibility, a warning message will appear whenever stair stepping occurs after compression. The message will not identify the location of the stair stepping. However, the location of the stair stepping with added pyramids often coincides with the location of the worst quality element (specifically, when considering the **Skewness** metric). For this reason, using the Meshing application's **Mesh Metric** (p. 133) feature to locate the worst quality element (based on **Skewness**) is also likely to locate the pyramids.
- The **Collision Avoidance** option is not used for **MultiZone** (p. 440) as the inflation layers are created within the blocking approach.

The following options are available:

- **None** - The **None** option does not check for layer collisions. Selecting this option speeds up inflation layer computation time. However, it can result in an invalid mesh and mesh failures. For these reasons, this option is not recommended.

  For 2D inflation, if a collision/proximity limitation is detected during layer creation, creation of inflation layers stops with the previous layer. (Inflation stops *completely*; contrast with **Stair Stepping** below.)

- **Layer Compression** - The **Layer Compression** option compresses inflation layers in areas of collision. In these areas, the defined heights and ratios are reduced to ensure the same number of layers throughout the entire inflation region. Generally, this option is best for avoiding the creation of pyramids in the mesh. **Layer Compression** is the default only when the **Physics Preference** (p. 96) is set to **CFD** and the **Solver Preference** (p. 99) is set to **Fluent**; otherwise, the default is **Stair Stepping**.

  For 2D inflation, if a collision/proximity limitation is detected during layer creation, inflation heights will shrink locally. If Fix First Layer (p. 176) is set to **Yes**, the First Layer Height (p. 169) will not be scaled.

  For details about the additional controls that appear when **Layer Compression** is selected, refer to the descriptions of the Fix First Layer (p. 176) and Gap Factor (p. 177) controls.

- **Stair Stepping** - Rather than compressing the prism layers, with **Stair Stepping** the prism layers are "stair stepped" in the proximity region to avoid collision and to maintain the gap defined by Gap Factor (p. 177). The **Stair Stepping** approach to inflation growth locally reduces inflation layers to avoid collisions, as well as bad quality elements in sharp or tight corners. The term "stair stepping" refers to the steps created between one layer and the next. Using this approach, special logic is used to fill the steps with pyramid and tetrahedron elements for prism steps, or prism, pyramid, and tetrahedron elements for hex steps. This special logic helps the mesher obtain a high-quality transition to the tetrahedral mesh. **Stair Stepping** is the default, unless the **Physics Preference** (p. 96) is set to **CFD** and the **Solver Preference** (p. 99) is set to **Fluent**, in which case the default is **Layer Compression**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For 2D inflation, if a collision/proximity limitation is detected during layer creation, creation of inflation layers stops *locally*. (Contrast with **None** above.)

An additional control, for **Gap Factor** (p. 177), appears when **Stair Stepping** is selected.

The figures below illustrate how the **Layer Compression** and **Stair Stepping** options differ.

Figure 59: Layer Compression vs. Stair Stepping Option (Full Mesh View)

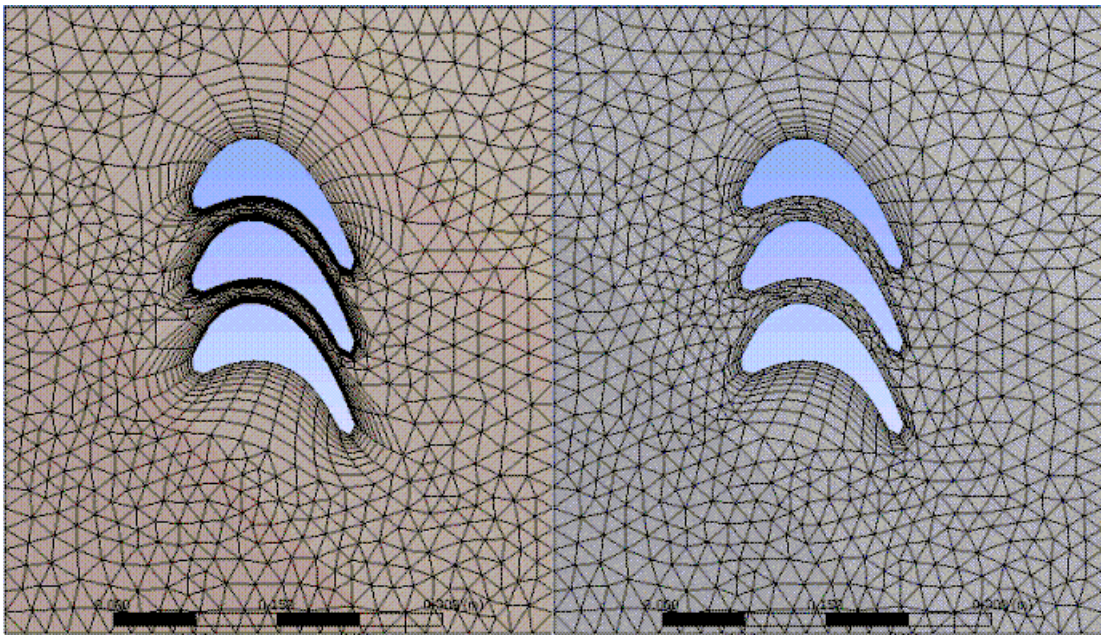
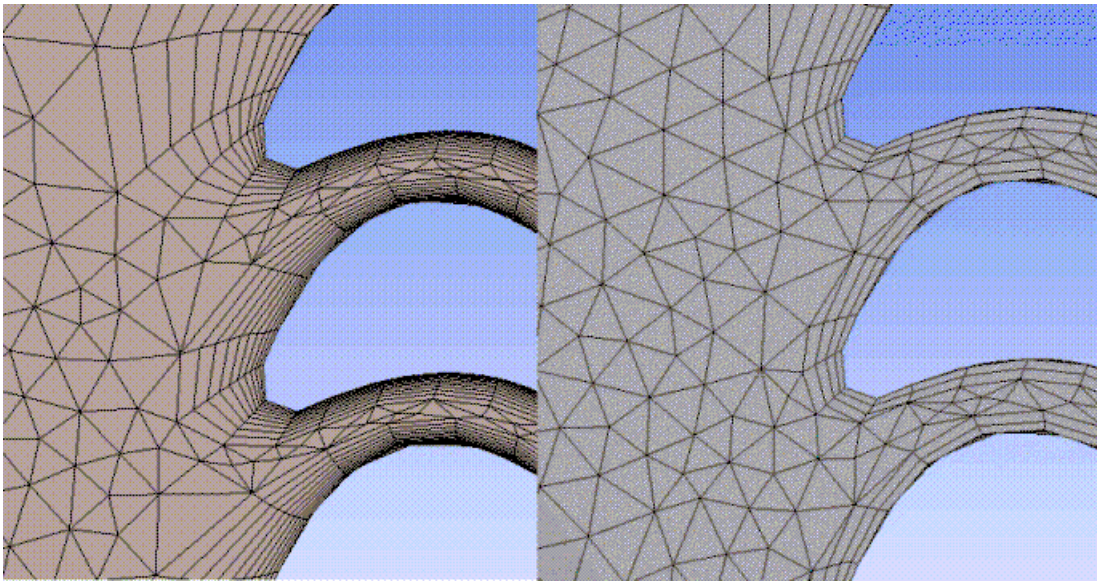


Figure 60: Layer Compression vs. Stair Stepping Option (Detail View)



Fix First Layer

The **Fix First Layer** control determines whether the heights or ratios of the first inflation layer will be modified to avoid collision. Valid values are **Yes** and **No**. The default is **No**. This option will not allow the value that is set for the First Layer Height (p. 169) control to be changed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Fix First Layer** control is applicable only when Collision Avoidance (p. 174) is **Layer Compression**.

Gap Factor

The **Gap Factor** control allows maintenance of the gap between intersecting prisms. Valid values are from **0** to **2**. The default is **0.5**. A value of **1** means a gap equal to the ideal tet cell height based on base face size in proximity to each other is maintained.

The **Gap Factor** control is applicable only when Collision Avoidance (p. 174) is **Layer Compression** or **Stair Stepping**

Note:

For **Explicit** physics preference, the default value is 0.1

.

Maximum Height over Base

The **Maximum Height over Base** control sets the maximum allowable prism aspect ratio (that is, the ratio of height over base of the base triangle). When the prism aspect ratio reaches this value, the height of the prisms stops growing. That is, new prisms continue to form, but the heights of the prisms will not increase. Valid values are from **0.1** to **5**. The default is **1.0**.

For 2D inflation, the **Maximum Height over Base** control helps to maintain a good size ratio. Once the inflation height is greater than the local mesh size multiplied by the value of **Maximum Height over Base**, the inflation height stops growing. New layers continue to form, but the heights of the inflation layers will not increase. New layers that form will be equal to the inflation base size multiplied by the value of **Maximum Height over Base**.

Growth Rate Type

The **Growth Rate Type** control determines the height of the inflation layers given the initial height and height ratio. The following options are available:

- **Geometric** - This is the default. With this option, the prism height of a particular layer is defined by $h*r^{(n-1)}$, where h = initial height, r = height ratio, and n = layer number. The total height at layer n is: $h(1–r^n)/(1–r)$.
- **Exponential** - With this option, the prism height of a particular layer is defined by $h*e^{(n-1)p}$, where h = initial height, p = exponent, and n = layer number.
- **Linear** - With this option, the prism height of a particular layer is defined by h(1+(n-1)(r-1)), where h = initial height, r = height ratio, and n = layer number. The total height at layer n is: nh((n-1)(r-1)+2)/2.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

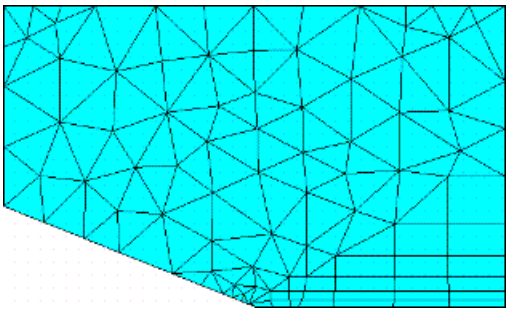
Maximum Angle

The **Maximum Angle** control determines prism layer growth around angles and when prisms will adhere (project) to adjacent surfaces/walls. If the inflated mesh involves extruding from one surface and not its neighbor, and the angle between the two surfaces is less than the specified value, the prisms (sides) will adhere (project) to the adjacent wall. Valid values are from **90** to **180** (degrees). Typically, a value between **120** and **180** is desirable. The default is **140**. Refer to the figures below for examples of maximum angle.

For 2D inflation, the **Maximum Angle** control determines whether an edge that is adjacent to an inflation edge can be imprinted with inflation mesh. If the angle between the two edges is smaller than the value of **Maximum Angle**, the inflation mesh will be imprinted on the adjacent edge. On the other hand, if the angle between the two edges is larger than the value of **Maximum Angle**, the inflation mesh will not be imprinted on the adjacent edge.

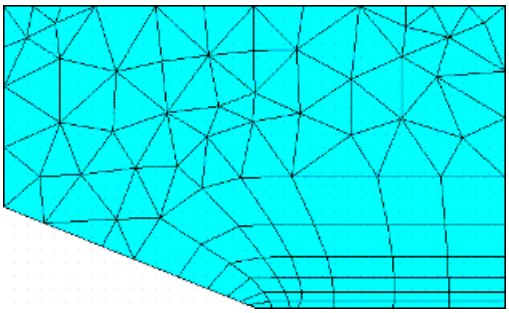
In Figure 61: Maximum Angle = 140 (p. 178), the angle between the planes is 158.2 (21.8) degrees. Since the maximum angle is less than the angle between the walls, the prism layers are capped with pyramids.

Figure 61: Maximum Angle = 140



In Figure 62: Maximum Angle = 180 (p. 178), the maximum angle exceeds the separation angle between the surfaces, so the prism remains attached to the adjacent surface.

Figure 62: Maximum Angle = 180



Fillet Ratio

The **Fillet Ratio** control determines whether a fillet proportional to the total height of a prism element will be created when a prism element is generated in the corner zone of a tetrahedral mesh. Creating a fillet proportional to the total height of the prism makes it possible to control the

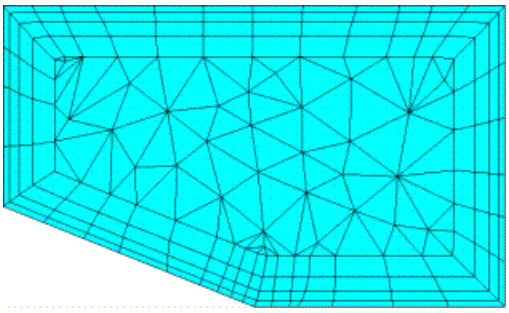
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

smoothness of the prism layer. Valid values are from **0** to **1** (decimal values are allowed). A value of **0** means no fillets. The default is **1**. Refer to the figures below for examples of fillet ratio.

Note:

For meshing corners with angles less than 60 degrees, there may not be space for a fillet.

Figure 63: Fillet Ratio = 0.0



In the figure below, the radius of the inner prism fillet is 0.5 times the height of the total prism thickness.

Figure 64: Fillet Ratio = 0.5

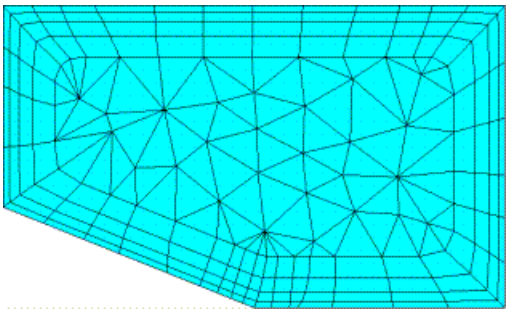
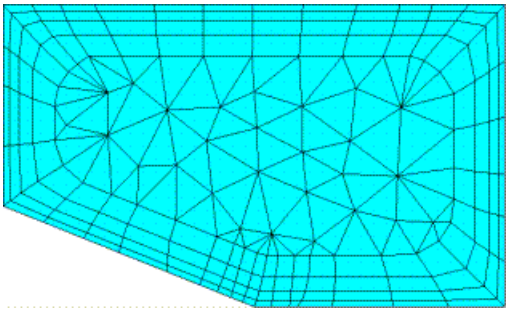


Figure 65: Fillet Ratio = 1.0



Use Post Smoothing

The **Use Post Smoothing** control determines whether post-inflation smoothing will be performed. Smoothing attempts to improve element quality by moving the locations of nodes with respect to

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

surrounding nodes and elements. Valid values are **Yes** and **No**. The default is **Yes**. When this control is set to **Yes**, the Smoothing Iterations (p. 180) control appears in the Details View.

Smoothing Iterations

The **Smoothing Iterations** control determines the number of post-inflation smoothing iterations that will be performed to improve the mesh. Valid values are from **1** to **20**. The default is 5.

The **Smoothing Iterations** control is applicable only when Use Post Smoothing (p. 179) is **Yes**.

Advanced Group

The **Advanced** group allows you to control these options:

Number of CPUs for Parallel Part Meshing

Straight Sided Elements

Rigid Body Behavior

Triangle Surface Mesher

Topology Checking

Pinch

Loop Removal

Number of CPUs for Parallel Part Meshing

Number of CPUs for Parallel Part Meshing: Sets the number of processors to be used for parallel part meshing. Using the default for specifying multiple processors will enhance meshing performance on geometries with multiple parts. For parallel part meshing, the default is set to **Program Controlled** or **0**. This instructs the mesher to use all available CPU cores. The Default setting inherently limits 2 GB memory per CPU core. An explicit value can be specified between **0** and **256**, where **0** is the default. Refer to Parallel Part Meshing (p. 472) for more details.

Straight Sided Elements

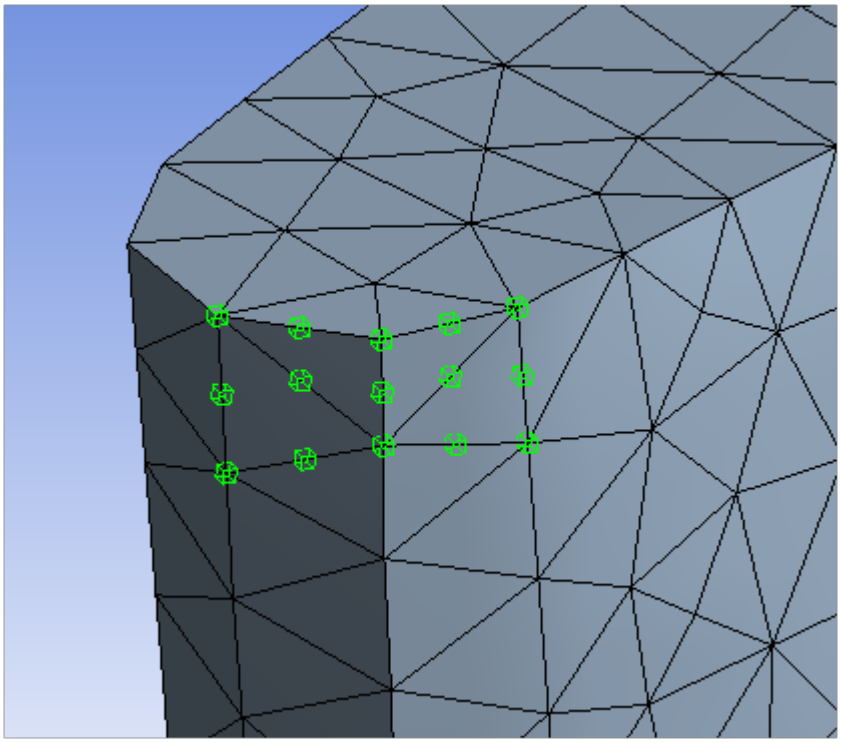
The **Straight Sided Elements** option (which is displayed when the model includes an enclosure from the DesignModeler application), specifies meshing to straight edge elements when set to **Yes**. You must set this option to **Yes** for Electromagnetic simulations.

This option may affect the placement of midside nodes if the **Element Order** option is set to **Quadratic**.

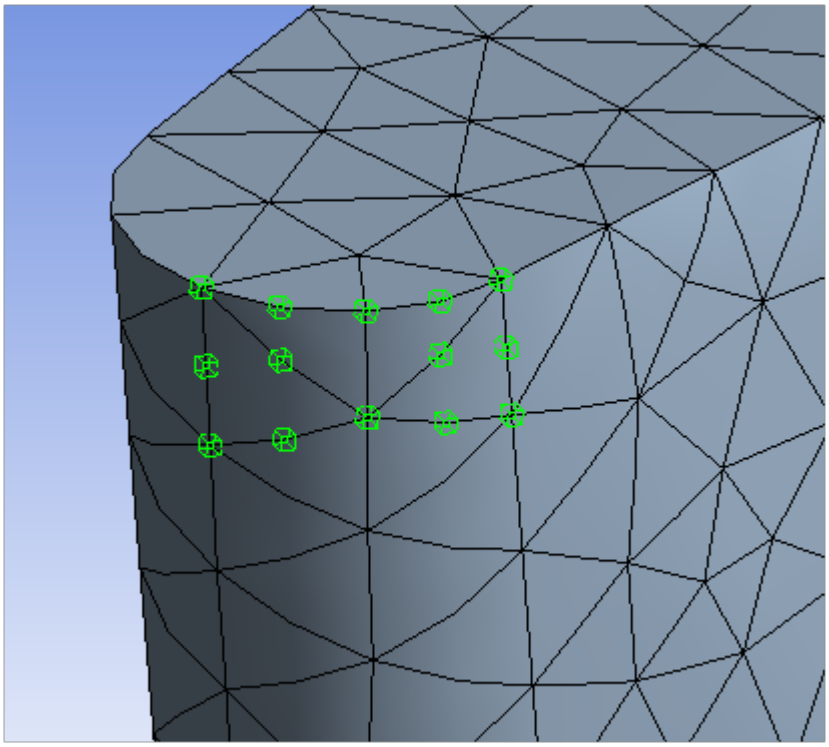
Example 3: Midside Node Placement with Straight Sided Elements

In this example, the mesh is generated with straight sided elements with midside nodes.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Generating the mesh without straight sided elements results in midside nodes that capture the curvature of the model. If an element edge corresponds to geometry that is curved, the element's edge will be curved. Likewise, if an element edge corresponds to geometry that is straight, the element's edge will be straight.



Straight Sided Elements is not available if the **Element Order** option is set to **Linear**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rigid Body Behavior

The **Rigid Body Behavior** option determines whether a full mesh is generated for a rigid body, rather than a surface contact mesh. **Rigid Body Behavior** is applicable to all body types. Valid values for **Rigid Body Behavior** are **Dimensionally Reduced** (generate surface contact mesh only) and **Full Mesh** (generate full mesh). The default is **Dimensionally Reduced** unless the **Physics Preference** (p. 96) is set to **Explicit**. For more information, refer to Rigid Body Meshing (p. 463).

For Explicit, the default behavior is **Full Mesh**, but **Dimensionally Reduced** is also available. The differences between the two options when the **Physics Preference** is set to **Explicit** are highlighted in the table below.

Explicit behavior	Full Mesh (default)	Dimensionally Reduced
Rigid properties	From Mesh, computed in Explicit Solver	From Geometry, computed in Mechanical
Mesh	Tets/hexas for solids, Quad/tria for shells	Quad/tria for both solids and shells

Triangle Surface Mesher

The **Triangle Surface Mesher** control determines which triangle surface meshing strategy will be used by patch conforming meshers. In general, the advancing front algorithm provides a smoother size variation and better results for skewness (p. 151) and orthogonal quality (p. 153). The following options are available:

- **Program Controlled** - This is the default. The mesher determines whether to use the Delaunay or advancing front algorithm based on a variety of factors such as surface type, face topology, and defeatured boundaries.
- **Advancing Front** - The mesher uses advancing front as its primary algorithm, but falls back to Delaunay if problems occur.

> **Note:**
>
> If the mesher falls back to Delaunay, the edge mesh from the advancing front algorithm could still be used and in some rare cases could lead to meshing failures. Switching to the **Program Controlled** option could fix the issue as the starting edge mesh could be better.

The figures below illustrate the difference between the **Program Controlled** and **Advancing Front** options.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 66: Triangle Surface Mesher = Program Controlled

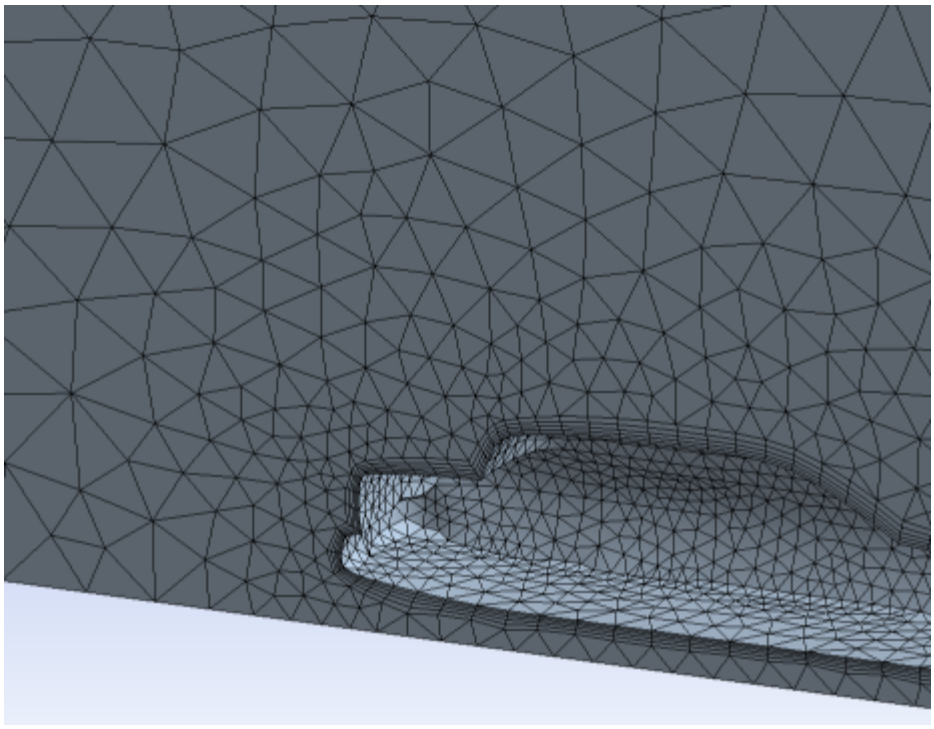
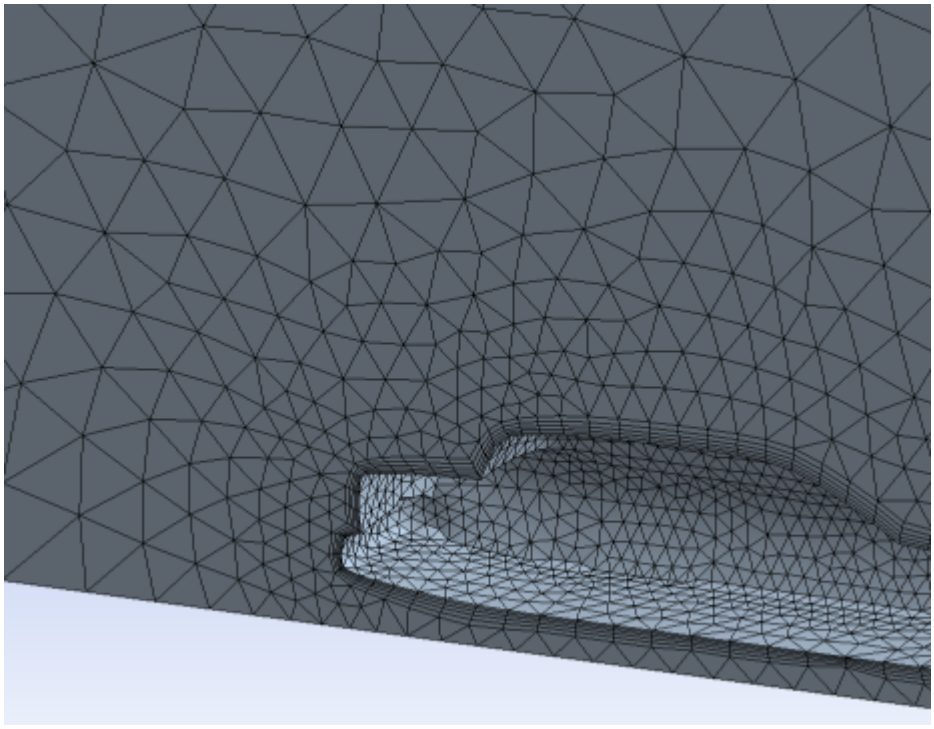


Figure 67: Triangle Surface Mesher = Advancing Front



Topology Checking

The **Topology Checking** option controls what happens when a user scopes an object (such as loads, boundary conditions, Named selections and so on) to geometry (bodies, faces, edges, and vertices)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

after the mesh has been generated. If **Topology Checking** is set to **Yes** (default), the software will check to see if the scoped geometry has mesh properly associated to it. If the associations are incorrect, the scoping of the object will force the mesh to be out of date. The mesh would need to be re-generated to get proper associations. If the associations are correct, the scoping is performed without any change to the mesh and the mesh stays up to date. Set **Topology Checking** to **No** to avoid the checks and always keep the mesh up to date.

Note:

- **Topology Checking** is defaulted to **Yes** as it is important to have proper associations between geometry and mesh in order to properly transfer the loads to the solver.
- The best practice is to define all loads and boundary conditions prior to meshing so that the topology is properly captured during meshing.

You can override the default of the **Topology Checking** control by setting the **Topology Checking** option on the **Options** panel (p. 389).

The following sections describe how the mesher handles Topology Protection:

Protecting Topology Defined Prior to Meshing

Protecting Topology Post Meshing

Protecting Topology Defined Prior to Meshing

Protected topology will result in having a better association between the mesh topology (nodes, faces, elements) and the geometry features (topology).That is,

- A geometry vertex should have a node associated to it.
- A geometry edge should have mesh nodes/edges associated to it.
- A geometry faces should have mesh nodes/faces/elements associated to it.
- A geometry body should have mesh nodes/elements associated to it.
- When you protect a group of faces, the mesher protects only the outer boundary of the group but it still defeatures the interior edges within the face group.

Since boundary conditions are ultimately applied to the mesh, it is important to have proper associations to get proper loading. Thus, if protected topologies are ignored, an error or warning message may be issued by the mesher.

There are two levels of topology protection, namely **hard** and **soft**.

- Hard protected topology can be defined by scoping Named Selections and/or Contact region objects to geometry (bodies, faces, edges, and vertices) and setting the **Protected** option to **Yes**. This instructs the mesher to give higher priority to those geometry features. Thus, during mesh generation, the outer boundaries of a collection of hard protected topologies will be maintained. The mesher will return an error state and message if the outer boundaries of the hard protected topologies cannot be protected.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Soft protected topology can be defined by scoping Named Selections to geometry (bodies, faces, edges, and vertices) and setting the **Protected** option to **Program Controlled**. For soft protected topologies, mesh-based defeaturing takes priority. Thus, the outer boundaries of soft protected topologies may be altered by the mesher. In such situations, mesher will return a warning message. You can right-click the warning message and use the **Show Problematic Geometry** option to visualize problematic geometries. You should verify that the mesh is acceptable in case of defeaturing of outer boundaries of the soft protected topologies.

Setting up Hard Protected Topology

- For contacts, set **Protected** to **Yes**.
- For Named Selections, set **Protected** to **Yes**.

Setting up Soft Protected Topology

For Named Selections, set **Protected** to **Program Controlled**. When **Program Controlled** is selected, the scoped object(s) will be considered as soft protected. The scoped objects will not receive additional protection by the mesher, even if the Named Selection is used for boundary conditions, symmetry, other types of loads for the solver, as well as match controls and hard sizing controls for meshing. Mesh-based defeaturing will have priority and the outer boundaries of these topologies may be altered by the mesher.

Resuming Legacy Databases (prior to Release 19.0)

If you resume a legacy database (prior to Release 19.0) with protected topologies:

- Contacts are resumed with **Protected** set to **No**.
- Named selections are resumed with **Protected** set to **Program Controlled**.

Note:

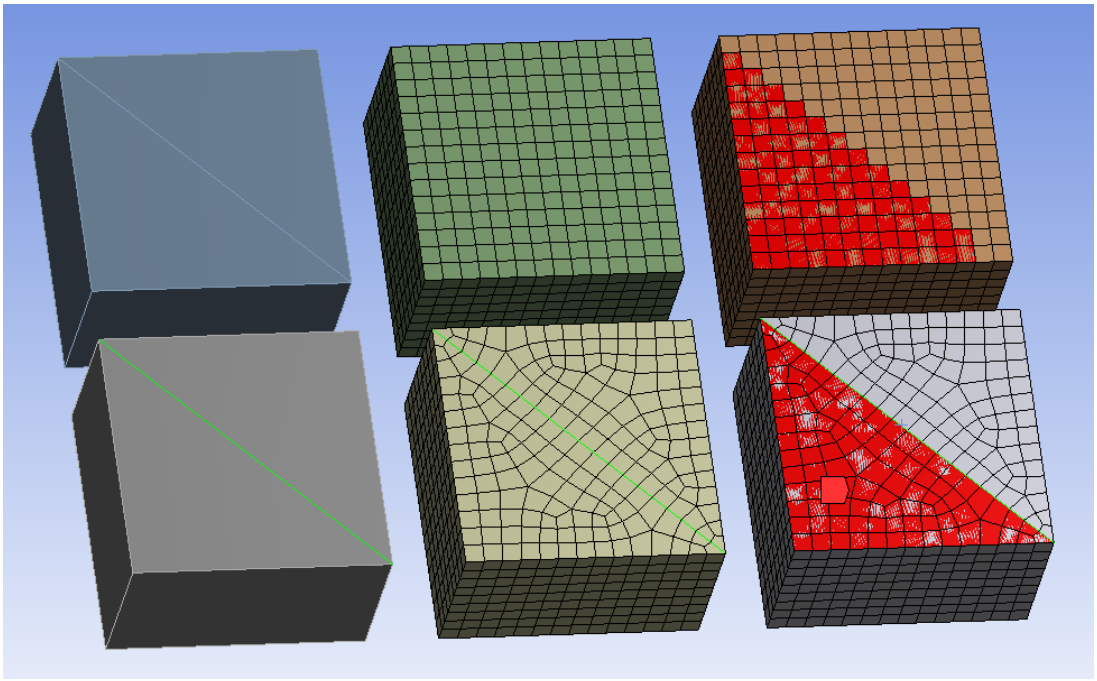
- If surface bodies have differing thicknesses, the edges between the faces will be protected unless using **MultiZone Quad/Tri** mesh method. If using the **MultiZone Quad/Tri** mesh method, use **Preserve Boundaries** = **All**, or put the faces into separate named selections.
- If you use the **MultiZone** mesh method and set **Preserve Boundaries** (p. 242) to **All**, the **MultiZone** method will protect all boundaries.
- Virtual Topology can still be used with patch independent meshing; however, the boundaries of the virtual cells do not have to be protected unless the virtual topology is scoped to something. In other words, the virtual cells replace the underlying geometry but follow the same protection rules.
- You should apply loads/boundary conditions prior to meshing as it is the most robust process to get the proper mesh to respect the boundary conditions.
- The mesh is associated to the geometry even for bodies, faces, edges, and vertices that are not protected, but in such cases there may not be the one-to-one relationship

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

that exists between the mesh and geometry for protected topology. For example, in Figure 68: Protecting Topology (p. 186) the highlighted edge is protected for the boxes in the bottom row, but not for the boxes in the top row.

In the two boxes on the right, the faces with red mesh indicate face mesh that is associated with the triangular face on the left of the geometry. Due to the sharp features of the box, the side faces all have a one-to-one relationship. You can check the association by using the **Plot Elements Attached to Named Selections** option.

Figure 68: Protecting Topology



Protecting Topology Post Meshing

After meshing, if a new object is scoped to the geometry, the associations to the mesh may not match (see Figure 68: Protecting Topology (p. 186)). In this case, the new object is added to the list of protected topologies, but you may or may not want to re-mesh because the associations may be fine. For example, if you scope a control to one of the side faces of the box, the associated mesh will be fine. However, if you select one of the triangular faces, the associated mesh may be problematic.

You can set the Topology Checking (p. 183) control in the **Advanced** group to control the re-meshing behavior. By default, **Topology Checking** is set to **Yes**, which forces a re-mesh.

- If **Topology Checking** is set to **Yes** (default), the mesh goes out-of-date, because the state manager must revalidate that all scoped topology is associated properly as protected topology. If you then attempt to re-mesh, the software runs the topology checks and ensures all protected topology is respected. If the topology checks are successful, the mesh is validated but not re-meshed. If the topology checks are unsuccessful, the software re-meshes the geometry, treating the newly scoped objects as protected topology.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If **Topology Checking** is set to **No**, the software does not check to ensure that the mesh is associated to the topology properly, so you must validate the associations manually if you have concerns.

Pinch

The **Pinch** feature lets you remove small features (such as short edges and narrow regions) at the mesh level in order to generate better quality elements around those features. The **Pinch** feature provides an alternative to Virtual Topology (p. 545), which works at the geometry level. The two features work in conjunction with one another to simplify meshing constraints due to small features in a model that would otherwise make it difficult to obtain a satisfactory mesh.

When **Pinch** controls are defined, the small features in the model that meet the criteria established by the controls will be "pinched out," thereby removing the features from the mesh. You can instruct the Meshing application to automatically (p. 193) create pinch controls based on settings that you specify, or you can manually (p. 340) designate the entities to be pinched. Pinch controls can be applied to solid and shell models, with certain restrictions as shown in the table below.

The **Pinch** feature is supported for the following mesh methods:

Volume Meshing:

- Patch Conforming (p. 208)
- Thin Solid Sweeping (p. 237)
- Hex Dominant Meshing (p. 233)

Surface Meshing:

- Quad Dominant (p. 282)
- All Triangles (p. 284)
- MultiZone Quad/Tri (p. 284)

The table below shows the types of model (solid or shell), mesh methods, pinch creation methods (auto or manual), and pinch behaviors that are supported for each type of pinch control.

Note:

With Ansys Workbench Release 16.0, post pinch behaviors are migrated into **Mesh Connections**. When you regenerate a mesh that was created using **Pinch Behavior** settings, the new mesh might report different results than the previous mesh.

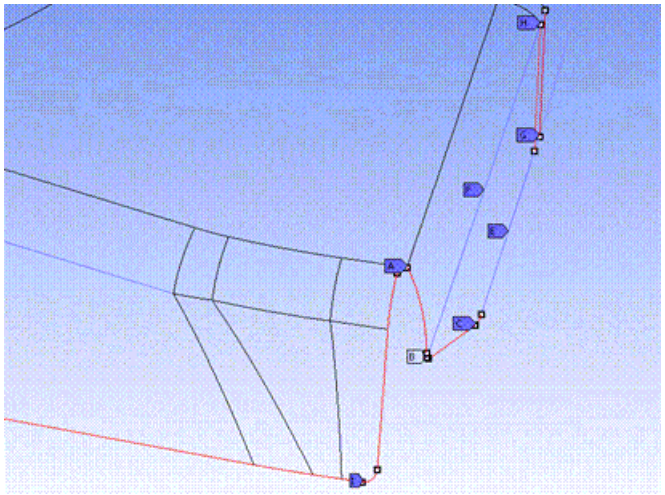
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entities in Pinch Control: Primary > Secondary	Model Type		Mesh Method						Pinch Creation Method	
	Solid	Shell	Patch Conforming	Thin Solid Sweeping	Hex Dominant	Quad Dominant	All Tri	MultiZone Quad / Tri	Auto Pinch	Manual Pinch
Edge > Edge	X	X	X	X	X	X	X	X	X	X
Edge > Vertex	X	X	X	X	X	X	X	X		X
Vertex > Vertex	X	X	X	X	X	X	X	X	X	X
Face > Edge		X				X	X	X		X
Face > Vertex		X				X	X	X		X

Examples of a Mesh With and Without Pinch

The figures below illustrate the effect of pinch controls on the mesh. The first figure shows where the pinch controls have been defined in the model. The second and third figures show the meshes that are generated without the pinch controls and with the pinch controls, respectively.

Figure 69: Locations of Pinch Controls



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 70: Mesh Generated Without Pinch Controls

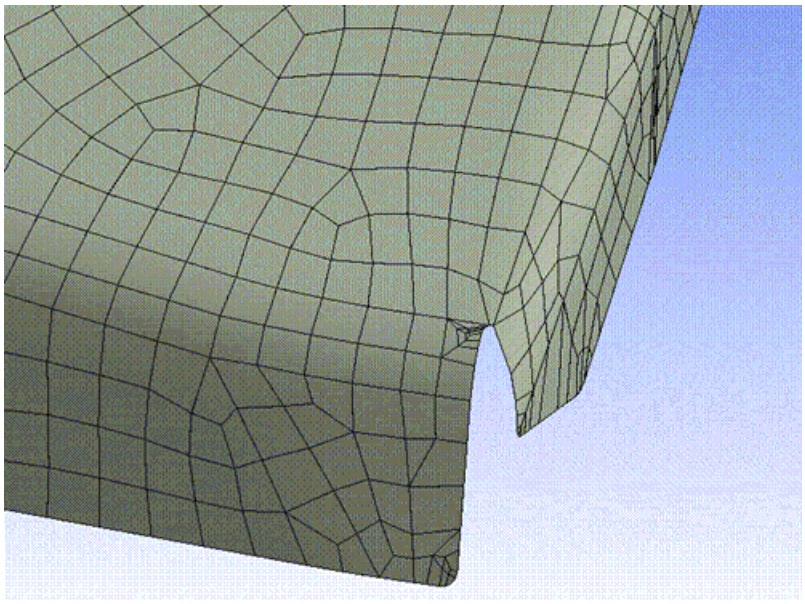
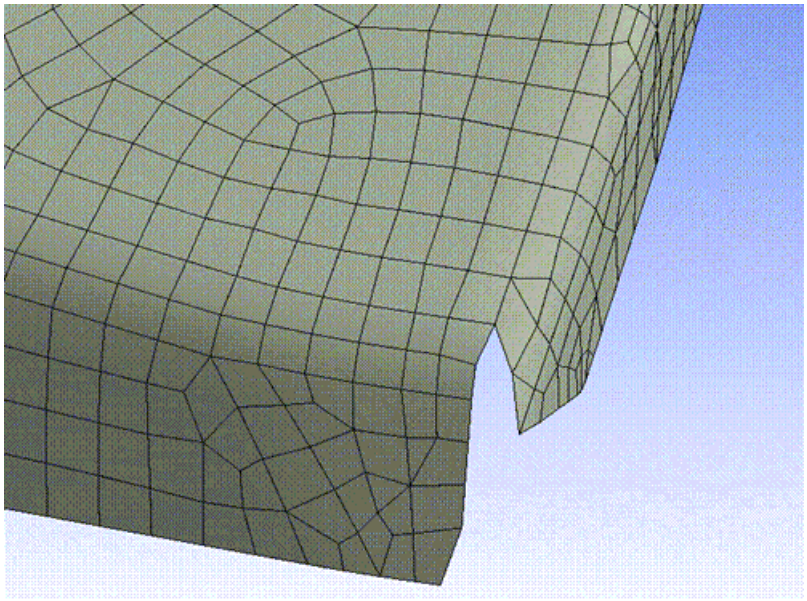


Figure 71: Mesh Generated With Pinch Controls



For More Information

For general information on applying pinch controls in combination with the various mesh method controls, refer to Meshing: Mesh Control Interaction Tables (p. 475).

Additional pinch topics include:

- Pinch Control Automation Overview
- How to Define Pinch Control Automation
- How to Define or Change Pinch Controls Manually
- Usage Information for Pinch Controls

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pinch Control Automation Overview

The **Advanced** group of global mesh controls appears in the Details View when the **Mesh** object is selected in the Tree Outline. By setting pinch options in the **Advanced** group, you can instruct the Meshing application to automatically generate pinch controls according to your settings. The Meshing application can generate the pinch controls based off shell thickness (for surface models only), or by identifying small features in the model that fall within a tolerance you specify.

Understanding the Automatic Pinch Control Algorithm

The Meshing application uses four major criteria for generating automatic pinch controls. These criteria (in order of importance) include:

1. Capture geometry of sheet bodies.
2. Edges adjacent to flat surfaces are primaries.
3. Cluster primaries together so that they are adjacent to one another.
4. All things being equal, longer edges are primaries.

Note:

- The primary geometry is the entity that retains the profile of the original geometry. The secondary geometry is the entity that changes in order to move towards the primary geometry. Depending on the tolerance, the pinch control will pinch out the entire secondary entity or only a portion of the secondary entity into the primary.
- The automatic pinch control algorithm supports only one primary for each pinch control.
- Once the automatic pinch control algorithm has paired two edges to use as a primary and a secondary in an automatic pinch control, the algorithm cannot use either of those same two edges as primary or secondary in any other automatic pinch control. For example, in the geometry shown in Figure 72: Automatic Pinch Control for Edges on Left; Manual Pinch Control Required for Edges on Right (p. 191), an automatic pinch control has been created for the annotated secondary (red) edges and the primary (blue) edge on the left side of the model. However, a pinch control will not be created automatically for the very similar configuration of edges on the right side of the model. For the Meshing application to be able to generate the mesh shown in Figure 73: Mesh Generated with Automatic Pinch Control and Manual Pinch Control on Similar Geometry (p. 191), a manual pinch control had to be created for the edges on the right.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 72: Automatic Pinch Control for Edges on Left; Manual Pinch Control Required for Edges on Right

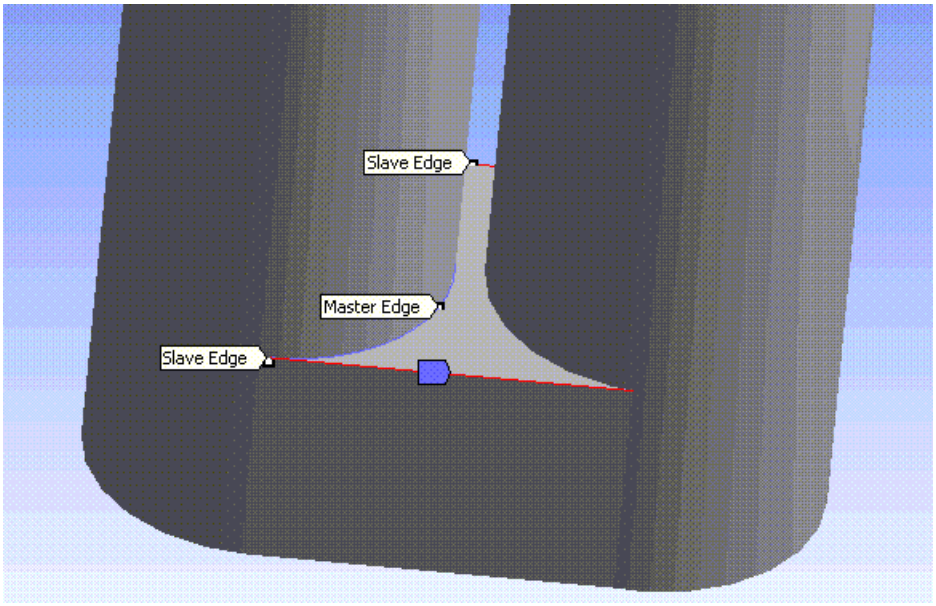
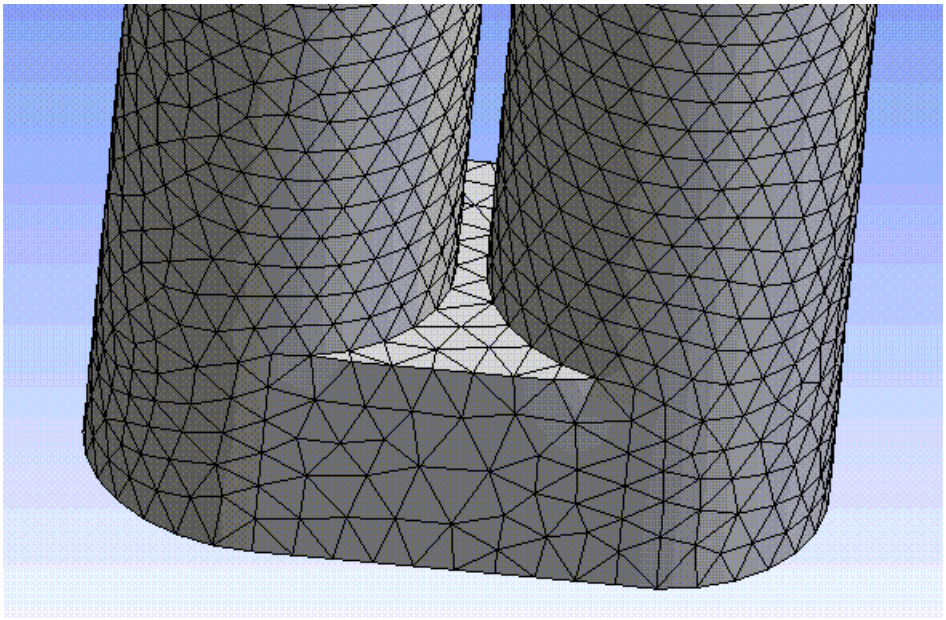


Figure 73: Mesh Generated with Automatic Pinch Control and Manual Pinch Control on Similar Geometry



The basic options for defining automatic pinch controls are described below.

Use Sheet Thickness for Pinch

The **Use Sheet Thickness for Pinch** control determines whether automatic pinch controls will be based on the shell thickness of the model, rather than on a specified pinch tolerance (p. 192). When the sheet thickness method of automatic pinch control is used, the pinch algorithm uses a pinch tolerance equal to 1/2 of the shell thickness.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Valid values are **Yes** and **No**. The default is **No**.

Note:

- The **Use Sheet Thickness for Pinch** control is available only for sheet (surface) models. If your model contains a combination of surface and solid bodies, the **Use Sheet Thickness for Pinch** control is available only if all solid bodies are suppressed (p. 528).
- You cannot use the sheet thickness method of automatic pinch control for a surface model that has no thickness defined or has a 0 (zero) thickness.

Pinch Tolerance

The **Pinch Tolerance** control allows you to specify a tolerance for the Meshing application to use when it generates automatic pinch controls. Vertex-vertex pinch controls will be created on any edge with a length less than the specified tolerance, and edge-edge pinch controls will be created on any face for which two edges are within proximity according to the specified tolerance. For the Meshing application to automate pinch control, you must specify a value for **Pinch Tolerance** unless **Use Sheet Thickness for Pinch** (p. 191) is set to **Yes**. Specify a value greater than **0.0**.

Note:

- The value that you specify for **Pinch Tolerance** should be smaller than the mesh size around the region in which the pinch control is being applied. For example, if an Edge Sizing control has been placed on an edge, a **Pinch Tolerance** value that is greater than that edge sizing may cause the mesher to fail.
- In general, the value that you specify for **Pinch Tolerance** should be greater than the value that appears in the **Sizing** > **Minimum Edge Length** (p. 120) field. Exceptions to this guideline include models without seam edges, such as elliptical prism, cylinder, and sphere models.
- Do not specify an overly high value for the **Pinch Tolerance** control. The **Pinch** feature allows the mesher to mesh over geometry features as if they were not there, and a tolerance that is set too high can cause inverted elements.
- When **Capture Curvature** and/or **Capture Proximity** is set to **Yes**, the default pinch tolerance is 90% of the value of **Curvature Min Size** (p. 113)/ **Proximity Min Size** (p. 115) (whichever is smaller). This differs from the tolerance used by the default mesh based defeaturing, refer to Mesh Defeaturing (p. 110) for details.

Generate Pinch on Refresh

The **Generate Pinch on Refresh** control determines whether pinch controls will be regenerated following a change made to the geometry (such as a change made via a DesignModeler application operation such as a merge, connect, etc.). If **Generate Pinch on Refresh** is set to **Yes** and you change the geometry, all pinch controls that were created automatically will be deleted and recreated based on the new geometry. If **Generate Pinch on Refresh** is set to **No** and you update the geo-

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

metry, all pinch controls related to the changed part will appear in the Tree Outline but will be flagged as undefined.

Valid values are **Yes** and **No**. The default is **No**.

Note:

Only **Automatic** pinch controls are regenerated. That is, if a pinch control has a **Scope Method** of **Manual** (either because it was created manually or because you made a change to an **Automatic** pinch control), the pinch control will never be regenerated on refresh. See Changing Pinch Controls Locally (p. 343) for information about making changes to pinch controls.

How to Define Pinch Control Automation (p. 193) provides the steps for defining automatic pinch controls.

How to Define Pinch Control Automation

The following sections provide the steps for defining pinch control automation. Pinch can be automated based on either shell thickness or a user-defined tolerance.

Note:

Use of pinch control automation will delete all existing pinch controls that have a **Scope Method** of **Automatic** before creating the new pinch controls.

Defining Pinch Control Automation Based on Shell Thickness

This section describes the steps for defining pinch control automation based on shell thickness. This procedure applies to sheet (surface) models only.

To define pinch control automation based on shell thickness:

1. In the Details View of the **Mesh** folder, expand the **Advanced** group of controls.
2. Set **Use Sheet Thickness for Pinch** (p. 191) to Yes.

   Notice that the value of the **Pinch Tolerance** (p. 192) control changes to **Based on Sheet Thickness** and is grayed out.
3. Change the value of the **Generate Pinch on Refresh** (p. 192) control if desired.
4. Right-click the **Mesh** folder and select **Create Pinch Controls** from the context menu.

   A pinch control object is automatically inserted into the Tree for each region containing features that meet the criteria established by the pinch control settings. To display details about an individual pinch control, highlight it in the Tree and information about it appears in the Details View. For information about making changes to this information, refer to Changing Pinch Controls Locally (p. 343).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining Pinch Control Automation Based on a Specified Pinch Tolerance

This section describes the steps for defining pinch control automation based on a tolerance that you specify.

To define pinch control automation based on pinch tolerance:

1. In the Details View of the **Mesh** folder, expand the **Advanced** group of controls.
2. Specify a value for **Pinch Tolerance** (p. 192).
3. Change the value of the **Generate Pinch on Refresh** (p. 192) control if desired.
4. Right-click the **Mesh** folder and select **Create Pinch Controls** from the context menu.

   A pinch control object is automatically inserted into the Tree for each region containing features that meet the criteria established by the pinch control settings. To display details about an individual pinch control, highlight it in the Tree and information about it appears in the Details View. For information about making changes to this information, refer to Changing Pinch Controls Locally (p. 343).

How to Define or Change Pinch Controls Manually

As an alternative to defining pinch control automation (p. 193), you can define local controls to pinch scoped entities. You can also make changes to pinch controls, regardless of whether they were created automatically or manually. For details, refer to Pinch Control (p. 340) in the local mesh controls section of the Meshing help.

Usage Information for Pinch Controls

Remember the following information when using the **Pinch** feature:

- The **Pinch** feature works on faces, edges, and vertices only, bodies cannot be pinched. Refer to the table in Pinch (p. 187) for restrictions related to entity types.
- The automatic pinch control algorithm supports only one primary for each pinch control. In manual pinch controls, you can specify multiple faces or multiple edges to act as primaries, but only one vertex can act as primary.
- When defining manual pinch controls, using the same primary in more than one pinch control is supported. This is true for all types of manual pinch controls: edge-edge, edge-vertex, vertex-vertex, face-edge, and face-vertex. When multiple pinch controls use the same primary, the aggregate of the pinch controls is used to determine the pinch. Note that this behavior differs from that of other mesh controls when multiples are specified. With other mesh controls, the control that appears lowest in the Tree is honored.
- If there are hard size constraints on a primary, the pinch control will be skipped completely. If there are hard size constraints on secondaries, only the secondaries with the constraints will be skipped. In either case, a warning message will be issued.
- If your model contains multibody parts and you want pinch controls to operate on selected parts/bodies only, you must first suppress (p. 528) the parts/bodies that you do not want the

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pinch controls to apply to. Then follow the steps outlined in How to Define Pinch Control Automation (p. 193).

Alternatively you can use mesh connections, which supports automatic and manual mesh connections. With mesh connections, you can change the settings in the Details View of the **Connection Group** folder to automatically generate mesh connections on scoped geometry only. For more information, see Connection Features and Operations in the Mechanical help.

- If the geometry fails to mesh correctly due to the pinched features, an error message is generated. To highlight the geometry that is responsible for the message, select the message, right-click, and select **Show Problematic Geometry** from the context menu.

- When a face contains an internal loop with a pinch control and the edges of the loop become a "single internal edge" due to pinching, the surface mesher may completely ignore the "single" edge (that is, the surface mesher may mesh over the edge). The reason that this may occur is that a pinch control never changes the topology of a model. When a surface mesher collects all boundary edge meshes before performing surface meshing, it considers the newly created "single" edge to be a regular edge rather than a hard edge, which most users would expect. As a result, all edge meshes along the "single" edge may be ignored.

- After pinch controls are generated:

  – If you highlight a pinch control in the Tree, the pinch region is flagged in the **Geometry** window. For more information, see the descriptions of **Primary Geometry** and **Secondary Geometry** pinch controls in Pinch Control Automation Overview (p. 190).

  – You can make changes to pinch controls whether they were generated automatically or created manually. To do so, in the **Mesh** folder, highlight the **Pinch** object that you want to change. As a result, the Details of the pinch appear in the Details View, where you can change its **Scope** and **Definition**. Making changes to a pinch that was generated automatically causes the value of the **Scope Method** control to change from **Automatic** to **Manual**. For details about defining or changing pinch controls manually, see Pinch Control (p. 340) in the local mesh controls section of the Meshing help.

- If a pinch control has a pinch tolerance defined for it that falls below one or more Hard (p. 310) scoped size controls, a warning will be issued. The warning will suggest that you either modify the pinch tolerance or remove any pinch control(s) in close proximity to the Hard size control(s) in question; otherwise, surface meshing may fail.

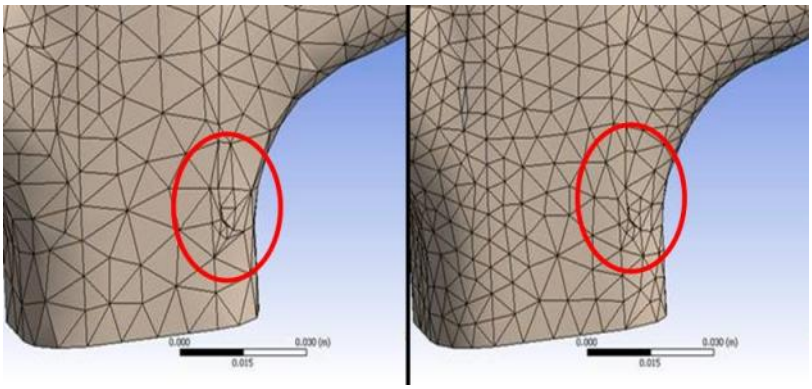
- There is no guarantee that features will be preserved when using pinch controls. For this reason, it is best practice to check the mesh where pinch controls have been defined close to features. If a problem exists in the mesh, flipping the primary and secondary entities will be sufficient to solve the problem in many cases.

- Pinch controls can be used for models involving multiple complications in one location (such as slivers, sharp angles, and short edges within the same pinch tolerance) as well as for models containing isolated problem spots. However, when used in combination with the Sizing Options (p. 89), pinch controls are best used for isolated problems. For example, refer to the meshes in the figure below, which show the results of applying pinch controls in combination with other sizing options. For the mesh on the left, a **Pinch Tolerance** (p. 192) of 3e-3 and a **Curvature Min Size** (p. 113) of 6.e-3 were specified. For the mesh on the right, a **Pinch Tolerance** of 3e-3 and a **Min Size** of 4.e-3 were specified. Neither is acceptable due to the presence of high aspect

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ratio triangles in the mesh. In such cases, the use of Virtual Topology or defeaturing within the DesignModeler application is recommended as an alternative to pinch.

Figure 74: Pinch Not Recommended for Models with Multiple Complications



- In a face-edge pinch control, the mesh on the edges within the specified tolerance is "snapped" to the primary face. You must choose the primary and secondaries in such a way that the elements on the face whose edges are defined as secondaries will be stretched onto the primary face. If the edges would be "squashed," no pinch will be created.
- When a face pinch control and a Face Meshing (p. 314) control are applied to the same face, the mesher attempts to generate a mapped mesh for the face. If the mesher cannot retain the mapped mesh pattern, it will generate a free mesh instead and issue a warning.
- When using a Face as the primary geometry, then the pinch control is applied post-processing and does not support mixed dimension meshing. When using an Edge as the primary geometry, then the pinch control is applied pre-processing, and is recommended in mixed dimension situations.
- If you apply a match control (p. 334) and a face-edge pinch control to the same topology, a warning is issued.
- Since **Pinch** objects cannot be duplicated, they cannot be used as template objects for the Object Generator.

Loop Removal

The **Advanced** group of global mesh controls appears in the Details View when the **Mesh** object is selected in the Tree Outline. The Meshing application automatically removes loops according to the criteria you specify for the loop removal options in this group. Prior to meshing, you can use the **Show Removable Loops** (p. 538) feature to preview the loops that will be removed according to the current settings.

The loop removal feature is supported for the following mesh methods:

Surface Meshing:

- Quad Dominant (p. 282)
- All Triangles (p. 284)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- MultiZone Quad/Tri (p. 284)

Note:

- The loop removal controls are passed to the MultiZone Quad/Tri method as described in MultiZone Quad/Tri Method Control (p. 284).
- If you are meshing with loop removal on (using the Quad Dominant or MultiZone Quad/Tri method), making changes to a loop after meshing (such as adding a load on a loop) invalidates the mesh and you will need to re-mesh. You should apply loads to the model before meshing when using these controls. Refer to Protecting Topology Defined Prior to Meshing (p. 184) for related information.

The options for defining loop removal are described below.

Sheet Loop Removal

The **Sheet Loop Removal** control determines whether loops will be removed (meshed over) by the mesher. When **Sheet Loop Removal** is set to **Yes**, the mesher removes any loop with a radius less than or equal to the value of **Loop Removal Tolerance**.

Valid values are **Yes** and **No**. The default is **No**.

Loop Removal Tolerance

The **Loop Removal Tolerance** control sets the tolerance for loop removal. Any loop with a radius less than or equal to the value of **Loop Removal Tolerance** will be meshed over by the mesher.

Specify a value greater than **0.0**.

Automatic Methods

The **Automatic Methods** describe how the mesh generated for the sheet and sweepable solid bodies when you do not apply a local mesh method to them. The **Automatic Methods** group includes:

Sheet Body Method

Sweepable Body Method

Sheet Body Method

The **Sheet Body Method** determines the mesh generated for sheet bodies when you do not apply a local mesh method to them. The available options are:

- **Quad Dominant**: Allows you to generate the quadrilateral mesh for the sheet bodies. The default value for **Sheet Body Method** is **Quad Dominant**.

  When **Sheet Body Method** is **Quad Dominant**, mesher cannot perform **Weld**, **Connect**, **Repair Topology**, **Deviation**, and **Quad Layer** controls for that sheet bodies.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the model or geometry has a manual connection, mesher applies the **Quad Dominant** method as the default **Sheet Body Method**, even when you set **Sheet Body Method** as **Prime Mesh**.

To perform **Weld**, **Connect**, **Repair Topology**, **Deviation**, and **Quad Layer** controls you need to scope those sheet bodies under **Automatic (PrimeMesh)** method.

- **Prime Mesh**: Allows you to generate the quadrilateral mesh for the sheet bodies.

  When **Sheet Body Method** is **Prime Mesh**, any mesh controls that are incompatible with the **Automatic (Prime Mesh)** method are ignored. For information on the **Prime Mesh**, refer to Automatic (PrimeMesh) (p. 293) limitations.

  When **Sheet Body Method** is **Prime Mesh,** mesher can perform **Weld**, **Connect**, **Repair Topology**, **Deviation**, and **Quad Layer** methods.

  Note:

  Prime Mesh does not support for **Inflation** control, **Match Mesh** control, **Vertex Types** and **Number of Divisions** in **Face Meshing** control, **Edge Biasing** in **Edge Sizing** control, and **Influence Volume** in **Face Sizing** control.

Sweepable Body Method

The **Sweepable Body Method** determines the mesh generated for the sweepable bodies when you do not apply a local mesh method to them. The available options are:

- **Sweep**: Allows you to generate the sweep mesh on the sweepable solid bodies.
- **MultiZone**: Allows you to generate the **MultiZone** mesh on the sweepable bodies. For information related to **MultiZone**, refer to MultiZone Method (p. 242).

Statistics Group

The **Statistics** group lets you view and request information about these options:

Nodes

Elements

Show Detailed Statistics

Nodes

The **Nodes** option provides a read-only indication of the number of nodes in the meshed model. If the model contains multiple parts or bodies, you can view the number of nodes in an individual part or body by highlighting it under the **Geometry** object in the Tree Outline.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Elements

The **Elements** option provides a read-only indication of the number of elements in the meshed model. If the model contains multiple parts or bodies, you can view the number of elements in an individual part or body by highlighting it under the **Geometry** object in the Tree Outline.

Show Detailed Statistics

The **Show Detailed Statistics** displays the total number of elements in each of the element types in the given model when set to **Yes**. The default is **No**. The available mesh statistics are **Corner Nodes, Mid Nodes**, **Orientation Nodes**, **Beam Elements**, **Beam3, Shell Elements**, **TriShell3**, **QuadShell4**, **Solid Elements**, **Tet10**, **Wedge15**, **Pyramid13**, **Solid Shell**, **Gasket Elements** and more.

Model Assembly Group

The **Model Assembly** group displays **Read Only** when you have assembled files from upstream sources (Mechanical Model, analysis systems, and so on).

Read Only is **Yes** by default. See the Mesh Modification section for more information.

Note:

When **Read Only** is set to **Yes**, the **Mesh Numbering** option is not available.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Meshing: Local Mesh Controls

Local mesh controls are available when you highlight a **Mesh** object in the tree and choose a tool from either the **Mesh Control** drop-down menu, or from first choosing **Insert** in the context menu (displayed when you right-click a **Mesh** object). You can specify the scoping of the tool in the tool's Details View under **Method** to either a **Geometry Selection** or to a **Named Selection**.

Note:

Be aware of the following items regarding mesh control tools:

- The Object Generator enables you to make one or more copies of a template object, scoping each to a different piece of geometry. When defining mesh controls, you can use the Object Generator to make copies of a template mesh control, which may reduce the necessity to manually define multiple related mesh controls. For details, refer to Generating Multiple Objects from a Template Object in the Mechanical help.

- For most mesh controls, the latest control that you add on a particular geometry overrides any prior controls that you already have added on that geometry. For example, if you apply a Sizing control setting of 0.5 to faces A,B,C then apply a setting of 1.0 to face B, faces A and C will retain the 0.5 setting, but the setting for face B will be 1.0. This is also useful when you want to force sweep many bodies of a multibody part and only tet mesh one or specify special sweeping options on one. For example, you can select all 1000 parts and then override one or 10 part(s) instead of picking 999 (990) and then selecting one (10).

  Exceptions include the MultiZone Quad/Tri and All Tetrahedrons - Patch Independent controls. For information about how these controls interact with other controls, refer to Meshing: Mesh Control Interaction Tables (p. 475), Interactions Between Mesh Methods (p. 475), and Interactions Between Mesh Methods and Mesh Controls (p. 478).

- If you suppress a mesh control tool, the Suppress symbol appears ("x" adjacent to the name of the tool) and **Suppressed** is set to **Yes** in the Details View of the tool. Situations can occur when you do not suppress a mesh control tool, and the Suppress symbol appears adjacent to the tool but **Suppressed** is set to **No** in the Details View of the tool. In these situations, refer to the mesh control's **Active** read-only field for the reason why the tool is suppressed. Examples are a control applied to a uniform surface body mesh (not supported), a control scoped to suppressed geometry, or a **Contact Sizing** control scoped to a suppressed **Contact Region**.

The following local mesh controls are available:

Method Control

Sizing Control

Contact Sizing Control

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Refinement Control
Face Meshing Control
Mesh Copy Control
Match Control
Pinch Control
Inflation Control
Gasket Control
Feature Suppress
Repair Topology
Connect
Weld
Quad Layer
Deviation
Geometry Fidelity

Method Control

The **Method Control** provides meshing algorithms used for meshing solid and sheet bodies. The **Method** control is valid only for a body. When you do not select any **Method Control**, application applies **Automatic Method** on all bodies to provide a successful automated mesh. By default, the application attempts to use hexahedral elements for solid bodies and quadrilateral element generation for sheet bodies. If a solid body is not sweepable, the body is meshed with the **Patch Conforming Tetrahedrons** mesher.

To set the values for the **Method** control,

On the Tree Outline, right-click **Mesh** and click **Insert > Method**.

Or

On the Tree Outline, click **Mesh** and click **Method** in the **Mesh Context** tab on the Ribbon.

In the Details view for the scoped local method, you can set **Method** based on whether you want to apply the method to a solid body or a sheet body. For more information, refer to:

Method Controls and Element Order Settings
Setting the Method Control for Solid Bodies
Setting the Method Control for Sheet Bodies

Method Controls and Element Order Settings

When setting the Method control (p. 202) to a scoped body, you can control whether meshes are to be created on the scoped body with midside nodes or without midside nodes by using the **Element Order** setting under **Definition** in the Details View. When setting the **Element Order** option for a scoped body, choices include **Use Global Setting**, **Linear**, and **Quadratic**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you select **Use Global Setting**, **Element Order** will be handled as dictated by the global **Element Order** option (p. 101). The remaining choices—**Linear** and **Quadratic**—have the same descriptions as their counterparts under the global **Element Order** option (p. 101). Setting **Element Order** to **Linear** or **Quadratic** for a scoped body will override the setting of the global **Element Order** option.

If the **Element Order** is set to **Quadratic**, and if **Straight Sided Elements** is set to **No**, the midside nodes will be placed on the geometry so that the mesh elements properly capture the shape of the geometry. However, if the location of a midside node might affect the mesh quality, the midside node may be relaxed to improve the element shape. Therefore, some midside nodes might not follow the shape of the geometry precisely.

For more information about how the **Straight Sided Elements** control affects midside nodes, see Straight Sided Elements (p. 180).

Mixed Order Meshing

Mixed order meshing is supported across bodies for the following mesh methods:

- For solid meshing:
  - Patch Conforming Tetrahedron (p. 208)
  - Patch Independent Tetrahedron (p. 210)
  - MultiZone (p. 242)
  - General Sweep (p. 399)
  - Thin Sweep (p. 406)
  - Hex Dominant (p. 233)
  - Cartesian (p. 261)
- For surface meshing:
  - Quad Dominant (p. 282)
  - All Triangles (p. 284)
  - MultiZone Quad/Tri (p. 284)
  - Automatic (PrimeMesh) (p. 287)

Note:

Mixed order meshing is not supported for **Pull**.

This means that when scoping one of these mesh methods to bodies in a multibody part, you can set the **Element Order** option to **Quadratic** (resulting in higher order elements) for some bodies and to **Linear** (resulting in lower order elements) for others.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mixed order meshing is supported whether you are performing Selective Meshing (p. 444) or meshing all of the bodies in the part at the same time. The behavior, and your resulting mesh, is dependent on the meshing order:

- For *simultaneous* meshing, the Quadratic bodies generally have precedence at the interface. In such cases, all of the elements in a lower order body that are adjacent to a higher order body will be higher order elements, thereby creating one layer of quadratic elements at the interface face. These elements will be higher order at the interface face but with dropped midside nodes where adjacent to the linear elements in the mesh.

  An exception occurs when a solid body and a sheet body share an interface. In this case, the solid body takes precedence and will be meshed first using its defined element order. Then the sheet body is meshed with midside nodes handled as described for **Selective Meshing**, below.

- For Selective Meshing (p. 444), the order in which you mesh the bodies determines the precedence. If you first mesh a linear body followed by meshing an adjacent quadratic body, then the linear body has precedence. Elements in the quadratic body that are adjacent to the linear body will be lower order elements. Midside nodes will be dropped from the quadratic elements at the interface face.

  If you first mesh a quadratic body followed by meshing an adjacent linear body, then the quadratic body has precedence. Elements in the linear body that are adjacent to the quadratic body will be higher order elements. Midside nodes will be added to the linear elements at the interface face.

The figures below illustrate an example of mixed order meshing. To obtain the mesh shown in Figure 75: Mixed Order Meshing of a Multibody Part (p. 205), the global **Element Order** option was set to **Quadratic**, resulting in a mesh of quadratic tet elements for the topmost body. The sweep method was applied to the remaining bodies, with the **Element Order** option set to **Linear** on the **Sweep Method** (p. 235) control. This resulted in a mesh of primarily linear hex/wedge elements for the swept bodies, with the hex/wedge elements that are attached to the common interface being mixed order (see Figure 76: Mixed Order Elements (p. 206)).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 75: Mixed Order Meshing of a Multibody Part

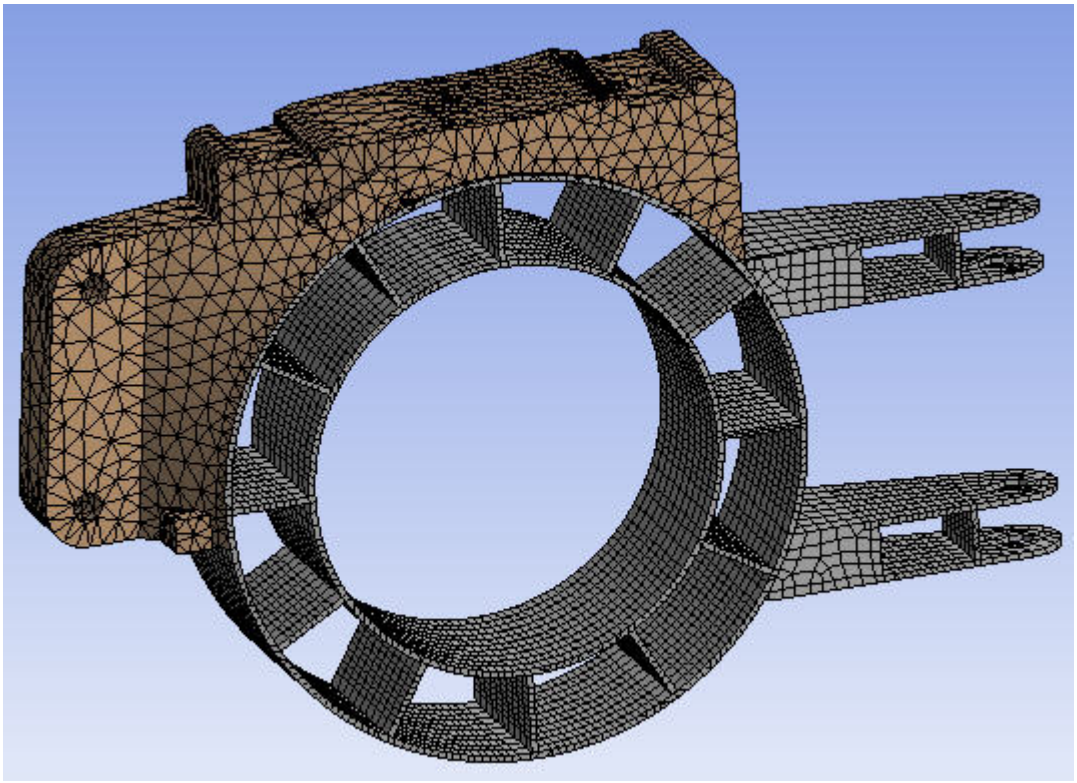
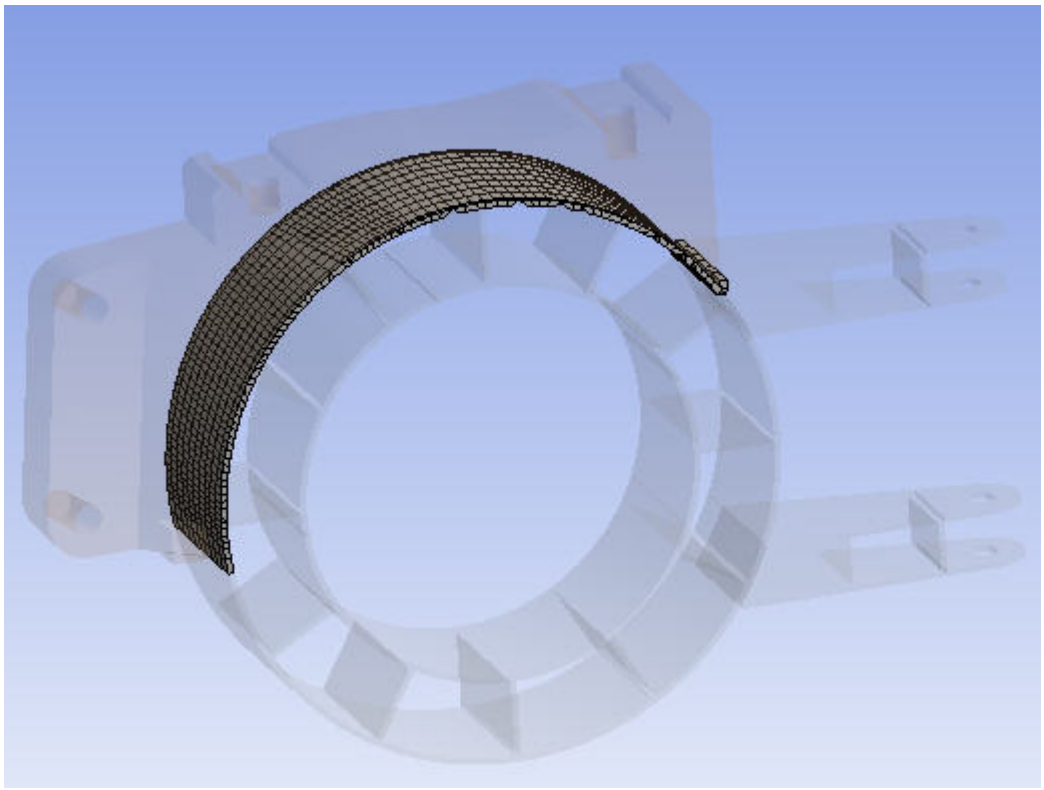


Figure 76: Mixed Order Elements (p. 206) shows the mixed order hex/wedge elements that are attached to quadratic pyramid elements at the interface. On the **Mesh Metrics** (p. 133) bar graph, mixed order elements are displayed as quadratic element types.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 76: Mixed Order Elements



Setting the Method Control for Solid Bodies

Method Controls available for solid bodies are:

- Automatic Method Control
- Tetrahedrons Method Control
- Hex Dominant Method Control
- Sweep Method Control
- MultiZone Method Control
- Cartesian Method Control
- Layered Tetrahedrons Method Control
- Particle Method
- Automatic (PrimeMesh)

Automatic Method Control

Automatic Method control performs sweeping for solid models and quadrilateral element generation for surface body models. When solid bodies cannot be swept, the body is meshed with the **Patch Conforming Tetrahedron** mesher.

On the Tree Outline, right-click **Mesh** and select **Show > Sweepable Bodies** to preview the bodies that can be swept meshed. To use the **MultiZone** mesh method instead of **Sweep** for bodies that are automatically detected as sweepable, click **Files > Options > Meshing**. Select **On** for **Use MultiZone for Sweepable Bodies**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To access the **Automatic Method** control,

On the Tree Outline, right-click **Mesh** and click **Insert > Method**.

Or

On the Tree Outline, click **Mesh** and click **Method** in the **Mesh Context** tab on the Ribbon.

The **Automatic Method** Details view has the following options:

Scope

- **Scoping Method**: Allows you to scope geometry bodies or named selection. The default value is **Geometry Selection**.
  - **Geometry Selection**: Allow you to scope the geometry bodies. When you select **Geometry Selection**, the **Geometry** allows you to select the geometry from the **Geometry** window.
  - **Named Selection**: Allow you to scope bodies grouped under a named selection.

Definition

- **Suppressed**: Allows you to suppress the selected control. The default value is **No**. When **Suppressed** is set to **Yes**, **Active** displays the status of the selected control. **Active** is read-only.
- **Method**: Allows you to select method.
- **Element Order**: Allows you to select the element order. The default value is **Use Global Setting**. For information about the **Element Order** option, refer Method Controls and Element Order Settings (p. 202).

Tetrahedrons Method Control

Tetrahedrons Method control creates a tetrahedral mesh. The **Algorithm** setting allows selection of a tetrahedral mesh creation method from the following options:

- Patch Conforming Algorithm for Tetrahedrons Method Control (p. 208)
- Patch Independent Algorithm for Tetrahedrons Method Control (p. 210)

To access the **Tetrahedrons Method** control,

On the Tree Outline, right-click **Mesh** and click **Insert > Method**.

Or

On the Tree Outline, click **Mesh** and click **Method** in the **Mesh Context** tab on the Ribbon. Select **Method** as **Tetrahedrons** in the **Automatic Method** Details view.

The **Tetrahedrons Method** Details view has the following options:

Scope

- **Scoping Method**: Allows you to scope geometry bodies or named selection. The default value is **Geometry Selection**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Geometry Selection**: Allow you to scope the geometry bodies. When you select **Geometry Selection**, the **Geometry** allows you to select the geometry from the **Geometry** window.

  - **Named Selection**: Allow you to scope bodies grouped under a named selection.

Definition

- **Suppressed**: Allows you to suppress the selected control. The default value is **No**. When **Suppressed** is set to **Yes**, **Active** displays the status of the selected control. **Active** is read-only.

- **Method**: Allows you to select method.

- **Algorithm**: Allows you to select the algorithm. The available options are **Patch Conforming** and **Patch Independent**. The default value is **Patch Conforming**.

- **Element Order**: Allows you to select the element order. The default value is **Use Global Setting**. For information about the **Element Order** option, refer Method Controls and Element Order Settings (p. 202).

Patch Conforming Algorithm for Tetrahedrons Method Control

The **Patch Conforming** Tetra mesh method is a Delaunay tetra mesher with an advancing-front point insertion technique used for mesh refinement. The Patch Conforming Tetra mesh method provides:

- Support for 3D inflation (p. 161)

- Built-in pyramid layer for conformal quad-tet transition (p. 462)

- Built-in growth and smoothness control. The mesher tries to create a smooth size variation based on the specified growth factor.

- Removes features under a certain size (Mesh Defeaturing (p. 110)).

Remember the following information when using the Patch Conforming Tetra mesh method:

- When **Patch Conforming Tetra** meshing (with **Physics Preference** (p. 96) of **CFD**) fails due to lack of available memory, an error message is issued. However, this error message does not identify insufficient memory as the cause because meshing stops before the memory limit is reached. You may not notice any unusual behavior.

Patch Conforming Method Details view has the following options:

Scope

- **Scoping Method**: Allows you to scope geometry bodies or named selection. The default value is **Geometry Selection**.

  - **Geometry Selection**: Allows you to scope the geometry bodies. When you select **Geometry Selection**, the **Geometry** allows you to select the geometry from the **Geometry** window.

  - **Named Selection**: Allows you to scope bodies grouped under a named selection.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Definition

- **Suppressed**: Allows you to suppress the selected control. The default value is **No**. When **Suppressed** is set to **Yes**, **Active** displays the status of the selected control. **Active** is read-only.
- **Method**: Allows you to select method.
- **Algorithm**: Allows you to select the algorithm. The available options are **Patch Conforming** and **Patch Independent**. The default value is **Patch Conforming**.
- **Element Order**: Allows you to select the element order. The default value is **Use Global Setting**. For information about the **Element Order** option, refer Method Controls and Element Order Settings (p. 202).

Advanced Improve Options

- **Aggressive Thin Face Collapse**: Allows you to defeature the thin faces when the size of the thin faces is less than twice of the target **Characteristics Length**, when **Aggressive Thin Face Collapse** is set to **Program Controlled**. **Aggressive Thin Face Collapse** is available only when **Physics Preference** is set to **Explicit**. The default value is **Program Controlled**.
- **Automatic Node Movement**: Allows you to move nodes to meet the target mesh quality criteria for a given **Physics Preference**. At the end of tetrahedral meshing, an optimization algorithm finds the elements exceeding the target quality and automatically move the nodes to improve the quality of the elements. When multiple quality targets exist, the node movement is performed sequentially on the different target criteria. For example, When **Physics Preference** is **Explicit**, performs **Automatic Node Movement** first for **Element Quality**, then **Aspect Ratio (Height)** and finally **Characteristic Length (LS-Dyna)**.

  The available options are:

  – **Program Controlled**: Performs **Automatic Node Movement** for the **Explicit** Physics Preference. The value of the **Physics Preference** option sets the default for **Program Controlled**, as detailed in the following table:

Physics Preference	Mechanical	Nonlinear Mechanical	Electromagnetics	CFD	Explicit
Program Controlled	Off	Off	Off	Off	On

  – **Conservative**: Allows you to meet the target quality without moving the nodes away from the surface.

  – **Aggressive**: Allows you to move nodes outside the boundary to meet target mesh quality for a given Physics Preference. When you set **Automatic Node Movement** to **Aggressive**, you receive a warning message for node movement.

  – **Custom**: Allows you to customize the **Feature Protection Angle**, **Restricted Node Movement to Surface**, and **Improvement Iterations** to meet the target quality for given Physics Preference.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ **Restricted Node Movements to Surface**: Allows you to constrain node movement to be only along the boundary when set to **Yes**. The default value is **Yes** for **Custom** and **Conservative**.

→ **Improvement Iterations**: Allows you to set the number of iterations performed by the algorithm to improve the target mesh quality. The default value is **3**.

→ **Feature Protection Angle**: Specifies the minimum angle at which nodes move to meet target quality for given **Physics Preference**. After volumes are meshed, if the dihedral angle between two triangular element faces is less than the specified **Feature Protection Angle**, the nodes on the edge between the element faces are protected during **Automatic Node Movement**. The default value is **120 degrees** for **Custom** and **Conservative** and **0 degrees** for **Aggressive**. If the angle is set to **0 degrees**, no nodes are protected during **Automatic Node Movement**. This is the most aggressive setting for quality improvement, causing nodes to locally move off features to attain quality targets where possible. If the angle is **180 degrees** this effectively protects all nodes in the surface mesh from moving. That is, **Automatic Node Movement** would only acts only on interior nodes.

Refinement Options

- **Refine at Thin Section**: Allows you the refine the number of element layers in the thin sections of the geometry when set to **Yes**. The default value is **No**.
  - **Number of Element Layers**: Provides the number of element layers in the thin section. **Number of Element Layers** is a read-only option. The default value is **2**.
- **Refine Surface Mesh**: Allows you to refine the surface mesh in the thin section of geometry when set to **Yes**. The default value is **No**. **Refine Surface Mesh** is available only when **Use Adaptive Sizing** is set to **No**.
  - **Minimum Thickness**: Allows you to set the minimum gap thickness to refine the surface mesh at thin sections with good quality elements. You should measure the thinnest section in your geometry and use this distance as input. The default value is half of the global **Element Size**.
  - **Face Proximity Gap Factor**: Allows you to refine or coarse the adjacent surface mesh.

Limitations

- **Automatic Node Movement** does not support **CFD** Physics Preference.

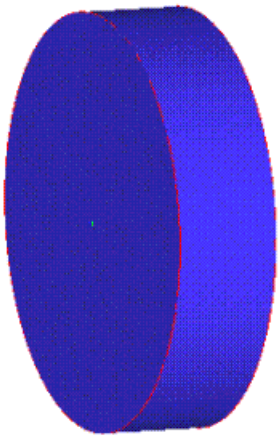
Patch Independent Algorithm for Tetrahedrons Method Control

The Patch Independent mesh algorithm for tetrahedrons is a "Top-Down" meshing algorithm based on the spatial subdivision algorithm which ensures refinement of the mesh where necessary, but maintains larger elements where possible, allowing faster computation. The Patch Conforming (PC) Tetrahedrons Algorithm is recommended for tetrahedral meshing but in some cases the Patch Independent Method can be helpful. For example, when meshing STL geometries or when geometry is dirty with badly formed surfaces which would require significant geometry clean-up to fix and mesh with Patch Conforming Tetrahedrons. After the root tetrahedron that encloses the entire geometry is initialized, the Patch Independent mesher subdivides the root tetrahedron to meet all element size requirements (that is, the prescribed local mesh sizes).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

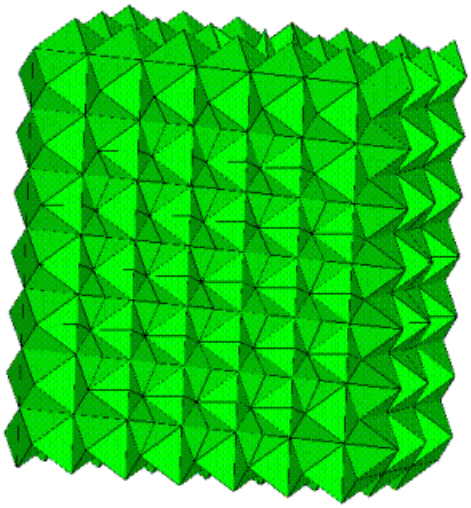
At each subdivision step, the edge length of the tetrahedron (=size) is divided by 2. This means that the prescribed sizes should all differ by factors that are an integer power of 2. The size of the root tetra is set to the smallest given size multiplied by 2^n. All other prescribed sizes are approximated by subdividing the root tetra. Refer to the series of figures below, which illustrate the process that is followed by the Patch Independent tetra mesher.

Figure 77: Geometry Input to Patch Independent Tetra Mesher



At this point, the Patch Independent tetra mesher balances the mesh so that elements sharing an edge or face do not differ in size by more than a factor of 2.

Figure 78: Full Tetrahedron Enclosing the Geometry



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 79: Full Tetrahedron Enclosing the Geometry in Wire Frame Mode

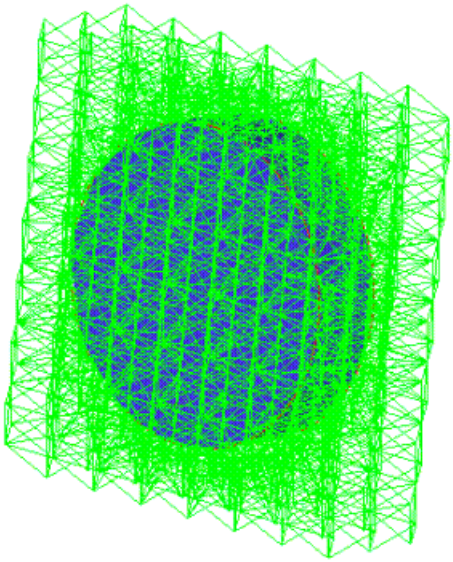
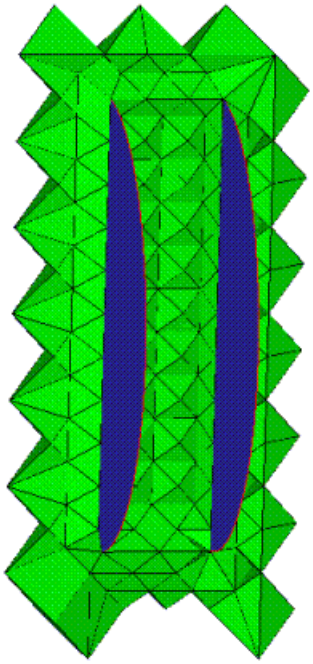


Figure 80: Cross-Section of the Tetrahedron



After this is done, the Patch Independent tetra mesher makes the mesh conformal. That is, it guarantees that each pair of adjacent elements shares an entire face. The mesh does not yet match the given geometry, so the mesher next rounds the nodes of the mesh to the prescribed points, prescribed curves, or model surfaces. The Patch Independent tetra mesher then "cuts away" all the mesh, which cannot be reached by a user-defined material point without intersection of a surface.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 81: Mesh After Capture of Surfaces and Separation of Useful Volume

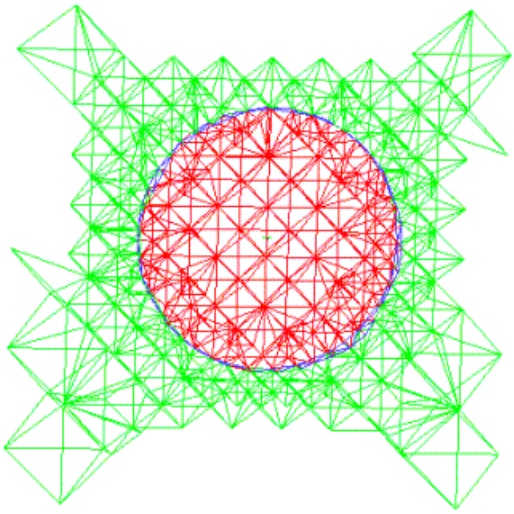
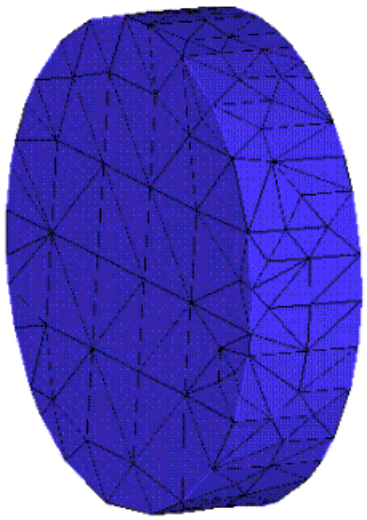


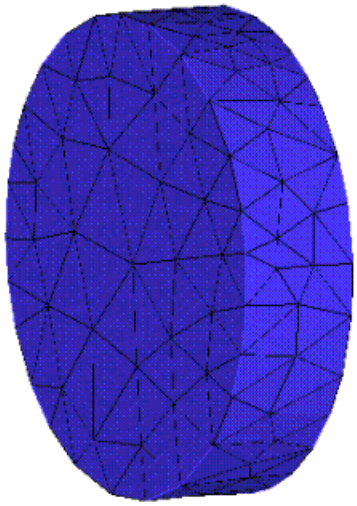
Figure 82: Final Mesh Before Smoothing



Finally, the mesh is smoothed by moving nodes, merging nodes, swapping edges and in some cases, deleting bad elements.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 83: Final Mesh After Smoothing



Note:

The **Patch Independent Tetrahedrons** method is being deprecated and will be removed in future releases.

The **Patch Independent** Method Details view has the following options:

Scope

- **Scoping Method**: Allows you to scope geometry or named selection. The default value is **Geometry Selection**. The available options are:
  - **Geometry Selection**: Allows you to scope the geometry for **Patch Independent Tetrahedrons** method.
  - **Named Selection**: Allows you to scope the available named selection for **Patch Independent Tetrahedrons**method.

Definition

- **Suppressed**: Allows you to suppress the selected control. The default value is **No**. When **Suppressed** is set to **Yes**, **Active** displays the status of the selected control. **Active** field is read-only.
- **Method** : Allows you to select the meshing method.
- **Algorithm**: Allows you to select the algorithm for meshing.
- **Element Order**: Defines the element order for meshing. Refer to Method Controls and Element Order Settings (p. 202).
- **Defined By**: Allows you to apply settings to define the mesh size. The available options are **Max Element Size** and **Approx Number of Elements per Part**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Max Element Size**: Specifies the size of the initial element subdivision. The default value depends on the sizing options selected:

  – When **Use Adaptive Sizing** is set to **No**, the default value of **Max Element Size** is inherited from the global **Max Size** (p. 110) value.

  – When **Use Adaptive Sizing** is set to **Yes**, the default value of **Max Element Size** is inherited from the global **Element Size** (p. 102) value.

  In either case, you can change the value if you want to apply a specific value locally. In such cases, the maximum size comes from the larger value of the global controls (that is, **Max Size** or **Element Size**, as described above) *or*, the largest scoped body size or face size that Patch Independent is also scoped to. A scoped edge size is not respected if it is larger than either the global size or the size on an attached face.

  With the **Patch Independent** mesh method, scoped body sizing is supported as follows:

  – If the local body size defined is smaller than the global maximum size, the scoped body size is assigned inside the volume.

  – If the global maximum size is smaller than any scoped body, face or edge sizing, the global maximum size (**Element Size** when **Use Adaptive Sizing** is set to **Yes**, **Max Size** when **Use Adaptive Sizing** is set to **No**) is changed to be the same as the largest sizing within the mesher. For example, if Patch Independent is defined on two bodies, and the setup is as follows:

    → Global Max Size = 4

    → Local body size scoped to Body1 = 8

    → No local body size is scoped to Body2
    The Patch Independent maximum size will be 8 and the global Max Size of 4 will be used for the sizing of Body2. You can paramterize **Max Element Size**.

    Note:

    The maximum element size inside the volume of Body2 could grow to 8. Because setting local sizings affects the largest element size in the model, you should avoid setting local sizes that are larger than the global maximum size.

- **Approx Number of Elements per Part**: Prescribes an approximate number of elements for the mesh. The default is 5.0e+005. Specifying a prescribed number of elements for the Patch Independent method is applicable only if the method is being applied to a single part. You can parameterize **Approx Number of Elements per Part**.

- **Feature Angle**: Specifies the minimum angle at which geometry features are captured. If the angle between two faces is less than the specified **Feature Angle**, the edge between the faces are ignored, and the nodes are placed without respect to that edge. If the angle between two faces is greater than the **Feature Angle**, the edge should be retained and mesh aligned and associated with it (note the edge could be ignored due to defeaturing, and so on). You can

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

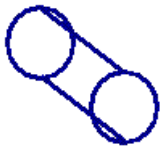
specify a value from 0 (capture most edges) to 90 (ignore most edges) degrees or accept the default of 30 degrees. You can parameterize the **Feature Angle**.

- **Mesh Based Defeaturing**: Ignores edges based on size. The default value is **Off**. When **Mesh Based Defeaturing** set to **On**, **Defeature Size** option is available.

  – **Defeature Size**: Allows you to enter a numerical value greater than 0.0. By default, the value of this local **Defeature Size** option is the same as the global Defeature Size (p. 111). When you specify a different value for the **Defeature Size**, it overrides the global value. Specifying a value of 0.0, resets the tolerance to its default. If multiple Patch Independent tetra mesh method controls are defined with different tolerances, the smallest tolerance is respected. You can parametrize the **Defeature Size**.

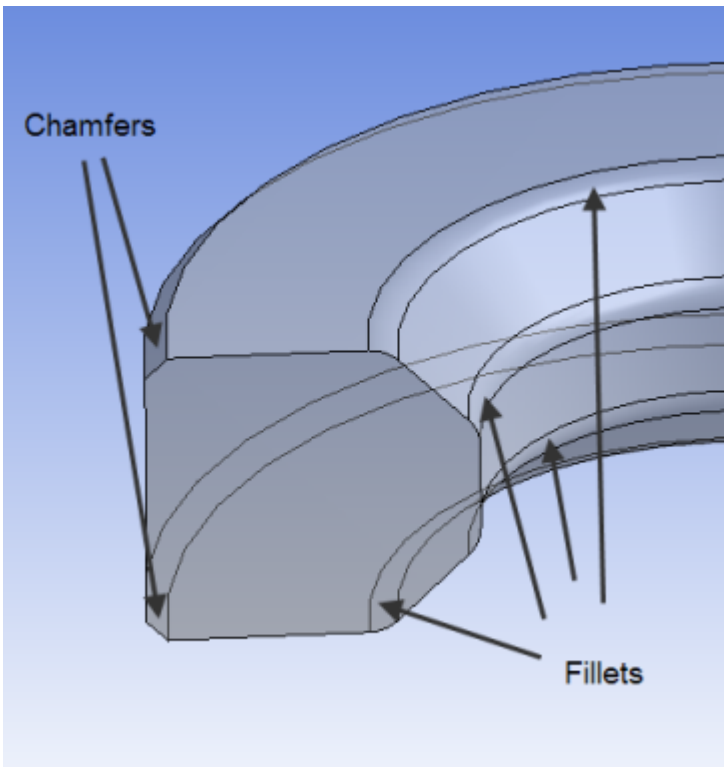
  Some basic cases include the following:

  – A small hole with a diameter smaller than the tolerance as shown below.



  No edges are dropped. You should defeature manually in this case.

  – Two approximately parallel spaced edges (fillet or chamfer), as shown below.



  To determine whether a face is a fillet or chamfer, the Patch Independent mesher evaluates the face's geometric features. To be considered as a fillet or chamfer:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ A face must be at least twice as long as it is width.

→ A fillet or chamfer has either three or four sides (that is, two long sides and one or two short sides), all with angles <= 135 degrees.

In the case of a fillet, which is a curved or rounded face, the angle between the fillet and a face attached to one of its long sides is 0 degrees (not 180 degrees). In contrast, a chamfer is a planar face and the angle between the chamfer and a face attached to one of its long sides is larger than 0 degrees.

For defeaturing of fillets or chamfers, the mesher considers the fillet or chamfer face as well as the faces adjacent to it (the faces attached to its long sides). The dihedral angles between these faces are evaluated to determine whether the attached edges of adjacent faces are respected (that is, whether nodes are placed with respect to the edges at the long sides of the fillet or chamfer).

There are three dihedral angles occurring at a fillet or chamfer:

→ One dihedral angle occurs between the two faces "touching," or adjacent to, the fillet or chamfer face. When this angle is compared with the **Feature Angle**, the angle is measured between the face normals at the imaginary edge where the two faces (virtually) meet.

→ Two dihedral angles occur between the fillet or chamfer face and the respective faces "touching," or adjacent to, the two long sides of the fillet orchamfer. The angles are evaluated as the angles between the face normals at the common edge of the fillet or chamfer and the attached face.

Defeaturing occurs as follows:

→ If the angle between the two faces adjacent to the fillet or chamfer face is greater than the **Feature Angle**, and the angles between the fillet or chamfer face and the faces attached to its long sides are less than the **Feature Angle**, and the minimum fillet or chamfer width is *greater than* the **Defeature Size**, both long sides or edges are respected.

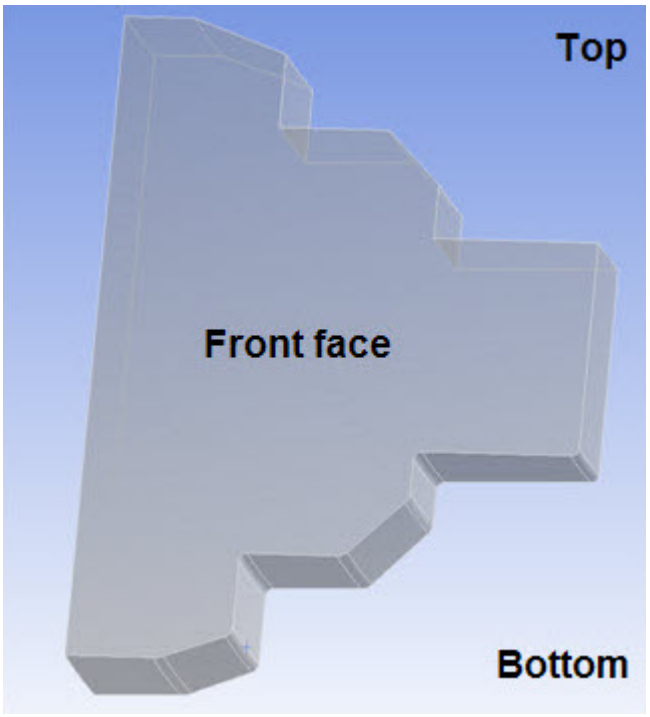
→ If the angle between the two faces adjacent to the fillet or chamfer face is greater than the **Feature Angle**, and the angles between the fillet or chamfer face and the faces attached to its long sides are less than the **Feature Angle**, and the minimum fillet or chamfer width is *less than* the **Defeature Size**, only one long side or edge is respected.

→ If only one angle between the fillet or chamfer face and the faces attached to its long sides is greater than the **Feature Angle**, only one long side or edge is respected.

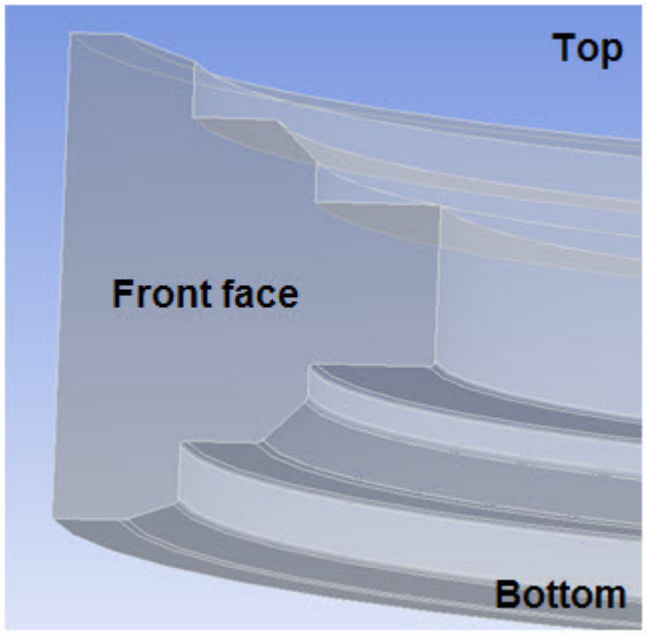
→ If none of the angles are greater than the **Feature Angle**, none of the long sides or edges are respected.

The following series of figures illustrates fillet or chamfer detection. In this example, a cross-section is revolved. The top and bottom of the section are identical, except the bottom has fillets or chamfers at each corner and the top does not. Because the definition of a fillet or chamfer is somewhat general, two cases are presented, each with a different angle of revolution. The angle of revolution is 5 degrees in the first case, as shown below. In this case, only the small faces fit the criteria of fillets or chamfers.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

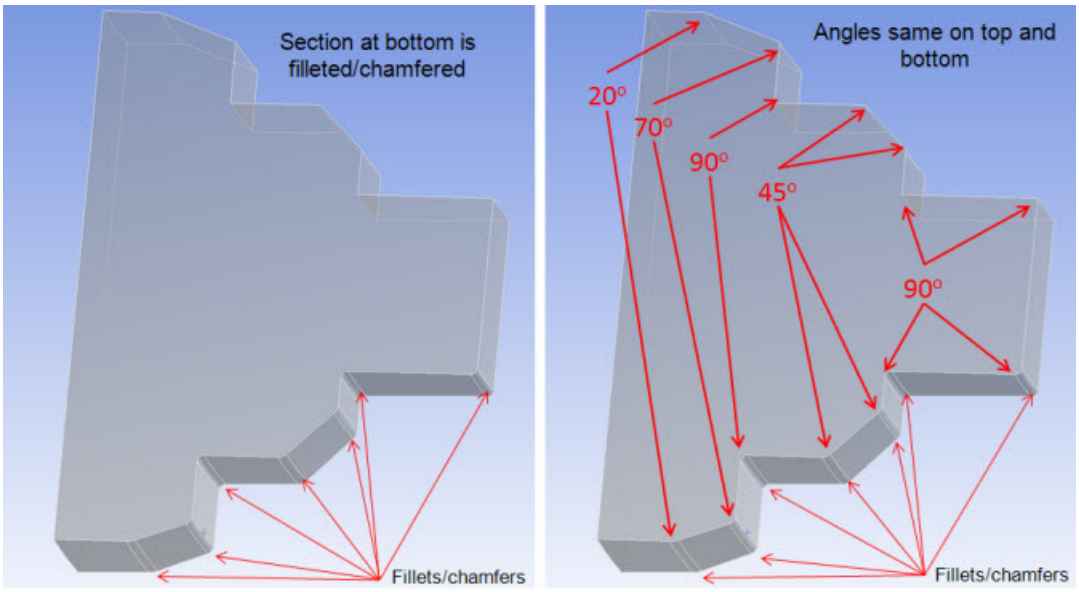


In the second case, the angle of revolution is 180 degrees, as shown below. In this case, all faces fit the criteria of fillets or chamfers, except for the front or back faces of the extrusion.



The figure below shows the angles that are considered for fillet or chamfer detection, and the small faces that are found to be fillets or chamfers.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

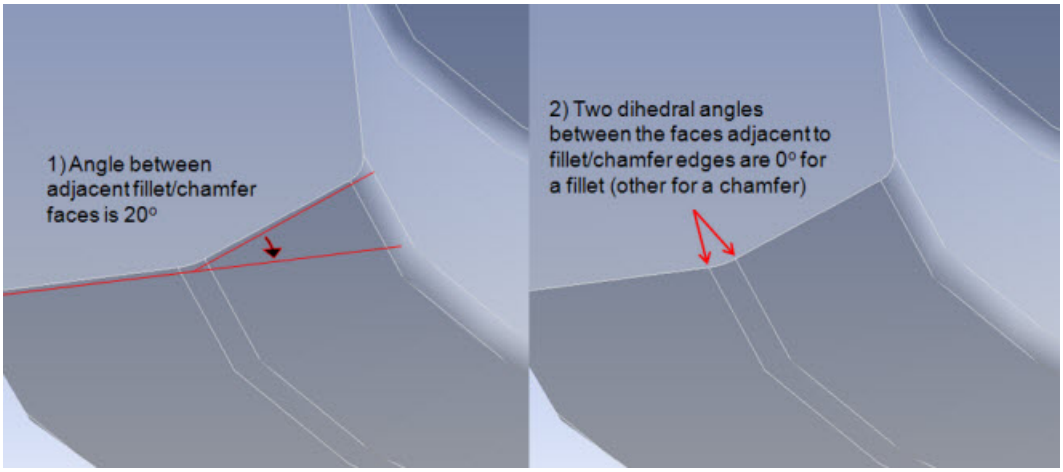


As described earlier, the angles that are considered for a given fillet or chamfer are:

1. the angle between adjacent fillet or chamfer faces.

2. the two angles attached to the fillet orchamfer.

Notice the angles in the figure below.



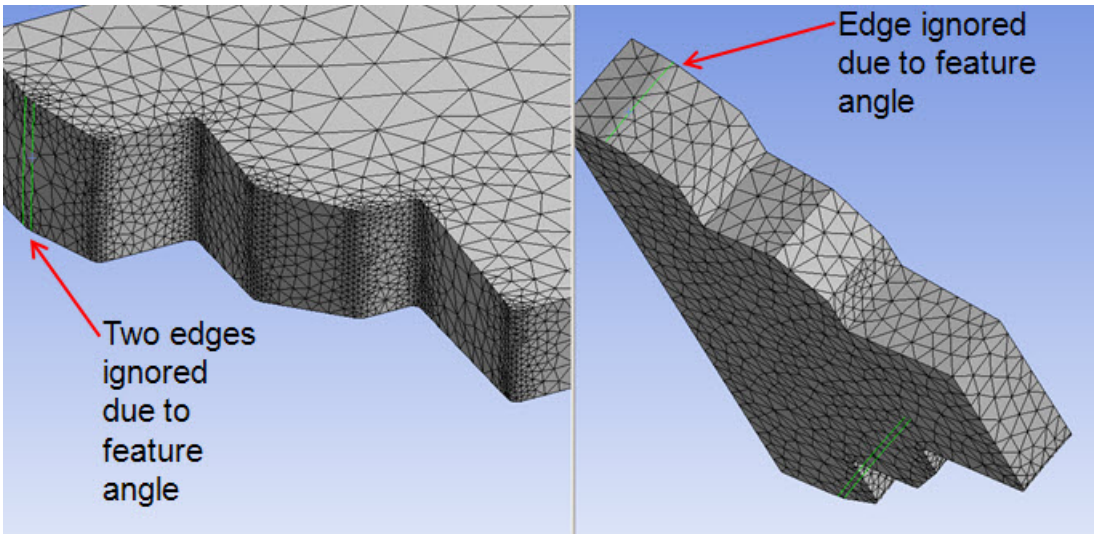
Notice the settings shown below, with **Feature Angle** set to 30 and **Mesh Based Defeaturing** turned off.

Advanced

Defined By	Max Element Size
Max Element Size	8. mm
Feature Angle	30.0 °
Mesh Based Defeaturing	Off
Refinement	Proximity and Curvature
Min Size Limit	1. mm

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the figure below, the highlighted edges are the edges that are ignored with the settings shown above. All edges are captured except for locations where the angle between faces or adjacent fillet/chamfer faces (two bottom edges) is 20 degrees. Changing the **Feature Angle** to a value below 20 results in the mesher capturing those edges, while increasing the angle results in more edges being ignored.

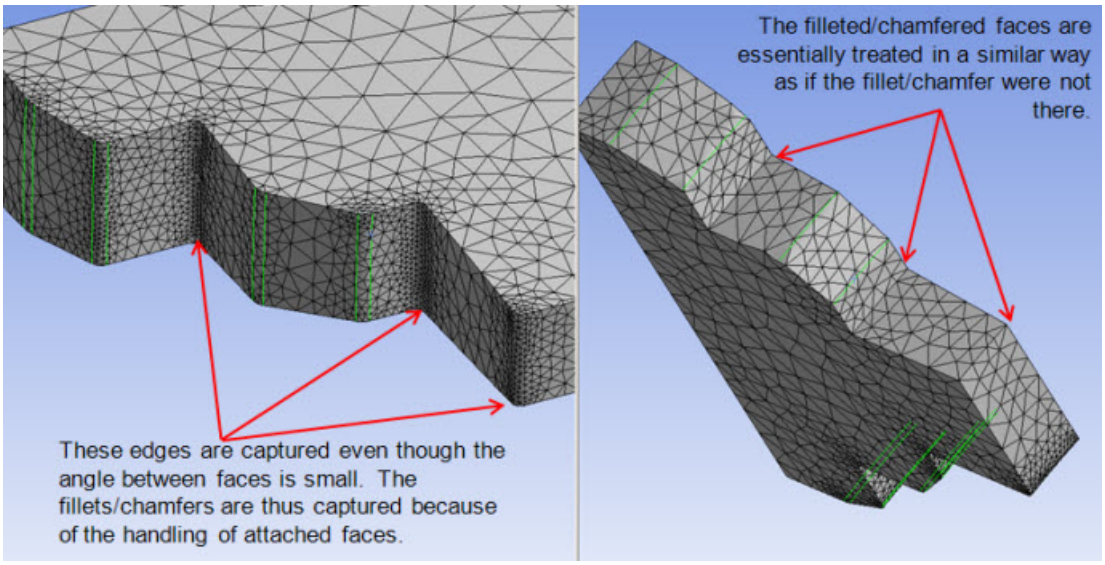


In the settings shown below, **Feature Angle** is changed to 80 but the other settings used before are retained.

Advanced	
Defined By	Max Element Size
Max Element Size	8. mm
Feature Angle	80.0 °
Mesh Based Defeaturing	Off
Refinement	Proximity and Curvature
Min Size Limit	1. mm

In the figure below, the highlighted edges are the edges that are ignored with the settings shown above. All edges are ignored except for those at angles of 90 degrees, both with or without fillets or chamfers.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

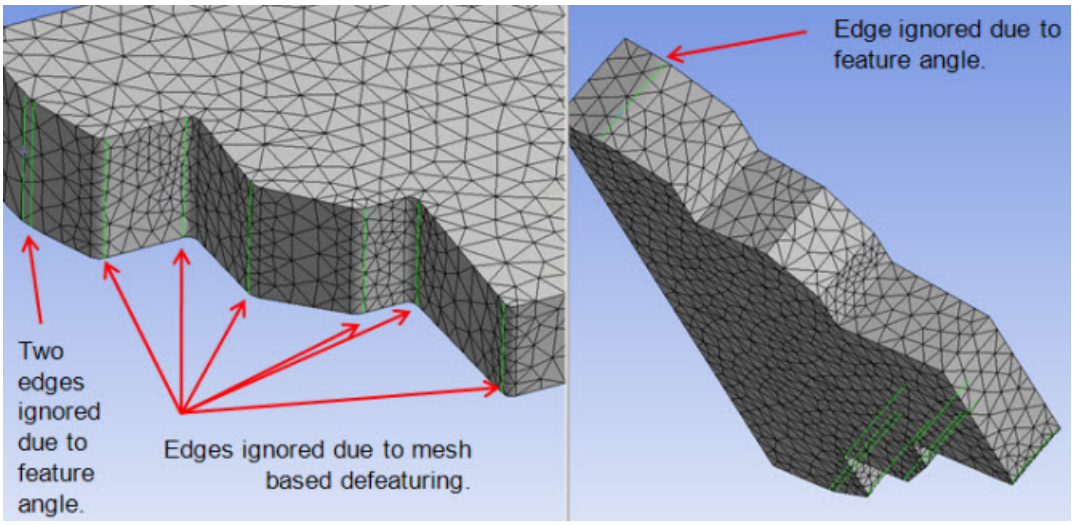


Now, consider the settings shown below. Here the **Feature Angle** is set back to 30, but **Mesh Based Defeaturing** is turned on. Both the **Defeature Size** and the **Min Size Limit** are set to 2.5 mm, which is larger than the bottom fillets or chamfers.

Advanced

Defined By	Max Element Size
Max Element Size	8. mm
Feature Angle	30.0 °
Mesh Based Defeaturing	On
Defeaturing Tolerance	2.5 mm
Refinement	Proximity and Curvature
Min Size Limit	2.5 mm

In the figure below, the highlighted edges are the edges that are ignored with the settings shown above. The same edges as before are ignored due to the feature angle, but in addition every other edge along the bottom fillets or chamfers is ignored.

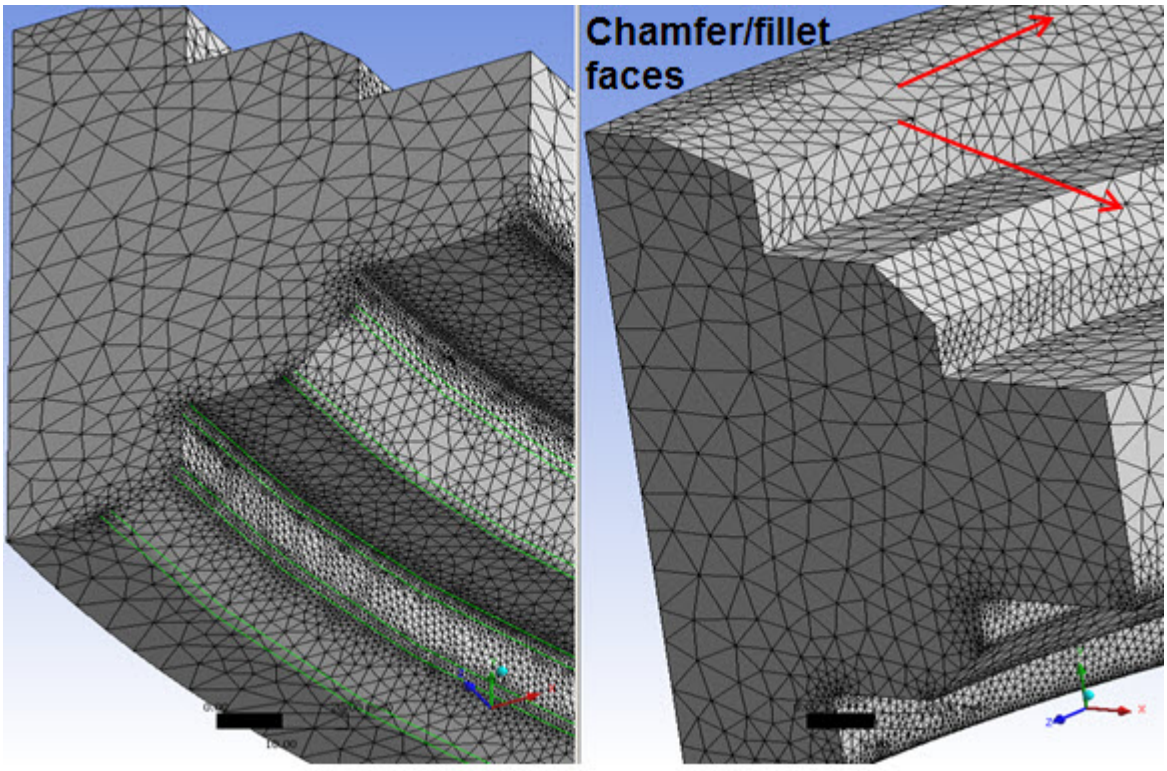


Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The last part of this example involves the case in which the angle of revolution is 180 degrees. Once again the **Feature Angle** is set to 80 but **Mesh Based Defeaturing** is turned off.

Advanced	
Defined By	Max Element Size
Max Element Size	8. mm
Feature Angle	80.0 °
Mesh Based Defeaturing	Off
Refinement	Proximity and Curvature
Min Size Limit	1. mm

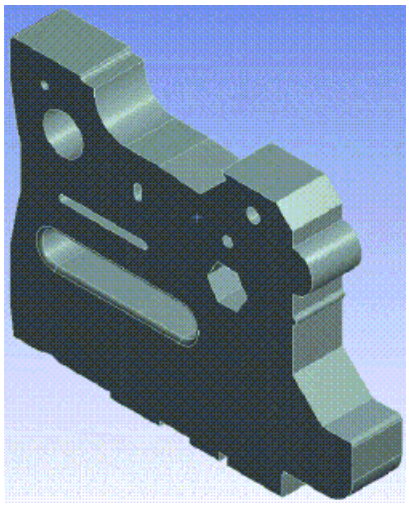
In the figure below, the highlighted edges are the edges that are ignored. With the settings shown above and the longer extrusion, more faces are found to be fillets or chamfers when compared to the case of the shorter extrusion. In comparison, the bottom section is identical as all faces are found to be fillets or chamfers (so the meshing behavior does not change). However, with the inclusion of all faces on top being considered chamfers, the meshing behavior does change.



The following series of figures shows examples of the Patch Independent Tetrahedron mesher with various settings. Figure (a) shows the base geometry.

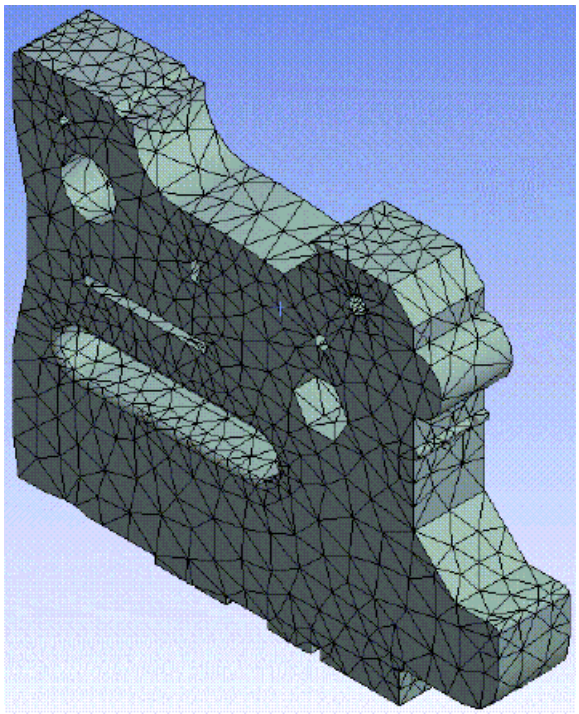
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 84: Example (a) Showing Base Geometry



Figures (b) through (f) below show examples of the Patch Independent Tetrahedron mesher under the conditions noted.

Figure 85: Example (b) Min Size Limit (Described Below) Set to 1



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 86: Example (c) Min Size Limit (Described Below) Set to 0.5

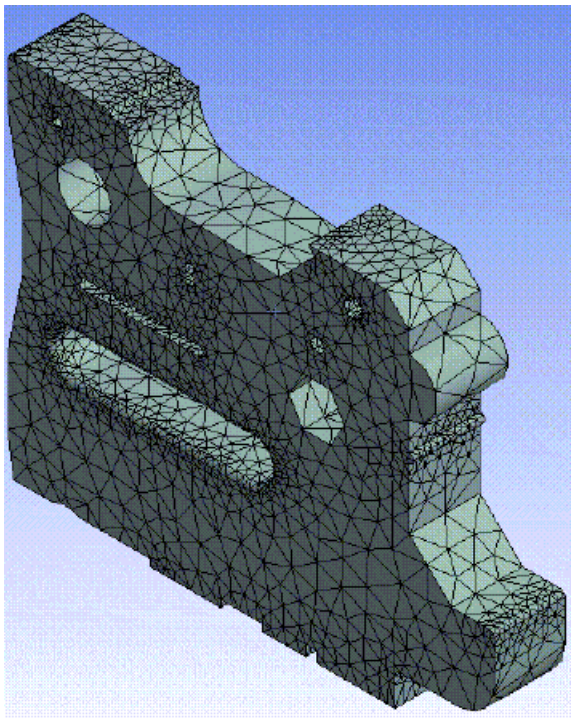
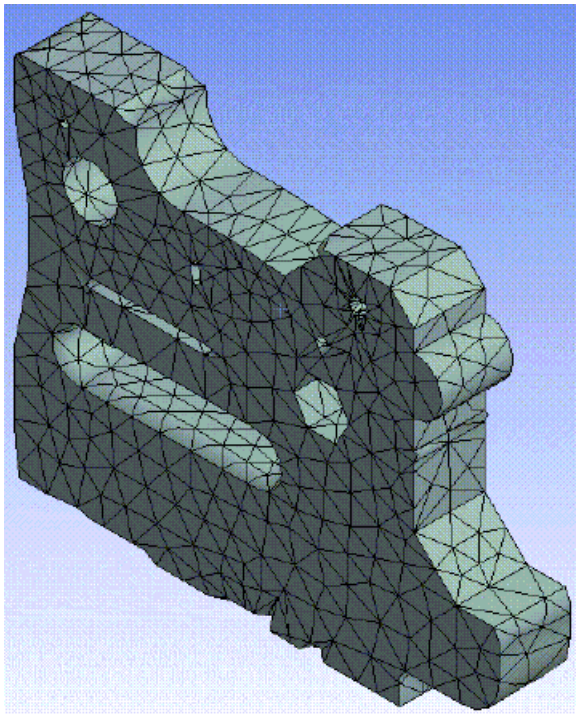


Figure 87: Example (d) Defeature Size Set to 1



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 88: Example (e) Defeature Size Set to 1 and Element Order Set to Linear

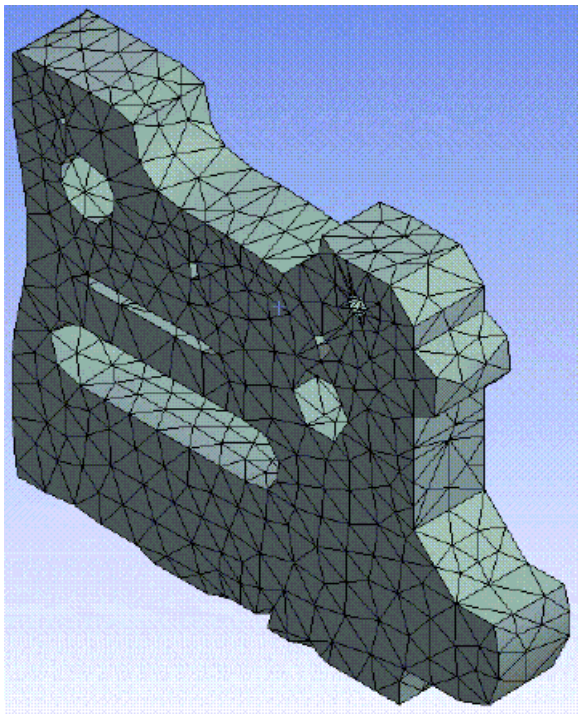
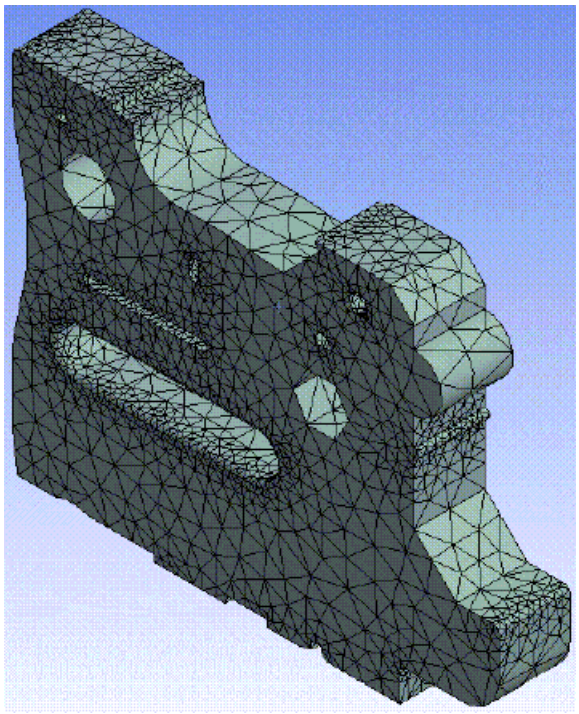


Figure 89: Example (f) Defeature Size Set to 1 and Min Size Limit Set to 0.5



- **Refinement**: Refines mesh automatically based on geometry curvature or proximity or both. This will result in larger elements on flat planar faces and smaller elements in areas of high curvature or within small gaps. The available options are **No**, **Proximity and Curvature**, **Curvature** and **Proximity**. The default value is **Proximity and Curvature**. For **Explicit** physics

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

preference, the default vale is **No**. When you select **Proximity and Curvature**, **Curvature** or **Proximity**, the **Min Size Limit** option is available.

– **Min Size Limit**: Prevents curvature or proximity based refinement from generating elements that are too small.

  Curvature or proximity based refinement subdivides the elements until this **Min Size Limit** is reached. However, projection to geometry and smoothing may push the size even smaller for some elements. The default value of **Min Size Limit** depends on whether **Use Adaptive Sizing** is set to **Yes** or **No**.

  When **Use Adaptive Sizing** is set to **No**, the default value of **Min Size Limit** is inherited from the global **Curvature Min Size** (p. 113) or **Proximity Min Size** (p. 115) control based on the **Refinement** type set. The mesher also uses the Min Size values defined locally.

  → When **Refinement** is set to **Proximity and Curvature** (default), the maximum of the global **Proximity Min Size** (p. 115) and **Curvature Min Size** (p. 113) is used as the default **Min Size Limit**.

  → When **Refinement** is set to **Proximity**, the global **Proximity Min Size** (p. 115) is used as the default **Min Size Limit**.

  → When **Refinement** is set to **Curvature**, the global **Curvature Min Size** (p. 113) is used as the default **Min Size Limit**.

  Note:

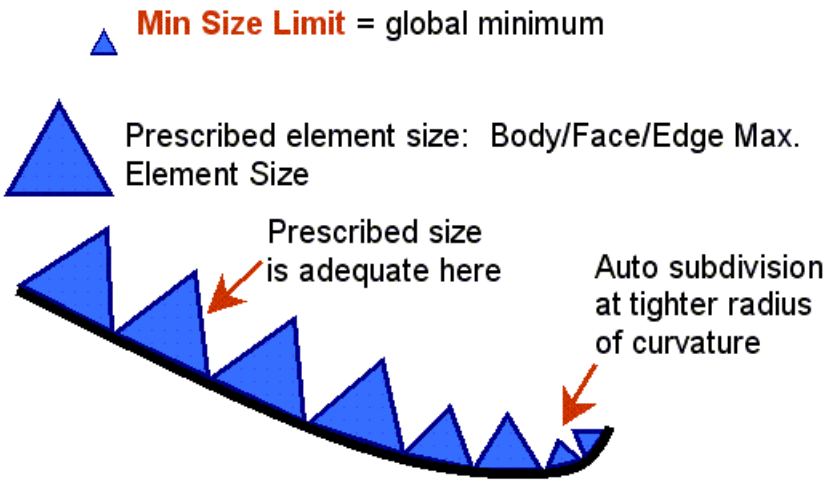
  The global **Curvature Min Size** (p. 113) or **Proximity Min Size** (p. 115) values are used to determine the default value for **Min Size Limit** based on the **Refinement** type set for the **Patch Independent** method control, irrespective of the global refinement type set (for example, even if **Capture Curvature** and **Capture Proximity** are both set to **No**).

  → When **Use Adaptive Sizing** is set to **Yes**, you must specify a value for **Min Size Limit**.

Note:

Also see Notes on Element Size Settings for the Patch Independent Tetra Mesher (p. 231).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

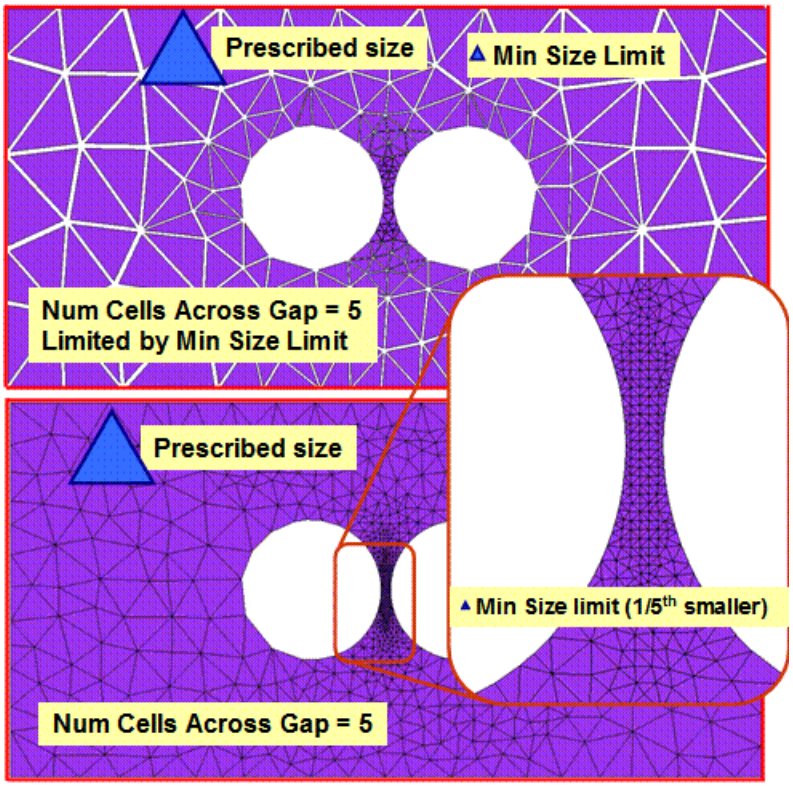


- **Num Cells Across Gap**: Specifies the number of cells desired in narrow gaps. This sets the goal for the proximity based refinement. The mesh will subdivide in tight regions toward this goal, but the refinement is limited by the **Min Size Limit**. It will not override this limit. **Num Cells Across Gap** is only available when **Refinement** is **Proximity and Curvature** or **Proximity**. The default value depends on the Sizing Options (p. 105):

  – When **Use Adaptive Sizing** is set to **No**, the default value of **Num Cells Across Gap** is inherited from the global **Num Cells Across Gap** (p. 115) value.

  – When **Use Adaptive Sizing** is set to **Yes**, the default value of **Num Cells Across Gap** is 3.

  In either case, you can change the value if you want to apply a specific value locally.



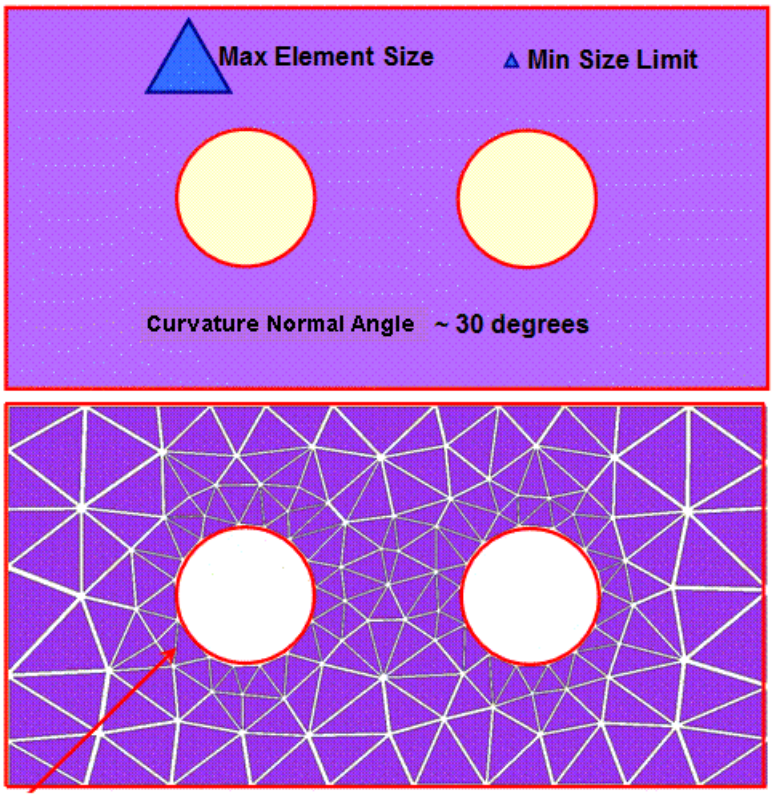
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Curvature Normal Angle**: Sets the goal for the curvature based refinement. The mesh will subdivide in curved regions until the individual elements span this angle. This refinement is also limited by the **Min Size Limit**. You can specify a value from 0 to 180. **Curvature Normal Angle** is available only when **Refinement** is set to **Proximity and Curvature** or **Curvature**. The default value depends on the Sizing Options (p. 105):

  – When **Use Adaptive Sizing** is **No**, the default value of **Curvature Normal Angle** is inherited from the global **Curvature Normal Angle** (p. 114) value.

  – When **Use Adaptive Sizing** is **Yes**, the default value of **Curvature Normal Angle** will be computed based on the value of the Span Angle Center (p. 112) global option.

  In either case, you can change the value to define specific values locally, but be aware that with Patch Independent Tet, only one value for curvature normal angle (num cells in gap) is used by the mesher. The smallest curvature normal angle (largest num cells in gap) is applied globally. A warning message will indicate this for you.



Refinement reaches Curvature Normal Angle goal well above Min Size Limit

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 90: Example (a) Showing Base Geometry

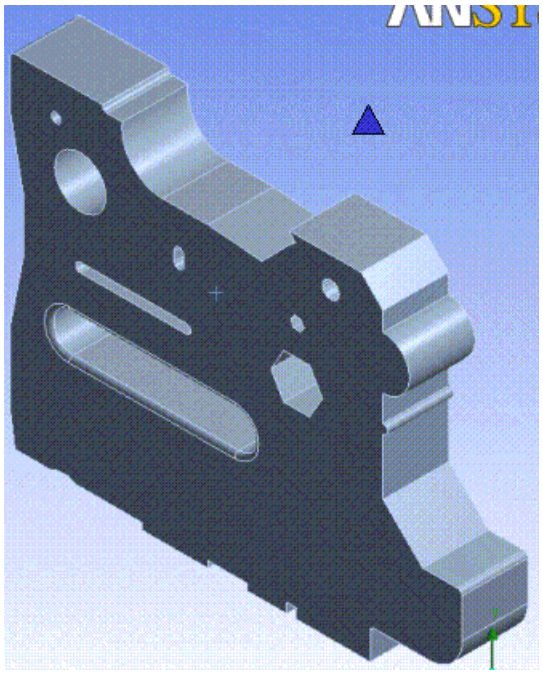
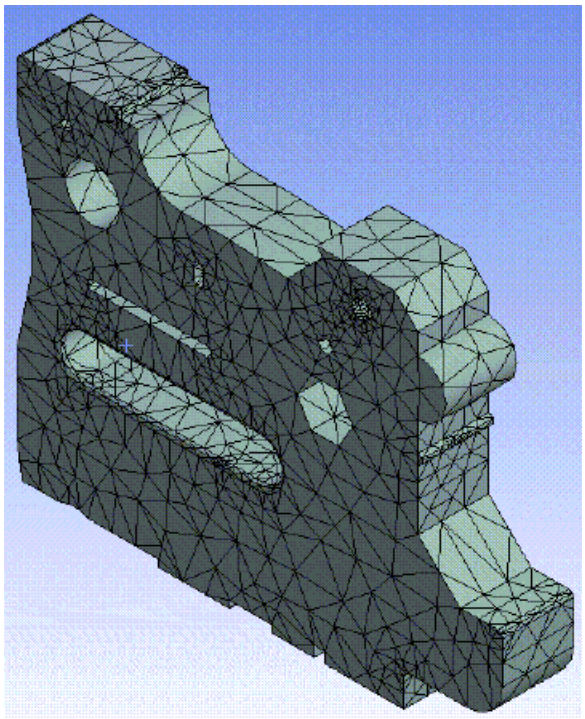
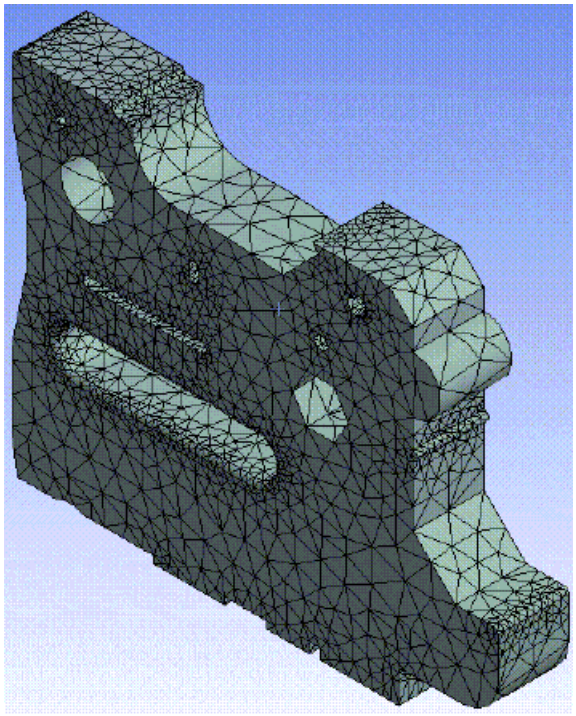


Figure 91: Example (b) Default Patch Independent Tetrahedron Mesher



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

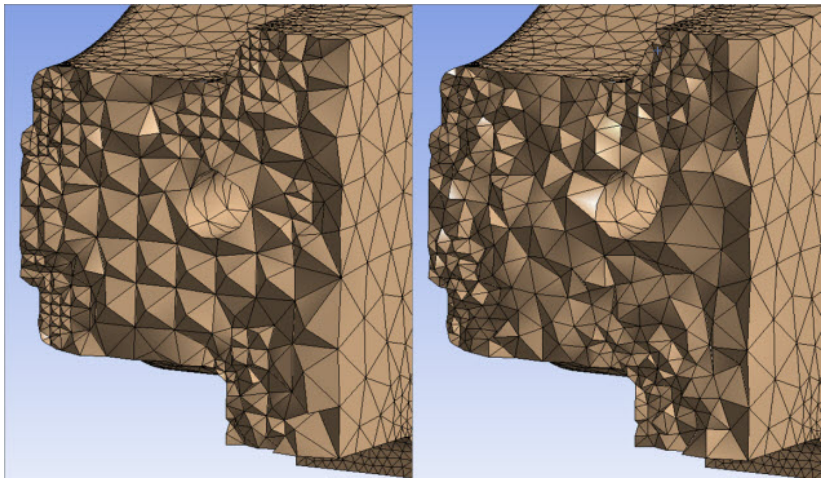
Figure 92: Example (c) Patch Independent Tetrahedron Mesher with Min Size Limit Set to Capture Curvature



- **Smooth Transition**: Determines whether the Octree volume mesh generated from the Patch Independent mesh method should be kept or whether it should be replaced with a Delaunay volume mesh starting from the Patch Independent surface mesh. The available options are **On** and **Off**. The default is **Off**. When **Smooth Transition** is set to **On**, the volume mesh is a Delaunay mesh. When **Smooth Transition** is set to **Off**, the volume mesh is an Octree mesh.

  Figure 93: Effect of Smooth Transition Setting (p. 230) illustrates the effect of setting **Smooth Transition** to **Off** (Octree volume mesh on the left) or **On** (Delaunay volume mesh on the right).

Figure 93: Effect of Smooth Transition Setting



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Growth Rate**: Represents the increase in element edge length 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. Specify a value from 1.0 to 5.0 or accept the **Default**. When set to **Default**, the value is the same as the global growth rate (p. 110). When **Use Adaptive Sizing** is set to **Yes**, the **Default** is set differently based on **Smooth Transition**. When **Smooth Transition** is **Off**, the default is 2.0. When **Smooth Transition** is **On**, the default is 1.2.

  Note:

  If **Smooth Transition** is **Off**, the growth rate is very approximate as the volume is filled with an Octree meshing approach which requires 2-to-1 transitions. Thus in such cases, the growth rate relates only to when the transitions occur through the mesh.

- **Minimum Edge Length**: Read-only indication of the smallest edge length in the part.
- **Match Mesh Where Possible**: The **Match Mesh Where Possible** control is applicable to contact definitions between faces. The available options are **Yes** and **No**. The default is **Yes**. If contact is defined by a single face that topologically belongs to two different bodies, setting this option to **Yes** has no effect. However, if there are independent faces on the two bodies, setting this option to **Yes** causes the Patch Independent mesh method to create nodes on both sides of the contact. The nodes are not connected but have identical coordinates.
- **Write ICEM CFD Files**: Sets options for writing Ansys ICEM CFD files. Refer to Writing Ansys ICEM CFD Files (p. 84) for details.

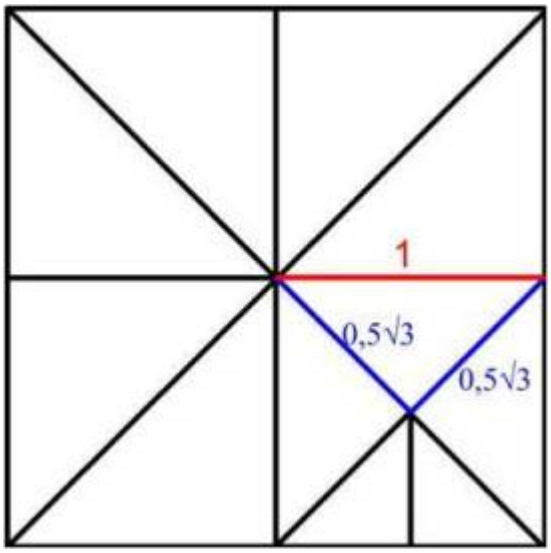
Notes on Element Size Settings for the Patch Independent Tetra Mesher

Remember these notes when using the Patch Independent tetra mesher:

- If you are specifying element sizes with the Patch Independent mesher, you may notice that some element edge lengths are less than the size that you have entered. For example, if your element size is 1, the resulting elements in a uniform tetrahedron mesh will have tetrahedron with edges of length $3^{1/2}/2$ and edges of length 1. A single tetrahedron in this mesh will have two edges of length 1 and four edges of length $3^{1/2}/2$. Two of the three dimensions of the bounding box of this tetrahedron will have length of 1 while the other dimension will have the length of 0.5. This correlates to an element size of 1.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 94: Element Edge Lengths Smaller Than Specified Element Size



- If you are using Curvature and Proximity Refinement, you may notice that your elements are always less than the maximum size specified. Element growth rates with this mesher are always based on powers of 2. For instance, if your minimum size limit is set to 1 and your maximum element size limit is set to 5, and you have curvature in your model that warrants curvature based mesh refinement down to the minimum element size, you will see that the largest elements are not size 5 but size 4. This happens because in order to maintain elements at the minimum size limit, the initial tetrahedron must be some power of 2 larger than the minimum element size, which in this example case is 1.
- Small features of Named Selections are checked in comparison to element size settings prior to meshing. If the minimum element size seems to be too big to capture the essential features of the geometry, a warning is provided if small entities could cause the mesher to fail.

Notes on Scoping for the Patch Independent Mesher

You can use the Patch Independent tetra mesh method in combination with other solid mesh methods in a multibody part, and the bodies are meshed with conformal mesh. Refer to Conformal and Non-Conformal Meshing (p. 21) for information about conformal meshing.

Notes on Virtual Topologies and the Patch Independent Mesher

Virtual topologies may affect the success of meshing with the Patch Independent tetra mesh method. Because virtual topologies are often a coarse approximation of the original faces or edges, the resulting small inaccuracies (gaps and overlaps) may cause the Patch Independent tetra mesher to miss some parts of the boundary of the virtual topology. As a result, the mesher may not accurately model the respected topology and may fail.

Since in general, the Patch Independent tetra mesh method does not require the use of virtual topologies to clean up the geometry, you can remove some of the problematic virtual topology and use Named Selections for boundary conditions instead, as appropriate.

Miscellaneous Notes for the Patch Independent Mesher

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Patch Independent tetra mesh method does not support mesh connections (p. 485), contact matches (p. 496), pinch controls (p. 187), match controls (p. 334), or face meshing controls (p. 314).

Hex Dominant Method Control

Hex Dominant Method creates a free hex-dominant mesh, meshing surfaces with quad elements, extruding them into volume for some distance and then filling the remaining space with unstructured element types.

Note:

When you perform **Explicit Analysis** with **LS-DYNA**, **Hex Dominant Method** is not recommended as it creates unsupported pyramid elements in the core.

To access the **Hex Dominant Method** control,

On the Tree Outline, right-click **Mesh** and click **Insert > Method**.

Or

On the Tree Outline, click **Mesh** and click **Method** in the **Mesh Context** tab on the Ribbon.

Select **Method** as **Hex Dominant Method** in the **Automatic Method** Details view.

Hex Dominant Method Details view has the following options:

Scope

- **Scoping Method**: Allows you to scope geometry bodies or named selection. The default value is **Geometry Selection**.
  - **Geometry Selection**: Allow you to scope the geometry bodies. When you select **Geometry Selection**, the **Geometry** allows you to select the geometry from the **Geometry** window.
  - **Named Selection**: Allow you to scope bodies grouped under a named selection.

Definition

- **Suppressed**: Allows you to suppress the selected control. The default value is **No**. When **Suppressed** is set to **Yes**, **Active** displays the status of the selected control. **Active** is read-only.
- **Method**: Allows you to select the method.
- **Element Order**: Allows you to select the element order. The default value is **Use Global Setting**. For information about the **Element Order** option, refer Method Controls and Element Order Settings (p. 202).
- **Free Face Mesh Type**: Allows you to select the shape of the elements used to fill the body. The available options are **Quad/Tri** or **All Quad**. The default is **Quad/Tri**.
  - **All Quad**: Allows you to create the quadrilateral mesh.
  - **Quad/Tri**: Allows you to create the mesh with both quadrilateral and triangular elements.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Control Messages**: Provides message when the **Hex Dominant Method** fails. You can click **Control Messages** to view the message.

Hex dominant meshing is more suitable for:

- Meshing bodies with large amounts of interior volume where stress concentrations are highest close to the surface of the bodies.
- Rigid body meshes where mesh quality is not important.
- Meshing bodies that require hex elements at the surface and where **Sweep** and **MultiZone** methods have failed to produce a mesh and when you lack time or the relevant skills to split the geometry to create successful mesh with **Sweep** or **MutliZone** methods.

Hex dominant meshing is less suitable for:

- Models where high quality hex mesh is required. For such cases, the recommendation is to use **MultiZone** or **Sweep** methods to achieve high quality results.
- Models where fast transition of the mesh can result in poor solution accuracy (such as CFD models). The Hex dominant approach can have very fast transitions at the core of the volume.
- Models where pyramid elements are invalid as the **Hex Dominant Method** produces a lot of pyramids in the core. For example, Explicit physics calculations.

Note:

- Mesh Matching for cyclic symmetry is not supported for hex dominant meshing.
- Workbench assists you in determining if hex dominant meshing is applicable to your situation. When you apply the **Hex Dominant** option on a body or group of bodies, Workbench calculates the normalized volume to surface area ratio (p. 234). If it detects a ratio less than 2, **Control Message** appears in a highlighted row under **Definition** in the Details View. If you click **Yes, Click To Display**, a warning message states that a low percentage of hex elements or poorly shaped hex elements may result. Suggestions are included for alternative meshing schemes.

  The normalized volume to surface area ratio is defined by the following expression:

  (Volume of body/(Surface area of body)$^{3/2}$)/factor

  where factor, the ratio for a unit sphere = (4/3 π)/(4 π)$^{3/2}$
- Adaptive refinement starting from a hex dominant mesh will result in remeshing of the structure with tetrahedrons.
- Use Body or Face Sizing to obtain more uniform face meshing, which leads to more hexes by volume.
- If you apply a local **Sizing** control (p. 296) to a solid body with a **Method** control set to **Hex Dominant** (p. 233) or **Sweep** (p. 235), or to a sheet body with a **Method** control set to **Quadrilateral Dominant** (p. 282), a near uniform quadrilateral mesh results on

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - all affected faces on a body meshed with **Hex Dominant**
  - the source face meshed with **Sweep**
  - all affected faces meshed with **Quadrilateral Dominant**

  To obtain even more of a uniform quadrilateral mesh, set the **Behavior** (p. 310) of the **Sizing** control to **Hard**.

- Meshing thin and complicated bodies (for example a cellular phone case), using **Hex Dominant Method** requires much smaller element sizes to give good quality hex elements. As a result, the number of elements may increase compared to using a tetrahedron mesh.

Sweep Method Control

Sweep method generates sweep mesh on "sweepable" bodies (p. 399) (including axis-sweepable bodies, which are not displayed when you use the **Show Sweepable Bodies** feature). The mesher fails, when a swept mesh cannot be generated on a body with a **Sweep Method** control.

Use the **Sweep** option when you require:

- a swept mesh.
- a swept mesh on a model that revolves around an axis where the source and target faces share topology.

To access the **Sweep Method**,

On the **Tree** Outline, right-click **Mesh** and click **Insert > Method**.

Or

On the **Tree** Outline, click **Mesh** and click **Method** in the **Mesh Context** tab on the Ribbon, then click **Method**. Set **Method** as **Sweep** in **Automatic Method** Details view.

Sweep Method Details view has the following options:

Scope

- **Scoping Method**: Allows you to scope geometry or named selection. The available options are **Geometry Selection** and **Named Selection**. The default value is **Geometry Selection**.
  - **Geometry Selection**: Allow you to scope the geometry bodies.
  - **Named Selection**: Allow you to scope the bodies grouped under a named selection.

Definition

- **Suppressed**: Allows you to suppress the method control. The default value is **No**. When **Suppressed** is set to **Yes**, the **Active** is available. **Active** is a read-only and provides the number of suppressed parts

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Method**: Allows you to select the method control.

When you select the **Sweep** option, the Details View expands to include additional settings, many of which are unique to this option. For basic usage that involves obtaining a swept mesh, the procedure is to apply a **Method Control** to one or more bodies, set **Method** to **Sweep**, and accept the default values of the various settings.

For advanced or specialized usage such as meshing thin models or axis sweeping, adjust the settings as needed. The following is a description of each of these settings.

- **Algorithm**: Allows you to select algorithm. The available options are **Program Controlled** and **Axisymmetric**. The default is **Program Controlled** .
  - **Program Controlled**: Applies to only on the traditional sweepable bodies which can be detected by **Show Sweepable Bodies**.
  - **Axisymmetric**: Applies to all axisymmetric models. Axisymmetric Sweeper supports shared topology. Axisymmetric Sweeper supports **Contact Sizing** (p. 311) and **Topology Protection** (p. 184).

Note:

Mesh Based Defeaturing (p. ?) is available only when **Axisymmetric** algorithm is selected.

- **Mesh In Center**: Defines the type of mesh in the centre of the Axisymmetric model. **Mesh in Center** is available only when **Algorithm** is **Axisymmetric**. The default value is **Hexahedra**. The available options are:
  - **Wedges**: Generates wedges at the center of the axisymmetric model.
  - **Hexahedra**: Generates O-Grid mesh at the centre of the axisymmetric model. You can specify the number of divisions for the O-Grid using the **O-Grid Divisions**.
- **Element Order**: Allows you to select element order for mesh. Refer to Method Controls and Element Order Settings (p. 202).
- **O-Grid Divisions**: Define the number of divisions per quadrant from the center of the O-Grid to the boundary (or outside) of the O-Grid. The default value for **O-Grid Divisions** is 4. **O-Grid Divisions** is available only when you select **Axisymmetric** algorithm.
- **Project Corners to Top**: Projects the nodes on the steps along the body of interest to the top surface of the Axisymmetric model. The default value is **Yes**. **Project Corners to Top** is available only when **Algorithm** is **Axisymmetric** .
- **Src/Trg Selection**: Defines the source and target selection type according to the following choices:
  - **Automatic**: Determines automatically the best source and target for the body.
  - **Manual Source**: Allows you to select the source and the program determines the target. **Manual Source** is useful when there are multiple source target pairs, and you want to specify the source to get the correct bias through the sweep direction. Another application is when

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

your cross section is changing, and the mesh quality would be better when sweeping from one side vs. another.

– **Manual Source and Target**: Allows you to select the source and target manually. The sweeper revolves the mesh around common edges or vertices. You can use **Manual Source and Target** when you want to sweep a body where the source and target faces share vertices or edges or both.

– **Automatic Thin** (p. 406): Allows you to select thin models and thin sheet metal parts and sweep a mesh through the thickness of the body. For **Automatic Thin**, the face with the largest face area is selected as the primary source and the algorithm determines the rest of the source faces. You can define multiple elements through the thickness using the **Sweep Num Divs** control. Biasing is not available. **User Defined Criteria** allows you to select the type of **Automatic Thin** sweep. This option is available only when the **Src/Trg Selection** is set to **Automatic Thin**. The available options are **Program Controlled** and **Protect Internal Edges**. When you select **Program Controlled**, the interior loops in the model are not preserved while performing thin sweep. When you select **Protect Internal Edges**, the interior loops in the model are preserved while performing thin sweep. An **Element Option** setting is included that instructs the solver to use the **Solid Shell** element where possible, or to always use a **Solid** element.

– **Manual Thin** (p. 406): The same restrictions apply as described above for **Automatic Thin**. However, with this choice, you can do *any* of the following:

→ Pick one source face and allow the program to determine the rest.

→ Pick all the source faces and allow the program to do nothing but mesh the source faces and sweep them to the target.

→ Pick multiple source faces and mesh one target face.

Note:

→ The **Sweep** mesh method does not support the **Manual Source**, **Manual Source and Target**, or **Manual Thin** settings for **Src/Trg Selection** if **Sweep** is applied to more than one part, even if you suppress all of the other parts.

→ In some cases, the thin model sweeper may want to swap source and target faces based on meshing conditions in neighboring bodies. In such cases, you receive a warning message.

Refer to Considerations for Selecting Source Faces for the Thin Model Sweeper (p. 407) for details.

→ To make source/target face selection easier, select **Annotation Preferences** from the Toolbar and then deselect **Body Scoping Annotations** in the **Annotation Preferences** option box to toggle the visibility of annotations in the **Geometry** window. For example, after scoping **Sweep** to a body, the body will be displayed using a blue solid annotation. Turn off the body scoping annotations, then select the source/target faces. For picking internal faces, the **Hide Faces** right-click option may help you to see inside a body. For example, you can select external faces in the **Geometry** window and then use the **Hide Faces** option to hide the selected faces (making it easier to select the internal faces).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Source Scoping Method**: Defines the method for choosing a source face or target face. **Source Scoping Method** is **Program Controlled** and read-only when you select **Src/Trg Selection** as **Automatic** or **Automatic Thin**. The available options are **Geometry Selection** and **Named Selection**.

  – **Geometry Selection**: Enables you to select sources or targets manually using the Source option.

  – **Named Selection**: Enables you to choose one **Named Selection** as a source or target using the **Source Named Selection** option.

- **Target Scoping Method**: Defines the method for scoping targets when **Src/Trg Selection** is **Manual Source** and **Target**. The available options are **Geometry Selection** and **Named Selection**.

  – **Geometry Selection**: Allows you to select the target from the **Geometry** window using **Target** option .

  – **Named Selection**: Allows you to select the target from the available named selections using **Target Named Selection** option.

- **Free Face Mesh Type**: Determines the shape of the elements used to fill the swept body (pure hex, pure wedge, or a combination of hex/wedge). The default value is **Quad/Tri**. The available options are :

  – **All Tri**: Fills the swept body with triangular elements.

  – **Quad/Tri**: Fills the swept body with quadrilateral or triangular elements.

  – **All Quad**: Fills the Swept body with quadrilateral elements.

  When **Src/Trg Selection** is **Automatic**, **Manual Source**, or **Manual Source and Target**,**Free Face Mesh Type** allows you to choose **All Tri**, **Quad/Tri**, or **All Quad** meshing

  When **Src/Trg Selection** is **Automatic Thin** or **Manual Thin**, **Free Face Mesh Type** allows you to choose **Quad/Tri** or **All Quad** meshing .

  Note:

  – If the source face is also a side face of another body, the face is always quad mapped.

  – When **Free Face Mesh Type** is set to either **Quad/Tri** or **All Quad** and the source face can be mapped meshed, the face is sometimes mapped meshed ignoring the applied sizing controls (such as **Contact Sizing** (p. 311), **Sphere of Influence** (p. 304),and more).

  – In some cases, when **Src/Trg Selection** is set to **Automatic**, automatically selected source face should be quad mapped to get a successful sweep mesh. In such cases, the value specified for **Free Face Mesh Type** may be ignored.

- **Type**: Allows you to specify whether to use **Number of Divisions** or **Element Size** through the sweep direction. The available options are:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Element Size**: Specifies the size of elements to be created in the sweep direction.

→ **Sweep Element Size**: Provides the size of elements created in the sweep direction.

– **Number of Division**: Specifies the number of elements to be created in the sweep direction.

→ **Sweep Num Divs**: Provides the number of elements to be created in the sweep direction.

When sweeping generalized bodies that share faces, the **Element Size** is a soft constraint on interval assignment and the **Number of Divisions** is a hard constraint. If you have conflicting **Number of Divisions** constraints, the sweeper will fail and yield a message. To obtain a regular mesh in the sweep direction, the guiding edges must have consistent lengths. You can define virtual split edges to achieve consistent lengths for these edges (see Creating and Managing Virtual Split Edges (p. 561)). Also see Sizing Control (p. 296) for more information.

Note:

The Type **Element Size** is not available for **Axisymmetric** algorithm.

- **User Defined Criteria**: Allows you to select the type of **Automatic Thin** sweep. **User Defined Criteria** is available only when the **Src/Trg Selection** is **Automatic Thin**. The default option is **Program Controlled**. The available options are:

  – **Program Controlled**: Performs thin sweep without preserving the interior loops in the model.

  – **Protect Internal Edges**: Performs thin sweep preserving the interior loops in the model.

- **Element Option**: Instructs the solver which the element option to be used. The default option is **Solid Shell**. The available options are **Solid Shell** and **Solid**.

  – **Solid Shell**: Instructs the solver to use the solid shell elements where possible.

  – **Solid**: Instructs the solver to always use a solid element.

- **Control Messages**: Provides message when the sweep method fails.

Advanced

- **Sweep Bias Type**: Specify bias in the same manner as edge biasing for the **Bias Type** setting in a **Sizing** (p. 296) mesh control. There is no graphical feedback for biasing on a **Method** control. Biasing direction is from the source to the target.

Note:

Sweep Bias Type is not available for **Axisymmetric** algorithm.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mesh Based Defeaturing**: Enable or disable the defeaturing of small features. The default value is **Off**. The available options are **On** and **Off**. When **Mesh Based Defeaturing** is **On**, features smaller than or equal to the value of **Defeature Size** are removed automatically.

  – **Defeature Size**: Specifies the tolerance value for defeaturing.

- **Fall-Back Mesh Type**: Provides the option to select the mesh type to be used when the **Axisymmetric Sweep** fails. The default option is **Program Controlled**. The available options are:

  – **Program Controlled**: Provides error message when **Axisymmetric Sweep** fails unless you choose tetrahedral meshing.

  – **None**: Provides error message when **Axisymmetric Sweep** fails.

  – **Tetrahedrons**: Generates tetrahedral mesh when **Axisymmetric Sweep** fails.

- **Constrain Boundary**: Specifies whether you want to allow the mesher to split the elements at the boundary of a swept mesh region to aid in meshing. Available for multibody parts only (both for general sweeping and thin sweeping). When **Constrain Boundary** is **Yes**, does not split the elements at the boundary while sweep meshing. (constrain boundary, no splitting is allowed). When **Constrain Boundary** is **No**, splits the elements at the boundary while sweep meshing. (do not constrain boundary, splitting is allowed). Choosing **Yes** prevents tets from entering the swept body. The default is **No**.

Note:

Constrain Boundary setting is not available for **Axisymmetric** algorithm.

Note:

- For gasket simulations, set the **Stiffness Behavior** of the body to **Gasket** and proceed with adjusting mesh settings as described in the Gasket Meshing section located under Gasket Bodies in the Mechanical application help.

- In models with swept regions, the sizing controls will affect the mesh gradation in the swept region. You can override this effect by specifying any **Sweep Bias** value (including a value of 1), **Sweep Element Size** value, or **Sweep Num Divs** value in the Details View when defining the sweep method.

- There is a system limitation when using the sweep method with the Size Function (p. 89). The Size Function may have nodes slightly off because the spacing is queried. The sweeper then tries to match that spacing, which may lead to unexpected mesh results.

- If you apply a local **Sizing Control** (p. 296) to a solid body with a **Method** control set to **Hex Dominant** (p. 233) or **Sweep** (p. 235), or to a sheet body with a **Method** control set to **Quadrilateral Dominant** (p. 282), a near uniform quadrilateral mesh will result on all affected faces on a body meshed with **Hex Dominant**, on the source face meshed with **Sweep**, and on all affected faces meshed with **Quadrilateral Dominant**.

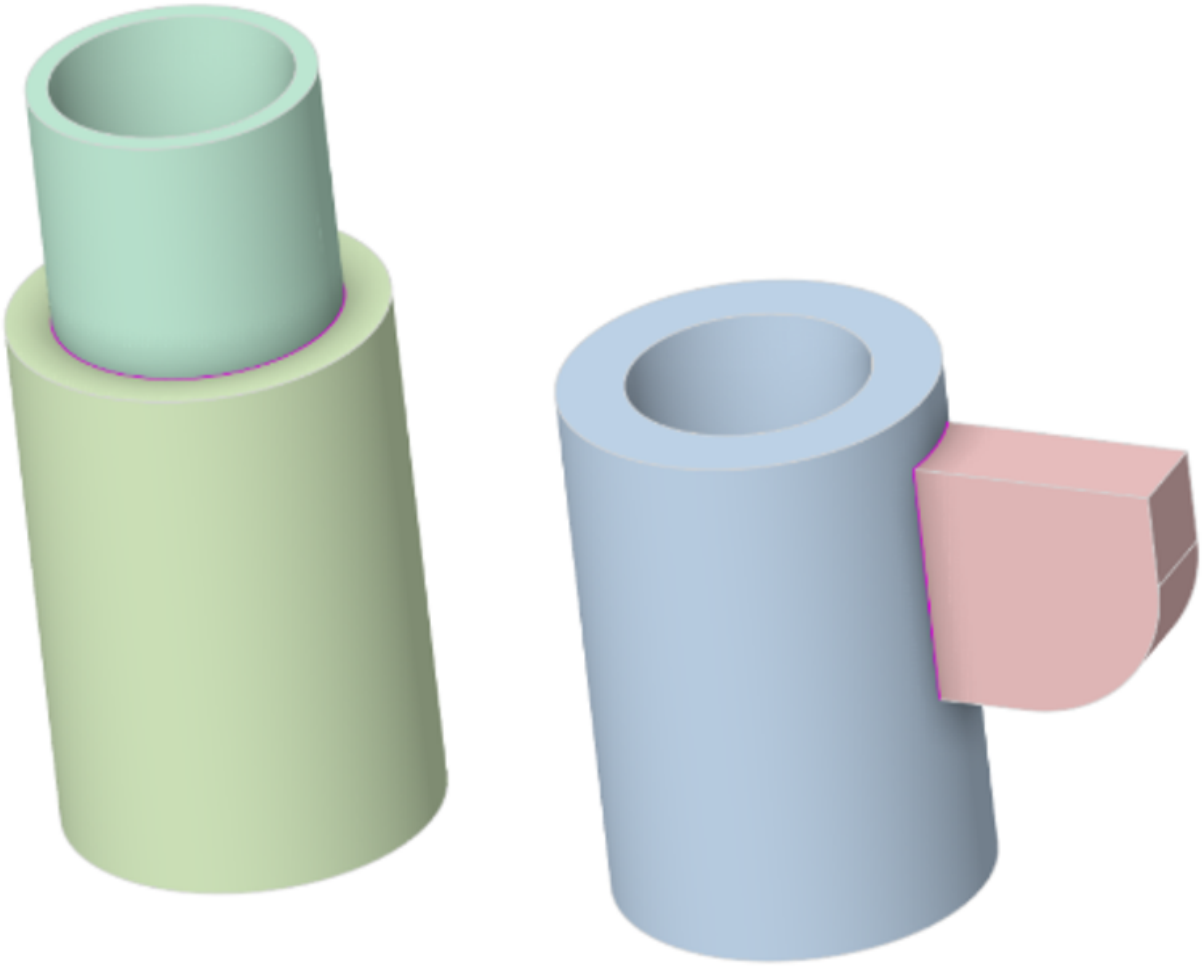
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To obtain even more of a uniform quadrilateral mesh, set the **Behavior** (p. 310) of the **Sizing Control** to **Hard**.

Limitations

Axisymmetric sweep has the following limitations:

- A warning message appears, when the nodes are off the mesh. You can click the warning message to locate that area on the model and can redefine mesh using **Mesh Based Defeaturing** (p. ?) for the particular area.
- **Axisymmetric** algorithm does not support **Surface of Influence**.
- **Axisymmetric** sweep fails when meshing shared topologies having a sweepable and a non-sweepable body (For example, bodies with imprints on sides). You may need to mesh the individual bodies separately.



1

2

Model 1 and 3 supports **Axisymmetric** sweep. **Axisymmetric** sweep fails on model 2 as the model has a side imprint that is not an axisymmetric model.

- In a multibody part, when you have **Axisymmetric** sweep and **MultiZone** method applied on shared topology bodies, and you mesh all bodies simultaneously, **Axisymmetric** sweep may fail if **MultiZone** fails to create O-Grid on the interfaces between the bodies where both methods are applied. Here, the mesher follows the following order while applying mesh methods on the bodies:

  1. MultiZone

  2. Axisymmetric Sweep

  3. Sweep

  4. Thin Sweep, Tetrahedron, Hex- Dominant

  In such cases, you may need to mesh the body by body. When you use **Generate Mesh on Selected Bodies** or **Mesh Worksheet** defining the meshing sequence for each body, **Axisymmetric** may fail in the following cases:

  – Executing low level mesh methods first.

  – Meshing bodies with large O-Grid patterns before the ones with small O-Grid pattern in solid bodies.

- **Axisymmetric** algorithm only supports some edge biasing along the axial direction.
- **Axisymmetric Sweep** cannot handle any body with internal voids.
- **Axisymmetric Sweep** does not support **Mixed Order Meshing**.

MultiZone Method Control

MultiZone Method Control provides automatic decomposition of geometry into mapped (sweepable) regions and free regions. When you select **MultiZone Method Control**, all regions are meshed with a pure hexahedral mesh if possible. To handle cases where a pure hex mesh is not possible, you can adjust your settings to generate sweep mesh in the structured regions and a free mesh in unstructured regions.

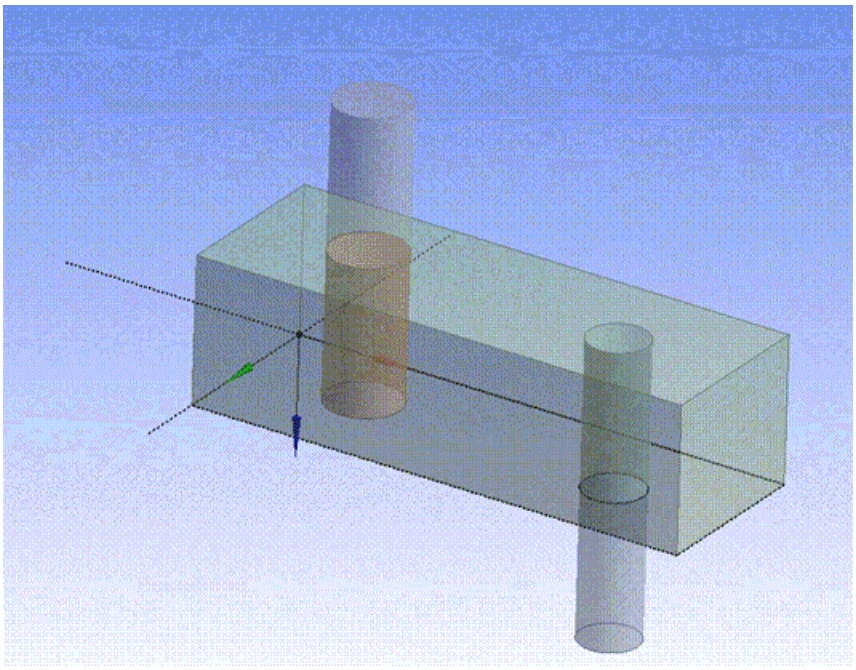
For example, using the **Sweep** mesh method, you would need to slice the part below into five bodies as shown to obtain a pure hex mesh:

Note:

This section describes method control settings. See MultiZone Meshing (p. 419) for detailed algorithm and usage information.

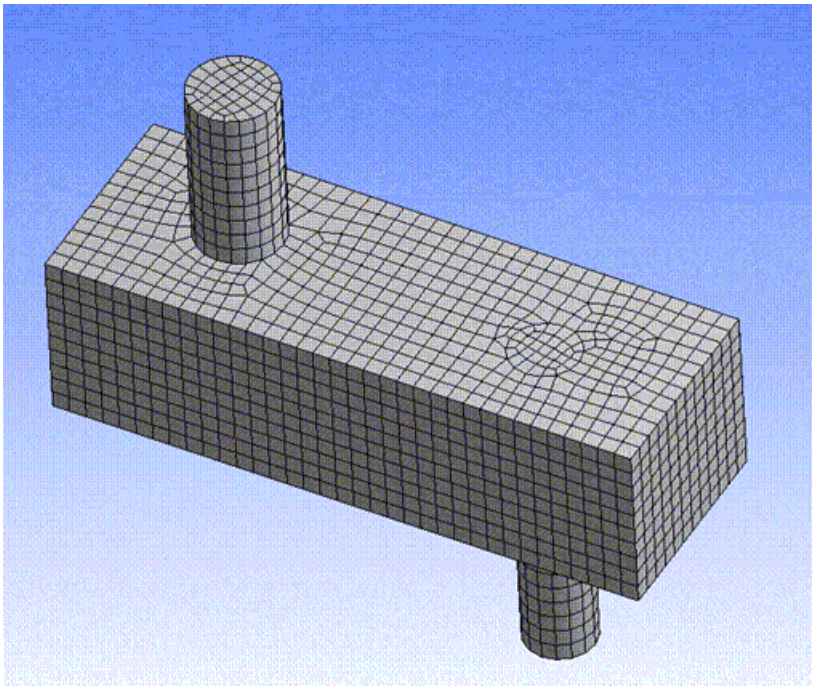
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 95: Sweep Method Would Require Slicing to Obtain Pure Hex Mesh



In contrast, using **MultiZone** requires no slicing. **MultiZone** automates the geometry decomposition and generates the pure hex mesh shown in Figure 96: MultiZone Generates Pure Hex Mesh without Slicing (p. 243).

Figure 96: MultiZone Generates Pure Hex Mesh without Slicing



When you select the **MultiZone** mesh method, the Details View expands to expose various settings, including several that are unique to **MultiZone**. For basic usage that involves obtaining a **MultiZone** mesh, the procedure is to apply a **Method Control** to one or more bodies, set **Method** to **MultiZone**, and accept the default values of the various settings.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For advanced or specialized usage, adjust the settings as needed. **MultiZone Method** Details view has the following options:

Scope

- **Scoping Method**: Allows you to scope geometry or named selection. The default value is **Geometry Selection**. The available options are:
  - **Geometry**: Allows you to scope the geometry for MultiZone method.
  - **Named Selection**: Allows you to scope the available named selections for MultiZone method.

Definition

- **Suppressed**: Allows you to suppress the method control. The default value is **No**. When **Suppressed** is set to **Yes**, the **Active** field is available. **Active** is a read only field and provides the number of suppressed parts.
- **Method**: Allows you to select the method.
- **Decomposition Type**: Allows you to define the decomposition type that MultiZone use to automatically break geometry into blocks. The available options are **Program Controlled** , **Thin Sweep**, **Cart Sweep**, **Standard** and **Medial Axis**. The default option is **Program Controlled**.
  - **Program Controlled**: Determines automatically the decomposition type for each body and meshes them accordingly. For single bodies, **Program Controlled** first mesh all axisymmetric bodies, then mesh thin bodies using **MultiZone Thin Sweep** and mesh the rest of the bodies with **MultiZone Standard** option. When you have multiple bodies in a multibody part and you generate mesh simultaneously for all the bodies, **Program Controlled** uses the same **Decomposition Type** for all connected bodies. If you mesh body by body using **Generate Mesh on Selected Bodies** or **Mesh Worksheet** defining the meshing sequence for each body, the mesher detects the body type and based on which the MultiZone decides the **Decomposition Type** to be used. Hence, meshing bodies together and individually can provide different results.

    Note:

    Program Controlled does not support cart sweep bodies. You may have to mesh the cart sweep bodies separately using **MultiZone Cart Sweep**.

  - **Thin Sweep**: Detects sweep source and target surfaces and creates the 2D blocking on the source surface. Then, the 2D blocking is pulled to 3D and associated to the target surfaces in an automatic fashion. You should apply **Thin Sweep Decomposition** to mesh geometries identified as thin-walled solids, as an alternative to using the typical approach

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

with **Standard** method. You can use the **Select Source/Target Faces** tool to select the sweep surfaces.

Note:

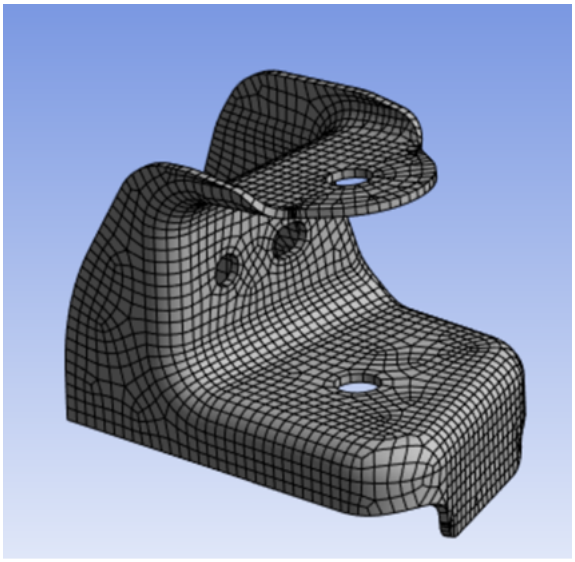
Partial selection of source and target faces may return partial and inconsistent mesh. Hence, full selection of sources and targets is recommended.

When you select **Decomposition Type** as **Thin Sweep**, the **Sweep Thickness** is available.

→ **Sweep Thickness**: Allows you to provide the thickness of the model. The default value is 0. If the value is 0, the application identifies the thickness automatically. If the mesh fails, you may have to provide the thickness. For models with varying thickness, you should provide thickness value slightly greater than the maximum thickness of the model. If you select the sweep source (top) and target (bottom) surfaces, the **Thickness** value is not used. You can parametrize **Sweep Thickness**.

Note:

The software automatically recognizes the source, side and target faces, then meshes the source set of faces and pulls the sheet blocking to the target set of faces. If the source, side, and target faces cannot be detected correctly, you receive a warning message to select the source or target faces manually.



When no side or only one side (or at least one surface on the side, algorithm checks and completes selection if required) of the solid is selected, the following priority is used to determine which (non-thin) side of the solid is a sweep source:

1. manually selected faces
2. shared faces or faces with input mesh
3. non-smooth faces

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. faces containing more features
5. faces with the largest area

The software attempts to automatically determine the side faces and remaining faces are the target set of faces.

Alternatively, select the source and target set of faces as the source faces. This should be two sets of connected faces. The remaining faces will be treated as side faces.

Refer Thin Sweep Limitations (p. 260) to view the limitations for **Thin Sweep** method in **MultiZone**.

– **Cart Sweep**: Applied to geometries that are axis aligned. **Cart Sweep** generates a Cartesian mesh, convert that Cartesian mesh to sweepable topologies and smooth it out.

For non-aligned geometries, you can define the sweep direction by selecting a face normal to the sweep direction. Use the **Select Source/Target Faces** tool to set the sweep direction.

Note:

Non-aligned geometries may require additional sizing adjustments when compared to those aligned to the global coordinate system.

You can set up a mapped mesh control on the source face to create all mapped blocks with the **Cart Sweep** method. The performance (time of mesh generation) depends on two points:

→ Time to generate the **Cartesian** mesh depends on the **Element Size** and **Cartesian Size**.

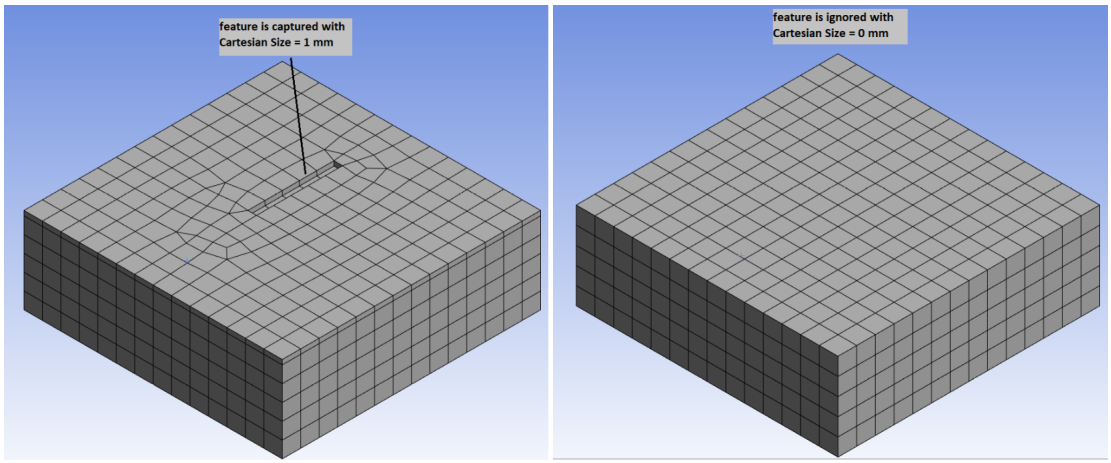
→ Time to convert from **Cartesian** to sweepable topology depends on the number of topological blocks or bodies created. Many stairsteps can slow things down. So, setting reasonable sizes and defeaturing non-axis-aligned features helps to improve performance.

When setting up the model, ensure that the sizes specified are appropriate for capturing the features. Unlike other mesh methods, reducing the **Element Size** may cause the mesh to fail. You should use **Element Size** to define the final mesh sizes.

When you select **Decomposition Type** as **Cart Sweep**, the **Cartesian Size** is available.

→ **Cartesian Size**: Defines the size above which features are captured in the mesh. Here, defeaturing is done based on **Cartesian Size**. You can set an explicit cartesian size to define the size that the Cartesian mesher uses to capture features. Use a smaller Cartesian mesh size to capture features without increasing final mesh size. If **Cartesian Size** is default, then mesh size will be used.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

– Removing fillets or chamfers helps to improve success of meshing.

– For selective meshing workflows, first mesh the bodies using the **Cart Sweep** method and then proceed to mesh the other bodies. The **Cart Sweep** method does not support pre-meshed adjacent bodies.

  Planar surface patches can be preserved with the **Cart Sweep** method by creating named selection. The Named Selections should include "protected" (case-insensitive) as a prefix or suffix to the name (for example, PROTECTED-SIDE, ProtectedFaces, protected-outlet).

Refer Cart Sweep Limitations (p. 260) to view the limitations for **Cart Sweep Method** in **MultiZone**.

- **Spacing Options**: Allows you to force split lines to better capture the model features. You can select the following options:

  – **Uniform**: Uses the specified element size for spacing.

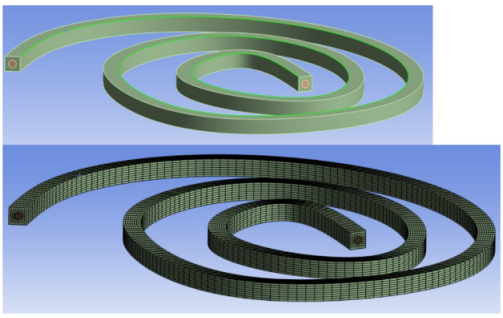
  – **Key-Points**: Check all vertices in the model and add split lines to follow the features, within the specified **Key-Points Tolerance**. It adjusts the mesh spacing between neighboring split lines so that the hex mesh follows the features. This aligns the cartesian mesh with feature lines and avoids clustering. When you select **Key-Points**, the **Key-Points Tolerance** is available and merges the nodes within the provided tolerance. The **Key-Points Tolerance** value should be less than or equal to the Cartesian Size. If you provide a **Key-Points Tolerance** value more than the Cartesian Size, the **Key-Points Tolerance** value gets reset to Cartesian Size.

  Note:

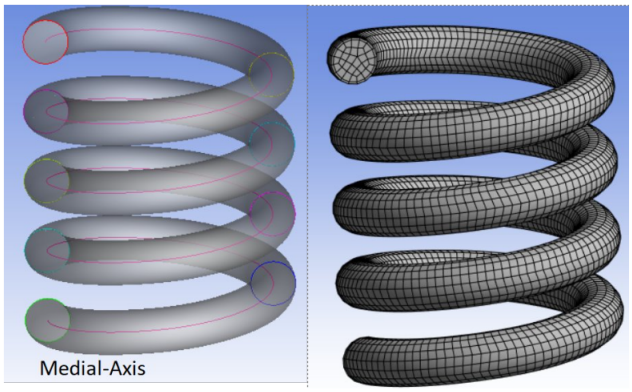
  Key-Points are the corner points of the features. Specifying Key-Points matches cartesian splits with the corner points so that the feature is respected.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

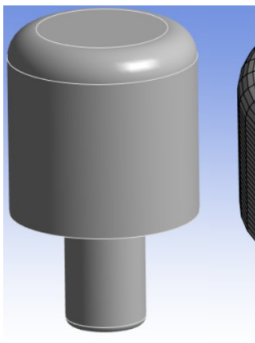
- **Standard**: Uses the standard **MultiZone Algorithm** (p. 420) for meshing which performs a surface blocking and attempts to break the volume into sweepable regions. If the software cannot find any sweepable regions,generates a free mesh.
- **Medial Axis**: Identifies the medial axis path, constructs surface blocking for the sectional profile, and extrudes along the medial axis path or revolves about the axis to create a 3D hexa blocking. You can determine the medial axis path for extrusion in any of the following ways:
  - Using contiguous surface curves that creates the geometry.
  - Computing the Medial-Axis curve from sectional circumcenters.
  - Using the axis of revolution for axis-symmetric bodies.



Medial Axis Path from Geometry Curves



Medial Axis Path from Circumcenters



Axisymm

Medial Axis determines the source and target automatically based on constraints. For models with multiple source faces, you can specify the source and target faces to avoid ambiguity. **Medial Axis** ignores the patch boundaries and you have to select **Preserve Boundaries** as **All** to respect protected topology along the extrusion curves.

When you apply **Medial Axis** decomposition type on a revolved body, the body is meshed with MultiZone axisymmetric meshing. Here, surface blocking for sectional profile is revolved around its axis to generate 3D hexa blocking. Here, **Sweep Element Size** controls the volume meshing and you can use **Defeature Tolerance** to remove the tiny edges and gaps in the geometry. Axisymmetric approach supports **Virtual Topology** and multi-body shared topology models. In multi-body shared topology models, all connected axisymmetric bodies must be meshed together. Axisymmetric rotation provides limited support for split revolved faces and expects at least one unsplit circular curve in the model to perform meshing. In such cases, you may use virtual topology to merge curves.

Medial Axis supports models with steps along the extrusion direction depending on the mesh size. For example, a change in pipe diameter. You can apply **Medial Axis** on long tubes or channels with and without guiding edges. You can also apply **Medial Axis** on fully or partially revolved bodies.

Medial Axis supports:

- Multi-Source faces to single target face extrusion
- Input mesh from adjacent meshed bodies.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - Inflation control
  - Facetted models

  Refer Medial Axis Limitations (p. 260) to view the limitations for **Medial Axis Method** in **MultiZone**.

- **Mapped/Swept Mesh Type**: Determines the shape of the elements used to fill structured regions. The default is **Hexa**. The available options are:
  - **Hexa**: Generates a mesh of all hexahedral elements for the part the method is scoped to.
  - **Hexa/Prism**: Generates a mesh of hexahedral and prism or wedge elements for the part the method is scoped to. The main difference between the **Hexa/Prism** option and **Hexa** is that for swept regions, the surface mesh can allow triangles for quality and transitioning. The triangles are later extruded to prisms or wedges.
  - **Prism**: Generates a mesh of all prism elements is generated for the part the method is scoped to. This option is sometimes useful if the source face mesh is being shared with a tet mesh, as pyramids are not required to transition to the tet mesh.
- **Surface Mesh Method**: Instructs **MultiZone** to use the **Program Controlled**, **Uniform**, **Pave** or **PrimeMesh Quad Dominant** method to create the surface mesh. The default method is **Program Controlled**.
  - **Program Controlled**: Automatically uses a combination of **Uniform** and **Pave** mesh methods depending on the mesh sizes set and face properties.

    Note:

    When you mesh a surface with more than thousand loops, **Program Controlled** automatically uses **Prime Quad Dominant Method** for meshing.

  - **Uniform**: Uses a recursive loop-splitting method which creates a highly uniform mesh. **Uniform** method is used when all edges have the same sizing and the faces being meshed do not have a high degree of curvature. This method provides good orthogonal mesh.
  - **Pave**: Uses a paving mesh method which creates a good quality mesh on faces with high curvature, when neighboring edges have a high aspect ratio. This approach is also more reliable to give an all-quad mesh.
  - **PrimeMesh Quad Dominant**: Uses the Prime quad-dominant surface meshing algorithm for multizone, which merges triangles into quads to create a surface mesh. **PrimeMesh Quad Dominant** method is suitable for models with thousands of loops. The method provides improved performance and fewer non-responsive state issues compared to other MultiZone surface

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

mesh methods. In some cases, it may deteriorate the mesh quality. You should select **Mapped/Swept Type** as **Hexa/Prism** when you apply **PrimeMesh Quad Dominant** method.

Note:

Note:**Automatic (PrimeMesh)** method and **PrimeMesh Quad Dominant** method in **MultiZone** are independent methods and may provide different results.

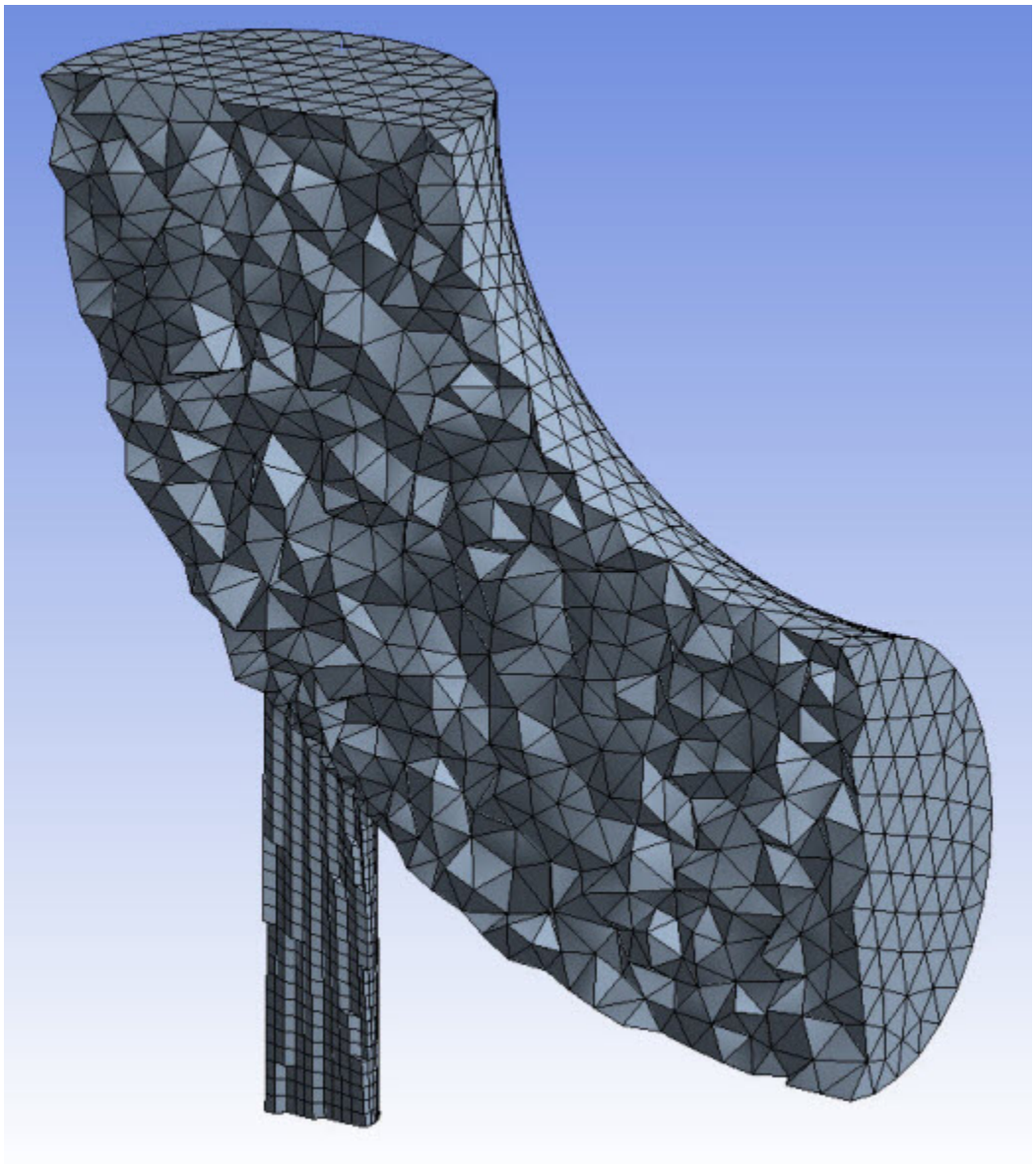
Note:

The **Surface Mesh Method** is applicable only to faces that are free meshed. If a face can be mapped meshed, it will be. See **Face Meshing Control** (p. 314) for more information.

- **Free Mesh Type**: Instructs **MultiZone** to allow a free mesh if it is not possible (without slicing) to generate a pure hex or hex/prism mesh. **Free Mesh Type** is available only when the **Decomposition Type** is **Standard**. The value of **Free Mesh Type** determines the shape of the elements used to fill unstructured regions. The default is **Not Allowed**. The available options are:
  - **Not Allowed**: Choose this option if you require a mapped mesh.
  - **Tetra**: Fills tetrahedral mesh on the regions of the model that cannot be meshed with a mapped mesh. Figure 97: Free Mesh Type = Tetra (p. 251) shows a **MultiZone** mesh that was generated when **Free Mesh Type** was set to **Tetra**. Notice the lower section is mapped meshed, and the upper section is free meshed because it cannot be map meshed. Refer to Patch Conforming Algorithm for Tetrahedrons Method Control (p. 208) for more information.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

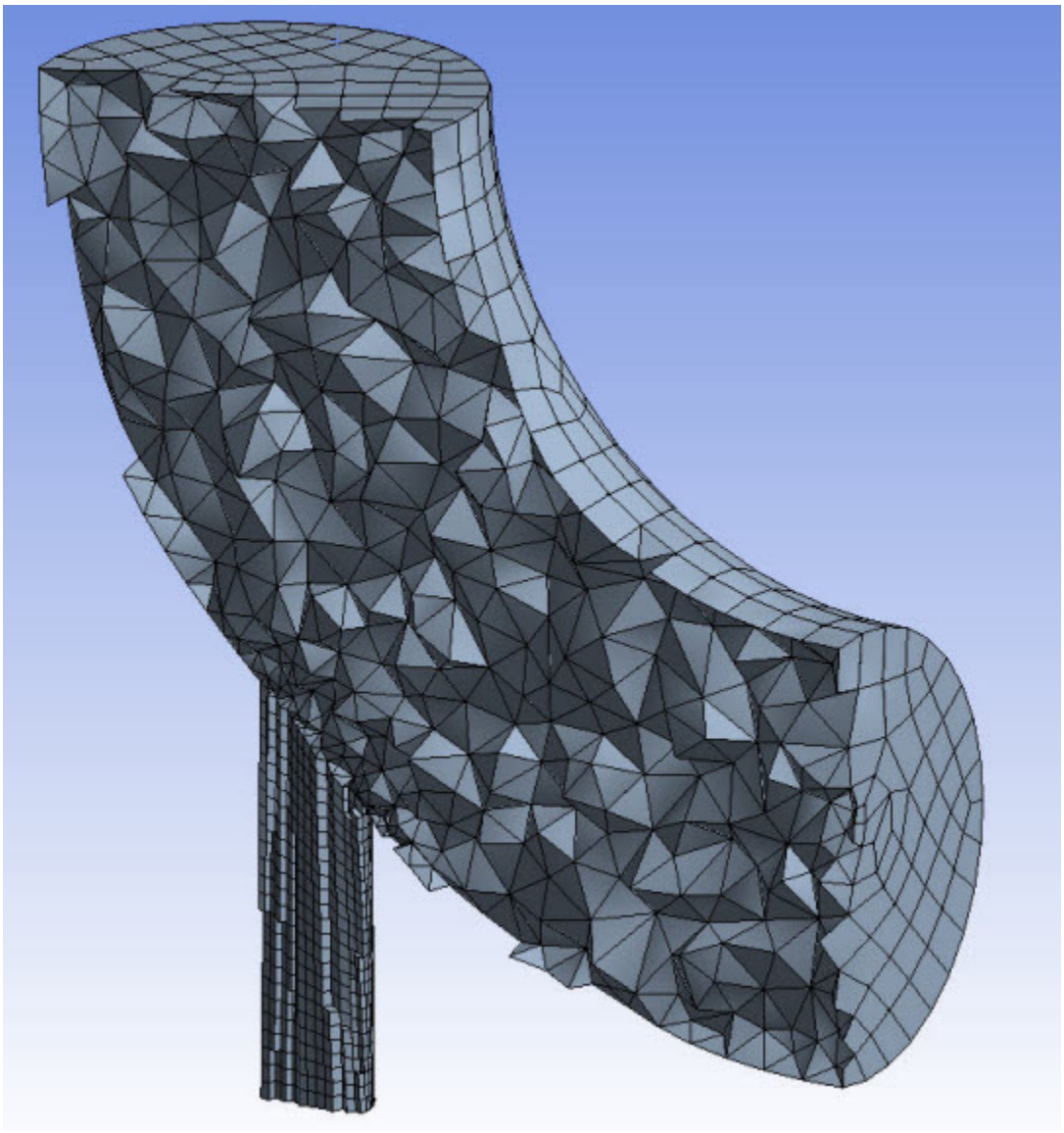
Figure 97: Free Mesh Type = Tetra



– **Tetra/Pyramid**: Fills tetrahedral mesh with pyramids at the faces on the regions of the model that cannot be meshed with a mapped mesh will be filled with a tetrahedral mesh with pyramids at the faces. Figure 98: Free Mesh Type = Tetra/Pyramid (p. 252) shows a **MultiZone** mesh that was generated when **Free Mesh Type** was set to **Tetra/Pyramid**. Notice the lower section is mapped meshed, and the upper section is free meshed because it cannot be map meshed. Refer to Patch Conforming Algorithm for Tetrahedrons Method Control (p. 208) for more information.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

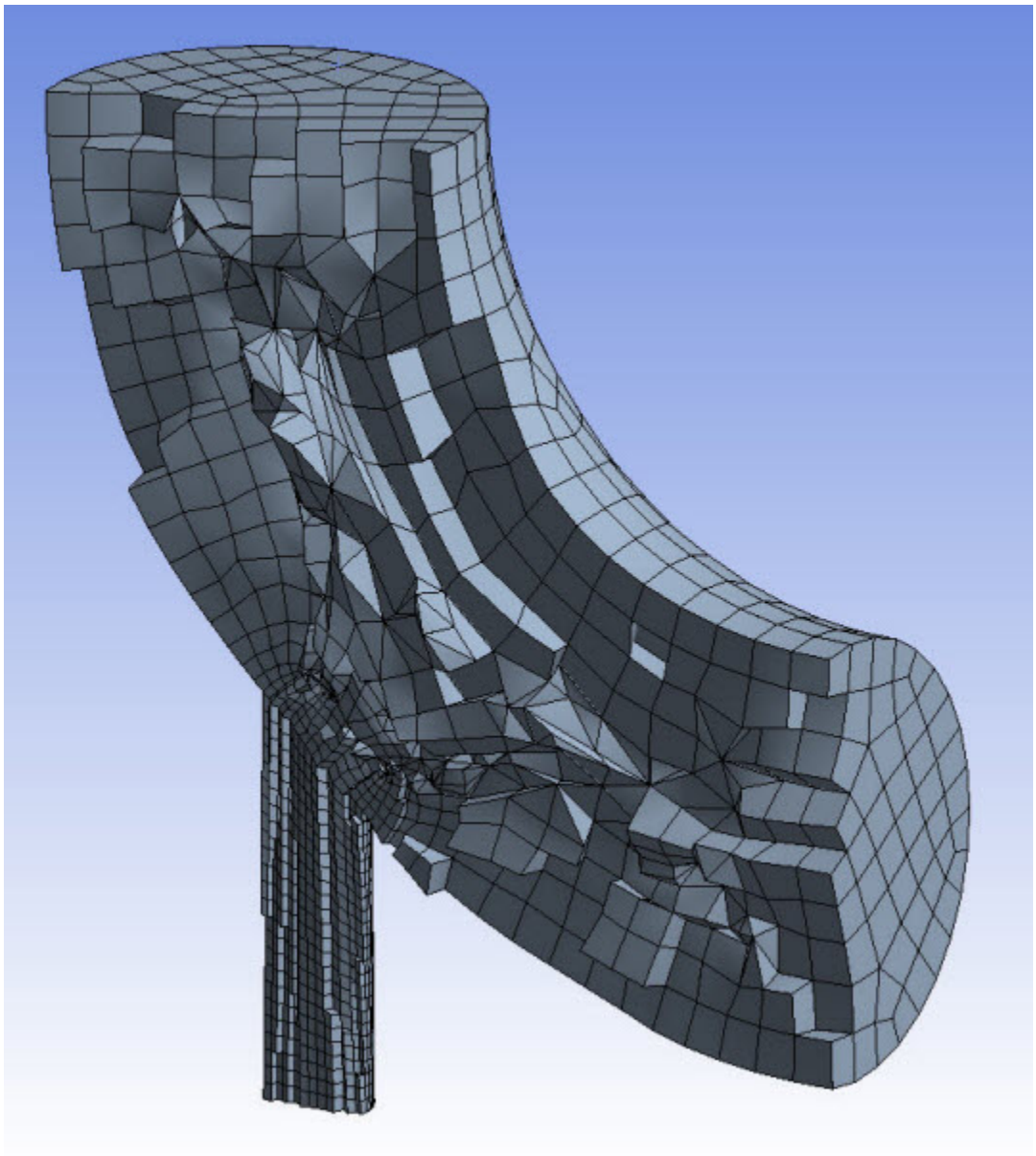
Figure 98: Free Mesh Type = Tetra/Pyramid



– **Hexa Dominant**: Fills hex dominant mesh on regions of the model that cannot be meshed with a mapped mesh. Figure 99: Free Mesh Type = Hexa Dominant (p. 253) shows a **MultiZone** mesh that was generated when **Free Mesh Type** was set to **Hexa Dominant**. Notice the upper section is mapped meshed, and the lower section is free meshed because it cannot be mapped meshed. Refer to Hex Dominant Method Control (p. 233) for more information.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

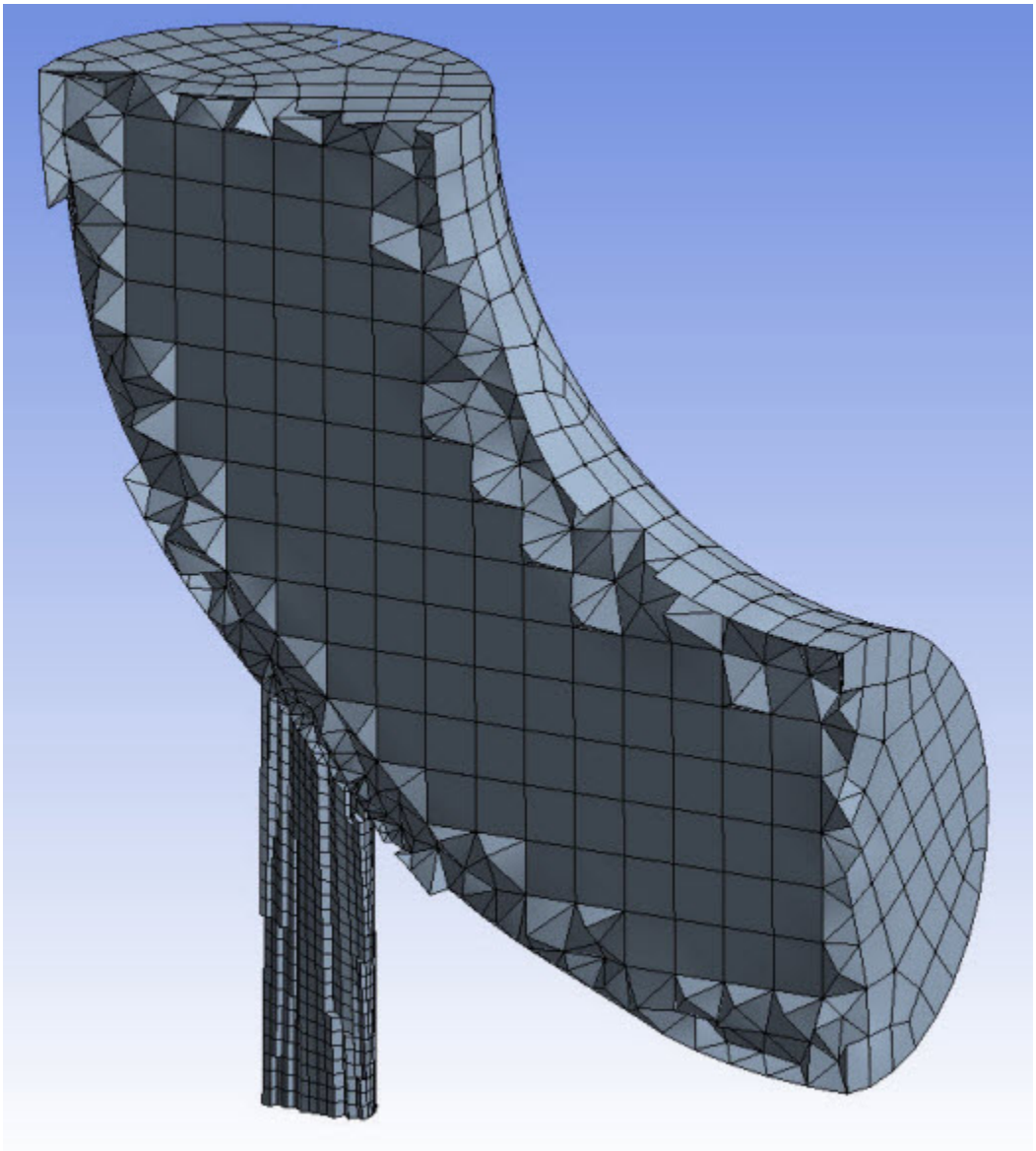
Figure 99: Free Mesh Type = Hexa Dominant



– **Hexa Core**: Fills hexa core mesh on the regions of the model that cannot be meshed with a mapped mesh. Hexa Core meshes can be generated where most of the volume is filled with a Cartesian array of hexahedral elements replacing the tetras. This is connected to the remainder of a prism/tetra hybrid by automatic creation of pyramids. Hexa Core allows for reduction in number of elements for quicker solver run time and better convergence. Figure 100: Free Mesh Type = Hexa Core (p. 254) shows a **MultiZone** mesh that is generated when **Free Mesh Type** was set to **Hexa Core**. Notice the upper section is mapped meshed, and the lower section is free meshed because it cannot be mapped meshed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 100: Free Mesh Type = Hexa Core



- **Element Order**: Refer to Method Controls and Element Order Settings (p. 202). The default is **Use Global Setting**.
- **Src/Trg Selection**: Defines the source and target selection type. The default is **Automatic**. The available options are:
  - **Automatic**: Determines the source and target automatically. The **Automatic** option works fine for simple sweep configurations, but if there are multiple levels of sweeps, it is often best to manually define the source faces.
  - **Manual Source**: Allows you to select the faces to be used as sources (and targets) using the specified**Source(s) and Target(s)**. **MultiZone** treats all sources or targets as sources, as imprinting can occur from either side. For more details, refer to the description of **Src/Trg Scoping Method**, **Source(s) and Target(s)**, and **Src/Trg Named Selection** below.

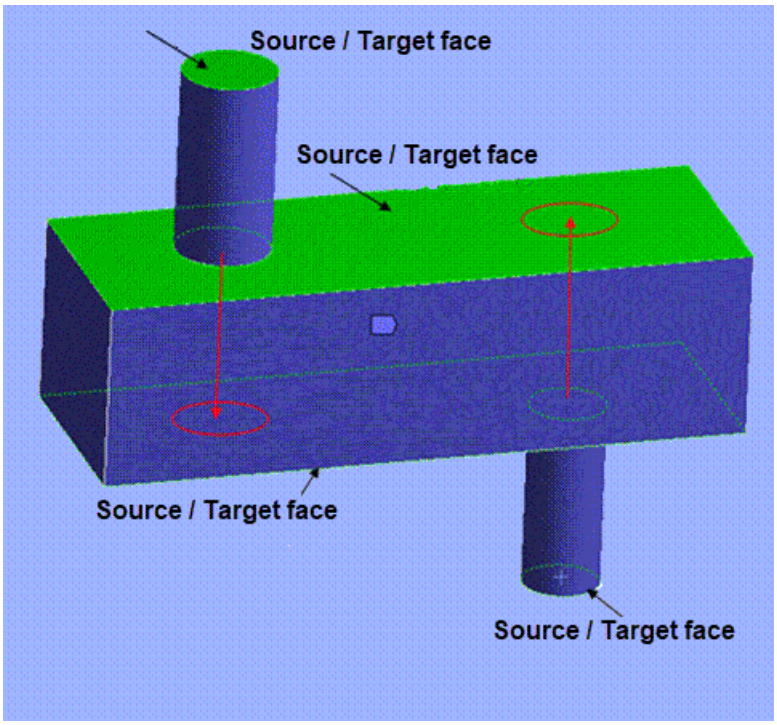
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Src/Trg Scoping Method**: Defines the method for choosing a source face. **Geometry Selection** enables you to select sources or targets manually using the **Source(s) and Target(s)** option. **Named Selection** enables you to choose one Named Selection as a source or target using the **Src/Trg Named Selection** option.
- **Source(s) and Target(s)**: Select the faces that need to be imprinted for proper geometry decomposition. **Source(s) and Target(s)** is available when you select **Geometry Selection** as your **Src/Trg Scoping Method**. The faces you select can be either "sources" or "targets," but all of them are treated as sources by **MultiZone**, as shown in Figure 101: Source / Target Face Selection for MultiZone (p. 255).

Note:

To make source face selection easier, select **Annotation Preferences** from the Toolbar and then deselect **Body Scoping Annotations** in the **Annotation Preferences** option box to toggle the visibility of annotations in the **Geometry** window. For example, after scoping **MultiZone** to a body, the body is displayed using a blue solid annotation. Turn off the body scoping annotations, then select the source faces. For picking internal faces, the **Hide Faces** right-click option may help you to see inside a body. For example, you can select external faces in the **Geometry** window and then use the **Hide Faces** option to hide the selected faces (making it easier to select the internal faces).

Figure 101: Source / Target Face Selection for MultiZone



- **Src/Trg Named Selection**: Choose an existing named selection to select the faces that need to be imprinted for proper geometry decomposition. **Src/Trg Named Selection** is only available when you select **Named Selection** as your **Src/Trg Scoping Method**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Sweep Size Behavior**: Enables you to set a **Sweep Element Size** to define the mesh spacing (default), or to select **Sweep Edges** to remove edges and prevent them from constraining the source faces.

  – **Sweep Element Size**: Enables you to set an element size to define the mesh spacing along the sweep path from source to target faces. If this control is set to a non-zero value, sizing controls applied to the selected bodies as curvature and proximity refinement, local sizing or both are ignored.

    The **Sweep Element Size** setting is ignored if hard size controls are applied to side edges or faces. If multiple bodies with the same sweep direction have different sizes set for **Sweep Element Size**, the smallest size is used and the others are ignored.

    Selecting the check box adds **Sweep Element Size** to the Workbench parameters, enabling you to use element size settings as a variable design point when creating multiple solutions.

  – **Sweep Edges**: This option should be used with an edge sizing control. The edge sizing control defines the distribution along the sweep path, and can also affect the source face. Use this option to remove the influence of the edge sizing from the source face mesh. That is, the edges selected will only influence the sweep path and not the source faces.

  – **Sweep Number of Divisions**: Allows you to provide the number of elements across the thickness. Available only when the **Decomposition Type** is **Thin Sweep**. **Sweep Number of Divisions** is not respected for cases where walls (side faces) are shared with source or target surfaces of connected bodies. You can parametrize **Sweep Number of Divisions**.

- **Element Option**: Allows you to select the element type for the solver to use. The available options are **Solid** and **Solid Shell**. The default value is **Solid**. **Element Option** is available only when the **Decomposition Type** is **Thin Sweep**.

  Note:

  In **Sweep Method** control, **Automatic Thin** and **Manual Thin** provide better orthogonal quality mesh than **MultiZone Thin Sweep** for **Solid Shell** element option for models with slanted side faces.

  Note:

  To choose the orientation of the Solid Shell elements, you can select the source face manually.

- **Preserve Boundaries**: Preserves only the protected topologies (See Protecting Topology Defined Prior to Meshing (p. 184)) or all features in the model. The default value is **Protected**. The available options are:

  – **Protected**: Preserves only the protected topologies at the boundary.

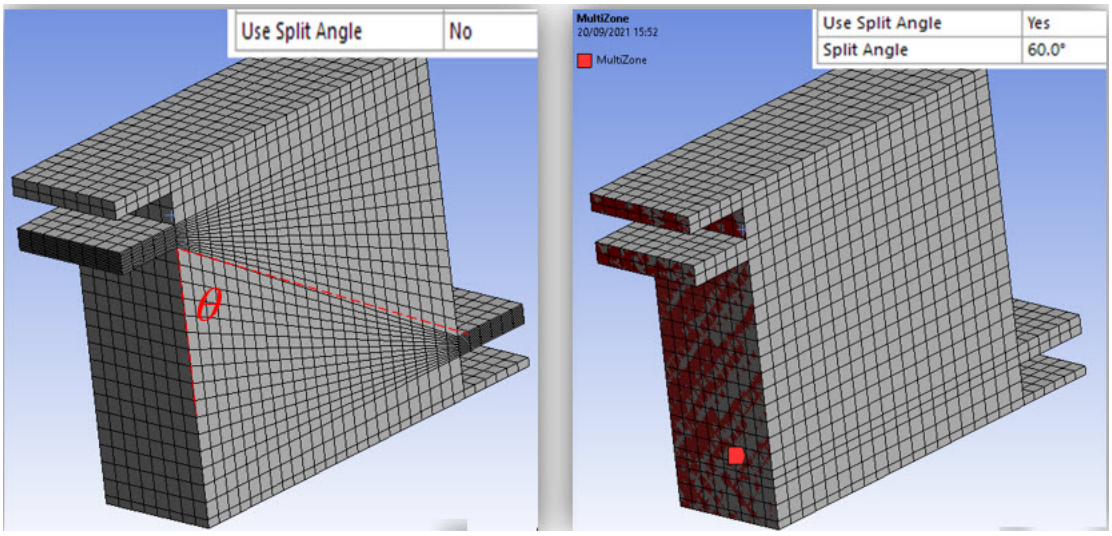
  – **All**: Preserves all topologies at the boundary.

- **Mesh Based Defeaturing**: Filters edges in or out based on size. The default value is **Off**. If **Mesh Based Defeaturing** set to **On**, a **Defeature Size** field appears where you may enter a numerical

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

value greater than 0.0. By default, the value of this local **Defeature Size** field is the same as the global Defeature Size (p. 111). If you specify a different value here, it overrides the global value. Specifying a value of 0.0 here, resets the tolerance to its default. Available only when the **Decomposition Type** is **Standard** and **Thin Sweep**.

- **Minimum Edge Length**: Read-only indication of the smallest edge length in the model.
- **Write ICEM CFD Files**: Sets options for writing Ansys ICEM CFD files. Refer to Writing Ansys ICEM CFD Files (p. 84) for details. The default is **No**.
- **Use Split Angle**: Decomposed regions are sent in blocks to the meshing algorithm. This option splits the edges of the decomposed regions to improve the mapped mesh quality and prevent skewed mesh on side faces. The default option is **No**. For **Physics Preference** set to **Explicit**, the default option is **Yes**. If the mesh is skewed, you can set the **Use Split Angle** to **Yes** to create new region splits which will create more orthogonal mesh. When **Use Split Angle** is set to **Yes**, the **Split Angle** is enabled and allows you specify the value for the split angle. The default value is **60** degrees. The splits are created at nodes where the angle of an end node of an edge (edge vector) to the corresponding end node of a parallel edge is less than the specified split angle. **Use Split Angle** is available only when the **Decomposition Type** is **Standard**.



- **Triangle Reduction**: Reduces the number of triangular elements while performing **PrimeMesh Quad Dominant** surface meshing. The default option is **Conservative**. **Triangle Reduction** is available only when **Surface Mesh Method** is **PrimeMesh Quad Dominant**. The available options are:
  - **None**: Generates quadrilateral mesh without reducing the triangle count on the whole body.
  - **Conservative**: Moves triangles near to the boundary nodes or edges away from the boundary and then reduces the triangle count on the whole body.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Aggressive**: Moves the interior triangles and the boundary triangles to form pairs of connecting triangles and then reduces the triangle count on the whole body. Hence, **Aggressive** removes more triangles than **Conservative**.

Note:

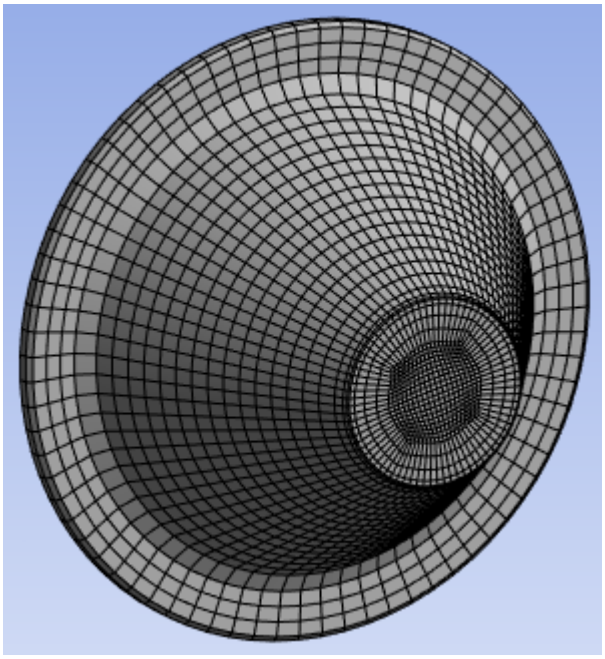
For detailed information about **MultiZone**, refer to MultiZone Meshing (p. 419). For general information on applying **MultiZone** in combination with other mesh method controls, refer to Meshing: Mesh Control Interaction Tables (p. 475).

Notes on Scoping for the MultiZone Mesh Method

- You can use the **MultiZone** mesh method in combination with other solid mesh methods in a multibody part, and the bodies are meshed with conformal mesh.
- If a multibody part contains some bodies that are scoped to be meshed with **MultiZone** and other bodies that are not scoped with any mesh method, then the other bodies are meshed with the default mesh method.

  Refer to Conformal and Non-Conformal Meshing (p. 21) for information about conformal meshing.
- For Torus geometry with sliver faces, set **Mesh Based Defeaturing** as **ON** to avoid mesh failure.
- **MultiZone** supports handling bodies where all three Decomposition Types are applied on different bodies. When you mesh multibodies with shared topology simultaneously, all bodies with **Cart Sweep** applied are meshed first, followed by bodies applied with **Thin Sweep** and finally the bodies applied with **Standard MultiZone**.
- **MultiZone** tries to mesh axisymmetric cases using the **Program Controlled Decomposition Type**. In case of failure, set the **Defeature Size** other than the default value. This forces the application to create a revolved profile blocking. This feature requires that all faces in the body to be 360 degree fully revolved but edges could be split. It does not support cases with cutouts in the revolved surface.

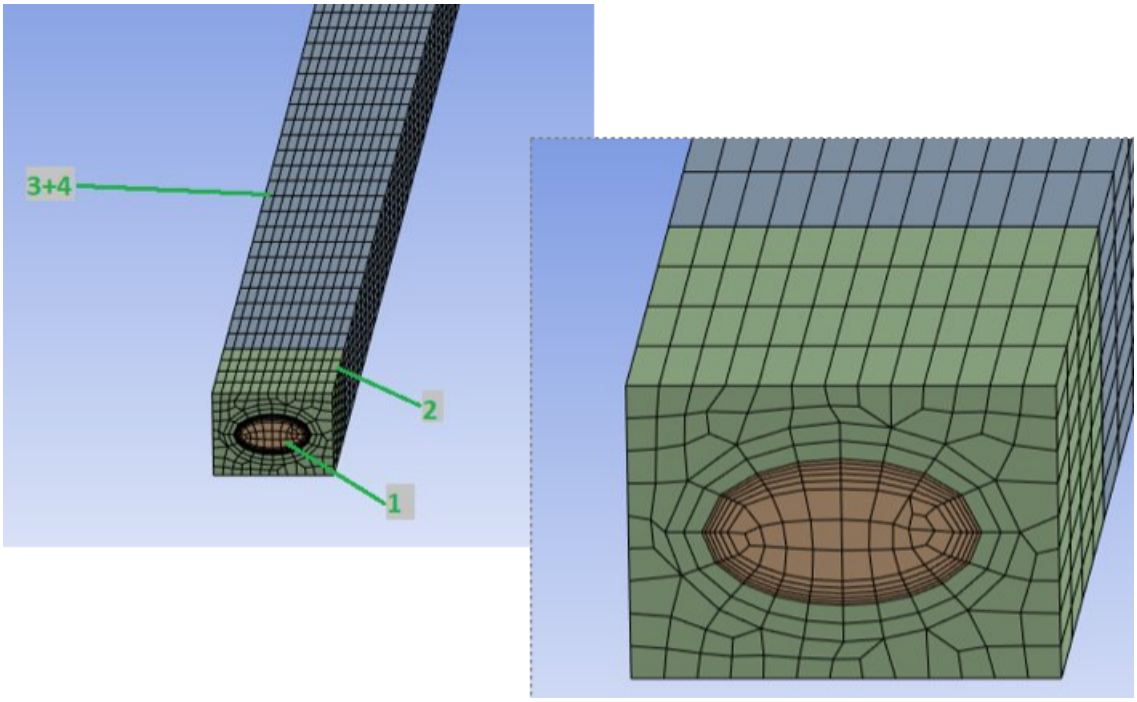
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- For long coil cases failure, you may perform the following:

  If the geometry is unclean, save the geometry in ACIS format and import the geometry to Discovery Modeling and save the database. Slice one end of the coil into a small piece. Now, apply Shared topology and clean up the geometry using **Repair** and **Prepare** tools in Discovery Modeling and save the database. Open the saved database in the **Mesh** component system in Ansys Workbench and apply mesh setting and perform meshing.

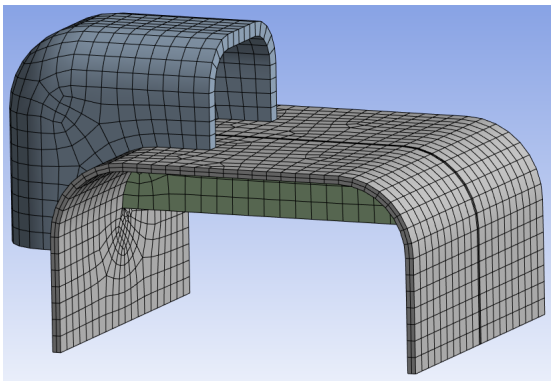
  Selective meshing may help sometimes. For example, mesh the inner body first, followed by the outer body and then two long bodies along the length of the coil simultaneously.



Thin Sweep Limitations

The limitations of **Thin Sweep** are:

- **Thin Sweep** does not fully support multibody part with shared topology meshed either with all bodies once or selective meshing.
- **Thin Sweep** does not support intersecting loops in double side imprinting.
- When you try to mesh all bodies in a model having a stack of the thin bodies, **Thin Sweep** may not recognize the bodies as thin bodies.
- The Sweep Number of Divisions is not respected for cases where walls (side faces) are shared with source or target surfaces of connected bodies. In the below example, the Sweep Number Of Divisions = 2 is not respected for top body.



Cart Sweep Limitations

The limitations of **Cart Sweep** are:

- **Cart Sweep** supports **Mapped Mesh Type** only with source face selection.
- **Cart Sweep** supports only **Preserve Boundaries** that are **Protected**.
- **Cart Sweep** may not work well with shared topology.

Medial Axis Limitations

The limitations of **Medial Axis** are:

- Medial Axis path should have a single start and a single end. That is, the path should not have any branch.
- **Medial Axis** may not support all protected topology.
- **Medial Axis** may exhibit slow performances when connecting to adjacent meshed bodies with many protected topologies. In such cases, try to maintain the medial axis body as single source to single target to improve the performance.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cartesian Method Control

Cartesian method creates unstructured hexa mesh of mostly uniform size, aligned to the specified coordinate system, and fits it to the geometry. The element size should be smaller than the thickness of the model to prevent the mesher from defeaturing (not capturing) that portion of the model. Alternatively, the defeaturing could be helpful to eliminate dirty geometry smaller than the element size. Cartesian Method is a body fitted meshing.

This method is useful when the geometry features align well with a coordinate system and a regular mesh is desired. Models for explicit dynamics, organic models (models without many feature edges), process industry and electronic components are good examples that could benefit from this mesh method. This method is also recommended for simulating the printing process in Additive Manufacturing.

Note:

The **Cartesian** method must be applied to an entire part. If a body in a multibody part is selected, all bodies in the part are added to the Geometry selection.

To access the **Cartesian Method** control,

On the Tree Outline, right-click **Mesh** and click **Insert > Method**.

Or

On the Tree Outline, click **Mesh** and click **Method** in the **Mesh Context** tab on the Ribbon. Select **Method** as **Cartesian** in the **Automatic Method** Details view.

The **Cartesian Method** Details view has the following options:

Scope

- **Scoping Method**: Allows you to scope geometry bodies or named selection. The default value is **Geometry Selection**.
  - **Geometry Selection**: Allow you to scope the geometry bodies. When you select **Geometry Selection**, the **Geometry** allows you to select the geometry from the **Geometry** window.
  - **Named Selection**: Allow you to scope bodies grouped under a named selection.

Definition

- **Suppressed**: Allows you to suppress the selected control. The default value is **No**. When **Suppressed** is set to **Yes**, **Active** displays the status of the selected control. **Active** is read-only.
- **Method**: Allows you to select method.

When you select the **Cartesian** Method option, the Details view expands to include additional settings, many of which are unique to this option. For basic usage that involves obtaining a Cartesian mesh, the procedure is to apply a **Method Control** to one or more bodies, set **Method** to **Cartesian**, and accept the default values of the various settings.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For advanced or specialized usage, adjust the settings as needed. The following is a description of each of these settings.

- **Element Order**: Allows you to select the element order. The default value is **Use Global Setting**. For information about the **Element Order** option, refer Method Controls and Element Order Settings (p. 202).

- **Type** - Allows you to specify how the edge size is determined. Edge size is considered a soft size, allowing for small variations depending on geometry and other settings. The available options are **Element Size** and **Number of Divisions**. The default value is **Element Size**.

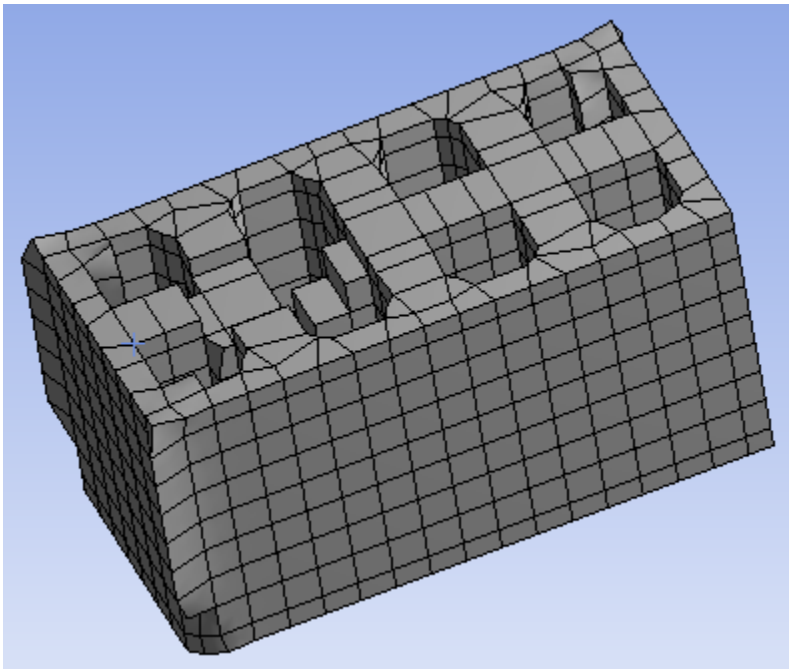
  – **Element Size**: Allows you to set the element size. You can use the Default setting, which is based on global element size, or manually set the element size. Select the check box to the left of the label to parameterize the **Element Size**.

  – **Number of Divisions**: Allows you to set manually number of divisions in Z-direction using **Number of Divisions in Z-Dir** or you can use the **Default** setting, which is obtained from the model's bounding box size in the z-direction divided by the global element size. The nominal size found from the computation of the number of divisions is also used for the spacing in the x and y dimensions.

    → **Number of Division in Z-Dir**: Allows you to set the number of divisions in the Z direction. You can parametrize **Number of Divisions in Z- Dir**.

- **Spacing Option**: Allows you to force split lines to better capture model features. The default value is **Uniform**. The available options are:

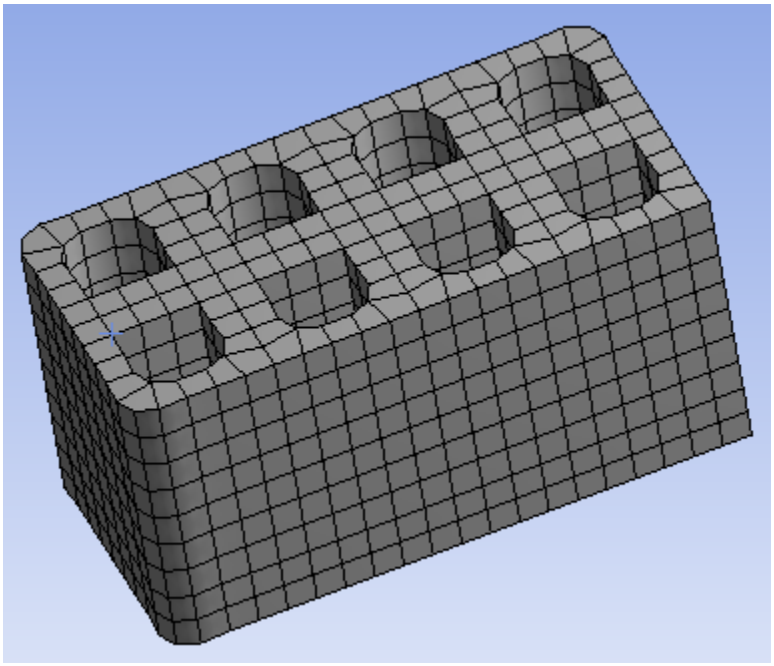
  – **Uniform**: Uses the specified element size for X, Y and Z spacing. This may result in distorted or missing elements.



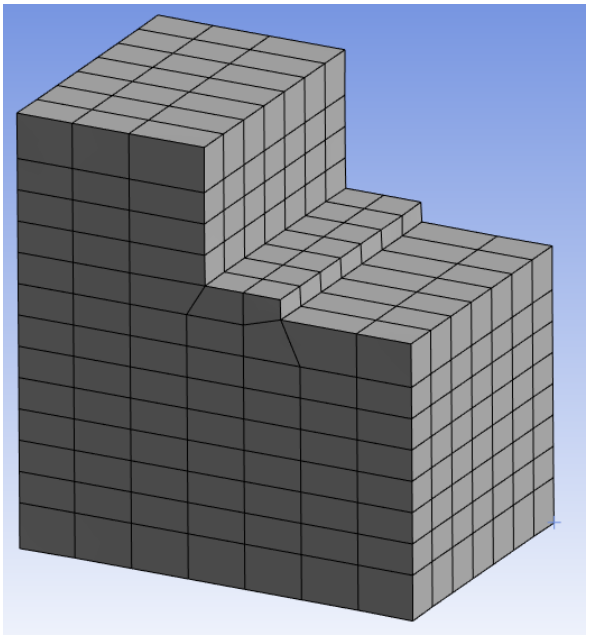
  – **Key-Points (Auto)**: Check all vertices in the model and add split lines to follow the features, within the specified **Tolerance**. Then, **Key-Points (Auto)** adjust the mesh spacing between neighboring split lines so that the hex mesh follows the features.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ **Key-Points Tolerance**: Allows you to set the tolerance to defeature small features and avoid over-refined meshes when corner features are not aligned.

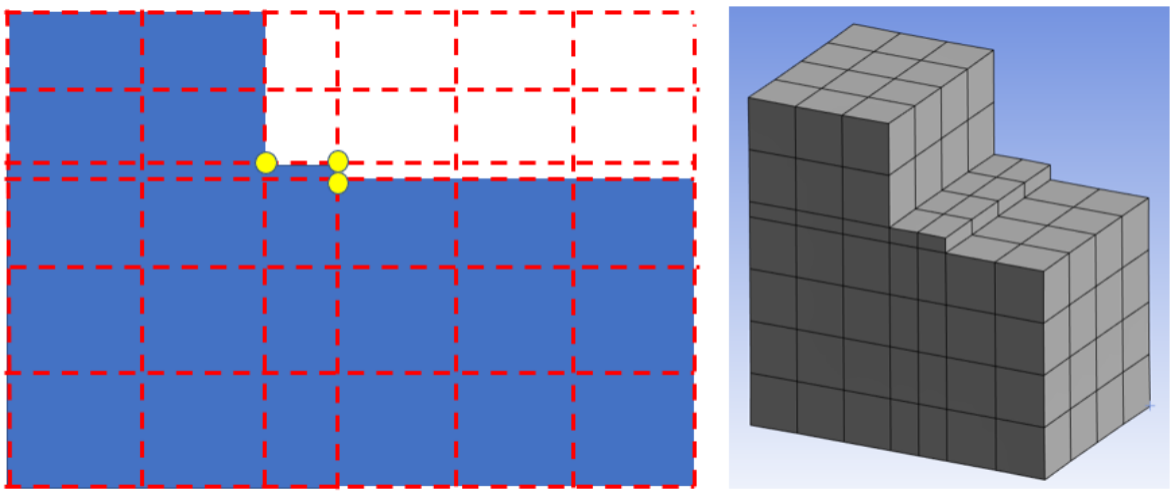


Example without Key-Points

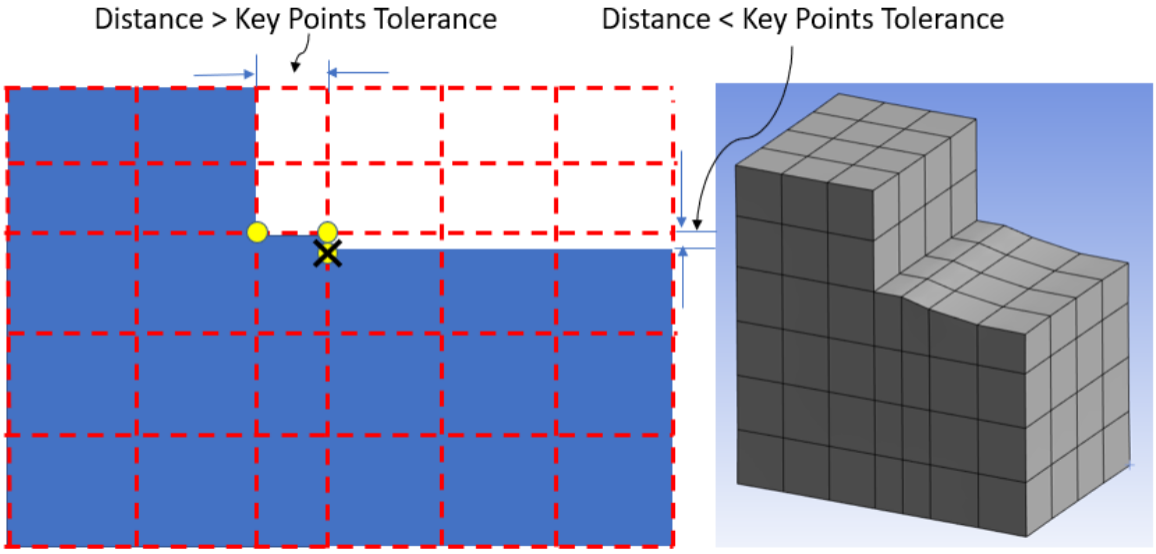


Example with Key-Points Tolerance equals zero

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Example with Key-Points Tolerance $\geq$ geometry step size



– **Key-Points (Manual)** - Allows you to select vertices manually to create split lines in each of the X, Y and Z directions. Mesh spacing is adjusted between neighboring splits. Additional split lines may be added by the mesher as required to follow the features. The available options for the Key-Points (Manual) in X, Y and Z direction are:

→ **Vertices in X**: Allows you to set vertices manually to create split lines in X-direction.

→ **Vertices in Y**: Allows you to set vertices manually to create split lines in Y-direction.

→ **Vertices in Z**: Allows you to set vertices manually to create split lines in Z-direction.

Note:

Key-Points are the corner points of the features. Specifying key-points matches cartesian splits with the corner points so that the feature is respected.

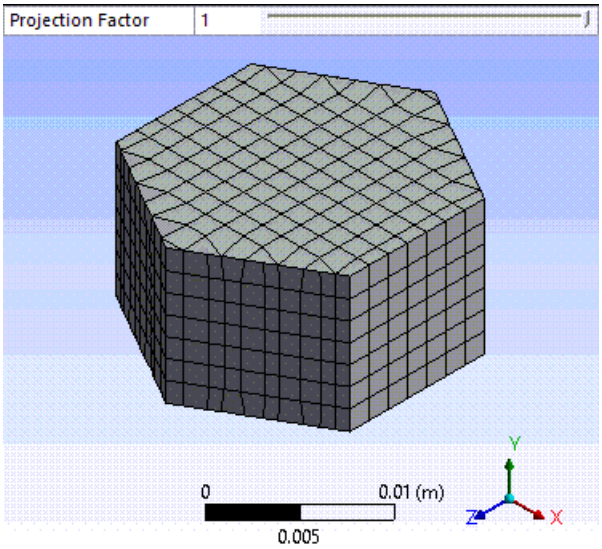
Advanced

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Projection Factor**: Allows you to set the balance between mesh quality and capturing the geometry.

  Set a value between **0** and **1**. If the **Projection Factor** is set to 0, the Cartesian mesh will have high quality hexa elements that approximate the geometry surface through stairstepping. Increasing the value of **Projection Factor** will force the mesh to more closely follow the geometry surface, but with reduced element quality. If the **Projection Factor** is set to 1, most of the boundary nodes will project to the geometry and perturb only those causing bad element quality. However, some hexa elements may have pairs of coplanar faces to fit a planar geometry surface.

  *The following demo is presented as an animated GIF. View online if you are reading the PDF version of the help. Interface names and other components shown in the demo may differ from those in the released product.*The image shows the effect of decreasing the **Projection Factor** for a simple geometry with two sides aligned with the global coordinate system.



  The default value of **Projection Factor** is 0 for Additive Manufacturing Process simulations. Specifically, if the AM Process object is present in the Project tree when a Cartesian method is added, the **Projection Factor** is automatically set to 0. When you want to generate supports from the meshed body, the **Projection Factor** *must* be set to 0.

- **Project in constant Z-Plane**: Allows you to use for print simulation in Additive Manufacturing.

  If enabled, the x and y coordinates of the Cartesian mesh are modified while maintaining a constant height in the Z-direction. The final mesh is much tighter to the geometry. This option is enabled by default if the AM Process object is present in the Project tree.

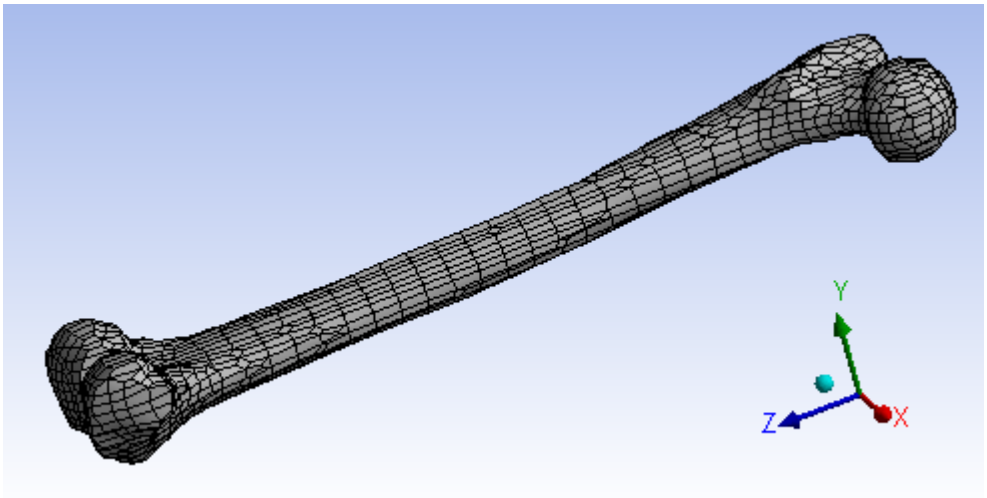
- **Stretch Factor in X**: Allows you to modify the Aspect Ratio of the hex mesh in the X-dimension.

- **Stretch Factor in Y**: Allows you to modify the Aspect Ratio of the hex mesh in the Y-dimension.

- **Stretch Factor in Z**: Allows you to modify the Aspect Ratio of the hex mesh in the Z-dimension.

  Stretch Factor is useful to reduce the element count for a geometry that may be elongated or shortened in one dimension. In this image, **Stretch Factor in Z** has been set to 3.0 while X and Y remain at 1.0. Mesh elements sizes can be up to about 3x larger in the z-dimension than x-dimension and y-dimension, within the constraint of the **Projection Factor**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

The process for applying the **Stretch Factor** depends on how edge sizing is determined. However the result on aspect ratio is the same.

– When **Type** = **Element Size**, the stretch factor applies a multiplier to the element size, aligned to a selected dimension. For example, when **Element Size** = 0.1 and stretch factors of 0.5, 1, and 2 are applied, then the dimensional mesh element size will be approximately 0.05, 0.1 and 0.2.

– When **Type** = **Number of Divisions**, the stretch factor scales the other dimensions while preserving the specified number of divisions in the z-dimension. For example, when stretch factors are 0.5, 1, and 2, then the nominal element sizes are scaled by approximately 1/4, 1/2 and 1 to preserve the number of divisions in the z-dimension.

Note:

Body Fitted Cartesian Method supports soft edge sizing.

- **Coordinate System**: Allows you to choose the coordinate system to which the mesh is aligned. The default value is **Global Coordinate System**.
- **Write ICEM CFD Files**: Allows you to set options for writing Ansys ICEM CFD files. Refer to Writing Ansys ICEM CFD Files (p. 84) for details.
- **Smooth Mesh Spacing**: Allows you to perform edge length based smoothing when **Smooth Mesh Spacing** is set to **Yes**. The default value is **Yes** when the **Physics Preference** is **Explicit**. For other **Physics Preference**, the default value is **No**.

Layered Tetrahedrons Method Control

Layered Tetrahedrons mesh method creates unstructured tetrahedral mesh in layers based on a specified layer height and fits it to the geometry.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

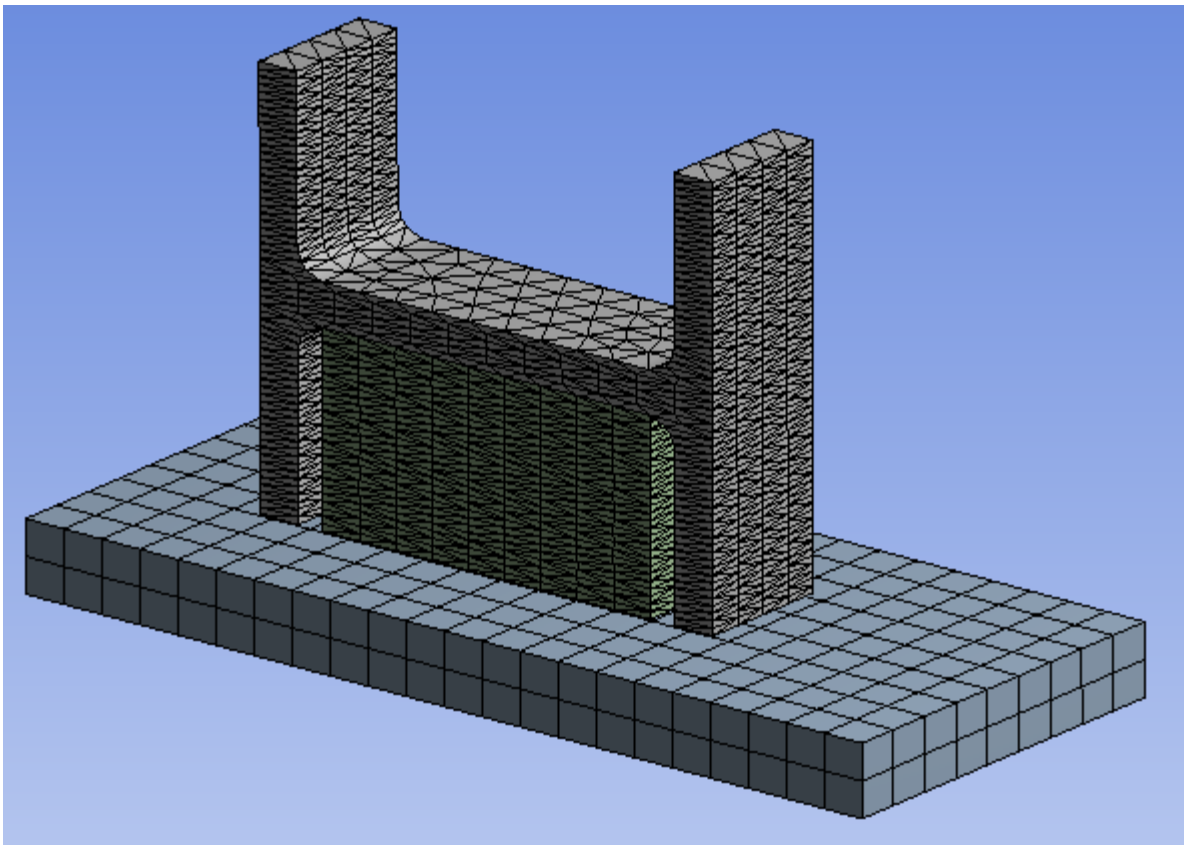
This method can be used for simulating the printing process in Additive Manufacturing as the build parts must conform to a mesh with fixed step sizes in the global Z direction.

The meshing process involves the following approach:

- Generates an initial surface mesh based on the defined settings.
- Generates mesh layers based on the specified layer height. The mesher generates the mesh layers starting from the Z-location of the plane.
- Projects nodes on the geometry within the specified tolerance (**Relative Tolerance**) of the layer plane to the plane during the initial layer operation.
- **Layered Tetrahedron** method preserves feature nodes and corner nodes determined based on the **Feature Angle** and the **Corner Angle**, respectively.
- Identifies problematic sliver faces based on **Sliver Triangle Height** and collapses or fixes sliver faces to improve quality.
- Identifies and inflates the geometry faces overlapping the layer planes based on the Overlapping angle. Additionally, geometry faces close to the layer planes which leads to bad quality tets are identified based on the **Layer Height** and proximity to the plane. These faces inflate away from the layer planes to create space for better quality tets based on the **Inflate Relative Tolerance** value specified.
- Fills the improved surface mesh with tetrahedral mesh conforming to the mesh layer. Thus, improves the tetrahedral mesh quality.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 102: Layered Tetrahedrons Mesh



Note:

- **Layered Tetrahedrons** Mesh Method does not support bodies with shared topology. Conformal mesh is not created even when the bodies have shared interface(s).
- This method cannot be used with mesh controls such as **Inflation**, **Refinement**, **Match Control**, **Pinch**, **Face Meshing** and **Edge Sizing** controls.
- The mesh is not associated back to geometry. See Association Using Named Selections (p. 271) for more details.

To access the **Layered Tetrahedrons Method** control,

On the Tree Outline, right-click **Mesh** and click **Insert > Method**.

Or

On the Tree Outline, click **Mesh** and click **Method** in the **Mesh Context** tab on the Ribbon.

Select **Method** as **Layered Tetrahedrons Method** in the **Automatic Method** Details view.

When you choose the **Layered Tetrahedrons** Method option, the Details view expands to include additional settings, many of which are unique to this option. For basic usage, the procedure is to

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

apply a Method Control to the body and set **Method** to **Layered Tetrahedrons**. Set the **Layer Height** and accept the default values of the various settings.

Meshing Recommendations

- The **Layered Tetrahedrons** method is available only when **Use Adaptive Sizing** is set to **No**.
- Minimum and maximum sizes should be decided based on the model.
  - The minimum size should be decided based on the features to be resolved and the **Layer Height**. You should set a value smaller than the **Layer Height**.
  - The **Element Size** should not be greater than 6 times the specified minimum size. The mesh quality reduces as the **Element Size** to **Min Size** ratio increases.
  - The **Max Size** does not influence the layered tet mesher.
- Set the **Growth Rate**, the recommended range of values is 1.2 to 2.
- The recommended range for the **Curvature Normal Angle** is up to 36 degrees.
- The **Sliver Triangle Height** is based on the minimum size (p. 113) specified. The default value is 10% of the minimum size. You should use a **Sliver Triangle Height** not more than 50% of min size.
- Increasing **Relative Tolerance** and **Inflate Relative Tolerance** can avoid generating thin pockets of mesh. The recommended range for **Relative Tolerance** is between 0.01 to 0.02. The recommended range for **Inflate Relative Tolerance** is between 0.1 to 0.3.

The **Layered Tetrahedrons** mesh method includes the following options:

Scope

- **Scoping Method**: Allows you to scope geometry bodies or named selection. The default value is **Geometry Selection**.
  - **Geometry Selection**: Allow you to scope the geometry bodies. When you select **Geometry Selection**, the **Geometry** allows you to select the geometry from the **Geometry** window.
  - **Named Selection**: Allow you to scope bodies grouped under a named selection.

Definition

- **Suppressed**: Allows you to suppress the selected control. The default value is **No**. When **Suppressed** is set to **Yes**, **Active** displays the status of the selected control. **Active** is read-only.
- **Method**: Allows you to select the method.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Element Order**: Allows you to select the element order. The default value is **Use Global Setting**. For information about the **Element Order** option, refer Method Controls and Element Order Settings (p. 202).

  Note:

  Only straight sided mid-nodes are available for quadratic element type.

- **Layer Height**: Allows you to set the height of steps in elements along the global Z direction.
- **Control Messages**: Provides a message when **Use Adaptive Sizing** is set to **Yes** and when layered tetrahedron mesh fails. You can click **Control Messages** to view the message.

Advanced

- **Generate Layers Using Facets**: Generates layered tetrahedron mesh for the given model using facets. When **Generate Layers Using Facets** is **Yes** this allows you to skip the surface mesh generation before slicing. Thus, slicing operation is performed directly on CAD. The default value is **No**.
- **Repair Facets**: Enables you to repair CAD facets. When set to **Yes**, **Repair Facets** allows you to fix Sliver facets and facet intersections aggressively. The default value is **No**.
- **Layer Start**: Generates the mesh layers from the Z-location of the plane. The default value is the min Z-coordinate of the bounding box enclosing the scoped bodies. For **Additive Manufacturing** simulations, more specifically, when the **AM Process** object is available in the **Project tree**, it automatically sets the value to the **Z-coordinate** of the top of the baseplate by default.
- **Relative Tolerance**: During the initial layer operation, projects nodes within the specified tolerance (relative tolerance) to the layer plane to the plane on the geometry plane. The default value is **0.01** (1%) which can be used for most cases. The recommended range of values is 0.01 to 0.02 (1-2%). For extreme cases, you may use a value of up to 0.05.
- **Inflate Relative Tolerance**: Improve the surface mesh in the thin layer regions after performing the layering operation. This tolerance moves nodes away to the specified value to improve the quality. The default value is **0.1** (10% of the layer height). The tolerance range of 0.1 and 0.3 help you to achieve good results. The largest acceptable value is 0.5, use this value carefully.
- **Overlapping Angle**: Identify and inflate the geometry faces overlapping the layer planes based on the provided angle. The default is **155 degrees**. The acceptable range is an angle greater than the **Feature Angle** and less than 180 degrees.
- **Defeature Layer Volume**: Specifies the threshold volume for defeaturing a thin end layer of cells. **Defeature Layer Volume** is used only if a very thin end layer is expected since such layers may cause mesh quality issues.
- **Aggressive Inflate Option**: Enables inflation of faces in the proximity of the layer planes to improve mesh quality in addition to inflation to resolve overlapping faces. This default value is **Yes**. In some cases, the aggressive inflation may cause some intersections in the surface mesh, in which case you can set the **Aggressive Inflate Option** to **No**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Aggressive Tetrahedrons Improvement**: Activates the tetrahedron mesh improvement routines. It allows you to remove the thin tetrahedrons formed during meshing. The default value is **No**. When set to **Yes**, allows the mesh to do aggressive tetrahedron improvements.
- **Sliver Triangle Height**: Identifies problematic sliver faces based on the provided height and collpases or fixes the face to improve quality. The **Sliver Triangle Height** is based on the specified minimum size (p. 113). The default value is 10% of the minimum size. Based on the model, you may need to increase the value. However, you should not use a value greater than 50% of the min size.
- **Feature Angle**: Determines feature nodes based on **Feature Angle** and are preserved during meshing. The default feature angle is **40 degrees**. The acceptable range is from 0 degrees to an angle less than the specified **Overlapping Angle**.
- **Corner Angle**: Determines corner nodes based on **Corner Angle** and are preserved during meshing. The default is **90 degrees**. The acceptable range is from 0 to 180 degrees.

Association Using Named Selections

The **Layered Tetrahedrons** method does not fully associate the mesh to the geometry. To have mesh associated to the geometry, define Named Selections on the faces on which association is required prior to meshing.

Note:

Edge and vertex named selections are not considered for mesh association.

Limitations

The **Layered Tetrahedrons** method has the following limitations:

- Multi-body parts with and without shared topology are not supported. You should separate them into individual parts in CAD.
- The model should not have other suppressed bodies.
- The mesh is not associated back to geometry. See Association Using Named Selections (p. 271) for more details.
- Only straight sided mid-nodes are available for quadratic element type.
- Contact must be used between two bodies even if they have a shared interface.

Note:

Layered Tetrahedron method does not generate conformal mesh when there is a shared interface between bodies.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Particle Method

Particle Method generates cloud of particles of specified diameter within the body. **Particle Method** is available only when the **Physics Preference** is set to **Explicit** in the **Mesh Details** view and for bodies with **Reference Frame** set to **Particle**.

Note:

- **Particle Method** does not support multi body parts.
- **Particle Method** supports only solid bodies.
- Solid bodies must have water-tight facets to be meshed with **Particle Method**.

To access the **Particle Method** control,

On the Tree Outline, right-click **Mesh** and click **Insert > Method**.

Or

On the Tree Outline, click **Mesh** and click **Method** in the **Mesh Context** tab on the Ribbon.

Select **Method** as **Particle** in the **Automatic Method** Details view.

Scope

- **Scoping Method**: Allows you to scope geometry bodies or named selection. The default value is **Geometry Selection**.
  - **Geometry Selection**: Allows you to scope the geometry bodies. When you select **Geometry Selection**, the **Geometry** allows you to select the geometry from the **Geometry** window.
  - **Named Selection**: Allows you to scope bodies grouped under a named selection.

Definition

- **Suppressed**: Allows you to suppress the selected control. The default value is **No**. When **Suppressed** is set to **Yes**, **Active** displays the status of the selected control. **Active** is read-only.
- **Method**: Allows you to select method.
- **Particle Diameter**: Allows you to define the diameter of particles to be generated.
- **Control Message**: Provides message when the **Particle Method** is scoped to an unsupported body. You can click **Control Message** to view the error message.

Note:

- If **Thick Shells and Beams** option is turned off, the particles are displayed as crosses based on their centers. To visualize the particles as spheres **Thick Shells and Beams** option must be turned on.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Error message for mesh failure referring to non-conformal facets means that the input body does not have water-tight facets. The input must be corrected in the geometry. Refaceting the geometry can help to resolve this issue in some cases or use the **Shrinkwrap** tool in Discovery or Discovery Modeling to get a fully connected faceted body before proceeding to Particle Meshing. Converting the body to faceted in the geometry tool and ensuring the faceted body is watertight is another way to resolve this issue.

Automatic (PrimeMesh)

Automatic (PrimeMesh) performs sweeping for solid models and generate quadrilateral elements for sheet body models, when **Solid-Shell Weld - Shared Nodes** is **No** under **Meshing** option in **Files > Options > Meshing**. When solid bodies cannot be swept, the body is meshed with the Patch Conforming Tetrahedron mesher. **Automatic (PrimeMesh)** creates non conformal mesh with contacts between the shell and the weld bodies.

Automatic (PrimeMesh) performs tetrahedral mesh for solid bodies even though bodies are sweepable and quadrilateral mesh for the sheet bodies, when **Solid-Shell Weld - Shared Nodes** is **Yes** under **Meshing** option in **Files > Options > Meshing**. **Automatic (PrimeMesh)** creates conformal mesh with contacts between the shell and the weld bodies.

Automatic (PrimeMesh) method can be scoped to all body types like solid, sheet and beam bodies. **Automatic (PrimeMesh)** uses global **Element Order** for meshing.

When **Solid-Shell Weld - Shared Nodes** is set to **No**,

- Geometry operations such as **Repair Topology**, **Connect** and so on are not performed on the solid bodies which are scoped to the **Automatic (PrimeMesh)**. For example, when you perform **Repair Topology** with merge face operation on two faces of the solid bodies, that operation is not executed.
- Weld generation between shell and solid bodies can be completed but the node connection between the weld and solid bodies are not conformal. Instead, contact is created during the solve.

When **Solid-Shell Weld - Shared Nodes** is set to **Yes**,

- Geometry operations such as **Repair Topology**, **Connect** and so on are performed on the scoped solid bodies which are scoped to the **Automatic (PrimeMesh)**.

To access **Automatic (PrimeMesh)**,

On the Tree Outline, right-click **Mesh** and click **Insert > Method**.

Or

On the Tree Outline, click **Mesh** and click **Method** in the **Mesh Context** tab on the Ribbon.

For solid bodies, Select **Method** as **Automatic (PrimeMesh)** in the **Automatic Method** Details view.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Automatic (PrimeMesh)** Details view has the following options:

Scope

- **Scoping Method**: Allows you to scope geometry bodies or named selection. The default value is **Geometry Selection**.
  - **Geometry Selection**: Allows you to scope the geometry bodies. When you select **Geometry Selection**, the **Geometry** allows you to select the geometry from the **Geometry** window.
  - **Named Selection**: Allows you to scope bodies grouped under a named selection.

Definition

- **Suppressed**: Allows you to suppress the selected control. The default value is **No**. When **Suppressed** is set to **Yes**, **Active** displays the status of the selected control. **Active** is read-only.
- **Method**: Allows you to select method.
- **Control Messages**: Provides an error message for errors or incompatibilities in the scoped geometries of the method. **Control Messages** is available only when **Use Adaptive Sizing** is set to **Yes**, the input geometry has symmetries or two **Automatic (PrimeMesh)** method is scoped to the same body. You can click **Control Messages** to view the error message.

Advanced

- **Shell Mesh Type**: Allows you to select the type of shell mesh. The default value is **Quadrilateral**.
  - **Triangle**: Allows you to generate a triangular mesh for shell bodies.
  - **Quadrilateral**: Allows you to generate a quad-dominant mesh for shell bodies.
    - → **Triangle Reduction**: Allows you to reduce the number of triangular elements while meshing with **Quadrilateral** mesh type. The available options are **None**, **Conservative**, and **Aggressive**. The default option is **Conservative**. **Triangle Reduction** is available only when **Shell Mesh Type** is **Quadrilateral**.
    - → **None**: Allows you to generate quadrilateral mesh without reducing the triangle count on the whole body.
    - → **Conservative**: Allows you to move the triangles near to the boundary nodes or edges away from the boundary and then reduces the triangle count on the whole body.
    - → **Aggressive**: Allows you to move the interior triangles and the boundary triangles to form pairs of connecting triangles and then reduces the triangle count on the whole body. Hence, **Aggressive** removes more triangles than **Conservative**.

Note:

Aggressive may reduce the quality of the mesh while removing triangles.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Auto Repair Topology**: Allows you to automatically repair topologies of the scoped bodies, when set to **Yes**. The default value is **No**. The available options are:

  – **Partial Defeature**: Allows you to partially merge the geometry by suppressing only a portion of the edge joining two faces based on **Thin Face Width**, **Sharp Angle** and **Feature Angle**.

  Partial Defeature is useful for retaining important geometric features without merging the faces completely at thin faces or sharp angles.

  When you set **Partial Defeature** to **Yes**, mesher automatically sets the **Remove Thin Faces** and **Remove Sharp Angle** to **Yes** by default and you cannot update **Remove Thin Faces** and **Remove Sharp Angle**.

  – **Remove Thin Faces**: Removes the thin faces by merging them with the neighboring faces below the specified tolerance. When **Remove Thin Faces** is **Yes**, the available option is :

  → **Thin Face Width**: Allows you to specify the width of the thin face. The default value for **Thin Face Width** is same as the **Connection Tolerance**.

  – **Collapse Short Edge**: Collapses the edges below the specified tolerance. Feature edges and Protected edges are not collapsed. The default value is **No**. When the **Collapse Short Edges** is **Yes**, the available option is:

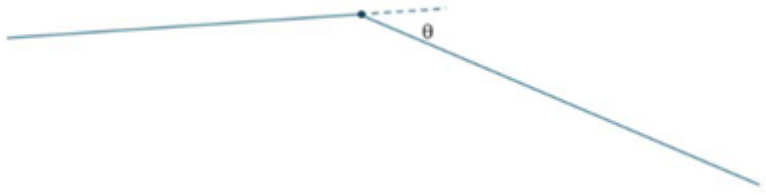
  → **Short Edge Length**: Allows you to specify the shortest edge length to perform collapse operation. The default value for **Collapse Short Edges** is same as the **Connection Tolerance**.

  – **Remove Sharp Angle Faces**: Removes the sharp angle faces below the specified angle tolerance by merging them to the neighboring faces. The default value is **No**. When **Remove Sharp Angle Faces** is **Yes**, the available option is:

  → **Sharp Angle**: Allows you to capture faces below the specified angle. The default value for **Sharp Angle** is **10 degrees**.

  – **Feature Angle**: Specifies the minimum dihedral angle at which the geometry features are repaired. The default value is **30 degrees**. You can provide any value between 0 to 180 degrees.

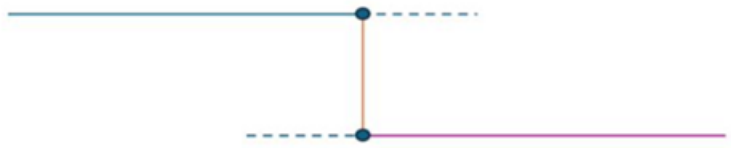
  Feature Angle is the dihedral angle measured as follows:

  When θ **= 0**, two surfaces are perfectly tangential, and the edges are not protected. Hence, the two surfaces can be merged, or edges can be suppressed, pinched or more.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When **30** $\leq \theta \leq$ **90** degrees, the edges between the two surfaces are protected and cannot perform merging or pinching at the location.



$\theta = 90$

When θ **> 90** degrees,the surfaces are not protected and prevents topology operations like merge or pinch on faces or edges.

Note:

When you scope same body or part with the **Auto Repair Topology** and **Repair Topology**, the Message window displays a warning message that multiple repair topology options overlap. Software executes controls with the smallest values first, which can lead to unexpected result.

- **Repair Tolerance**: Allows you to repair and collapse the smallest edges of faces. The default value is the same as the global **Defeature Size**.

  When **Repair Tolerance** is default and you modify global **Defeature Size**, the mesher automatically updates the **Repair Tolerance**.

  When you modify **Repair Tolerance**, the mesher does not automatically update the global **Defeature Size**.

  Note:

  When you provide a high value of **Repair Tolerance**, it may lead to mesh failure. When you scope same body or part with the **Repair Tolerance** in **Automatic (PrimeMesh) Method** and **Connect** control, the Message window displays a warning message that **Connection Tolerance** overrides **Repair Tolerance** in **Automatic (PrimeMesh) Method**.

Best Practices

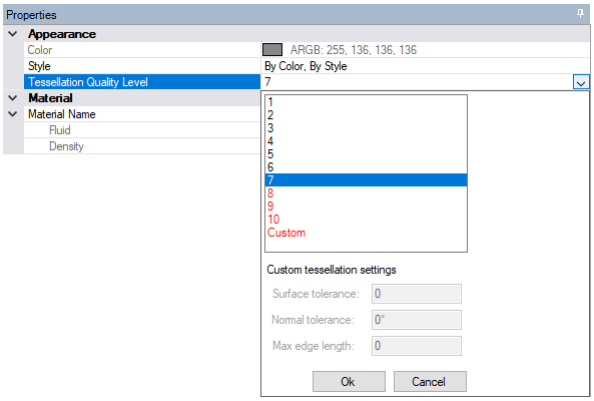
The recommended inputs for **Automatic (PrimeMesh)** are as follows:

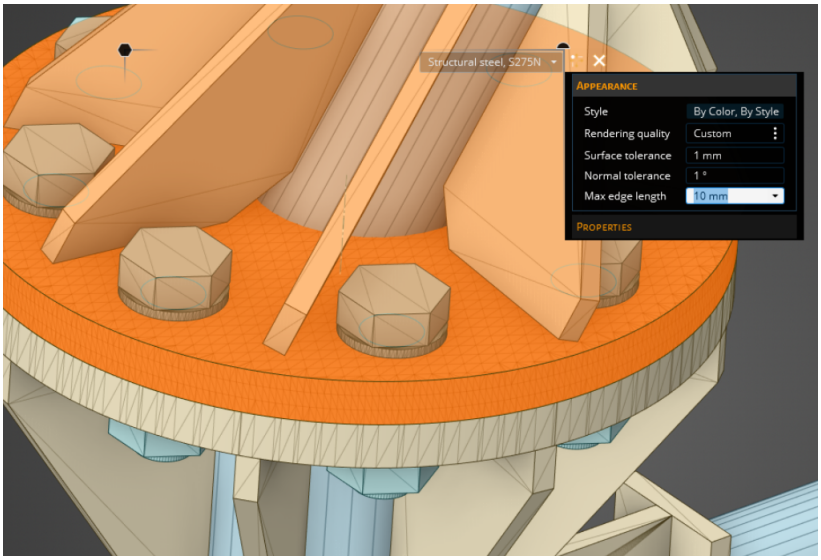
Set the facet quality to 7 before importing model into Mechanical. You can set the facet quality in Discovery Modeling by navigating through **SpaceClaim > SpaceClaim Options > Rendering Quality**. The facet quality in DesignModeler can be set by navigating through **DesignModeler > Tools > Options >Graphics > Facet Quality**.

- For bodies with large cylindrical faces, a facet quality of 10 may be required for adequate geometric representation during meshing. To avoid rendering the whole model with this

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

value, you can set this value on each body using **Body Properties** in Discovery Modeling or under **Appearance** in Discovery.





- In Discovery Modeling, DesignModeler or in other upstream CAD package, **Extend** operation should be performed. The largest connection gap to be resolved should be smaller than the specified **Element size**.
- Remove overhangs in upstream CAD. Overhangs are small penetrations between two surfaces. When unresolved overhangs can lead to mesh failure or mesh quality issues.
- Imprinting edges or performing share topology at CAD level is not recommended. If CAD model has edges already imprinted, then use **Repair Topology** (p. 355) to remove them.
- If the model has faces with missing facets, then you should fix them in upstream CAD.

The suggested general connection strategy is for clean models that do not have much difference between the smallest and the largest gap to be connected across. The best practices to follow while connecting two entities are as follows:

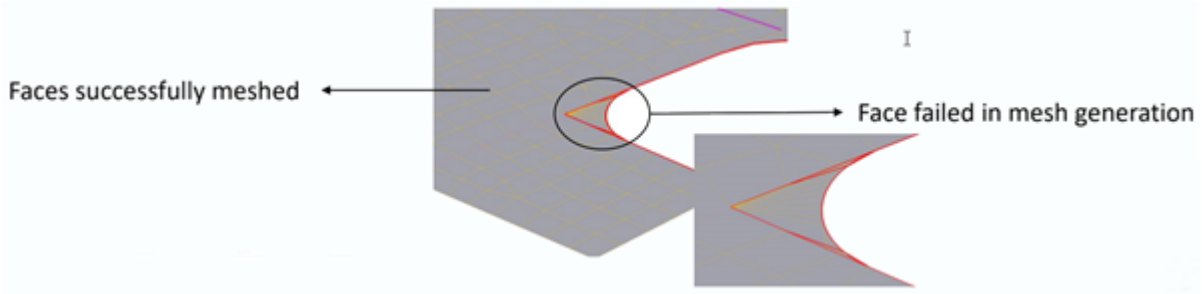
- Find the smallest and largest gap between the entities to be connected.
  - Define a list of connection tolerance such that the smallest gap is less than the smallest connection tolerance and the largest connection tolerance is greater than the largest gap.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - Check for any warning message on unconnected edges reported during meshing.
  - Right-click the warning message and select **Show Problematic Geometry**. Use model **Walk** to analyze the unconnected regions.
  - Create named selections on unconnected entities.

Remember the following while performing **Automatic (PrimeMesh)**:

- The gap size between the entities to be connected must be smaller than the **Element Size**. For example, if the biggest gap size to be resolved is 1 mm then the element size must be greater than 1 mm.
- Faces (width) and edges (length) of the connected entities below the connection tolerance(s) are defeatured.
- Mesh is not associated with defeatured entities (edge, face, body/part).
- Mesh is not associated with vertices.
- In **Automatic (PrimeMesh)**, the **Sheet Loop Removal** only works if the connection is made through the **Connect** control.
- In case of quadrilateral mesh failure, triangle mesh is generated with a warning message displayed.
  - Right-click the warning message and select **Show Problematic Topology** to display the bodies on which triangle mesh was generated.
  - If triangle mesh also fails, then the mesh failure is reported and CAD facets will be displayed.



Limitations

The limitations of **Automatic (PrimeMesh)** are as follows:

- Overlapping beams are not supported. Mesh in the overlapping portion of the beams will be assigned to either one of the involved beams.
- **Mapped Meshing** does not support mapping of annular region.
- **Weld** and Mapped control override all other sizing and local control settings.
- **Automatic (PrimeMesh)** does not support multi-body parts which includes solid body parts.

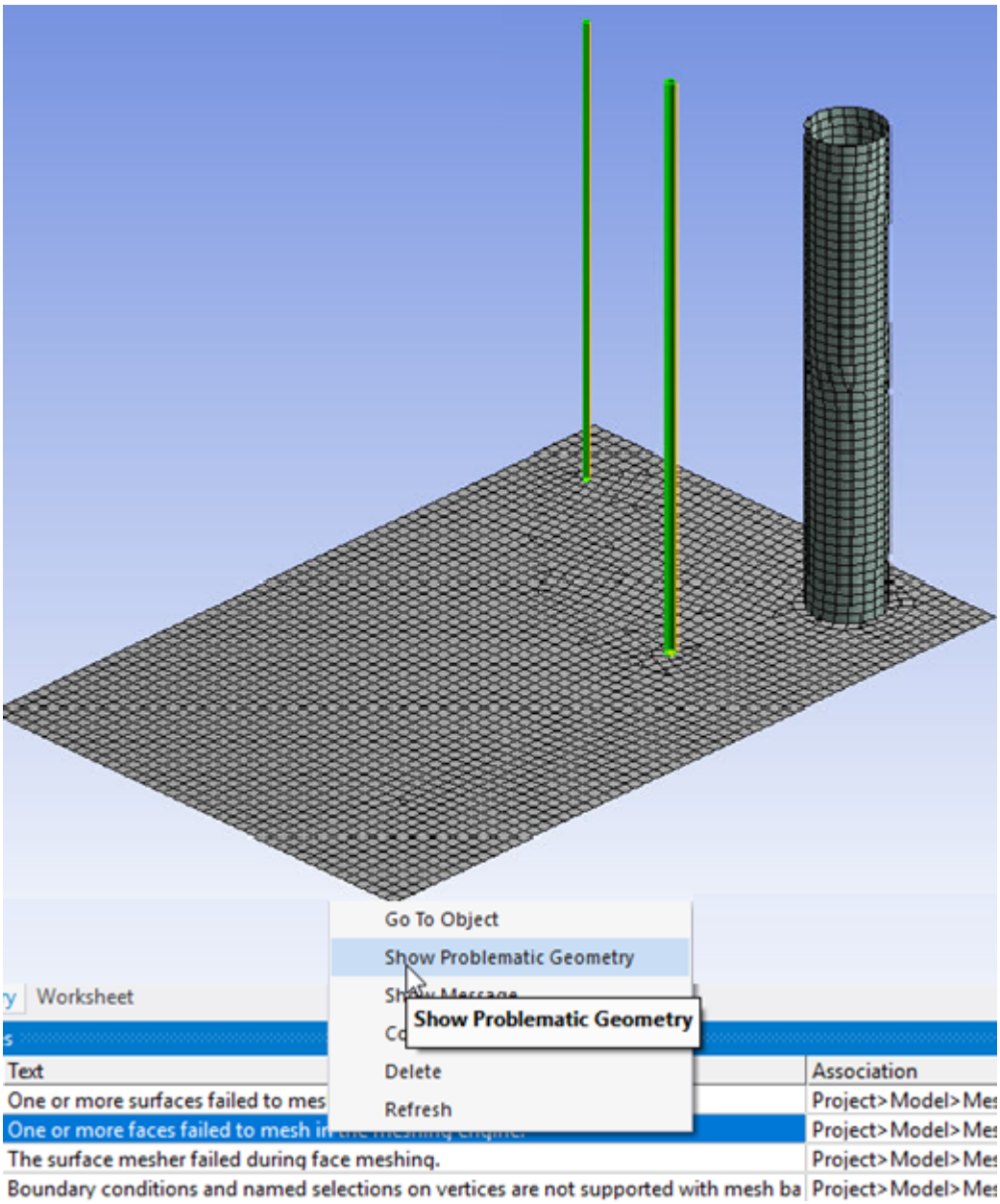
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Automatic (PrimeMesh)** does not create conformal meshes between solids.
- **Automatic (PrimeMesh)** does not support **Layered Tetrahedrons Method**.
- **Automatic (PrimeMesh)** does not support global **Defeature Size**, when **Mesh Defeaturing** is set to **Yes** in the **Sizing** option. In **Automatic (PrimeMesh)**, you must use Repair Topology (p. 355) to defeature the geometry. Also, in **Repair Topology** set **Repair Thin Faces Options** and **Repair Short Edges Options** to **Yes** for automatically repairing short edges and thin faces.
- **Automatic (PrimeMesh)** does not support **Inflation** control, **Match** control, **Pinch** control, **Adaptive Sizing**, **BOI Size**, **Vertex Size** (Sphere of Influence).
- **Automatic (PrimeMesh)** does not support quality targets provided to the mesher.
- **Automatic (PrimeMesh)** Method applies **Face Meshing** and local **Sizing** controls after performing any merge and suppress operations defined under **Auto Repair Topology** or **Repair Topology** control. In such cases, **Face Meshing** and local **Sizing** controls are applied to a larger composite face than the initial scoped faces. To avoid faces being modified by topology repair operations, you can apply a **Named Selection** to the faces with **Protected** as **Yes**.

Ease of Use Features

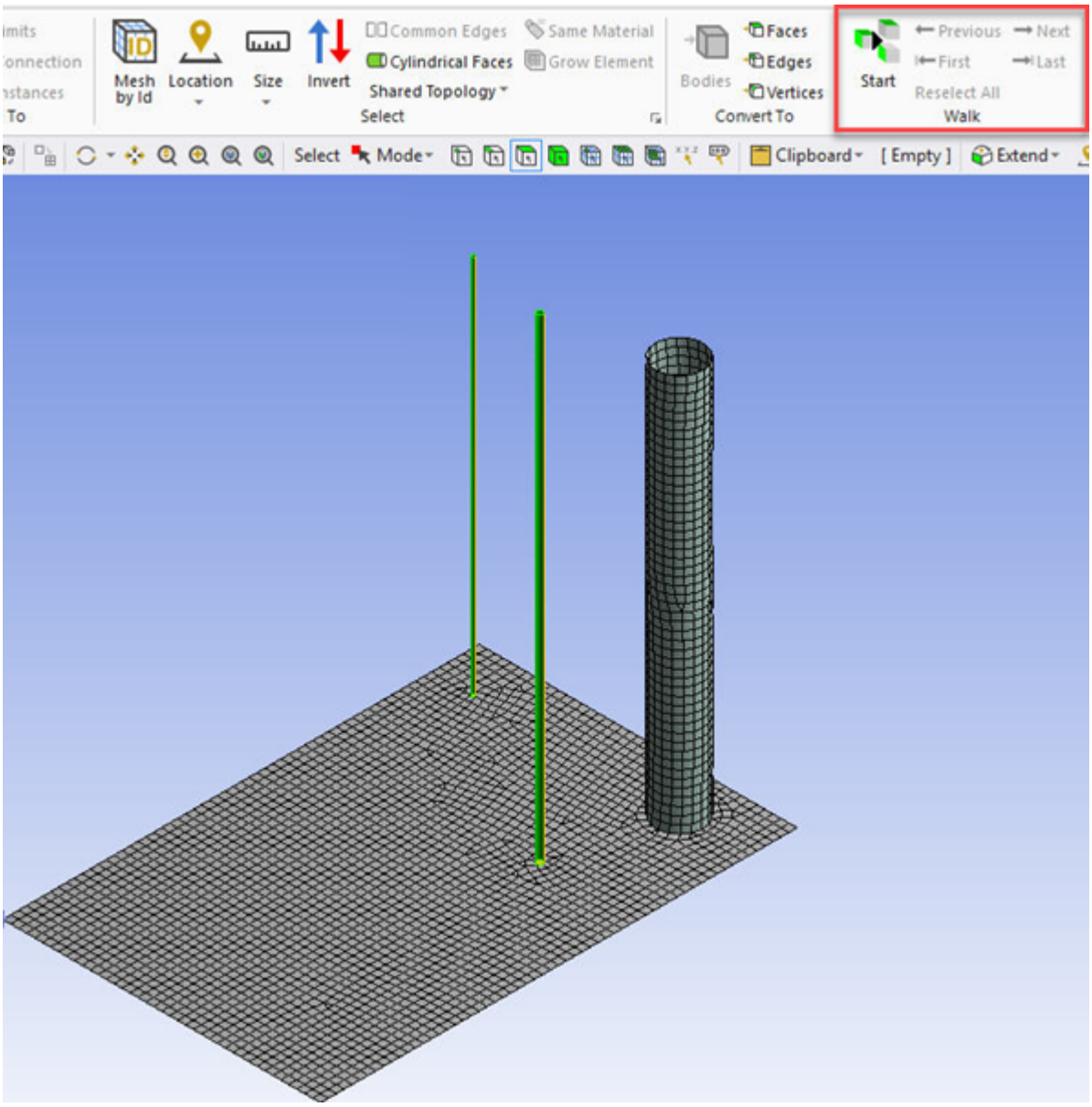
When an error or warning such as unconnected edges or mesh failure occurs, you can right-click the corresponding message and select **Show Problematic Geometry** to highlight the location at which error occurred in the model.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

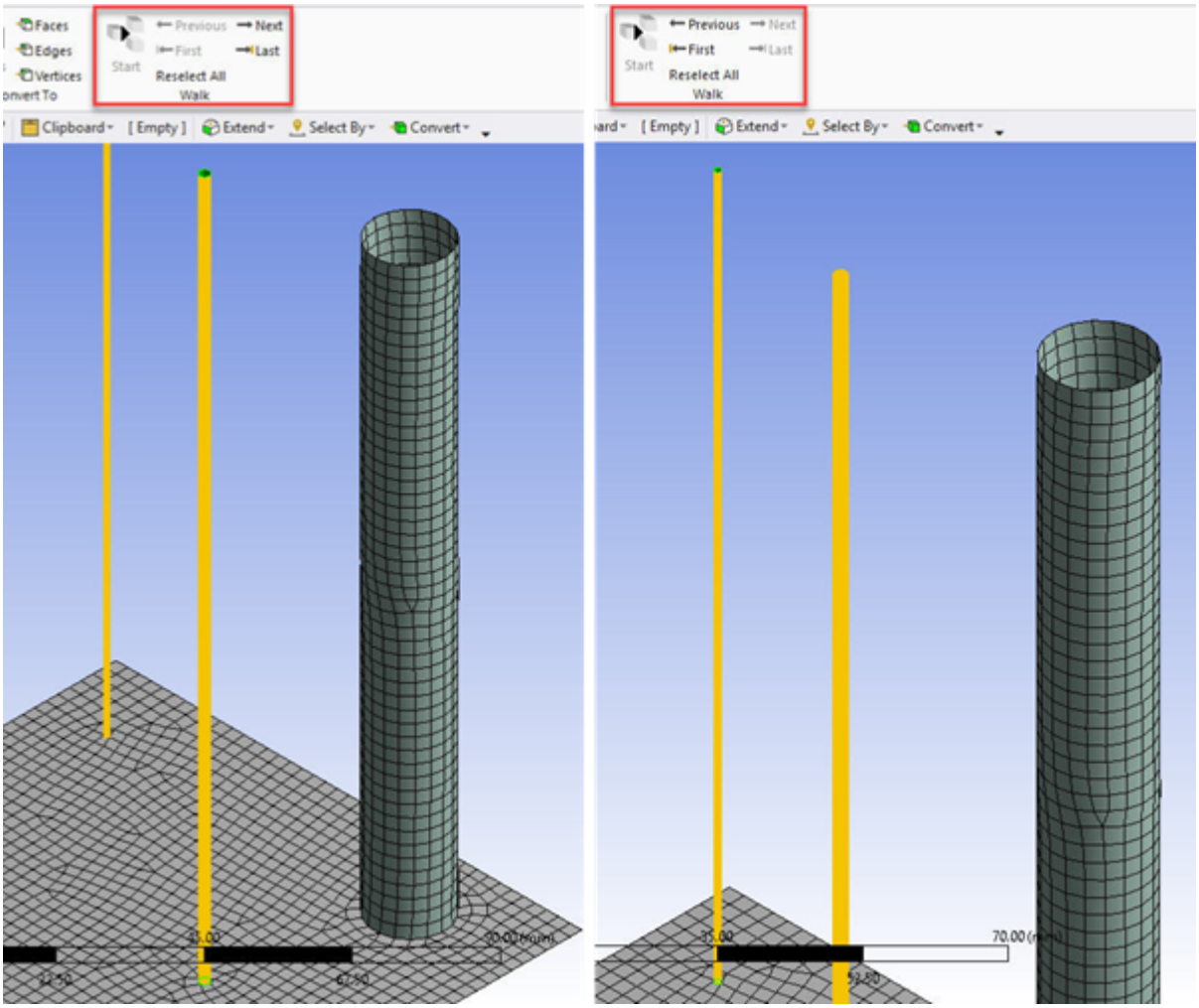


You can click **Selections** tab in the ribbon and click **Start** in **Walk** group to walk through the error locations. You can use **Next** to walk to the next location. You can click **Previous** to go back to the previous error location.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can view the unconnected faces after meshing. To view the unconnected faces, right-click the Geometry window. Select **Diagnostics > Find Unconnected Faces**. The unconnected faces get highlighted in the model.

When entities scoped to **Named Selection** with **Protected** set to **Yes** are modified while performing **Automatic (PrimeMesh)**, you will receive a warning message.

Setting the Method Control for Sheet Bodies

Method Controls available for sheet bodies are:

Quadrilateral Dominant Method Control

Triangles Method Control

MultiZone Quad/Tri Method Control

Automatic (PrimeMesh)

Quadrilateral Dominant Method Control

Quadrilateral Dominant Method creates the free quad mesh on the body. To access the **Quadrilateral Dominant method**,

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

On the Tree Outline, right-click **Mesh** and click **Insert > Method**.

Or

On the Tree Outline, click **Mesh** and click **Method** in the **Mesh Context** tab on the Ribbon.

Scope

- **Scoping Method**: Allows you to scope geometry bodies or named selection. The default value is **Geometry Selection**.
  - **Geometry Selection**: Allows you to scope the geometry bodies. When you select **Geometry Selection**, the **Geometry** allows you to select the geometry from the **Geometry** window.
  - **Named Selection**: Allows you to scope bodies grouped under a named selection.

Definition

- **Suppressed**: Allows you to suppress the selected control. The default value is **No**. When **Suppressed** is set to **Yes**, **Active** displays the status of the selected control. **Active** is read-only.
- **Method**: Allows you to select method.
- **Element Order**: Allows you to select the element order. The default value is **Use Global Setting**. For information about the **Element Order** option, refer Method Controls and Element Order Settings (p. 202).
- **Free Face Mesh Type**: Allows you to select the shape of the elements used to fill the body. The available options are **Quad/Tri** or **All Quad**. The default is **Quad/Tri**.
  - **All Quad**: Allows you to create the quadrilateral mesh.
  - **Quad/Tri**: Allows you to create the mesh with both quadrilateral and triangular elements.

Note:

- If you are using the Quadrilateral Dominant mesh method with inflation (p. 345) and the Size Function (p. 105) is on, the mesh size of the last inflation layer will be used for the corresponding Quadrilateral Dominant boundary mesh size.
- If you apply a local **Sizing** control (p. 296) to a solid body with a **Method** control set to **Hex Dominant** (p. 233) or **Sweep** (p. 235), or to a sheet body with a **Method** control set to **Quadrilateral Dominant** (p. 282), a near uniform quadrilateral mesh will result on all affected faces on a body meshed with **Hex Dominant**, on the source face meshed with **Sweep**, and on all affected faces meshed with **Quadrilateral Dominant**. To obtain even more of a uniform quadrilateral mesh, set the **Behavior** (p. 310) of the **Sizing** control to **Hard**.
- If you are meshing a multibody part that contains a mix of line bodies and surface bodies, all surface bodies and all line bodies that share edges with surface bodies will be meshed with the selected surface mesh method. Any remaining line bodies (where only vertices are shared with surface bodies) will always be meshed with the **Quadrilateral Dominant** mesh method.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Triangles Method Control

Triangles Method creates a triangular mesh. To access the **Triangles Method**,

On the Tree Outline, right-click **Mesh** and click **Insert > Method**.

Or

On the Tree Outline, click **Mesh** and click **Method** in the **Mesh Context** tab on the Ribbon.

Select **Method** as **Triangles** in the **Quadrilateral Dominant** Details view.

Scope

- **Scoping Method**: Allows you to scope geometry bodies or named selection. The default value is **Geometry Selection**.
  - **Geometry Selection**: Allows you to scope the geometry bodies. When you select **Geometry Selection**, the **Geometry** allows you to select the geometry from the **Geometry** window.
  - **Named Selection**: Allows you to scope bodies grouped under a named selection.

Definition

- **Suppressed**: Allows you to suppress the selected control. The default value is **No**. When **Suppressed** is set to **Yes**, **Active** displays the status of the selected control. **Active** is read-only.
- **Method**: Allows you to select method.
- **Element Order**: Allows you to select the element order. The default value is **Use Global Setting**. For information about the **Element Order** option, refer Method Controls and Element Order Settings (p. 202).

MultiZone Quad/Tri Method Control

MultiZone Quad/Tri Method creates a mesh of quads,triangles or both over the entire part of the selected body.

MultiZone Quad/Tri is a patch independent method.

To access the **MultiZone Quad/Tri Method**,

On the Tree Outline, right-click **Mesh** and click **Insert > Method**.

Or

On the Tree Outline, click **Mesh** and click **Method** in the **Mesh Context** tab on the Ribbon.

Select **Method** as **MultiZone Quad/Tri** in the **Quadrilateral Dominant** Details view.

Scope

- **Scoping Method**: Allows you to scope geometry bodies or named selection. The default value is **Geometry Selection**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Geometry Selection**: Allows you to scope the geometry bodies. When you select **Geometry Selection**, the **Geometry** allows you to select the geometry from the **Geometry** window.

  - **Named Selection**: Allows you to scope bodies grouped under a named selection.

Definition

- **Suppressed**: Allows you to suppress the selected control. The default value is **No**. When **Suppressed** is set to **Yes**, **Active** displays the status of the selected control. **Active** is read-only.

- **Method**: Allows you to select method.

- **Surface Mesh Method**: Instructs **MultiZone Quad/Tri** to use the **Program Controlled**, **Uniform**, or **Pave** method to create the mesh. The default method is **Program Controlled**

  - **Program Controlled**: Automatically uses a combination of **Uniform** and **Pave** mesh methods depending on the mesh sizes set and face properties.

  - **Uniform**: Uses a recursive loop-splitting method which creates a highly uniform mesh. This option is generally good when all edges have the same sizing and the faces being meshed do not have a high degree of curvature. The orthogonality of the mesh from this method is generally very good.

  - **Pave**: Uses a paving mesh method which creates a good quality mesh on faces with high curvature, and also when neighboring edges have a high aspect ratio. This approach is also more reliable to give an all-quad mesh.

Note:

The **MultiZone Quad/Tri** method ignores the Sizing Options (p. 105) when **Surface Mesh Method** is set to **Uniform**. In such cases, **Element Size** acts as a hard size.

- **Element Order** - Allows you to select the element order. Refer to Method Controls and Element Order Settings (p. 202).

- **Free Face Mesh Type** - Determines the shape of the mesh elements. The default value is **Quad/Tri**. **Free Face Mesh Type** has the following options:

  - **All Tri**: Allows you to create the triangular mesh.

  - **Quad/Tri**: Allows you to create the mesh with both quadrilateral and triangular elements.

  - **All Quad**: Allows you to create the quadrilateral mesh.

- **Element Size** - Allows you to specify the element size used for the selected geometry. Applicable only when **Surface Mesh Method** is set to **Uniform**. Otherwise, uses the global **Element Size** (p. 102).

Advanced

- **Preserve Boundaries**: Allows you to protect the boundaries from the defeature size when set to **Protected** and when set to **All** it respects all the features. The default value is **Protected**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mesh Based Defeaturing**: "Filters" edges in or out based on size and angle. The default value based on global Mesh Defeaturing (p. 111). This local Mesh Based Defeaturing setting is the same as the setting of the global Mesh Defeaturing control. When **Mesh Based Defeaturing** is On, a **Defeature Size** field appears.

  – **Defeature Size**: Allows you to set the defeature size for mesh defeaturing. **Defeature Size** is available only when **Mesh Defeaturing** is set to **Yes**. The default value is based on the global Mesh Defeaturing (p. 111). When you specify a value different from the global Defeature Size, it overrides the global value. Specifying a value of **0.0** resets the **Defeature Size** to its default. A recommended setting is at least one-half the value set for **Element Size** to assure a successful mesh. You can parametrize **Defeature Size**.

  Note:

  When the global Mesh Defeaturing (p. 111) control is on but **Use Adaptive Sizing** is set to **Yes**, the default defeaturing performed for **MultiZone Quad/Tri** includes defeaturing based on the dihedral angle between the faces as well as edge length defeaturing based on the smallest element size set by the user.

- **Sheet Loop Removal**: Removes holes on surface bodies based on size. If set to **Yes**, a **Loop Removal Tolerance** field appears. The default value is **No**.

  – **Loop Removal Tolerance**: Sets the tolerance for loop removal. You can specify a numerical value greater than **0.0**. By default, the value of this local **Loop Removal Tolerance** field is the same as the global Loop Removal Tolerance (p. 197). When you specify a value different from the global **Loop Removal Tolerance**. It overrides the global value. Holes with boundary conditions applied are not removed from the mesh. Any boundary conditions applied to holes that were removed from the mesh will not be respected by the solver.

- **Minimum Edge Length**: Read-only indication of the smallest edge length in the part.

- **Write ICEM CFD Files**: Sets options for writing Ansys ICEM CFD files. Refer to Writing Ansys ICEM CFD Files (p. 84) for details.

Usage Information for the MultiZone Quad/Tri Mesh Method Control

The following usage information is applicable to the **MultiZone Quad/Tri** mesh method:

- **MultiZone Quad/Tri** does not support base mesh caching, so a change to inflation controls requires remeshing.

- Using **MultiZone Quad/Tri** may allow meshing over very small bodies in a multibody part. This may lead to a solver error if a body load is associated with that body. If this is the case, you must suppress the body (p. 528) before solving your model.

- Surface bodies with specified variable thickness are not protected topology. To prevent faces and their boundaries from being meshed over, create an individual Named Selection for each thickness.

- The **MultiZone Quad/Tri** mesh method supports mesh connections (p. 485).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You can use the **MultiZone Quad/Tri** mesh method in combination with other surface mesh methods in a multibody part, and meshes the bodies with conformal mesh. If you select the **MultiZone Quad/Tri** mesh method to mesh a multibody part that contains a mix of line bodies and surface bodies, all surface bodies and all line bodies that share edges with surface bodies will be meshed with the selected method. Any remaining line bodies (where only vertices are shared with surface bodies) will be meshed with the **Quadrilateral Dominant** (p. 282) mesh method. Refer to Conformal and Non-Conformal Meshing (p. 21) for more information about conformal meshing.
- When meshing multibody parts having solid bodies and sheet bodies with some faces shared, it is best to mesh the shared faces with **MultiZone Quad/Tri**. If the faces are meshed using the solid body mesh method control, the **MultiZone Quad/Tri** body will often fail.

Caution:

Multiple environments with different loadings may over-constrain the **MultiZone Quad/Tri** mesher such that the mesher may not be able to return a mesh for the given inputs. If discretization error is not an issue, the mesher will be less constrained if you duplicate the model and change the environment instead of adding multiple environments under the same model.

Automatic (PrimeMesh)

Automatic (PrimeMesh) performs sweeping for solid models and generate quadrilateral elements for sheet body models, when **Solid-Shell Weld - Shared Nodes** is **No** under **Meshing** option in **Files > Options > Meshing**. When solid bodies cannot be swept, the body is meshed with the Patch Conforming Tetrahedron mesher. **Automatic (PrimeMesh)** creates non conformal mesh with contacts between the shell and the weld bodies.

Automatic (PrimeMesh) performs tetrahedral mesh for solid bodies even though bodies are sweepable and quadrilateral mesh for the sheet bodies, when **Solid-Shell Weld - Shared Nodes** is **Yes** under **Meshing** option in **Files > Options > Meshing**. **Automatic (PrimeMesh)** creates conformal mesh with contacts between the shell and the weld bodies.

Automatic (PrimeMesh) method can be scoped to all body types like solid, sheet and beam bodies. **Automatic (PrimeMesh)** uses global **Element Order** for meshing.

When **Solid-Shell Weld - Shared Nodes** is set to **No**,

- Geometry operations such as **Repair Topology**, **Connect** and so on are not performed on the solid bodies which are scoped to the **Automatic (PrimeMesh)**. For example, when you perform **Repair Topology** with merge face operation on two faces of the solid bodies, that operation is not executed.
- Weld generation between shell and solid bodies can be completed but the node connection between the weld and solid bodies are not conformal. Instead, contact is created during the solve.

When **Solid-Shell Weld - Shared Nodes** is set to **Yes**,

- Geometry operations such as **Repair Topology**, **Connect** and so on are performed on the scoped solid bodies which are scoped to the **Automatic (PrimeMesh)**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To access **Automatic (PrimeMesh)**,

On the Tree Outline, right-click **Mesh** and click **Insert > Method**.

Or

On the Tree Outline, click **Mesh** and click **Method** in the **Mesh Context** tab on the Ribbon.

For sheet bodies, Select **Method** as **Automatic (PrimeMesh)** in the **Quadrilateral Dominant** Details view.

Scope

- **Scoping Method**: Allows you to scope geometry bodies or named selection. The default value is **Geometry Selection**.
  - **Geometry Selection**: Allows you to scope the geometry bodies. When you select **Geometry Selection**, the **Geometry** allows you to select the geometry from the **Geometry** window.
  - **Named Selection**: Allows you to scope bodies grouped under a named selection.

Definition

- **Suppressed**: Allows you to suppress the selected control. The default value is **No**. When **Suppressed** is set to **Yes**, **Active** displays the status of the selected control. **Active** is read-only.
- **Method**: Allows you to select method.
- **Control Messages**: Provides an error message for errors or incompatibilities in the scoped geometries of the method. **Control Messages** is available only when **Use Adaptive Sizing** is set to **Yes**, the input geometry has symmetries or two **Automatic (PrimeMesh)** method is scoped to the same body. You can click **Control Messages** to view the error message.

Advanced

- **Shell Mesh Type**: Allows you to select the type of shell mesh. The default value is **Quadrilateral**.
  - **Triangle**: Allows you to generate a triangular mesh for shell bodies.
  - **Quadrilateral**: Allows you to generate a quad-dominant mesh for shell bodies.
    - → **Triangle Reduction**: Allows you to reduce the number of triangular elements while meshing with **Quadrilateral** mesh type. The available options are **None**, **Conservative**, and **Aggressive**. The default option is **Conservative**. **Triangle Reduction** is available only when **Shell Mesh Type** is **Quadrilateral**.
    - → **None**: Allows you to generate quadrilateral mesh without reducing the triangle count on the whole body.
    - → **Conservative**: Allows you to move the triangles near to the boundary nodes or edges away from the boundary and then reduces the triangle count on the whole body.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ **Aggressive**: Allows you to move the interior triangles and the boundary triangles to form pairs of connecting triangles and then reduces the triangle count on the whole body. Hence, **Aggressive** removes more triangles than **Conservative**.

Note:

Aggressive may reduce the quality of the mesh while removing triangles.

- **Auto Repair Topology**: Allows you to automatically repair topologies of the scoped bodies, when set to **Yes**. The default value is **No**. The available options are:

  – **Partial Defeature**: Allows you to partially merge the geometry by suppressing only a portion of the edge joining two faces based on **Thin Face Width**, **Sharp Angle** and **Feature Angle**.

  Partial Defeature is useful for retaining important geometric features without merging the faces completely at thin faces or sharp angles.

  When you set **Partial Defeature** to **Yes**, mesher automatically sets the **Remove Thin Faces** and **Remove Sharp Angle** to **Yes** by default and you cannot update **Remove Thin Faces** and **Remove Sharp Angle**.

  – **Remove Thin Faces**: Removes the thin faces by merging them with the neighboring faces below the specified tolerance. When **Remove Thin Faces** is **Yes**, the available option is :

  → **Thin Face Width**: Allows you to specify the width of the thin face. The default value for **Thin Face Width** is same as the **Connection Tolerance**.

  – **Collapse Short Edge**: Collapses the edges below the specified tolerance. Feature edges and Protected edges are not collapsed. The default value is **No**. When the **Collapse Short Edges** is **Yes**, the available option is:

  → **Short Edge Length**: Allows you to specify the shortest edge length to perform collapse operation. The default value for **Collapse Short Edges** is same as the **Connection Tolerance**.

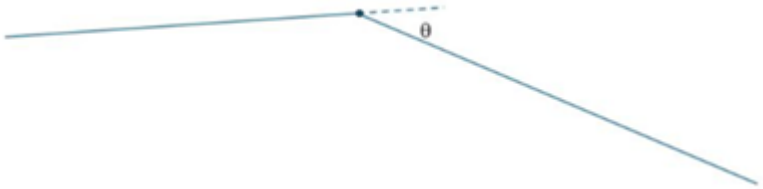
  – **Remove Sharp Angle Faces**: Removes the sharp angle faces below the specified angle tolerance by merging them to the neighboring faces. The default value is **No**. When **Remove Sharp Angle Faces** is **Yes**, the available option is:

  → **Sharp Angle**: Allows you to capture faces below the specified angle. The default value for **Sharp Angle** is **10 degrees**.

  – **Feature Angle**: Specifies the minimum dihedral angle at which the geometry features are repaired. The default value is **30 degrees**. You can provide any value between 0 to 180 degrees.

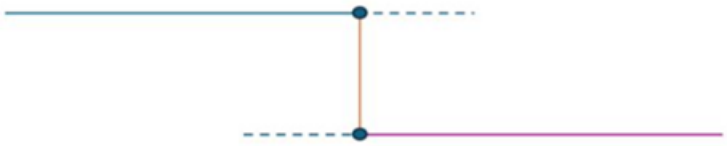
  Feature Angle is the dihedral angle measured as follows:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When θ **= 0**, two surfaces are perfectly tangential, and the edges are not protected. Hence, the two surfaces can be merged, or edges can be suppressed, pinched or more.

When **30** $\leq \theta \leq$ **90** degrees, the edges between the two surfaces are protected and cannot perform merging or pinching at the location.



θ = 90

When θ **> 90** degrees,the surfaces are not protected and prevents topology operations like merge or pinch on faces or edges.

Note:

When you scope same body or part with the **Auto Repair Topology** and **Repair Topology**, the Message window displays a warning message that multiple repair topology options overlap. Software executes controls with the smallest values first, which can lead to unexpected result.

- **Repair Tolerance**: Allows you to repair and collapse the smallest edges of faces. The default value is the same as the global **Defeature Size**.

  When **Repair Tolerance** is default and you modify global **Defeature Size**, the mesher automatically updates the **Repair Tolerance**.

  When you modify **Repair Tolerance**, the mesher does not automatically update the global **Defeature Size**.

  Note:

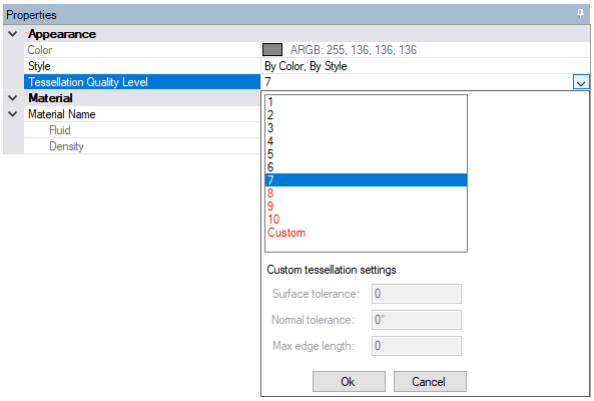
  When you provide a high value of **Repair Tolerance**, it may lead to mesh failure. When you scope same body or part with the **Repair Tolerance** in **Automatic (PrimeMesh) Method** and **Connect** control, the Message window displays a warning message that **Connection Tolerance** overrides **Repair Tolerance** in **Automatic (PrimeMesh) Method**.

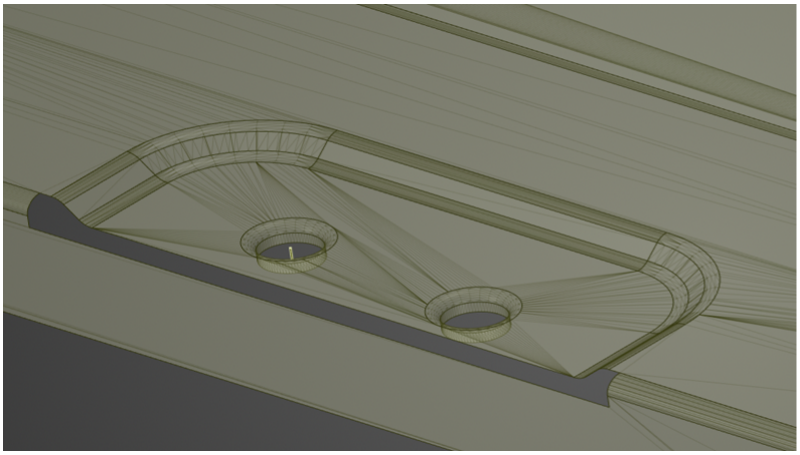
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

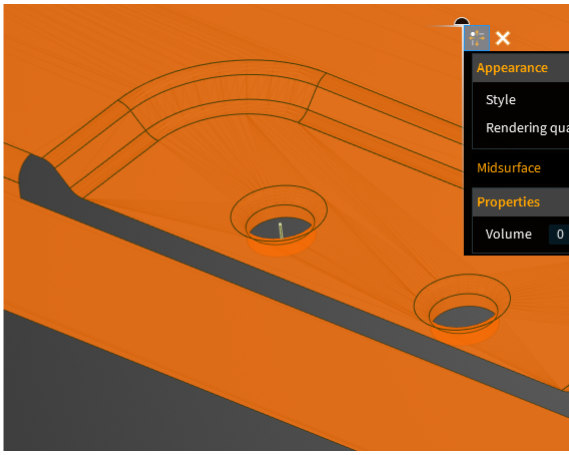
Best Practices

The recommended inputs for **Automatic (PrimeMesh)** are as follows:

- Set the facet quality to 7 before importing model into Mechanical. You can set the facet quality in Discovery Modeling by navigating through **SpaceClaim > SpaceClaim Options > Rendering Quality**. The facet quality in DesignModeler can be set by navigating through **DesignModeler > Tools > Options >Graphics > Facet Quality**.
- For bodies with large cylindrical faces, a facet quality of 10 may be required for adequate geometric representation during meshing. To avoid rendering the whole model with this value, you can set this value on each body using **Body Properties** in Discovery Modeling or under **Appearance** in Discovery.







- In Discovery Modeling, DesignModeler, or in other upstream CAD package, **Extend** operation should be performed. The largest connection gap to be resolved should be smaller than the specified **Element size**.
- Remove overhangs in upstream CAD. Overhangs are small penetrations between two surfaces. When unresolved overhangs can lead to mesh failure or mesh quality issues.
- Imprinting edges or performing share topology at CAD level is not recommended. If CAD model has edges already imprinted, then use **Repair Topology** (p. 355) to remove them.
- If the model has faces with missing facets, then you should fix them in upstream CAD.

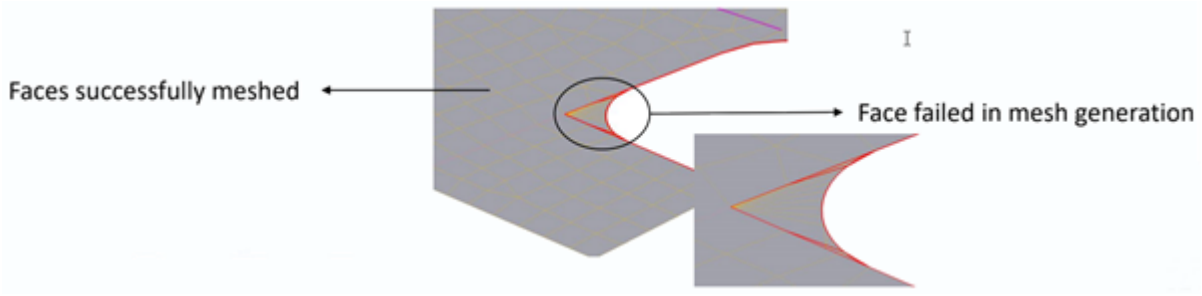
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The suggested general connection strategy is for clean models that do not have much difference between the smallest and the largest gap to be connected across. The best practices to follow while connecting two entities are as follows:

- Find the smallest and largest gap between the entities to be connected.
  - Define a list of connection tolerance such that the smallest gap is less than the smallest connection tolerance and the largest connection tolerance is greater than the largest gap.
  - Check for any warning message on unconnected edges reported during meshing.
  - Right-click the warning message and select **Show Problematic Geometry**. Use model **Walk** to analyze the unconnected regions.
  - Create named selections on unconnected entities.

Remember the following while performing **Automatic (PrimeMesh)**:

- The gap size between the entities to be connected must be smaller than the **Element Size**. For example, if the biggest gap size to be resolved is 1 mm then the element size must be greater than 1 mm.
- Faces (width) and edges (length) of the connected entities below the connection tolerance(s) are defeatured.
- Mesh is not associated with defeatured entities (edge, face, body/part).
- Mesh is not associated with vertices.
- In **Automatic (PrimeMesh)**, the **Sheet Loop Removal** only works if the connection is made through the **Connect** control.
- In case of quadrilateral mesh failure, triangle mesh is generated with a warning message displayed.
  - Right-click the warning message and select **Show Problematic Topology** to display the bodies on which triangle mesh was generated.
  - If triangle mesh also fails, then the mesh failure is reported and CAD facets will be displayed.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Limitations

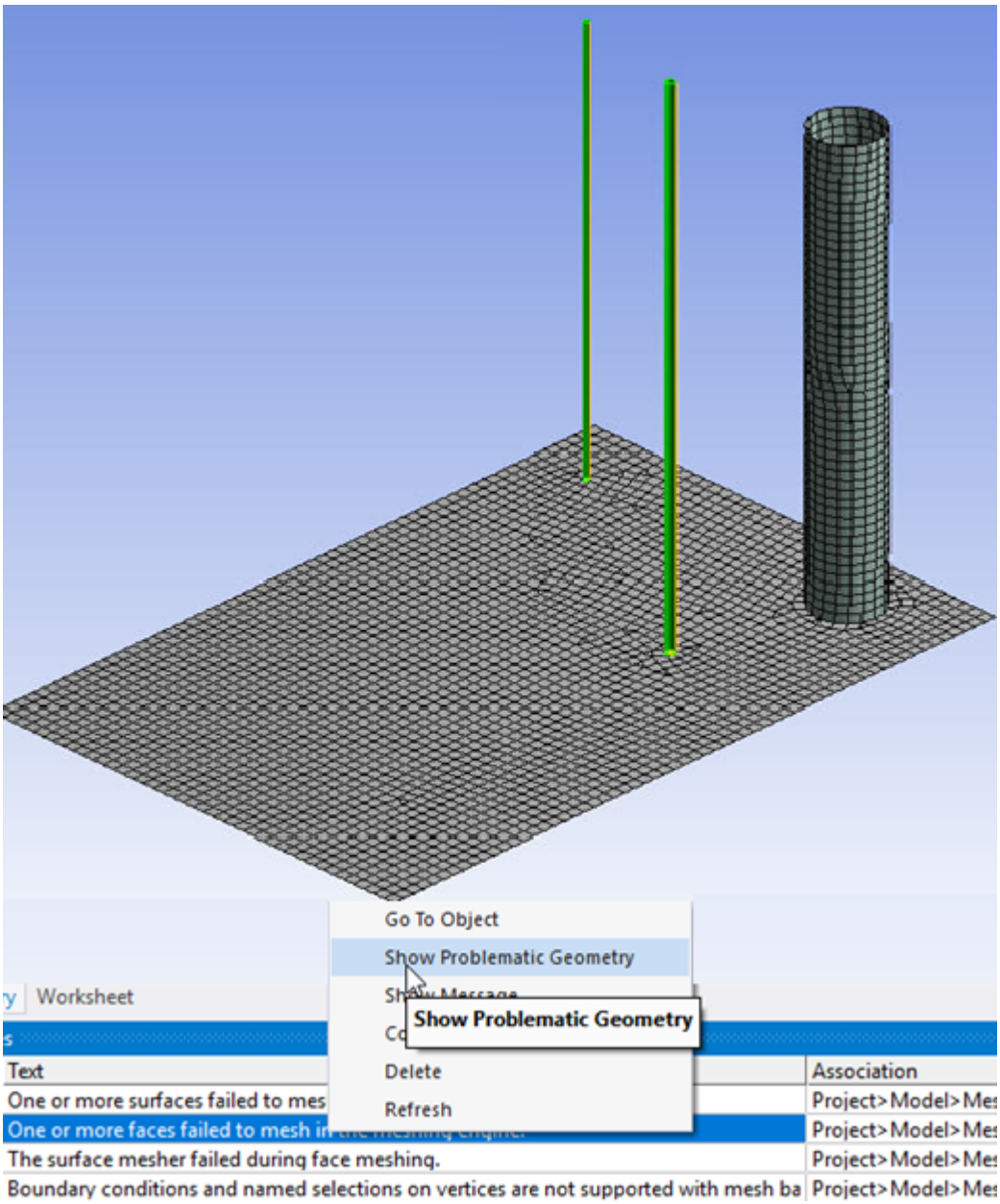
The limitations of **Automatic (PrimeMesh)** are as follows:

- Overlapping beams are not supported. Mesh in the overlapping portion of the beams will be assigned to either one of the involved beams.
- **Mapped Meshing** does not support mapping of annular region.
- **Weld** and Mapped control override all other sizing and local control settings.
- **Automatic (PrimeMesh)** does not support multi-body parts which includes solid body parts.
- **Automatic (PrimeMesh)** does not create conformal meshes between solids.
- **Automatic (PrimeMesh)** does not support **Layered Tetrahedrons Method**.
- **Automatic (PrimeMesh)** does not support global **Defeature Size**, when **Mesh Defeaturing** is set to **Yes** in the **Sizing** option. In **Automatic (PrimeMesh)**, you must use Repair Topology (p. 355) to defeature the geometry. Also, in **Repair Topology** set **Repair Thin Faces Options** and **Repair Short Edges Options** to **Yes** for automatically repairing short edges and thin faces.
- **Automatic (PrimeMesh)** does not support **Inflation** control, **Match** control, **Pinch** control, **Adaptive Sizing**, **BOI Size**, **Vertex Size** (Sphere of Influence).
- **Automatic (PrimeMesh)** does not support quality targets provided to the mesher.

Ease of Use Features

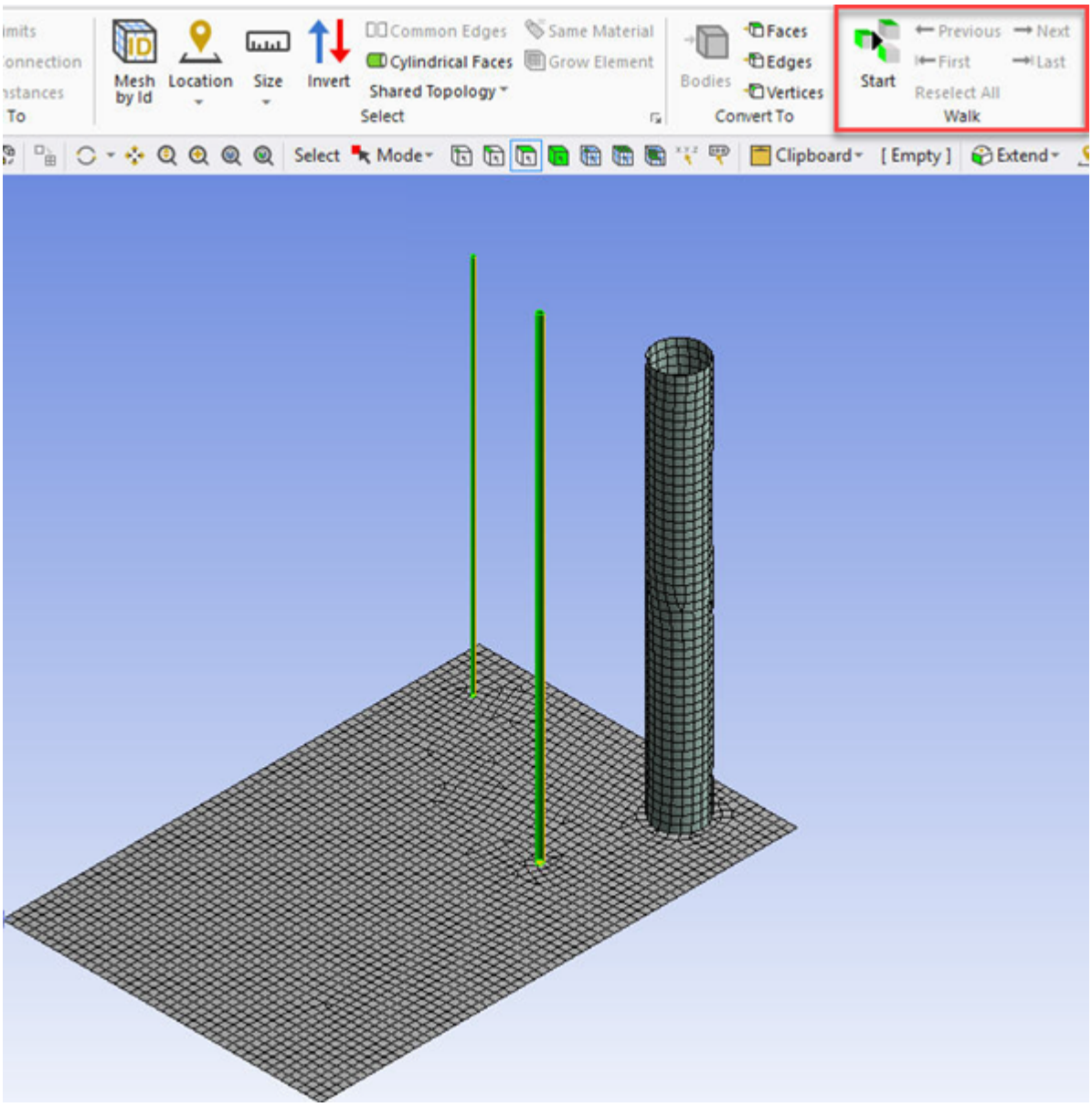
When an error or warning such as unconnected edges or mesh failure occurs, you can right-click the corresponding message and select **Show Problematic Geometry** to highlight the location at which error occurred in the model.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

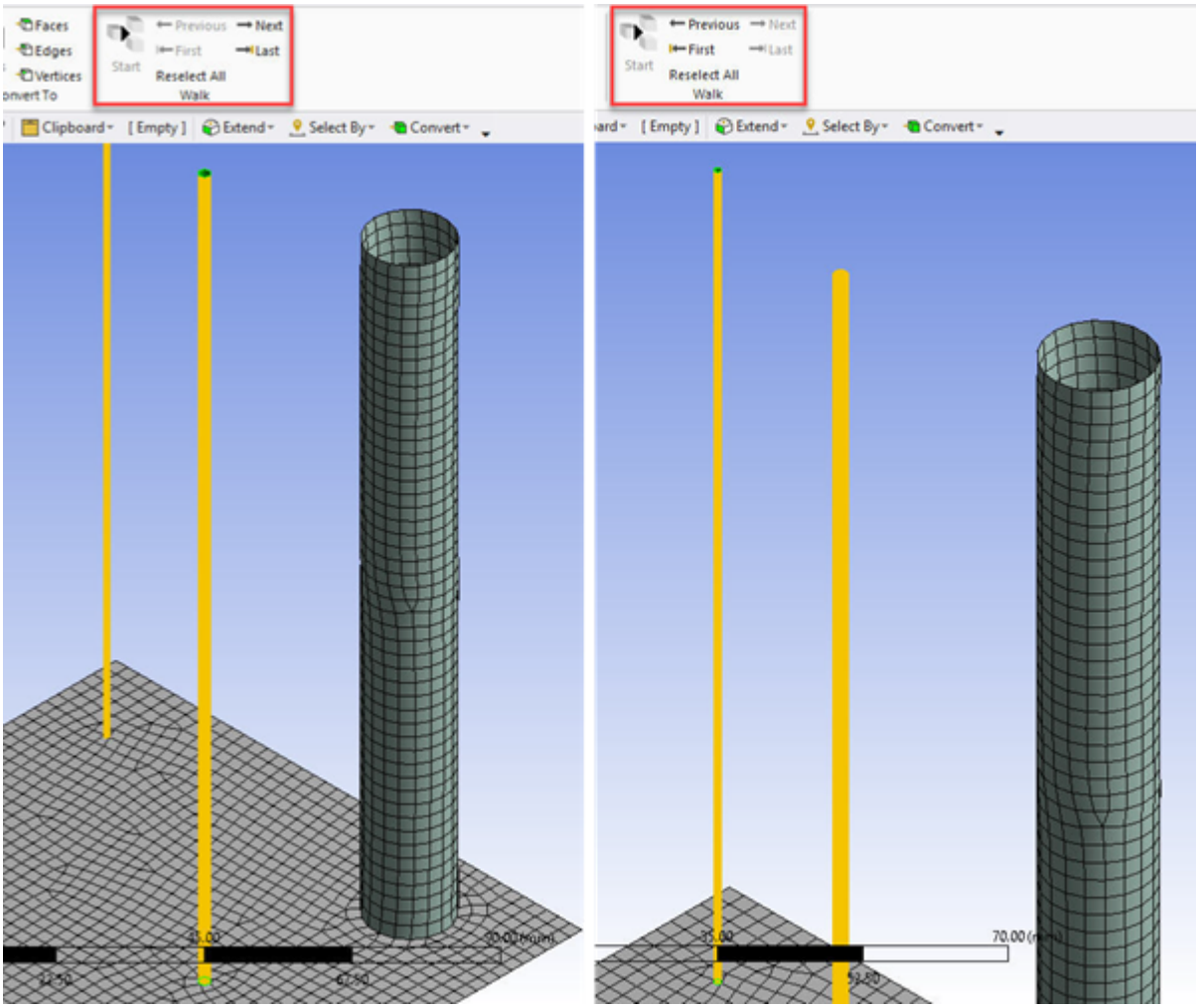


You can click **Selections** tab in the ribbon and click **Start** in **Walk** group to walk through the error locations. You can use **Next** to walk to the next location. You can click **Previous** to go back to the previous error location.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can view the unconnected faces after meshing. To view the unconnected faces, right-click the Geometry window. Select **Diagnostics > Find Unconnected Faces**. The unconnected faces get highlighted in the model.

When entities scoped to **Named Selection** with **Protected** set to **Yes** are modified while performing **Automatic (PrimeMesh)**, you will receive a warning message.

Sizing Control

The Sizing control sets:

- The element size for a selected body, face, or edge.
- The number of divisions along an edge.
- The element size within a user-defined "sphere of influence" that can include a selected body, face, edge, or vertex. This control is recommended for local mesh sizing. The control must also be attached to a coordinate system if it is to be scoped to anything other than a vertex.
- The element size within a user-defined "body of influence." The body of influence will influence the mesh of the body to which it is scoped, but the body of influence itself will not be meshed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The scale factor for a selected body, face, or edge. The scale factor enables you to define the local element, minimum, and defeature sizes as factors of the global element size.
- The minimum mesh sizing used for a selected body, face, or edge. This setting overrides the default global sizing.

The Sizing control is described in the following sections:

Notes on Element Sizing
Applying a Local Sizing Control
Descriptions of Local Sizing Control Options

Notes on Element Sizing

Remember the following notes when using the Sizing control:

- Visual aids are available to assist you. When you pick an edge, the edge length is displayed. A circle is displayed adjacent to the cursor whose diameter indicates the current setting in the **Element Size** field. The scale ruler is displayed below the graphic and provides a good estimate of the scale of the model. Also for edge sizing, if you specify a bias, and if you set **Element Size** to a value other than **Default**, the size control will be displayed graphically with the initial mesh density (including any specified bias) in the **Geometry** window.
- **When Applying Sizes to Faces:** Faces adjacent to a face that has a scoped size control applied to it respect the source as part of the sizing control. Meshes on the adjacent faces will transition smoothly to the size on the scoped face. When size controls that have differing sizes are on adjacent entities, the adjacent topology receives the smallest size.
- **When Applying Sizes to Edges:** If possible, the meshing algorithm places the requested number of divisions on the specified edge. Otherwise, the algorithm adjusts the number to allow a successful mesh generation. When **Behavior** is set to Hard, the number of divisions is given a higher priority and the mesher may fail rather than adjust the number of divisions. In case of failure, inspect the messages for more information as to what constraints may be causing it.

  Note:

  – In **MultiZone**, a closed edge is represented by more than one blocking topological edges. Therefore, user-specified edge node distribution size may not be respected. However, the number of intervals will still be respected.

  – For split curves in rotated edge association, only interval count, not distribution, will be respected.

- **When Sweeping (p. 399):** Consider the following when applying size controls to source and target geometry:

  – If your sizing controls are scoped to either the source or target face, the mesher will transfer the size control to the opposite face. If you have a size control on both faces, the size on one of the faces will be used. That face is automatically determined by the software. However the size on the edges of the target face will not be affected if no sizes are explicitly defined on these edges.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Edge sizing applied to a target face is respected only if **Behavior** is **Hard**.

– If you have a sphere of influence on a possible source or target face, the face with the most spheres will be chosen as the source face. The edge mesh of the source face affected by the sphere of influence will not affect the target face. This may prevent the model from sweeping with acceptable element quality. To avoid this, place the sphere of influence on the edges of both the source and target face.

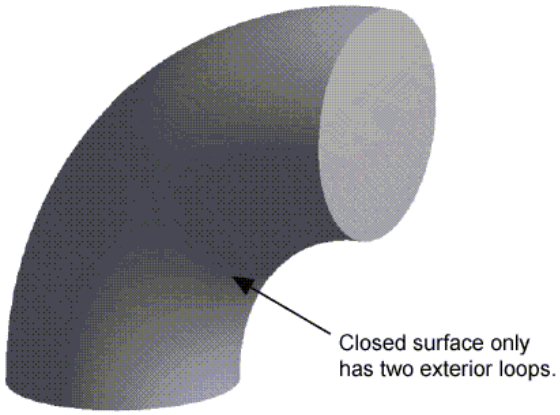
– Applying sizes, regardless of type (that is, size, number of divisions, sphere of influence), to the edges of possible source and target faces will only affect the faces that use these edges.

If you want to control a side area, the problem must be properly constrained such that the interval assignment does not override your size control. The divisions on the edge may decrease in order to make the body sweepable. When using a meshing process other than swept meshing, the divisions can only increase. When applying a size to a part that is sweepable, the resulting mesh may have fewer divisions on the edge than specified due to the interval assignment logic of the sweepers.

When sweeping a model, if you use the **Sphere of Influence** sizing control and the sphere is not touching the edges of a side area or is totally enclosed in the body, the sphere will have no effect.

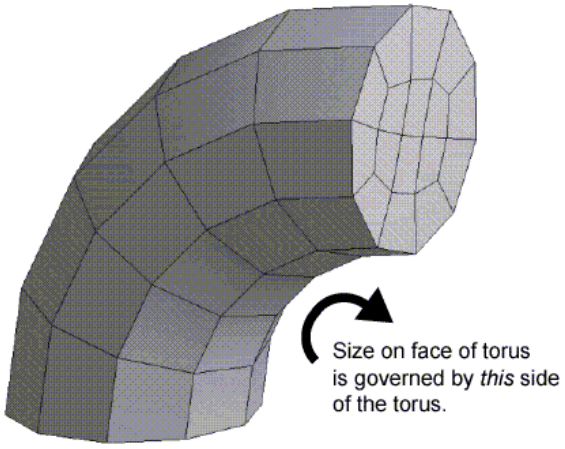
When sweeping a closed torus (shown below) with an applied size on the face of the torus, the number of divisions that will result on the torus is governed by the arc length between the caps of the surface on the inside of the torus.

Figure 103: Sweeping a Closed Torus



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 104: Resulting Mesh for Closed Torus



- Using the **Sphere of Influence** sizing control may not have any effect on the generated mesh if the control is scoped to the Body of a Line Body.
- In general, users are discouraged from defining a body of influence and a sphere of influence such that the regions of influence overlap. In cases where elements fall within overlapping bodies/spheres of influence, elements will be created using the **Sphere of Influence** sizing that appears lowest in the Tree.
- Regardless of the value for the sizing control you set, other factors such as edge and face curvature and the proximity of small features may override the effect of the sizing control.
- When using mapped face controls, or **Sweep** or **MultiZone** parallel edge assignments are handled automatically for mapped faces. That is, for a mapped face, there are two sets of parallel edges. If you increase or decrease the sizing on one edge, the same increase or decrease occurs automatically on the other edge to ensure a mapped mesh is possible. If a model contains a row of mapped faces (such as the sides of a box), you can set a number of elements on one edge and the same number of elements will be forced on all side/parallel edges. Setting **Behavior** to **Hard** will give that edge priority. In case of conflict between two edges that should be parallel but have different hard sizing controls assigned to them, the software will either fail and print a message about the conflict, or a free mesh will be given for that face.
- When using **MultiZone**:
  - If you scope bias settings to parallel edges, and the sizings create conflicts, they are applied according to the following priority:
    1. Double bias edge (highest priority)
    2. Single bias edge
    3. No bias edge
  - Setting **Behavior** to **Hard** on an edge gives that edge a higher priority. That is, a hard edge sizing takes priority over a soft edge sizing, but among sizing controls that are hard or soft, the order of priority is double, single, then no biasing.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you apply a local **Sizing** control (p. 296) to a solid body with a **Method** control set to **Hex Dominant** (p. 233) or **Sweep** (p. 235), or to a sheet body with a **Method** control set to **Quadrilateral Dominant** (p. 282), a near uniform quadrilateral mesh will result on all affected faces on a body meshed with **Hex Dominant**, on the source face meshed with **Sweep**, and on all affected faces meshed with **Quadrilateral Dominant**. To obtain even more of a uniform quadrilateral mesh, set the **Behavior** of the **Sizing** control to **Hard**.
- If several sizing controls are attached to the same edge, face, or body, the last control is applied. If a sizing control is placed on an edge and then another is placed on a face or body that contains that edge, the edge sizing takes precedence over the face or part sizing.
- If you have adjusted the element size, then changed length units in a CATIA or ACIS model, when you choose **Update** or **Clear Generated Data** at a Model or Project node in the Tree Outline, you may need to re-adjust the element size. The sizing control does not automatically re-adjust to match this situation.
- When using **MultiZone Quad/Tri** (p. 284) and **Surface Mesh Method** is set to **Uniform**, the **Element Size** will take priority over local face or body sizings unless the specified face or body size is smaller than the **Element Size**. This means you can use face or body sizing to obtain a finer mesh, but not to obtain a coarser mesh.
- For a table summarizing the behaviors of local sizing controls when used with various mesh methods, refer to Interactions Between Mesh Methods and Mesh Controls (p. 478).

Applying a Local Sizing Control

You can set the local sizing on a body, face, edge, or vertex.

1. To access the **Sizing Control**:

   Right-click a body, face, edge, or vertex, and then select **Insert** > **Sizing**.

   Or

   On the Tree Outline, right-click the **Mesh** object and select click **Insert** > **Sizing**.

   Or

   On the Tree Outline, right-click Mesh and click **Insert** > **Sizing**.

2. If necessary, in the Details view, define the scope of the selection:

To apply local sizing to...	Do this...
A geometry selection	1. Click **Scoping Method** and select **Geometry Selection**. 2. Select a body, face, edge, or vertex. 3. In the **Geometry** field, click **Apply**.
A named selection	1. Click **Scoping Method** and select **Named Selection**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To apply local sizing to...	Do this...
	2. Select a **Named Selection**.

3. Do one of the following, depending on what entities you selected and how you want to control the mesh sizing:

If you selected...	And you want to...	Do this...
Bodies, faces, or edges	Specify an element size to control mesh sizing	1. In the **Type** field, select **Element Size**. 2. Define the **Element Size** (p. 303).
Bodies, faces, or edges	Specify a scale factor to define the local minimum and defeature sizes as factors of the global element size	1. In the **Type** field, select **Factor of Global Size**. 2. Define the **Factor of Global Size** (p. 306).
Bodies, faces, edges, or vertices	Apply mesh sizing within the confines of a sphere	1. In the **Type** field, select **Sphere of Influence**. 2. Define the **Sphere of Influence** (p. 304).
Bodies	Apply mesh sizing by specifying an element size and controlling the mesh density based on neighboring bodies of influence	1. In the **Type** field, select **Body of Influence**. 2. Define the **Bodies of Influence** (p. 305). 3. Define the **Element Size** (p. 303).
Edges	Control mesh sizing according to a discrete number of divisions along the edge	1. In the **Type** field, select **Number of Divisions**. 2. Define the **Number of Divisions** (p. 306).

4. If necessary, define any advanced options.

   The values of the advanced options are set by default. You can change these values to apply greater control over the local sizing on an entity.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more information on the advanced options, see Descriptions of Local Sizing Control Options (p. 302).

Note:

If **Use Adaptive Sizing** is set to **No**, any local size applied to an entity is also applied to all lower topology entities.

Descriptions of Local Sizing Control Options

Use the options described below to further define local mesh sizing. The choices that are available depend on the selected topology, physics preference, and as noted in the individual option descriptions. You can set the following local sizing options:

Definition Options:

- Element Size (p. 303)
- Sphere of Influence (p. 304)
- Body of Influence (p. 305)
- Factor of Global Size (p. 306)
- Number of Divisions (p. 306)

Advanced Options:

- Defeature Size (p. 307)
- Defeature Size Scale (p. 307)
- Influence Volume (p. 307)
- Affected Distance (p. 307)
- Capture Curvature (p. 308)
- Capture Proximity (p. 308)
- Growth Rate (p. 309)
- Local Min Size (p. 309)
- Curvature Min Size Scale (p. 309)
- Curvature Normal Angle (p. 309)
- Proximity Min Size (p. 309)
- Proximity Min Size Scale (p. 310)
- Proximity Gap Factor (p. 310)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

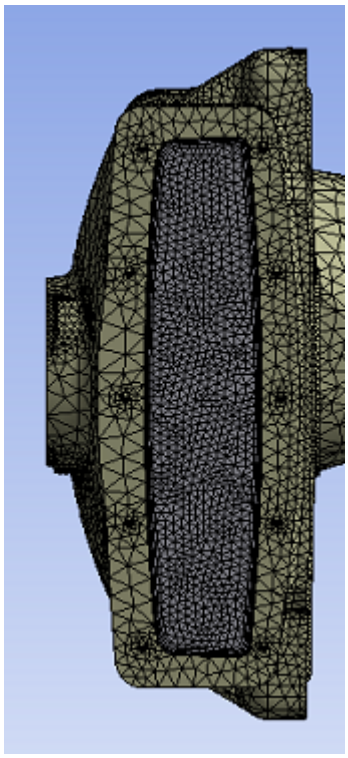
- Proximity Size Function Sources (p. 310)
- Behavior (p. 310)
- Bias Type and Bias Option (p. 310)

Element Size

If you selected a body, face, or edge, **Element Size** (default) is one of the available options in the **Type** field. Enter a positive value (decimals are allowed) in this field. Smaller values generate more divisions. A value of "0" instructs the sizing control to use its defaults.

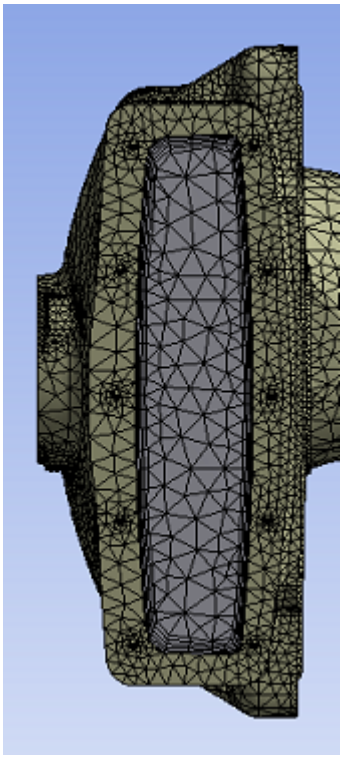
The following series of figures shows the effect of the **Element Size** option applied to the central face at 5mm.

Element Size set to 5mm.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element Size default.



Additional details about the **Element Size** option differs depending on the sizing options set. The description of **Element Size** is as follows:

- For an *edge*, **Element Size** is the maximum size on the edge. It takes priority over the global Max Size (p. 110), and any face or body sizing control that includes the edge(s). If two edge sizing controls are attached to the same edge, the latter size control takes priority.
- For a *face*, **Element Size** is the maximum size on the face, which is then propagated down to the edges (unless a more local edge control is assigned). The value of **Element Size** takes priority over Max Size (p. 110) and any body size control that includes the face(s). If two face sizing controls are attached to the same face, the latter size control takes priority.
- For a *body*, **Element Size** is the maximum size on the body, *and* the maximum sizes on faces and edges of the body (unless a more local face or edge size control is assigned). The value of **Element Size** takes priority over the global Max Size (p. 110). If the body size behavior is **Hard** (p. 310), the value of **Element Size** also takes priority over the global Max Size (p. 110).

Sphere of Influence

The **Sphere of Influence** option is available in the **Type** field after you select an entity such as a body, face, edge, or vertex.

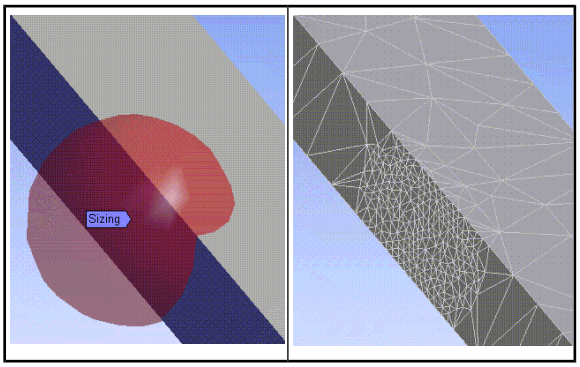
If the **Sphere of Influence** is scoped to a body or vertex, the **Sphere of Influence** affects the entire body regardless of sizing options being used. If the **Sphere of Influence** is scoped to a face or edge and **Use Adaptive Sizing** is set to **Yes**, the **Sphere of Influence** only affects the face(s) or edge(s) that are scoped to the control and the transition away from those entities. If the **Sphere of Influence** is scoped to a face or edge and any other sizing option is used, the **Sphere of Influence** will affect the whole body (not just scoped face(s) or edge(s)).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

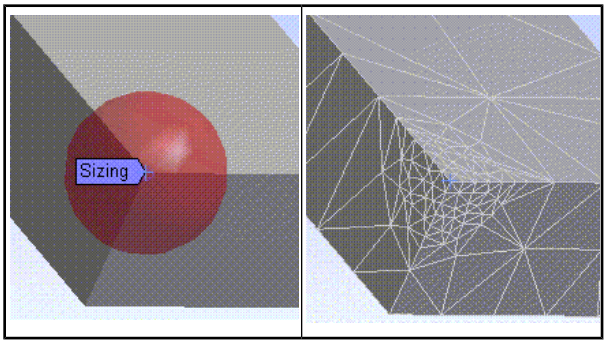
Although the **Behavior** option is not available for **Sphere of Influence**, **Sphere of Influence** behaves as a **Hard** setting. That is, in the vicinity of a **Sphere of Influence**, the **Sphere of Influence** sizing overrides pre-existing sizing information regardless of whether the pre-existing sizes are larger or smaller than the **Sphere of Influence** sizing. This is in contrast to the **Body of Influence** (p. 305) option, which behaves as a **Soft** setting.

For bodies, faces, and edges, **Sphere of Influence** allows you to apply mesh sizing within the confines of a sphere in space that you define as follows:

1. Create a local coordinate system whose origin you intend to be the center of the sphere.
2. Select this coordinate system in the **Sphere Center** field.
3. Enter the radius of the sphere in the **Sphere Radius** field.
4. Enter a value in the **Element Size** field. The element size will be applied to all topologies within the confines of the sphere. For example, if you are applying the element size to a face, the size will also be applied to the edges of that face, and to the vertices of those edges, but only within the confines of the sphere. An example is shown below.



If you selected a vertex, the *only* option available in the **Type** field is **Sphere of Influence**. The description is the same as presented above except that the center of the sphere *is* the vertex. There is no need to create or use a local coordinate system to define the center of the sphere. After applying element size to a vertex using **Sphere of Influence**, the element size is applied to all topologies connected to that vertex, such as all edges and faces containing that vertex, if they fall within the sphere. An example is shown below.



Body of Influence

The **Body of Influence** option is available in the **Type** field if you selected a body and **Use Adaptive Sizing** is set to **No**. Using this option, you can set one body as a source of another body (that is, a

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Body of Influence). The **Body of Influence** will influence the mesh density of the body that it is scoped to, but it will not be a part of the model geometry nor will it be meshed. **Body of Influence** bodies are noted in the Details View of each prototype.

Although the **Behavior** option is not available for **Body of Influence**, **Body of Influence** behaves as a **Soft** setting. That is, in the vicinity of a **Body of Influence**, the specified **Body of Influence** sizing must be larger than the smallest Curvature Min Size (p. 113)/Proximity Min Size (p. 115) *and* smaller than the global Max Size (p. 110) to have an effect on the mesh size distribution. The specified **Body of Influence** sizing imposes a local maximum size on all elements that are inside the boundary of the body. This is in contrast to the **Sphere of Influence** (p. 304) option, which behaves as a **Hard** setting.

Remember the following notes when using **Body of Influence**:

- If your source body is a sphere, it is best for you to use the **Sphere of Influence** option instead of **Body of Influence**. **Body of Influence** is intended for non-spherical bodies.
- In general, users are discouraged from defining a **Body of Influence** and a **Sphere of Influence** such that the regions of influence overlap. In cases where elements fall within overlapping bodies/spheres of influence, elements will be created using the **Sphere of Influence** sizing that appears lowest in the Tree.
- You cannot apply loads, mesh controls, etc. on bodies of influence. Bodies of influence are used only to influence the sizing controls and therefore only sizing attributes can be applied to them.
- You can suppress and unsuppress bodies of influence.
- If you are using **Body of Influence** with **Match Control**, be aware that the body of influence will not be copied from one matched entity to the other. As a workaround, you can copy the body in the DesignModeler application and use both bodies as your source.

Factor of Global Size

If you select a body, face, or edge, **Factor of Global Size** is one of the available options in the **Type** field (unless **Use Adaptive Sizing** is set to **Yes**). Choose **Factor of Global Size** and enter a value for **Element Size Factor** to define the local minimum and defeature sizes as factors of the global element size. If local sizings are defined as a factor of global size and you modify the **Element Size** (p. 102), the local sizings are recalculated.

Number of Divisions

If you select an edge, the options available in the **Type** field are **Element Size** and **Sphere of Influence**, along with the **Number of Divisions** option. Choosing **Number of Divisions** and entering a value in the **Number of Divisions** field is an alternative to choosing **Element Size** if you are interested in having the mesh be sized according to a discrete number of divisions along an edge. If you set **Number of Divisions** to a value greater than 1000, the number of divisions will not be drawn on the edge in the **Geometry** window.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defeature Size

For body sizing and face sizing controls, sets the local tolerance for defeaturing. Features smaller than or equal to this value will be removed when the mesh is generated. You can specify any value greater than 0.0.

If the local sizing is **Uniform**, the default local **Defeature Size** is set to the smaller of the following two values:

- Global **Defeature Size**
- 50% of the value of the local **Element Size**

For all other local sizing controls, the default local **Defeature Size** is set to the smaller of the following two values:

- Global **Defeature Size**
- 50% of the value of the local **Curvature Min Size** or **Proximity Min Size** (whichever is smaller)

For more information about setting the defeature size, see Defeature Size (p. 111).

Defeature Size Scale

Specify the scale factor for the defeature size. The default values comes from the scale factors for **Mechanical Defeature Size Factor** and **CFD Defeature Size Factor** depending on the physics preference (**CFD** physics preference uses the **CFD Defeature Size Factor**, other physics preferences use the **Mechanical Defeature Size Factor**). The default value or the value that you specify is multiplied by the global element size to determine the local defeature size.

Influence Volume

When **Element Size** is selected, the **Influence Volume** appears with **Yes** or **No** option. The default option is **No**. If you select **Yes**, the **Affected Distance** field appears. When influencing the volume, the face size control generates spheres of influence internally by automatic determination of radius and size based on the **Affected Distance**.

Note:

- Cartesian method does not support **Influence Volume** meshing.
- **Layered Tetrahedrons Method** does not support **Influence Volume** meshing.
- When **Adaptive Sizing** is set to **Yes**, **Influence Volume** option does not have effect.

Affected Distance

When **Influence Volume** is set as **Yes**, the **Affected Distance** option appears. You can enter the distance up to which the defined **Element Size** will affect the volume mesh.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Capture Curvature and Capture Proximity

If a global sizing option is defined for the model, then you can set a local sizing on a body, face, or edge to further refine the local entity's sizing. When you set a local sizing on an entity, the local sizing options default to the same values that are defined for the global size function. You can change any of these values, and the local values will take precedence over the global values when the mesh is generated.

For example, if **Capture Curvature** and/or **Capture Proximity** are selected, and the local sizing is **Uniform** (or vice versa), the local sizing option takes precedence.

Example 4: Setting a Local Sizing Option

If the global sizing option is set to **Curvature** with a 20° **Curvature Normal Angle**, and you set a local sizing of **Curvature** on a face with a 60° **Curvature Normal Angle**, the local **Curvature Normal Angle** will take precedence when the mesh is generated. In this example, the face with the local sizing control will use a **Curvature Normal Angle** of 60° even though the global value is 20°.

If using **Nonlinear Mechanical** or **CFD** physics preference, or when **Use Adaptive Sizing** is set to **No**; the following options are available and work as follows:

- **Capture Curvature** = **Yes** turns on the Curvature-based Sizing (p. 107).
- **Capture Proximity** = **Yes** turns on the Proximity-based Sizing (p. 107).
- If **Capture Curvature** and **Capture Proximity** are both set to **Yes**, then both Curvature-based Sizing (p. 107) and Proximity-based Sizing (p. 107) will be used.
- If **Capture Curvature** and **Capture Proximity** are both set to **No**, then Uniform (p. 108) sizing will be used.

Note:

Local sizing limitations include:

- If **Use Adaptive Sizing** is set to **Yes**, then you cannot set a local size function.
- If the global sizing option is set to **Capture Proximity** or both **Capture Proximity** and **Capture Curvature**, and the local sizing option is set to **Capture Curvature** or both **Capture Proximity** and **Capture Curvature** (or vice versa), the mesh on the entity is generated as though both **Capture Proximity** and **Capture Curvature** are set locally.
- If the global sizing option is set to **Capture Proximity**, and the local sizing option is set to **Capture Curvature** (or vice versa), the option that has the smallest size specified for it (either **Proximity Gap** for the **Proximity** sizing, or **Curvature Normal Angle** for the **Curvature** sizing) takes precedence.
- The local **Proximity** sizing is not supported for sheet models. If both the local **Proximity** and **Curvature** sizing is defined on a sheet model, only the **Curvature** sizing takes effect.
- Local proximity sizing is not supported for edge sizing controls. To set a local edge sizing control, you must choose either **Capture Curvature** or **Uniform** (both **Capture Curvature** and **Capture Proximity** set to **No**).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Growth Rate

You may specify a growth rate for the scoped soft size of an entity (*body, face, or edge only*). The description of the scoped **Growth Rate** is the same as that of the global growth rate (p. 110) that you can set in the Details View when the **Mesh** object is selected in the Tree Outline. However, the growth rate you specify for a scoped entity *must always be smaller than or equal to* the specified global growth rate (p. 110). **Growth Rate** is not available for **Sphere of Influence**. Specifying a growth rate for a face or body affects the growth on the face or on the boundary of the body, and its effect continues outside of the scoped entity as well.

Local Min Size

You can set **Local Min Size** to specify a value that takes priority over the global **Min Size** when you select a *body*, *face*, or *edge*. If **Local Min Size** is set to **Default**, the mesher uses either the global **Min Size** or the local **Element Size** (if defined), whichever is smaller. This setting is available when **Capture Curvature** is set to **Yes**, and is useful when you want to refine the mesh based on curvature. When you set the **Local Min Size**, the mesher refines from the **Element Size** to the **Local Min Size** in curved areas, but retains the **Element Size** for flat areas.

Note:

Local Min Size may not be respected by the Sweeping or MultiZone Methods due to the interval assignment used to generate structured meshes. The min size is used on source faces, but not necessarily on side faces.

Curvature Min Size Scale

Specify the scale factor for the curvature minimum size. The value that you specify is multiplied by the global element size to determine the local curvature minimum size. The default is determined by the **Mechanical Min Size Factor** or **CFD Min Size Factor** option (p. 392) in the Options dialog box, depending on the physics preference.

Curvature Normal Angle

You may specify **Curvature Normal Angle** for a scoped entity (*body, face, or edge only*). **Curvature Normal Angle** is the maximum allowable angle that one element edge is allowed to span given a particular geometry curvature. Available only when **Capture Curvature** is set to **Yes**. You can specify a value from 0 to 180 degrees to override the global setting (where a value of 0 resets the option to its default). The default for the global setting is calculated based on the Physics Preference (p. 96).

For more information, see Curvature Normal Angle (p. 114).

Proximity Min Size

This option allows you to specify a minimum size to be used in a local proximity sizing calculation, in addition to the **Local Min Size** (p. 309). By default, **Proximity Min Size** is set equal to the default of **Local Min Size**. You can accept the default or specify a value greater than 0. For more information, see Proximity Min Size (p. 115).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Proximity Min Size Scale

Specify the scale factor for the proximity minimum size. The value that you specify is multiplied by the global element size to determine the local proximity minimum size. The default is determined by the **Mechanical Min Size Factor** or **CFD Min Size Factor** option (p. 392) in the Options dialog box, depending on the physics preference.

Proximity Gap Factor

Specify the minimum number of layers of elements to be generated in gaps when **Capture Proximity** is set to **Yes**.

Proximity Size Function Sources

This option is available if **Capture Proximity** is set to **Yes**. It determines whether regions of proximity between faces and/or edges are considered when proximity sizing calculations are performed.

For more information, see Proximity Size Function Sources (p. 115).

Behavior

If the global option **Use Adaptive Sizing** is **Yes**, or the local sizing is Uniform (both **Capture Curvature** and **Capture Proximity** set to **No**), the **Behavior** option is available for bodies, faces, and edges.

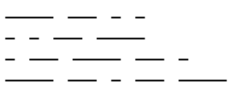
You can specify either **Soft** (default) or **Hard**, but the effect depends on the type of mesh being generated. Typically, **Hard** results in a stricter size setting than **Soft**. For example, with a tri or tet mesh, the **Hard** setting would not allow as much transition from one element size to another element size due to influences of sizing on neighboring objects. With a **Hard** setting, the element size defined would be maintained for the object to which the setting is scoped. With a **Soft** setting, the element size defined may be modified in order to respect other size settings in the vicinity of the object to which the setting is scoped.

You should take care when applying **Hard** sizes. With a mapped quad/tri or swept mesh, a **Hard** setting forces the interval edge assignment on the object to be met, and if the mesher cannot achieve the setting, the mesh could fail.

Bias Type and Bias Option

For edges only, use **Bias Type** to adjust the spacing ratio of nodes on an edge. This feature is useful for any engineering problem where nodes need to be clustered on an edge or group of edges, or if there is a need to bias the mapped mesh of a face towards a specific direction with respect to the edges of the face. **Bias Type** can be used with all meshers except the **Patch Independent Tetrahedron** method. To use **Bias Type**, choose one of the four pre-determined patterned options depicted pictorially from the **Bias Type** drop-down menu:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

The **MultiZone** meshing method respects all **Bias Type** options. The nodes generally follow the distribution. However, some optimization can be done to improve transition with other mesh, or to improve the quality. Setting the **Behavior** to **Hard** will give higher priority to the bias settings defined than the optimization.

Then specify a **Bias Option**. The drop-down menu enables you to choose **No Bias**, **Bias Factor**, or **Smooth Transition**:

- **Bias Factor** is defined as the ratio of the largest edge to the smallest edge. To set the Bias Factor, choose **Bias Factor** from **Bias Option** and enter a value into the field to define the ratio.
- **Smooth Transition** is defined as Growth Rate = Bias Factor^(1/(n-1)), where *n* is the number of divisions. To define Smooth Transition, choose **Smooth Transition** from **Bias Option**, then enter a value for **Growth Rate**.

Note:

If **Behavior** is set to **Hard**, then the number of divisions and the bias cannot be changed by the mesher. If **Behavior** is set to **Soft**, then the edge divisions can be changed but the edge will be initially meshed with the specified **Bias Factor**.

If you want to select multiple edges to apply sizing, but some of the edges do not have the same orientation, you can use the **Reverse Bias** option to manually select all of the edges. **Reverse Bias** is available when the control is applied to the edges, defined as an **Element Size** or **Number of Divisions**, and has a bias towards one of the vertices ("- -- --- ----" or "---- --- -- -").

To apply **Reverse Bias**, select a group of edges, then choose **Reverse Bias** and click **Apply**. Only edges that are part of the main scoping of the control are applied. All others are ignored.

To undo **Reverse Bias**, select an edge that is not part of the main scoping. Then choose **Reverse Bias** and click **Apply**.

Contact Sizing Control

Contact Sizing creates elements of relatively the same size on bodies from the faces of a face to face or face to edge contact region. This control generates spheres of influence (p. 304) internally with automatic determination of radius and size (if **Resolution** is selected for **Type**). You may want to apply a method control (p. 202) on sweepable bodies to force the elements to be tetrahedron in the case where the sweeper is not providing enough local sizing near your contact region. Your swept mesh may be quite dense if the contact size is small on the source and target faces of the body. You may also see very little effect on swept bodies in the case where a contact size is applied to a very small region of a large source face.

To access the **Contact Sizing**,

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

On the Tree Outline, right-click **Mesh** and click **Insert > Contact Sizing**.

Or

On the Tree Outline, click **Mesh** and click **Method** in the **Mesh Context** tab on the Ribbon.

Or

Right-click a body, face, edge, or vertex, and then select **Insert > Contact Sizing**.

Or

On the Tree Outline, drag a **Contact Region** or **Contacts** folder from the **Connections** to **Mesh**.

Or

On the Tree Outline, right-click **Contact Region** from the **Connections** and click **Create > Contact Sizing**.

The **Contact Region** Details view has the following options:

Scope

- **Contact Region**: Allows you to scope the contact region.

Definition

- **Suppressed**: Allows you to suppress the selected control. The default value is **No**. When **Suppressed** is set to **Yes**, the **Active** field displays the status of the selected control. The **Active** field is read-only.
- **Type**: Allows you to select the type of contact meshing. The default value is. The available options are:
  - **Resolution**: Allows you to specify a value for relative size. The default value is **4**. You can parametrize **Resolution**.
  - **Element Size**: Specify an element size to control mesh sizing. Enter a positive value (decimals are allowed) in this field. Smaller values generate more divisions. When you select **Element Size**, the **Element Size**options allow you to specify the value for absolute size. The default value is **0 mm**. When you specify a value different from the global Defeature Size, it overrides the global value. You can parametrize **Element Size**.

Note:

- You can select two bodies in the Geometry window and use the **Go To** > Contact Sizing Common to Selected Bodies option to identify any contact sizing controls that exist between the two bodies. This feature provides an easy way for you to delete the common controls.
- Because **Contact Sizing** objects cannot be duplicated, they cannot be used as template objects for the Object Generator.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Refinement Control

Refinement controls specify the maximum number of times you want an initial mesh to be refined. You can specify refinement controls for faces, edges, and vertices.

To access the **Refinement Control**,

On the Tree Outline, right-click Mesh and click **Insert** > **Refinement**.

Or

On the Tree Outline, click **Mesh** and click **Refinement** in the **Mesh Context** tab on the Ribbon.

Or

Right-click a face, edge, or vertex, and then select **Insert** > **Refinement**

The **Refinement** Details View has the following options:

Scope

- **Scoping Method**: Allows you to scope geometry or named selection. The default value is **Geometry Selection**.
  - **Geometry Selection**: Allow you to scope the geometry bodies. When you select **Geometry Selection**, the **Geometry** allows you to select the geometry from the Geometry window.
  - **Named Selection**: Allow you to scope bodies grouped under a named selection.

Definition

- **Suppressed**: Allows you to suppress the method control. The default value is **No**. When **Suppressed** is set to **Yes**, the **Active** provides the number of suppressed parts and is a read-only option.
- **Refinement**: Specify the maximum number of times you want an initial mesh to be refined. The default value is **1**. You can parametrize **Refinement**. When you attach several entities, the last control applied takes precedence.

Some refinement controls can override or affect other refinement controls that have been applied to connected topology. A face refinement control overrides a refinement control on any of the face's edges or vertices. An edge refinement control overrides a refinement control on either of the edge's vertices. Basically, a refinement control will lower the value of an overridden control by its own value.

For example, consider a face refinement control with a refinement value of 1, where one of the face's edges has a refinement control with a value of 2, and one of the edge's vertices has a refinement control with a value of 2. In this example, the face refinement control reduces the value of the edge refinement control by 1, and it also reduces the value of the vertex refinement control by 1. The edge refinement

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

control now has a value of 1, so it reduces the vertex's refinement control by 1. Now the vertex refinement control has a value of zero, which essentially means the refinement control has no effect.

Note:

- Refinement controls are not available for the **MultiZone**, **Patch Independent Tetra**, **Cartesian**, **Layer Tetrahedron** or **MultiZone Quad/Tri** mesh methods. If you are using the **Automatic Method** (p. 206) and you have enabled the Use MultiZone for Sweepable Bodies (p. 389) option, refinement controls on sweepable bodies behave similarly to how they behave when the **Sweep** mesh method is used.
- In the following scenarios, refinement controls are automatically suppressed:
  - When automatic inflation (p. 163) (either Program Controlled (p. 164) or All Faces in Chosen Named Selection (p. 165)) is used with refinement in the same model.
  - When local inflation (p. 345) is used with refinement in the same body or in the same part.
- If you apply a refinement control to a part that was either swept meshed (p. 235) or hex dominant meshed (p. 233), and then you delete the refinement control, the intermediate tetrahedral mesh is retained unless you invalidate the state of the part (for example, by clearing the database). An intermediate tetrahedral mesh is created when you try to refine non-tetrahedral solid elements.
- Refinement controls are not supported on shared faces between solid bodies and sheet bodies in a multibody part.
- Refinement controls are not supported for Mixed Order Meshing (p. 461).
- Special processing of refinement operations occurs when you use the **Mesh** worksheet to create a selective mesh history. Refer to Using the Mesh Worksheet to Create a Selective Meshing History (p. 448) for details.

Face Meshing Control

Face meshing controls enable you to generate a free or mapped mesh on selected faces. The Meshing application determines a suitable number of divisions for the edges on the boundary face automatically. If you specify the number of divisions on the edge with a **Sizing** control, the Meshing application attempts to enforce those divisions.

To set the **Face Meshing** controls, highlight **Mesh** in the Tree Outline, and right-click to view the menu. Select **Insert> Face Meshing**. You can also click **Mesh** in the Tree Outline, and select the **Mesh Control** Context Toolbar, then select **Face Meshing** from the drop-down menu.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Definition>Mapping is set to **Yes** by default, also exposing **Constrain Boundary** (p. 315) and **Advanced** (p. 320) settings. If Mapping is set to **No**, the Mesher will perform a free mesh and the **Constrain Boundary** and **Advanced** settings are not available.

Note:

To assist you in defining face meshing controls, you can use the **Show Mappable Faces** (p. 542) feature to select all mappable faces automatically and highlight them in the **Geometry** window.

Mapped Face Meshing is supported for the following mesh methods:

Volume Meshing:

- Sweep (p. 235)
- Patch Conforming Tetrahedron (p. 208)
- Hex Dominant (p. 233)
- MultiZone (p. 242)

Surface Meshing:

- Quadrilateral Dominant (p. 282)
- Triangles (p. 284)
- MultiZone Quad/Tri (p. 284)
- Automatic (PrimeMesh) (p. 287)

Face Meshing control topics include:

Setting Basic Face Meshing Controls for Mapped Meshing
Understanding Advanced Mapped Face Meshing Controls
Notes on Face Meshing Controls for Mapped Meshing
Example Usage

Note:

For general information about applying mapped Face Meshing controls in combination with the various mesh method controls, refer to Meshing: Mesh Control Interaction Tables (p. 475).

Setting Basic Face Meshing Controls for Mapped Meshing

This section describes the steps for setting basic **Face Meshing** controls for mapped face meshing.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To set basic Face Meshing controls for mapped meshing:

1. Insert a mapped face meshing control by highlighting **Mesh** in the Tree and right-clicking to view the menu. Select **Insert > Face Meshing**.

   In the **Mapped Face Meshing Details** view, under**Definition, Mapped Mesh** is set to **Yes** by default.

2. Under the **Definition** , select **Method** as **Quadrilaterals** or **Triangles: Best Split**. The **Triangles: Best Split** option is available only for sheet models.

3. Specify the **Internal Number of Divisions** to provide number of divisions across annular regions or seamless cylinders. (The **Internal Number of Divisions** option is activated when the **Face Meshing** control **Definition** is set to **Mapped Mesh** and the control is scoped to faces made up of two loops.) The default value is **0**.

4. Select **Constrain Boundary** to specify whether you want to allow the mesher to split the boundary of a mapped mesh region to aid in meshing of adjacent faces. You can select **Yes** (constrain boundary; no splitting is allowed) or **No** (do not constrain boundary; splitting is allowed). The default is **No**. See Notes on Face Meshing Controls for Mapped Meshing (p. 327) for related information.

5. Right-click the **Mesh** object in the Tree and click **Generate Mesh**to generate the mesh.

Note:

When **Mapped Meshing** fails on a surface, the mesher automatically meshes the surface using **Aggressive Tri Reduction**.

To access the **Face Meshing**,

On the **Tree** Outline, right-click **Mesh** and click **Insert > Face Meshing**.

Or

On The **Tree** Outline, click **Mesh** and click **Face Meshing** in the **Mesh Context** tab on the **Ribbon**.

Or

Right-click the face and select **Insert > Face Meshing**.

Mapped Face Meshing Details view has the following options:

Scope

- **Scoping Method**: Allows you to scope geometry or named selection. The default value is **Geometry Selection**. The available options are:

  – **Geometry Selection**: Allow you to scope the geometry bodies. When you select **Geometry Selection**, the **Geometry** allows you to select the geometry from the **Geometry** window

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Named Selection**: Allow you to scope the named selection from the available named selections.

Definition

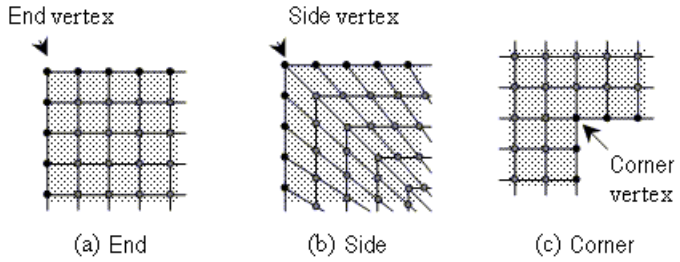
- **Suppressed**: Allows you to suppress the method control. The default value is **No**. When **Suppressed** is **Yes**, the**Active** provides the number of suppressed parts and is read-only.
- **Mapped Mesh**: Generates the structured mesh for the selected geometry faces when set to **Yes**. When **Mapped Mesh** is **No**, maps the faces automatically and generates quad dominant mesh.
- **Internal Number of Divisions**: Allows you to set the number of divisions across annular regions or seamless cylinders. The default value is **0**.

Note:

Internal Number of Divisions is available only when **Mapped Mesh** is **Yes**.

- **Method**: Allows you to select the shape of the elements used to fill the body. The default value is **Quadrilaterals**. When you set **Method** as **Quadrilaterals**, it creates the quadrilateral mesh on the face. When you set **Method** as **Triangles: Best Split**, it creates the triangular mesh on the face.
- **Constrain Boundary**: Specify whether you want to allow the mesher to split the boundary of a mapped mesh region to aid in meshing of adjacent faces. The default value is **No**.]]When **Constrain Boundary** is **Yes**, it allows you to split the boundary of a mapped mesh region. When **Constrain Boundary** is **No**, it allows you to split the boundary of a mapped mesh region.

Figure 108: Face Vertex Types



Advanced

- **Specific Sides**: Creates the face mesh such that three mesh element edges intersect at the vertex (see (b) in Figure 108: Face Vertex Types). The Meshing application treats the two topological edges that are adjacent to the vertex as a single edge for the purposes of meshing.
- **Specified Corners**: Creates the face mesh such that four mesh element edges intersect at the vertex (see (c) in Figure 108: Face Vertex Types). Assigning the Corner vertex type to a vertex whose adjacent edges form an angle less than 180° creates an unnecessarily bad quality mesh (although the mesh will be valid).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Specific Ends**: Creates the face mesh such that only two mesh element edges intersect at the vertex (see (a) in Figure 108: Face Vertex Types). As a result, the mapped and submapped face mesh patterns on both sides of the End vertex terminate at the edges adjacent to the vertex. Assigning the End vertex type to a vertex whose adjacent edges form an angle greater than 180° is likely to result in mesh failure.

  For detailed information regarding advanced features, refer Understanding Advanced Mapped Face Meshing Controls (p. 320)

- **MultiZone Semi-Structured**: Improves the mesh distribution, applies to only **MultiZone Meshing**. The default value is **No**.

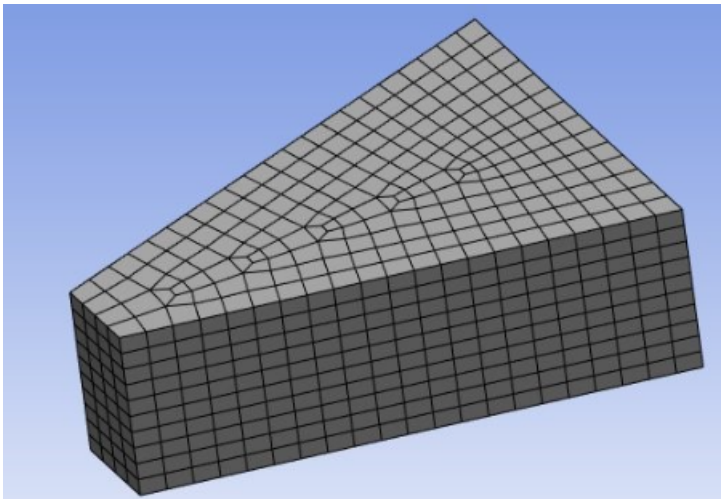
  When **MultiZone Semi-Structured** is **Yes**, the available option is:

  – **Transition Type**: Allows you to specify the type of transition. The available options are:

    → **Transition (Default)**: Transitions mapped mesh such that the two edges along the transition path have the same edge sizing and the other two edges should have a transitional size of the form:

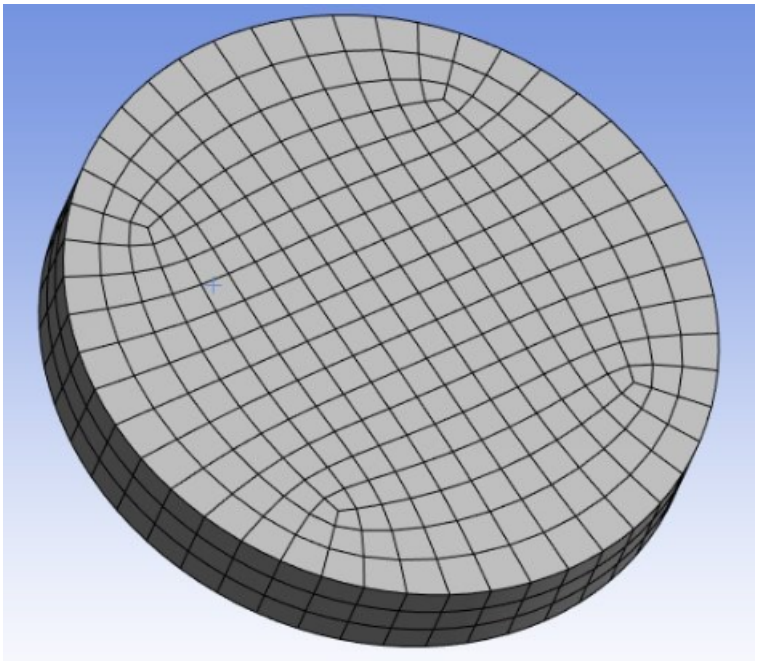
    edge1 = edge2 +/- 2*n

    where, n is the number of transitions.

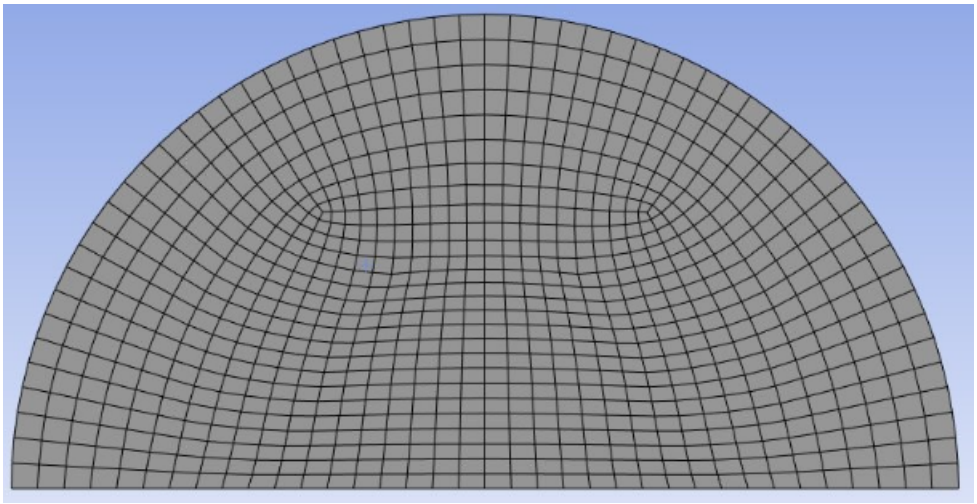
    

    → **Circle**: Creates an O-grid for circular or elliptical type face.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

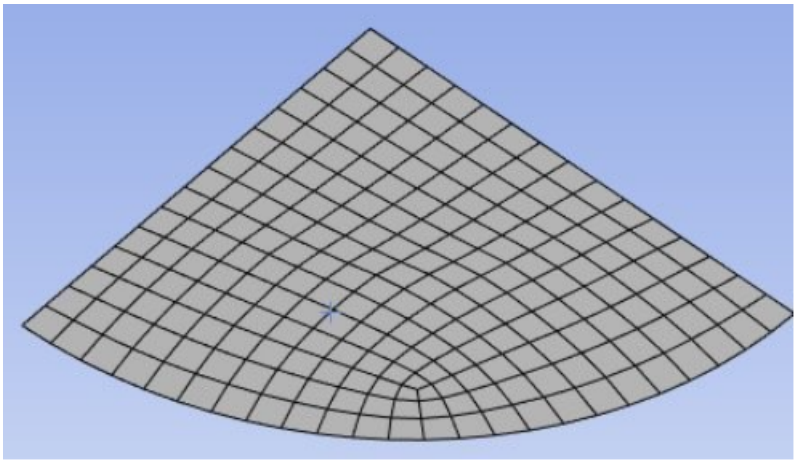


→ **Half-Circle**: Creates a C-grid for semi-circular or semi-elliptical type face.



→ **Quarter-Circle**: Creates an L-grid for quarter-circle or quarter-ellipse type faces.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Understanding Advanced Mapped Face Meshing Controls

When you apply advanced mapped Face Meshing controls for mapped meshing to a face, the Meshing application divides the face into one or more mappable regions and creates a mapped mesh in each region. Advanced mapped face meshing controls are subject to restrictions related to vertex types (p. 321) and restrictions related to edge mesh intervals (p. 322).

The *advanced* Face Meshing controls for mapped meshing are supported for the following mesh methods only:

Volume Meshing:

- Sweep (p. 235)
- Patch Conforming Tetrahedron (p. 208)
- Hex Dominant (p. 233)
- MultiZone (p. 242)

Surface Meshing:

- Quadrilateral Dominant (p. 282)
- Triangles (p. 284)
- MultiZone Quad/Tri (p. 284)

Advanced mapped Face Meshing topics include:

Restrictions Related to Vertex Types

Restrictions Related to Edge Mesh Intervals

Selecting Faces and Vertices

Effect of Vertex Type on Face Meshes

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Advanced Face Meshing Controls for Mapped Meshing

Note:

For general information on applying Face Meshing controls for mapped meshing in combination with the various mesh method controls, refer to Meshing: Mesh Control Interaction Tables (p. 475).

Restrictions Related to Vertex Types

To constitute a submappable face, a face must possess only End, Side, Corner, and Reversal vertices. In addition, the total number of End vertices, N_E, must satisfy the following equation:

$$N_E = 4 + N_C + 2N_R$$

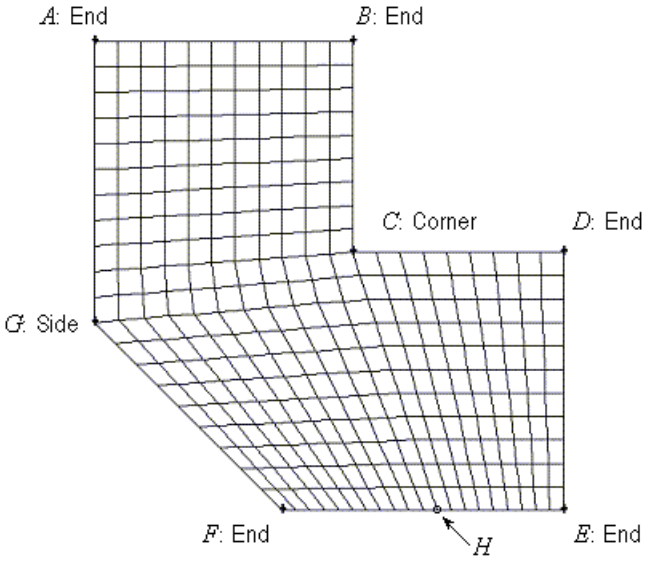
where N_C and N_R are the total numbers of Corner and Reversal type vertices, respectively, on the face. That is, for every Corner type vertex, the face must possess an additional End vertex, and for every Reversal vertex, the face must possess two additional End vertices.

Note:

You cannot specify Reversal vertices. Reversal vertices are used internally by the Meshing application to determine whether the face is mappable.

The shape of the mesh generated by means of the advanced face meshing controls depends on the type and arrangement of vertex types on the face. As an example of the effect of vertex types, consider the face shown in Figure 105: Inside Corner Vertex (p. 321), which consists of a planar L-shaped face, one corner of which is truncated at an angle.

Figure 105: Inside Corner Vertex



In Figure 105: Inside Corner Vertex (p. 321), the inside corner vertex (*C*) is designated as a Corner vertex, therefore, in order to be submappable, the face must possess five End type vertices (*A*, *B*,

D, *E*, and *F*). The advanced mapped Face Mesh control divides the face into the following two mapped regions:

- *A*, *B*, *C*, *H*, *F*, *G*
- *C*, *D*, *E*, *H*

Note:

If you enforce an advanced mapped face mesh control on a face, the Meshing application evaluates the face with respect to its vertex type designations. If the vertex types do not meet the criteria outlined above, the Meshing application attempts to change the vertex types so that the face is submappable.

For most submappable faces, there are multiple configurations of vertex types that satisfy the vertex type criteria. Each vertex type configuration results in a unique node pattern for the submapped mesh. When the Meshing application automatically changes vertex types, it attempts to employ the configuration that minimizes distortion in the mesh. To enforce a specific node pattern on a submapped mesh, manually select the vertices such that they meet the advanced mapped mesh control vertex type criteria outlined above. (See Selecting the Vertex Type and Picking Vertices (p. 323).)

Restrictions Related to Edge Mesh Intervals

If you specify a bias on the edge of a face before applying an advanced mapped Face Mesh control to the face, you must specify the bias on all parallel edges of the face.

Selecting Faces and Vertices

To use advanced mapped Face Mesh controls on a face, you must do the following:

- Select the face upon which the vertex types are to be defined
- Select the vertex type (using the **Specified Sides**, **Specified Corners**, and **Specified Ends** controls (p. 325))
- Pick the vertices to which the vertex type specification is to be applied

Selecting the Face

The Meshing application vertex types are specific to the faces upon which they are set. Therefore, to specify the type designation of an individual vertex, you must first select a face to be associated with that vertex. An individual vertex may possess as many vertex type designations as the number of faces to which it is attached. For example, it is possible for a vertex to possess a Side type designation with respect to one face and an End type designation with respect to another, as long as two separate mapped face meshing controls are defined for the two faces. For more information, refer to Setting Advanced Face Meshing Controls for Mapped Meshing (p. 325).

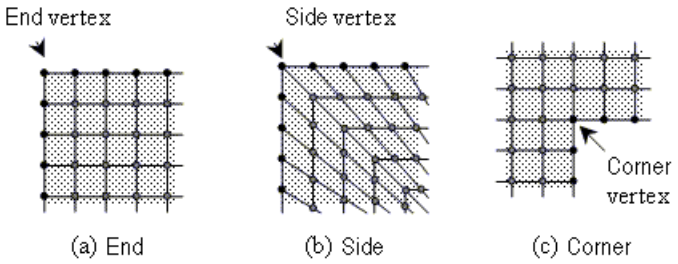
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting the Vertex Type and Picking Vertices

The structure of any face mesh in the vicinity of an individual vertex on its boundary is a function of the vertex type. There are three vertex types that you can specify.

- End
- Side
- Corner

Figure 106: Face Vertex Types



An individual vertex may possess only one vertex type designation. For example, you cannot designate a vertex as type "side" and also designate that same vertex as type "end." For more information, refer to Setting Advanced Face Meshing Controls for Mapped Meshing (p. 325).

Each vertex type differs from the others in the following ways:

- The number of face mesh lines that intersect the vertex
- The angle between the edges immediately adjacent to the vertex

The following table summarizes the characteristics of the vertex types shown in Figure 106: Face Vertex Types (p. 323).

Note:

If a face has only 4 vertices and 4 edges, the maximum for the range of the angle of a Side vertex type is 179°, and the acceptable range shifts accordingly.

Vertex Type	Intersecting Grid Lines	Range of Angle Between Edges
End	0	0° — 135°
Side	1	136° — 224°
Corner	2	225° — 314°
Reversal	3	315° — 360° (You cannot specify Reversal vertices. The range for Reversal vertices is used internally by the Meshing application to determine whether the face is mappable.)

The following sections describe the general effect of the End, Side, and Corner vertex types on the shape of the face mesh in the vicinity of a specified vertex.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

End Vertex Type

When you specify a vertex as the End vertex type (**Specified Ends** control (p. 325)), the Meshing application creates the face mesh such that only two mesh element edges intersect at the vertex (see (a) in Figure 106: Face Vertex Types (p. 323)). As a result, the mapped and submapped face mesh patterns on both sides of the End vertex terminate at the edges adjacent to the vertex. Assigning the End vertex type to a vertex whose adjacent edges form an angle greater than 180° will likely result in mesh failure.

Side Vertex Type

When you specify a vertex as the Side vertex type (**Specified Sides** control (p. 325)), the Meshing application creates the face mesh such that three mesh element edges intersect at the vertex (see (b) in Figure 106: Face Vertex Types (p. 323)). The Meshing application treats the two topological edges that are adjacent to the vertex as a single edge for the purposes of meshing.

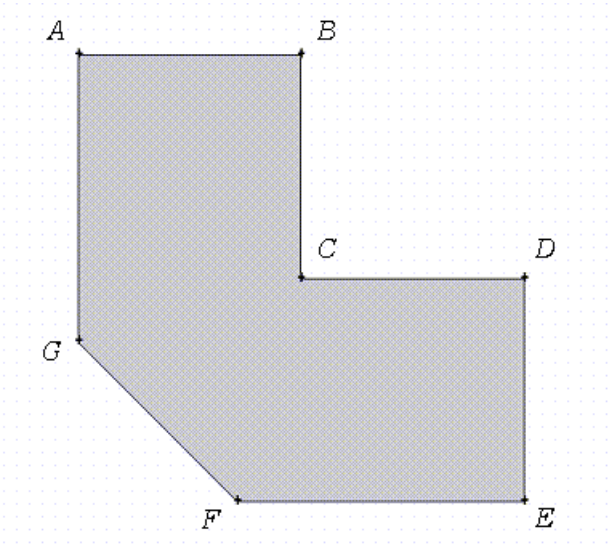
Corner Vertex Type

When you specify a vertex as the Corner vertex type (**Specified Corners** control (p. 325)), the Meshing application creates the face mesh such that four mesh element edges intersect at the vertex (see (c) in Figure 106: Face Vertex Types (p. 323)). Assigning the Corner vertex type to a vertex whose adjacent edges form an angle less than 180° will create an unnecessarily bad quality mesh (although the mesh will be valid).

Effect of Vertex Type on Face Meshes

As an example of the general effects of vertex types on face meshes, consider the planar face shown in Figure 107: Seven-sided Planar Face (p. 324). The following two examples illustrate the effects of different vertex type specifications applied to vertices *C*, *F*, and *G* on the shape of the resulting mesh.

Figure 107: Seven-sided Planar Face

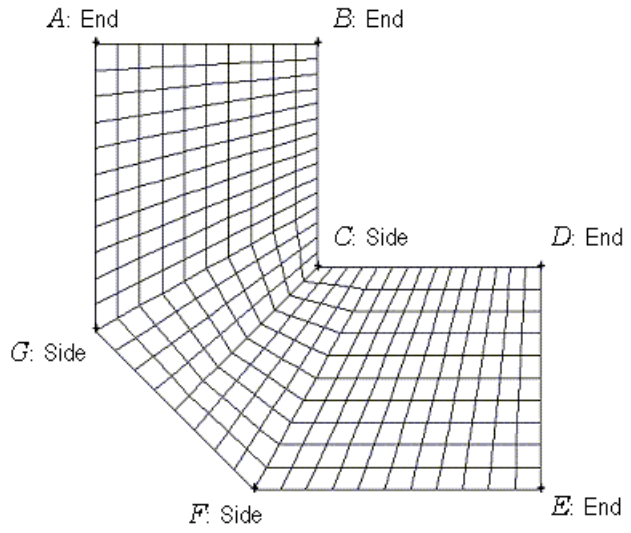


In Figure 108: Example Face Mesh—Side Inside Corner Vertex (p. 325), vertices *C*, *F*, and *G* are specified as Side vertices. Therefore, the Meshing application treats sides *BCD* and *EFGA* as if each were a

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

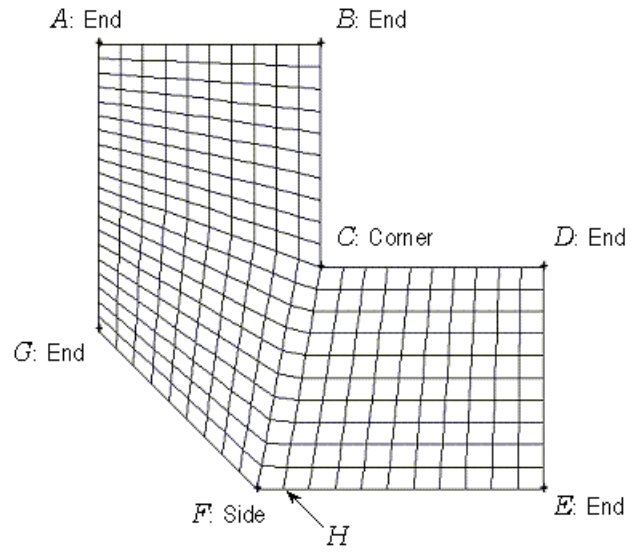
single edge. As a result, the entire face represents a mappable region, and the Meshing application creates a single checkerboard pattern for the mesh.

Figure 108: Example Face Mesh—Side Inside Corner Vertex



In Figure 109: Example Face Mesh—Corner Inside Corner Vertex (p. 325), vertices *C*, *F*, and *G* are specified as Corner, Side, and End type vertices, respectively. As a result, the face is submappable, and the Meshing application creates two separate checkerboard patterns for the mesh. The upper-left submapped region is defined by the polygon *ABCHFG*. The lower-right submapped region is defined by *CDEH*. For both regions, the node at point *H* serves as an End type vertex for the purposes of mesh creation.

Figure 109: Example Face Mesh—Corner Inside Corner Vertex



Setting Advanced Face Meshing Controls for Mapped Meshing

This section describes the basic steps for setting Face Meshing controls for mapped meshing.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To set advanced Face Meshing controls for mapped meshing:

1. Insert a **Face Meshing** control by highlighting **Mesh** in the Tree and right-clicking to view the menu. Select **Insert> Face Meshing**. **Definition>Mapped Mesh** is set to **Yes** by default.
2. Select the face upon which the vertex types are to be defined by scoping the face in the **Mapped Face Meshing** Details View. (Refer to Selecting the Face (p. 322) for more information.)
3. For the **Definition> Method** control, choose **Quadrilaterals** or **Triangles: Best Split.**
4. Enter additional **Definition** settings, as desired, in the Details View.
5. Use the **Specified Sides**, **Specified Corners**, and **Specified Ends** controls in the **Advanced** section of the Details View to select the desired vertices in the **Geometry** window and apply your selections. To do so, pick the desired vertex/vertices in the **Geometry** window and then click the **Specified Sides**, **Specified Corners**, or **Specified Ends** control to assign the vertex/vertices to the desired vertex type. (Refer to Selecting the Vertex Type and Picking Vertices (p. 323) for more information.)

Note:

If you select a vertex by mistake and want to de-select it, click the control in question in the **Advanced** section of the Details View, clear the selection by clicking in an "empty" portion of the **Geometry** window, and then click **Apply**. For example, assume that you mistakenly selected 1 vertex for the **Specified Corners** control. To clear the selection:

- In the **Specified Corners** control in the **Advanced** section of the Details View, click your selection (that is, the text "1 Vertex").

  The **Apply**/**Cancel** buttons will appear in the **Specified Corners** control and the vertex will be highlighted in green in the **Geometry** window.

- Click in an empty portion of the **Geometry** window.
- Click **Apply** in the **Specified Corners** control.

Note:

An individual vertex may possess as many vertex type designations as the number of faces to which it is attached. For example, it is possible for a vertex to possess a Side type designation with respect to one face and an End type designation with respect to another, as long as two separate mapped face meshing controls are defined for the two faces. Conversely, a single mapped Face Meshing control cannot specify the same vertex as more than one vertex type. That is, you cannot designate a vertex as type Side and also designate that same vertex as type End in a single mapped face meshing control. If you attempt to do so, the second and any subsequent assignments for that vertex will result in the control being highlighted in yellow in the **Advanced** section of the Details View, and you will not be able to

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

generate a mesh. If this occurs, use the procedure noted above to de-select the unwanted vertex assignment(s).

6. Generate the mesh by right-clicking the **Mesh** object in the Tree and selecting **Generate Mesh**.

Notes on Face Meshing Controls for Mapped Meshing

Remember the following notes when using the Face Meshing controls for mapped meshing:

- The blue status icon that may appear in the Tree Outline indicates that a mapped mesh cannot be provided on the scoped topology. One of three scenarios triggers the icon:

  1. The face cannot be map meshed.
  2. The quality of the mapped mesh was not acceptable and a free mesh was generated.
  3. If **Constrain Boundary** (p. 315) is set to **Yes**, the mesher will fail if the boundary of a mapped mesh must be modified.

- For mixed/solid shell parts, a Face Meshing control cannot be scoped to a sheet face if the face is adjacent to a solid body. In such cases, the meshing of the sheet face will fail.

- To assist you in working with mapped face meshing, you may want to use the **Show Mappable Faces** (p. 542), **Show Sweepable Bodies** (p. 537), and/or **Preview Surface Mesh** (p. 531) features. The **Show Mappable Faces** feature selects all mappable faces automatically and highlights them in the **Geometry** window. By using the **Show Sweepable Bodies** feature, you can find out whether bodies are sweepable (before and after modifying vertex types). By using the **Preview Surface Mesh** feature, you can verify that your mesh settings are correct.

- When sweeping (p. 399):

  – If the sweep method is applied to a body and mapped face meshing is defined for either the body's source or target face, the sweep mesher will fail if a mapped mesh cannot be obtained for the face.

  – When mapped face meshing is defined for a side face, the mapped mesher will loosen its tolerances on determining whether a face is mappable.

  – When sweeping and using *advanced* mapped meshing controls (p. 322), you must set vertex types for both the source and target faces.

- When a face has only 4 vertices and 4 edges, and mapped Face Meshing controls are applied, the only factor that will determine a successful mapped mesh is element quality.

Note:

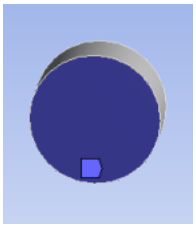
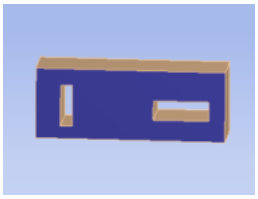
It is often helpful to use the **Show Vertices** option to ensure edges are complete and do not have unintended segmentation. If an edge is segmented, it could mean that a face you think should be successfully mapped actually has 5 vertices and 5 edges. To

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

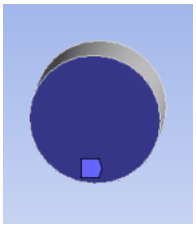
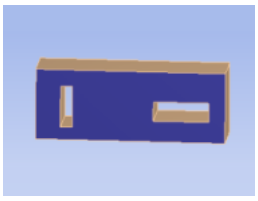
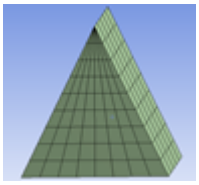
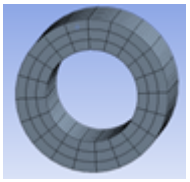
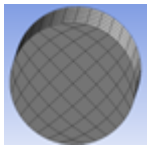
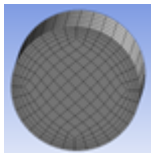
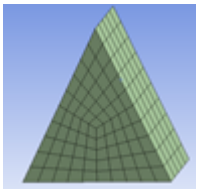
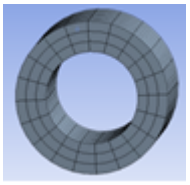
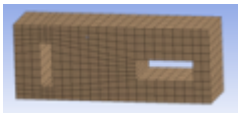
help resolve such issues, you can define a virtual edge (p. 546) or use advanced mapped Face Meshing controls (p. 320).

- An effective technique for mapped meshing on surface bodies is to select all faces on the body and let the mesher determine which faces should be map meshed and which faces should be free meshed.
- If the mapped Face Meshing controls are attached to faces with exactly two boundary edges or two sets of boundary edges, an additional option, **Internal Number of Divisions**, is available. This option allows you to specify the number of layers of elements that will be generated between the two boundary edges.
  - If there is a conflict between **Internal Number of Divisions** and a **Face** sizing control, the mapped face control's Internal Number of Divisions value will take priority.
  - If there is a conflict between **Internal Number of Divisions** and a parallel **Edge** sizing control, the Sweep mesh method will respect the Internal Number of Divisions unless the size is hard, in which case it will return an error notifying you of the conflict. The **MultiZone** mesh method will respect the **Edge** sizing control.
  - If there is a conflict between the **Internal Number of Divisions** and the number of divisions on a Sweep mesh method, the Sweep mesh method will return an error notifying you of the conflict.
  - See Face Characteristics for Annular faces in the table below for more information.

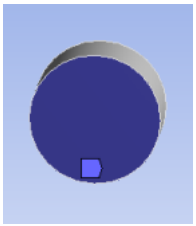
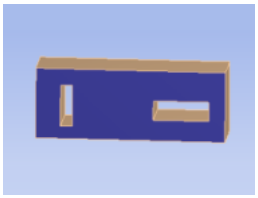
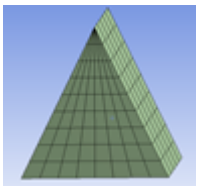
The table below provides an overview of types of faces and how various mesh methods handle when mapped Face Meshing controls are applied to them.

Mesh Method	Face Characteristic			
	Circular 	**Triangle**	**Annular**	**Internal Loops** 
• Sweep (p. 235) • Thin Sweep (p. 406) • Hex Dominant (p. 233) • Patch Conforming Tetrahedron (p. 208)	Not supported. You must free mesh these faces.	Not supported. You must free mesh these faces. As side faces, triangle faces can be mapped to obtain a wedge mesh	Supported. In the Details View, the **Internal Number of Divisions** (p. 315) option is activated so you can specify the number of	Not supported. You must free mesh these faces, but note the following: If there is just one internal loop, it is treated as an

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh Method	Face Characteristic			
	Circular 	**Triangle**	**Annular**	**Internal Loops** 
• Quadrilateral Dominant (p. 282) • Triangles (p. 284)		at one corner, depending on source face selection. 	divisions across the annular region. The option is set to **3** in the example below: 	annular case. For example, if the model above were split in half, you would have a square annulus which would mesh similar to the circular annulus model.
• MultiZone (p. 242) • MultiZone Quad/Tri (p. 284)	Supported, but mesh quality is poor.  You can use MultiZone or Mapped Mesh to obtain an O-Grid for better quality in corners. 	Supported. Triangle faces are submapped to tri primitives.  As side faces, triangle faces can be mapped to obtain a wedge mesh at one corner, depending on source face selection.	Supported. In the Details View, the **Internal Number of Divisions** (p. 315) option is activated so you can specify the number of divisions across the annular region. The option is set to **3** in the example below: 	Ignored for source faces. However, internal loops are supported for side faces. If this example is meshed top to bottom, and has mapped faces defined on the sides, it meshes as follows:  For more information on using mapped Face Meshing with

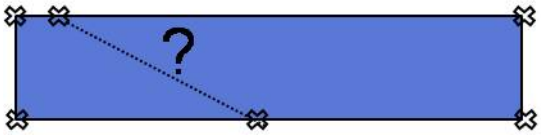
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh Method	Face Characteristic			
	Circular	**Triangle**	**Annular**	**Internal Loops**
				
				side faces, see Side Face Handling of Imprinted Regions (p. 432).

With fillets, chamfers, or large or small angles

- It is not always clear which vertex(es) should be used as the corner(s). Using virtual topology (p. 545) to merge edges or split faces, or using mapped face meshing vertex controls may help.

- When there are a large number of segments along a side. This situation may create difficulties in assigning incremental edge assignments that lead to good quality mesh. In these cases, adding a face split is a good way to ensure the interval edge assignment is done correctly.



Concerns over the mappability of faces may be different depending on whether the faces are source/target faces or side faces. For sweeping to be successful (regardless of whether Sweep (p. 235), Thin Sweep (p. 406), or MultiZone (p. 242) is used), all side faces (that is, all non-source/target faces) must be mappable. Ensuring side face mappability is most critical to ensure a successful swept mesh. Ensuring a source face can be mapped should be a lower priority, and your strategy for sweeping should account for this difference. You can use the **Show>Mappable Faces** (p. 542) option to help determine mappability of faces.

In addition to the mappability of each individual side face, the collective set of faces for a given side may present a problem. For example, for a collective set of faces, all parallel edges need to have the same number of divisions from the top to the bottom of the sweep. Ensuring all side faces of a swept body are mappable does not always ensure the body is sweepable. For example, if parallel edges of a mapped face change in direction, a source edge could become a side edge and make the body impossible to sweep. Also, edge splits on one face need to propagate through the collective set of faces while maintaining a reasonable quality mesh. Reducing the number of edge splits may simplify the sweeping and lead to a better quality mesh. This effect is the result of the simplification in the

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

"interval edge assignment," or the requirement for the mesher to have the same number of divisions along the sweep path.

Extending splits through the set of side faces may also help the mesher with the incremental edge assignments, as well as with constraining the grid lines along the sweep path to control the mesh quality. The imprinting that occurs with **MultiZone** creates further complications. See Side Face Handling of Imprinted Regions (p. 432) for more information.

Example Usage

For an example problem demonstrating how to use face meshing, see Brake-Squeal Analysis in the *Mechanical Technology Showcase: Example Problems*.

Mesh Copy Control

The **Mesh Copy** control enables you copy mesh from one body to another. **Mesh Copy** reduces the mesh setup time for repetitive bodies or parts. Association to CAD is maintained after performing mesh copy.

Mesh controls are scoped only to the source anchor body. When the mesh is generated, the source anchor body is meshed and copies the mesh to targets. Mesh Copy automatically merges nodes of bodies when both bodies belong to the same part or bodies have a shared interface.

To access the **Mesh Copy** control,

On the **Tree** Outline, right-click **Mesh** and click **Insert > Mesh Copy**.

or

On the **Tree** Outline, click **Mesh** and click **Mesh Copy** in the **Mesh Context** tab on the Ribbon.

Right-click on the face in the **Geometry** window and then select **Insert > Mesh Copy**.

Mesh Copy Details view has the following options:

Scope

- **Scoping Method**: Allows you to select the geometry selections or named selections for scoping. The default value is **Geometry Selection**.
  - **Geometry Selection**: Allows you to scope the geometry faces or bodies. When you select **Geometry Selection**, **Source Anchor** allows you to select the face or body as source from the **Geometry** window.
  - **Named Selection**: Allows you to scope the faces or bodies grouped under a named selection. When you select **Named Selection**, **Source Anchor** allows you to select the appropriate **Named Selection** as source.
- **Target Scoping**: Allows you to scope the target faces or bodies. The available options are **Geometry Selection** and **Named Selection**. The default value is **Geometry Selection**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Geometry Selection**: Allows you to scope the selected geometry. When you select **Geometry Selection**, the **Target Anchors** allows you to select the target faces or bodies from the **Geometry** window.

– **Named Selection**: Allows you to select the available name selection. When you select **Named Selection**, the **Target Anchors** allows you select the target faces or bodies from the available named selections.

Definition

- **Suppressed**: Allows you to suppress the method control. The default value is **No**. When **Suppressed** is **Yes**, the **Active** provides the number of suppressed parts and is read-only.
- **Node Merge Tolerance Option**: Allows you to select the node merge tolerance options. The default option is **Program Controlled**. The available options are :

  – **Program Controlled**: Performs node merge within the shared topologies with a default tolerance. The default tolerance is 10 percent of the global **Element Size**.

  – **Percentage of Element Size**: Performs node merge within the shared topologies with the provided percentage of element size.

  – **Absolute Value**: Performs node merge on shared topologies based on the absolute distance. The default absolute value is calculated as 0.1 * global **Element Size**.

Note:

While scoping the source and target anchors, note the following:

- The source and target face area should be the same.
- The associated source and target bodies should have the same volume.
- The source and target configuration should be identical (for example, a circle to rectangle mapping is incorrect, even if the face area is the same).
- If the above is not true, the mesh may be copied, but the nodes may not be associated properly to the target bodies, or the copied mesh transformation may be incorrect.

Figure 110: Mesh Copy Scope (p. 333) shows the set up of the Source face (blue) and Target Anchor faces (red) for the Mesh Copy control. Figure 111: Generated Mesh (p. 333) shows the mesh that was generated.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 110: Mesh Copy Scope

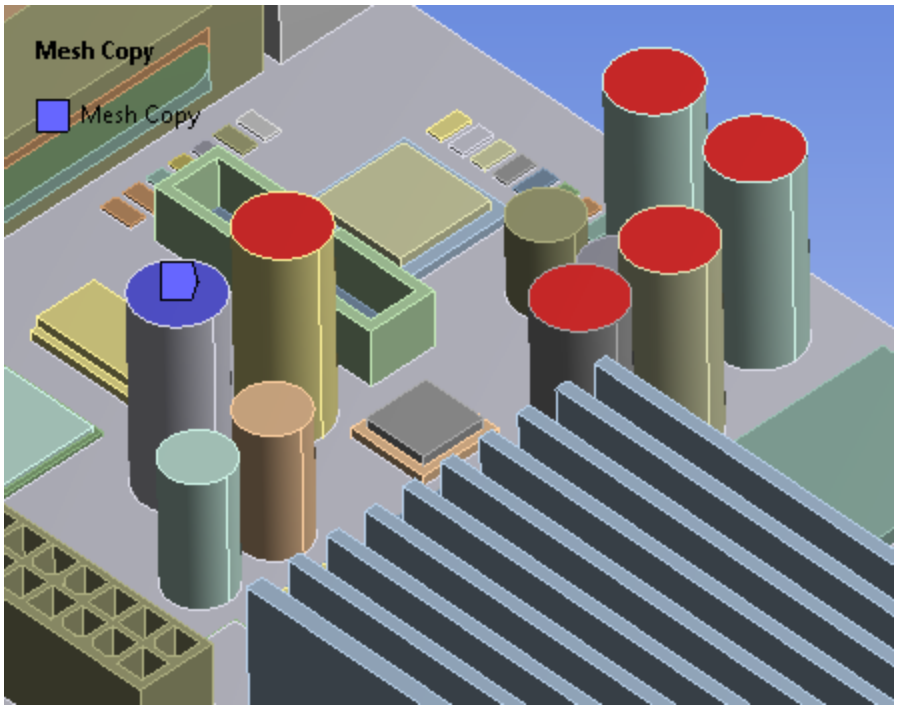
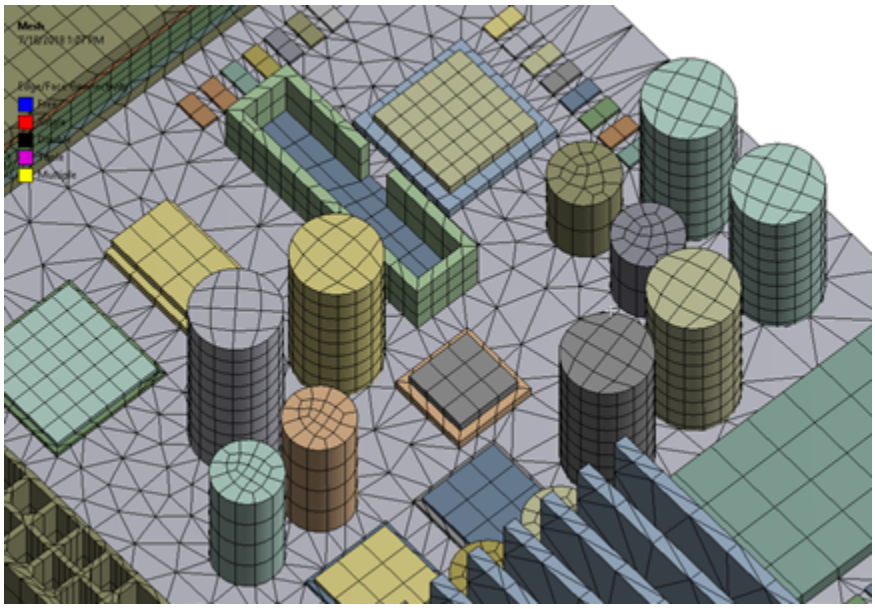


Figure 111: Generated Mesh



Note:

- **Mesh Copy** does not support previewing the surface mesh, or previewing inflation.
- **Mesh Copy**supports multibody parts with share topology. Bodies which are scoped to a **Mesh Copy** control are meshed first, and then the remaining bodies are meshed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mesh Copy** supports Selective meshing (p. 444) only if the source bodies are meshed before the target bodies. If the target bodies are meshed first, the mesh is not copied.
- The **Mesh Copy** target body is a reflection of the source. Element connectivities are different. This may cause the solver to fail. Choosing a different target anchor face might help in some cases. True reflection is not supported.
- **Mesh Copy** does not support refinement controls.
- A body can be scoped to **Mesh Copy** control target scoping, and mesh **Method Controls** geometry scoping at the same time.
- The mesh copy algorithm gives higher priority to **Method Control** when a body is scoped to **Method Controls** geometry scoping and **Mesh Copy** target scoping.
- The source body does not copy the sizing controls set up on the target bodies.

Match Control

The **Match Control** matches the mesh on two or more faces or edges in a model. The Meshing application provides two types of match controls—cyclic (p. 336) and arbitrary (p. 338).

To access the **Match control**, right-click **Mesh** on the Tree Outline. Click **Insert > Match Control**. You can also click **Mesh** on the **Tree Outline**, click the **Mesh** Context Toolbar on the ribbon, then click **Match Control**.

The **Match Control** is supported for the following mesh methods:

Volume Meshing:

- Sweep (p. 235)
- Patch Conforming (p. 208)
- MultiZone (p. 242)

Surface Meshing:

- Quad Dominant (p. 282)
- All Triangles (p. 284)
- Automatic (PrimeMesh) (p. 287)

Remember the following information when using the match control feature:

- Edge meshes are matched for sheet, 2D, and 3D bodies. Face meshes are matched across bodies.
- A single match control with one high face and one low face cannot be applied across multiple parts. If there are multiple faces on the high side and multiple faces on the low side, the software does its best to match the high and low sides on a part-by-part basis. For example, the match control will support situations in which there are two parts, each having one face on the high side and one face

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

on the low side (for a total of two high faces and two low faces). However, for more complex situations, you must be careful to ensure the proper matching is done.

- Matching will fail if the high and low faces are on two separate bodies that have other bodies (being meshed with a method other than Sweep) or a space between them.
- The faces or edges that you select must be topologically and geometrically the same.

  This means:

  – They have the same number of vertices on the high and low sides.

  – They have similar surface area or length.

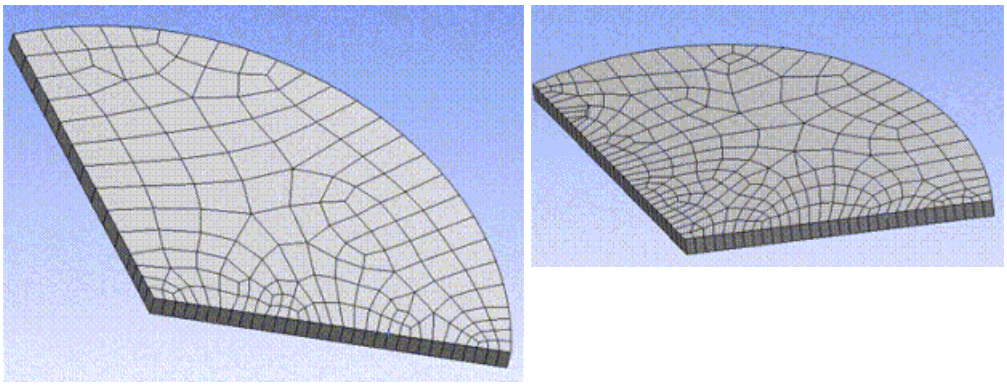
  – The high and low sides have similar transformation.

    The mesher will move nodes away from the geometry to meet the transformation.
- If the low and high sides of a geometry do not match, the high side geometry is meshed and the low side will be meshed using the transformation from the match control. In this case, the low side geometry and mesh might be slightly different, so a warning is displayed prompting you to check this deviation and decide whether it is acceptable or that you should modify the geometry.
- Multiple match controls can be associated with a single entity, but multiple associations can result in conflicts among match controls. If a conflict occurs, the Meshing application issues an error message, and matching fails. For example, a match control conflict may occur if the two faces adjacent to an edge have two different match controls applied to them. If the two match controls use two different sets of coordinate systems the mesher may produce an error even if the transformation between the two match controls is the same. However, for this case, if the two match controls refer to the same set of coordinate systems there won't be any conflict.
- A match control can only be assigned to one unique face pair. Assigning the same face as **High/Low Geometry** in more than one match control is not supported. If multiple match controls assign the same face as a **High/Low Geometry** entity, the match control that appears lowest in the Tree is honored.
- Match controls are not respected with refinement or adaptivity.
- When you perform tet meshing on a body with match control applied, adjacent bodies get meshed concurrently, even if a single body is chosen using **Selective Meshing**.
- When match is used with the Sizing Options (p. 105), the effect of a sizing on the high or low side will be transferred bidirectionally from the high side to the low side and vice versa. This means that if the low side has a sizing control and the high side does not, the Sizing control will use the low sizing control on the high side.
- Match controls on faces are supported with Pre inflation (p. 172), regardless of whether inflation is set to Program Controlled (p. 164) or has been set through any global or local inflation definition. In contrast, match controls on edges are not supported with Pre inflation. Match controls (both faces and edges) are not supported with Post inflation (p. 173). For non-supported cases, Ansys Workbench automatically suppresses/disables the Match Control feature.
- Match controls are not enforced when previewing inflation (p. 534).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You cannot apply a match control to topology on which a face-edge pinch (p. 187), mesh connection, or symmetry control has been applied. In any of these cases, an error will be issued when you generate the mesh.
- Match controls on edges are not supported for the MultiZone (p. 242) mesh method. When match controls on faces are used with MultiZone, only one periodic or cyclic transformation is supported (MultiZone can support multiple match controls, as long as they use the same coordinate system and have the same angle/translation). In addition, MultiZone does not support matching of free meshed regions (p. 244).
- Match controls can be used with thin sweeping (p. 406), as shown in the figures below. In the figure on the left, a match control was applied to the top and bottom faces. In the figure on the right, a match control was applied to the side faces.

Figure 112: Match Controls Used with Thin Sweeping



Match control topics include:

Cyclic Match Control

Arbitrary Match Control

Note:

For general information on applying match controls in combination with the various mesh method controls, refer to Meshing: Mesh Control Interaction Tables (p. 475).

Cyclic Match Control

The cyclic matching process involves copying the mesh of the first selected faces or edges in the **Match Control** (the **High Geometry Selection** scoped in the Details View of the **Match Control**) to the second selected faces or edges in the control (the scoped **Low Geometry Selection**).

You can scope a Cyclic Match Control to either a geometry selection or a named selection.

If you want to automatically generate solver constraints for periodic mesh in addition to matching the mesh, you should use the **Symmetry** feature in the Mechanical application.

1. Insert a Match Control by right-clicking the Mesh object in the Tree Outline and selecting **Insert** > **Match Control**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the Details view, define the scope of the selection:

To apply a match control to...	Do this...
A geometry selection	1. Click **Scoping Method** and select **Geometry Selection**. 2. Select one or more faces or edges, and then in the **High Geometry Selection** field, click **Apply**. 3. Select one or more faces or edges, and then in the **Low Geometry Selection** field, click **Apply**.
A named selection	1. Click **Scoping Method** and select **Named Selection**. 2. In the **High Boundary** and **Low Boundary** fields, select the appropriate named selections.

3. In the Details View, select **Cyclic** as the **Transformation** type and a coordinate system with its z-axis aligned to the axis of rotation for the geometry.

4. Generate the mesh by right-clicking the **Mesh** object and selecting **Generate Mesh** from the context menu.

Note:

When a match is successful, the number of elements in the matched faces/edges will be the same, and there will be a direct one-to-one mapping between their nodes. A small blue status icon ( ) appears to the left of the **Match Control** object icon in the Tree Outline if the **Match Control** fails on the face or edge pair.

When a cyclic **Match Control** is used together with **Sizing** (p. 296) controls, the controls on the high side have the higher precedence. Whatever controls are on the high faces or edges will be honored on the low faces or edges in the **Match Control**. **Sizing** controls applied to the low faces or edges will be honored only if the high side does not have the same controls, and only if the sizing is applied *directly* on the low topology (that is, applying the sizing on connected topologies will have no effect).

Note:

The Meshing application inserts match controls for periodic regions automatically. See the Match Meshing and Symmetry (p. 462) section for more information.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Arbitrary Match Control

This feature lets you select multiple pairs of faces or edges in a model to create a match control that will consequently generate exactly the same mesh on the high geometry as it does on the low geometry. You can select a set of high faces belonging to different parts, and low faces belonging to different parts, as long as the high and low pairs are on the same body. So, for example, you can have a single match control consisting of multiple pairs of faces or edges across bodies.

However, unlike cyclic match controls (p. 336), which require you to select a coordinate system with its z-axis of rotation aligned to the geometry's axis of rotation, for arbitrary match controls the faces or edges to be matched can be arbitrarily located, and the match control is based on two coordinate systems that you select.

You can scope an Arbitrary Match Control to either a geometry selection or a named selection.

If you want to automatically generate solver constraints for periodic mesh in addition to matching the mesh, you should use the **Symmetry** feature in the Mechanical application.

1. Insert a Match Control by right-clicking the Mesh object in the Tree Outline and selecting **Insert** > **Match Control**.

2. In the Details view, define the scope of the selection:

To apply a match control to...	**Do this...**
A geometry selection	1. Click **Scoping Method** and select **Geometry Selection**. 2. Select one or more faces or edges, and then in the **High Geometry Selection** field, click **Apply**. 3. Select one or more faces or edges, and then in the **Low Geometry Selection** field, click **Apply**.
A named selection	1. Click **Scoping Method** and select **Named Selection**. 2. In the **High Boundary** and **Low Boundary** fields, select the appropriate named selections.

3. Change the value of the **Transformation** control to **Arbitrary**.

4. Choose the coordinate systems for the selected high and low geometry entities. The applicable settings in the Details View are:

   - **High Coordinate System**: Choose the coordinate system for the faces/edges assigned by the **High Geometry Selection** control.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Low Coordinate System**: Choose the coordinate system for the faces/edges assigned by the **Low Geometry Selection** control.

Note:

All the coordinate systems currently defined for the model appear in the **High Coordinate System** and **Low Coordinate System** drop-down menus. You may choose coordinate systems from the list, or you may need to define additional coordinate systems. For the match control to be honored, the coordinate systems that you choose must be defined such that a valid transformation matrix can be calculated. Else, the two coordinate systems must be created such that when the coordinates of every point of the **Low Geometry Selection** in the **Low Coordinate System** are placed into the **High Coordinate System**, the high and low faces/edges match exactly. Refer to the Coordinate Systems Overview in the Mechanical help for information on coordinate systems and how to create them.

---

5. Generate the mesh by right-clicking on the **Mesh** object and selecting **Generate Mesh** from the context menu.

   When a match is successful, the number of elements in the matched faces/edges will be the same, and there will be a direct one-to-one mapping between their nodes. If the mesh is generated but the match was unsuccessful, a small blue status icon ( ) displays to the left of the **Match Control** object icon in the Tree Outline.

Example 5: Arbitrary Mesh Matching

The figures below show an example of arbitrary mesh matching. Figure 113: Coordinate Systems for Arbitrary Mesh Matching (p. 339) shows the selected coordinate systems, and Figure 114: Matched Mesh (p. 340) shows the resulting matched mesh. Edge and face sizings were also applied.

Figure 113: Coordinate Systems for Arbitrary Mesh Matching

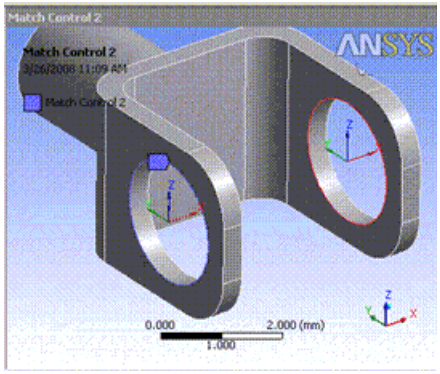
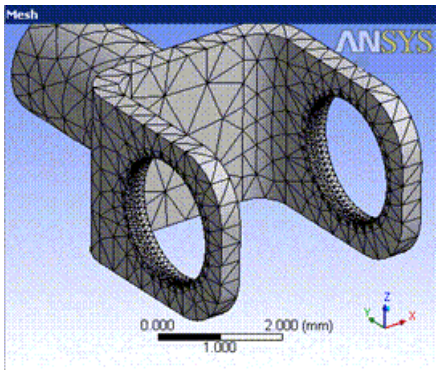


Figure 114: Matched Mesh



Pinch Control

The **Pinch** control lets you remove small features (such as short edges and narrow regions) at the mesh level in order to generate better quality elements around those features.

When **Pinch** controls are defined, the small features in the model that meet the criteria established by the controls will be "pinched out," thereby removing the features from the mesh. You can instruct the Meshing application to automatically create pinch controls based on settings that you specify (as described in Pinch Control Automation Overview (p. 190)), or you can manually designate the entities to be pinched, as described below in Defining Pinch Controls Locally (p. 340).

Local pinch control topics include:

Defining Pinch Controls Locally

Changing Pinch Controls Locally

For an overview of pinch controls and details on pinch control automation, refer to Pinch (p. 187). For general information on applying pinch controls in combination with the various mesh method controls, refer to Meshing: Mesh Control Interaction Tables (p. 475). For general information about pinch control usage, see Usage Information for Pinch Controls (p. 194).

Defining Pinch Controls Locally

This section describes the steps for defining pinch controls locally.

To define pinch controls locally:

1. Click the **Mesh** folder in the Tree Outline. Right-click and select **Insert** > **Pinch** from the context menu.

   Or

   Click **Mesh Control** on the toolbar and choose **Pinch** from the menu.

   A pinch control object is inserted into the Tree.

2. In the **Geometry** window, pick one or more faces, one or more edges, or one vertex that you want to define as the Primary. (A Primary geometry is an entity that retains the profile of the

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

original geometry.) Refer to Pinch (p. 187) for a table that summarizes the valid entities you can pick for each type of pinch control.

3. Use either of these methods to apply your selection:

   - Right-click in the **Geometry** window to display the context menu and select **Set As Pinch Primary**.
   - Click in the **Primary Geometry** field in the Details View.

   The pinch region is flagged in the **Geometry** window. The color of each selected entity changes to blue to identify it as primary geometry. If you want to adjust your selections, you can re-pick geometry and then apply the new selections to overwrite the existing primary geometry. If using the context menu method to apply selections, you can also pick additional geometry and select **Add To Pinch Primary** to add the geometry to the existing primary geometry. Also see Changing Pinch Controls Locally (p. 343).

4. In the **Geometry** window, pick one or more edges or vertices that you want to define as the secondary. (A secondary geometry is an entity that changes in order to move towards the primary geometry. Depending on the tolerance, the pinch control will pinch out the entire secondary entity or only a portion of the secondary entity into the primary. Faces cannot be defined as secondary.)

5. Use either of these methods to apply your selection:

   - Right-click in the **Geometry** window to display the context menu and select **Set As Pinch Secondary**.
   - Click in the **Secondary Geometry** field in the Details View.

   The color of each selected edge/vertex changes to red to identify it as secondary geometry. If you want to adjust your selections, you can re-pick geometry and then apply the new selections to overwrite the existing secondary geometry. If using the context menu method to apply selections, you can also pick additional geometry and select **Add To Pinch Secondary** to add the geometry to the existing secondary geometry. Also see Changing Pinch Controls Locally (p. 343).

6. Change the value of the **Suppressed** control if desired.

   By default, the value of **Suppressed** is **No**. If you change the value to **Yes**, this pinch control has no effect on the mesh (that is, the small features you picked for this pinch control are not removed and will affect the mesh). In addition, an **Active** control with a read-only setting of **No, Suppressed** appears under the **Suppressed** control when **Suppressed** is set to **Yes**.

7. Change the value of the **Tolerance** control if desired.

   By default, the value of **Tolerance** is based on the global pinch control tolerance (p. 192). If you specify a different value here, it overrides the global value.

8. Change the value of the **Snap to Boundary** control if desired.

   By default, the value of **Snap to Boundary** is **Yes**. **Snap to Boundary** is applicable only for pinch controls in shell models for which a face has been defined as the primary geometry and one or more edges are defined as the secondary. In such cases, when the value of **Snap to Boundary** is **Yes** and the distance from a secondary edge to the closest mesh boundary of the primary face

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is within the specified snap to boundary tolerance, nodes from the secondary edge are projected onto the boundary of the primary face. The joined edge will be on the primary face along with other edges on the primary face that fall within the defined pinch control tolerance. Refer to the figures below to see the effect of the **Snap to Boundary** setting.

Note:

For edge-to-edge pinch controls in shell models, the snap tolerance is set equal to the pinch tolerance internally and cannot be modified.

9. Change the value of the **Snap Type** control if desired. **Snap Type** appears only when the value of **Snap to Boundary** is **Yes**.

   - If **Snap Type** is set to **Manual Tolerance** (the default), a **Snap Tolerance** field appears where you may enter a numerical value greater than 0. By default, the **Snap Tolerance** is set equal to the pinch tolerance but it can be overridden here.
   - If **Snap Type** is set to **Element Size Factor**, a **Primary Element Size Factor** field appears where you may enter a numerical value greater than 0. The value entered should be a factor of the local element size of the primary topology.

Note:

When you define a pinch control locally, the value of **Scope Method** is **Manual** (read-only). If you make changes to a pinch control that was created through pinch control automation (p. 193), the value of the **Scope Method** field for that pinch control changes from **Automatic** to **Manual**.

Figure 115: Snap to Boundary Set to Yes (p. 343) shows the mesh when **Snap to Boundary** is set to **Yes** (default).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 115: Snap to Boundary Set to Yes

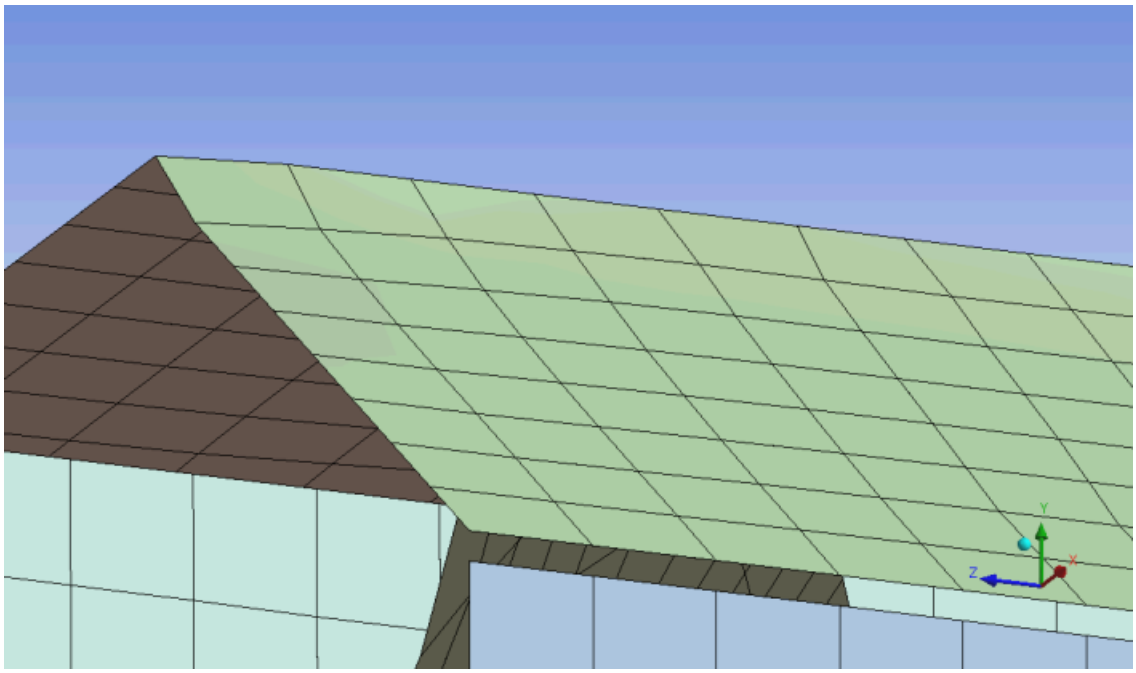
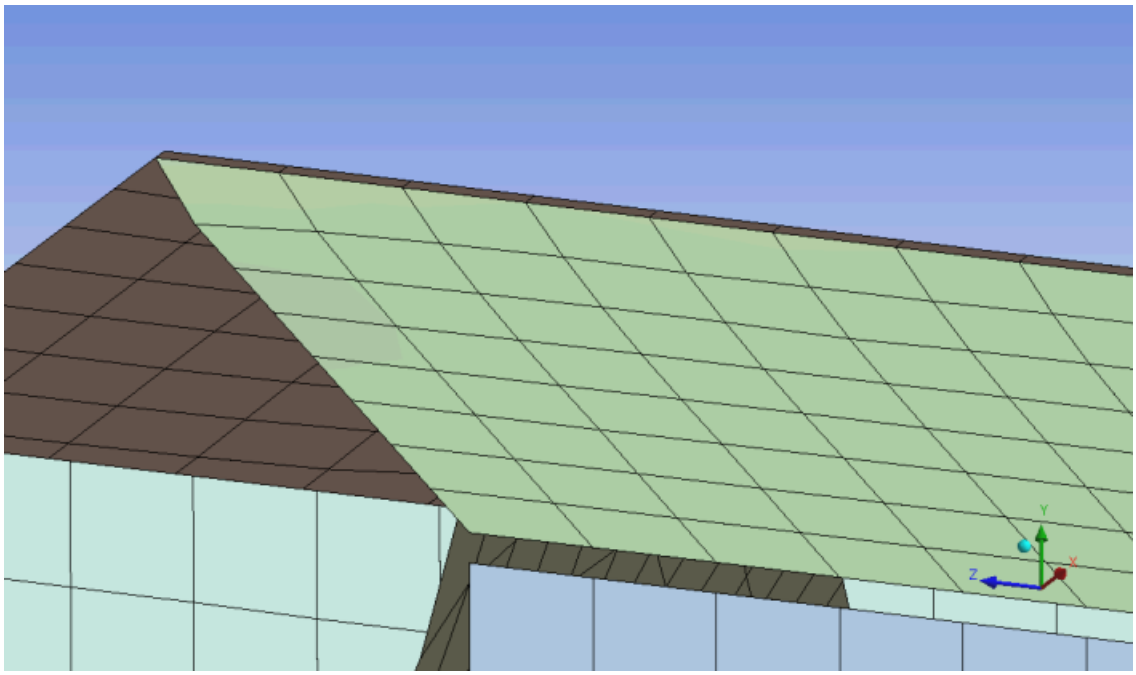


Figure 116: Snap to Boundary Set to No (p. 343) shows the mesh for the same model when **Snap to Boundary** is set to **No**.

Figure 116: Snap to Boundary Set to No



Changing Pinch Controls Locally

This section describes the steps for changing pinch controls locally. You can make changes to pinch controls regardless of whether they were created automatically or manually. You can select multiple pinch controls to make the same changes to all selected pinch controls at one time.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To change pinch controls locally:

1. In the Tree Outline, select the pinch control(s) that you want to change.
2. In the **Geometry** window, pick one or more faces, one or more edges, or one vertex that you want to define as primary. Refer to Pinch (p. 187) for a table that summarizes the valid entities you can pick for each type of pinch control.
3. Use either of these methods to apply your selection:

   - Right-click in the **Geometry** window to display the context menu and select **Set As Pinch Primary** or **Add To Pinch Primary**.
   - Click in the **Primary Geometry** field in the Details View.

   The pinch region is flagged in the **Geometry** window. The color of each selected entity changes to blue to identify it as primary geometry. If you want to further adjust your selections, you can re-pick geometry and then apply the new selections to overwrite or add to the existing primary geometry. To add to the geometry, you must use **Add To Pinch Primary** in the context menu.
4. In the **Geometry** window, pick one or more edges or vertices that you want to define as secondary. (Faces cannot be defined as secondary.)
5. Use either of these methods to apply your selection:

   - Right-click in the **Geometry** window to display the context menu and select **Set As Pinch Secondary** or **Add To Pinch Secondary**.
   - Click in the **Secondary Geometry** field in the Details View.

   The color of each selected edge/vertex changes to red to identify it as secondary geometry. If you want to further adjust your selections, you can re-pick geometry and then apply the new selections to overwrite or add to the existing secondary geometry. To add to the geometry, you must use **Add To Pinch Secondary** in the context menu.
6. Change the value of the **Suppressed** control if desired.

   By default, the value of **Suppressed** is **No**. If you change the value to **Yes**, this pinch control has no effect on the mesh (that is, the small features you picked for this pinch control are not removed and will affect the mesh). In addition, an **Active** control with a read-only setting of **No, Suppressed** appears under the **Suppressed** control when **Suppressed** is set to **Yes**.
7. Change the value of the **Tolerance** control if desired.

   By default, the value of **Tolerance** is based on the global pinch control tolerance (p. 192). If you specify a different value here, it overrides the global value.
8. Change the value of the **Snap to Boundary** control if desired.

   By default, the value of **Snap to Boundary** is **Yes**. **Snap to Boundary** is applicable only for pinch controls in shell models for which a face has been defined as the primary geometry and one or more edges are defined as the secondary. In such cases, when the value of **Snap to Boundary** is **Yes** and the distance from a secondary edge to the closest mesh boundary of the primary face is within the specified snap to boundary tolerance, nodes from the secondary edge are projected

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

onto the boundary of the primary face. The joined edge will be on the primary face along with other edges on the primary face that fall within the defined pinch control tolerance. Refer to Figure 115: Snap to Boundary Set to Yes (p. 343) and Figure 116: Snap to Boundary Set to No (p. 343) to see the effect of the setting.

Note:

For edge-to-edge pinch controls in shell models, the snap tolerance is set equal to the pinch tolerance internally and cannot be modified.

9. Change the value of the **Snap Type** control if desired. **Snap Type** appears only when the value of **Snap to Boundary** is **Yes**.

   - If **Snap Type** is set to **Manual Tolerance** (the default), a **Snap Tolerance** field appears where you may enter a numerical value greater than 0. By default, the **Snap Tolerance** is set equal to the pinch tolerance but it can be overridden here.
   - If **Snap Type** is set to **Element Size Factor**, a **Primary Element Size Factor** field appears where you may enter a numerical value greater than 0. The value entered should be a factor of the local element size of the primary topology.

Note:

If you make changes to a pinch control that was created through pinch control automation (p. 193), the value of the **Scope Method** field for that pinch control changes from **Automatic** to **Manual**.

Inflation Control

Inflation is useful for CFD boundary layer resolution, electromagnetic air gap resolution or resolving high stress concentrations for structures. It is supported for the mesh methods listed in the section Inflation Group (p. 161). You can use local inflation mesh controls to apply inflation to specific boundaries. When an inflation control is scoped to a solid model, every scoped geometry must have a boundary defined for it. The settings of the local inflation controls will override global inflation control (p. 161) settings.

You can define local inflation controls either by inflating a method or by inserting individual inflation controls.

Inflating a Method

1. Insert a mesh method.
2. Associate (scope) the desired bodies with the method.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Right-click the method and choose **Inflate This Method** from the menu.

   Note:

   For example, if you right-click a **Tetrahedrons** method control or a **Sweep** method control and specify the source, then an **Inflate This Method** menu option is available. Choosing this option inserts an inflation control on every body to which the **Tetrahedrons** meshing method with the selected **Algorithm** is applied, or on every face to which the **Sweep** method is applied. Similarly, an inflation control will be inserted into the Tree Outline for each body/face. (Sweeping with inflation is the same as inflation with tetrahedrons except that with sweeping you pick faces and edges instead of bodies and faces.)

4. Highlight one of the inflation controls that was inserted into the Tree Outline.

5. Change the value of the **Suppressed** control if desired.

   By default, the value of **Suppressed** is **No**. If you change the value to **Yes**, this inflation control has no effect on the mesh. In addition, an **Active** control with a read-only setting of **No, Suppressed** appears under the **Suppressed** control when **Suppressed** is set to **Yes**.

6. Use either of these methods to specify the inflation boundaries:

   - In the Details View, set **Boundary Scoping Method** to **Geometry Selection**, pick the entities in the **Geometry** window, and click the **Boundary** field in the Details View to **Apply**.
   - In the Details View, set **Boundary Scoping Method** to **Named Selections**, select a Named Selection from the **Boundary** drop-down, and press **Enter**.

     Note:

     – To select multiple Named Selections to be used as inflation boundaries, press and hold the **Ctrl** key while selecting the Named Selections from the **Boundary** drop-down, and then press **Enter**.

     – If none of the predefined Named Selections include the correct topology to be used as an inflation boundary, no Named Selections will be available in the drop-down. For example, if you scoped a surface body with the method in step 2, a Named Selection containing an edge must exist. Otherwise, you cannot select anything from the drop-down.

7. Specify additional inflation options as desired. For details about options, refer to the notes below and to Inflation Group (p. 161).

8. Repeat steps 4 through 7 for each inflation control in the Tree Outline.

Inserting Individual Inflation Controls

1. Optionally, select the desired bodies or faces in the **Geometry** window.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Use either of these methods to insert the inflation control:

   - Click **Mesh Control** on the toolbar and choose **Inflation** from the menu.
   - Right-click in the **Geometry** window and choose **Insert** > **Inflation** from the menu.

3. If you selected the bodies or faces in step 1, go directly to step 4. If not, use either of these methods to scope inflation to the desired bodies or faces:

   - In the Details View, set **Scoping Method** to **Geometry Selection**, pick the entities in the **Geometry** window, and click the **Geometry** field in the Details View to **Apply**.
   - In the Details View, set **Scoping Method** to **Named Selection**, and select a Named Selection from the **Named Selection** drop-down.

4. Change the value of the **Suppressed** control if desired.

   By default, the value of **Suppressed** is **No**. If you change the value to **Yes**, this inflation control has no effect on the mesh. In addition, an **Active** control with a read-only setting of **No, Suppressed** appears under the **Suppressed** control when **Suppressed** is set to **Yes**.

5. Use either of these methods to specify the inflation boundaries:

   - In the Details View, set **Boundary Scoping Method** to **Geometry Selection**, pick the entities in the **Geometry** window, and click the **Boundary** field in the Details View to **Apply**.
   - In the Details View, set **Boundary Scoping Method** to **Named Selections**, select a Named Selection from the **Boundary** drop-down, and press **Enter**.

   Note:

   – To select multiple Named Selections to be used as inflation boundaries, press and hold the **Ctrl** key while selecting the Named Selections from the **Boundary** drop-down, and then press **Enter**.

   – If none of the predefined Named Selections include the correct topology to be used as an inflation boundary, no Named Selections will be available in the drop-down. For example, if you selected a face in step 1, a Named Selection containing an edge must exist. Otherwise, you cannot select anything from the drop-down.

6. Specify additional inflation options as desired. For details about options, refer to the notes below and to Inflation Group (p. 161).

Notes on Defining Local Inflation Controls (2D Only)

In most cases, the controls in the global Inflation (p. 161) group apply to both 3D and 2D inflation, and the values set globally will be populated to the local inflation controls. Changes that you make to the local inflation settings will override the global settings. Exceptions and special considerations for defining 2D local inflation are described here.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When defining 2D local inflation, the available options for the **Inflation Option** (p. 166) control are **Smooth Transition** (default), **First Layer Thickness**, and **Total Thickness**. If the **Inflation Option** control is set to **First Aspect Ratio** or **Last Aspect Ratio** globally, it will be set to **Smooth Transition** locally.

  – When **Inflation Option** is **Smooth Transition**, you can set values for Transition Ratio (p. 168), Maximum Layers (p. 169), and Growth Rate (p. 169).

  – If you select **First Layer Thickness**, you can set values for First Layer Height (p. 169), Maximum Layers (p. 169), and Growth Rate (p. 169). The value of **Growth Rate** is used to calculate the heights of the successive layers, as follows:

    Growth Rate = h_{n+1} / h_n

    where h_n = height of layer n

  – If you select **Total Thickness**, you can set values for Number of Layers (p. 169), Growth Rate (p. 169), and Maximum Thickness (p. 169). The first layer height is computed based on these three values. The heights of the successive layers are computed using the same formula shown above under **First Layer Thickness**.

  When 2D inflation is applied to a 3D model (that is, to the face of a 3D body), the local value of **Maximum Layers** will be set equal to the global value by default. However, when it is applied to a 2D model (i.e, to the face of a surface body), the local value of **Maximum Layers** will be set to 2.

- If you are using the Quadrilateral Dominant (p. 282) mesh method with inflation and the Size Function (p. 105) is on, the mesh size of the last inflation layer will be used for the corresponding Quadrilateral Dominant boundary mesh size.

Notes on Defining Local Inflation Controls (3D and 2D)

- To make inflation boundary selection easier, select **Annotation Preferences** from the Toolbar and then deselect **Body Scoping Annotations** in the **Annotation Preferences** option box to toggle the visibility of annotations in the **Geometry** window. For example, after scoping inflation to a body, the body will be displayed using a blue solid annotation. Turn off the body scoping annotations, then select the desired faces as boundaries. For picking internal faces, the **Hide Faces** right-click option may help you to see inside a body. For example, you can select external faces in the **Geometry** window and then use the **Hide Faces** option to hide the selected faces (making it easier to select the internal faces).

- Multiple **Inflation** controls can be scoped to the same body or face with different inflation options on the faces/edges.

- If the mesh method is **Automatic**, the **Patch Conforming** tetrahedron method will be used for inflation on a body and the **Sweep** method will be used for inflation on a face.

- If the mesh method is **Cartesian** (3D only) and **Physics Preference** is set to **CFD**, then three boundary layers are created with total thickness proportional to element size. If **Cartesian** and physics is not **CFD**, then a single boundary layer is added with thickness proportional to element size.

- In the following scenarios, using inflation results in automatic suppression of the refinement (p. 313) control:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - When automatic inflation (p. 163) (either Program Controlled (p. 164) or All Faces in Chosen Named Selection (p. 165)) is used with refinement in the same model
  - When local inflation (p. 345) is used with refinement in the same body or in the same part

- For information on setting global inflation controls and descriptions of all of the individual inflation controls, refer to Inflation Group (p. 161). For steps to follow to assign inflation depending on the selected mesh method, refer to Inflation Controls (p. 453). For general information on applying inflation controls in combination with the various mesh method controls, refer to Meshing: Mesh Control Interaction Tables (p. 475).

Gasket Control

Gasket Control applies a sweep mesh in a chosen direction and drops midside nodes on gasket elements that are parallel to the sweep direction.

For information regarding **Gasket Mesh** from the **Geometry** refer to Gasket Meshing.

Gasket Control is available only when Body object's Stiffness **Behavior** is set to **Gasket**.

Or

When there is **Static Structural** is available in the **Model Tree**.

To access **Gasket Control**,

On the **Tree Outline**, right-click **Mesh** and click **Insert** > **Gasket**.

Or

On the **Tree Outline**, click **Mesh** and click **Gasket** in the **Mesh Context** tab on the **Ribbon**.

Or

When Body object's **Stiffness Behavior** is set to **Gasket**, **Gasket Control** is automatically available on the Tree.

Gasket Control Details View has the following options:

Scope

- **Scoping Method**: Allows you to scope geometry bodies or named selection. The default value is **Geometry Selection**.
  - **Geometry Selection**: Allow you to scope the geometry bodies. When you select **Geometry Selection**, the Geometry allows you to select the geometry from the **Geometry** window.
  - **Named Selection**: Allow you to scope bodies grouped under a named selection.

Definition

- **Element Order**: Allows you to select element order for **Gasket Control**. The default value is **Use Global Setting**. For information on the **Element Order** option, refer to Method Controls and Element Order Settings.

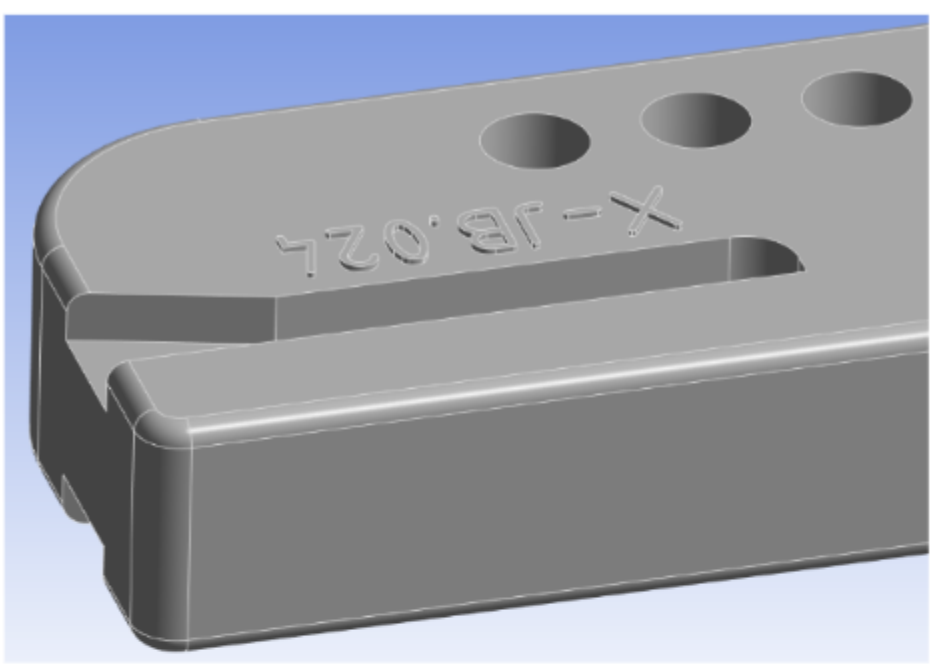
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

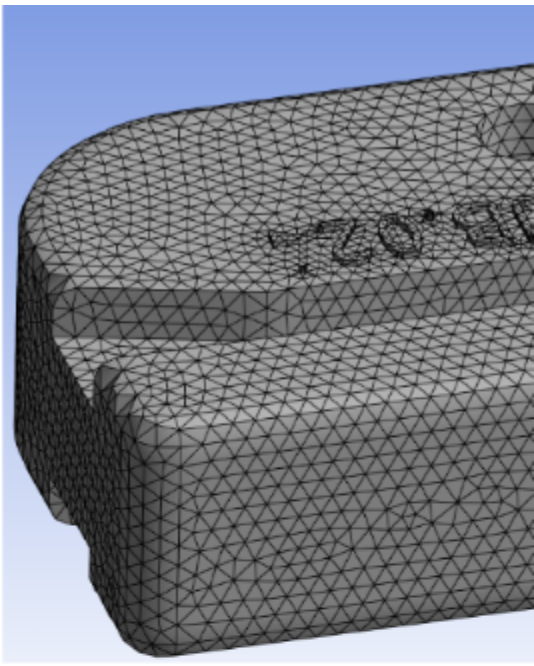
- **Source Scoping Method**: Defines the method for choosing a source face. **Geometry Selection** enables you to select sources or targets manually using the **Source option**. **Named Selection** enables you to choose one **Named Selection** as a source or target using the **Source Named Selection** option.
- **Free Face Mesh Type**: Allows you to select the type of face mesh. The default value is **Quad/Tri**.
  - **Quad/Tri**: Allows you to generate the quad and triangular mesh on the source face.
  - **All Quad**: Allows you to generate all quadrilateral mesh on the source face.
  - **All Tri**: Allows you to generate all triangular mesh on the source face.

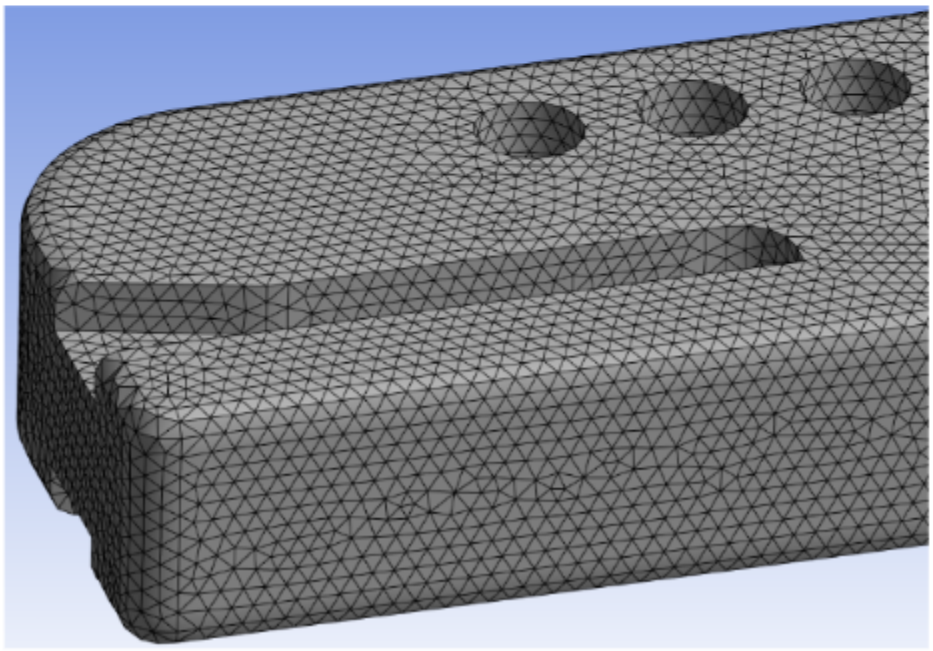
Feature Suppress

Feature Suppress enables you to remove small features at the mesh level. The small features may create bad quality mesh while meshing. Hence, you need to remove such small features to generate a good quality mesh. A typical example of a small feature may be an embossed (or indented) logo as shown below:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.







To access **Feature Suppress** control:

- Right-click **Mesh** object and click **Insert > Feature Suppress**.

  or

- On the Ribbon, click **Feature Suppress** under **Controls** group in **Mesh Context** tab.

When you click **Feature Suppress**, the **Details** view displays the **Feature Suppress** options:

Scope

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

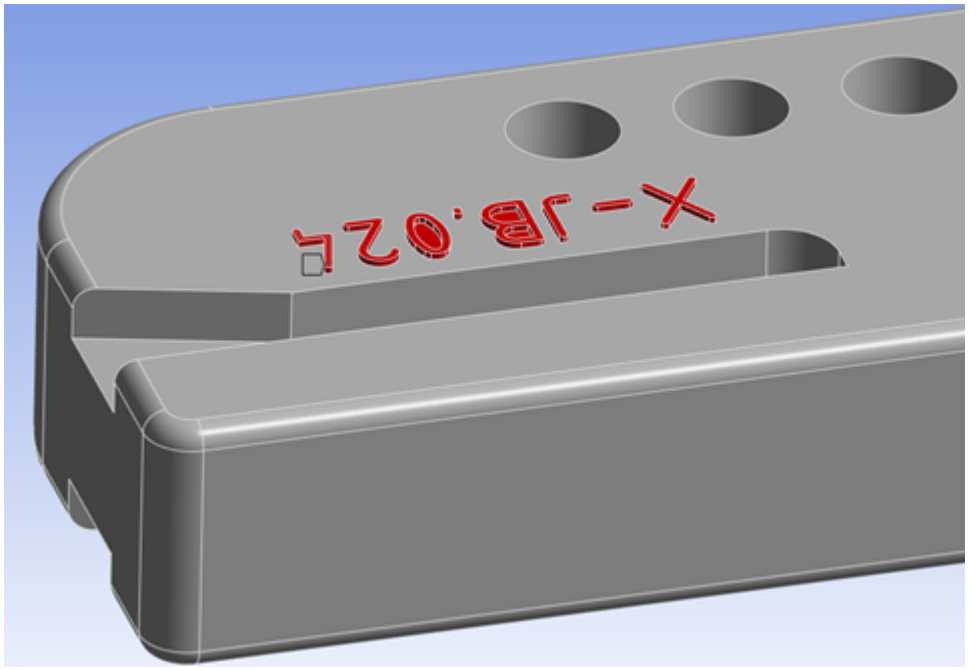
Scoping Method: Allow you to scope the bodies having features that need to be suppressed. The available body scoping methods are **Geometry Selection** and **Named Selection**.

- **Geometry Selection**: Allow you to scope the geometry bodies.
- **Named Selection**: Allow you to scope the geometry bodies to be suppressed in the named selection.

Definition

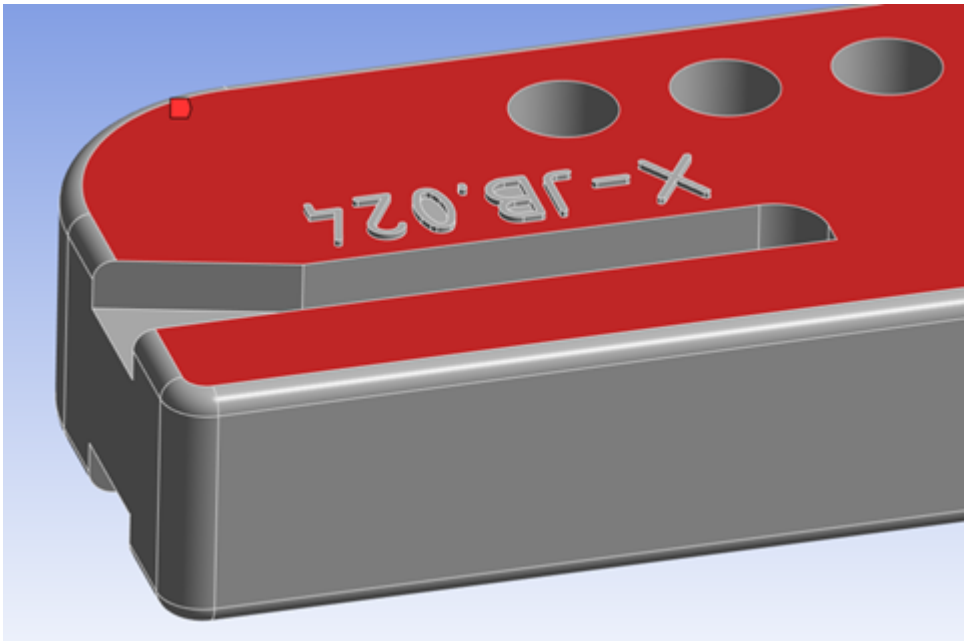
Source Selections: Allow you select the methods used for feature suppression. The available options are **Automatic**, **Defeature Faces** and **Parent Faces**. The default option is **Automatic**.

- **Automatic**: Automatically detect the features to be suppressed on a body. The default value of the **Feature Height** used for feature suppression is also calculated automatically.
- **Defeature Faces**: Allow you to select the faces to be suppressed during meshing.



- **Parent Faces**: Allows you to select the common neighboring face (parent) to the feature faces that needs to be suppressed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Source Scoping Method: Allow you to scope the defeatured faces or parent faces depending on the selected method. The available scoping methods are **Geometry Selection** and **Named Selection**.

- **Geometry Selection**: Allow you to select the defeatured faces or parent faces. When you select **Geometry Selection**, the **Source Geometry** allows you to select the feature to be suppressed from the **Geometry** window.
- **Named Selection**: Allow you to scope the defeatured faces or parent faces in the named selection. When you select **Named Selection**, the **Source Named Selection** allows you to select the feature to be suppressed from the available named selections.

Feature Height: Allows you to specify the height of the feature to be suppressed based on the maximum distance between parent faces and faces considered for suppression. The image below shows the same. You may use a value slightly larger than a locally measured edge length, as the distance between the parent faces and faces considered for suppression can vary for a given feature.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

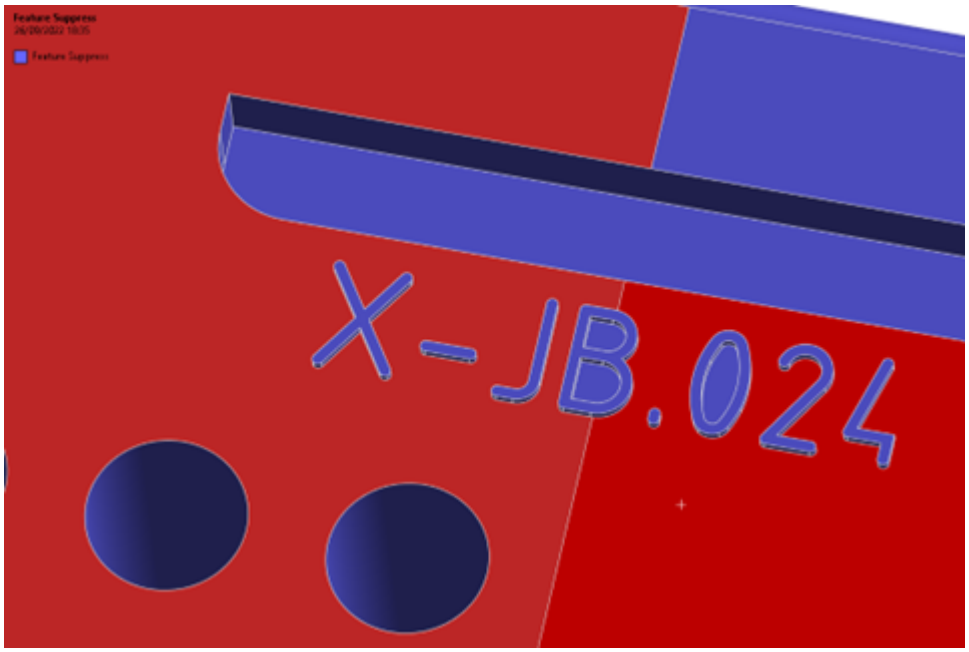
When you select the **Method** as **Automatic** or **Parent Faces**, the **Feature Height** is available. For the **Automatic** method, the default value chosen is 80% of the **Element Size**. For **Parent Faces** method, the default value is –1. It suppresses all features enclosed by the parent face.

Feature Suppress control allow you to preview the suppressed faces using the Show Suppressible Features (p. 539) before suppressing them.

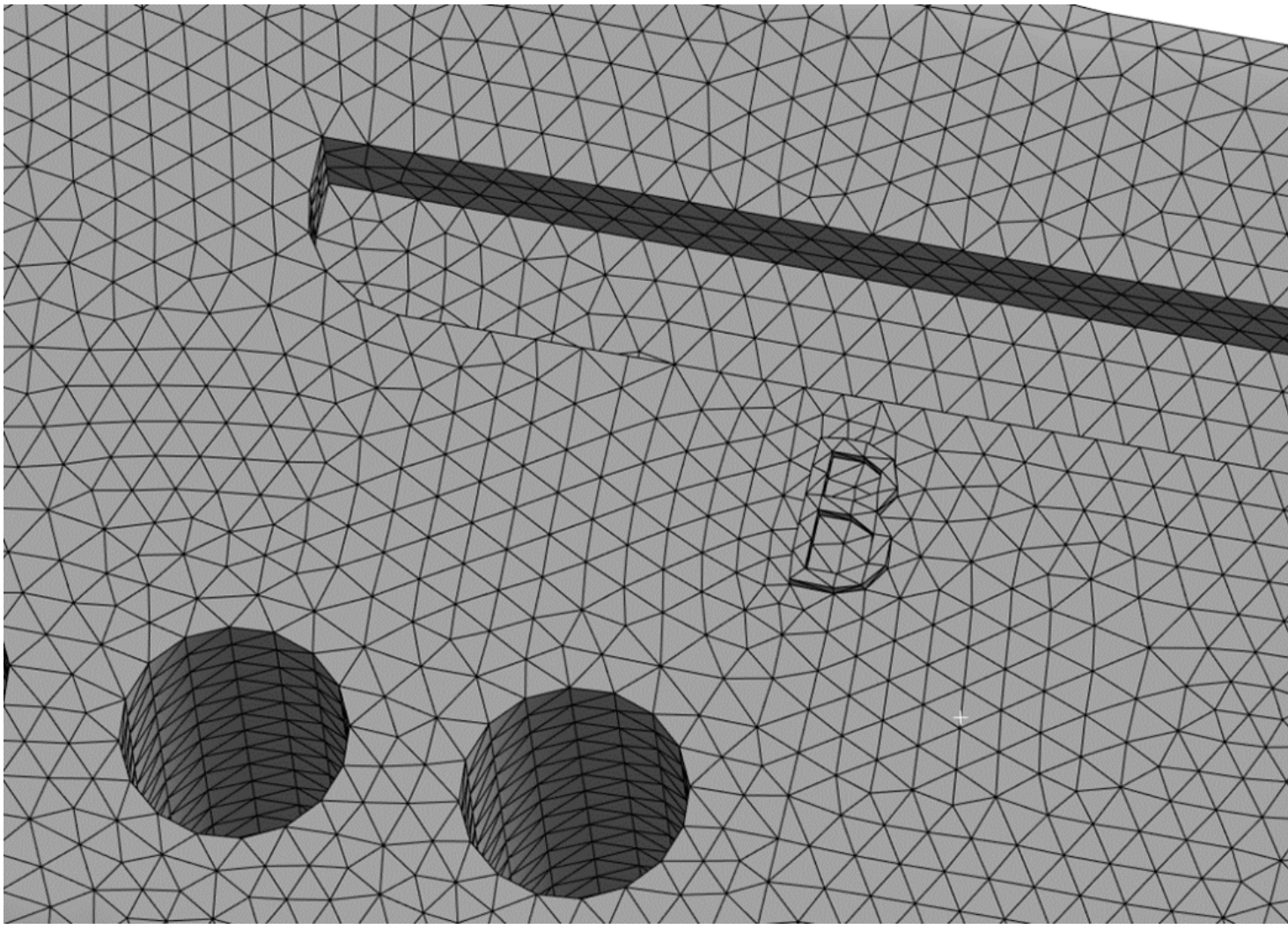
Limitation

Feature Suppress control has the following limitations:

- **Feature Suppress** cannot be applied on areas with protected topology. **Feature Suppress** considers the following as **Protected Topology**:
  - **Named Selection** group with **Protected** set to **Yes**.
  - geometric faces in Face Meshing controls.
  - geometric faces with applied sizing controls.
- **Feature Suppress** does not work if parent faces are split around the feature to be removed. You cannot use **Virtual Topology** to merge such faces and you can merge the faces at the geometry level using geometry modeling tool such as Discovery or Discovery Modeling.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Feature Suppress** supports only solid bodies and not sheet bodies.
- **Feature Suppress** supports only **Patch Conforming Tetrahedron** and **Hex Dominant** mesh methods.
- **Feature Suppress** does not support **Virtual Topology**.

Repair Topology

Repair Topology allows you to repair topologies of the scoped entities.

All **Repair Topology** bodies should be scoped under **Automatic (PrimeMesh)** method. If not, scope the bodies under **Automatic (PrimeMesh)** to perform **Repair Topology** operation.

To Access Repair Topology:

1. Right-click the **Mesh** folder in the Tree Outline.
2. Select **Insert > Repair Topology**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following options are available in the **Details** view for **Repair Topology**:

Definition

- **Suppressed**: Allows you to suppress the **Repair Topology** control. The default value is **No**. When **Suppressed** is set to **Yes**, the **Active** field displays the status of the **Repair Topology** control. The **Active** field is read-only.

Merge Faces Options

- **Merge Faces**: Merges the selected faces. The default value is **No**. When the value is set to **Yes**, the following option appears:

  – **Scoping**: Allows you to select either **Geometry Selection** or **Named Selection**. The default value is **Geometry Selection**. When you select **Geometry Selection**, **Merge Faces Geometry** allows you to scope the face geometry. When you select **Named Selection**, you can select multiple named selections from the list of available face named selections. After selecting, you must press **Enter** to apply the selection. Based on the **Scoping** selected, the number of faces used for the respective selection is displayed.

Suppress Edges Options

- **Suppress Edges**: Suppress the selected edges. The default value is **No**. When the default value is set to **Yes**, the **Scoping** allows you to scope **Geometry Selection** or **Named Selection**. The default value is **Geometry Selection**. When you select **Geometry Selection**, **Suppress Edges Geometry** allows you to scope geometry selection. When you select **Named Selection**, allows you to scope the named selection.

Partial Defeature Options

- **Partial Defeature**: Allows you to partially merge the geometry by suppressing only a portion of the edge joining two faces based on **Thin Face Width**, **Sharp Angle** and **Feature Angle**. **Partial Defeature** is useful for retaining important geometric features without merging the faces completely at thin faces or sharp angles.

  When you set **Partial Defeature** to **Yes**, the mesher automatically sets the **Remove Thin Faces** and **Remove Sharp Angle** to **Yes** by default, and you cannot update **Remove Thin Faces** and **Remove Sharp Angle**. When **Partial Defeature** set to **Yes**, the available option is:

  – **Scoping**: Allows you to select the scoping method. The default value is **Geometry Selection**. **Scoping** allows you to select **Geometry Selection** or **Named Selection**. When you select **Geometry Selection**, **Partial Defeature Geometry** allows you to scope the thin faces in the geometry. When you select **Named Selection**, allows you to scope the thin faces in the named selection. **Partial Defeature** scoping is used for **Remove Thin Face Width** and **Remove Sharp Angle options**.

Repair Thin Faces Options

- **Remove Thin Faces**: Removes the thin faces by merging them with the neighboring faces. The default value is **No**. When **Remove Thin Faces** is**Yes**, the **Thin Face Width** and **Scoping** options are displayed.

  – **Thin Face Width**: Allows you to specify the width of the thin face. The default value for **Thin Face Width** is same as the **Connection Tolerance**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Scoping**: Allows you to select the scoping method. The default value is **Geometry Selection**. **Scoping** allows you to select **Geometry Selection** or **Named Selection**. When you select **Geometry Selection**, **Thin Face Geometry** allows you to scope the thin faces in the geometry. When you select **Named Selection,** allows you to scope the thin faces in the named selection. **Scoping** is available only when **Partial Defeature** is set to **No**.

Repair Short Edges Options

- **Collapse Short Edges**: Collapses the edges below the specified tolerance. Feature edges and Protected edges are not collapsed. The default value is **No**. When the **Collapse Short Edges** is set to **Yes**, the **Short Edge Length** and **Scoping**option are displayed.

  – **Short Edge Length**: Allows you to specify the shortest edge length to performs collapse operation.

  – **Scoping**: Allows you to select the scoping method. The default value is **Geometry Selection**. When **Scoping** is **Geometry Selection**, the **Short Edge Geometry** is displayed that allows you to apply the scoping to the selected edges. When **Scoping** is **Named Selection**, allows you to select the available named selections. **Scoping** is available only when **Partial Defeature** is set to **No**.

Repair Sharp Angle Options

- **Remove Sharp Angle Faces**: Removes the sharp angle faces below the specified angle tolerance by merging them to the neighboring faces. The default value is **No**. When the value is set to **Yes**, the **Sharp Angle** and **Scoping** options appear.

  – **Sharp Angle**: Allows you to capture faces below the specified angle. The default value for **Sharp Angle** is 10 degrees.

  – **Scoping**: Allows you to select the scoping method. The default value is **Geometry Selection**. When **Scoping** is **Geometry Selection**, the **Sharp Angle Geometry** allows you to apply the scoping to the selected faces. When **Scoping** is **Named Selection**, allows you to select the available named selections.

Repair Pinch Faces Options

- **Pinch Faces**: Removes small features at the mesh level. The default value is **No**. When set to **Yes**, the **Pinch Tolerance** and **Scoping** options appear.

  – **Pinch Tolerance**: Allows you to pinch the faces below the specified tolerance. The default value of **Pinch Tolerance** is same as the **Connection Tolerance**.

  – **Scoping**: Allows you to select the scoping method. The default value is **Geometry Selection**. When **Scoping** is **Geometry Selection**, the **Pinch Faces Geometry** allows you to apply the scoping to the selected faces. When **Scoping** is **Named Selection**, allows you to select the available named selections.

Fill Hole Options

- **Fill Hole**: Fills the selected holes. The default value is **No**. When set to **Yes**, **Scoping** appears and allows you to select either **Geometry Selection** or **Named Selection**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

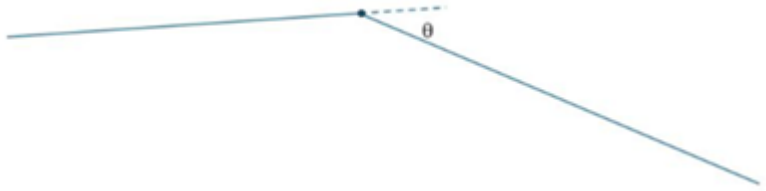
– **Geometry Selection**: Allows you to scope only edges for fill hole option. The default value is **Geometry Selection**.

– **Named Selection**: Allows you to scope edge-based name selection for fill hole option.

Advanced

- **Feature Angle**: Specifies the minimum dihedral angle at which the geometry features are repaired. The default value is **30 degrees**. You can provide any value between 0 to 180 degrees.

  Feature Angle is the dihedral angle measured as follows:

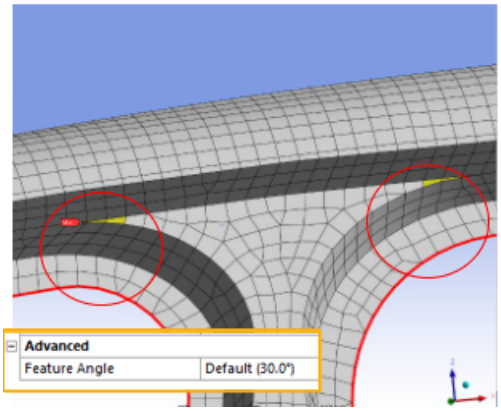


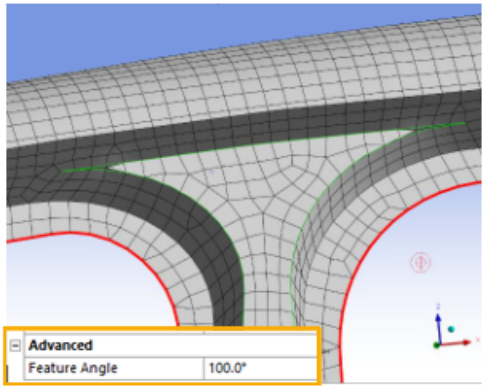
  When θ**=0**, two surfaces are perfectly tangential, and the edges are not protected. Hence, the two surfaces can be merged, or edges can be suppressed, pinched or more.

  When **30** $\leq \theta \leq$ **90** degrees, the edges between the two surfaces are protected and cannot perform merging or pinching at the location.

  θ = 90

  When θ **> 90** degrees,the surfaces are not protected and prevents topology operations like merge or pinch on faces or edges.





Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can right-click the **Mesh**, select **Diagnostics > Find Unconnected Edges** to display the unconnected edges. You can also access the overlapping faces by selecting **Diagnostics > Find Overlapping Faces**.

Note:

- **Repair Topology** operations are performed during meshing. Hence, the changes are not reflected on the input geometry. You should carefully examine mesh association to geometry when using repair topology.
- **Repair Topology** operations respects **Protected Entities**. Hence, **Repair Topology** options do not alter **Protected Entities**.

Connect

Connect control allows you to create conformal mesh between the selected entities.

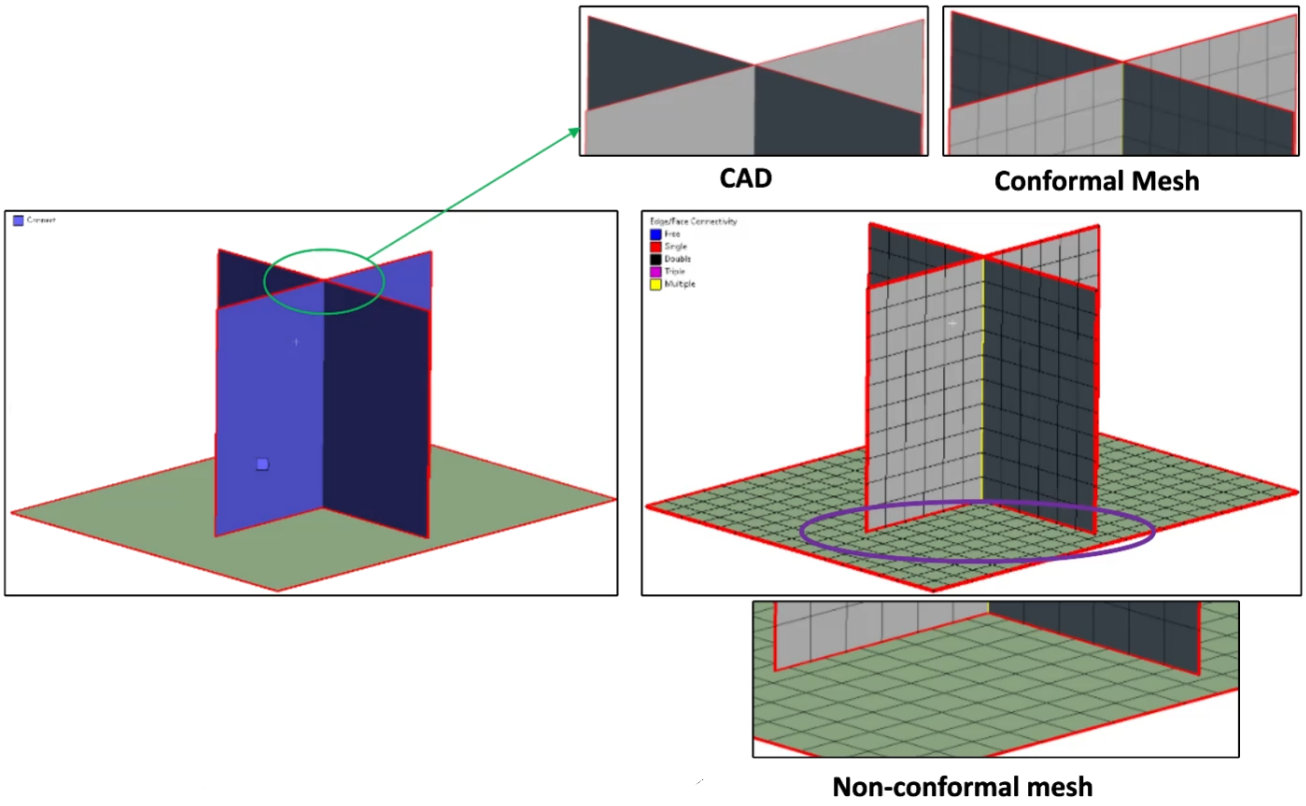
All **Connect** bodies should be scoped under **Automatic (PrimeMesh)** method. If not, scope the bodies under **Automatic (PrimeMesh)** to perform **Connect** operation.

To access **Connect** control, right-click **Mesh** object and click **Insert > Connect**. When you click **Connect**, the **Details** view displays the **Connect** options.

Details of "Connect" - Connect

Scope	
Scoping Method	Geometry Selection
Geometry	7 Faces
Use Worksheet	No
Definition	
Suppressed	No
Multiple Connection Steps	No
Connection Tolerance	Default (4.8848e-004 m)
Connection Size	Default (4.8848e-003 m)
Connection Option	All To All
Coplanar Angle Tolerance	Default (25.0°)
Perform Intersections	Yes

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

- **Connect** control does not support scoping to solid entities.
- All legacy databases saved with **Batch Connections** will resume with **Connect** control.

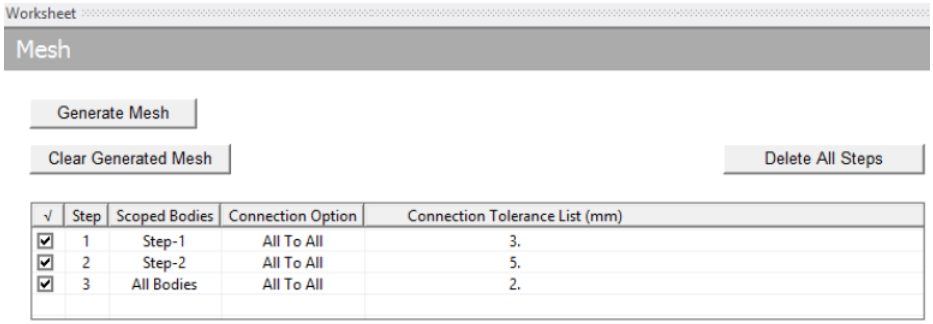
Scope

- **Scoping Method**: Allows you to select **Geometry** or **Named Selection** for scoping.
- **Use Worksheet**: Allows you to scope using worksheet when set to **Yes**. The default value is **No**. When **Use Worksheet** is set to **Yes**, the connections are made using worksheet.

  Connections Using Worksheet

  Connections can also be performed using **Worksheet**. **Worksheet** allows you to define the order in which connections are performed and allows you to define varying connection parameters. **Worksheet** is context sensitive. To perform connections using **Worksheet**, you must be in the **Mesh** folder. Click **Worksheet** in **Tools** under the **Home** tab to open the **Worksheet**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

You can activate, setup and visualize mesh worksheet only from the Mesh folder.

Worksheet allows you to scope named selections. Named selections can either be body based or face based. For each step you can change connection option and specify connection tolerance(s). The available connection options are:

- **All to All**: Connects all possible entities in the scope during the connection process. When **All to All** is selected, sliver faces and short edges below the connection tolerance value are removed as a part of the connection process.
- **Free to All**: Connects only unconnected edges to rest of the entities in the scope.
- **Free to Free**: Connects only unconnected edges in the scope.

The steps involved in **Worksheet** based connection are as follows:

1. Right- click the **Worksheet**.
2. Click **Add** to add a step.
3. Specify scope defined by named selection from the drop-down list in the **Scoped Bodies** field. **All Bodies** is a named selection available by default.
4. Select the **Connection Option**.
5. Specify connection tolerance or list of tolerances for the step. The connection tolerance(s) behaves same as global connection tolerance and list of tolerance.
6. Add as many steps as required for the connection.

When **Generate Mesh** is clicked connections are established in the sequence defined in the worksheet. The mesh is generated after all the connections are made.

Note:

All Bodies is available by default under scoped bodies in the **Worksheet**. This named selection is scoped to all bodies in the assembly.

Definition

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Suppressed**: Allows you to suppress the **Connect** control. The default value is **No**. When **Suppressed** is set to **Yes**, the **Active** field displays the status of the **Connect** control. The **Active** field is read-only.
- **Multiple Connection Steps**: Allows you to provide multiple values for connection tolerance. The default value is **No**. When you set **Multiple Connection Steps** to **No**, **Connection Tolerance** field is available to provide the tolerance value for connection. When you set the **Multiple Connection Steps** to **Yes**, the **Connection Tolerance List** is available to provide multiple values for connection tolerance. You can specify any number of connection tolerance values separated by a space. The first value in **Connection Tolerance List** performs face to face intersections, short edge removal, thin face removal and so on. From the second tolerance value onwards only the unconnected (free) edges are considered for performing connections.
- **Connection Size**: Defines the size with which you discretize the edges before connecting them. By default, the connection size is same as **Element Size**.
- **Connection Option**: Allows you to select the connect options. The available connection options are **All to All**, **Free to All**, **Free to Free**. For more details, check Connections Using Worksheet (p. ?)
- **Coplanar Angle Tolerance**: Checks whether the two faces to be connected are in the same plane or not. When the faces are in the same plane, intersection is not performed. The value of **Coplanar Angle Tolerance** is in degrees.
- **Perform Intersections**: Allows you to skip intersection while connecting the selected entities. The default value is **Yes**.

Note:

When you use **Connect and Mesh Selected Entities** on legacy files which have partially saved mesh, the previously saved mesh gets cleared. You may have to regenerate mesh for the entire model.

Weld

Weld control allows you to create weld bodies and, or generate layers of quad elements on the weld faces (angled and normal faces), in between the angled and normal faces and on the faces that are welded together along the edges shared with the angled and normal faces. When you select **Source** as **Mesh** and **Create using** options like **Curves**, **Curves and Bodies** or **Curves and Faces**, angled and normal faces are created during meshing using defined **Weld Curve**. The faces are associated to a weld body that is created during meshing. The weld body is available under the **Geometry** tree object. The weld body name is same as that of the **Weld** control. One weld body gets created per **Weld Curve**. **Weld** supports coincident welds and solid shell welds. In **Files > Options > Meshing**, when **Solid-Shell Weld – Shared Nodes** is **No** under **Meshing**, creates non conformal mesh with contacts between the shell and the weld bodies. When **Solid-Shell Weld – Shared Nodes** is **Yes**, creates conformal mesh without contacts between the shell and the weld bodies. The default value is **No**.

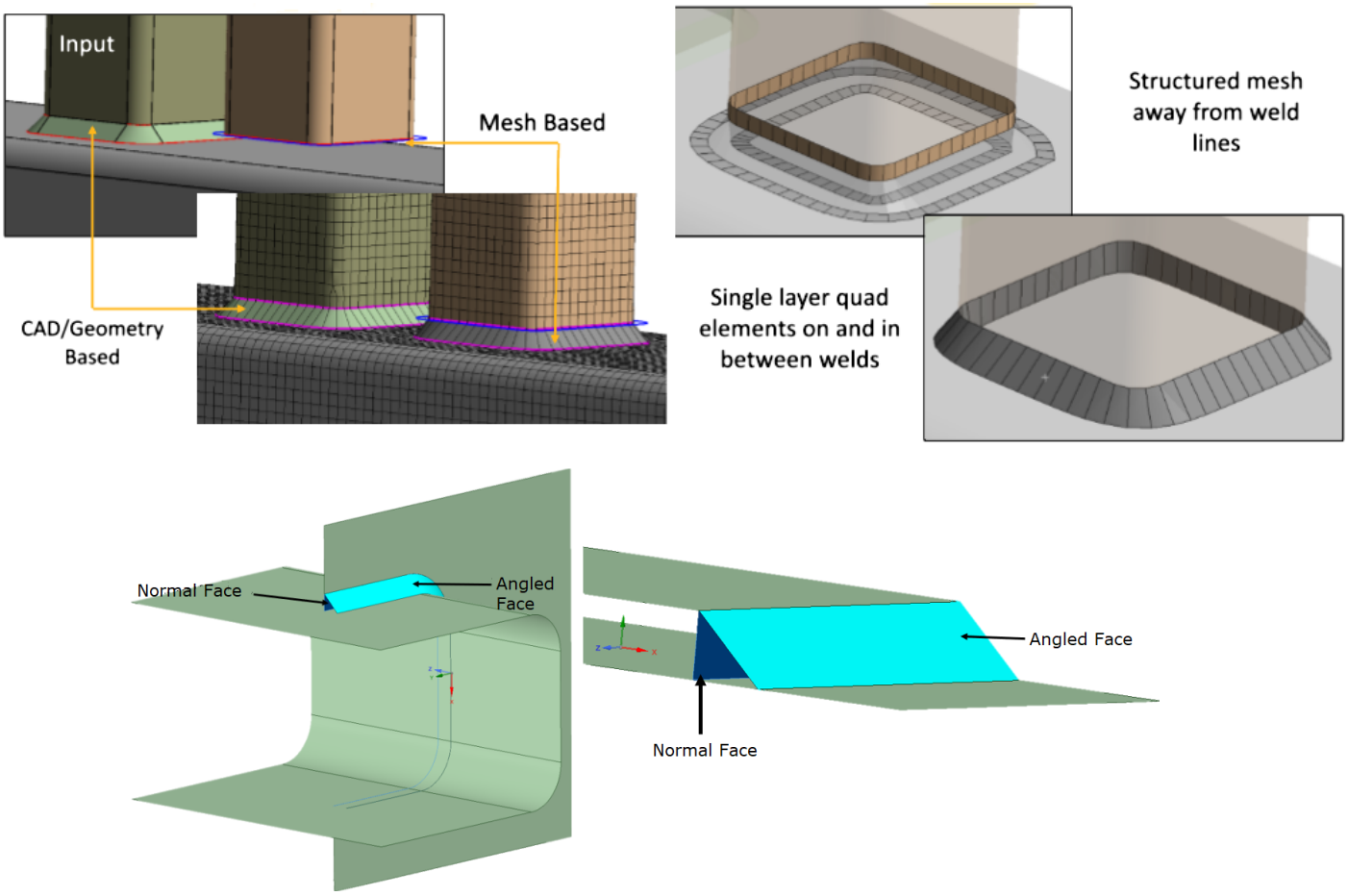
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

All **Weld** bodies should be scoped under **Automatic (PrimeMesh)** method. If not, scope the bodies under **Automatic (PrimeMesh)** to perform **Weld** operation.

Note:

Weld bodies created at the mesh level are deleted, when the following occurs:

- Mesh is cleared.
- Geometry is updated.



You can access the weld control as follows:

1. On the Tree Outline, click **Mesh** object.
2. Right-click the **Mesh** object in the Tree Outline.
3. Select **Insert > Weld**.

Also, you can access **Weld** from **Mesh Context** tab when you click **Mesh** object on the Tree Outline.

When you have a defined **Weld** object on the **Tree**, you can right-click **Mesh** and select **Export... > Weld Definition File** to write the FE-Safe weld definition files at the provided location.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

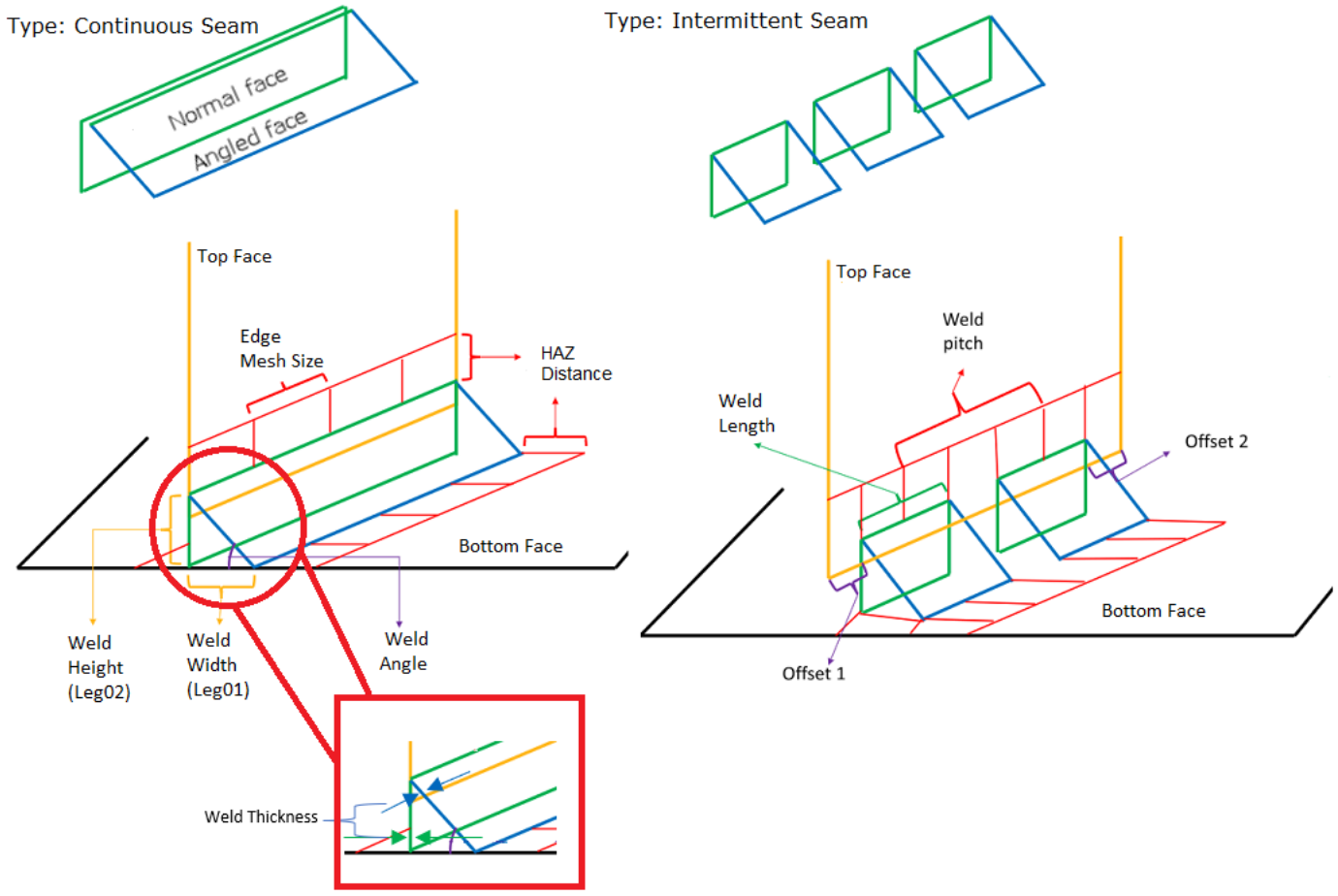
Weld Details view has the following options:

Scope

- **Scoping Method**: You can use **Geometry Selection** or **Named Selections** for scoping. The default value is **Geometry Selection**. **Scoping Method** is not available only when the **Create Using** is set to **Curves**.
- **Type**: Allows you to select the type of weld control. The available types are:
  - **Continuous Seam**: Allows you to perform continuous welding.
  - **Intermittent Seam**: Allows you to perform discontinuous welding. When you select **Type** as **Intermittent Seam**, the **Source** is **Mesh** by default and is a read only field.

Note:

Intermittent Seam modeled as **Mesh Independent** is treated as **Continuous Seam** as it is in contact with solids.



- **Source**: Allows you to select **Geometry** or **Mesh** for welding. **Source** represents where the weld faces are modeled. You can select **Geometry**, for weld faces created in CAD. You can select **Mesh**, when the weld entities are to be created during mesh generation.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Modeled As**: Allows you to choose or define how weld entities are modeled. When the **Source** is **Geometry** and **Type** is **Continuous Seam**, you need to define what the weld faces represent. The available options are:

  – **Angled**: Creates mesh only on the angled face for the weld.

  – **Normal**: Creates mesh only on the normal face for the weld.

  – **Normal and Angled**: Creates mesh on the angled or normal faces with layers of quad mesh on both angled and normal with the user defined HAZ distance on top and bottom faces. When you select **Normal and Angled**, the **Normal Surface** and **Angled Surfaces** fields are available.

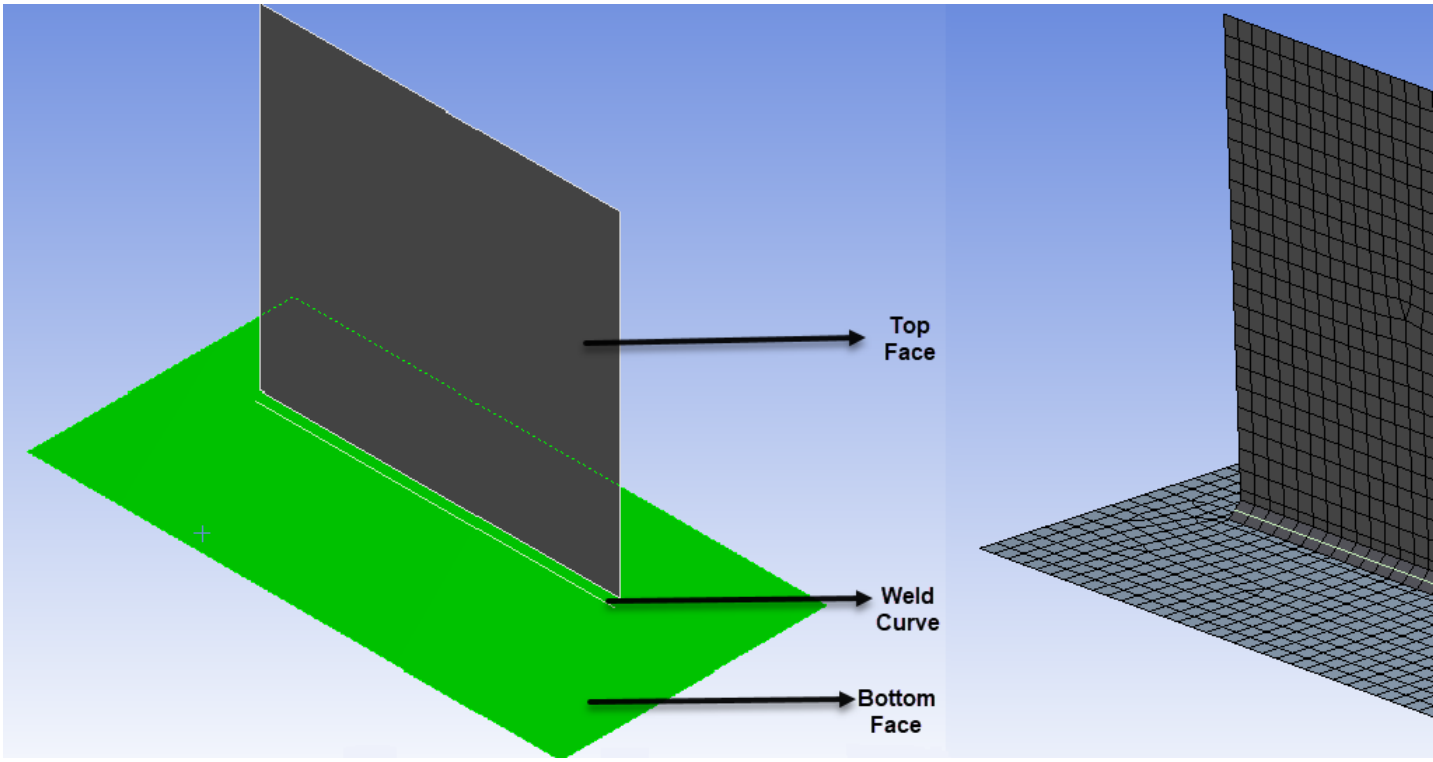
  – **Parent Bodies Connection**: Connects the top and bottom faces without creating quad layers on the angled or normal face. When you select **Parent Bodies Connection**, the **Bottom Faces** and **Top Faces** fields are available.

  When you select the **Source** as **Mesh**, the available options are:

  – **Normal and Angled**: Creates both angled and normal face for the weld.

  – **Angled**: Creates only the angled face for the weld.

  – **Normal**: Creates only the normal face for the weld.

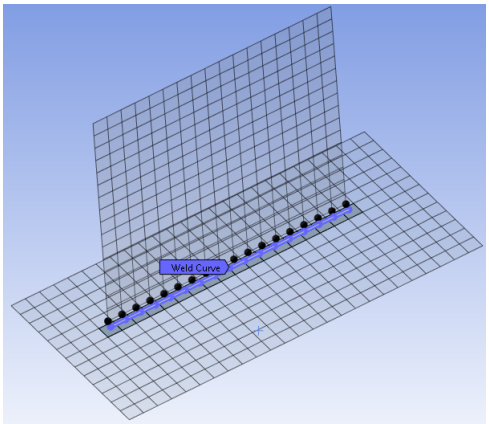


Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **1D**: Creates normal only weld using beams. Here beams are conformally connected to the top and bottom faces. For input edge or curves, the beams created are represented as a single body. **1D** is available only when the **Source** is **Mesh**.

> **Note:**
>
> **1D** does not support **Weld Element Rows**.

– **Mesh Independent**: Creates mesh without weld faces or HAZ layers. **Mesh Independent** is available only when the **Source** is **Mesh**. Nodes on the top faces are attached to the elements on the bottom face which act as contact points. To view the element faces attached to the nodes, click **Show Mesh** in the **Display** tab after generating mesh. **Mesh Independent** supports Solid-Solid weld creation.



- **Weld Element Rows**: Allows you to split elements from weld faces. You can provide the number of splits required. The default value is 1. The value ranges from 1 to 2.
- **Create Using**: Allows you to create weld using **Curves**, **Curves and Bodies**, **Curves and Faces**, **Edges**, **Edges and Bodies** or **Edges and Faces**. Available only when the **Source** is **Mesh**.
  – **Curves**: Allows you to pick a beam body and use it as weld curve. Software automatically determines the faces in proximity of the defined weld curve to be welded. When you select **Curves**, the **Weld Curve** option is available.
  – **Curves and Bodies**: Allows you to define bodies to be welded together in addition to the weld curve. When you select **Curves and Bodies**, the **Bottom Bodies**, **Top Bodies**, **Weld Curve** options are available.
  – **Curves and Faces**: Allows you to define faces to be welded together in addition to the weld curve. This option gives you better control on defining the scope for weld creation during mesh generation. When you select **Curves and Faces**, the **Bottom Faces**, **Top Faces**, **Weld Curve** options are available.
  – **Edges**: Allows you to scope geometry edges to create welds. The edges scoped to a weld control or in a row of worksheet must be continuous. When you select **Edges**, the **Edges** option is available.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Edges and Bodies**: Allows you to define bodies to be welded together in addition to the geometry edges. When you select **Edges and Bodies**, the **Bottom Bodies**, **Top Bodies**, **Edges** options are available.
  - **Edges and Faces**: Allows you to define faces to be welded together in addition to the geometry edges. This option gives you better control on defining the scope for weld creation during mesh generation. When you select **Edges and Faces**, the **Bottom Faces**, **Top Faces**, **Edges** options are available.

Note:

If the **Curves** or **Edges** option is providing incorrect result for weld creation, then using **Curves and Faces** or **Edges and Faces** is recommended.

- **Angled Direction**: Allows you to specify the direction of the angle creation. The available options are **Normal**, **Reversed** and **Both**.
  - **Normal**: Allows you to create an angled face in the same direction of the face normal direct for edges-based inputs and along the weld curve for curve-based input.
  - **Reversed**: Allows you to create an angled face in the opposite direction of the face normal direct for edge-based input and opposite to the weld curve for curve-based input.
  - **Both**: Allows you to create an angled face along both the same direction and opposite direction of the face normal direct for edge-based inputs and both the same and opposite sides of the weld curve for curve-based input.
- **Use Worksheet**: Allows you to use worksheet for weld controls. **Worksheet** enables you to combine multiple weld controls in a single weld control. The default value is **No**. For more details, see Weld Worksheet (p. 376).
- **Curve Scoping**: Available only when the **Source** is **Mesh** and **Creating Using** is set to either **Curves**, **Curves and Bodies** or **Curves and Faces**. The available options are **Geometry Selection** and **Body Selection**. The default value is **Geometry Selection**. **Body Selection** allows you to select the weld curve from the list of beam bodies.

Note:

When you use beam bodies as weld curve, **Treatment** is automatically set to **Construction Body**. **Treatment** does not reset automatically, when the beam body is removed from the **Weld Curve** scope.

Definition

- **Suppressed**: Suppress the weld control. The default value is **No**. When the value is set to **Yes**, the **Active** options appears with a read-only setting **No,Suppressed**.
- **Adjust Weld Height**: Allows you to adjust the weld height. The default value is **No**. When the **Adjust Weld Height** is set to **Yes**, the **Weld Height (Leg02) Assignment** is available.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Weld Height (Leg02) Assignment**: Allows you to select the weld height assignment. The available options for **Weld Height (Leg02) Assignment** are **User Defined** and **Expression**.

→ **User Defined**: Allows you to define the weld height. When you select **User Defined**, the **Weld Height (Leg02)** is available. **Weld Height (Leg02)** allows you to define the height of the weld created during mesh generation. The default value is calculated based on the average of the sheet thicknesses of the bodies connected by the weld. If the sheet thicknesses of the bodies are not available, no height adjustment is performed during mesh generation.

→ **Expression**: Allows you to calculate the weld height based on the expression. When you select **Expression**, the **Weld Height (Leg02) Expression** allows you to calculate the weld height based on the average thickness of the up and down faces. The **Weld Height (Leg02) Expression** should be based on t1 and t2, where t1 is the average thickness of all up faces and t2 is the average thickness of all down faces. **Weld Height (Leg02) Expression** allows only variables t1 and t2 as input and does not allow other variables.

Note:

Variables t1 and t2 must be in lower case. Expression supports min, max and scientific numbering. Expression does not support any special characters other than the following mathematical operators like '+', '-', '/', '*', '^', '(', ')'. For Expression, you must specify constants in geometric unit and not in Workbench display units.

- **Weld Length**: Allows you to define the length of each weld in the intermittent seam. The **Weld Length** is available only when the selected **Type** is **Intermittent Seam**. There is no default value for weld length. Thus, you must define the weld length manually.
- **Weld Pitch**: Allows you to define the pitch of the weld. **Weld Pitch** is the distance from mid-point of one weld to the mid-point of adjacent weld in an intermittent seam weld. **Weld Pitch** is therefore the sum of the weld length and the gap between each of the welds. There is no default value for **Weld Pitch**. Therefore, you must define the **Weld Pitch** manually. **Weld Pitch** must be greater than **Weld Length**.
- **Number of Welds**: Allows you to define the number of intermittent seam welds on the **Weld Curve** or **Edge**. You can also specify the number of intermittent seam welds. The intermittent seam welds are created based on the specified **Weld Length** and **Weld Pitch**. Once the defined **Number of Welds** is achieved, no further welds are created on the scoped **Weld Curve** or **Edges**.

Note:

Weld Pitch and **Number of Welds** are optional. You can either provide **Weld Pitch** or **Number of Welds** value while performing weld operation.

- **Offset 1**: Allows you to specify the distance between the start vertex and the first intermittent seam weld. The default value is 0.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Offset 2**: Allows you to specify the distance between the last intermittent seam weld and the end vertex in the scoped **Weld Curve** or **Edges**. The default value is 0.

- **Creation Criteria**: Allows you to define the criteria for the weld creation. The field is only available when the **Source** is **Mesh**. The default value is **Width Based**. The available options are:

  – **Angle Based**: Allows you to control the angle between the angled face and bottom face. When you select **Angle Based** for creating weld, the **Weld Angle** appears. The default value is 45 degrees. You can select any value from 30 to 60 degrees for Weld Angle.

  Note:

  Normal face is always created perpendicular to the bottom face.

  – **Width Based**: Allows you to create welds of constant width along the weld line. When you select **Width Based** for creating weld, the available option is **Weld Width (Leg01) Assignment**. **Weld Width (Leg01) Assignment** allows you to select the type of weld width assignment. The available options are **User Defined** and **Expression**. The default value is **User Defined**.

  → **User Defined**: Allows you to provide the weld width. When you select **User Defined**, the **Weld Width (Leg01)** is set to **Default**. The **Default** value for **Weld Width (Leg01)** is equal to **Weld Height (Leg02)**. If the **Weld Height (Leg02)** is not available, the **Default** is calculated as the default **Angle Based**.

  → **Expression**: Allows you to calculate the weld width based on the **Weld Width (Leg01) Expression**. When you select **Expression**, the **Weld Width (Leg01) Expression** calculates the weld width based on the expression. The expression should be based on t1 and t2, where t1 is the average thickness of all up faces and t2 is the average thickness of all down faces. Expression allows only variables t1 and t2 as input and does not allow other variables.

  Note:

  Variables t1 and t2 must be in lower case. Expression supports min, max and scientific numbering. Expression does not support any special characters other than the following mathematical operators like '+', '-', '/', '*', '^', '(', ')'. For Expression, you must specify constants in geometric unit and not in Workbench display units.

- **Edge Mesh Size Assignment**: Allows you to select the edge mesh size assignment. The available options are **User Defined** and **Expression**. The default is **User Defined**.

  – **User Defined**: Allows you to define the mesh size used to discretize the edges defined as weld lines. The default value for **Edge Mesh Size** is same as the global **Element Size**.

  – **Expression**: Allows you to calculate the edge mesh size based on the Edge Mesh Size Expression. **Edge Mesh Size Expression** calculates the edge mesh size based on the average thickness

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of the up and down faces. The expression must have only t1 and t2 as variables, where t1 is the average thickness of all up faces and t2 is the average thickness of all down faces.

Note:

Variables t1 and t2 must be in lower case. Expression supports min, max and scientific numbering. Expression does not support any special characters other than the following mathematical operators '+', '-', '/', '*', '^''(', ')'. For Expression, you must specify constants in geometric unit and not in Workbench display units.

Note:

The **Edge Mesh Size** should be reduced for welds involving high curvature.

- **Create HAZ Layer**: Allows you to create the HAZ layer for the weld control when set to **Yes**. The default value is **Yes**. When **Create HAZ Layer** is set to **Yes**, the following options are available:
  - **HAZ Distance Option**: Allows you to define the HAZ Distance. The default value is the **Global Element Size**.The available options are **Distance Per HAZ** and **Total Distance**.
    - → **Distance Per HAZ**: Allows you to specify the height for each quad layer.
    - → **Total Distance**: Allows you to provide the total distance for which the HAZ layers are created. Here, **Total Distance** by **Number of HAZ** along with **HAZ Growth Rate** determines the height of each quad layer.
  - **HAZ Distance Assignment**: Allows you to select the type of HAZ distance assignment. When you set **Create HAZ Layer** to **Yes** , the **HAZ Distance Assignment** is available with **User Defined** option as default. The available options are **User Defined** and **Expression**.
    - → **User Defined**: Allows you to provide the **HAZ Distance**. The **HAZ Distance** allows you to define the height for each quad layer. The default value is the **Global Element Size**. When you select **User Defined**, the **HAZ Distance Top Plate** allows you to specify the height of each quad layer for the top plate based on the selected **HAZ Distance Option**. **HAZ Distance Bottom Plate** allows you to specify the height of each quad layer for the bottom plate based on the selected **HAZ Distance Option**.**HAZ Distance Top Plate** and **HAZ Distance Bottom Plate** can be parameterized.
    - → **Expression**: Allows you to calculate the HAZ distance based on the expression. The **HAZ Distance Expression Top Plate** allows you to calculate HAZ distance for top plate based on the expression. The **HAZ Distance Expression Bottom Plate** allows you to calculate HAZ distance for bottom plate based on the expression. The **HAZ Distance Expression Top Plate** and **HAZ Distance Expression Bottom Plate** should be based on t1 and t2, where t1 is the average thickness of all up faces and t2 is the average thickness of all down faces.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HAZ Distance Expression Top Plate and **HAZ Distance Expression Bottom Plate** allow only variables t1 and t2 as input and do not allow other variables.

Note:

Variables t1 and t2 must be in lower case. Expression supports min, max and scientific numbering. Expression does not support any special characters other than the following mathematical operators like '+', '-', '/', '*', '^', '(', ')'. For Expression, you must specify constants in geometric unit and not in Workbench display units.

- **Number Of HAZ**: Allows you to define the number of quad layers generated from the weld lines. The default value for **Number Of HAZ** is **1** and the maximum number of layers that can be generated is **3**. When **Number Of HAZ** is greater than one, **Layer Transition Type** allows you to specify the type of mesh elements to be created during the layer transition.
- **HAZ Growth Rate**: Allows you to define the rate at which the HAZ layer grows. The default value is 1.2

- **Generate End-Caps**: Allows you to generate triangular end-caps at the free ends of the welds. The default value is **Yes**.

Note:

- End-caps are not created when **Modeled As** is set to **Normal**.
- End-caps are created when **Modeled As** is set to **Angled**. End-caps are usually created in plane of an angled face. However, if there is no enough space to create in-plane end-caps, then they can be created out of angled face plane.

- **Generate Named Selection**: Creates FE named selection. The default value is **No**. The available options are:
  - **Weld**: Creates FE named selection on the weld body for the respective mesh control. This named selection is added under Named Selection tree-node with the name same as that of the weld control. The default value is **No**.
  - **HAZ Layer 1**: Creates named selection on mesh elements for the first offset layer.
  - **HAZ Layer 2**: Creates named selection on mesh elements for the second offset layer.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **HAZ Layer 3**: Creates named selection on mesh elements for the third offset layer.

Note:

The **HAZ Layer** option created depends on the **Number Of HAZ**. That is, if the **Number of HAZ** is set to 1, only **HAZ Layer 1** option is created. The **HAZ Layer 2** and **HAZ Layer 3** options are not created when the **Number of HAZ** is 1.

Note:

You can parameterize **Edge Mesh Size**, **Weld Height (Leg02)**, **Weld Pitch**, **Weld Width (Leg01)**,**Number of Welds**, **Offset 1**, **Offset 2**, **HAZ Distance**, **Layer Height**, **Weld Angle**, **Weld Length** only when **Use Worksheet** is set to **No.** For more information, see Working with Meshing Application Parameters (p. 87).

Mechanical Properties

- **Material**: Allows you to define the material for the weld bodies created during meshing. You can only select the defined materials for weld bodies from the drop-down menu. The default option is **None**.
- **Cross Section**: Allows you to assign cross section to weld bodies **Modeled As 1D**. This option is available only when **Modeled As** is set to **1D**.
- **Thickness Assignment**: Allows you to select different options to define the thickness of weld bodies created during meshing. The available options are:
  - **Program Controlled**: Allows the application to control the thickness of the weld bodies during meshing. The default value is Program Controlled. Here thickness is calculated as follows:

    Weld Thickness = (Average thickness of top and bottom weld bodies) / $\sqrt{2}$
  - **Min Parent Thickness Factor**: Allows you to specify the minimum thickness factor for weld bodies during meshing. Here thickness is calculated as follows:

    Weld Thickness= Min Parent Thickness Factor * min (Average thickness of all top and bottom faces for the created weld)
  - **Max Parent Thickness Factor**: Allows you to define the maximum thickness factor for weld bodies during meshing. Here thickness is calculated as follows:

    Weld Thickness= Max Parent Thickness Factor * max (Average thickness of all top and bottom faces for the created weld).
  - **Both**: Allows you to specify the minimum and maximum thickness factor for weld bodies during meshing. Here thickness is calculated as follows:

    Weld Thickness= Min Parent Thickness Factor * min(Average thickness of all top and bottom faces for the created weld) + Max Parent Thickness Factor * max(Average thickness of all top and bottom faces for the created weld).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **User Defined**: Allows you to define the **Thickness** value for weld bodies during meshing.

– **Expression**: Allows you to calculate thickness of weld based on the expression. The weld thickness is calculated as the average thickness of all up and down faces. The expression should be based on t1 and t2, where t1 is the average thickness of all up faces and t2 is the average thickness of all down faces. **Expression** allows only variables t1 and t2 as input and does not allow other variables.

Note:

Variables t1 and t2 must be in lower case. Expression supports min, max and scientific numbering. Expression does not support any special characters other than the following mathematical operators like '+', '-', '/', '*', '^', '(', ')'. For Expression, you must specify constants in geometric unit and not in Workbench display units.

Note:

The **Mechanical Properties** options are not available when the **Source** is **Geometry**.

Advanced

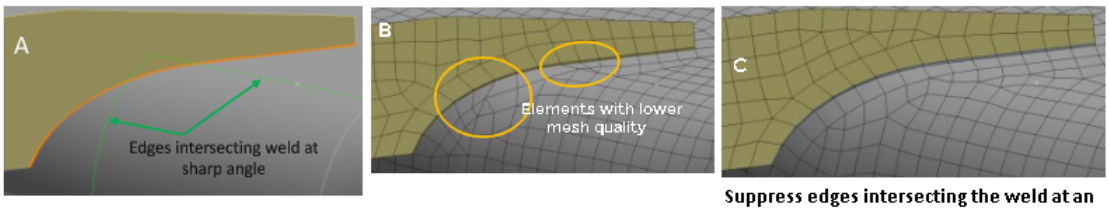
- **Relaxation**: Relaxes the HAZ layered failure. The available options are **Conservative** and **None**. The default value is **Conservative**.

  – **Conservative** : Provides warning for HAZ layered failures.

  – **None**: Provides error for HAZ layered failure.

  – **Aggressive**: Provides warning for weld face creation failures and HAZ layered failures. **Aggressive** option allows you treat failing welds as **Mesh Independent** irrespective of the setting or **Type** to handle weld face failures.

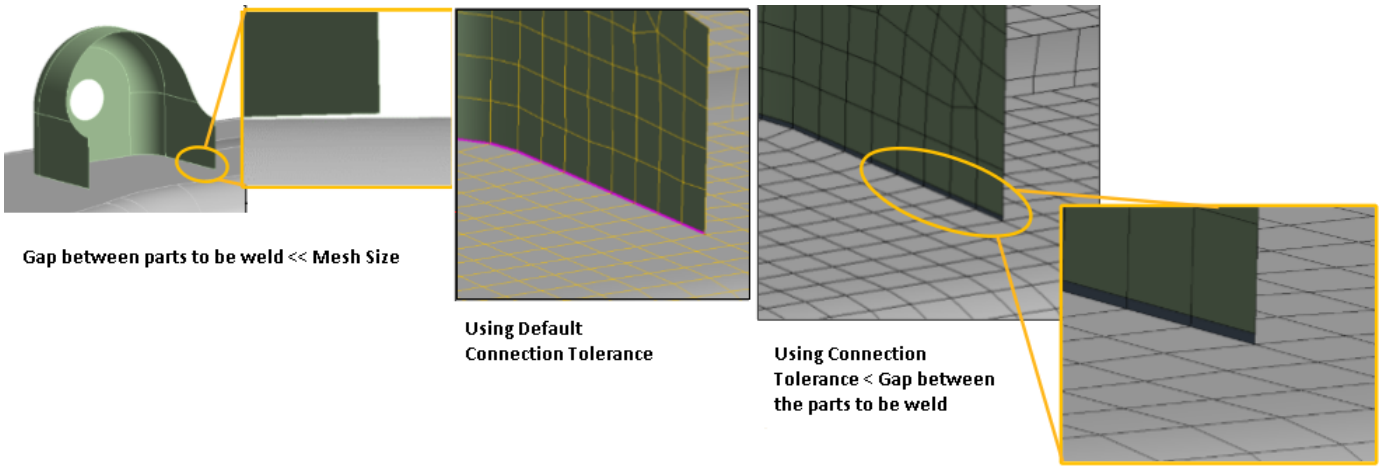
- **Sharp Angle**: Allows you to automatically merge the bottom faces if the weld faces intersect them at an angle less than the specified value. The default value is **90** degrees. The **Sharp Angle** allows you to provide any value from **0** to **90** degrees.



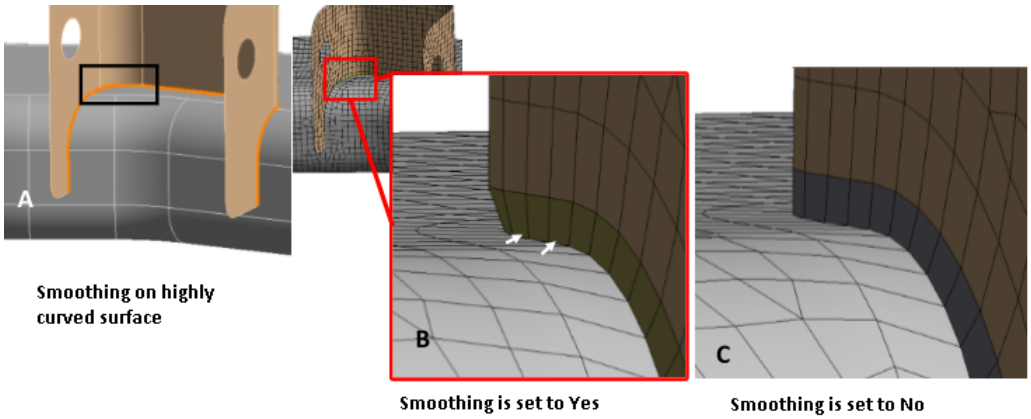
Suppress edges intersecting the weld at an acute angle and improves mesh quality

- **Connection Tolerance**: Allows you to connect the weld bodies created during mesh generation to the parent metal bodies. Available only when the **Source** is **Mesh**. When two parts to be welded together are close to each other, the **Connection Tolerance** should be less than the gap between the parts to be weld.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Smoothing**: Available only when the **Source** is **Mesh**. The default value is **Yes**. **Smoothing** should be set to **No** for highly curved surfaces to achieve better results.



- **Butt Weld**: Mark the weld elements and join them along their edges on the same plane when **Butt Weld** is **Yes**. **Butt Weld** is available only for **Mesh Independent Weld**.
- **Lap Weld Angle Tolerance**: Allows you to adjust angled face direction by specifying appropriate angle. **Lap Weld Angle Tolerance** is available only when the **Source** is **Mesh** and **Modeled As** is either **Angled** or **Normal and Angled**. The default value is **5** degree. The acceptable range of **Lap Weld Angle Tolerance** is **0** to **90** degrees.
- **Pinball Radius**: Allows you to calculate contact formulation while solving. **Pinball Radius** is available only when **Modeled As** is **Mesh Independent**. The default value is **Element Size**.
- **Layer Transition Type**: Allows you to provide the mesh element type to be created during layer transition. **Layer Transition Type** is available only when the **Number Of HAZ** is greater than one. The default value is **Quad**. The available options are:
  - **Quad**: Allows you to create the quadrilateral mesh.
  - **Quad/Tri**: Allows you to create the mesh with both quadrilateral and triangular elements.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

– **Quad/Minimal Tri**: Allows you to create the mesh with more quadrilateral and less triangular elements.

Note:

Quad/Minimal Tri pattern is used only in limited scenarios where good quality elements can be created.

Note:

Cross hair is implemented for Edge Mesh Size, HAZ Distance and Connection Tolerance, the diameter of the cross hair is equal to the value of the corresponding parameter.

Weld Graphics Enhancements

Preview Weld Geometry: Allows you to preview the weld faces before meshing. It also allows you to view the errors and warnings for weld before meshing. You can access the **Preview Weld Geometry** from the **Mesh** object and**Weld** controls.

Preview Weld Mesh: Allows you to generate the weld mesh on the selected controls for preview. You can access **Preview Weld Mesh** from **Mesh** object and **Weld** control. **Preview Weld Mesh** creates mesh only for the weld faces and its corresponding up and down faces. You can also right-click any column in the **Worksheet** to access **Preview Weld Mesh**.

Color Coding

Weld helps you to visualize the status of the weld created based on the color of the weld control. The blue color weld control denotes successful weld creation. The yellow color denotes the warning and partial error in the weld creation. The red color denotes the failure in weld creation. The same color coding applies to the weld worksheet. When you select a weld control in a row in the worksheet, the selected weld control is highlighted in green color in the **Geometry** window and vice versa. The light blue color denotes the weld curve or weld in an unknown mesh state.

Show Mesh

Show Mesh displays the Weld and HAZ layer elements as solid and other elements as translucent. To access **Show Mesh**, in the **Display** tab click **Show > Show Mesh**.

Zoom to Weld

Zoom to Weld allows you to zoom to fit the scoped references of weld component. To access the **Zoom to Weld**, right-click the **Geometry** window and click **Zoom to Weld**. **Zoom to Weld** is automatically applied to the selected items in worksheet.

Seam Weld Review

Seam Weld Review allows you to hide the parts that are not scoped to weld. To access **Seam Weld Review**, in the **Display** tab click **Show > Seam Weld Review**.

Imported Data Highlight

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

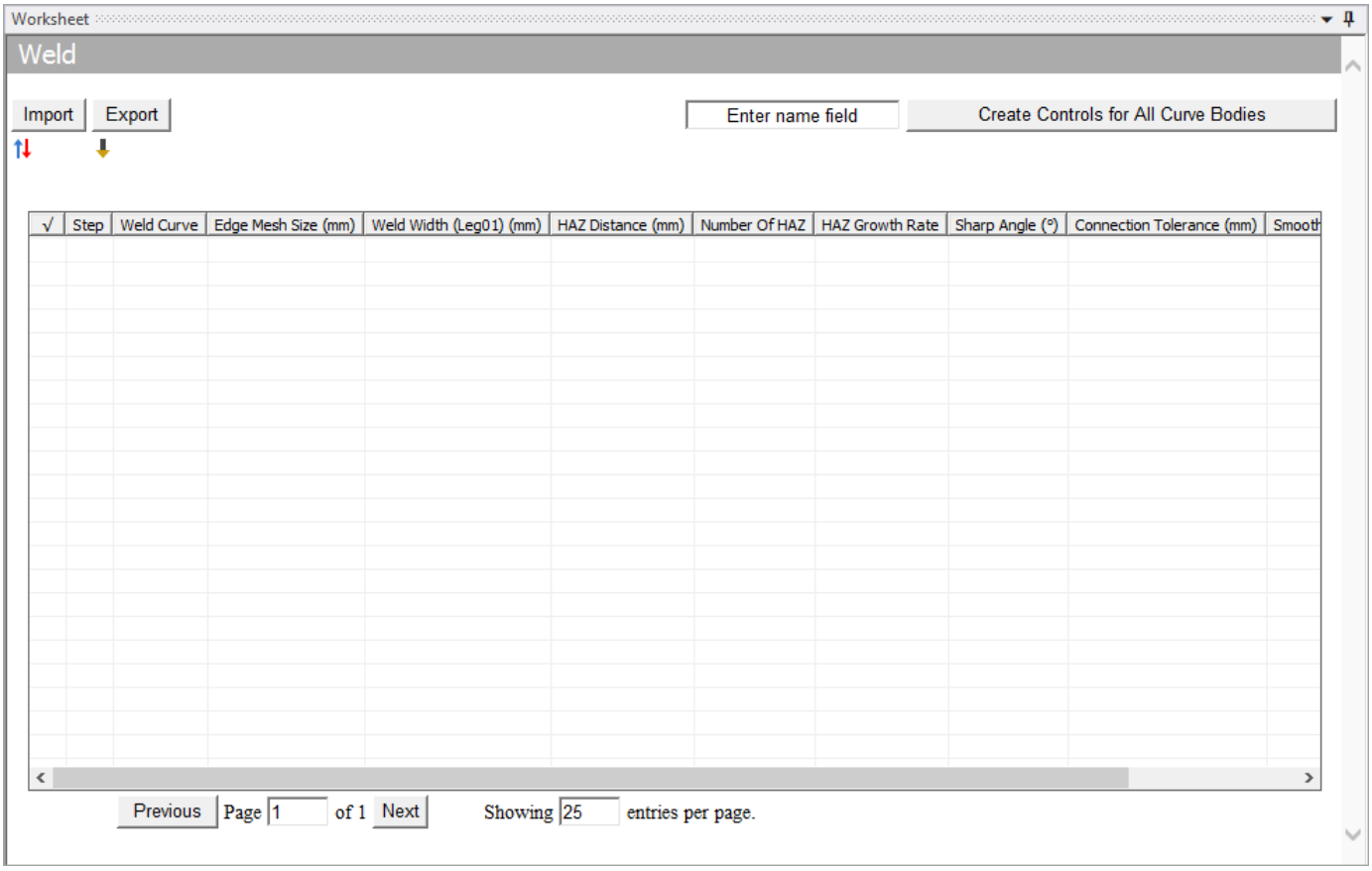
Imported Data Highlight enables you to pick and highlight the weld components both in the geometry window and in the worksheet. You can access **Imported Data Highlight** from the **Graphics** toolbar.

Note:

Weld supports Scripting. Scripting allows you to diagnose the weld errors and you can edit the Weld APIs to correct the same. For more detailed information on **Weld Scripting**, refer **Accessing and Manipulating the Mesh Object** and for more information on **Weld Worksheet Scripting**, refer **Working with Weld Worksheet**.

Weld Worksheet

Worksheet allows you to scope the weld controls. When **Use Worksheet** is set to **Yes, Scope** with **Worksheet** appears in the **Details** view which is a read-only field. The **Worksheet** opens and allows you to create weld controls. **Worksheet** provides different options based on the selected **Source** and **Type**. The options available in **Worksheet** are read only in the **Details** view. You can define these options in the **Worksheet**.



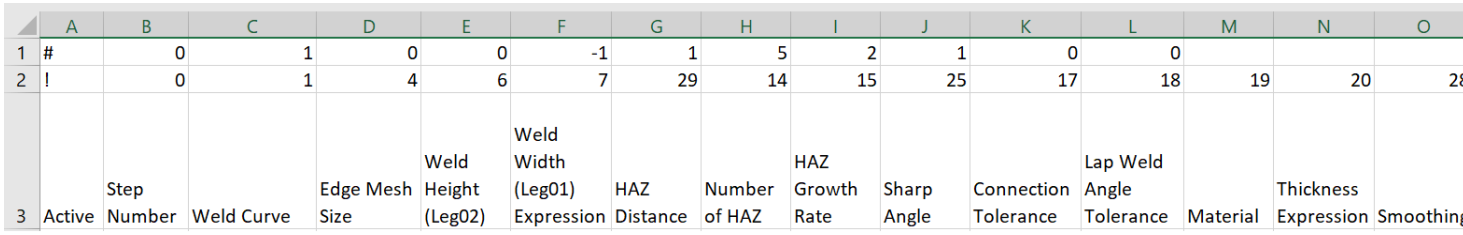
Note:

Worksheet allows you to create weld controls for curved bodies using **Body Selection**.

Worksheet has the following features:

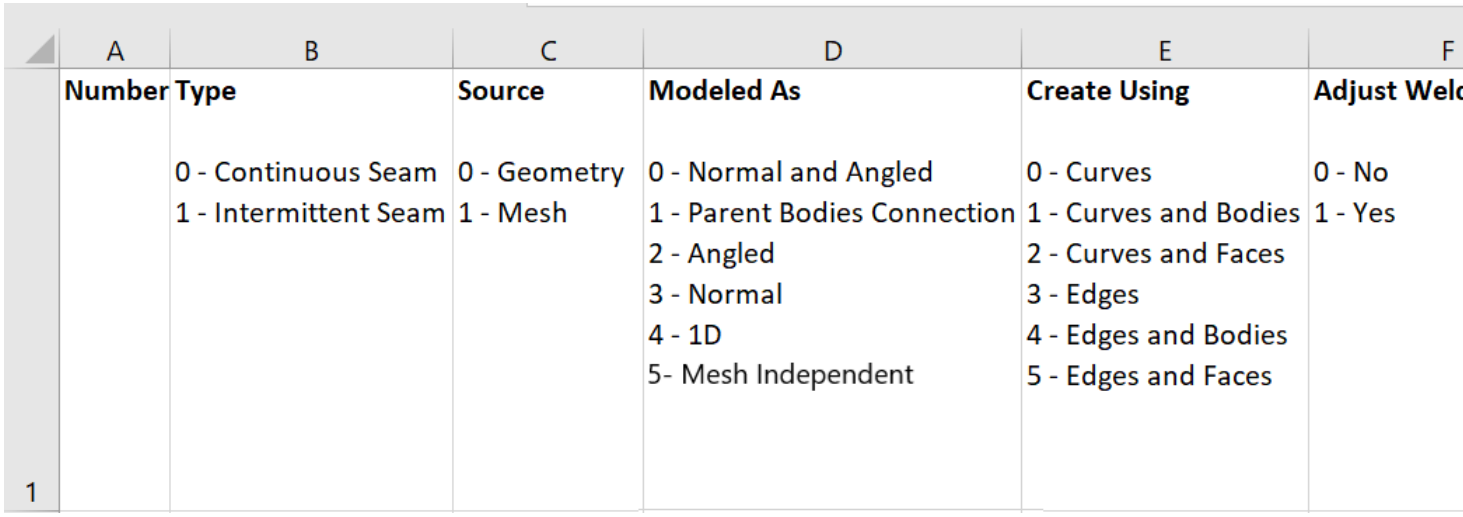
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Import**: Allows you to import the worksheet parameters as .CSV file format. You can also edit the CSV file if required. The .CSV file import format should be as given below:



	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O
1	#	0	1	0	0	-1	1	5	2	1	0	0			
2	!	0	1	4	6	7	29	14	15	25	17	18	19	20	28
3	Active	Step Number	Weld Curve	Edge Mesh Size	Weld Height (Leg02)	Weld Width (Leg01) Expression	HAZ Distance	Number of HAZ	HAZ Growth Rate	Sharp Angle	Connection Tolerance	Lap Weld Angle Tolerance	Material	Thickness Expression	Smoothing

Rows 1 and 2 are used to populate the weld control Details view. The values in Row 1 and 2 should not be modified (Row 1 and Row 2 are for internal use only. Row 2 and Row 3 are not related. You must be using Row 3 onwards for working with worksheet). The values in Row 1 represents the following:



	A	B	C	D	E	F
	Number	**Type**	**Source**	**Modeled As**	**Create Using**	**Adjust Weld**
1		0 - Continuous Seam 1 - Intermittent Seam	0 - Geometry 1 - Mesh	0 - Normal and Angled 1 - Parent Bodies Connection 2 - Angled 3 - Normal 4 - 1D 5- Mesh Independent	0 - Curves 1 - Curves and Bodies 2 - Curves and Faces 3 - Edges 4 - Edges and Bodies 5 - Edges and Faces	0 - No 1 - Yes

In Row 2, the numbers indicate the columns in worksheet. The details are as follows:

– 0 - Active

– 1 - StepNumber

– 2 - Bottom Face/Body/ Normal surface

– 3 - Top Face/Body/Angled surface

– 4 - Weld Curve

– 5 - Edge

– 6 - Edge Mesh Size

– 7 - Weld Height (Leg02)

– 8 - Weld Angle

– 9 - Weld Length

– 10 - Weld Pitch

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 11 - Number of Welds
- 12 - Offset1
- 13 - Offset2
- 14 - HAZ Distance
- 15 - Number of HAZ Layers
- 16 - Generate End-caps
- 17 - Sharp Angle
- 18 - Connection Tolerance
- 19 - Lap Weld Angle Tolerance
- 20 - Thickness Value
- 21 - Min Thickness Factor
- 22 - Max Thickness Factor
- 23 - Smoothing
- 24 - HAZ Growth Rate
- 25 - Cross Section Id
- 26 - Intersection Tag
- 27 - Thickness Expression
- 28 - Weld Width (Leg01) Expression

Note:

- Named selections for edges should be unique.
- Named selection generated are always prefixed with **weld**_ in the name.
- For each row, the edges scoped to the named selection must be continuous.
- If welds are defined using **Curves**, then the names of weld curves must be unique.
- **Material** is not available in Worksheet.

- **Export**: Allows you to export the worksheet parameters as .CSV file format.
- : Allows you to sort all the selected weld curves in the worksheet or the **Geometry** window in ascending order and displays it on the first page of the worksheet.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- : Allows you to sort rows based on its status.
- **Enter name field**: Allows you search weld curves using the text entered in the textbox. When you enter the text in the **Enter name field** and click **Create Controls for All Curve Bodies**, worksheet displays all weld curves with the entered text.
- **Create Controls for All Curve Bodies**: Allows you to create all weld curves. Also, records the operation and allows you to playback the same for reference.
- **Previous**: Allows you to move to the previous page of the worksheet from the current page.
- **Next**: Allows you to move to the next page of the worksheet from the current page.
- **Showing entries per page**: Displays the number of allowed entries per page. This is an auto-populated entry.

Note:

You can click the header of the worksheet columns to sort the items of the respective column.

Error Handling using Worksheet

Worksheet enables you to handle errors and failures efficiently. The **Worksheet** rows highlighted in yellow indicates warning messages. The red highlighted rows indicate error messages. The grey highlighted rows indicate the invalid weld or suppressed geometry. The allows you to send the weld for meshing. White is for active and valid weld with valid geometry. Only active rows can be provided as input for meshing. When your input to worksheet has a suppressed body, the weld worksheet row is highlighted in grey but will be active.

When you set the **Use Worksheet** to **Yes** and right-click the **Weld** object in the tree view, the following options are available:

- **Deactivate Problematic Worksheet Entries**: Allows you to deactivate the row which is causing error while meshing.

Right-click the **Worksheet**, the following options are available:

- **Add**: Allows you to add a row to the worksheet.
- **Delete**: Allows you to remove a row in the worksheet.
- **Activate all Selections**: Allows you to activate the multiple rows selected.
- **Deactivate all Selections**: Allows you to deactivate the multiple rows selected.
- **Go To Selected Items in Tree**: Allows you to go to the selected worksheet item in the tree.
- **Promote to Weld Control**: Allows you to create new weld control for the selected worksheet row. These rows are added to a new worksheet created for the new weld control.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Paste From Clipboard**: Allows you to copy the text from clipboard to worksheet.
- **Preview Weld Mesh**: Allows you to generate the weld mesh on the selected controls for preview from the worksheet.
- **Preview Weld Geometry**: Allows you to preview the weld faces before meshing from the worksheet. It also allows you to view the errors and warnings for weld before meshing.

Note:

Weld Worksheet allows you to select the faces, edges and bodies only through named selections and you cannot add faces, edges and bodies to the weld worksheet selection from the **Geometry** window.

Weld Limitations

The **Weld** control has the following limitations:

- **Weld** control does not support **Connect and Mesh Selected Entities** option.
- Combined length of all angled faces and combined length of all normal faces should be same.
- **Weld** control overrides all other sizing and local control settings.
- The selection option for **Weld Curve** does not support multiple bodies.
- CAD edges scoped in a weld control or in one row of weld control worksheet must be continuous.
- **Weld** control does not support intersecting welds for all sources.
- **Weld** Control does not support triangular mesh, when you select **Mesh Type** as **Triangles**.
- **Weld Worksheet** allows you to import only files with plain CSV format.
- The **Weld** control does not support writing Weld Definition File for multiple edges or closed loop selections.
- Solid Shell welds could fail for solver if **Solid-Shell Weld – Shared Nodes** is **Yes** under **Meshing** in **Files > Options > Meshing**.

Quad Layer

Quad Layer control allows you to create layer of quadrilateral elements around holes and edges. **Quad Layer** control enables you to scope the edge loops or edge based named selections. You must use **Automatic (PrimeMesh)** method to scope the parent bodies of edges used in **Quad Layer** control.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can right-click **Mesh** and click **Preview > Quad Layer** to preview the quad layer before mesh generation. Also, you can right-click **Quad Layer**, and click **Preview > Quad Layer**.

Note:

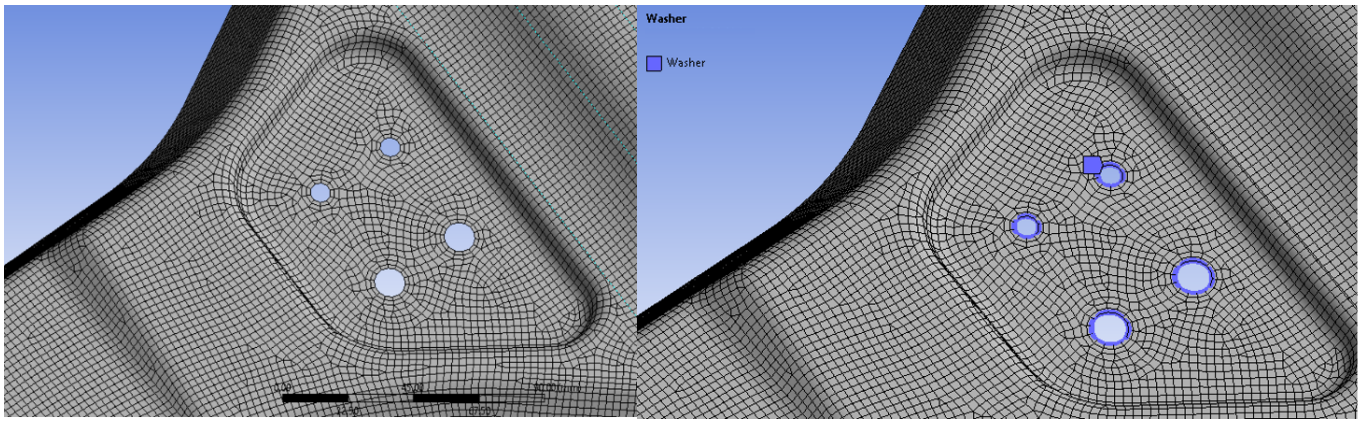
When you use **Preview > Quad Layer**, you cannot preview the applied edge sizing.

To access the **Quad Layer** control,

Right-click **Mesh** object and click **Insert > Quad Layer**.

or

Click **Mesh** on the Tree Outline and click **Quad Layer** in the **Mesh Context** tab on the Ribbon.



When you click **Quad Layer**, the Details view displays the **Quad Layer** options:

Type

- **Quad Layer Type**: Allows you to select the type of quad layer control. The available option is **Washer** and **Edge Loop**.
  - **Washer**: Enables you to create quad layers around the circular edges.
  - **Edge Loop**: Enables you to create quad layers on circular and non-circular edges.

Scope

- **Scoping Method**: Allows you to select the scoping method for washer control. The default method is **Geometry**.
  - **Geometry**: Allows you to scope the edges.
  - **Named selection**: Allows you to scope the edge based named selection.
- **Method**: Allows you to define the method when the **Quad Layer Type** is **Washer**. The available options are **Basic** and **Advanced**.
  - **Basic**: Compute the edge meshes on the washer and provides better transition. **Basic** method automatically decides the growth rate and width of the washer. When you select **Basic**, the available options are **Minimum Number of Divisions** and **Number of Washer**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Layers. **Basic** method requires you to provide minimal inputs. The number of divisions around the hole as well as washer layer heights is computed to give the best mesh based on the element size and the hole size.

Note:

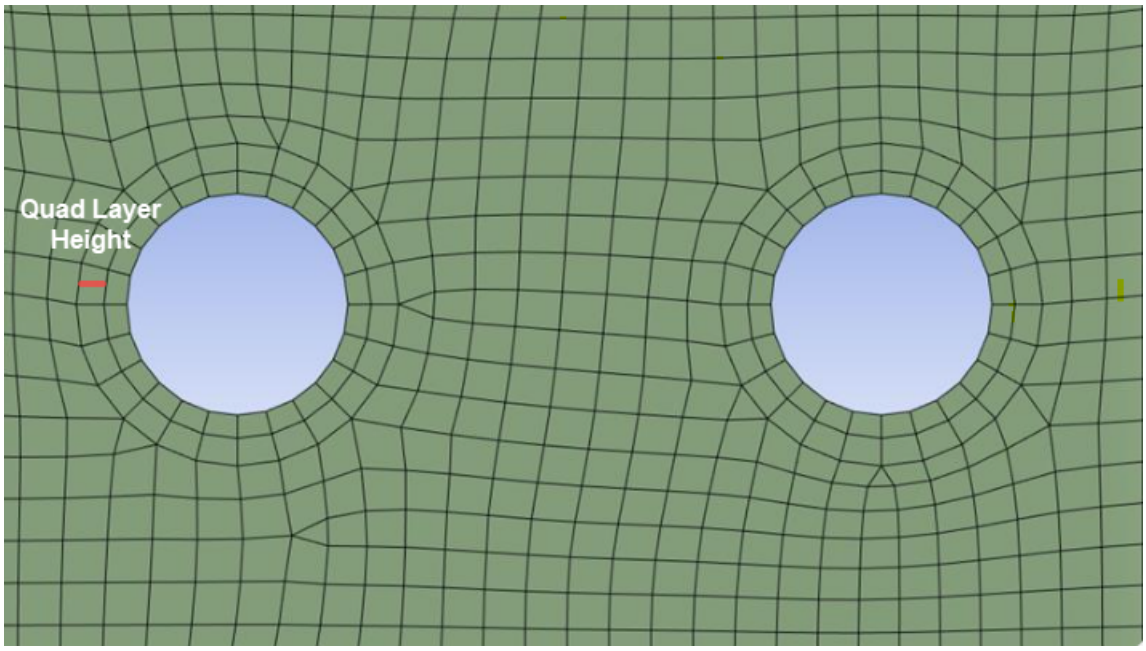
When more than one washer layer is specified in the **Basic** method, in some cases the width of the washer does not increase with the number of layers. This happens to accommodate the minimum number divisions specified and to get a smooth transition of the elements from washer layer to the adjacent mesh.

– **Advanced**: Allows you to create the washer layers as per your preference. When you select **Advanced**, the available options are **Number of Divisions**, **Number of Washer Layers** and **Type**.

Definition

- **Suppressed**: Allows you to suppress the washer control. The default value is **No**. When **Suppressed** is set to **Yes**, the **Active** field displays the status of the washer control. The **Active** field is read-only.
- **Control Messages**: Provides an error message when you try to apply control on washer edges which already have an applied control. The default value is **No**. When you try to apply a control on the Washer edge having an applied control, the **Control Message** is set to **Yes** and the corresponding error message is provided.
- **Minimum Number of Divisions**: Displays the minimum number of divisions required for the washer layer. It determines the minimum number of nodes on the washer layer. The default value is 6. The **Minimum Number of Divisions** option is only available when the **Method** is **Basic**. It is a lower bound on the **Number of Divisions**, so that you can avoid the automatic setting of a smaller number of divisions than an acceptable value.
- **Number of Divisions**: Allows you to divide the washer edge into the specified the number of divisions. The application recommends, you use an even number of divisions (6, 8, 12, ...) to get the best mesh flow in your quad mesh.
- **Number of Washer Layers**: Allows you to specify the required number of quad layers to be created. The default value is **1** and the maximum allowed number of layers is **3**.
- **Type**: Allows you to define the type of washer layer height. The available options are **Washer Diameter** and **Washer Layer Height**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



  - **Washer Diameter**: Allows you to specify the washer layer height as a factor of diameter for holes. When the **Type** is set to **Washer Diameter**, the **Scale of Diameter** is available. The **Scale of Diameter** allows you to define the washer height as factor of diameter. The default value is **1.5**

> **Note:**
>
> When you provide a **Scale of Diameter** value less than or equal to one, **Washer** takes the default value.

  - **Washer Layer Height**: Allows you to specify the height of the washer layers. The default value is same as the **Element Size**. Available only when the **Type** is **Washer Layer Height**.

- **Growth Rate**: Allows you to specify the width ratio between the adjacent washer layers. This option is available only when the **Number of Washer Layers** is greater than **1**. The default value is **1.0**

- **Element Size**: Allows you to provide the element size of the created quad mesh. **Element Size** is available only when the **Quad Layer Type** is **Edge Loop**.

- **Number of Quad Layers**: Allows you to provide the number of quad layers to be generated. **Number of Quad Layers** is available only when the **Quad Layer Type** is **Edge Loop**. The default value is **1**. The maximum allowed number of quad layers is **4**.

- **Layer Height**: Allows you to specify the height for each quad layer. **Layer Height** is available only when the **Quad Layer Type** is **Edge Loop**.

- **Quad Layer Direction**: Allows you to select how to specify the quad layer direction. The available options are:

  - **Program Controlled**: Creates quad layers on all connected faces around the input edges.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Scope Faces**: Allows you to create quad layers on the scoped faces. When you select **Scope Faces**, **Faces Scoping Method** allows you to scope the face using **Geometry Selection** or **Named Selection**. When you select **Geometry Selection**, **Face Selection** allows you to select the faces from the **Geometry** window for scoping. When you select **Named Selection**, **Face Named Selection** allows you to select the faces from the available named selections for scoping.

Advanced

– **Relaxation**: Relaxes the quad layer failures. The available options are **None** and **Conservative**. When you select **None**, generated mesh fails on quad layer failure. When you select **Conservative**, generates successful mesh even when quad layer fails.

– **Auto-Defeature**: Allows you to partially suppress edges or edges around the holes to obtain good quality quad layers. The default value is **No**. When **Auto-Defeature** is **Yes**, allows you to specify the **Proximity Tolerance**.

– **Proximity Tolerance**: Specifies the absolute distance to defeature around the edges to generate successful quad layer. The default value for proximity tolerance is calculated as:

Proximity Tolerance = Total Layer Height + (0.8*Element Size)

– **Layer Transition Type**: Allows you to specify the mesh element type created during layer transition. **Layer Transition Type** is available only when the **Number Of Washer Layers** or **Number of Quad Layers** is greater than one. The default value is **Quad**. The available options are:

→ **Quad**: Allows you to create the quadrilateral mesh.

→ **Quad/Tri**: Allows you to create the mesh with both quadrilateral and triangular elements.

→ **Quad/Minimal Tri**: Allows you to create the mesh with more quadrilateral and less triangular elements.

Note:

Quad/Minimal Tri Layer Transition Type are used only in limited scenarios where good quality elements can be created.

Limitations

Quad Layer has the following limitations:

- **Quad Layer** control supports only quadrilateral mesh.
- When the edge sizes on the hole boundary are smaller than the allowed minimum size, you should use a smaller value for the **Minimum Number of Divisions** to get a good quality mesh for the washer layers.
- **Washer** control can be applied only on circular holes. **Washer** control automatically filters out non-circular holes while meshing. Hence, you should use **Washer** control on circular holes only.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Quad/Minimal Tri Layer Transition Type** are used only in limited scenarios where good quality elements can be created.

Deviation

Deviation control enables you to set the mesh resolution in the primary curvature direction for fillets on shell (surface) bodies. Mesh resolution along the fillet is taken from global or local mesh size controls. **Deviation** control enables you to scope faces or face based named selections.

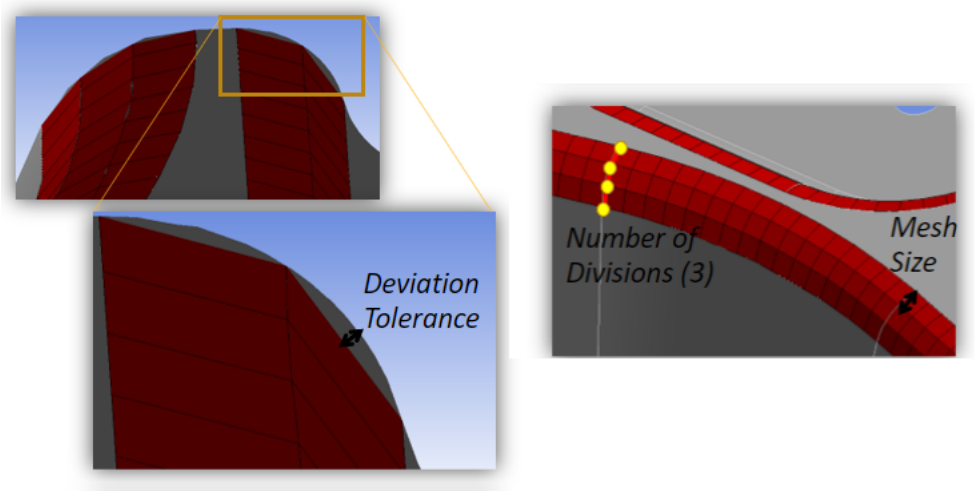
All **Deviation** bodies should be scoped under **Automatic (PrimeMesh)** method. If not, scope the bodies under **Automatic (PrimeMesh)** to perform **Deviation** operation.

To access the **Deviation** control,

Right-click **Mesh** object and click **Insert > Deviation**.

or

Click **Mesh** on the Tree Outline and click **Deviation** in the **Mesh Context** tab on the Ribbon.



When you click **Deviation**, the Details view displays the deviation options:

Scope

- **Scoping Method**: Allows you to select the scoping method for **Deviation** control. The default method is **Geometry**.
  - **Geometry**: Allows you to scope the faces.
  - **Named Selection**: Allows you to scope the face based named selection.

Definition

- **Suppressed**: Allows you to suppress the **Deviation** control. The default value is **No**. When **Suppressed** is set to **Yes**, the **Active** field displays the status of the **Deviation** control. The **Active** field is read-only.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Type**: Allows you to define the type of deviation control specifications. The available options are **Deviation Tolerance**, **Mesh Size**, **Number of Divisions**.
  - **Deviation Tolerance**: Allows you to specify the maximum distance between the mesh and the curvature of the fillet. You can provide smaller **Deviation Tolerance** to get a better curvature resolution.
  - **Mesh Size**: Allows you to specify the mesh size on the shorter sides of the fillets. The default value for **Mesh Size** is 75% of the chosen **Element Size**.
  - **Number of Divisions**: Allows you to specify the number of divisions on the curved sides of the faces.

Limitations

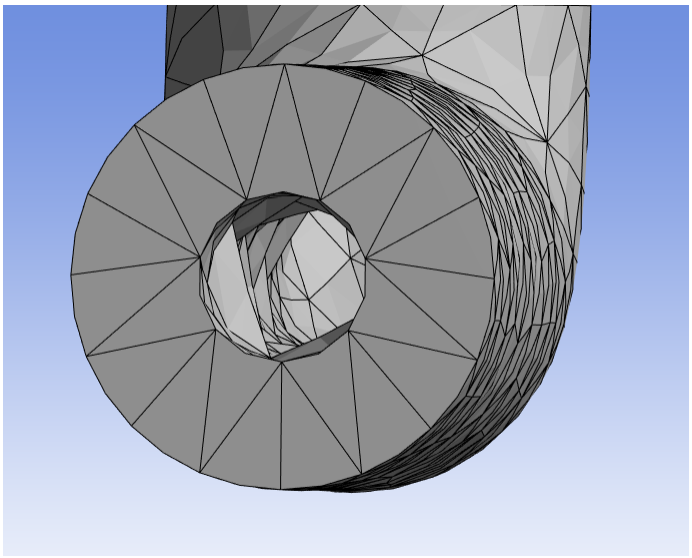
- In the **Mesh** Details view, when the **Capture Curvature** is set to **Yes**, the **Deviation** control is not considered while meshing.
- **Deviation** control supports only quadrilateral mesh.
- **Deviation** control does not work on fillets having sharp angle edges.
- **Deviation** control only meshes continuous body. It does not mesh any exclusion or inclusion in the model.
- When **Deviation** control cannot be applied, the default quadrilateral dominant mesh is generated.
- When there is an extra topological node on one side of the shorter edge of the fillet, the node gets projected to the other shorter edge of the fillet to get all quadrilateral mesh for **Deviation** control.
- The **Deviation** control generates quadrilateral mesh on triangular and polygonal shaped fillets.

Geometry Fidelity

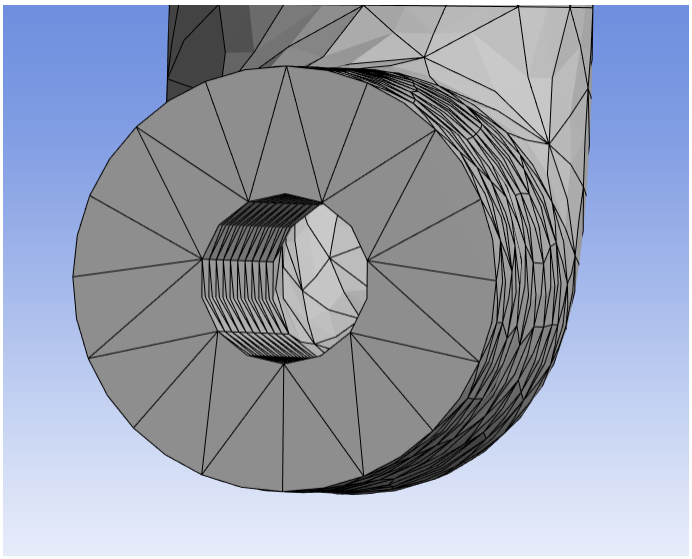
Geometry Fidelity ensures the nodes of the mesh lie on the scoped geometry including midside nodes for quadratic elements. **Geometry Fidelity** adds constraint on the mesh which cause deterioration in quality in some situations due to the forced projection. In such cases, you should refine the mesh at the same locations with local size control.

The following image shows the face without **Geometry Fidelity**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following image shows the face with **Geometry Fidelity**



To access **Geometry Fidelity**,

On the Tree Outline, right-click the **Mesh** and click **Insert > Geometry Fidelity**.

Or

Right-click on the Geometry window and click **Insert > Geometry Fidelity**.

Geometry Fidelity Details view has the following options:

Scope

- **Scoping Method**: Allows you to scope geometry faces or named selection. The default value is **Geometry Selection**.
  - **Geometry Selection**: Allow you to scope the faces. When you select **Geometry Selection**, the **Geometry** allows you to select the faces from the **Geometry window**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Named Selection**: Allow you to scope faces grouped under a named selection.

Definition

Suppressed: Allows you to suppress the **Geometry Fidelity** control, when set to **Yes**. The default value is **No**.

Limitations

- **Geometry Fidelity** supports only for Patch Conforming Algorithm for Tetrahedrons Method Control (p. 208).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Meshing: Options

You can control the behavior of functions in the Meshing application through the **Options** dialog box. For more information, refer to:

Accessing the Options Dialog Box
Meshing Options on the Options Dialog Box
Licensing Option
Mesh Quality Option

Accessing the Options Dialog Box

To access the Meshing application options (p. 389):

1. From the main menu, choose **Files > Options**. An **Options** dialog box appears and the options are displayed on the left.
2. Click a specific option on the left (either **Meshing** or **Export**).
3. Change any of the option settings by clicking directly in the option field on the right. You will first see a visual indication for the kind of interaction required in the field (examples are drop-down menus, secondary dialog boxes, direct text entries).
4. Click **OK**.

Note:

- To cancel your changes and restore the default settings, click **Reset** on the **Options** dialog box.
- Option settings within a particular language are independent of option settings in another language. If you change any options from their default settings, then start a new Workbench session in a different language, the changes you made in the original language session are not reflected in the new session. You are advised to make the same option changes in the new language session.

Meshing Options on the Options Dialog Box

The options that appear in the right pane of the **Options** dialog box depend on which category is selected in the left pane:

- When **Meshing** is selected in the left pane, these subcategories appear:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - Meshing
  - Virtual Topology
  - Sizing
  - Quality
  - Inflation
- When **Export** is selected in the left pane, these subcategories appear:
  - CGNS
  - Ansys Fluent

Meshing

- **Highlight Topology Being Meshed if Possible**: Controls the default for highlighting of topologies during mesh processing. When **Yes** (default), the topology that is currently being processed by the mesher is highlighted in the **Geometry** window, which may help with troubleshooting (p. 580). This highlighting is not supported for the **Patch Independent Tetra** or **MultiZone** mesh methods, or when assembly meshing is being used. Refer to Generating Mesh (p. 528) for details.
- **Allow Selective Meshing**: Allows/disallows selective meshing. Choices are **Yes** and **No**. The default value is **Yes**. Refer to Selective Meshing (p. 444) for details.
- **Number of CPUs for Meshing Methods**: Specifies the number of processors to be used by the meshing operation. There is no counterpart setting in the Details View. Specifying multiple processors will enhance the performance of the **MultiZone Quad/Tri**, **Patch Independent Tetra**, and **MultiZone** mesh methods. This option has no effect when other mesh methods are being used. You can specify a value from **0** to **256** or accept the default. The default is **0**, which means the number of processors will be set automatically to the maximum number of available CPUs. The **Number of CPUs** option is applicable to shared memory parallel (SMP) meshing (multiple cores; not supported for clusters). Refer to Parallel Part Meshing (p. 472).
- **Default Physics Preference**: Sets the default option for the **Physics Preference** (p. 96) in the Details View of a **Mesh** object. Choices are **Mechanical**, **Nonlinear Mechanical**, **Electromagnetics**, **CFD**, **Explicit**, and **Hydrodynamics**.
- **Default Method**: Sets the default **Method** setting in the Details View of a **Method** (p. 202) control object. This option only affects **Method** controls that are added manually. Choices are **Automatic (Patch Conforming/Sweeping)**, **Patch Independent**, and **Patch Conforming**. When the geometry is attached, the default method of all options except **Automatic** will be scoped to all parts in the assembly. The **Automatic** option has no effect.

  Note:

  Changing the **Default Method** changes the default mesh method for all future analyses, regardless of analysis type.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Default Sheet Body Method**: Sets the default method for the **Sheet Body Method** in the Details view of a Mesh object. The available options are **Quad Dominant** and **Prime Mesh**. You can change this value in the **Automatic Methods** group under Mesh Details view.
- **Use MultiZone for Sweepable Bodies**: If set to **On**, the mesher uses the MultiZone (p. 419) method instead of General Sweeping (p. 399) for sweepable bodies. The default setting is **Off**. See MultiZone for Sweepable Bodies (p. 422) for more information.
- **Solid –Shell Weld- Shared Nodes**: Generates weld between shell and solid bodies but the node connection between the weld and solid bodies are not conformal. Instead, bonded contact is created while solving when **Solid –Shell Weld- Shared Nodes** is **No**. When **Solid –Shell Weld- Shared Nodes** is **Yes**, creates conformal mesh without contacts between the shell and weld bodies.
- **Topology Checking**: Sets the default value for the Topology Checking (p. 183) control. The default value is **Yes**.
- **Verbose Messages from Meshing**: Controls the verbosity of messages returned to you. If set to **On** and you are meshing a subset of bodies, the message "These bodies are going to be meshed" appears, and you can click the right mouse button on the message to see the bodies. The default is **Off**. Regardless of the setting, when meshing completes and any bodies failed to mesh, the message "These bodies failed to be meshed" appears, and you can click the right mouse button to see them.
- **Number of CPUs for Parallel Part Meshing**: Sets the default number of processors to be used for parallel part meshing. You can change this value in the **Advanced** (p. 93) group under the Details View. Using the default for specifying multiple processors will enhance meshing performance on geometries with multiple parts. For parallel part meshing, the default is set to **Program Controlled** or **0**. This instructs the mesher to use all available CPU cores. The Default setting inherently limits 2 GB memory per CPU core. An explicit value can be specified between **0** and **256**, where **0** is the default. Refer to Parallel Part Meshing (p. 472) for more details.
- **Show Only Non-Zero Mesh Element Statistics**: Enables you to view the counts for all element types even the ones that do not exist in the mesh when set to **No**. The default value is **Yes**.
- **Redraw Graphics after Each Mesh Generation**: Displays the graphics for every step in the **Mesh Worksheet** (p. 448) after completing the mesh generation for the respective step, when set to **Yes**. The default value is **Yes**. **Redraw Graphics after Each Mesh Generation** applicable only when **Mesh Worksheet** is created and active.
- **Skip Part Remesh in Worksheet Meshing**: Skips remeshing the failed rows in the worksheet in case of mesh failure and continues meshing the consecutive rows when **Skip Part Remesh in Worksheet Meshing** is **Yes**. The default value is **No**.

Virtual Topology

Merge Edges Bounding Manually Created Faces: Sets the default value for the **Merge Face Edges** setting in the Details View of a **Virtual Topology** object. Choices are **Yes** and **No**. The default value is **Yes**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sizing

- **Adaptive Resolution**: Sets the resolution for mesh sizing when **Use Adaptive Sizing** is set to **Yes**. The default setting is **Program Controlled**. The range of values that can be set is 0 to 7, with the mesh resolution changing from coarse (**0**) to fine (**7**). See Resolution (p. 109) for more information.
- **Mechanical Min Size Factor (Default: 0.01)**: Sets your preference for the scale factor that will be used to calculate the default minimum size when the physics preference is **Mechanical**, **Electromagnetics**, or **Explicit**. The value that is specified here is multiplied by the global element size to determine the default minimum size.
- **CFD Min Size Factor (Default: 0.01)**: Sets your preference for the scale factor that will be used to calculate the default minimum size when the physics preference is **CFD**. The value that is specified here is multiplied by the global element size to determine the default minimum size.
- **Explicit Min Size Factor (Default: 0.5)**: Sets your preference for the scale factor that will be used to calculate the default minimum size when the physics preference is **Explicit**. The value that is specified here is multiplied by the global element size to determine the default minimum size.
- **Mechanical Defeature Size Factor (Default: 0.005)**: Sets your preference for the scale factor that will be used to calculate the default defeature size when the physics preference is **Mechanical**, **Electromagnetics**, or **Explicit**. The value that is specified here is multiplied by the global element size to determine the default defeature size.
- **CFD Defeature Size Factor (Default: 0.005)**: Sets your preference for the scale factor that will be used to calculate the default defeature size when the physics preference is **CFD**. The value that is specified here is multiplied by the global element size to determine the default defeature size.
- **Explicit Defeature Size Factor (Default: 0.1)**: Sets your preference for the scale factor that will be used to calculate the default defeature size when the physics preference is **Explicit**. The value that is specified here is multiplied by the global element size to determine the default defeature size.
- **Bounding Box Factor (Default: 0.05)**: Helps set the default Element size as follows: (Bounding Box Diagonal * Bounding Box Factor = Default Element size). This is only used when **Use Adaptive Sizing** is set to **No** and only when solid or both solid and sheet bodies are present in the model. **Adaptive** sizing uses its own default.
- **Surface Area Factor (Default: 0.125)**: Helps set the default Element size as follows: sqrt(Average Surface Area) * Surface Area Factor = Default Element Size). This is only used when **Use Adaptive Sizing** is set to **No** and sheet bodies are present in the model. **Adaptive** sizing uses its own default.
- **Explicit Particle Diameter Factor (Default: 0.5)**: Sets your preference for the scale factor that will be used to calculate the default Particle Diameter when the physics preference is **Explicit**. The value that is specified here is multiplied by the global Particle Diameter to determine the default Particle Diameter.
- **MultiZone Sweep Sizing Behavior**: If set to **Use Size Function**, then any applied sizing controls (curvature and proximity refinement, and/or local sizing) are evaluated in all directions of the selected bodies. If set to **Ignore Size Function**, curvature and proximity refinement and/or local sizing along the sweep path are ignored and the spacing along the sweep path is determined either by the global sizes or local sizes that are explicitly set on edges along the sweep direction.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also override the sizing by locally specifying hard edge sizes or using the **Sweep Size Behavior** control in the **MultiZone** method control to locally adjust the sizing for the bodies the control is scoped to. See MultiZone Method Control (p. 242) for more information.

- **Propagate Global to Local Settings**: Enables you to use the global values for **Capture Curvature** and **Capture Proximity** as default local values when set to **Yes**. The default value is **Yes**.

Quality

- **Check Mesh Quality**: Sets the default quality behavior (p. 121) with respect to how the mesher responds to error and warning limits. Choices are:
  - **Default** - With this setting, the behavior changes as appropriate when you change the setting of **Physics Preference**.
  - **Yes, Errors** - If the meshing algorithm cannot generate a mesh that passes all error limits, an error message is printed and meshing fails.
  - **Yes, Errors and Warnings** - If the meshing algorithm cannot generate a mesh that passes all error limits, an error message is printed and meshing fails. In addition, if the meshing algorithm cannot generate a mesh that passes all warning (target) limits, a warning message is printed.
  - **No** - Mesh quality checks are done at various stages of the meshing process (for example, after surface meshing prior to volume meshing). The **No** setting turns off most quality checks, but some minimal checking is still done. In addition, even with the **No** setting, the target quality metrics are still used to improve the mesh. The **No** setting is intended for troubleshooting (p. 579) and should be used with caution as it could lead to solver failures or incorrect solution results.
  - **Mesh Quality Worksheet**: Perform mesh checks after mesh completion based on user defined warning and error limits set in the **Mesh Quality Worksheet**.
- **Mechanical Error Limit**: Sets the default error limit (p. 127) when **Physics Preference** is set to **Mechanical**. Choices are **Standard Mechanical** and **Aggressive Mechanical**.
- **Target Element Quality (0 = Program Default)**: Sets the default target element quality (p. 139). When you modify this value, the value you enter becomes the new default for the Target Element Quality (p. 130) in the Details View. You can enter 0 on the **Options** panel to revert to the program default.
- **Target Skewness (0 = Program Default)**: Sets the default target skewness (p. 151). When you modify this value, the value you enter becomes the new default for the Target Skewness (p. 131) in the Details View. You can enter 0 on the **Options** panel to revert to the program default. For a tetrahedral mesh, you should not set **Target Skewness** to a value < 0.8.
- **Target Jacobian Ratio (Corner Nodes) (0 = Program Default)**: Sets the default target Jacobian ratio (p. 142). When you modify this value, the value you enter becomes the new default Target Jacobian Ratio (Corner Nodes) (p. 131) in the Details View. You can enter 0 on the **Options** panel to revert to the program default.
- **Target Aspect Ratio (Height) (0 = Program Default)**: Sets the default target explicit aspect ratio. When you modify this value, the value you enter becomes the new default for the **Target Aspect Ratio (Height)** in the Details View. You can enter 0 on the **Options** panel to revert to the program default.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inflation

Inflation-related options that can be set on the **Options** dialog box include:

- **Maximum Height over Base** (p. 177)
- **Gap Factor** (p. 177)
- **Growth Rate Type** (p. 177)
- **Maximum Angle (Degrees)** (p. 178)
- **Fillet Ratio** (p. 178)
- **Use Post Smoothing** (p. 179)
- **Smoothing Iterations** (p. 180)

CGNS

- **File Format**: Sets the file format to be used for CGNS Export (p. 64) operations. There is no counterpart setting in the Details View. Choices are:
  - **ADF** (default) - Exports the mesh in ADF (Advanced Data Format).
  - **HDF5** - Exports the mesh in HDF5 (Hierarchical Data Format version 5).
- **CGNS Version**: Sets the CGNS library version to be used for CGNS Export (p. 64) operations. There is no counterpart setting in the Details View. Choices are **4.3** (the default), **3.3**, **3.2**, **3.1**, **3.0**, **2.5**, **2.4**, **2.3**, **2.2**, and **2.1**.
- **Export Unit**: Defines the unit of measurement for the mesh when exported to CGNS. The default is **Use Project Unit**, which means the mesh is not scaled. If you change this to another value (centimeter, millimeter, micrometer, inch, or foot), the mesh is scaled according to the export unit you select.

Ansys Fluent

- **Format of Input File (*.msh)**: Sets the file format to be used for Fluent Mesh Export (p. 43) operations. There is no counterpart setting in the Details View. Choices are **Binary** (the default) and **ASCII**.
- **Auto Zone Type Assignment**: When set to **On** (the default), zone types are automatically assigned, as described in Fluent Mesh Export (p. 43). When set to **Off**, assigns all boundary zones as the default WALL, enabling you to assign your own zone type assignments for Fluent Mesh Export (p. 43) operations.

Licensing Option

In the Mesh System, you can access **Meshing License** options from **File > Licensing**. The **License Options** allows you to set your license preferences. The available License Options are:

- **CFD PrepPost**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Ansys Mechanical Enterprise**
- **Ansys Mechanical Premium**
- **Ansys Mechanicl Pro**
- **Ansys Mechanical Enterprise PrepPost**
- **Ansys LS-DYNA**

From **2021 R2** release, Ansys Meshing does not support **Advanced Meshing**(advanced_meshing license feature). However, you can access the **Advanced Meshing** in the earlier versions of Ansys Meshing providing backward compatibility.

Meshing App sets the **CFD PrepPost** by default. You can also disable the license by selecting the license to be disabled and click the **Disable** button. You can access the legacy features by selecting the **Show Legacy Licenses** and the following license options are available:

- **CFD PrepPostPro**
- **Ansys Professional NLT**
- **Ansys Structural**
- **Ansys Professional NLS**
- **Ansys Multiphysics**
- **Ansys ExplicitSTR**
- **Ansys Emag**
- **Ansys AUTODYN PrepPost**
- **Ansys Mechanical/Emag**

All meshing features and options are available for the supported license features, but Fracture Meshing is available only for Ansys Enterprise License.

Mesh Quality Option

Mesh Quality options can be accessed from **File > Options > Meshing > Mesh Quality**. The **Mesh Metrics** are displayed and you can set them as per your preference.

- **Element Quality**: When you select **Based on Physics Preference**, the **Element Quality** is available for the following physics preference **Explicit, Mechanical, Nonlinear Mechanical, CFD, Electromagnetics, Hydrodynamics**. You can enable the **Element Quality** visibility in the **Mesh Details** view for the given mesh metric by selecting **On** and disable visibility by selecting **Off** as per your preference.
- **Aspect Ratio**: When you select **Based on Physics Preference**, the **Aspect Ratio** is available for the following physics preference **Mechanical, Nonlinear Mechanical, CFD, Electromagnetics,**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Hydrodynamics. You can enable the **Aspect Ratio** visibility in the **Mesh Details** view for the given mesh metric by selecting **On** and disable visibility by selecting **Off** as per your preference.

- **Jacobian Ratio (MAPDL)**: When you select **Based on Physics Preference**, the **Jacobian Ratio (MAPDL)** is available for the physics preference **Electromagnetics**. You can enable the **Jacobian Ratio (MAPDL)** visibility in the **Mesh Details** view for the given mesh metric by selecting **On** and disable visibility by selecting **Off** as per your preference.
- **Jacobian Ratio (Corner Nodes)**: When you select **Based on Physics Preference**, the **Jacobian Ratio (Corner Nodes)** is available for the physics preference **Explicit, Mechanical, Nonlinear Mechanical, Electromagnetics**. You can enable the **Jacobian Ratio (Corner Nodes)** visibility in the **Mesh Details** view for the given mesh metric by selecting **On** and disable visibility by selecting **Off** as per your preference.
- **Jacobian Ratio (Gauss Point)**: When you select **Based on Physics Preference**, the **Jacobian Ratio (Gauss Point)** is available for the physics preference **Mechanical, Nonlinear Mechanical, Electromagnetics**. You can enable the **Jacobian Ratio (Gauss Point)** visibility in the **Mesh Details** view for the given mesh metric by selecting **On** and disable visibility by selecting **Off** as per your preference.
- **Warping Factor**: When you select **Based on Physics Preference**, the **Warping Factor** is available for the physics preference **Electromagnetics, Hydrodynamics**. You can enable the **Warping Factor** visibility in the **Mesh Details** view for the given mesh metric by selecting **On** and disable visibility by selecting **Off** as per your preference.
- **Parallel Deviation**: When you select **Based on Physics Preference**, the **Parallel Deviation** is available for the physics preference **Electromagnetics**. You can enable the **Parallel Deviation** visibility in the **Mesh Details** view for the given mesh metric by selecting **On** and disable visibility by selecting **Off** as per your preference.
- **Maximum Corner Angle**: When you select **Based on Physics Preference**, the **Maximum Corner Angle** is available for the following physics preference **Explicit, Mechanical, Nonlinear Mechanical** and **Electromagnetics**. You can enable the **Maximum Corner Angle** visibility in the **Mesh Details** view for the given mesh metric by selecting **On** and disable visibility by selecting **Off** as per your preference.
- **Skewness**: When you select **Based on Physics Preference**, the **Skewness** is available for the following physics preference **Mechanical, Nonlinear Mechanical, CFD, Electromagnetics, Hydrodynamics**. You can enable the **Skewness** visibility in the **Mesh Details** view for the given mesh metric by selecting **On** and disable visibility by selecting **Off** as per your preference.
- **Orthogonal Angle**: When you select **Based on Physics Preference**, the **Orthogonal Angle** is available for the following physics preference **CFD, Electromagnetics**. You can enable the **Orthogonal Angle** visibility in the **Mesh Details** view for the given mesh metric by selecting **On** and disable visibility by selecting **Off** as per your preference.
- **Characteristic Length (Autodyn)**: When you select **Based on Physics Preference**, the **Characteristic Length (Autodyn)** is not available for any of the physics preference. You can enable the **Characteristic Length (Autodyn)** visibility in the **Mesh Details** view for the given mesh metric by selecting **On** and disable visibility by selecting **Off** as per your preference
- **Minimum Tri Angle**: When you select **Based on Physics Preference**, the **Minimum Tri Angle** is available for the following physics preference **Explicit, Mechanical, Nonlinear Mechanical**

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and **Hydrodynamics**. You can enable **Minimum Tri Angle** visibility in the **Mesh Details** view for the given mesh metric by selecting **On** and disable visibility by selecting **Off** as per your preference.

- **Maximum Tri Angle**: When you select **Based on Physics Preference**, the **Maximum Tri Angle** is available for the following physics preference **Explicit, Mechanical, Nonlinear Mechanical** and **Hydrodynamics**. You can enable the **Maximum Tri Angle** visibility in the **Mesh Details** view for the given mesh metric by selecting **On** and disable visibility by selecting **Off** as per your preference.
- **Minimum Quad Angle**: When you select **Based on Physics Preference**, the **Minimum Quad Angle** is available for the following physics preference **Explicit, Mechanical, Nonlinear Mechanical** and **Hydrodynamics**. You can enable the **Minimum Quad Angle** visibility in the **Mesh Details** view for the given mesh metric by selecting **On** and disable by selecting **Off** as per your preference.
- **Maximum Quad Angle**: When you select **Based on Physics Preference**, the **Maximum Quad Angle** is available for the following physics preference **Explicit, Mechanical, Nonlinear Mechanical** and **Hydrodynamics**. You can enable the **Maximum Quad Angle** visibility in the **Mesh Details** view for the given mesh metric by selecting **On** and disable visibility by selecting **Off** as per your preference.
- **Warping Angle**: When you select **Based on Physics Preference**, the **Warping Angle** is available for the following physics preference **Explicit, Mechanical, Nonlinear Mechanical** and **Hydrodynamics**. You can enable the **Warping Angle** visibility in the **Mesh Details** view for the given mesh metric by selecting **On** and disable visibility by selecting **Off** as per your preference.
- **Tet Collapse**: When you select **Based on Physics Preference**, the **Tet Collapse** is available for the following physics preference **Explicit, Mechanical** and **Nonlinear Mechanical**. You can enable the **Tet Collapse** visibility in the **Mesh Details** view for the given mesh metric by selecting **On** and disable visibility by selecting **Off** as per your preference.
- **Aspect Ratio (Height)**: When you select **Based on Physics Preference**, the **Aspect Ratio (Height)** is available only for physics preference **Explicit**. You can enable the **Aspect Ratio (Height)** visibility in the **Mesh Details** view for the given mesh metric by selecting **On** and disable visibility by selecting **Off** as per your preference.
- **Minimum Element Edge Length**: When you select **Based on Physics Preference**, the **Minimum Element Edge Length** is available for the following physics preference **Explicit, Mechanical** and **Nonlinear Mechanical**. You can enable the **Minimum Element Edge Length** visibility in the **Mesh Details** view for the given mesh metric by selecting **On** and disable visibility by selecting **Off** as per your preference.
- **Maximum Element Edge Length**: When you select **Based on Physics Preference**, the **Maximum Element Edge Length** is available for the following physics preference **Explicit, Mechanical** and **Nonlinear Mechanical**. You can enable the **Maximum Element Edge Length** visibility in the **Mesh Details** view for the given mesh metric by selecting **On** and disable visibility by selecting **Off** as per your preference.
- **Characteristic Length (LS-DYNA)**: When you select **Based on Physics Preference**, the **Characteristic Length (LS-DYNA)** is available only for physics preference **Explicit**. You can enable the **Characteristic Length (LS-DYNA)** visibility in the **Mesh Details** view for the given mesh metric by selecting **On** and disable visibility by selecting **Off** as per your preference.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Meshing: Specialized Meshing

You can use the meshing features in Ansys Workbench to perform various types of specialized meshing. For more information, refer to:

Mesh Sweeping
MultiZone Meshing
Selective Meshing
Inflation Controls
Mesh Refinement
Mixed Order Meshing
Contact Meshing
Winding Body Meshing
Wire Body Meshing
Pyramid Transitions
Match Meshing and Symmetry
Rigid Body Meshing
Thin Solid Meshing
CAD Instance Meshing
Meshing and Hard Entities
Baffle Meshing
Parallel Part Meshing

Mesh Sweeping

This method of meshing complements the free mesher. If a body's topology is recognized as sweepable, the body can be meshed very efficiently with hexahedral and wedge elements using this technique. The number of nodes and elements for a swept body is usually much smaller than ones meshed with the free mesher. In addition, the time to create these elements is much smaller.

Workbench will automatically check to see if the body fulfills the topological requirements for sweeping. It will then choose **two faces** that are topologically on the opposite sides of the body. These faces are called the source and target faces. Workbench will mesh the source face with quadrilateral and triangular

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

elements and then copy that mesh onto the target face. It then generates either hexahedral or wedge elements connecting the two faces and following the exterior topology of the body.

Note:

- This information applies to general sweeping. For requirements and usage information specific to thin model sweeping, see Thin Model Sweeping (p. 406).
- For descriptions of the sweep option settings, see Sweep Method Control (p. 235).

Requirements for General Sweeping

A body cannot be swept if any of these conditions exist:

- There is a completely contained internal void in the body.
- If more than one source or target face exists on a body. (You must slice the body so that the sub-bodies produced may have one-to-one mapping of source and target bodies or use the MultiZone Method or Thin Sweep that support many-to-one configuration of source and target bodies.)
- A source and target pair cannot be found. That is, the sweeper cannot find at least one path from a source surface to a target surface connected by edges or closed surfaces.
- If a **Sizing** control (p. 296) is used on a body with hard edge sizing and the source and target faces contain hard divisions which are not the same for each respective edge.

When sweeping it is only necessary to apply hard divisions to one leg of the sweep path. If the path has multiple edges, you should apply your controls to that path.

If the sweep path is shared by another body and that path lies on the body's source or target face then more hard divisions may be needed to constrain the sweeper.

When using Virtual Topology with sweeping, avoid creating virtual cells that result in a fully closed surface. Fully closed surfaces cause difficulties for the swept mesher and may result in poor meshes. When selecting adjacent faces for inclusion in a virtual cell, it is best to use Virtual Topology to merge some (but not all) of the faces. A good approach is to use Virtual Topology for the smaller faces, but omit any larger faces from the virtual cell.

To preview any bodies that can be swept meshed, click Mesh on the Tree Outline and right-click the mouse. Select **Show > Sweepable Bodies** from the context menu to display bodies that fulfill the requirements of a sweepable body. However, even if these requirements are met, the shape of the body may at times still result in poorly shaped elements. In these cases, the tetrahedron mesher is used to mesh the body.

The **Show Sweepable Bodies** feature only displays bodies that can be swept in terms of topology where the source and target are not adjacent on an axis. It cannot automatically determine axis-sweepable bodies. However, these bodies can be meshed if a Sweep mesh method is applied and source and target faces are defined. A sweepable body may not be Sweep meshed if the body geometry is not suitable.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Show Mappable Faces is a good tool to diagnose side faces. All side faces should be mappable, but if they are not found to be mappable, it indicates there may be a problem with the topology. For help in diagnosing problems when using the **Sweep** method, refer to the description of the Edge group in the Mechanical help. This toolbar provides access to features that are intended to improve your ability to distinguish edge and mesh connectivity.

Other Characteristics of General Sweeping

Other characteristics of sweeping include the following:

- The general sweeper ignores the **Proximity Gap Factor** (p. 115) setting, which is used to help define the proximity-based sizing (p. 107).
- Hard entities (p. 468) are not supported for the general sweeper.
- If the sweep method is applied to a body and a mapped Face Meshing (p. 314) control is defined for either the body's source or target face, the sweep mesher will fail if a mapped mesh cannot be obtained for the face. See Notes on Face Meshing Controls for Mapped Meshing (p. 327) for related information.
- The source and target faces do not have to be flat or parallel.
- If the topology of the source and target face is the same, the sweeping operation will often succeed even if the shape of the source face is different from the shape of the target face. However, drastically different shapes can cause element shape failures.
- Sweeping does not require your model to have a constant cross section. However, the best results are obtained for constant or linearly varying cross sections.
- For swept meshes with inflation (p. 454) and match control (p. 334), inflation is performed ahead of the match mesh and sweeping. This can affect the sizings on the match controls, which can in turn lead to meshing failure. Therefore, when using both match controls and inflation with sweeping, it might improve meshing robustness if you assign hard edge sizings (p. 300) to the high and low edges of the source face for the sweep.

Figure 117: Axis Sweep Representation

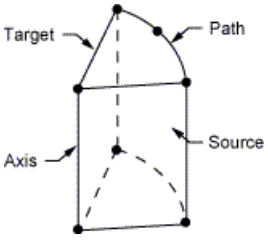
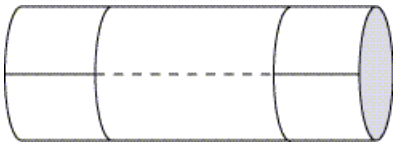


Figure 118: Edge Only Sweep Path

Figure 119: Edge Plus Closed Surface Sweep Path



Rules Followed By the General Sweeper

In deciding which area should be designated as the source area for general sweeping, the program uses the following rules, in the order as listed below. The sweeper will check all the rules until it finds a rule to use. Once a higher order rule is used, all the lower rules will not be considered. For example, if none of the first five rules apply, it will check against rule 6. In this case, if a face is a plane (flat) and the other face is not a plane (not flat), the flat face will be picked as the source and the test will be terminated.

1. **Manually set control - specify both source and target:** The source and target for sweeping will be exactly as you specify. This is the fastest way of meshing. It will eliminate searching for a possible source and target. For axis-sweeping, this method *must* be used. If a face is a source of another body and is not picked as a source of the current body, the aforementioned face will be used as a source.
2. **Manually set control - specify source:** Once the user specifies a source area, the program will try to find the target suitable to the source. The source will be exactly as specified.
3. **Face Mesh control:** The program finds the face with a mapped **Face Mesh** control applied to it.
4. **Number of loops:** The face with the largest number of loops will be picked as source face.
5. **Number of lines:** The face with the largest number of lines will be picked as source face.
6. **Flat face:** A flat face has higher priority for being a source face.
7. **Less sharing:** In most cases, a face might be used by one or a maximum of two bodies. If every one is flat (plane), the one used by the least number of bodies (that is, used by just one body) will be picked as source face.
8. **Larger area:** The largest area will be picked as the source.

Topological Requirements of the General Sweeper

The general sweeper must have at least one path between the source face and target face. The side faces of the sweep do not need to be singular but they must all be submappable and have single loops. The source face cannot be a closed analytic such as a full cylinder, torus or sphere. However, partial analytics are acceptable as source and target faces.

Note:

Creo Parametric creates unique topological models that no other CAD system creates. In all other CAD systems, non-periodic faces can have only one exterior topological loop. On the other hand, models in Creo Parametric can have non-periodic faces with multiple exterior

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

loops. This type of topology does not pose a problem for the free meshers in the Meshing application. However, it does pose a problem for the general sweeper. As noted above, side faces of the sweep must have single loops. They cannot have multiple exterior loops because if they do, a single path from the source to the target cannot be determined.

Importing the model into the DesignModeler application breaks the face with multiple exterior loops into multiple faces with single loops because the DesignModeler kernel does not support the Creo Parametric topology. Exporting the model from Creo Parametric to IGES or STEP format will also resolve this issue.

Figure 120: Example (a) Showing Invalid Closed Cylindrical Face as Source Face

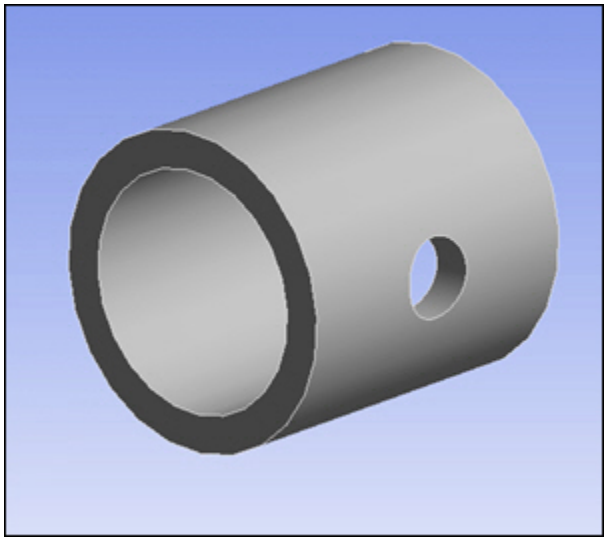
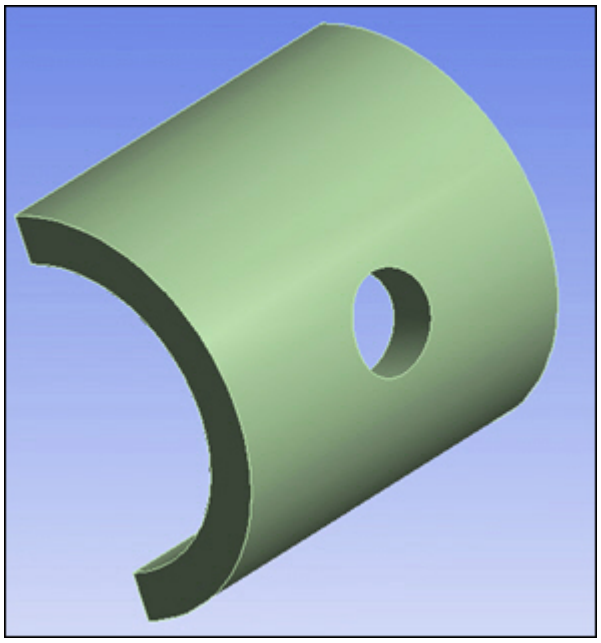
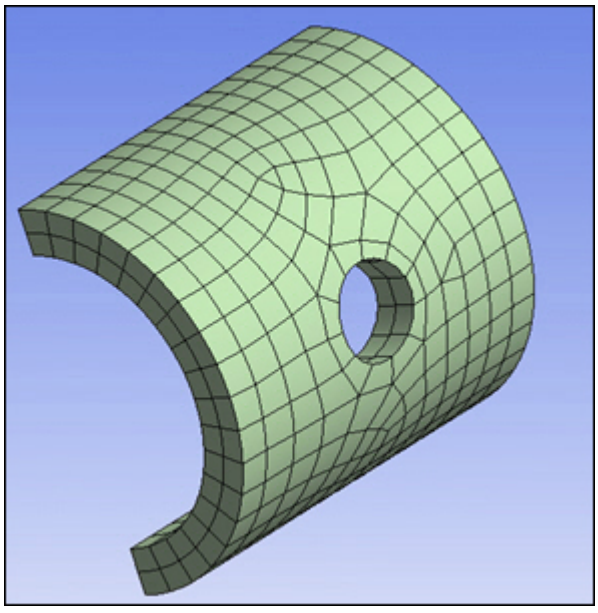


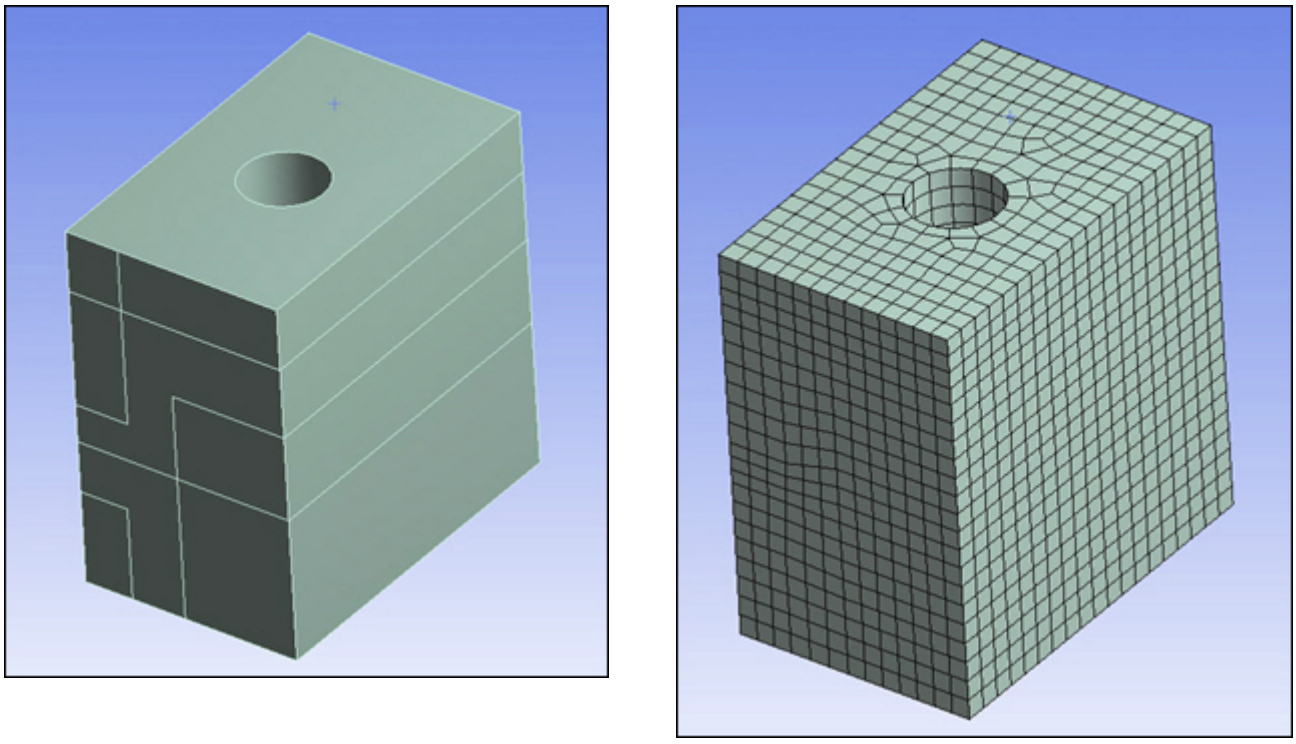
Figure 121: Example (b) Valid Open Cylindrical Face as Source Face





Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

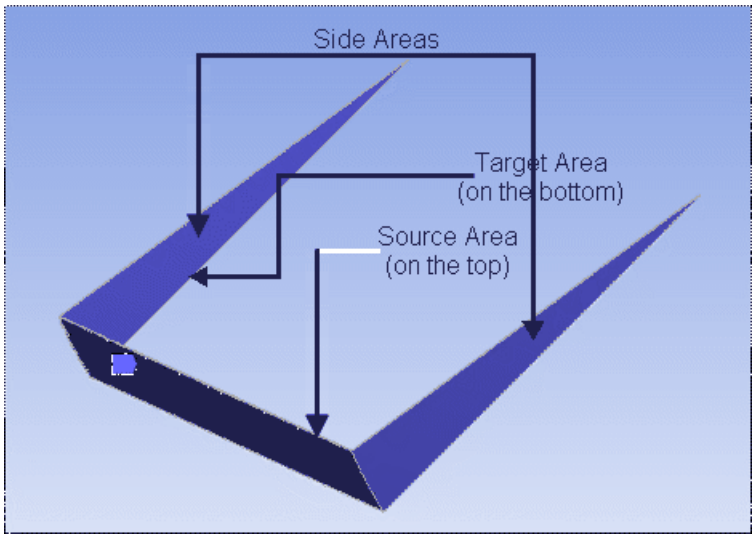
Figure 122: Example (c) Multiple Connected Side Faces



Using General Sweep to Mesh a Narrow Channel Body

The series of images below illustrates the use of general sweep, along with mapped face meshing (p. 314) and hard edge sizing (p. 300) controls, to mesh a narrow channel body. Figure 123: Axial Sweep Model (p. 404) shows the source, target, and side areas of the axial sweep model used in this example.

Figure 123: Axial Sweep Model

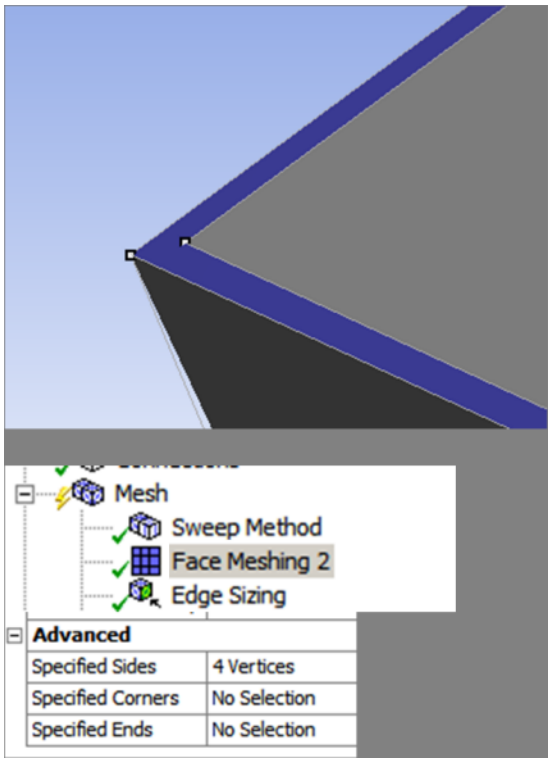


Because the source and target areas to be meshed are a narrow channel and you want them to be meshed with map mesh, it may present difficulties to the mesher. In Figure 124: Axial Sweep Model:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Face Meshing Control (p. 405), a mapped **Face Meshing** control is defined on the source face. Four vertices (two on each side area) have been selected for the **Specified Sides** control.

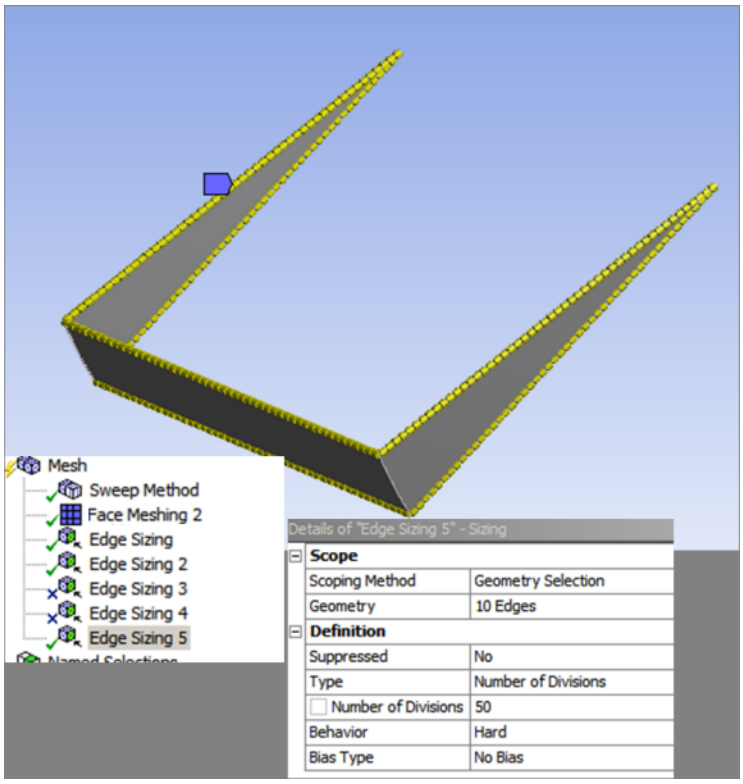
Figure 124: Axial Sweep Model: Face Meshing Control



In cases similar to this example, the key to obtaining a successful mesh is the definition of a hard edge sizing control to make the two paired parallel edges. As shown in Figure 125: Axial Sweep Model: Hard Edge Sizing Control (p. 406), set the **Type** to **Number of Divisions** and enter a value in the **Number of Divisions** field (in this case, 50). The **Hard** option ensures the number of divisions are the same on the pair of edges.

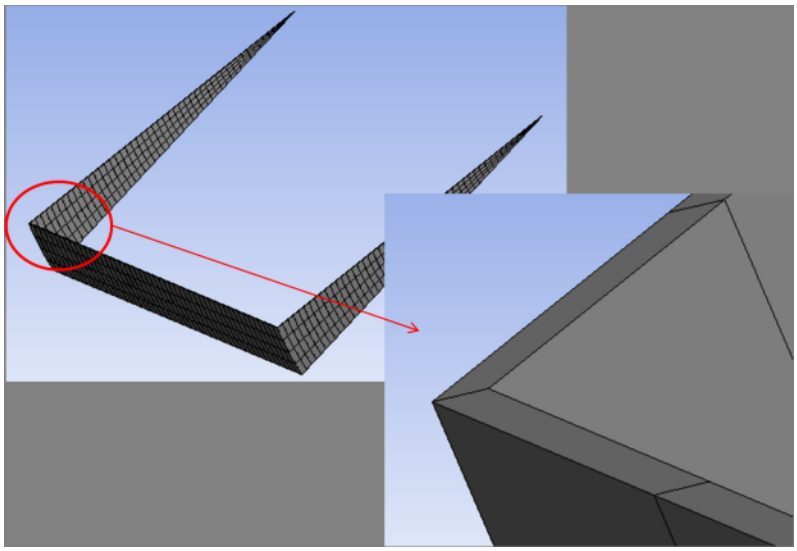
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 125: Axial Sweep Model: Hard Edge Sizing Control



Finally, Figure 126: Axial Sweep Model: Meshed (p. 406) shows the mesh obtained using the settings described above.

Figure 126: Axial Sweep Model: Meshed



Thin Model Sweeping

Similar to the behavior of the general sweeper (p. 399), the thin model sweeper creates a structured hexahedral/wedge mesh, but for a thin model. It meshes one side of the thin solid (the source), and then sweeps the mesh to the other side (the target). Unlike the general sweeper, the thin model

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

sweeper does not require a topological one-to-one match of source to target. The model may have multiple source and/or target surfaces. (Refer to Topological Requirements of the Thin Model Sweeper (p. 408) for examples.) In addition, the thin model sweeper can perform some edge defeaturing and allowing it to mesh models that have reasonably small features.

Requirements and usage information specific to the thin model sweeper include the following:

- The model must be thin—if the model is too thick, the thin model sweeper algorithm may fail.
- The source(s) and target(s) cannot touch each other.
- The model must have an obvious "side" that is perpendicular to the source and target, all of the side areas must connect directly from source to target.
- Mesh controls defined on the target may not be respected.
- Multibody parts are supported.
- For multibody parts, only one division through the thickness is possible. For single body parts, you can define multiple elements through the thickness using the **Sweep Num Divs** control in the Details View of the **Sweep Method**. (See steps below.)
- The thin model sweeper ignores the **Proximity Gap Factor** (p. 115) setting, which is used to help define the proximity-based sizing (p. 107). Using the proximity-based sizing in combination with the thin model sweeper may lead to an unnecessarily long computation time.
- If two bodies intersect to make a "T" connection, the thin model sweeper does not require that a mapped mesh control be defined at the junction of the two bodies.
- The **Preview Source and Target Mesh** and **Preview Surface Mesh** features do not support the thin model sweeper.
- See Notes on Face Meshing Controls for Mapped Meshing (p. 327) for information about using mapped Face Meshing controls with the thin model sweeper.

Considerations for Selecting Source Faces for the Thin Model Sweeper

The thin model sweeper meshes one side of a thin solid (the source), and then sweeps the mesh to the other side (the target). You can control which side the mesher uses as the source by selecting source faces manually. (To do so, set the **Src/Trg Selection** control to **Manual Thin** as described below.)

For most geometries, you can select just 1 of the faces in the complete set of faces that you want to be used as the source set, and the mesher will properly identify the other faces that are a part of that source set. However, for more complicated models (such as those containing multibody parts), you need to select all source faces in the source set in order for the mesher to be successful in finding the complete set of source faces.

A general rule of thumb is if you can select a single face and then extend the selection to its limits, the mesher can also identify the proper complete set of source faces. (For details about extending selections, refer to the description of the Extend Selection command in the Mechanical help.) If the geometry contains sharp angles that make the limit extension selection difficult, it will also be difficult for the mesher to use a single face for the source face definition, and you should select the complete set of source faces.

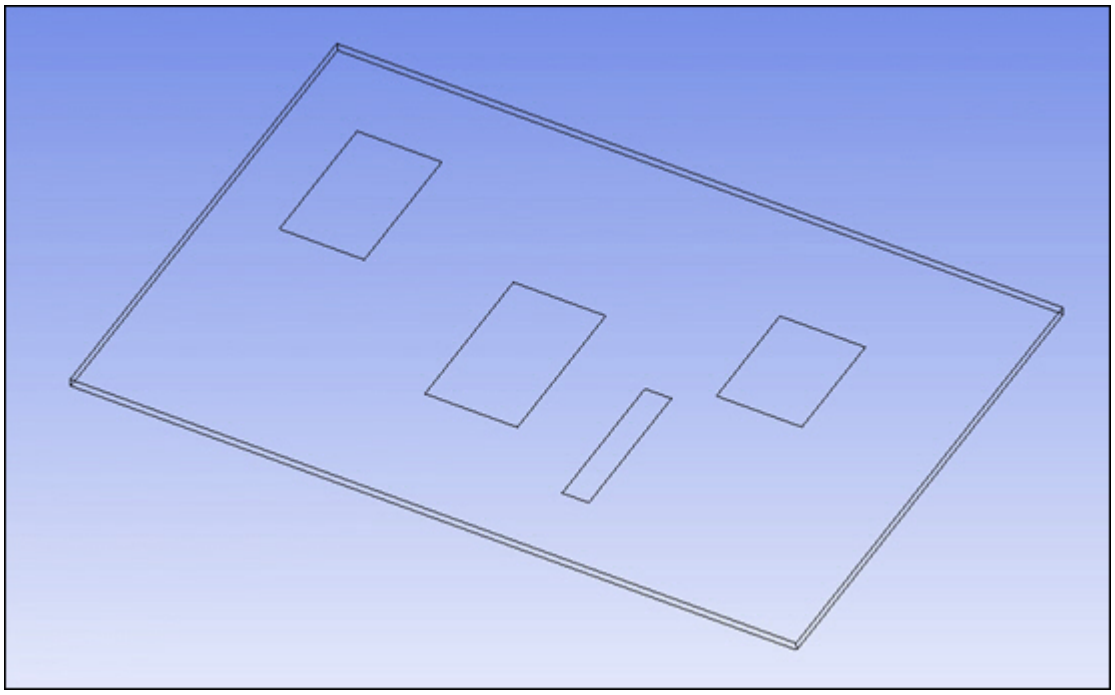
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Topological Requirements of the Thin Model Sweeper

If a thin model mesh fails, turn on the Edge Coloring > By Connection option to see if the edge connectivity is unusual. In some cases, the geometry connectivity may not be as expected, and this may create problems during meshing. These problems can be fixed in the DesignModeler application, the CAD package, or possibly through the use of virtual topologies (p. 545)

The thin model sweeper supports M source faces to N target faces, where M and N can be any positive whole numbers. Between source faces and target faces, there must be "side faces." The angles between side faces and either source faces or target faces must be sharp enough that the faces are NOT considered to be smoothly connected. Therefore, a knife with a thin blade would not be appropriate for thin model sweeping because the cutting edge (blade) does not form a "side face." During the thin model sweeping meshing process, the features (vertices, edges, and faces) on the target may not be preserved and therefore, you should avoid applying boundary conditions to the target. The side faces must connect to both source and target. No edges or vertices are allowed on side faces. In this sense, no hard edges (p. 468) on side faces are allowed. Side edges must connect directly from source to target. You can use virtual topology to eliminate some features.

Figure 127: Example (a) N Source to 1 Target or 1 Target to N Source Topology



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 128: Example (b) N Source to N Target Topology

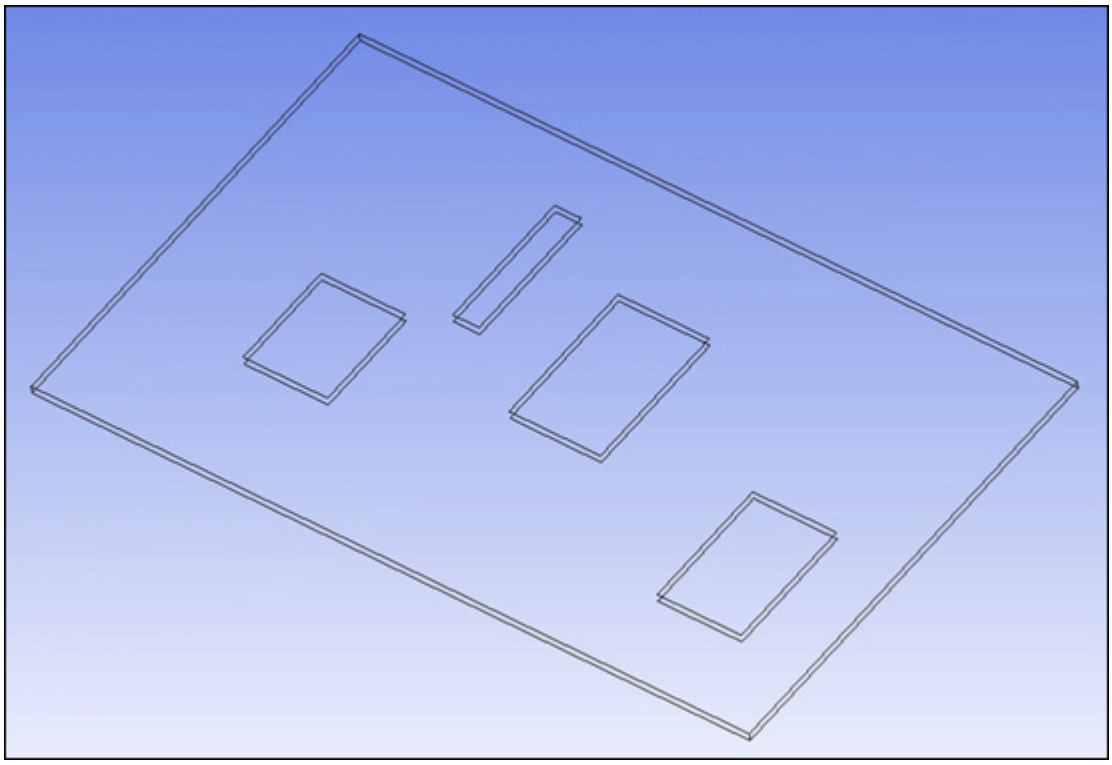
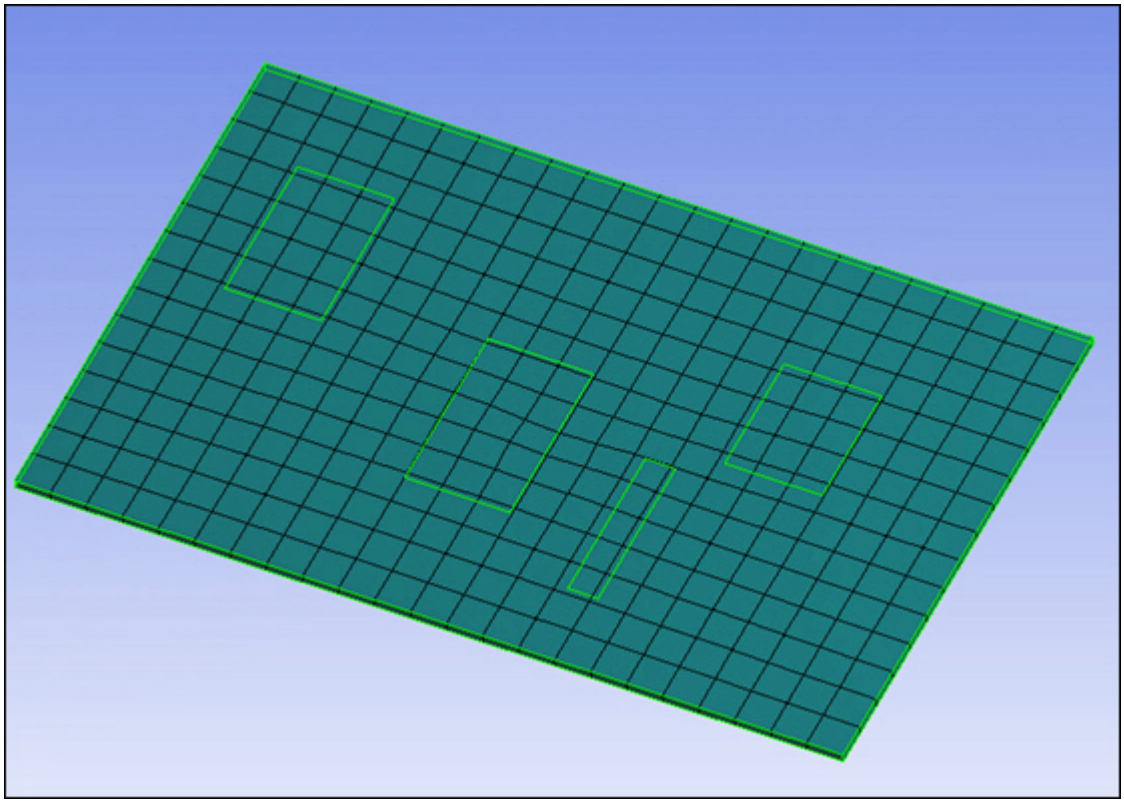


Figure 129: Example (c) 1 Source to N Target Mesh



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 130: Example (d) N Source to 1 Target Mesh

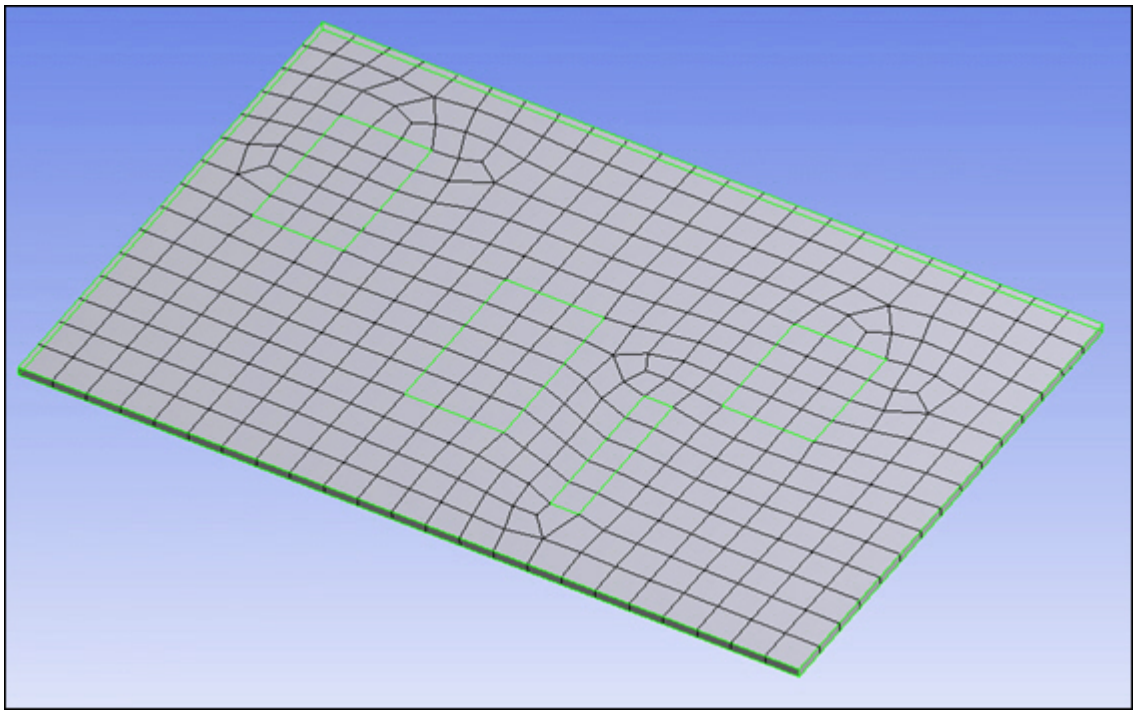
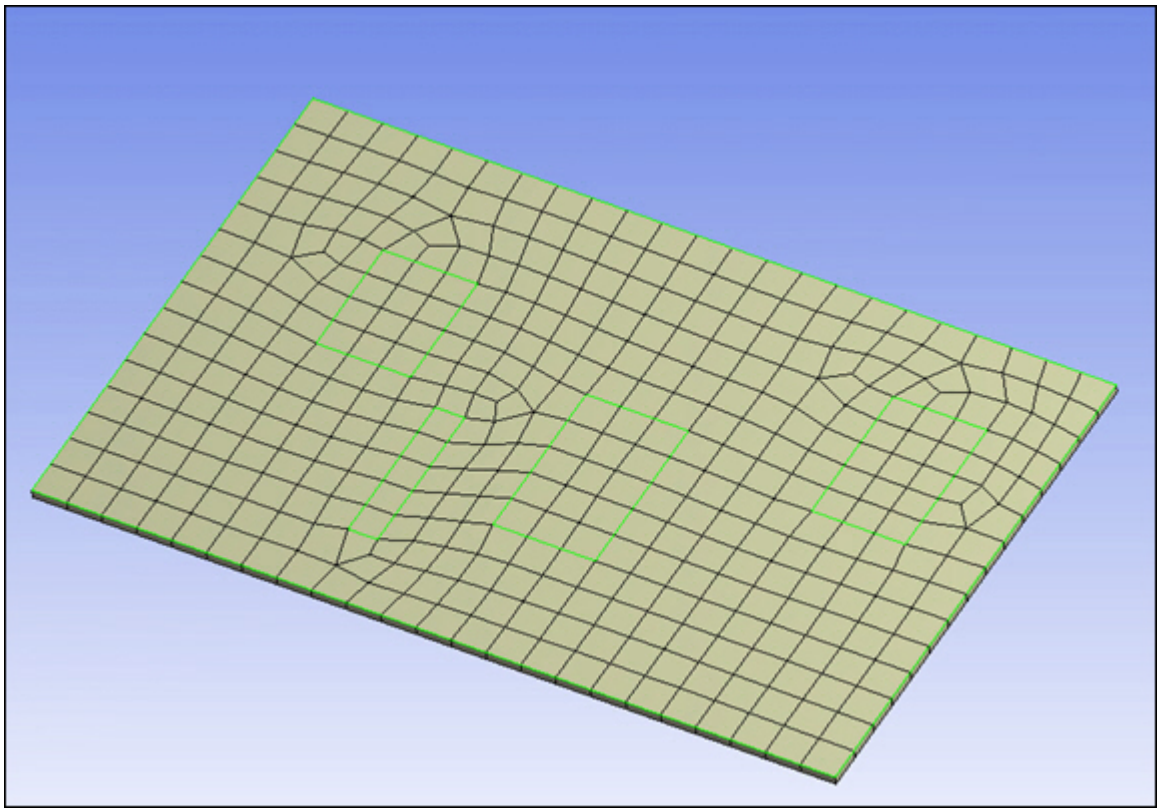


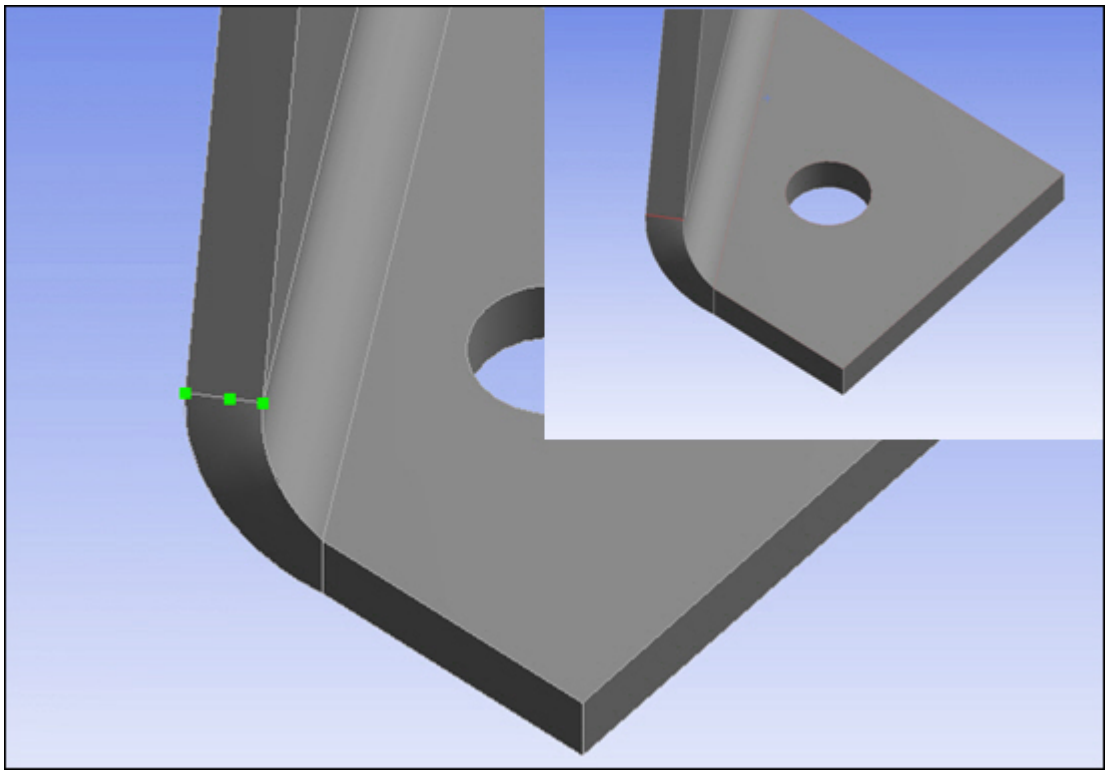
Figure 131: Example (e) N Source to N Target Mesh



Use Virtual Topology to create a single edge between source and target faces.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 132: Using Virtual Topology to Create Single Edge Between Source/Target Faces

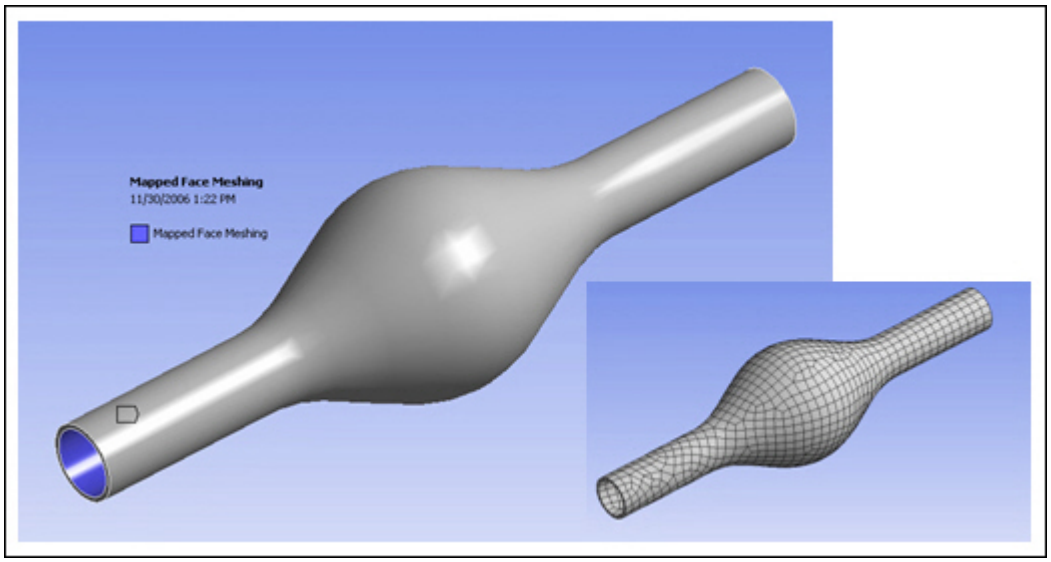


Mesh Controls and the Thin Model Sweeper

Mesh Controls applied on the target faces/edges are ignored. Only mesh controls applied to the source faces/edges are respected.

In example (a) below, the Mapped Face Control is ignored because it is applied to the target face.

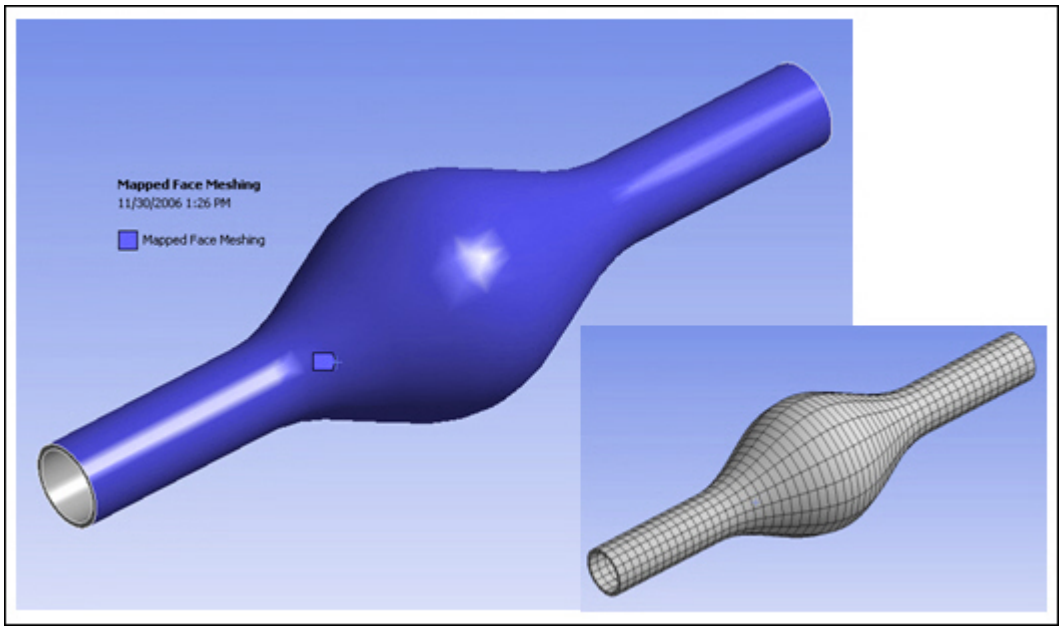
Figure 133: Example (a) Mapped Face Control Applied to Target Is Ignored



In example (b) below, the Mapped Face Control is respected because it is applied to the source face.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 134: Example (b) Mapped Face Control Applied to Source Is Respected



Thin Model Sweeping for Single Body Parts

This section provides the basic steps for using thin model sweeping to mesh a single body part.

To use the Thin Model Sweeper to mesh a single body part:

1. Click the **Mesh** object in the Tree and select **Insert> Method** from the context menu.
2. Scope the **Method** control to the thin body.
3. In **Details> Definition**, set **Method** to **Sweep**.
4. Set **Src/Trg Selection** to **Manual Thin** or **Automatic Thin**.

   Although **Automatic Thin** may work for simple cases, you may need to select **Manual Thin** depending on the complexity of the model.
5. If you selected **Manual Thin**, scope the source face(s), remembering the recommendations provided in Considerations for Selecting Source Faces for the Thin Model Sweeper (p. 407).
6. Enter additional sweep option settings, as desired, in the Details View. These may include **Free Face Mesh Type**, **Sweep Num Divs**, and **Element Option**. For descriptions of these options, see Sweep Method Control (p. 235).
7. Define other mesh controls, as desired.
8. Generate the mesh.

Figure 135: Thin Solid Sweeper Used to Mesh a Single Body Part (p. 413) shows a model of a timing cover that consists of a single body. The thin solid sweeper was used to mesh the body. To obtain this mesh, **Free Face Mesh Type** was set to **Quad/Tri**, **Sweep Num Divs** was set to **2**, and **Element Option** was set to **Solid Shell**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 135: Thin Solid Sweeper Used to Mesh a Single Body Part

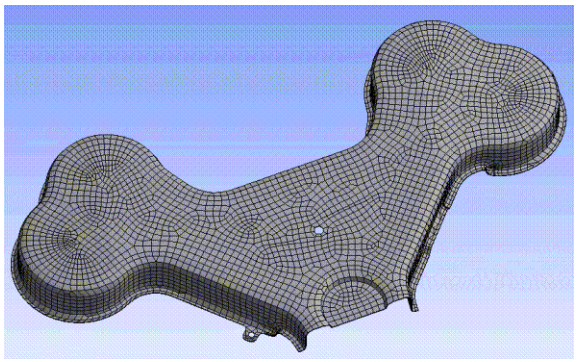
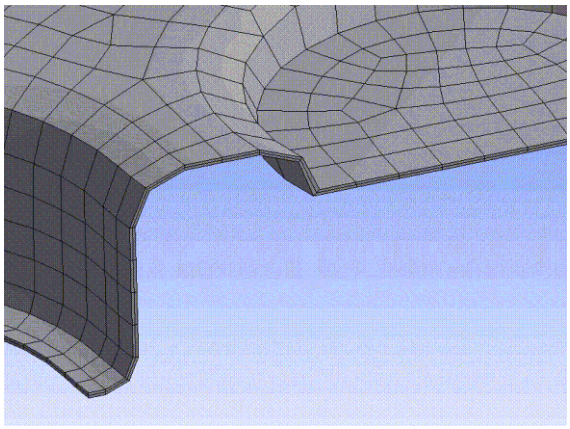


Figure 136: Thin Solid Sweeper Used to Mesh a Single Body Part: Detail (p. 413) shows detail of the timing cover. The **Sweep Num Divs** setting of **2** is apparent in this view.

Figure 136: Thin Solid Sweeper Used to Mesh a Single Body Part: Detail



Thin Model Sweeping for Multibody Parts

This section provides the basic steps for using thin model sweeping to mesh multibody parts. You can define thin sweep for each thin body in the multibody part.

To use the Thin Model Sweeper to mesh a multibody part:

1. Select a thin body in the **Geometry** window, right-click, and select **Insert> Method**.
2. Set **Method** to **Sweep**.
3. Set **Src/Trg Selection** to **Manual Thin** or **Automatic Thin**.

   Although **Automatic Thin** may work for simple cases, you may need to select **Manual Thin** depending on the complexity of the model.
4. If you selected **Manual Thin**, scope the source face(s) of the thin body, remembering the recommendations provided in Considerations for Selecting Source Faces for the Thin Model Sweeper (p. 407).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Enter additional sweep option settings for the thin body, as desired, in the Details View. These may include **Free Face Mesh Type** and **Element Option**. For descriptions of these options, see Sweep Method Control (p. 235).

6. If the part contains multiple thin bodies, repeat step 1 through step 5 for each.

7. If the part contains any thick sweepable bodies, repeat step 1 through step 5 for each, but set **Src/Trg Selection** to **Automatic**, **Manual Source**, or **Manual Source and Target** (depending on complexity of the model).

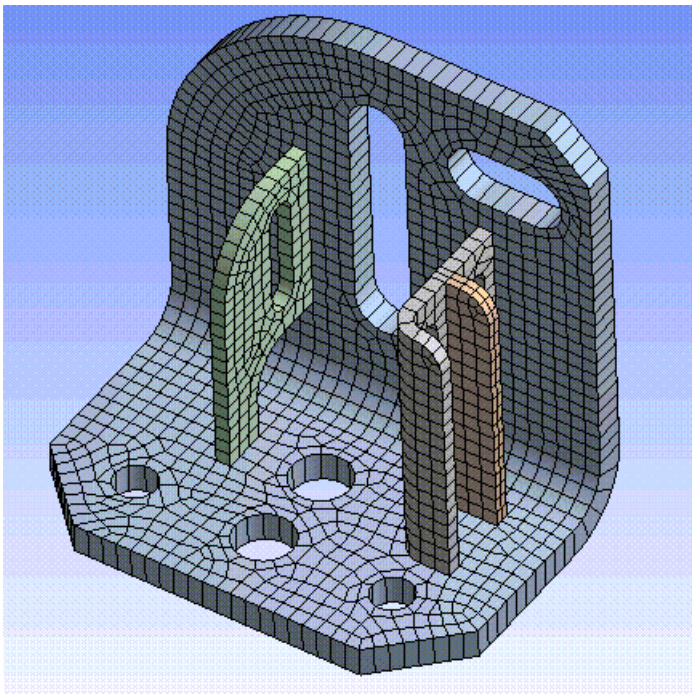
8. If the part contains any non-sweepable bodies, define mesh methods for each, if desired. If the mesh methods are left undefined, the Meshing application will determine the most appropriate methods to use for the non-sweepable bodies.

9. Define other mesh controls, as desired.

10. Generate the mesh.

Figure 137: Thin Solid Sweeper Used to Mesh a Multibody Part (p. 414) shows a model of a bracket that consists of four bodies. The thin solid sweeper was used to mesh the bodies. To obtain this mesh, **Free Face Mesh Type** was set to **Quad/Tri** and **Element Option** was set to **Solid**.

Figure 137: Thin Solid Sweeper Used to Mesh a Multibody Part



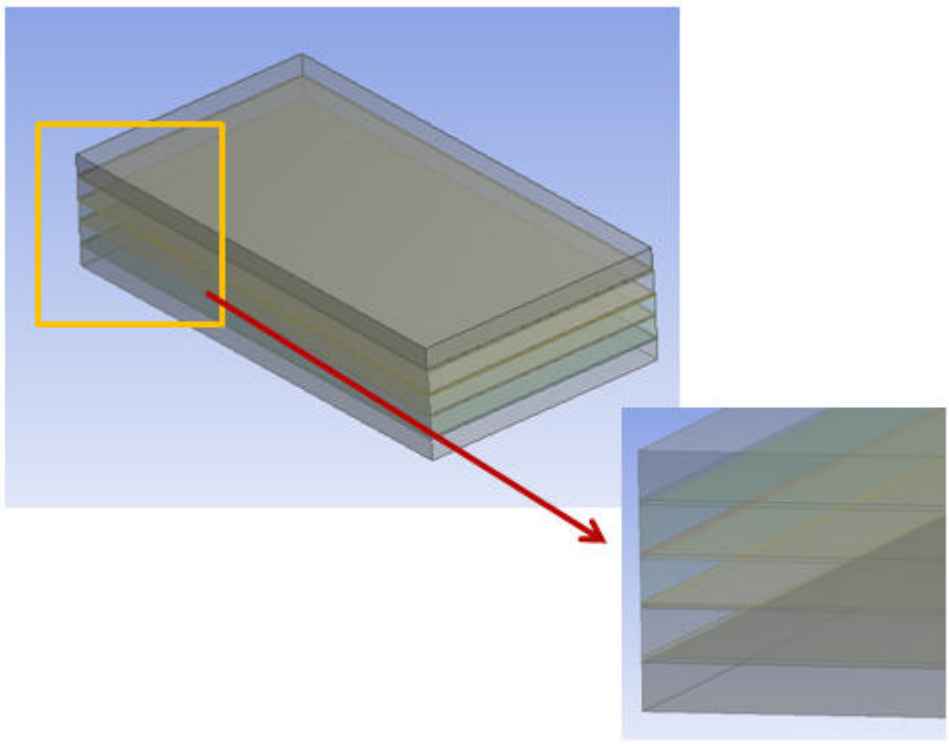
Additional Considerations for Using the Thin Model Sweeper

This section describes several models and scenarios to consider before using the thin model sweeper.

The first example involves a multibody part that models a laminated composite material, as shown below. Defining source faces for such models may be confusing.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 138: Thin Solid Sweeper and Laminated Composite Models

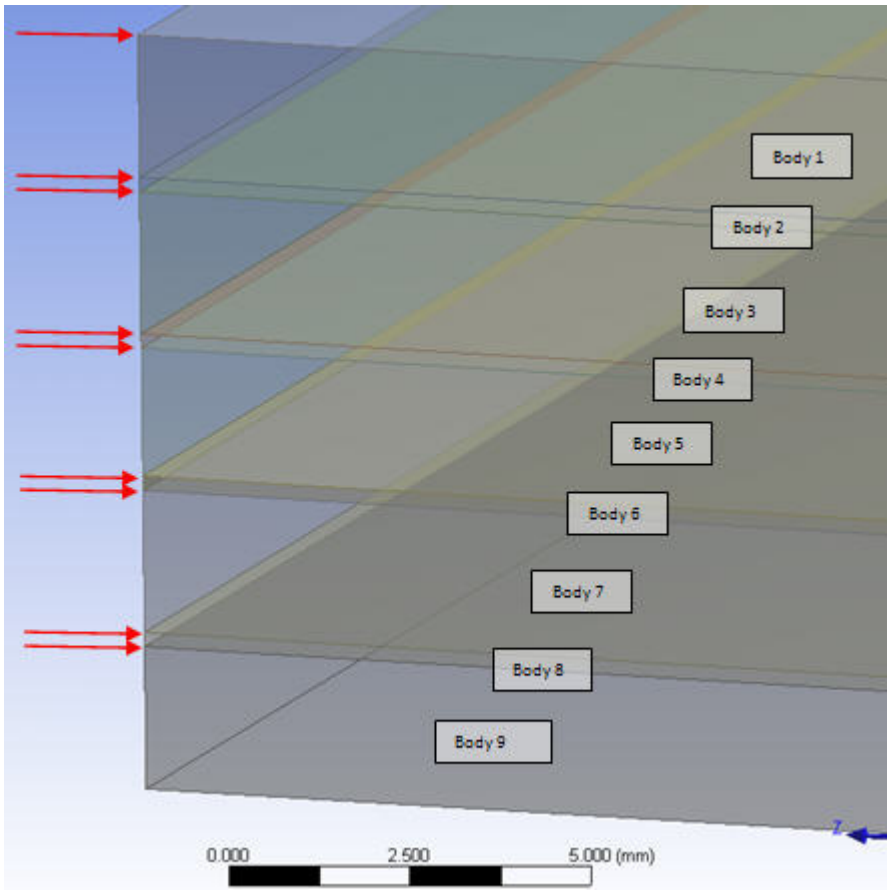


The part contains nine bodies. Assume that the **Manual Thin** option for **Src/Trg Selection** will be applied to each of them. With the **Manual Thin** option, source faces must be defined for each selected body (each body must have at least one face selected as its source face). There are different ways that you can select faces to meet this requirement, and it is logical to assume that defining nine source faces (one for each body) is one way that will work. However, in cases of laminated composites, you must specify *every other face* as a source face.

Consider the figure below, in which nine faces (indicated by arrows) are defined as source faces for the nine bodies. As illustrated by the figure, Body 1 has two faces defined as source faces, and the same is true for bodies 2 through 8. This source face definition causes ambiguity for the thin sweep mesher, which will have trouble determining a target face in bodies 1 through 8 and may fail.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

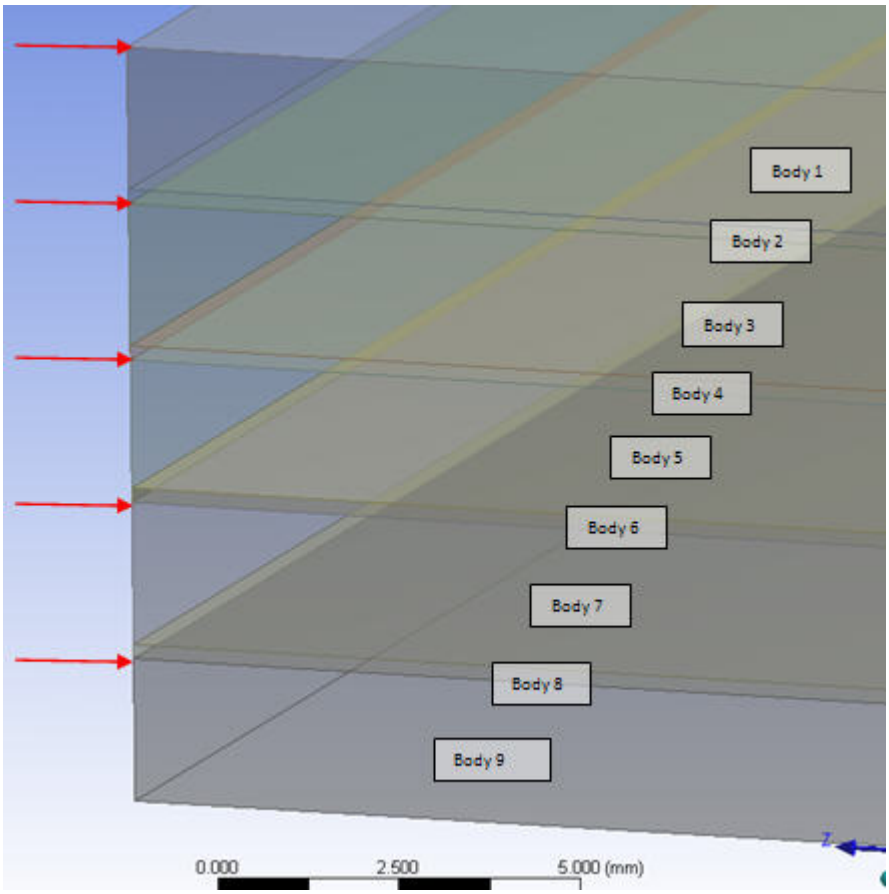
Figure 139: Ambiguous Source Face Definition for Laminated Composite Model



Now look at the figure below. Here every other face has been selected as a source face, for a total of five. With this source face definition, each body still has one face selected to be its source face, so the requirement for **Manual Thin** source face selection has been met. With this source face definition, the thin sweep mesher will have no problem determining target faces for each of the bodies.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 140: Recommended Source Face Definition for Laminated Composite Model



Note:

For cases in which the **Automatic Thin** option can be used, an alternative method to consider is to apply the **Manual Thin** option to only one body, define its source face, and apply the **Automatic Thin** option to the remaining bodies.

Before using thin solid sweeping, remember that the mesher meshes one side of faces and then sweeps the mesh to the second side of faces. Consider Figure 141: Thin Solid Sweeper Limitation (p. 418), which shows a model containing three plates. In the thin sweep operation, the edges that are common to two source faces are present on the source side. If the edges are different on the opposite side, the mesher will use the nodes from the source side in the mesh on the opposite side anyway. Thus, if there are features on the non-source side that are unique and need to be captured, you should not use the thin solid sweeping approach, as the mesher will ignore these features.

In Figure 141: Thin Solid Sweeper Limitation (p. 418), there is no valid way to mesh the middle plate with the thin solid sweep method, as there is an imprint coming from both the plate above and the plate below the middle plate, unless:

1. The plate is decomposed (sliced) to ensure all target face(s) have a corresponding source face.

2. The multibody part is broken into single parts (non-conformal mesh at common interfaces).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Some other mesh method is used. (In Figure 141: Thin Solid Sweeper Limitation (p. 418), a tet mesh method is used.)

4. The source and target faces have similar pairs, and the source faces are selected properly (described below).

Figure 141: Thin Solid Sweeper Limitation

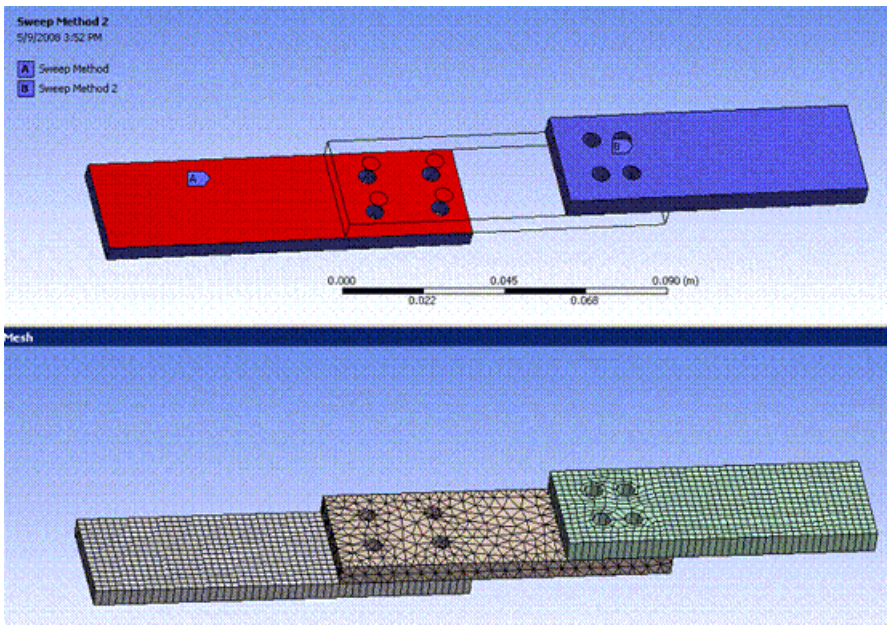
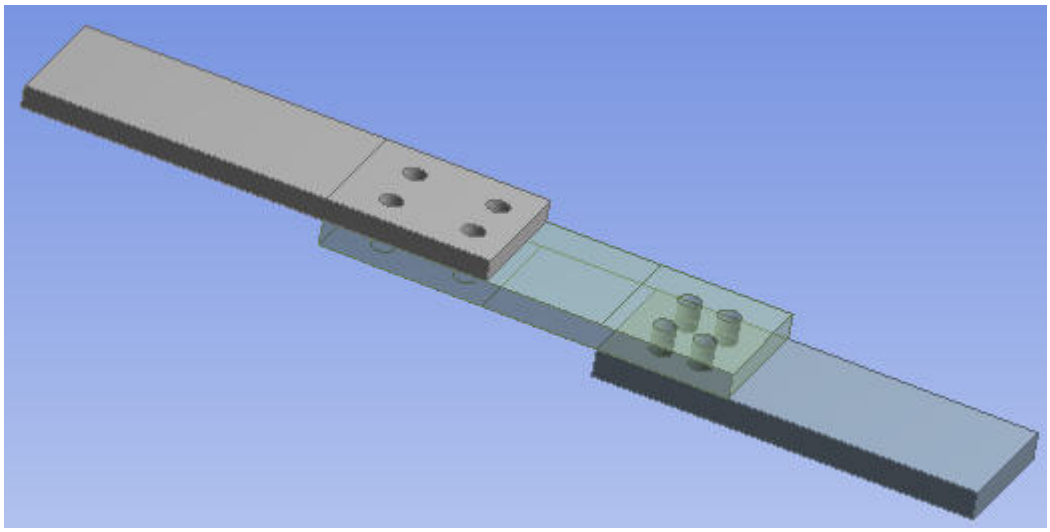


Figure 142: Adding Face Projections (Splits) in the DesignModeler Application (p. 418) illustrates an alternative approach to meshing the model above. In the DesignModeler application, the Projection feature allows face(s) to be split so that the source and target pairs will align. (For this model, the Edges on Face type of projection was used.)

Figure 142: Adding Face Projections (Splits) in the DesignModeler Application



With the addition of the face splits, the model can be meshed successfully with the thin solid sweep method by applying the **Manual Thin** option for **Src/Trg Selection** to all three bodies, and defining the top surface of each body as its source faces, as shown below. In this example, two faces are selected

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

as source faces for the body on the left, three for the middle body, and two for the body on the right. Defining the source faces in this way ensures that everything is meshed from one side of the multibody part to the other.

Figure 143: Defining Source Faces when Face Splits Are Present

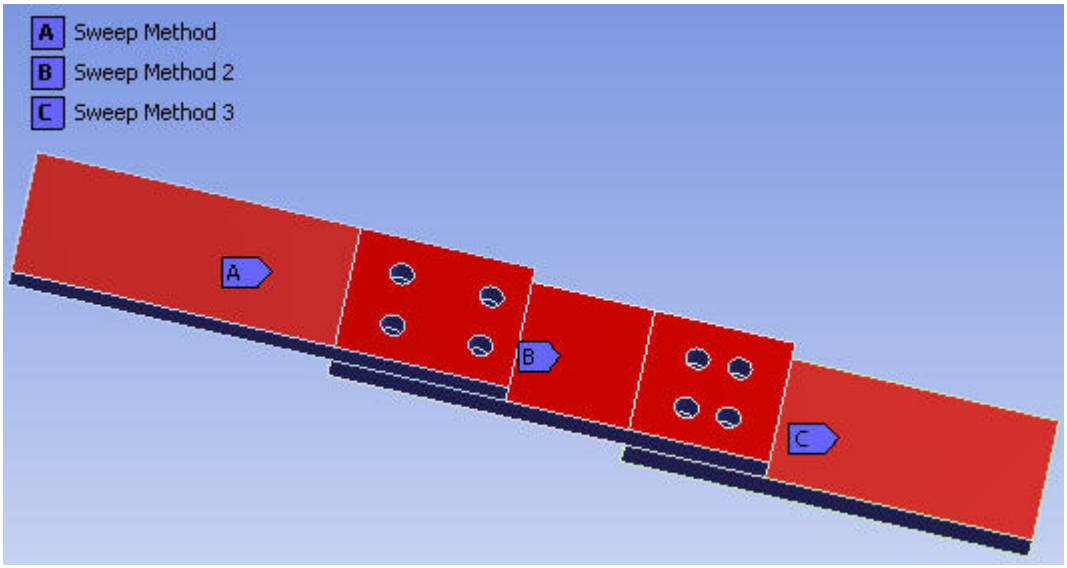
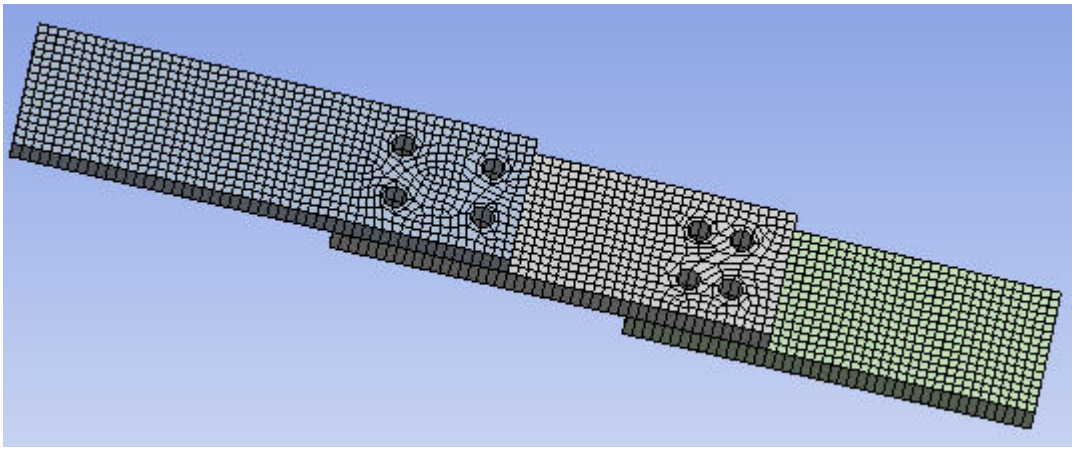


Figure 144: Three Plates Model Meshed with Thin Solid Sweeper (p. 419) shows the meshed model.

Figure 144: Three Plates Model Meshed with Thin Solid Sweeper



MultiZone Meshing

The **MultiZone** mesh method provides automatic decomposition of geometry into mapped (structured/sweepable) regions and free (unstructured) regions. It automatically generates a pure hexahedral mesh where possible and then fills the more difficult to capture regions with unstructured mesh. The **MultiZone** mesh method and the **Sweep** mesh method operate similarly. However, **MultiZone** has capabilities that make it more suitable for a class of problems for which the **Sweep** method would not work without extensive geometry decomposition.

MultiZone meshing topics include:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- MultiZone Method Control (p. 242)
- MultiZone Algorithms (p. 420)
- Using MultiZone (p. 423)
- MultiZone Support for Inflation (p. 440)
- MultiZone Limitations and Hints (p. 442)

MultiZone Algorithms

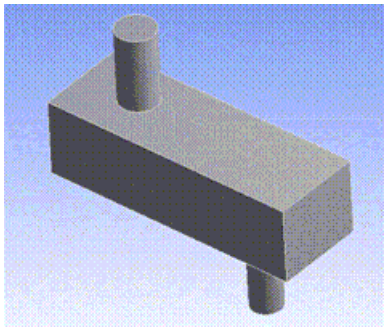
The **MultiZone** mesh method, which is based on the blocking approach used in Ansys ICEM CFD Hexa, starts by automatically blocking out surfaces. If the surface blocking can form a closed volume, then the volume may be filled automatically with a number of structured, swept, or unstructured blocks. The structured blocks can be filled with **Hexa**, **Hexa/Prism**, or **Prism** elements and the unstructured blocks can be filled with **Tetra**, **Hexa Dominant**, or **Hexa Core** elements depending on your settings, as described in MultiZone Method Control (p. 242).

The blocking algorithm and the meshing algorithm used to generate a **MultiZone** mesh are detailed below.

MultiZone Blocking Algorithm

The blocking algorithm used to generate a **MultiZone** mesh can be described as follows. The series of figures illustrates the process, assuming the geometry shown in Figure 145: Blocking Algorithm—Sample Geometry (p. 420).

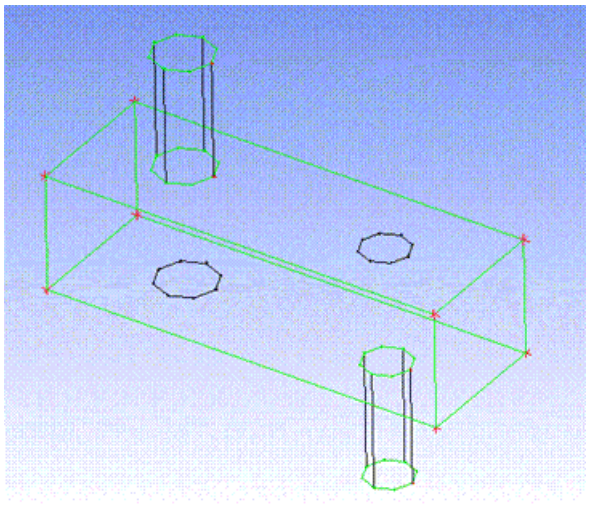
Figure 145: Blocking Algorithm—Sample Geometry



1. Creates automatic surface blocking (2D) by performing a surface analysis. In this step the algorithm:
   - Uses the **MultiZone Quad/Tri** mesh method to generate 2D blocking.
   - Uses mapped controls to try to force mapping. In general, surfaces with four sides are mapped (structured) and other surface patches are free (unstructured).
   - Uses Src/Trg face selection to try to imprint faces into the source/target faces. (**MultiZone** treats all sources/targets as sources, as imprinting can occur from either side. For additional details, refer to MultiZone Source Face Selection Tips (p. 426).)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

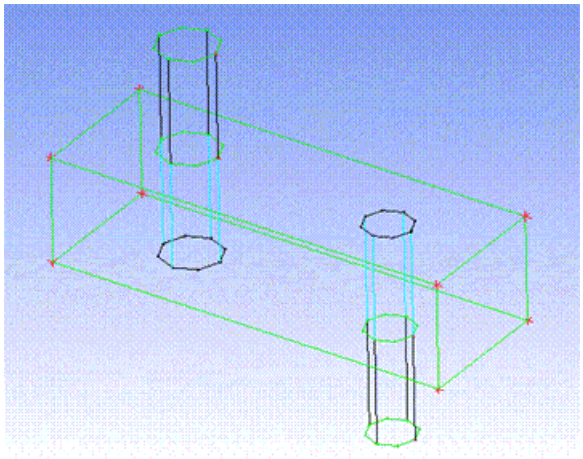
Figure 146: Blocking Algorithm—Step 1: 2D Blocking



2. Creates a **MultiZone** (3D) structure. In this step the algorithm:

   - Processes surface blocks to match swept sections.
   - Connects surface blocks to create volume blocking (3D) using unique heuristic approaches to avoid some of the traditional limitations of sweeping algorithms.
   - Resolves the volume region with structured, swept, and unstructured blocks. Mapped faces become structured blocks and free faces become unstructured blocks.

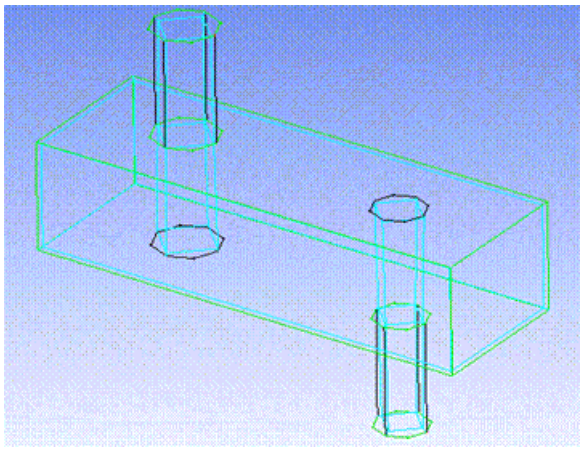
Figure 147: Blocking Algorithm—Step 2: 3D Blocking



3. O-Grid creates boundary blocks automatically, and the algorithm extrudes O-Grid to create inflation.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 148: Blocking Algorithm—Step 3: Inflation



Note:

Inflation can be performed on a combination of side and/or source faces.

MultiZone Meshing Algorithm

The meshing algorithm used to generate a **MultiZone** mesh can be described as follows:

- Assigns intervals by transferring sizing from the topology to blocking.
- Meshes structured faces (those that can be mapped meshed) using transfinite interpolation and unstructured (free) faces with all quadrilateral or a combination of quadrilateral/triangle elements.
- Interpolates, sweeps, and fills the volume mesh for the structured, swept, and unstructured blocks respectively. The structured blocks can be filled with **Hexa**, **Hexa/Prism**, or **Prism** elements and the unstructured blocks can be filled with **Tetra**, **Hexa Dominant**, or **Hexa Core** elements depending on your settings, as described in MultiZone Method Control (p. 242).

MultiZone for Sweepable Bodies

If using Use MultiZone for Sweepable bodies (p. 389), you can use the Show Sweepable Bodies (p. 537) setting to identify bodies that will be meshed using the **MultiZone** method. Although **MultiZone** can mesh other, more complex bodies, it does not attempt this unless you manually insert **MultiZone** mesh method controls.

Other notes about the behavior of **MultiZone** when meshed in this way:

1. When meshing using **Automatic** approach, **MultiZone** uses **Preserve Boundaries** = **All** (Note this is different than the default when scoping a body to a MultiZone mesh method) and no defeaturing is done.
2. If **MultiZone** fails to generate a valid mesh due to errors in the mesh quality or topology, the body will be meshed with **Free Mesh Type** = **Tetra**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Bodies scoped to an **Automatic** Mesh Method will also be meshed in the same way.

Using MultiZone

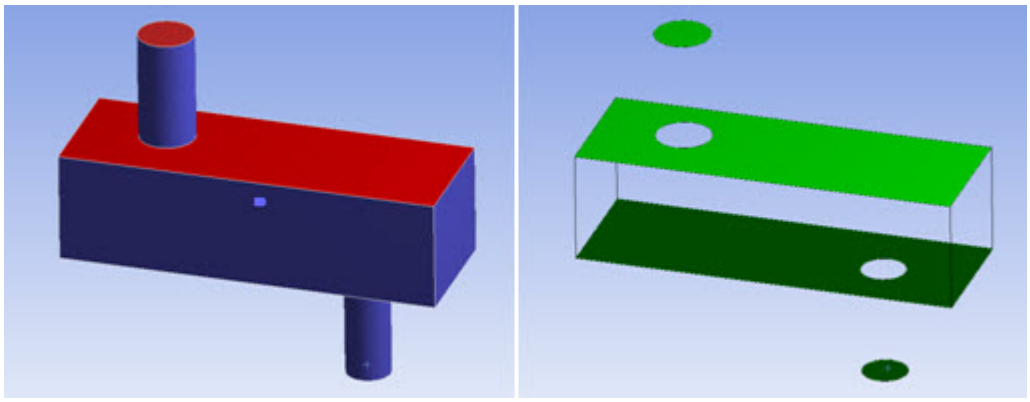
To help you to determine whether the **MultiZone** mesh method is the most appropriate method for your problem and to ensure blocking is constructed properly, you must consider the following characteristics of the problem:

- Sources
- Sweep paths (side faces)
- Intersections (levels)

To classify a **MultiZone** problem, consider which faces will be sources and in turn, which sources will cause imprinting. (Note that imprinting, which is described in MultiZone Source Face Imprinting Guidelines (p. 427), has its own classifications that further define the problem.)

In Figure 149: Classifying the Problem: Sources (p. 423), four faces have been selected as sources.

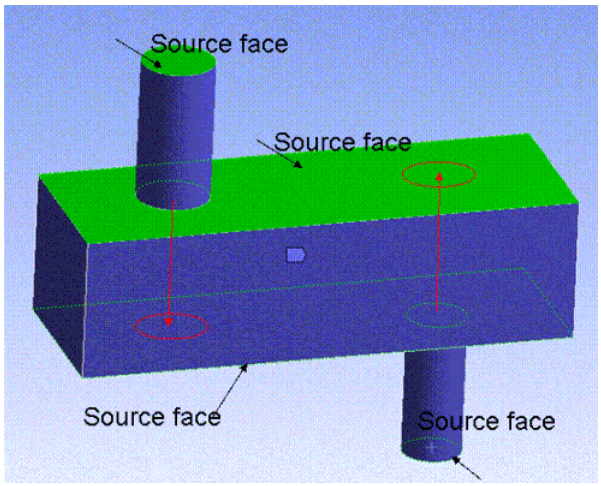
Figure 149: Classifying the Problem: Sources



You must also consider the sweep path, or sides, of the mesh. This includes how many potential sweep directions exist in the problem and if there is more than one, how imprints should be handled. See Figure 150: Classifying the Problem: Handling of Paths and Imprints (p. 424).

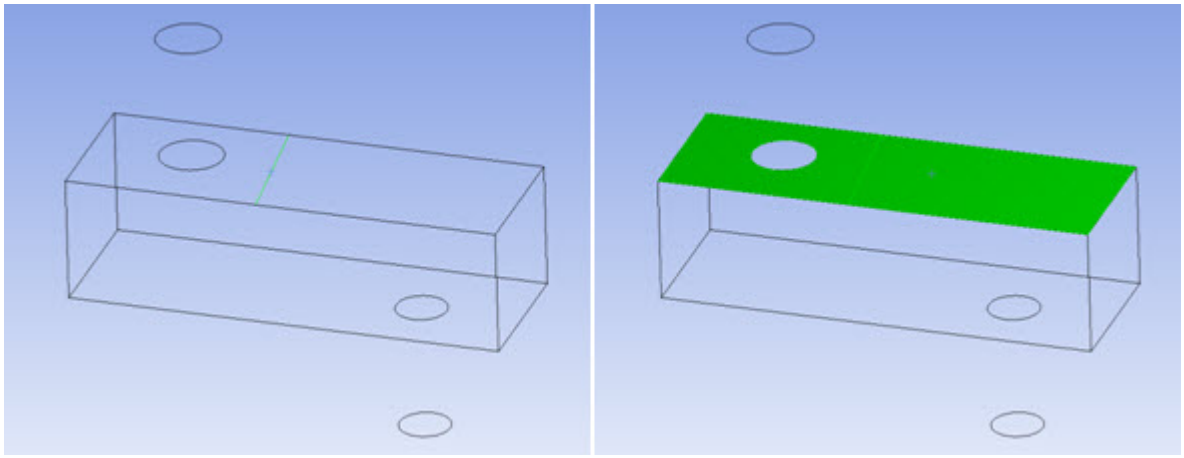
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 150: Classifying the Problem: Handling of Paths and Imprints



A collective set of attached faces in a body is used for imprinting. For example, if one of the faces in this model has a split in it as shown in Figure 151: Collective Source Faces (p. 424), the number of source faces increases by one. However, the set of source faces remains the same.

Figure 151: Collective Source Faces



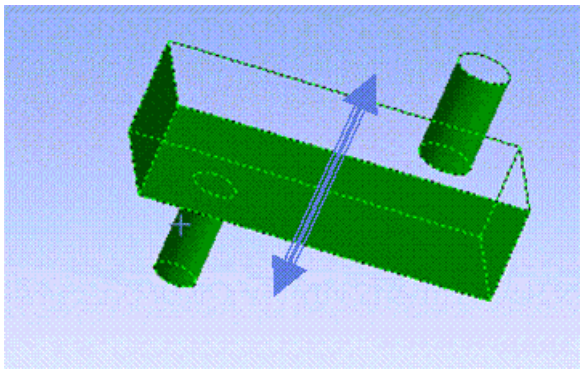
Thus, there are still four levels of source faces. However, now there are additional considerations necessary for imprinting, and handling of side faces.

All faces that are not sources are side faces, and they make up the path. In a simple box example, the box could be swept in any one of three directions, but it is the source selection that determines the path. If no source is selected, **MultiZone** determines the path arbitrarily.

The model shown in Figure 152: Classifying the Problem: Sweep Path (p. 425) shows one clear sweep direction. The green faces are the side faces.

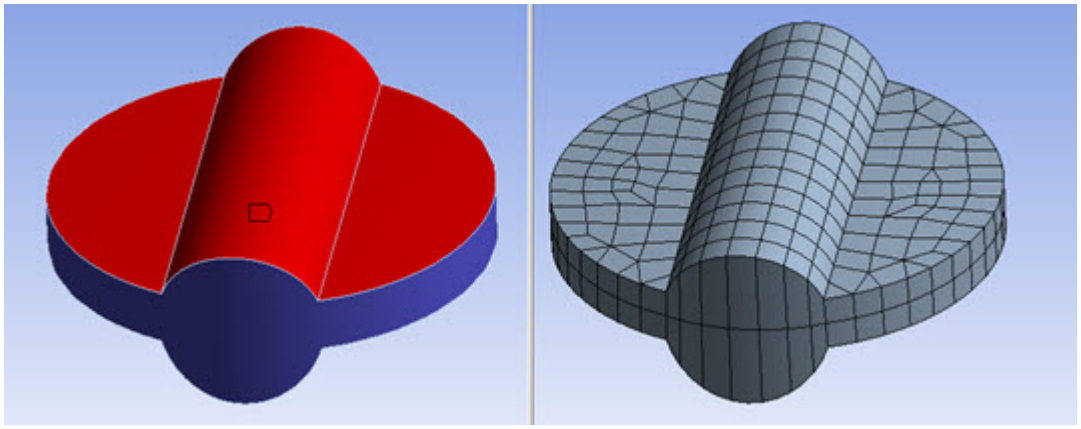
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 152: Classifying the Problem: Sweep Path



By default in **MultiZone**, the mesher is set to automatically determine the sweep path. Since 2D regions and 3D regions are decomposed automatically, this allows the mesher the freedom to choose paths that may not be possible without first subdividing the model. For example, in cases like the valve body shown below, with a traditional sweep meshing approach you would either need to split this body into three regions (center region with path along the axis, and two half cylinders with path top to bottom), or mesh the whole body from top to bottom.

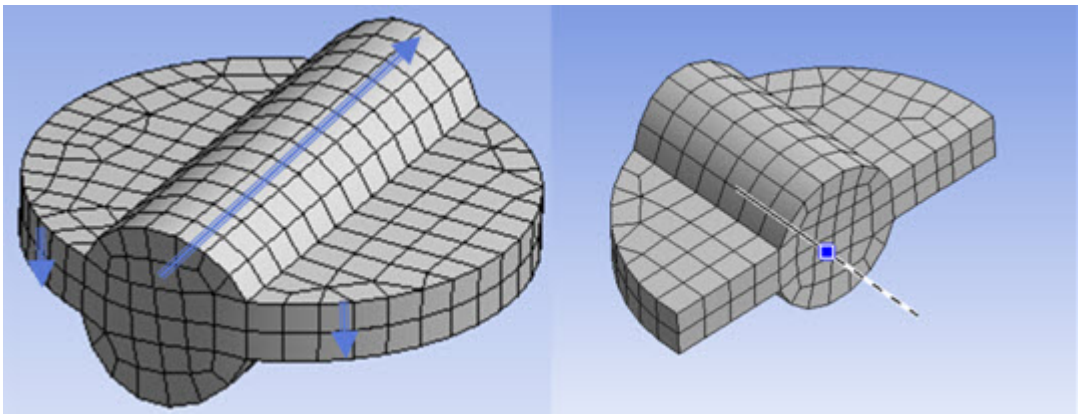
Figure 153: Valve Body: Traditional Approach



With **MultiZone**, the mesher determines that certain portions of the cylindrical region face need to be mapped as a side face, and other portions of it that need to be considered as source faces, and therefore it can decompose the sweep paths automatically.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 154: Valve Body: Automatic Source Faces with MultiZone



In general, you should allow **MultiZone** to define its own sweep path. However, if the path is obvious and there are clear source and side faces, or if **MultiZone** is not able to determine the sweep path on its own, you can guide the mesher by setting manual source faces and ensuring side faces are mappable.

For more information, refer to:

MultiZone Source Face Selection Tips

MultiZone Source Face Imprinting Guidelines

MultiZone Face Mappability Guidelines

Using Virtual Topology to Handle Fillets in MultiZone Problems

MultiZone Source Face Selection Tips

When you choose the **MultiZone** mesh method, the Details View expands to expose various settings (p. 242), one of which is **Src/Trg Selection**. **Src/Trg Selection**, which defines the source and target selection type for the **MultiZone** mesh method, can be set to either **Automatic** (the default) or **Manual Source**. Remember these points when selecting faces:

- Select both source and target faces. **MultiZone** internally decides which faces to take as sources and which to take as targets.
- To do imprinting of face loops, all swept section faces (that is, those that are perpendicular to the sweep direction) should be selected.
- Source faces will generally be meshed with a free mesh. Depending on the geometry, you can add mapped face meshing controls (p. 314) to these faces to get mapped meshing.
- At an interface in a multibody part (in which two bodies share the same face), there is not always a distinction of which body the face belongs to. When using **MultiZone** in such cases, setting the face to be inflated may affect both bodies because both bodies are meshed with **MultiZone** at the same time.
- To make source face selection easier, select **Annotation Preferences** from the Toolbar and then deselect **Body Scoping Annotations** in the Annotation Preferences option box to toggle the visibility of annotations in the **Geometry** window. For example, after scoping **MultiZone** to a body, the body will be displayed using a blue solid annotation. Turn off the body scoping annota-

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tions, then select the source faces. For picking internal faces, the **Hide Faces** right-click option may help you to see inside a body. For example, you can select external faces in the **Geometry** window and then use the **Hide Faces** option to hide the selected faces (making it easier to select the internal faces).

Refer to Figure 101: Source / Target Face Selection for MultiZone (p. 255) for more information.

MultiZone Source Face Imprinting Guidelines

Imprinting has its own classifications that further define the problem. Models often exhibit combinations of these different situations when trying to sort out the sweep paths between pairs of source face(s).

When selecting source faces, it is important to recognize that the pairs of source faces need to have proper matches (that is, each level must be resolved with other levels). If source faces cannot be paired, imprinting must be done by the software to create proper pairs. The software supports only certain types of imprinting, so it is important to understand these cases.

Imprinting classifications include:

Internal Loops

Boundary Loops

Multiple Internal Loops

Multiple Connected Internal Loops

Parallel Loops

Intersecting Loops

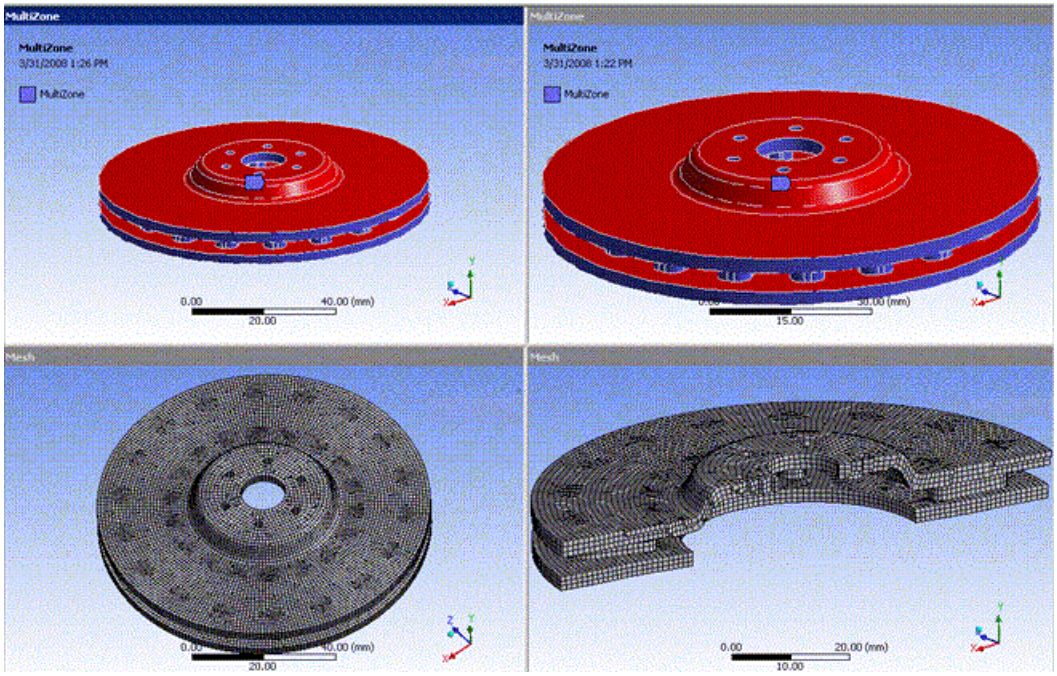
It is also important to note that if you set the **Free Mesh Type** (p. 242) to anything other than **Not Allowed**, some of the imprinting will not be done, as this can lead to some problems in the creation of the free blocks. For example, on some models you could get a pure hex mesh if you set **Free Mesh Type** to **Not Allowed**, but not get a pure hex mesh if you set **Free Mesh Type** to some other value.

Internal Loops

Internal loops are internal regions that imprint to other faces, such as the fins in the rotor model shown below.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

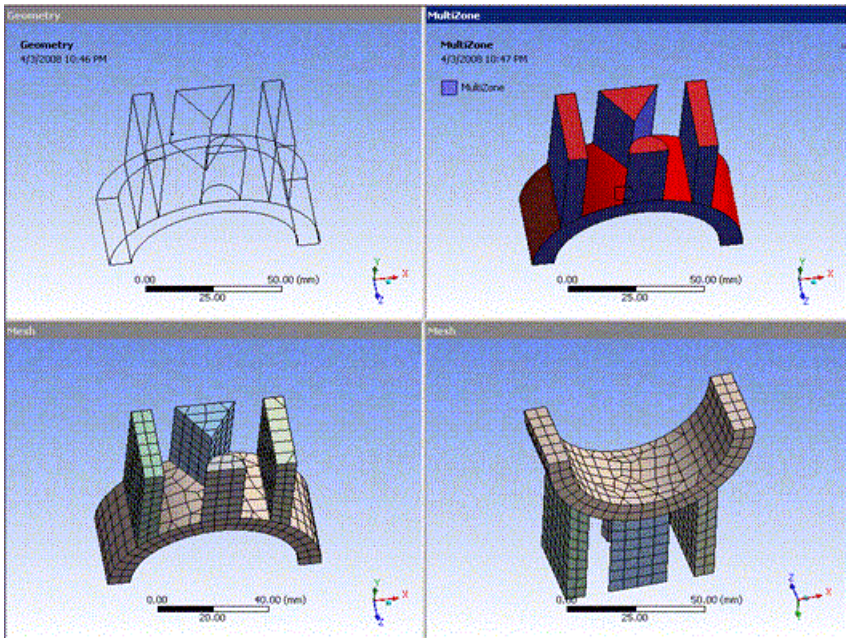
Figure 155: Source Imprinting Classifications: Internal Loops



Boundary Loops

Boundary loops are splits in a face that extend to the boundary, such as those shown in the model below.

Figure 156: Source Imprinting Classifications: Boundary Loops

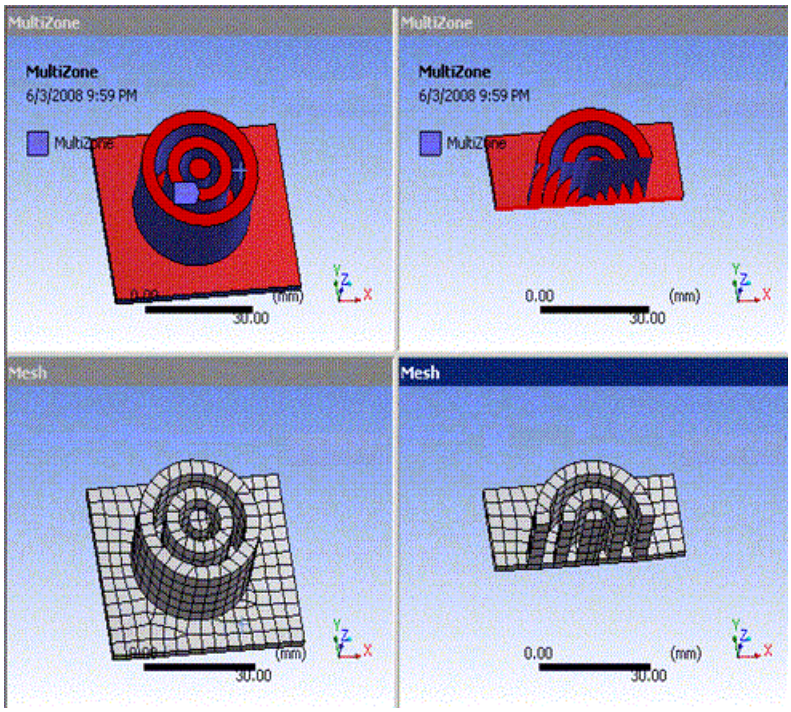


Multiple Internal Loops

Some models exhibit multiple internal loops, such as those in the model below.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

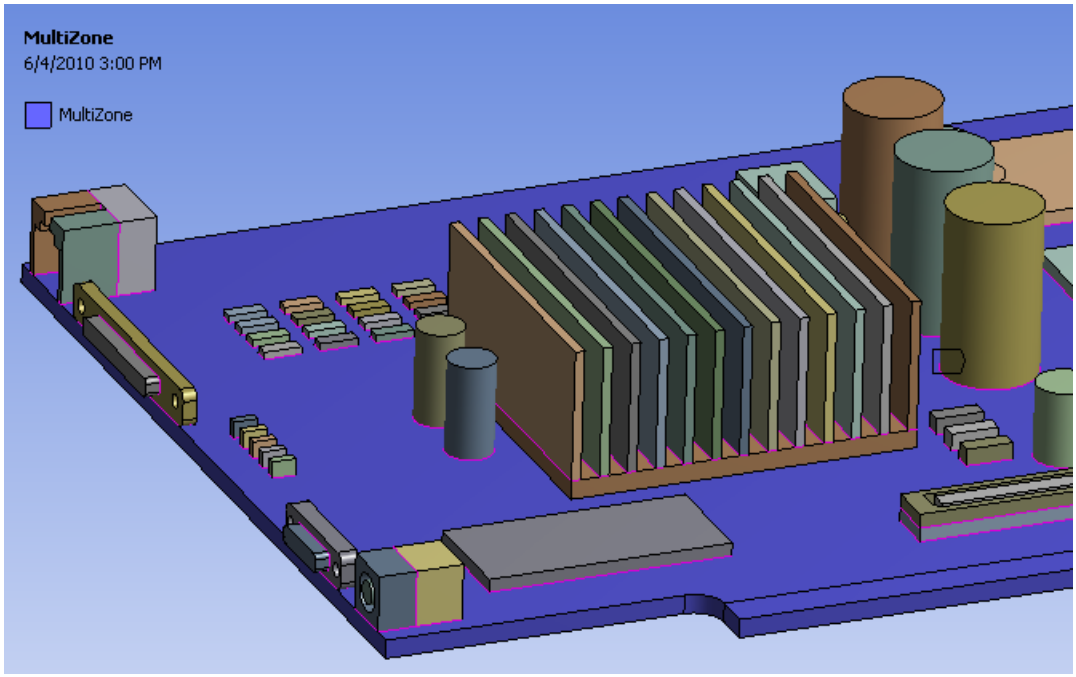
Figure 157: Source Imprinting Classifications: Multiple Internal Loops



Multiple Connected Internal Loops

Some models exhibit multiple connected internal loops, such as those in the model of the circuit board below. The **MultiZone** method has been applied to the underlying board, which is highlighted in blue. The Edge Coloring>By Connection feature is on in this view.

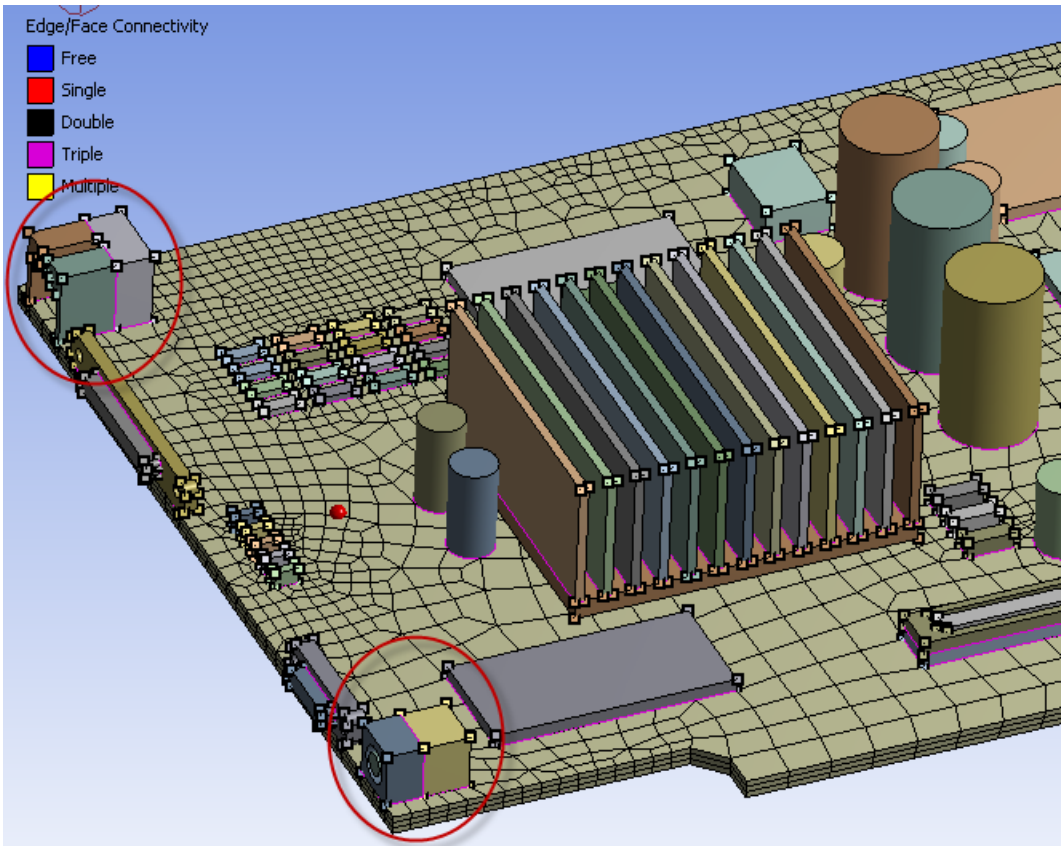
Figure 158: Source Imprinting Classifications: Multiple Connected Internal Loops View 1



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the figure below, the body corresponding to the underlying board was selected and meshed. In this view, both the Edge Coloring>By Connection and Show Vertices features are on.

Figure 159: Source Imprinting Classifications: Multiple Connected Internal Loops View 2

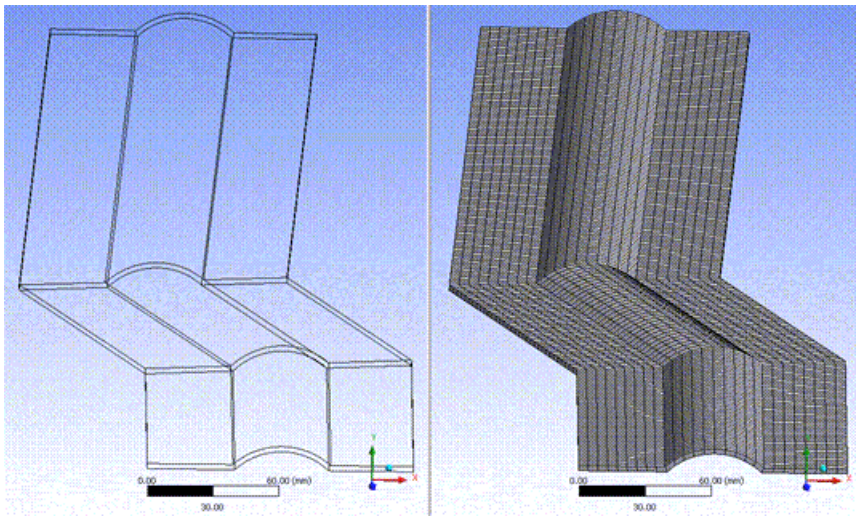


Parallel Loops

Some models exhibit parallel loops, such as those in the model below. If the software can match all pairs these models should work; however, the angle between pairs can cause problems (due to difficulties finding the pairs), and if there are non-matched pairs they will cause problems. For non-matched pairs, virtual topologies may solve the problem. For these kinds of examples, using automatic source face selection is often more robust than manual source face selection.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

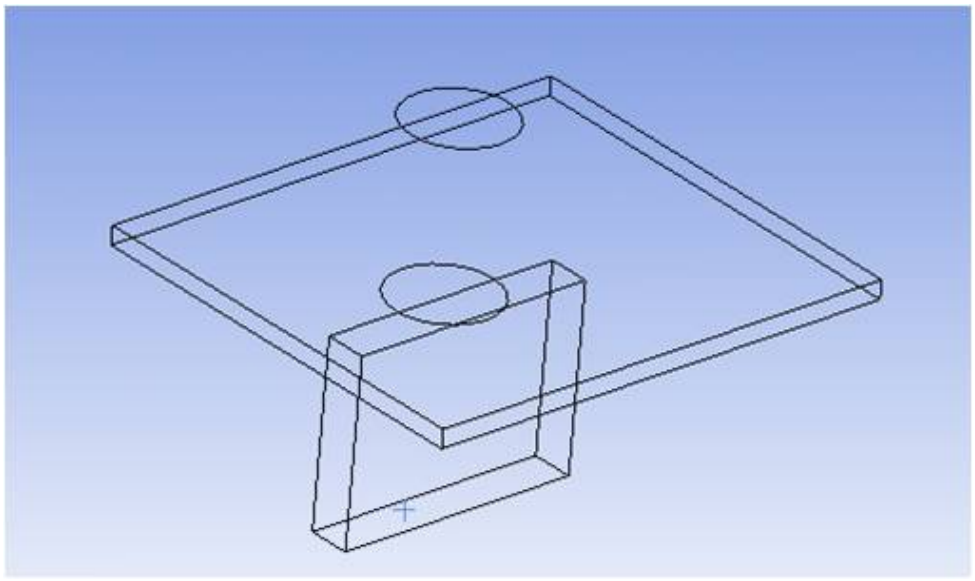
Figure 160: Source Imprinting Classifications: Parallel Loops



Intersecting Loops

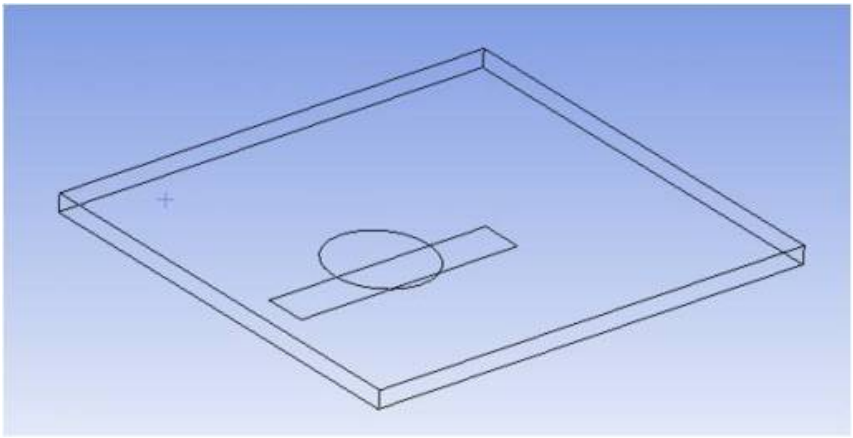
Two views of a model with intersecting loops are shown below. Notice that the sources on the top and bottom would intersect when being imprinted. These types of cases are not supported yet. You can use geometry decomposition as a workaround.

Figure 161: Source Imprinting Classifications: Intersecting Loops View 1



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 162: Source Imprinting Classifications: Intersecting Loops View 2



MultiZone Face Mappability Guidelines

To construct a swept or all hex mesh on typical geometries, side faces generally have to be able to be mapped or submapped. In most cases, if manual source faces are selected with **MultiZone**, all other faces are treated as side faces and are mapped/submapped if possible. However, certain model characteristics can lead to problems in mapping/submapping side faces, and ultimately lead to meshing failure. You can use virtual topologies to correct most issues and obtain successful side face handling and mesh generation. In some cases, you may also want source faces to be mapped/submapped, but that should be a lower priority.

In addition, imprinted regions must have a clear path of connecting side faces and handling intersections throughout the path.

Note:

If all faces of a body have a face meshing control set to **Mapped**, MultiZone will perform additional steps to ensure an all-mapped hex mesh is created. This can be important for cases where source faces of one body are side faces of an adjacent body so the entire body must be meshed with mapped faces in order to mesh the full model.

For guidelines that explain some common situations and steps you can take to resolve faces into mappable regions, see:

- Face Meshing Control (p. 314)
- Side Face Handling of Imprinted Regions (p. 432)

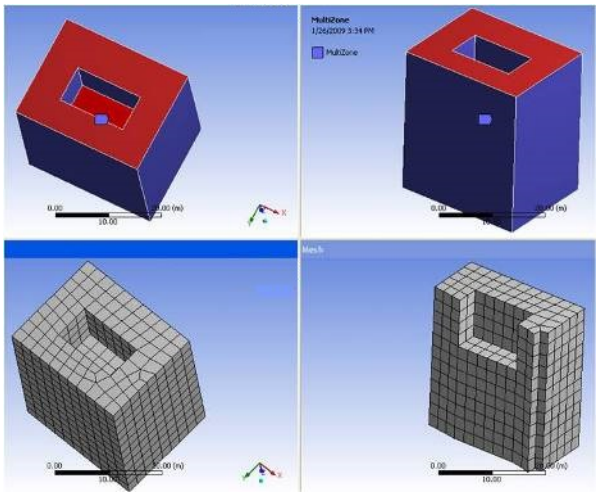
Side Face Handling of Imprinted Regions

When an imprint is made to connect two sets of source faces, side faces are constructed in the process. (See Figure 150: Classifying the Problem: Handling of Paths and Imprints (p. 424).) Generally, the pairs of imprints create a natural set of side faces that are mappable. However, if there are several sets of side faces along the sweep path, the interval edge assignment of the internal mapped faces can become tricky. Because there are not physical faces or edges to help define the interval edge assignment, you are sometimes better off adding more decomposition to help control this (either by splitting some of the external faces, or by slicing the geometry).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The simple cutout case shown below illustrates this.

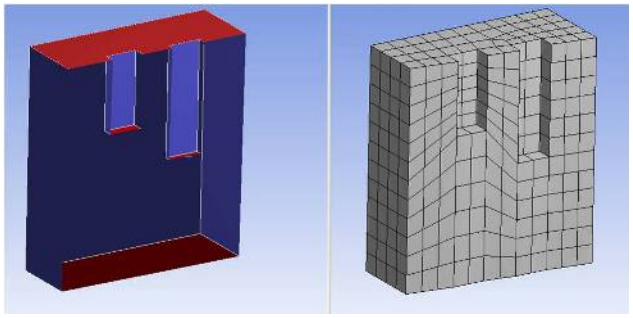
Figure 163: Simple Cutout Case



In the case above, there is a set of source faces at top, center, and bottom. Internal side faces are constructed from the center to the bottom and the interval edge assignment for these internal edges is found by subtracting the number of divisions for the top to center region of sweep path from the entire sweep path.

However, in cases in which the internal cutouts are at multiple levels or the sides do not provide a clear connected path, **MultiZone** could have difficulties constructing side faces with appropriate interval edge assignment unless you set the **Free Mesh Type** (p. 242) to something other than **Not Allowed**, or you perform manual geometry decomposition. Refer to the figure below for an example showing cutouts at multiple levels.

Figure 164: Cutouts at Multiple Levels



In the figure above, notice that cutouts at different levels are meshed at the same level with regard to the grid lines.

When there are intersections between the imprints and the side faces along the sweep path, some complications may arise. For example, the case shown in Figure 165: Intersections Between Levels and Sides (p. 434) has one clear sweep path (top to bottom), six levels (sets of sources), and a need for imprints.

The legend below explains the symbols and display options used in Figure 165: Intersections Between Levels and Sides (p. 434).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

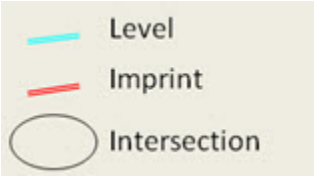
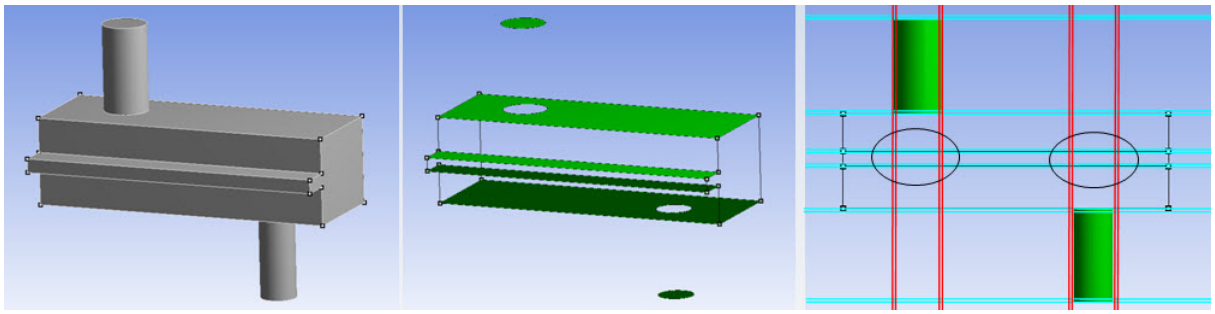
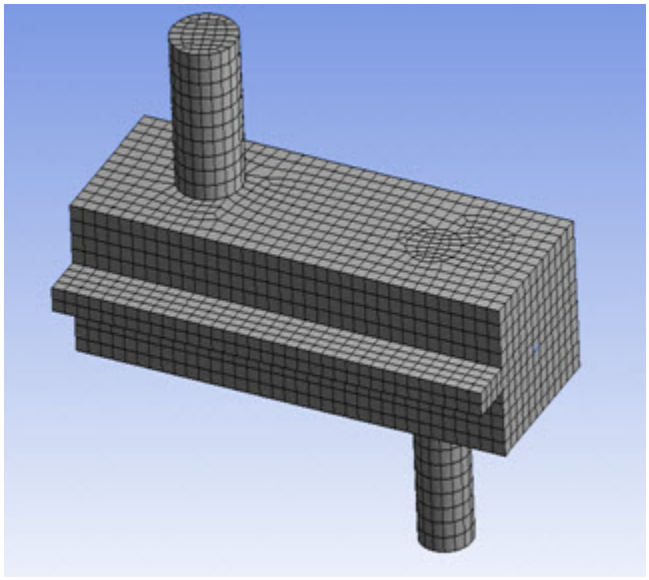


Figure 165: Intersections Between Levels and Sides



To construct a swept mesh on this model, **MultiZone** needs to resolve the intersections between the cylinders, and the side rib. In this case, meshing is successful because the sources are well-defined, and the side face handling is clear (side faces can be submapped appropriately).

Figure 166: Meshed Model



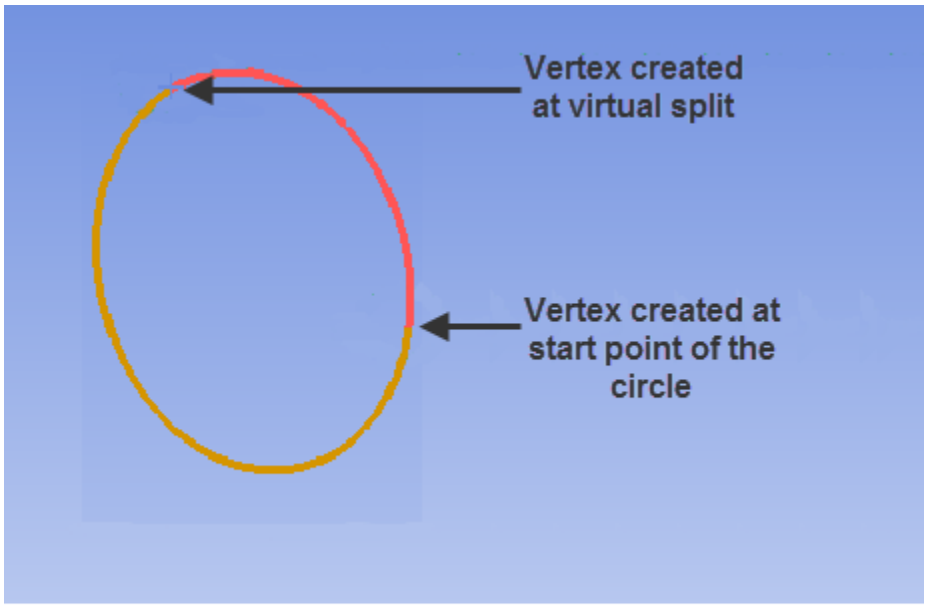
However, if the sources are not well-defined (or incomplete), or the side faces cannot be mapped, **MultiZone** will have difficulties. To fix such problems, add extra decomposition or mesh with the **Free Mesh Type** option.

Cylindrical Side Faces

Cylinders have a start and end point which, at times, can affect the quality of the meshing. When you add a first virtual edge to split a circle (the side face of a cylinder), you create two vertices: one where you split the circle and one at the end point, as shown below.

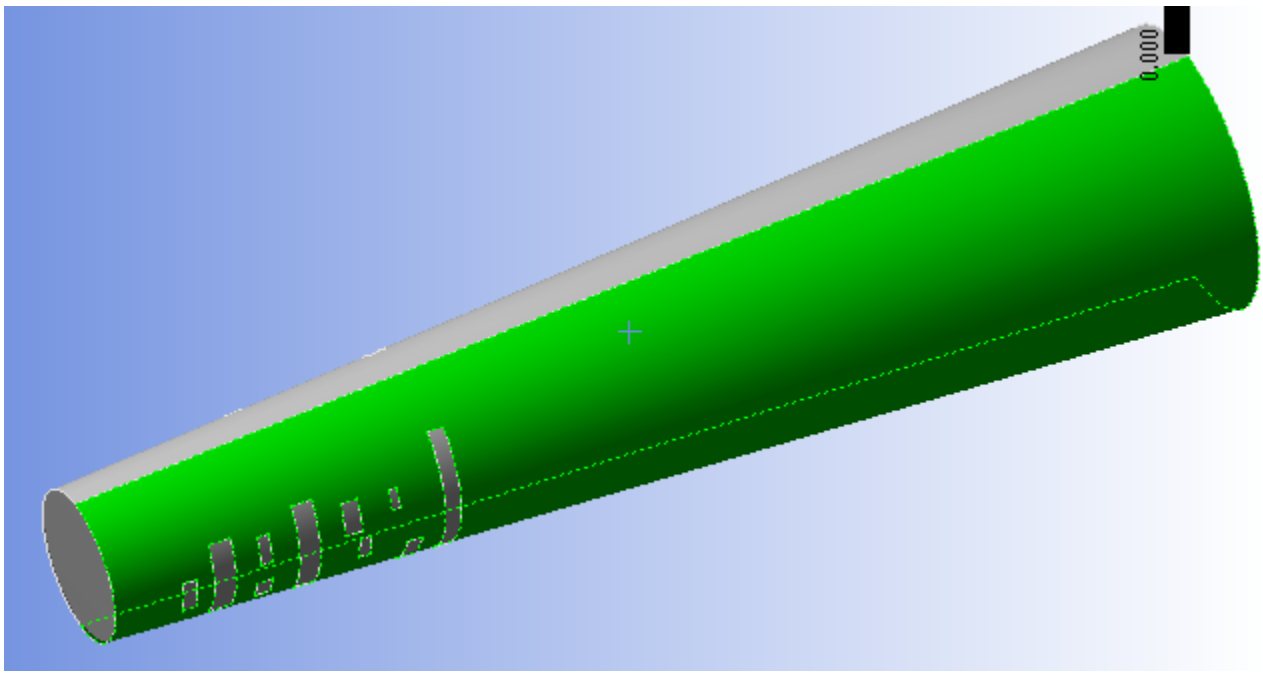
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 167: Vertices in a Split Circle



If **MultiZone** fails to mesh a cylindrical surface with cut-outs folded at greater than 180 degrees, splitting the cylinder along the seam to get a sub-mapped face mesh may help. If you use this method, check to ensure that the split does not hamper mesh quality.

Figure 168: 360 ˚ Cutout



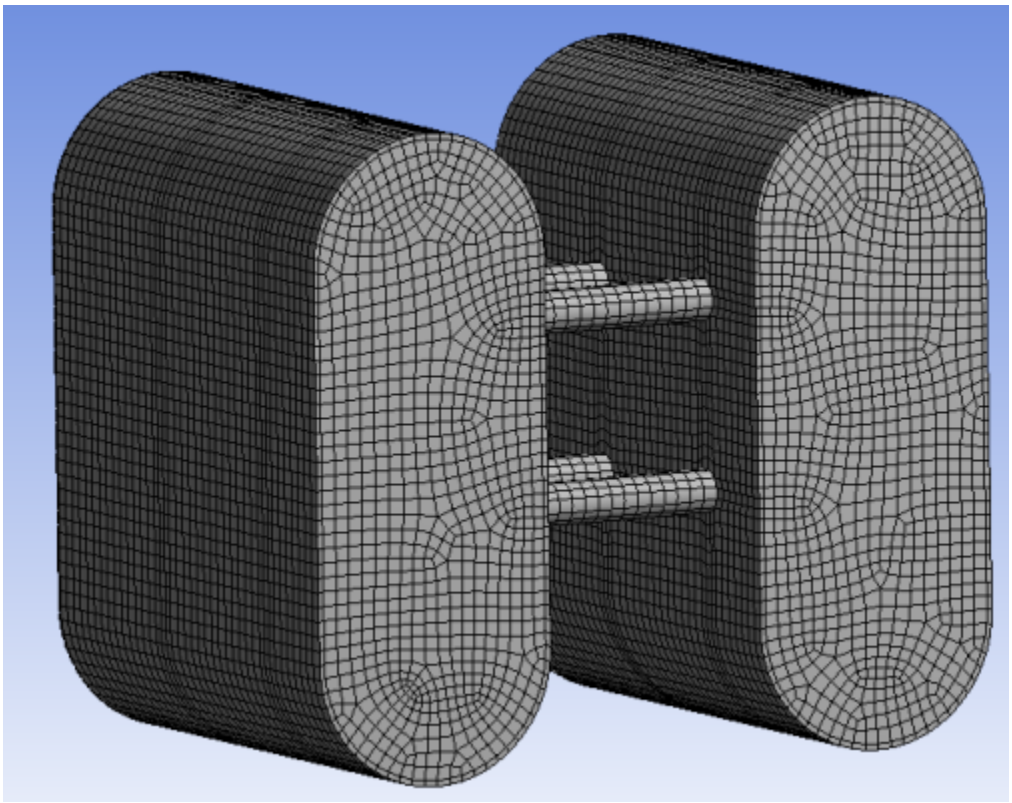
For more information about virtual split edges, see Creating and Managing Virtual Split Edges (p. 561).

Internal Loop Side Faces

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When internal loops exist along the side faces of the sweep path, as shown in Figure 169: Internal Loops along Side Faces of the Sweep Path (p. 436) below, the following tips might help:

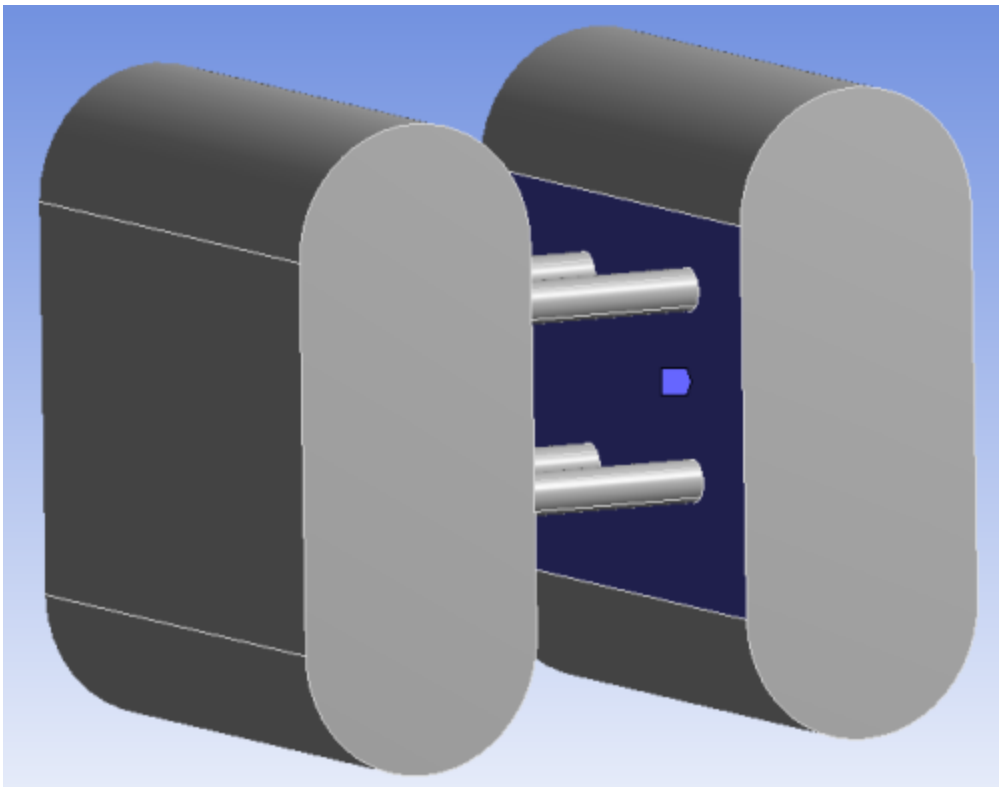
Figure 169: Internal Loops along Side Faces of the Sweep Path



1. Assign a mapped face control to the side face(s) with the internal cutouts to help ensure those faces are mapped.

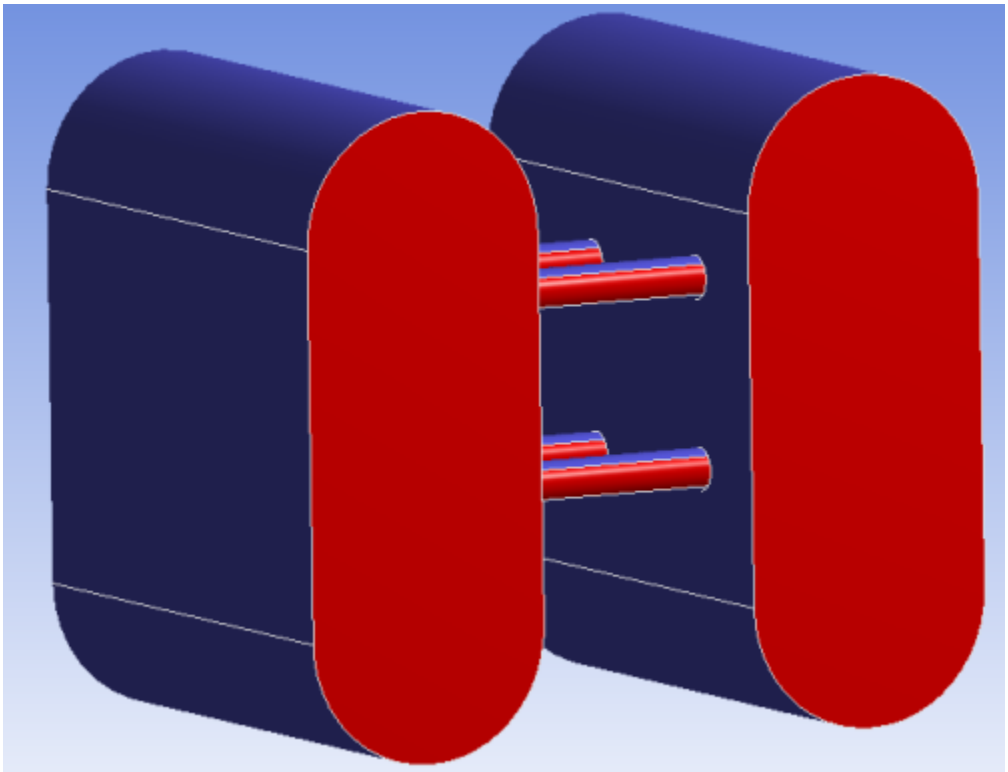
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 170: Map Face Control Assigned to Side Faces



2. Assign front/back of connecting faces as source faces for MultiZone.

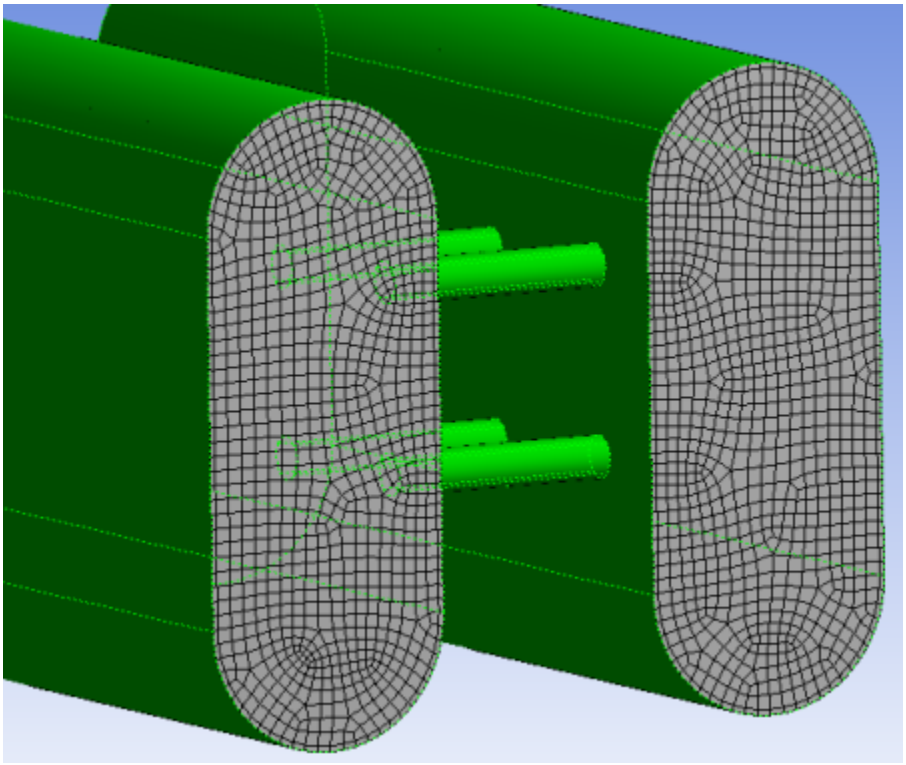
Figure 171: Connecting Faces Assigned as Source Faces



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

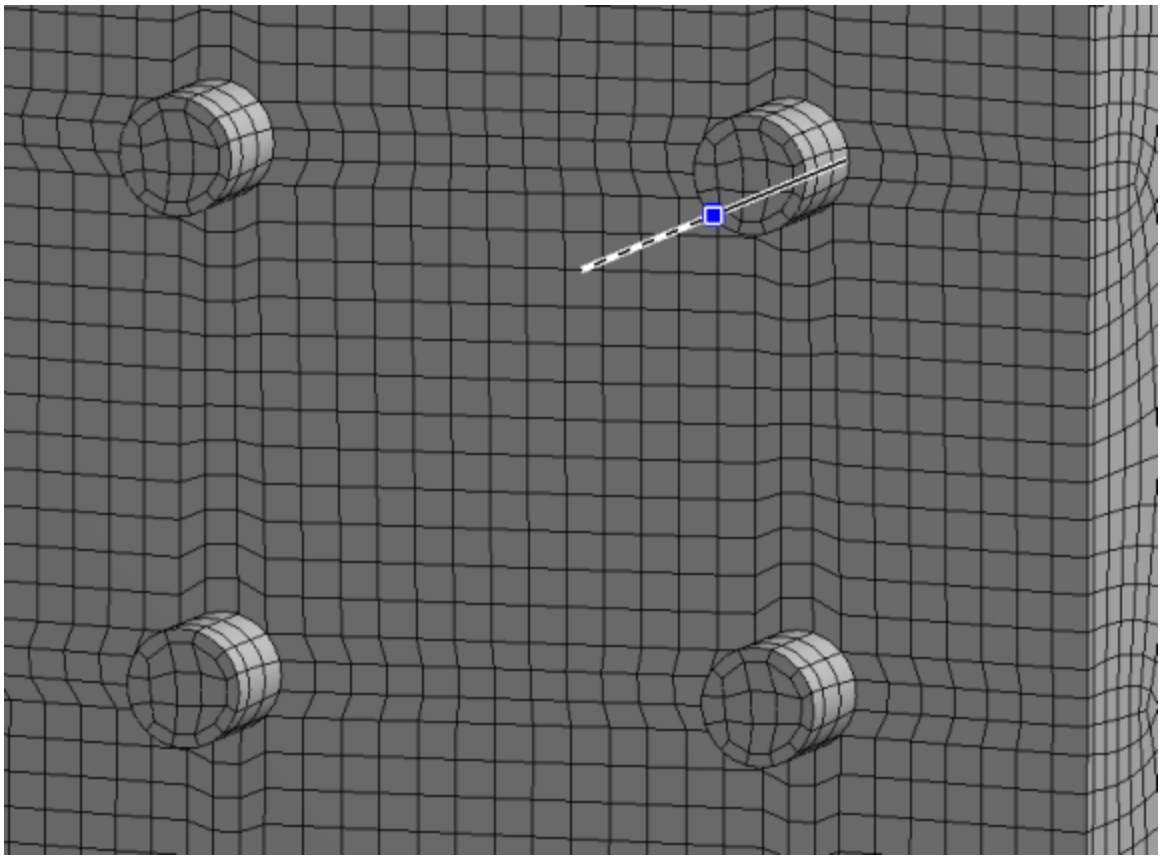
3. If side faces are cylindrical, use inflation to get reasonable quality mesh.

Figure 172: Using Inflation on Cylindrical Side Faces



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

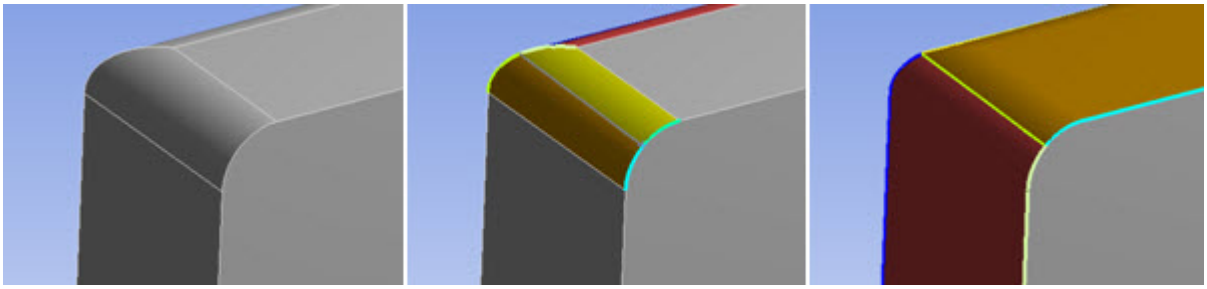
Figure 173: Using Inflation on Cylindrical Side Faces



Using Virtual Topology to Handle Fillets in MultiZone Problems

The presence of fillets in a model can cause problems for **MultiZone**, since arguably a fillet could be either a side face or a source face and the mesh quality in the corner could be problematic. Ideally, a fillet should be split in half, with one half going to the source face and the other half going to the side face.

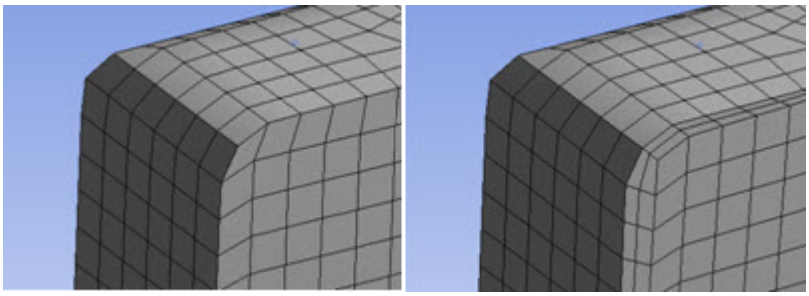
Figure 174: Fillets and MultiZone



In addition, inflation is generally recommended to improve the high angled elements that would be formed along the edge that is shared between the source and side faces. Inflation allows the mesh to transition away from the boundary and reduce the angle. See Figure 175: Fillets and Inflation (p. 440).

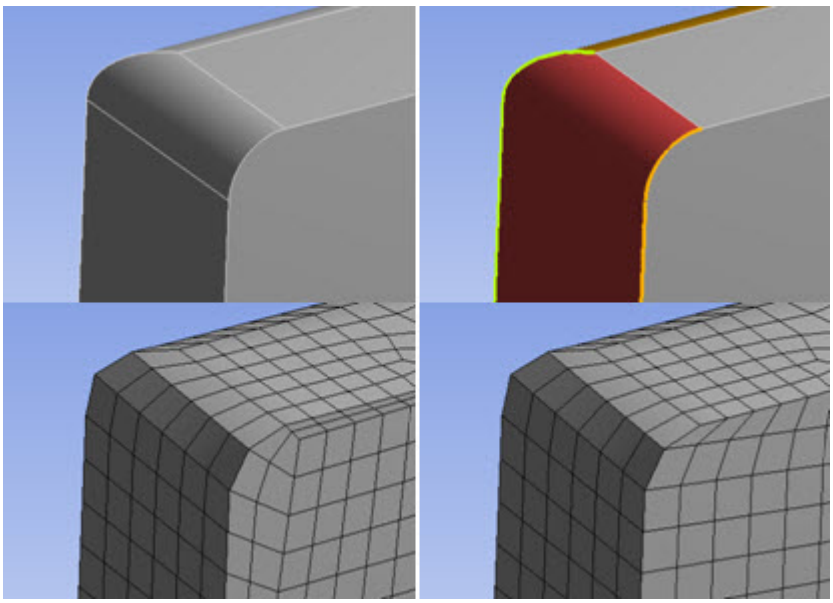
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 175: Fillets and Inflation



Alternatively, the fillet could be removed by assigning the fillet to the side face through a virtual topology operation.

Figure 176: Fillets as Side Faces



Refer to Meshing: Virtual Topology (p. 545) for more information.

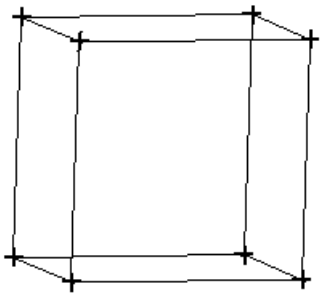
MultiZone Support for Inflation

The Inflation Option (p. 166) setting determines the heights of the inflation layers. For the **MultiZone** mesh method, you can set **Inflation Option** to **Smooth Transition**, **Total Thickness**, or **First Layer Thickness**. **First Aspect Ratio** and **Last Aspect Ratio** are not supported.

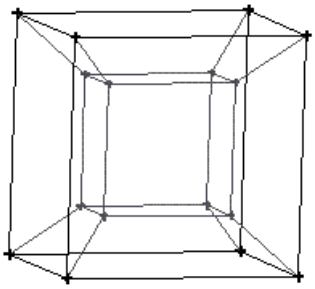
The **Inflation Option** for **MultiZone** is set to **Smooth Transition** by default. The approach **Smooth Transition** uses for computing each local initial height for **MultiZone** differs from the approach used for tet mesh methods. This is because an inflated tet mesh contains different types of volume elements (tets and prisms where the ratio takes into consideration the difference in volume based on element shape), while in an inflated **MultiZone** mesh the elements generally will be the same type.

Inflation in **MultiZone** is done by offsetting the topology to construct inflation regions called an O-Grid. In the O-Grid creation, only the faces that are being inflated will be offset. For example, in a simple box (or sphere) like the following:

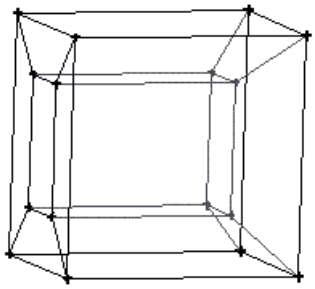
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If all faces are inflated, you would get a complete O-Grid.



If you only select some faces of the box, the faces that are not inflated do not get an O-Grid region, and the inflation layers attach to that face. In the following image, all faces are inflated except the left face.



Inflation controls define:

- How thick the O-Grid region is.
- The sizing along the O-Grid.

When the **Smooth Transition** option is used with **MultiZone**, the O-Grid edge length varies based on the number of elements, and the local last inflation height is computed as Transition Ratio (p. 168) * local_mesh_size. As with other mesh methods when **Smooth Transition** is used, the inflation layers are created using the values of the Transition Ratio (p. 168), Maximum Layers (p. 169), and Growth Rate (p. 169) controls.

Note:

- Because the **MultiZone** mesh method uses an O-Grid technique for inflation, it cannot stairstep or peel back layers if there is insufficient room for all layers. Because of this, the default values for **Smooth Transition** inflation could be aggressive, depending on

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the model. Reducing the number of layers or switching to a different type of inflation definition may be more suitable for some models.

- **MultiZone** supports Program Controlled (p. 164) inflation.
- The Collision Avoidance (p. 174) option is not used for **MultiZone** as the inflation layers are created within the blocking approach.

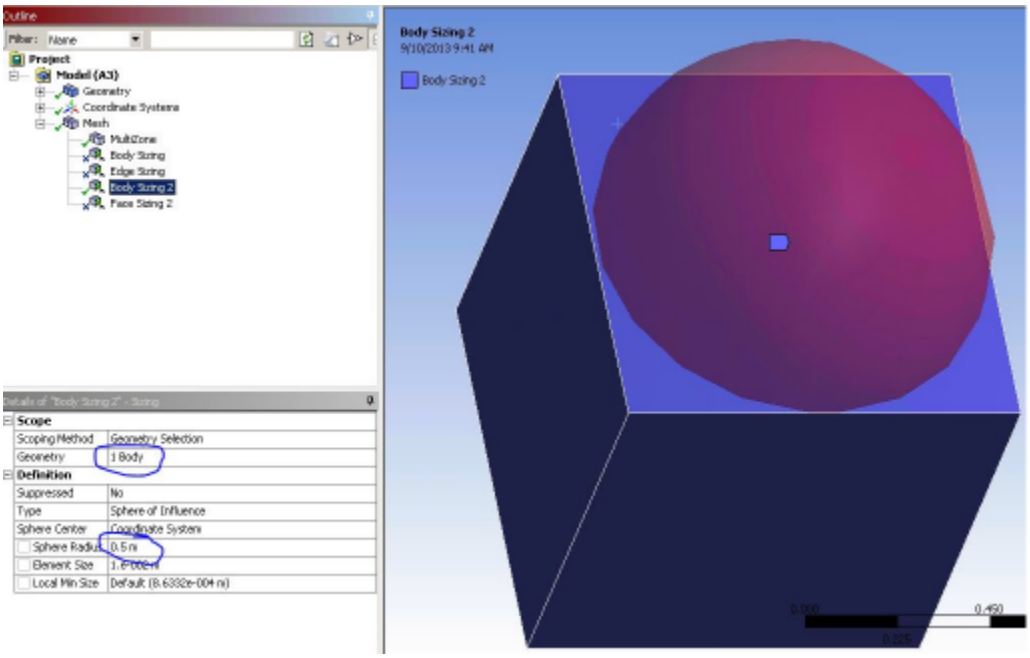
MultiZone Limitations and Hints

MultiZone has the following limitations:

- Pinch controls (p. 187) are not supported.
- Only three-dimensional bodies can be used as Bodies of Influence (BOIs) (p. 305).
- The hard-sizing behavior of Sphere of influence (SOI) (p. 304) controls is not supported. You can define a refinement with an SOI control but not a coarsening. The SOI element size is only applicable in the mesh if it is smaller than the local mesh size without SOI.
- BOI controls are interpreted as an agglomerate of SOI controls that fill the body of influence. Therefore, on the boundaries of that body, the size might be a little larger because the location is not in one of the spheres.
- The scope is not used for the SOI/BOI definition. The SOI/BOI does not act only on the scoped topology, but on all topologies within the region of the SOI/BOI.
- In **MultiZone**, size controls (p. 304) are first applied on the edges. For structured blocks, the mapped mesh of structured blocks is completely determined by the mesh on the attached edges. You cannot apply refinement or coarsening to the faces as you can for unstructured blocks. Therefore, to see the impact of the SOI on a mapped face, you must increase the radius of the SOI or lower the Growth Rate (p. 110) to influence the mesh size on the edges.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 177: Sphere of Influence on Face that Doesn't Intersect Edges



- In **MultiZone**, a closed edge is represented by more than one blocking topological edges. Therefore, user-specified edge node distribution size may not be respected. However, the number of intervals will still be respected,
- For split curves in rotated edge association, only interval count, not distribution, will be respected.
- Some inflation cases are not supported. For example, when the inflation does not form a closed loop (which would lead to non-conformal mesh that is not allowed). In these cases the inflation will be skipped. Also see MultiZone Support for Inflation (p. 440).

Tips:

Some tips to follow while performing **MultiZone** meshing:

- The Show Sweepable Bodies (p. 537) feature is a good tool to detect bodies that should mesh with MultiZone.
- There is no access to underlying blocks except by writing out the Ansys ICEM CFD files (p. 84).
- For help in diagnosing problems when using the **MultiZone** method, refer to the description of the Edge group in the Mechanical help. This toolbar provides access to features that are intended to improve your ability to distinguish edge and mesh connectivity.
- See Handling General MultiZone Meshing Failures (p. 593) for more information.

For additional information, refer to Interactions Between Mesh Methods and Mesh Controls (p. 478) (Patch Independent Mesh Methods table).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selective Meshing

Using selective meshing, you can selectively pick bodies and mesh them incrementally. After meshing a body, you can mesh the entire part or assembly or continue meshing individual bodies. To generate the rest of the mesh in the model, use the **Generate Mesh** (p. 528) feature.

The following mesh methods are supported:

- For solid meshing:
  - Patch Conforming Tetrahedron (p. 208)
  - Patch Independent Tetrahedron (p. 210)
  - MultiZone (p. 242)
  - Sweep (p. 235)
  - Hex Dominant (p. 233)
- For surface meshing:
  - Quad Dominant (p. 282)
  - All Triangles (p. 284)
  - MultiZone Quad/Tri (p. 284)

Remember the following information when using selective meshing:

- Selective meshing is enabled by default. You can use the **Allow Selective Meshing** (p. 389) option to disable it. See Disabling Selective Meshing (p. 448) for information about Meshing application behaviors when selective meshing is disabled.
- Selective meshing is not persistent for a geometry update or re-mesh operation. However, you can use the **Mesh** worksheet to create a selective meshing history so that your meshing steps can be repeated in the desired sequence. Otherwise, you must manually re-mesh bodies in the desired sequence. Refer to Using the Mesh Worksheet to Create a Selective Meshing History (p. 448) for details.
- When using the **Preview Surface Mesh** (p. 531), **Preview Source and Target Mesh** (p. 533), or **Preview Inflation** (p. 534) feature during selective meshing, the previewed mesh will be discarded when you perform a subsequent preview or full mesh operation. The previewed mesh will not be used to seed the subsequent mesh operation.
- After meshing, the meshed status icon (p. 579) appears in the Tree Outline for a meshed body within the **Geometry** folder, or for a multibody part whose child bodies are all meshed. If you make changes after meshing that invalidate (p. 579) the mesh for an individual body (such as adding sizing to the body), you will need to re-mesh that body only.
- In a multibody part, if any child bodies have been meshed and refined, another child body is unmeshed, and you subsequently mesh the unmeshed body, the mesh state of all refined bodies in the part will be invalidated and re-meshed during mesh generation. Similarly, if one body is unmeshed and refinement is needed on another, generating the mesh will result in meshing and refinement of the entire

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

part. In addition to cases involving refinement, this behavior applies in cases where post inflation is used.

- When meshing a body that is part of a symmetry object, match control (p. 334), or pinch control (p. 187), all bodies to which the control is applied need to be meshed at the same time. Also, if a body that is part of a symmetry object, mesh connection object, contact match object, match control, or pinch control fails to mesh, the body will have an invalid mesh state (p. 579) that will propagate to all other bodies that are part of the respective object/control.

- Mesh connections (p. 485), contact matches (p. 496), post inflation (p. 170), and refinement (p. 313) are not performed until all body meshes have been generated. These operations will either be performed when the last body is meshed through selective meshing, or in the last step of the **Generate Mesh** (p. 528) operation.

- Selective meshing is boundary constrained. That is, if you add a size control to a face that is adjacent to an up-to-date body, the edges of that face will be recovered from the existing mesh. Due to the boundary constraints, the mesher cannot split the edges to aid in meshing and will fail if it attempts to do so.

- When you mix mesh methods in multibody parts, the manner in which topology shared by multiple bodies is protected depends on whether adjacent bodies are being meshed with Patch Independent methods (Patch Independent Tetrahedron (p. 210), MultiZone (p. 242), or MultiZone Quad/Tri (p. 284)) and/or Patch Conforming methods (Patch Conforming Tetrahedron (p. 208), Sweep (p. 235) [general or thin], or Hex Dominant (p. 233)):

  – The interface between a Patch Conforming method and a Patch Conforming method is not protected.

  – The interface between a Patch Conforming method and a Patch Independent method is completely protected.

  – Only the boundary is protected at the interface between a Patch Independent method and a Patch Independent method.

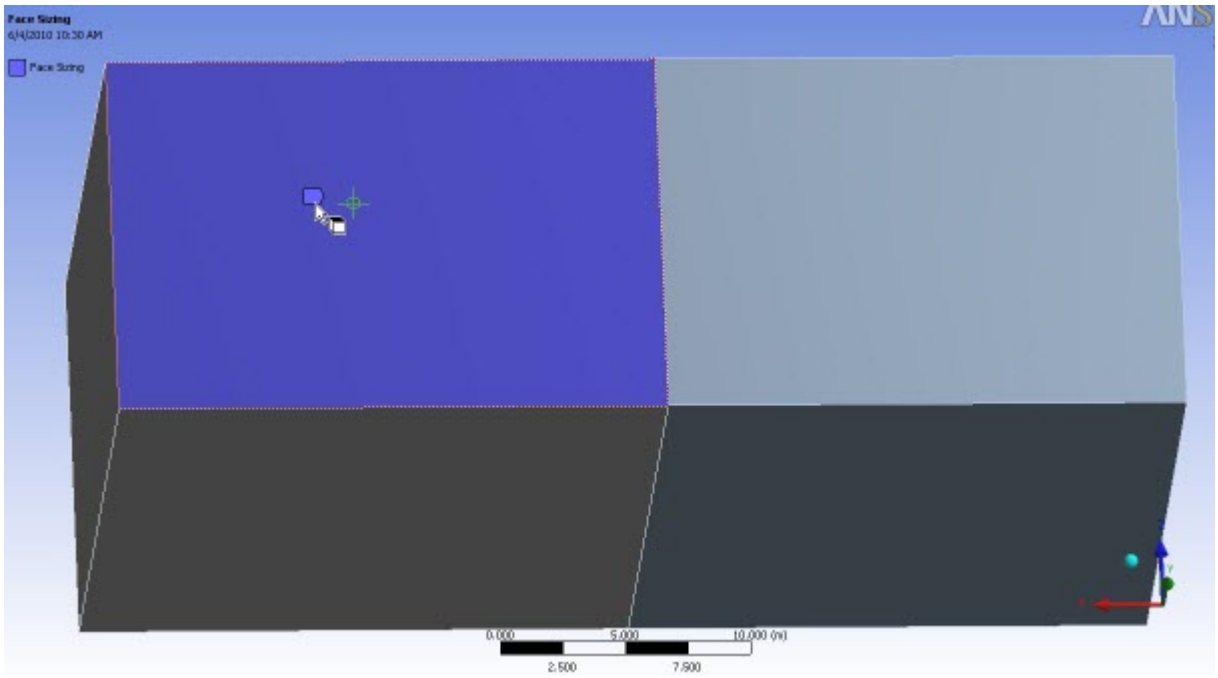
- You can use the **Verbose Messages from Meshing** (p. 389) option to control the verbosity of messages returned to you. Depending on the setting, before meshing a message reports the subset of bodies that is going to be meshed and/or after meshing a message reports the subset of bodies that failed to mesh.

- Size controls on neighboring bodies are not considered, if you are performing selective meshing. This limitation is applicable to all mesh methods that support selective meshing. However, its impact may differ depending on the methods being used.

  For example, consider the simple model below, which consists of two boxes to which the Patch Independent Tetra mesh method has been applied. A local size control that defines a much smaller **Element Size** than the global size has been scoped to the top face of the box on the left.

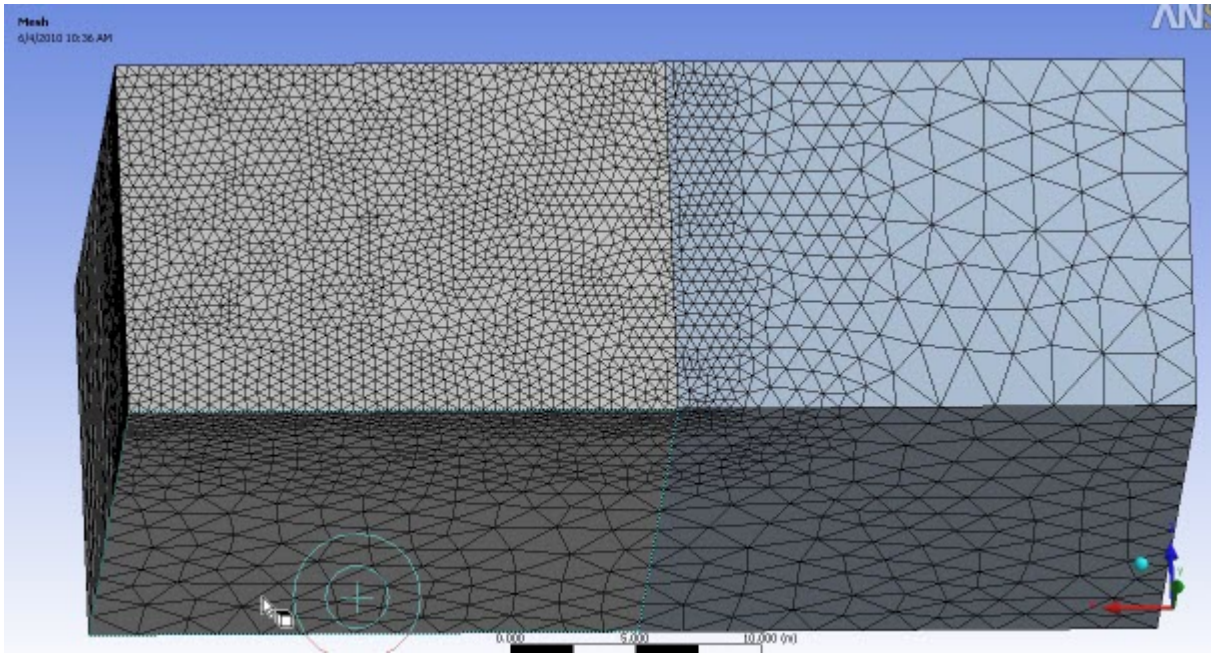
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 178: Two Boxes with Sizing on One Face



When the mesh is generated in one step (for the entire part rather than body by body), there is a smooth transition from the fine element size to the coarse element size, as shown in Figure 179: Mesh Generated for Entire Part (p. 446).

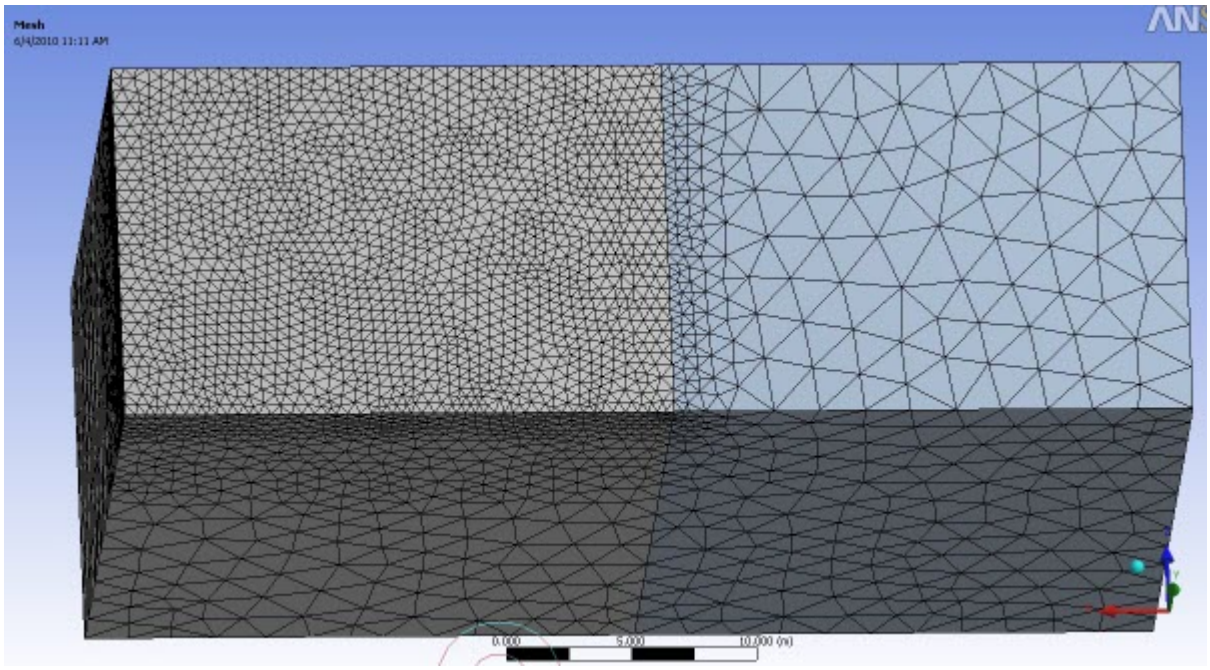
Figure 179: Mesh Generated for Entire Part



However, the mesh will differ if selective (body by body) meshing is performed. Figure 180: Selective Meshing: Left Body First (p. 447) shows the mesh when the body on the left is meshed first, and the body on the right is meshed second. In this case although the results are different than those in the figure above, the mesh may still be acceptable because the impact of the local size control on the left body has influenced the boundary mesh of the right body.

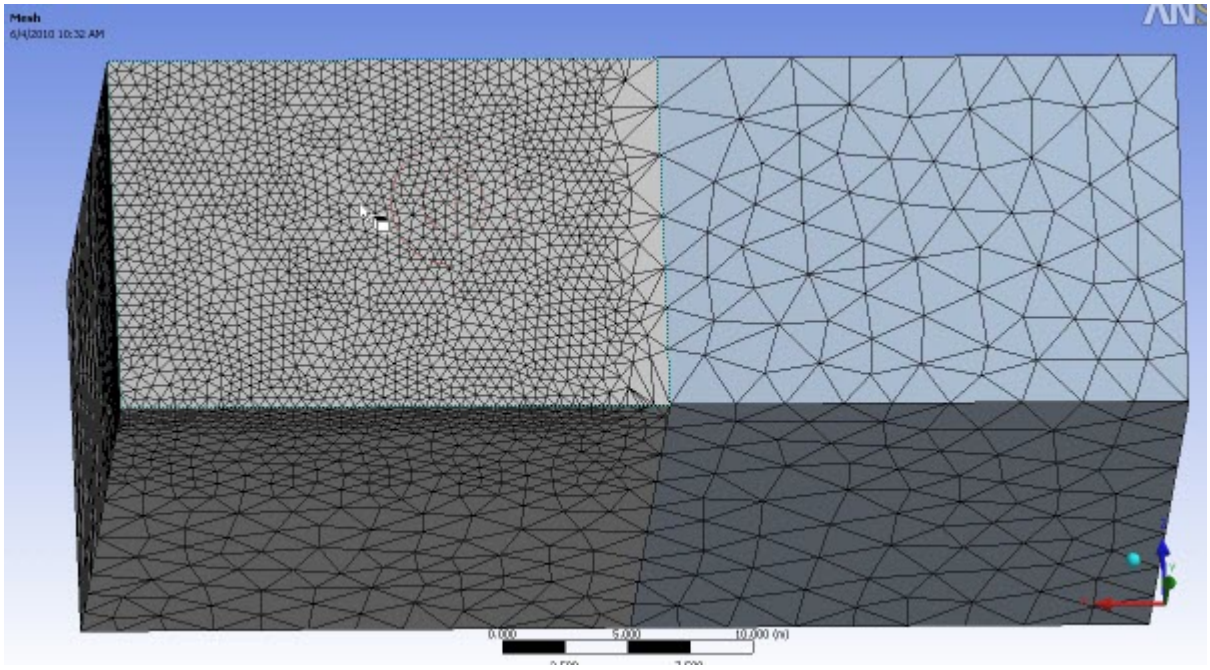
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 180: Selective Meshing: Left Body First



In Figure 181: Selective Meshing: Right Body First (p. 447), the body on the right was meshed first, and the body on the left was meshed second. When this meshing sequence is used, the mesh on the right body does not recognize the size control that is scoped to the body on the left. This results in a coarse mesh on the right body with the transition region occurring on the left body.

Figure 181: Selective Meshing: Right Body First



- When **Mesh Based Connection** is set to **Yes**,**Selective Meshing** is not supported.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Disabling Selective Meshing

Set the **Allow Selective Meshing** (p. 389) option to **No** to disable selective meshing. The Meshing application behaves as follows when selective meshing is disabled:

- If you make changes after meshing that invalidate (p. 579) the mesh for an individual body in a multibody part (such as adding sizing to the body), the mesh for all bodies in the part is invalidated and you will need to re-mesh all bodies.
- If one body in a multibody part is suppressed and you mesh all the other bodies, unsuppressing the suppressed body invalidates (p. 579) the mesh for all the bodies within that part. When you regenerate the mesh, all the bodies within that part will be re-meshed. If the model contains multiple parts, bodies in the other parts will not be affected and will not be re-meshed.
- The **Generate Mesh** (p. 528), **Preview Surface Mesh** (p. 531), and **Clear Generated Data** (p. 541) RMB menu options are unavailable for individual bodies in multibody parts in the Tree Outline. To use these features for a multibody part, you must right-click at the part level in the Tree Outline.
- There is no **Parts>** flyout menu for the **Generate Mesh** (p. 528), **Preview Surface Mesh** (p. 531), and **Clear Generated Data** (p. 541) RMB menu options in the **Geometry** window. If you select one of these options, the action is performed on the entire part.

Note:

In some cases, the mesh may be generated in a selective fashion (body by body) even if selective meshing is currently disabled. For example, if you use selective meshing to mesh some of the bodies in a part, then disable selective meshing, and then generate the mesh *and the mesh process does not invalidate any bodies*, the mesh is generated using selective meshing processes. To avoid this behavior, you can use the **Clear Generated Data** (p. 541) feature or force a change of the mesh state on the part. Non-selective meshing will be used for all subsequent meshing.

Using the Mesh Worksheet to Create a Selective Meshing History

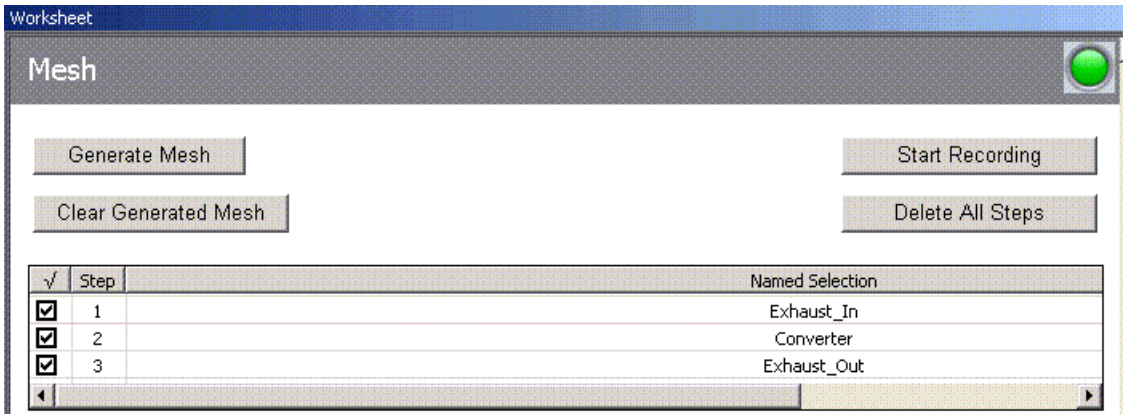
When you perform selective meshing, you control the sequence in which bodies are meshed. You can use the **Mesh** worksheet to create a selective meshing history, so that your meshing steps can be repeated in the desired sequence for any geometry update or re-mesh operation. Figure 182: Mesh Worksheet (p. 449) shows a sample **Mesh** worksheet. Each row in the worksheet corresponds to a step in the meshing sequence. When you generate the mesh, the software processes the steps one by one. For each step, the software selects the bodies identified by the specified Named Selection and meshes those bodies using the meshing controls that are applied to them.

Note:

The **Mesh** worksheet is a helpful meshing tool, but it does not manage mesh state. State management is handled at the prototype (Body object) and mesh levels. Mesh state is visible in the Tree Outline in the **Geometry** folder and in the **Mesh** folder. See Understanding States (p. 583) for more information.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 182: Mesh Worksheet



There are two ways to create a selective meshing history:

- By recording meshing steps as you perform them
- By adding meshing steps to the worksheet manually

To create a selective meshing history by recording your steps:

1. Click the **Mesh** object in the Tree Outline.
2. Do one of the following:

   Right-click the **Mesh** object and select **Start Recording** from the context menu. As a result, the worksheet opens in recording mode automatically.

   -or-

   Click the **Worksheet** button on the toolbar and click the **Start Recording** button on the worksheet.

3. Move the worksheet to the desired location to dock it. The location will persist whenever the **Mesh** object or one of its child objects is highlighted in the Tree Outline. For example, you may want to dock the worksheet alongside the **Geometry** window, allowing you to view both at once.

   Note:

   The worksheet is not dockable on Linux platforms. On Linux, you can drag the worksheet off the Meshing application interface, and it will then appear in its own separate window.

4. Do one of the following:

   Select one or more bodies or parts in the **Geometry** window, right-click, and select **Generate Mesh on Selected Bodies**. If you did not dock the worksheet, you may need to click the **Worksheet** button on the toolbar or the **Graphics** tab at the bottom of the worksheet to return to the **Geometry** window first.

   -or-

   Select one or more bodies or parts in the Tree Outline, right-click, and select **Generate Mesh**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

As a result, a Named Selection for the selected set is generated (named **Meshing_1**, **Meshing_2**, **Meshing_3**, and so on) and is added to the worksheet automatically.

During recording, other steps in the worksheet are ignored while the mesh for the selected set is being generated. See below for additional notes on recording and playback behaviors.

5. Repeat step 4 for each meshing step in the desired sequence.
6. When you are done meshing bodies/parts, do one of the following:

   Right-click the **Mesh** object and select **Stop Recording** from the context menu.

   -or-

   Click the **Stop Recording** button on the worksheet.

Note:

You do not have to record the meshing for all bodies, only those for which you want to record the meshing order. See Mesh Worksheet Recording and Playback Behaviors (p. 450) for details.

To create a selective meshing history manually:

1. Click the **Mesh** object in the Tree Outline.
2. Click the **Worksheet** button on the toolbar.
3. Move the worksheet to the desired location to dock it. The location will persist whenever the **Mesh** object or one of its child objects is highlighted in the Tree Outline. For example, you may want to dock the worksheet alongside the **Geometry** window, allowing you to view both at once.

Note:

The worksheet is not dockable on Linux platforms. On Linux, you can drag the worksheet off the Meshing application interface, and it will then appear in its own separate window.

4. Add a row to the worksheet by right-clicking on the table and selecting **Add** from the context menu.
5. In the new row, click the **Named Selection** column and select a Named Selection from the **Named Selection** drop-down.
6. Repeat steps 4 and 5 for each meshing step in the desired sequence.

Mesh Worksheet Recording and Playback Behaviors

Remember the following information regarding recording and playback behaviors:

- During recording, the button in the upper-right corner of the worksheet is red. When recording is stopped, the button is green.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When you start recording, the software checks for meshed bodies and verifies that each meshed body is accounted for in the worksheet. If any extraneous meshed bodies are found, recording does not occur and an error message is issued. You must clear the mesh entirely (or at least clear the mesh from the bodies that are not being used by the worksheet) before you can begin recording. To do so, use the **Clear Generated Data** (p. 541) option or click the **Clear Generated Mesh** button on the worksheet.
- When you start recording, the states of the bodies being used by the worksheet are checked. If any body is not in a meshed state, recording does not occur and an error message is issued. You must bring the mesh up-to-date before you can begin recording. To do so, right-click the last step in the worksheet and select **Generate Mesh Through This Step** from the context menu. See below for more information about using this option.
- As steps are being recorded, they are appended to the existing steps in the worksheet.
- Other steps in the worksheet are ignored while the mesh for the selected set is being generated.
- After you have entered recording mode, recording stops when the mesh is up-to-date. An exception to this behavior occurs if you record one or more steps but then suppress all remaining unmeshed bodies. In such cases, the mesh will be up-to-date but recording will not stop automatically. You must click **Stop Recording**.
- During recording/playback, post mesh operations (including mesh connections (p. 485), contact matches (p. 496), post inflation (p. 170), and refinement (p. 313)) do not occur until the last step in the worksheet, after all meshing is complete.
- If you select the Preview Surface Mesh (p. 531), Preview Source and Target Mesh (p. 533), or Preview Inflation (p. 534) feature, recording stops and a warning message is issued.
- The Generate Mesh (p. 528) option is context sensitive. It behaves differently depending on where you invoke it. If you invoke it from the **Mesh** object in the Tree Outline (RMB on **Mesh** folder > **Generate Mesh**), it behaves similar to the **Preview** options in that recording stops. If you invoke it from the **Geometry** object (RMB on **Geometry** folder > **Generate Mesh**) or from the **Geometry** window (RMB > **Generate Mesh On Selected Bodies**), the operation is treated as a selective meshing step and is recorded.
- If you insert a step manually, recording stops and a warning message is issued.
- If you delete a step manually, recording stops and a warning message is issued.
- If the mesh fails, recording stops and a warning message is issued.
- To replay steps incrementally, right-click in the row of interest and select **Generate Mesh Through This Step** from the context menu. As a result, recording stops, any existing mesh is cleared, and all meshing steps prior to and including the selected step are replayed in the **Geometry** window.

Mesh Worksheet Named Selection Behaviors

Remember the following information regarding Named Selection behaviors:

- Only Named Selections that consist of bodies are selectable from the worksheet's **Named Selection** drop-down.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If the model contains bodies that are not included in any Named Selection, these bodies are meshed last.
- If a Named Selection is used by the worksheet, the **Used by Mesh Worksheet** field in the Details View for that Named Selection is set to **Yes**, even if the corresponding worksheet step is inactive.
- For Named Selections that are generated automatically by the worksheet, the **Send to Solver** (p. 78) option in the Details View is set to **No** by default. The default is **Yes** for Named Selections that you create. The worksheet respects the user-defined settings. That is, if you create a Named Selection, retain the **Send to Solver** setting of **Yes**, and subsequently use that Named Selection in the worksheet, the Named Selection will be passed to the solver as expected.
- If a change to a Named Selection being used by the worksheet causes the Named Selection to become underdefined or invalid, the corresponding worksheet step will be deactivated without invalidating the worksheet. Reasons a Named Selection may become underdefined or invalid include:
  - You change the scope of a Named Selection from body to face/edge
  - A geometry update changes the scope of a Named Selection, or causes the software to try to delete the Named Selection
- If you delete a named selection manually, recording stops and a warning message is issued.
- From the Tree Outline, you can suppress/unsuppress Named Selections or bodies included in Named Selections that are being used by the worksheet, and the corresponding worksheet step will be de-activated/activated accordingly without invalidating the worksheet.

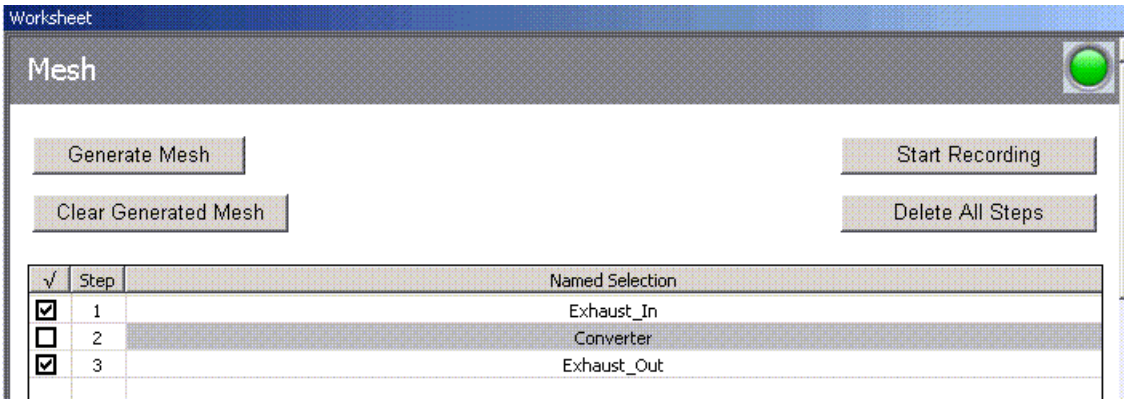
  You also can activate/deactivate steps directly on the worksheet. By default, a check mark ( ) appears on the worksheet, meaning steps that correspond to all unsuppressed bodies/Named Selections are active. To deactivate a single step, clear the corresponding check box. To deactivate all active steps, click and it is replaced by . Toggling between and activates/deactivates the steps corresponding to all unsuppressed bodies/Named Selections. Toggling step activation on the worksheet has no effect on the suppressed status of the corresponding bodies/Named Selections in the Tree Outline.

  When a step is inactive, its row in the worksheet is grayed out. Inactive steps are skipped during mesh generation and other worksheet operations.

  To obtain the example shown in Figure 183: Mesh Worksheet Step Deactivation (p. 453), the Converter Named Selection was suppressed in the Tree Outline.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 183: Mesh Worksheet Step Deactivation



Miscellaneous Points to Remember

- To delete a row from the worksheet, right-click in the row of interest and select **Delete** from the context menu. The worksheet, including the meshing sequence, will be updated automatically.
- Click the **Delete All Steps** button to clear all data from the worksheet.
- If the mesh fails at any point in the process, the process terminates but returns as much of the mesh as possible.
- If you change the worksheet after you mesh, those changes will not be reflected in your meshing state as the worksheet does not affect meshing state. The changes will take effect the next time you generate a mesh.
- If you select bodies to mesh individually (by using the Generate Mesh On Selected Bodies (p. 528) option in the **Geometry** window, the Generate Mesh (p. 528) option in the **Geometry** folder, or by using the Preview Surface Mesh (p. 531), Preview Source and Target Mesh (p. 533), or Preview Inflation (p. 534) feature), the Meshing application ignores the worksheet and generates the mesh for the selected bodies.
- If you select **Part** or **Body** objects in the **Geometry** folder in the Tree Outline, right-click, and then select Generate Mesh (p. 528) from the context menu, the Meshing application ignores the worksheet and generates the mesh for the selected parts/bodies.
- When you resume a saved project from an older release, you should generate worksheet based named selections manually after meshing to update the named selections.
- When generating a mesh from the **Mesh Worksheet**, if the last step fails, mesher attempts to remesh the entire part, including bodies not listed in the named selection.

Inflation Controls

Inflation is useful for CFD boundary layer resolution, electromagnetic air gap resolution or resolving high stress concentrations for structures.

The following sections provide the high-level steps to follow to assign inflation depending on the selected mesh method.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For information on setting global inflation controls and descriptions of all of the individual inflation controls, refer to Inflation Group (p. 161). Alternatively, you can use local inflation mesh controls to apply inflation to specific boundaries. For details, refer to Inflation Control (p. 345). For general information on applying inflation controls in combination with the various mesh method controls, refer to Meshing: Mesh Control Interaction Tables (p. 475).

Inflation Controls With Sweeper

Inflation is a pre process for the sweeper (p. 235). The source face is meshed and then inflated before sweeping with the sweeper.

Note:

- Inflation is supported only on the source face(s) of the sweep (that is, inflation on the side faces). Inflation away from the source face(s) is not supported.
- You do not have to select a source face for sweeping with inflation. You can simply pick faces for inflation and the Meshing application will internally place a Sweep method on the adjacent bodies using the inflated faces as the sources (unless another method already exists).

To add boundary layers to a source face for sweeping:

1. Apply a **Method** control to a body.
2. Set **Method** to **Sweep**.
3. Set **Src/Trg Selection** to **Manual Source** or to **Manual Source and Target**.
4. Scope the source (and target if **Manual Source and Target** was selected).
5. Set **Free Face Mesh Type** to **All Quad**, **All Tri**, or **Quad/Tri**. Your selection determines the shape of the elements used to fill the swept body (pure hex, pure wedge, or a combination of hex/wedge respectively). The boundary region of the source/target faces will always be meshed with quad layers. Refer to Figure 184: Sweep Method With Inflation: Hex Fill (p. 455) and Figure 185: Sweep Method With Inflation: Wedge Fill (p. 456).
6. Enter additional sweep settings, as desired, in the Details View.
7. Select the source face and insert an **Inflation** control.
8. Select the outer boundary edges of the source face for inflation (p. 346) (the edges that you want inflation to grow away from).
9. Enter additional inflation settings, as desired, in the Details View.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

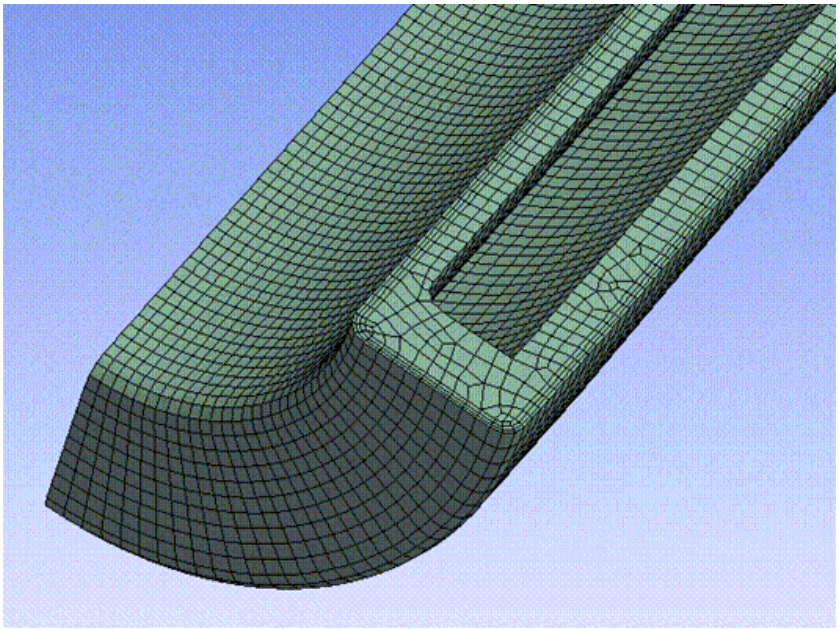
10. Mesh the body.

Note:

If the target face has a different number of edges than the source face, the bias of the boundary layer may not be transferred correctly.

To obtain the mesh shown in Figure 184: Sweep Method With Inflation: Hex Fill (p. 455), **Free Face Mesh Type** was set to **All Quad**. Notice the boundary region is meshed with quad layers.

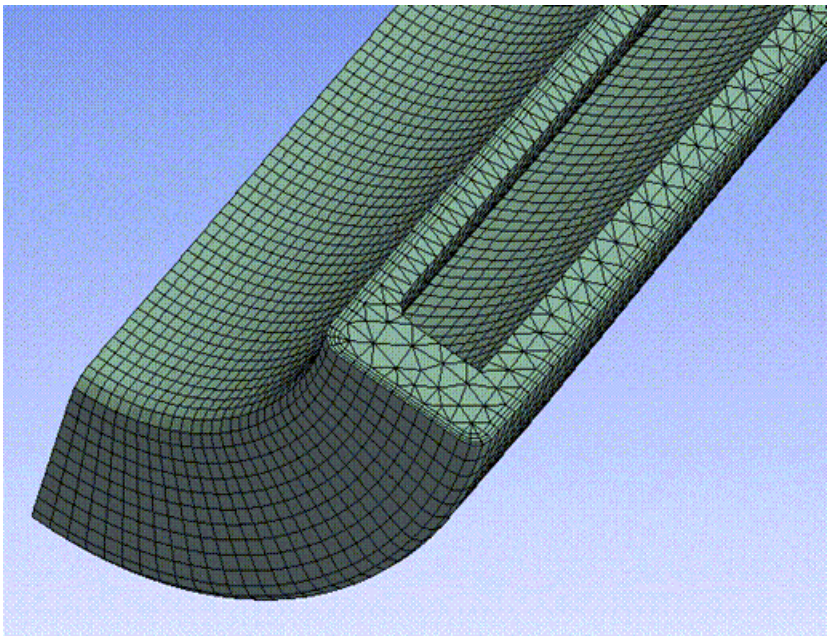
Figure 184: Sweep Method With Inflation: Hex Fill



To obtain the mesh shown in Figure 185: Sweep Method With Inflation: Wedge Fill (p. 456), **Free Face Mesh Type** was set to **All Tri**. Notice the boundary region is meshed with quad layers.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 185: Sweep Method With Inflation: Wedge Fill



Inflation Controls With Patch Conforming Mesher

Inflation can be either a pre process or a post process for the patch conforming (p. 208) mesher.

To add boundary layers to a face using the Patch Conforming Mesher:

1. Apply a **Method** control to a body.
2. Set **Method** to **Tetrahedrons**.
3. Set the tetrahedrons **Algorithm** to **Patch Conforming**.
4. Select the body and insert an **Inflation** control.
5. Select the faces to be inflated (p. 346) (the faces that you want the inflation layers to grow away from).
6. Enter additional settings, as desired, in the Details View.
7. Mesh the body.

Inflation Controls With Patch Independent Mesher

Inflation is a post process for the patch independent (p. 210) mesher after it has created the tetrahedron elements.

To add boundary layers to a face using the Patch Independent Mesher:

1. Apply a **Method** control to a body.
2. Set **Method** to **Tetrahedrons**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Set the tetrahedrons **Algorithm** to **Patch Independent**.
4. Set the **Min Size Limit**.
5. Select the body and insert an **Inflation** control.
6. Select the faces to be inflated (p. 346) (the faces that you want the inflation layers to grow away from).
7. Enter additional settings, as desired, in the Details View.
8. Mesh the body.

Inflation Controls With MultiZone

To add boundary layers to a face using the MultiZone (p. 242) Mesher:

1. Apply a **Method** control to a body.
2. Set **Method** to **MultiZone**.
3. Select the body and insert an **Inflation** control.
4. Select the faces to be inflated (p. 346) (the faces that you want the inflation layers to grow away from).
5. Enter additional settings, as desired, in the Details View.
6. Mesh the body.

For more information, see MultiZone Support for Inflation (p. 440).

Inflation Controls With MultiZone Quad/Tri Mesher

To add boundary layers to a face using the MultiZone Quad/Tri (p. 284) Mesher:

1. Apply a **Method** control to a body.
2. Set **Method** to **MultiZone Quad/Tri**.
3. Select a body or face and insert an **Inflation** control.
4. Select the edges to be inflated (p. 346) (the edges that you want inflation to grow away from).
5. Enter additional settings, as desired, in the Details View.
6. Mesh the body.

Note:

Base mesh caching is not supported for **MultiZone Quad/Tri**, so a change to inflation controls requires remeshing.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inflation Controls With Quadrilateral Dominant or All Triangles Mesher

Inflation is a pre process for the quadrilateral dominant (p. 282) mesher or all triangles (p. 284) mesher.

To add boundary layers to a face using the Quadrilateral Dominant or All Triangles Mesher:

1. Apply a **Method** control to a body.
2. Set **Method** to **Quadrilateral Dominant** or **Triangles**.
3. Select a body or face and insert an **Inflation** control.
4. Select the edges to be inflated (p. 346) (the edges that you want inflation to grow away from).
5. Enter additional settings, as desired, in the Details View.
6. Mesh the body.

Inflation Controls With Cartesian Mesher

Inflation is a **Pre** process only for the Body Fitted Cartesian (p. 261) mesher. For **CFD** physics only, three layers are created with total thickness proportional to **Element Size**; for other physics preferences, only only layer is created.

To add boundary layers to a body using the Cartesian Mesher:

1. Apply a **Method** control to a body.
2. Set **Method** to **Cartesian**.
3. Select the **Method** and add an **Inflation** (p. 345) control.
4. Enter additional settings, as desired, in the Details View.
5. Mesh the body.

Inflation Handling Between Bodies With Different Methods

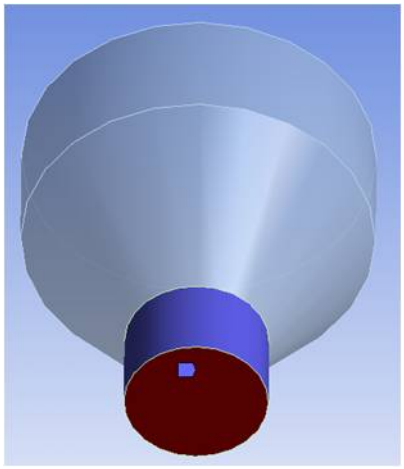
The inflation handling between bodies where one body is meshed with the sweep method and one body is meshed with the patch conforming tetrahedral method requires some special consideration to ensure inflation layers propagate through the common interface. There are two such cases to consider:

- The case in which the common interface of two bodies is also a source/target face of the swept body
- The case in which the common interface of two bodies is also a side face of the swept body

The model below will be used to explain the first case, in which the common interface of two bodies is also a source/target face of the swept body.

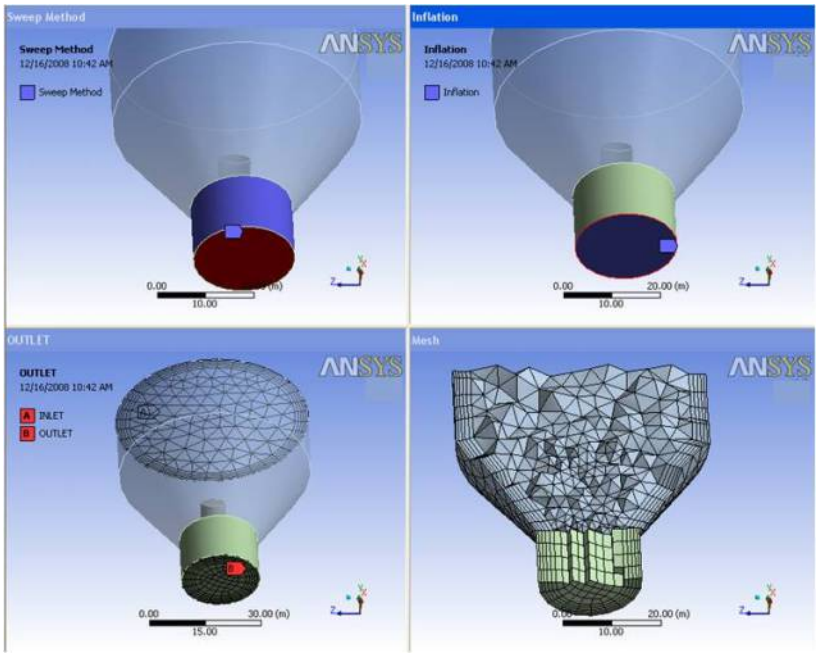
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 186: Swept Body Shares Source/Target Face With Tet Body



In this case, inflation on the patch conforming tetrahedral method must be defined off the faces of the wall (not common interface), or by using Program Controlled (p. 164) inflation (which ignores faces in Named Selections and common interfaces between bodies). The swept body needs the source face to be selected, and 2D inflation must be defined on the source face. Since 2D inflation does not support the Smooth Transition (p. 166) option, it is best to use another option so that the inflation between bodies will properly align.

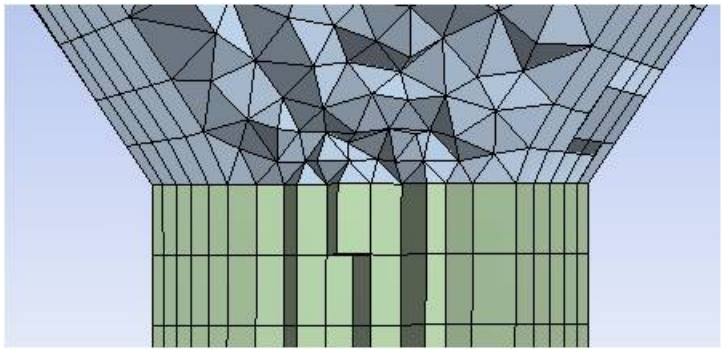
Figure 187: Defining Inflation for a Swept Body Sharing Source/Target Face With Tet Body



After properly setting up the model and ensuring the inflation of the tet body and the swept body have similar near-wall spacings, a mesh can be generated where the inflation layers will pass from one body to the next with proper connections on the common interface, as shown below.

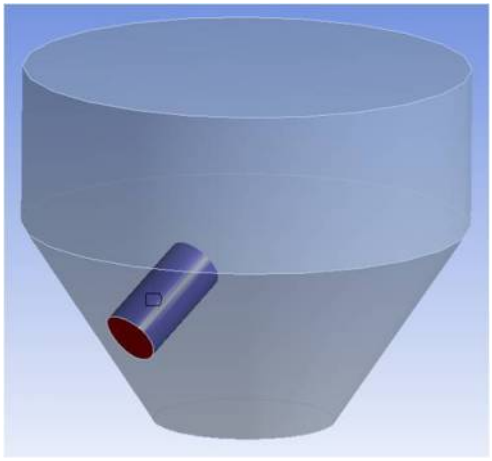
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 188: Detail of Proper Connections on the Common Interface



The model below will be used to explain the second case, in which the common interface of two bodies is also a side face of the swept body.

Figure 189: Tet Body Surrounds Swept Body



In this case, inflation on the patch conforming tetrahedral method must be defined off the faces of the wall (not common interface), or by using Program Controlled (p. 164) inflation (which ignores faces in Named Selections and common interfaces between bodies). To properly align the inflated tet mesh to the side faces of the swept body, a biasing must be used along the sweep direction. Since the biasing along the sweep direction uses a different formulation than the inflation biasing, the following notes may be helpful.

The sweep bias ratio is the ratio of largest edge to smallest edge along sweep path, the growth ratio for inflation is a factor of the growth from the first element to the second element, etc. These relate as described below. The equation to get the inflation growth rate to align to the swept body is:

$$I_r = S_b^{(1/N-1)}$$

where

S_b = Sweep Bias

N = Number of Divisions along sweep

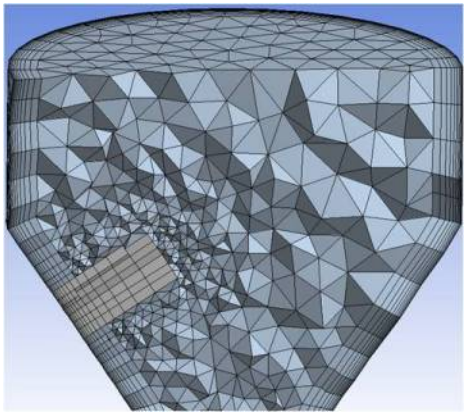
I_r = Inflation Growth Rate

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Also, to get proper alignment between the first layer of inflation and sweep, you need to use the first layer height of the swept mesh as first layer height for sweep.

Because the first layer height is computed by the software from the length of the sweep path, the sweep bias, and the number of divisions along the sweep—and there is no easy way to get the length of the sweep path—you should mesh the swept body first, measure the first layer height, and use this value as input for the First Layer Height (p. 169) option when defining inflation controls. The inflation growth rate can then be calculated using the formula above. This leads to well-aligned layers between the sweep and tet regions, as shown below.

Figure 190: Detail of Well-aligned Layers Between the Swept and Tet Regions



Inflation and Baffles

The Meshing application provides support for meshing 0–thickness walls, or baffles, as non-manifold faces of a solid body. Inflation is supported. See Baffle Meshing (p. 470) for more information.

Mesh Refinement

Mesh refinement is a postprocess in the mesh generation process in which the elements on the selected topology are split. This is useful for local mesh sizing control. See the Refinement Control (p. 313) section for more information.

Mixed Order Meshing

The **Method Controls and Element Order Settings** allow you to specify whether the mesh to be generated for a given body is **Quadratic** (high order) or **Linear** (low order). **Mixed Order Meshing** refers to meshing a multibody part having shared topology with some bodies as **Quadratic** and some bodies as **Linear**. In such situations, "transitional elements" are required to connect the mesh at any linear-to-quadratic interface. These "transitional elements" are treated as quadratic elements with dropped midside nodes, and the side of the interface on which they appear is determined by your meshing process. **Mixed Order Meshing** is not supported for **Axisymmetric Sweep**. See Method Controls and Element Order Settings (p. 202) for more information.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Contact Meshing

Enhancing convergence or quality of results for structural contact analysis may require the use of contact sizing to create similarly sized meshes where faces and bodies are coming in contact. See the Contact Sizing Control (p. 311) section for more information.

For information about using contact meshing for rigid bodies, refer to Rigid Body Meshing (p. 463).

Winding Body Meshing

Winding body meshing creates special element types depending on the attributes given to bodies in the DesignModeler application. No mesh controls are supported for winding bodies because of the nature of the required mesh.

Wire Body Meshing

Wire body meshing meshes the wire bodies in an assembly, respecting any mesh controls applied to the edges of the wire body.

Note:

Wires and beams are both considered to be line bodies and are handled in the same way by the mesher.

Pyramid Transitions

Pyramid transitions occur when a swept or hex dominant body abuts, that is, shares faces with, a body meshed with tetrahedrons. The mesher will try to insert the pyramids into the body meshed with tetrahedrons. If that is not possible, the hexahedron at the boundary will be split into pyramids and tetrahedrons to create a conformal mesh between the two bodies. Pyramids will also be formed at the interface of an end cap of inflation on quad surface mesh and a tet body.

Match Meshing and Symmetry

For parts that are symmetric about a cylindrical axis, you can match the mesh by using either cyclic mesh matching or the **Symmetry** feature in the Mechanical application. The following table describes when to use each method:

Table 2: Mesh Matching for Symmetrical Parts

If you want to...	Do this...
Match the mesh, but you do not want to automatically generate solver constraints for periodic mesh	Apply a **Cyclic Mesh Match** control (p. 336).
Match the mesh, and automatically generate solver constraints for periodic mesh	Define the symmetry in the model by applying the necessary Symmetry regions, Periodic regions, or Cyclic regions.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When **Periodic Region** or **Cyclic Region** objects exist in the **Symmetry** folder, match face mesh controls will be created internal to the mesher. If the mesher cannot match the mesh on the objects in the **Symmetry** folder, it will return a failure or informational message.

Related topics include:

- For a description of the **Symmetry** folder and its support of **Periodic Region** and **Cyclic Region** objects, see Symmetry in the Mechanical help.
- For an overview of the match control and its limitations, see Match Control (p. 334).
- For general information on applying match controls in combination with the various mesh method controls, see Meshing: Mesh Control Interaction Tables (p. 475).

Rigid Body Meshing

Rigid body meshing simplifies the representation of a model by reducing it to the contact regions and the centroid of the model. That is, when a part is defined as a rigid body, the mesher knows to mesh only the contact regions, and to create a single mass element at the centroid of the model. (The centroid is represented by an **Inertial Coordinate System**. Refer to the discussion of Rigid Bodies in the Mechanical help for more information.)

Rigid body meshing can be used for both 2D and 3D contact. For 2D models, only the edges of the rigid surface in contact are meshed because the surface mesh is not needed for the analysis. Similarly, for 3D models, only the faces of the rigid body in contact are meshed because the volume mesh is not needed. The elements used to mesh the contact surfaces can be quadrilateral or triangular in shape, with or without midside nodes.

The following surface mesh methods can be applied to rigid bodies:

- Quadrilateral Dominant (p. 282)
- Triangles (p. 284)

Rigid body contact meshing respects mapped face controls (p. 314) and sizing (p. 296) controls. If 2D inflation (p. 161) is applied, inflation layers are generated for the surfaces in contact.

If a method control is scoped to a rigid body, the **Method** control is set to **Quadrilateral Dominant** by default, but you can change the value to **Triangles**. When **Method** is set to **Quadrilateral Dominant**, an additional option called the **Free Face Mesh Type** control is available for most analyses and can be set to either **Quad/Tri** (default) or **All Quad**.

For Transient Structural, Rigid Dynamics, and Explicit Dynamics analyses, certain default behaviors related to rigid body meshing differ depending on analysis type and solver. Additionally, the **Free Face Mesh Type** option is replaced by the **Rigid Face Mesh Type** option to determine the default element shape

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for rigid body face meshing. The table below provides information you should be aware of when selecting one of these analysis types from the Toolbox and adding it to a Workbench project.

Table 3: Rigid Body Meshing: Default Behaviors for Rigid Dynamics, Transient Structural, and Explicit Dynamics Analyses

Standard Analysis Type	Solver	Element Order (p. 101)	Straight Sided Elements (p. 180)	Rigid Body Behavior (p. 182)	Rigid Face Mesh Type
Rigid Dynamics	Rigid Body Dynamics	-	Not applicable	Full Mesh	Quad/Tri
Transient Structural	Mechanical APDL	Program Controlled	No	Dimensionally Reduced	Quad/Tri
Explicit Dynamics	Autodyn	Linear	Not applicable	Full Mesh	Quad/Tri
LS-DYNA	LS-DYNA	Linear	Not applicable	Full Mesh	Quad/Tri

For information about generating a full mesh on rigid bodies instead of a surface mesh, refer to the description of the Rigid Body Behavior (p. 182) control.

Using 2D Rigid Body Contact Meshing

This section describes the basic steps for using 2D rigid body contact meshing.

To define a 2D rigid body for contact meshing:

1. Open the model in the Mechanical application.
2. In the Tree, expand the **Geometry** object so that the body objects are visible.
3. Click the body that you want to define as the rigid body.
4. In the **Details> Definition** view for the body, change the value of the Stiffness Behavior control to **Rigid**.

   Note:

   When you change the **Stiffness Behavior** to **Rigid**, an **Inertial Coordinate System** object is added to the Tree automatically. This **Inertial Coordinate System** represents the centroid of the body.

5. If desired, change the value of the **Element Order** (p. 101) control.
6. Generate the mesh by right-clicking on the **Mesh** object in the Tree and selecting **Generate Mesh**.

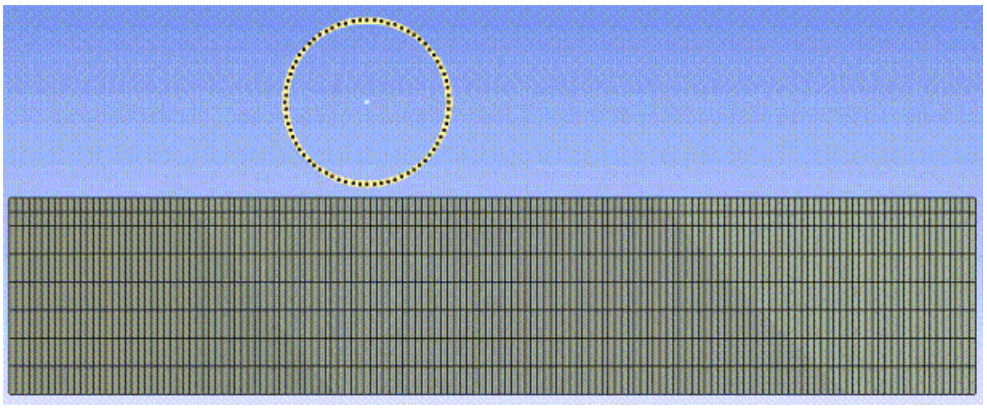
   Note:

   The mesh for the 2D rigid body is created only in the contact region (edges in contact). See the figure below for an example.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the figure below, which shows a model of a slab and a cylinder, the cylinder has been defined as a rigid body. When the mesh is generated, the cylinder is meshed with line elements as shown.

Figure 191: 2D Rigid Body Contact Meshing



Using 3D Rigid Body Contact Meshing

This section describes the basic steps for using 3D rigid body contact meshing.

To define a 3D rigid body for contact meshing:

1. Open the model in the Mechanical application.
2. In the Tree, expand the **Geometry** object so that the body objects are visible.
3. Click the body that you want to define as a rigid body.
4. In the **Details> Definition** view for the body, change the value of the Stiffness Behavior control to **Rigid**.

   Note:

   When you change the **Stiffness Behavior** to **Rigid**, an **Inertial Coordinate System** object is added to the Tree automatically. This **Inertial Coordinate System** represents the centroid of the body.

5. If you wish to control the mesh method, insert a mesh method by right-clicking on the **Mesh** object in the Tree and selecting **Insert> Method**.

   Note:

   The Automatic method appears in the Details View.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. In the Details View, scope the mesh method to the rigid body.

Note:

By default, the **Method** control is set to **Quadrilateral Dominant** for rigid bodies, but you can change the value to **Triangles**. When **Method** is set to **Quadrilateral Dominant**, an additional option called the **Free Face Mesh Type** control is available and can be set to either **Quad/Tri** (default) or **All Quad**.

7. If desired, change the value of the **Element Order** (p. 101) control.

8. Generate the mesh by right-clicking on the **Mesh** object in the Tree and selecting **Generate Mesh**.

Note:

The mesh for the 3D rigid body is created only in the contact region (faces in contact).

Thin Solid Meshing

Thin solid meshing is useful for thin solid bodies where one element through the thickness is desired. This meshing also takes advantage of the Mechanical APDL application's SOLSH190 element or the LS-DYNA thick shell element .

- It may be advantageous to use a **Sizing** control on the faces/body along with mapped Face Meshing controls to give a uniform mesh.
- Virtual Topology (p. 545) may be necessary to satisfy the topological criterion for thin solid meshing.

CAD Instance Meshing

The Meshing application supports pattern instances that have been defined for part features or assembly components in a CAD system, such as Creo Parametric, Parasolid, or Solid Edge. These instance definitions remain in the CAD system. When a model with instances is read in to Workbench, the geometry is read once and then copied for each instance. Similarly, when meshing, the Meshing application generates the mesh once and then copies it for each instance. Support for pattern instances provides these benefits:

- Improved geometry import speed because only one instance of a part is read in
- Improved meshing speed because only one instance of a part is meshed, copies of the first instance's mesh are used for the remaining instances

Remember the following information when working with instances:

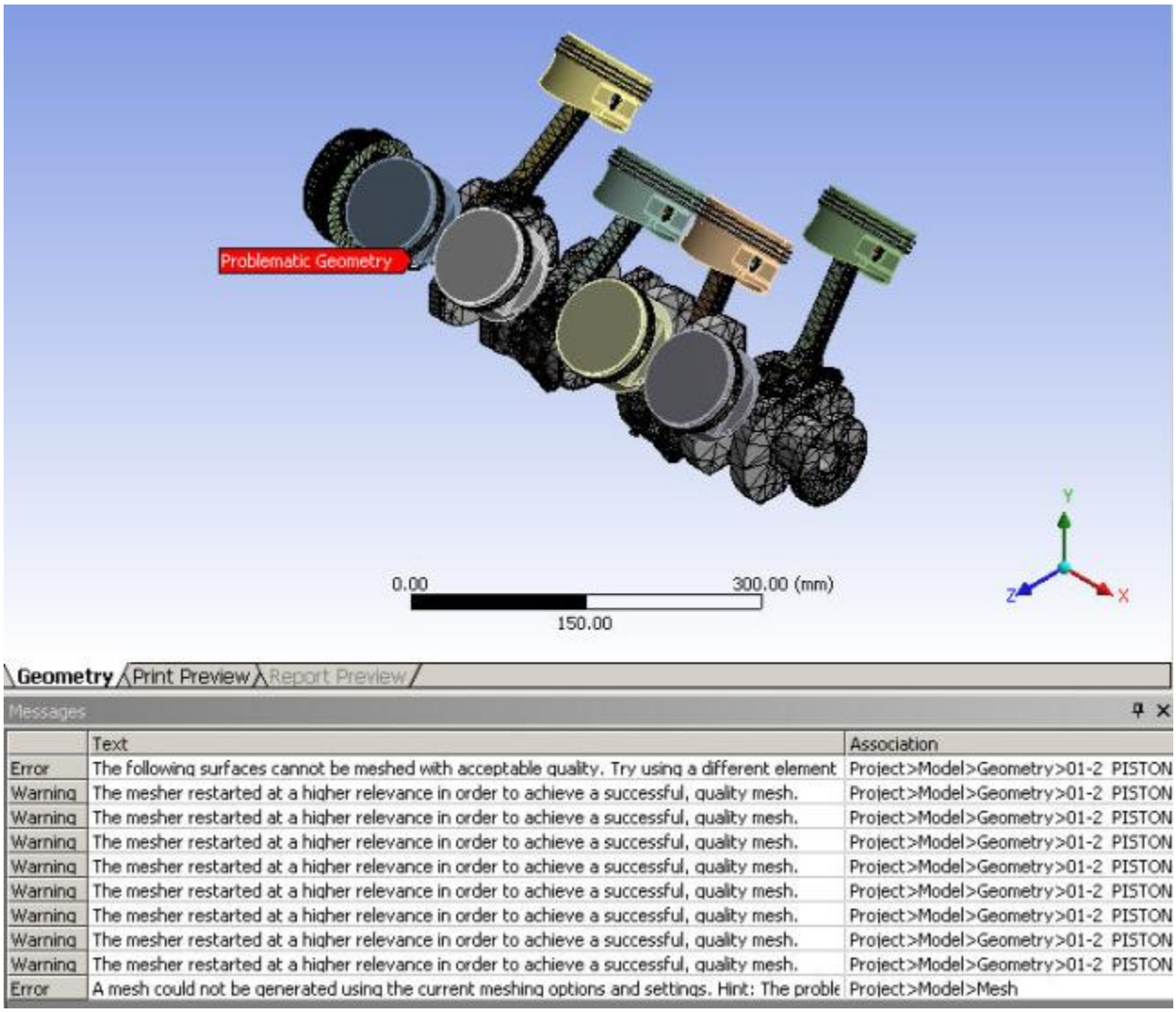
- Instances of *bodies* are not supported. Single body part instancing is supported, as well as certain variations of instancing of multibody parts (for example, multibody parts can be instanced, but a body cannot be instanced within a single part). For more information, refer to the discussion of feature modeling's effect on instance data in the DesignModeler help.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you apply a mesh control to a part that is instanced, each instance must have the same control applied to it or the part will be meshed individually. For example, if your model consists of four parts that are instanced but only one part has an edge sizing control applied to it, that part will be meshed individually and the other three parts will be meshed once and instanced.
- Instancing is not supported for the following controls. Each part that has any of these controls applied to it will be meshed individually, regardless of whether the control in question is applied to all instances:
  - Sphere of influence (p. 304) sizing
  - Body of influence (p. 305) sizing
  - Contact sizing (p. 311)
  - Rigid body meshing (p. 463) (presence of any rigid bodies in the part)
- You can use the **Extend Selection** command to select a set of instances, which can be useful for performing tasks such as applying loads or creating a Named Selection. To select a set of instances, first select one instance, then click the **Extend Selection** drop-down menu and choose **Extend to Instances**. **Extend to Instances** searches for all remaining instances that are defined for the currently selected instance. The **Extend to Instances** menu option is enabled only when pattern instances are defined in the current model.
- When using mesh methods or controls that require both body scoping and face scoping (such as Sweep (p. 235) with source face selection or inflation (p. 345)), you can use the Object Generator to set up the model for instancing. For details, refer to Generating Multiple Objects from a Template Object in the Mechanical help.
- Because the mesh is copied from the first instance to the remaining instance, it follows that each instance will be meshed with the same number of nodes and elements. To confirm this, after CAD instances are successfully meshed, click the **Geometry** object in the Tree Outline. Then click the **Worksheet** toolbar button. When the **Worksheet** appears, click the **Nodes** heading, which sorts the data on that column and allows you to view the matching numbers of nodes and elements for each instance.
- Error handling for instances is also copied. For example, if the mesher fails on one instance, all instances will fail. However, you will receive a warning message for each instance. Refer to Figure 192: Error Handling for Instances (p. 468), which shows a model containing eight identical pistons.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 192: Error Handling for Instances



	Text	Association
Error	The following surfaces cannot be meshed with acceptable quality. Try using a different element	Project>Model>Geometry>01-2 PISTON
Warning	The mesher restarted at a higher relevance in order to achieve a successful, quality mesh.	Project>Model>Geometry>01-2 PISTON
Warning	The mesher restarted at a higher relevance in order to achieve a successful, quality mesh.	Project>Model>Geometry>01-2 PISTON
Warning	The mesher restarted at a higher relevance in order to achieve a successful, quality mesh.	Project>Model>Geometry>01-2 PISTON
Warning	The mesher restarted at a higher relevance in order to achieve a successful, quality mesh.	Project>Model>Geometry>01-2 PISTON
Warning	The mesher restarted at a higher relevance in order to achieve a successful, quality mesh.	Project>Model>Geometry>01-2 PISTON
Warning	The mesher restarted at a higher relevance in order to achieve a successful, quality mesh.	Project>Model>Geometry>01-2 PISTON
Warning	The mesher restarted at a higher relevance in order to achieve a successful, quality mesh.	Project>Model>Geometry>01-2 PISTON
Warning	The mesher restarted at a higher relevance in order to achieve a successful, quality mesh.	Project>Model>Geometry>01-2 PISTON
Error	A mesh could not be generated using the current meshing options and settings. Hint: The proble	Project>Model>Mesh

Meshing and Hard Entities

The two types of hard entities are hard edges and hard points. Hard entities are usually defined in the DesignModeler application or a CAD system.

A hard point is an embedded point in a face or edge. Hard points are captured by nodes during the meshing process. Hard points are not supported for the following mesh methods or controls:

- General sweeping (p. 399)
- 3D inflation (p. 161)

A hard edge is an embedded edge in a face. There are two main types of embedded edges:

- An edge in which one vertex of the edge touches the face boundary, but the other vertex does not touch a face boundary

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- An edge in which neither of the vertices touches the face boundary

Hard edges are not supported for the following mesh methods or controls:

- General sweeping (p. 399)
- 2D or 3D inflation (p. 161)

Note:

- If the methods and controls listed above are required in your mesh, insert a Virtual Topology and use the **Simplify Faces** (p. 552) option to remove the hard entities.
- Other mesh methods have certain limitations in how hard points and hard edges are handled. For more information, refer to Limitations of Using Hard Entities with Other Mesh Controls (p. 469).

Spot Welds

Spot welds are used to connect individual surface body parts together to form surface body model assemblies, just as contact is used for solid body part assemblies. Spot welds are usually defined as hard points in the DesignModeler application or a CAD system:

- When a model is imported into the Meshing application, the mesher simply treats the hard points as embedded points.
- Upon import to the Mechanical application, spot welds are automatically generated where hard points are defined in the model.

For related information, refer to Point in the DesignModeler help and Spot Welds in the Mechanical help.

Limitations of Using Hard Entities with Other Mesh Controls

Limitations of hard entities include the following:

- 3D inflation (p. 161) does not support hard entities of either type. If inflation is applied, a warning message is issued to indicate the hard entities were ignored.
- 2D inflation (p. 161) supports hard points only.
- General Sweeping (p. 399) does not support hard points.
- The MultiZone Quad/Tri (p. 284) and MultiZone (p. 242) mesh methods do not respect hard entities unless their topology is protected (p. 184).
- For the MultiZone (p. 242) mesh method only, the faces that contain the hard entities must be selected as source faces for the hard entities to be respected.
- Hard edges may exist accidentally in a CAD model due to Boolean operations with tight tolerances or other such operations. These accidental hard edges may be undesired, in which case you

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

should remove them by using the **Virtual Topology: Face Simplify** feature or defeaturing them within the DesignModeler application or a CAD system.

Baffle Meshing

The Meshing application provides support for meshing 0-thickness walls, or baffles, as non-manifold faces of a solid body. For such models, you do not have to adjust the mesh size to capture the thin regions.

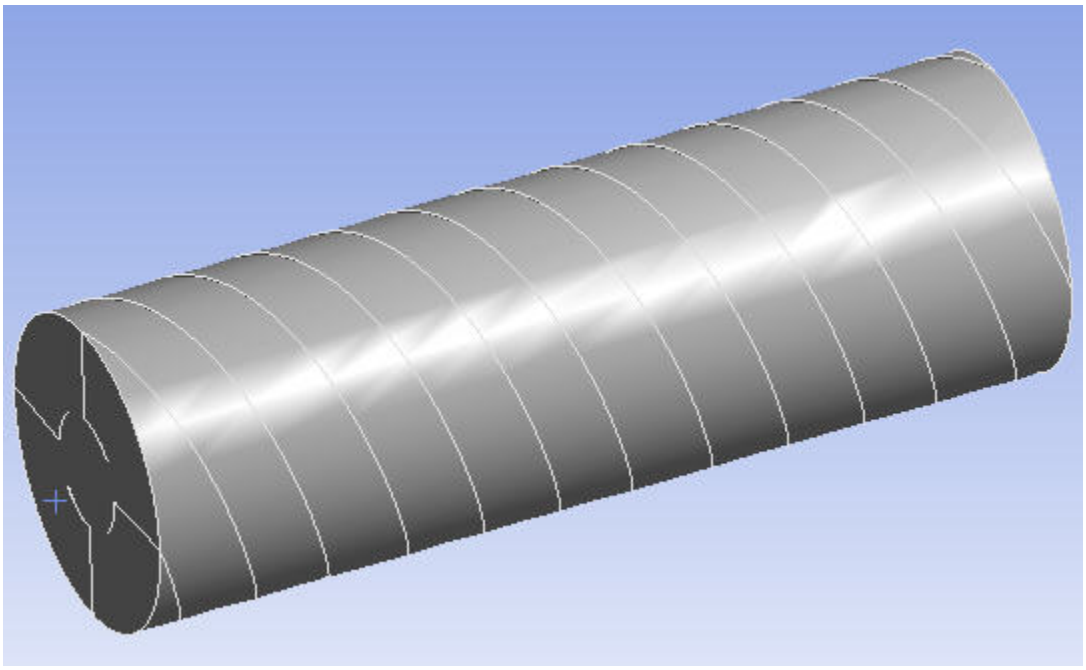
Characteristics and limitations of baffle meshing include:

- For part/body level meshing, baffle meshing is supported by the Patch Conforming Tetra (p. 208), Patch Independent Tetra (p. 210), and MultiZone (p. 242) mesh methods only. If you apply any other mesh method to a body containing baffles, the mesh method will be suppressed, and the reason (not supported) will be reported in the **Active** read-only field in the Details View. In such cases, the body will be meshed with the Patch Conforming Tetra mesh method.
- When the MultiZone mesh method is used, the body with a baffle must be meshed with a free mesh of tetrahedral elements. For this reason, you must set the **Free Mesh Type** to **Tetra** for bodies with baffles.
- The Patch Conforming Tetra and Patch Independent Tetra mesh methods support inflation of baffles. The MultiZone mesh method does not support inflation of baffles.
- When the Patch Conforming Tetra mesh method is used, inflation layers will stair step (p. 174) at free boundary edges of the baffles.
- When the Patch Independent Tetra mesh method is used and Collision Avoidance (p. 174) is set to **Stair Stepping**, inflation layers will stair step at free boundary edges of the baffles. However, if **Collision Avoidance** is set to **Layer Compression**, full prism columns appear at the free boundary edges.
- **Program Controlled** (p. 164) inflation is supported (that is, if you select **Program Controlled** inflation, baffles are automatically selected to be inflation boundaries unless they are in a Named Selection).
- Only two-sided growth cases for inflation are supported.
- Pyramid transitions are supported.
- Prism/pyramid elements are not supported for meshing crossed/intersecting baffles.
- There is a single set of nodes on the internal face.

Figure 193: Cylinder Containing Baffles (p. 471) shows a model of a cylinder that contains a series of baffles.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 193: Cylinder Containing Baffles



In Figure 194: Section Cut Showing Baffle Meshing (p. 471), the model was meshed using **Program Controlled** inflation. Since the baffles were not in a Named Selection, they were automatically selected to be inflation boundaries. A section plane was activated to view a section cut through the model, and the baffle faces were selected in the **Geometry** window.

Figure 194: Section Cut Showing Baffle Meshing

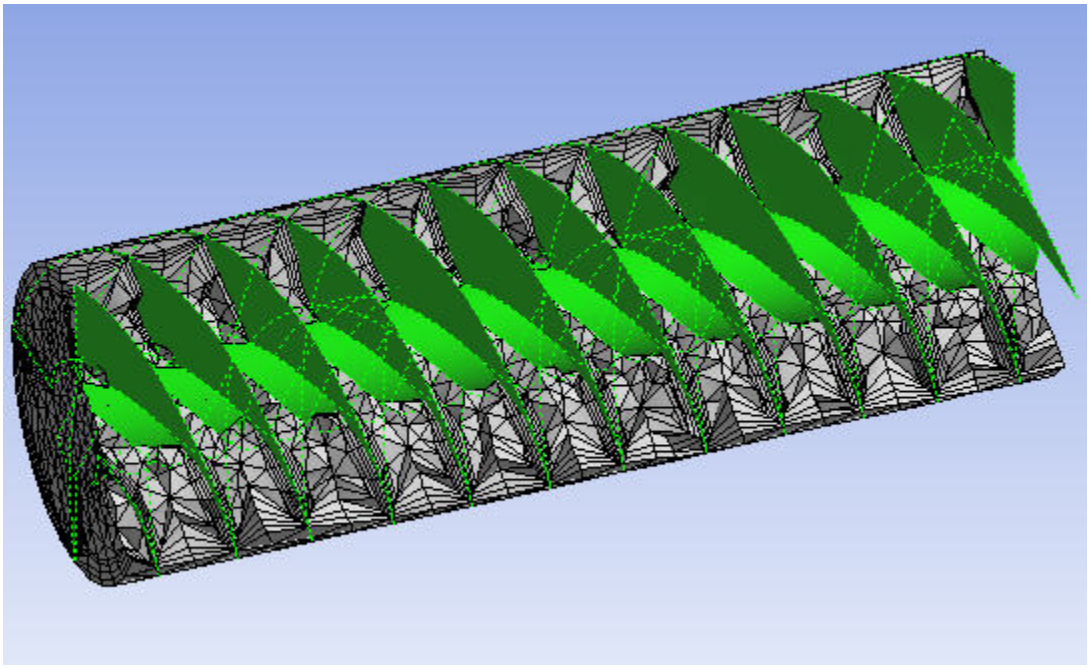
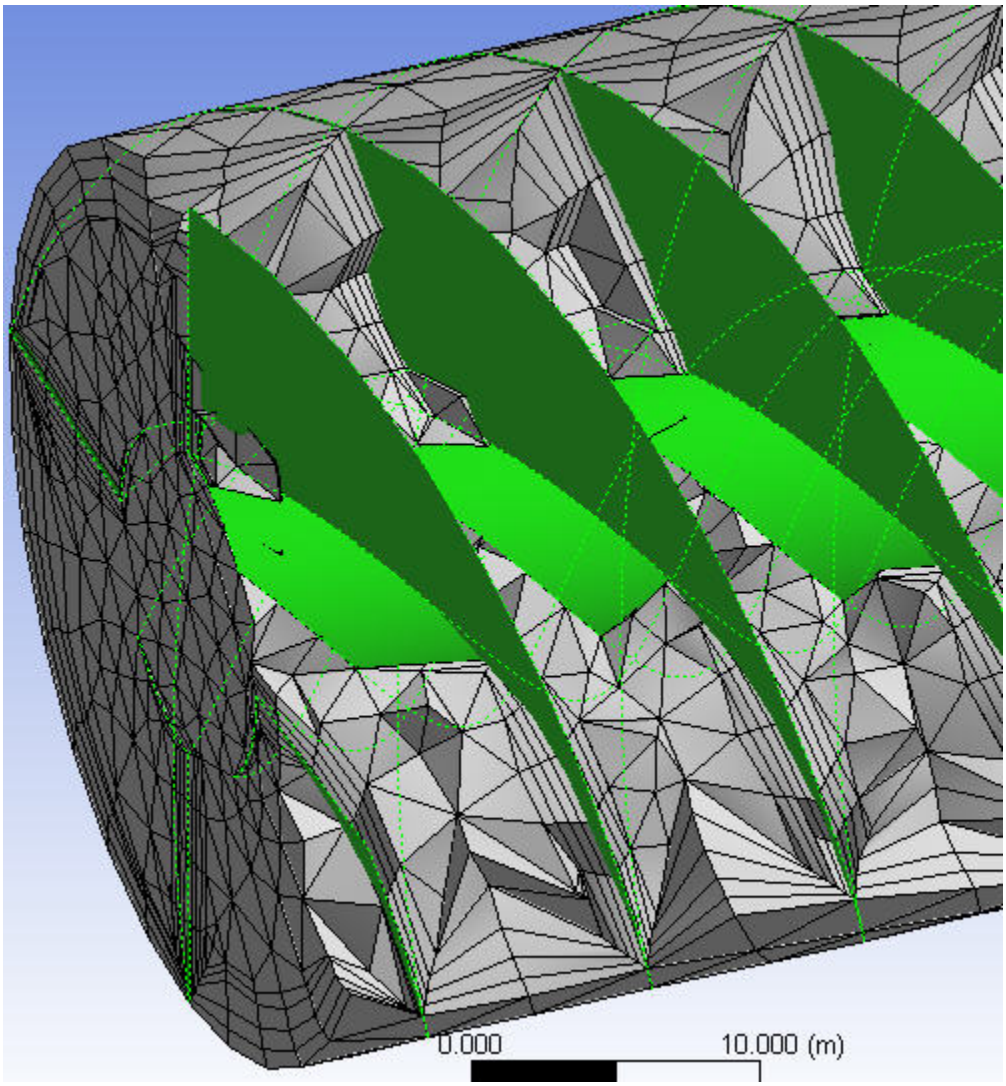


Figure 195: Detail of Inflation on Baffles (p. 472) shows a detailed view of the inflation layers on the baffles.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 195: Detail of Inflation on Baffles



Parallel Part Meshing

You can control three mechanisms in Ansys Workbench that operate in a parallelized manner:

- Remote Solve Manager Design Point updates. Refer to RSM Configuration.
- Parallel Part meshing: **Tools**> **Options**> **Number of CPUs for Parallel Part Meshing**
- Individual mesh methods (**MultiZone Quad/Tri**, **Patch Independent Tetra**, and **MultiZone** only): **Tools Options > Number of CPUs for Meshing Methods**

For the most efficient use of machine resources, it is important that the running processes do not over-saturate the processing cores or the available memory. You must allocate processing cores to each of these mechanisms in a way that provides the most benefit for your workflow. When parallel part meshing is invoked with the default number of CPUs, it automatically uses the cores of all available CPUs with a minimum of 2 gigabytes per CPU core.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parallel Part Meshing Best Practices

Best practices include:

- Know how many physical processing cores are available.
- If you are using Remote Solve Manager (RSM), meshing is done serially. This option cannot be overridden.
- For non-RSM Design Point updates, meshing is done serially by default. You can override this option by setting the option **Number of CPUs for Parallel Part Meshing** explicitly under **Meshing** > **Advanced Options**.
- If you are using more than one processor for individual meshing methods, be sure to set a balance between the two meshing options if you are using methods that support the **Number of CPUs for Meshing Methods** option (**MultiZone Quad/Tri**, **Patch Independent Tetra**, and **MultiZone**). They should be set to an explicit value greater than 0.

  For example, if you are using an 8 core system, setting **Number of CPUs for Parallel Part Meshing** to **3** and **Number of CPUs for Mesh Methods** to **3** will provide a good balance. If the mesh methods that you typically use support the **Number of CPUs for Meshing Methods** option, setting the **Number of CPUs for Parallel Part Meshing** to **2** and the **Number of CPUs for Meshing Methods** to **4** or **5** may potentially provide more benefit.
- If you are using parallel part meshing only, you can set the **Number of CPUs for Parallel Part Meshing** to **0**. In such cases, the software uses as many cores as possible.
- For Parallel Part meshing, you should turn off hyper-threading as this may lead to degradation of parallel performance.
- Note that Parallel Part meshing does not support the following mesh controls:
  - Contact Sizing
  - Fracture
  - Mesh Match via Symmetry
  - Morph Service/Morphing
  - Pinch
  - Post Connection
  - Post-Inflation
  - Preview Surface Mesh/Preview Inflation
  - Retry
  - Refinement

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Meshing: Mesh Control Interaction Tables

This section presents the effects of applying combinations of mesh controls on the same part or body. Topics include the meshing implication when one mesh method is applied in combination with another mesh method, and the effects of applying various mesh controls in combination with the various mesh methods.

Interactions Between Mesh Methods

Interactions Between Mesh Methods and Mesh Controls

Interactions Between Mesh Methods

The tables below present the effects of meshing two or more bodies in a multibody part using a combination of different mesh methods:

- Using combinations of surface mesh methods
- Using combinations of solid mesh methods
- Applying a single 3D inflation control on more than one solid body when a combination of mesh methods has been scoped to the bodies
- Applying a 3D inflation control on a solid body when more than one mesh method has been scoped to the body

---

Note:

- The Cartesian (p. 261) mesh method operates at the part level, and does not support interactions with other mesh methods. If one body in a multibody part is scoped to be meshed with the Cartesian mesh method, all bodies will be added to the scoping.
- Refer to Conformal and Non-Conformal Meshing (p. 21) for information about conformal meshing.

The table below describes the automatic sequencing of surface mesh methods when two mesh methods are being used. If all four methods are being used, the automatic sequence is:

1. Automatic (PrimeMesh) (p. 287)
2. All Triangles (p. 284)
3. Quad Dominant (p. 282)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. MultiZone Quad/Tri (p. 284)

Note:

If you are performing selective meshing, you control the sequence. Refer to Selective Meshing (p. 444) for usage notes.

Surface Mesh Method	Surface Mesh Method			
	All Triangles (p. 284)	**Quad Dominant (p. 282)**	**MultiZone Quad/Tri (p. 284)**	**Automatic (PrimeMesh) (p. 287)**
All Triangles (p. 284)	N/A	**All Triangles** first	**All Triangles** first	**Automatic (PrimeMesh)** only
Quad Dominant (p. 282)	**All Triangles** first	N/A	**Quad Dominant** first	**Automatic (PrimeMesh)** only
MultiZone Quad/Tri (p. 284)	**All Triangles** first	**Quad Dominant** first	N/A	**Automatic (PrimeMesh)** only
Automatic (PrimeMesh) (p. 287)	**Automatic (PrimeMesh)** only	**Automatic (PrimeMesh)** only	**Automatic (PrimeMesh)** only	N/A

The table below describes the automatic sequencing of solid mesh methods when two methods are being used. If more than two methods are being used, the automatic sequence is:

1. Automatic (PrimeMesh) (p. 287)
2. MultiZone (p. 242)
3. General Sweep (p. 399)
4. Thin Sweep (p. 406)
5. Hex Dominant (p. 233)
6. Patch Conforming Tetra (p. 208)
7. Patch Independent Tetra (p. 210)

Note:

- If you are performing selective meshing, you control the sequence. Refer to Selective Meshing (p. 444) for additional usage notes.
- During automatic sequencing of solid mesh methods when inflation has been applied, **Post** inflation is always applied last and uses as its input mesh the complete currently existing part mesh.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solid Mesh Method	Solid Mesh Method						
	Hex Dominant (p. 233)	**General Sweep (p. 399)**	**Thin Sweep (p. 406)**	**Patch Conforming Tetra (p. 208)**	**Patch Independent Tetra (p. 210)**	**MultiZone (p. 242)**	**Auto (P eMes**
Hex Dominant (p. 233)	N/A	**General Sweep** first	**Thin Sweep** first	**Hex Dominant** first	**Hex Dominant** first	**MultiZone** first	**Auto (Prim** f
General Sweep (p. 399)	**General Sweep** first	N/A	**General Sweep** first	**General Sweep** first	**General Sweep** first	**MultiZone** first[1]	**Auto (Prim** f
Thin Sweep (p. 406)	**Thin Sweep** first	**General Sweep** first	N/A	**Thin Sweep** first	**Thin Sweep** first	**MultiZone** first	**Auto (Prim** f
Patch Conforming Tetra (p. 208)	**Hex Dominant** first	**General Sweep** first	**Thin Sweep** first	N/A	**Patch Conforming Tetra** first	**MultiZone** first	**Auto (Prim** f
Patch Independent Tetra (p. 210)	**Hex Dominant** first	**General Sweep** first	**Thin Sweep** first	**Patch Conforming Tetra** first	N/A	**MultiZone** first	**Auto (Prim** f
MultiZone (p. 242)	**MultiZone** first	**MultiZone** first[1]	**MultiZone** first	**MultiZone** first	**MultiZone** first	N/A	**Auto (Prim** f
Automatic (PrimeMesh) (p. 287)	**Automatic (PrimeMesh)** first	**Automatic (PrimeMesh)** first	**Automatic (PrimeMesh)** first	**Automatic (PrimeMesh)** first	**Automatic (PrimeMesh)** first	**Automatic (PrimeMesh)** first	N

1–While mixing **Sweep** and **MultiZone** mesh methods, pre-meshed faces may be used in these ways:

- Mapped faces can be supported as side faces when **MultiZone** or **Sweep** is used to mesh subsequent bodies. The pre-meshed faces may have been generated using either **General Sweep** or **MultiZone**. There are limitations on how the face is mapped. Simple mapped faces (that is, 4-sided) are supported. However, more complicated submapped cases may cause problems.
- Mapped faces can be supported as source faces.
- Free faces (where mesh does not have a quad mapped pattern) can be supported as source faces only.

The table below describes how inflation is handled if you apply a single 3D inflation control on more than one solid body when a combination of mesh methods has been scoped to the bodies.

Method 1	Method 2	Supported Inflation Algorithm
Patch Conforming Tetra (p. 208)	Patch Independent Tetra (p. 210)	Post (p. 173) inflation only
Patch Conforming Tetra (p. 208)	MultiZone (p. 242)	Pre (p. 172) inflation only[1]
Patch Independent Tetra (p. 210)	MultiZone (p. 242)	No inflation allowed; inflation is suppressed

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Method 1	Method 2	Supported Inflation Algorithm
Patch Conforming Tetra (p. 208)	Patch Independent Tetra (p. 210) (+ Method 3 of MultiZone (p. 242))	No inflation allowed; inflation is suppressed
General Sweep (p. 399)	Any other method	No inflation allowed; inflation is suppressed
Thin Sweep (p. 406)	Any other method	No inflation allowed; inflation is suppressed
Hex Dominant (p. 233)	Any other method	No inflation allowed; inflation is suppressed
Automatic (PrimeMesh) (p. 287)	Any other method	No inflation allowed; inflation is suppressed

1– In such cases involving **MultiZone**, the value of the **Inflation Algorithm** control displays as **Pre** but an O-grid-based algorithm specific to **MultiZone** is used. As with the **Pre** inflation algorithm, the mesh is inflated during the meshing process.

The table below describes how inflation is handled if you apply a 3D inflation control on a solid body when more than one mesh method has been scoped to the body. In such cases, the method control that appears lowest in the Tree is respected and therefore inflation is handled as it would normally be handled for that method.

Lowest Method in Tree	Supported Inflation Algorithm(s)
Patch Conforming Tetra (p. 208)	Post (p. 173) or Pre (p. 172) inflation
Patch Independent Tetra (p. 210)	Post (p. 173) inflation only
MultiZone (p. 242)	Pre (p. 172) inflation only[1]
General Sweep (p. 399)	No inflation allowed; inflation is suppressed
Thin Sweep (p. 406)	No inflation allowed; inflation is suppressed
Hex Dominant (p. 233)	No inflation allowed; inflation is suppressed
Automatic (PrimeMesh) (p. 287)	No inflation allowed; inflation is suppressed

1– In such cases involving **MultiZone**, the value of the **Inflation Algorithm** control displays as **Pre** but an O-grid-based algorithm specific to **MultiZone** is used. As with the **Pre** inflation algorithm, the surface mesh is inflated first and then the rest of the volume mesh is generated.

Interactions Between Mesh Methods and Mesh Controls

The tables in this section present the effects of applying various mesh controls in combination with the various mesh methods, and include:

- Using mesh controls with the solid meshing methods

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Using mesh controls with the surface meshing methods

The table below describes the effects of applying the mesh control on the left with each of the solid meshing methods.

Mesh Control	Solid Meshing Methods					
	Patch Conforming Tetrahedron (p. 280)	Patch Independent Tetrahedron (p. 288)	MultiZone (p. 242)	Thin Solid Sweep (p. 233)	Cartesian (p. 261)	Layered Tetrahedron (p. 266)	Automatic (PrimeMesh) (p. 287)[a]
Body Sizing Control (p. 296)	Supported	Supported	Supported	Supported	Supported	Supported	Supported
Face Sizing Control (p. 296)	Supported	Supported	Supported	Supported	Supported	Supported	Supported
Edge Sizing Control (p. 296)	Supported	Supported	Supported	Supported	Supported	Not supported	Supported
Sphere of Influence Control (p. 304)	Supported	Supported	Supported, but only influences source face(s).	Supported, but only influences source face(s).	Not supported	Not supported	Not supported
Body of Influence Control (p. 305)	Supported	Supported	Supported, but only influences source face(s).	Supported, but only influences source face(s).	Not supported	Not supported	Not Supported
Contact Sizing Control (p. 311)	Supported	Supported	Supported	Supported	Not supported	Not supported	Not Supported
Refinement Control (p. 313)	Not supported	Not supported	Supported	Supported	Not supported	Not supported	Not supported
Mapped Face Control (p. 314)	Not supported	Supported	Supported	Supported	N/A. All faces essentially mapped.	Not supported	Not Supported
Match Control (p. 324)	Not supported	Supported, with limitations (p. 334).	Supported	Not supported	Not supported	Not supported	Not supported
Pinch Control (p. 340)	Not supported	Not supported	Not supported	Supported on source and targets.	Not supported	Not supported	Not supported
Inflation Control (p. 345)	Supported	Supported	Supported	Not supported on source	Supported	Not supported	Not supported

Mesh Control	Solid Meshing Methods						
	Patch Conforming Tetrahedron (p. 280)	Patch Independent Tetrahedron (p. 280)	MultiZone (p. 242)	Thin Solid Sweep (p. 233)	Cartesian (p. 261)	Layered Tetrahedron (p. 266)	Automatic (PrimeMesh) (p. 287)[a]
				source via 2D inflation.			
Feature Suppression (p. 350)	Not Supported	Not supported	Not supported	Not supported	Not supported	Not supported	Not supported
Axisymmetric Mesh	Not supported	Not supported	Supported with limitations (p. 258)	Not supported	Not supported	Not supported	Not supported
Geometry Fidelity (p. 386)	Not Supported	Not supported	Not supported	Not supported	Not supported	Not supported	Not supported

[a] Effect of applying mesh controls with **Automatic (PrimeMesh)** for solid bodies when **Shell-Solid Weld -Shared Nodes** is set to **Yes**. For more information See Automatic (PrimeMesh) (p. 287).

The table below describes the effect of applying the mesh control on the left with each of the surface meshing methods.

Mesh Control	Surface Mesh Methods			
	All Triangles (p. 284)	Quad Dominant (p. 282)	MultiZone Quad/Tri (p. 284)	Automatic (PrimeMesh) (p. 287)
Body Sizing Control (p. 296)	Size control affects elements on body and lower topological entities.		Supported	Supported
Face Sizing Control (p. 296)	Size control affects elements on face and lower topological entities.		Supported	Supported
Edge Sizing Control (p. 296)	Size control affects element edge lengths on edge.		Supported	Supported
Sphere of Influence Control (p. 304)	Inserts elements of specified size within sphere.		Not supported	Not supported
Body of Influence Control (p. 305)	Inserts elements of specified size within body. Only available when Size Function is on.		Not supported	Not supported
Contact Sizing Control (p. 311)	Inserts spheres of influence on contact faces in regions within contact tolerance.		Not supported	Not supported
Refinement Control (p. 313)	Refines elements as post-process.		Not supported	Not supported
Mapped Face Control (p. 314)	Mapped faces are meshed before any other faces. Interval assignment may affect edge divisions. No sphere of influence support. No mesh based defeaturing or pinch support.		Supported	Not Supported

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh Control	Surface Mesh Methods			
	All Triangles (p. 284)	**Quad Dominant (p. 282)**	**MultiZone Quad/Tri (p. 284)**	**Automatic (PrimeMesh) (p. 287)**
Match Control (p. 334)	Edge meshes are matched for sheet, 2D, and 3D bodies. Face meshes are matched across bodies. Match controls cannot be applied across multiple parts. Additional restrictions (p. 334) apply.		Not supported	Not supported
Pinch Control (p. 340)	Supported		Not supported	Not supported
Inflation Control (p. 345)	Supported		Supported	Not Supported
Weld (p. 362)	N/A	N/A	N/A	Supported
Repair Topology (p. 355)	N/A	N/A	N/A	Supported
Connect (p. 359)	N/A	N/A	N/A	Supported
Deviation (p. 385)	Not supported	Not supported	Not supported	Supported
Quad Layer (p. 380)	Not supported	Not supported	Not supported	Supported
Pull (p. 516)	Supported	Supported	Supported	Supported
Geometry Fidelity (p. 386)	Not supported	Not supported	Not supported	Not supported

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Meshing: Miscellaneous Tools

The miscellaneous meshing tools described in the following sections include:

Generation of Contact Elements
Renaming Mesh Control Tools
Mesh Numbering
Mesh Editing
Common Display Features

Generation of Contact Elements

When you load a model into the Meshing or Mechanical application, by default, connections are found between parts that have faces in proximity of each other. Depending on the application, you may want the boundaries common to two parts to be similar, so that contact definitions or non-conformal interface definitions may be more accurate. To get common boundaries between parts in an assembly, you should first imprint all the parts with each other in Discovery Modeling or DesignModeler (p. 21). Then, when you edit the model in the Meshing or Mechanical application, you should define specific contact conditions.

One of those conditions is tolerance, which controls the extent of contact between parts in an assembly. Tolerance is set as a percentage of the bounding box of the assembly. The bounding box is the smallest volume that the assembly will fit in. You can change the tolerance (between -100 and 100) in the **Options** dialog box under the Mechanical application's **Connections** category.

The higher the number, the tighter the tolerance. A loose tolerance generally increases the number of contact faces and areas of contact between parts, while a tight tolerance will decrease the number of contact faces.

Each face of a part is checked against the faces of other parts in the assembly. A connection is generated between any faces within the tolerance. You can use overlap tolerances to further limit which faces are in contact if you want only the faces that fully overlap to be found in contact.

When solving in the Mechanical solver, the elements for the two sets of faces that make up a contact pair are compared. Contact elements are generated for element pairs that are within the tolerance, but element pairs outside the tolerance are ignored.

Recommendations for Defining Contact for CFD Analyses

CFD users should be aware of the following recommendations:

- The **Auto Detect Contact On Attach** option controls whether contact detection is computed upon geometry import. If you do not want contact detection to be computed, make sure that it is disabled by selecting **Tools> Options** from the Ansys Workbench main menu, and then se-

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

lecting either the Mechanical or Meshing category as appropriate. The option is enabled by default in both applications.

- If you are an Ansys Fluent user, you generally want to imprint all the parts with each other in Discovery Modeling or DesignModeler (p. 21) as mentioned above. Failing to imprint parts may lead to connections that have cyclic redundancy and may fail to output to the solver.
- For Ansys Fluent users, a boundary zone type of *INTERFACE* is assigned automatically to the contact source and contact target entities that compose contact regions at the time of mesh export. See Special Cases (p. 54) for details.

Renaming Mesh Control Tools

You can rename any of the mesh control tool objects to include the name assigned to the part or body. To do this, right-click the object and choose **Rename Based on Definition** from the context menu. For example, if you scope a **Refinement** tool to a body named **Tube** and choose **Rename Based on Definition**, the mesh control tool name changes from **Refinement** to **Refinement on Tube**. The name change is reflected both in the tree and as a label on the body.

Mesh Numbering

The Mesh Numbering feature allows you to renumber the node and/or element numbers of a generated meshed model consisting of flexible parts. The feature is useful when exchanging or assembling models and could isolate the impact of using special elements such as super elements. For details, refer to Mesh Numbering in the Mechanical help.

In the **Details** view, when the **Read Only** is set to **Yes** under the **Model Assembly** group, you cannot access **Mesh Numbering**. When the mesh is read only, mesh renumbering fails and provides an error message.

Mesh Editing

Mesh Editing enables you to improve or refine the quality of a mesh and more efficiently create continuous, conformal meshes for large models with multiple parts. You can move individual nodes, merge nodes together, match nodes, or use mesh connections to join the meshes of topologically disconnected surface bodies and solids.

If the nodes on two different parts are coincident (there are duplicate nodes at the same location), it is faster to merge the nodes to join them. If the nodes are at different locations, you should use mesh connections or contact matches.

Mesh Connections work only for sheet bodies, and **Contact Matches** work only for solid bodies. You can use **Node Move** and **Node Merge** for solid, sheet, and line bodies.

Note:

Ansys DesignSpace licenses do not support the Mesh Editing feature.

The Mesh Editing feature is described in the following sections:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inserting a Mesh Edit Object
Mesh Connections
Contact Matches
Node Merge
Node Move
Pull

Inserting a Mesh Edit Object

Mesh Editing tools, such as Mesh Connections (p. 485), Contact Matches (p. 496), Node Merge (p. 508), and Node Move (p. 512) are listed in the tree hierarchy as children of the Mesh Edit object.

To insert a **Mesh Edit** object:

- Right-click the **Model** object and choose **Insert > Mesh Edit**.

  If you have already created a **Mesh Edit** object, the option will not be available from the **Insert** menu.

- Right-click the **Mesh** object and choose **Insert**, then choose any of the following:

  – Mesh Connection Group

  – Manual Mesh Connection (for manual mesh connections)

  – Contact Match Group

  – Contact Match (for manual mesh contact match)

  – Node Merge Group

  – Node Merge

  – Node Move

  – Pull

  A parent **Mesh Edit** object is created automatically.

- On the Mesh Edit toolbar, select **Mesh Edit**, and then select an option.

  A parent **Mesh Edit** object is created automatically.

When you add a **Manual Mesh Connection**, **Contact Match**, or **Node Merge** object, the corresponding group object is created as well.

Mesh Connections

Mesh connections enable you to join the meshes of topologically disconnected surface bodies that may reside in different parts. They are an alternate option to connecting the geometry (for example, by using the DesignModeler application to repair small gaps). However, geometry tolerances are

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tighter than the tolerances used by mesh connections and often lead to problems in obtaining conformal mesh.

Scoping

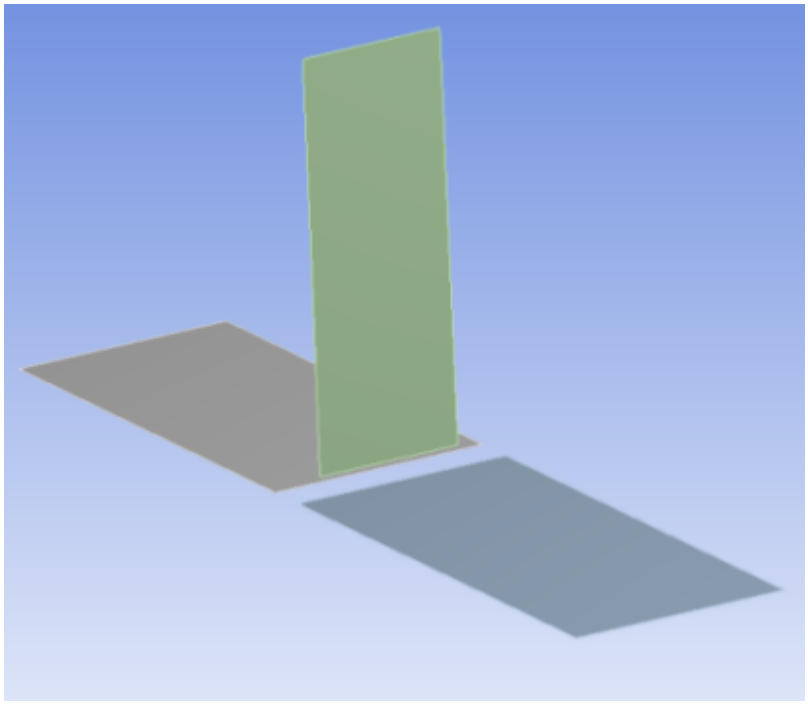
With mesh connections, the connections are made at the mesh level and tolerance is based locally on mesh size. Connections are made edge-to-face or vertex-to-face, they connect edge(s) or vertices on face(s) to another face to pinch out the gap and create conformal mesh between the edge(s) and face(s).

Since mesh connections are a post mesh process–the mesh is pinched in a separate step after meshing is complete–the base mesh is stored to allow for quicker updates. That is, if you change a mesh connection or meshing control, only local re-meshing is required to clean up the neighboring mesh.

Note:

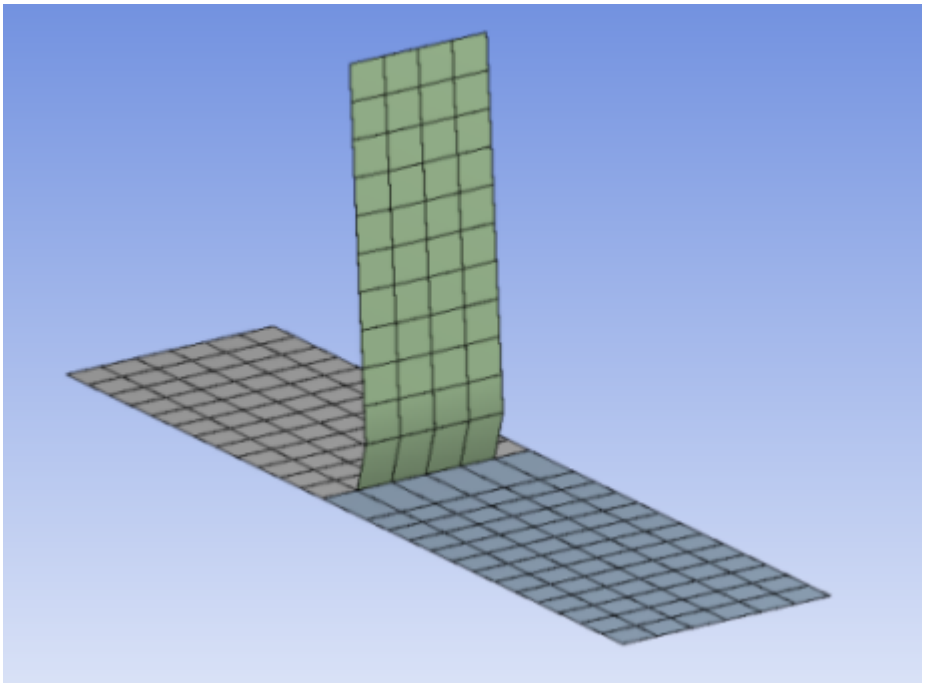
With Ansys Workbench Release 16.0, post pinch behaviors are migrated into **Mesh Connections**. When you regenerate a mesh that was created using **Pinch Behavior** settings, the new mesh might report different results than the previous mesh.

Surface Bodies With No Shared Topology:



Same Surface Bodies With Edge-To-Edge Mesh Connection Established:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Treatment of Legacy Databases

Resuming a legacy database (prior to v16), the application moves **Mesh Connection** objects and the associated **Connection Group** folders that contain them to the new **Model**-level **Mesh Edit** parent folder.

Application

To apply mesh connections:

1. Insert Mesh Connection objects automatically or manually.

   - Mesh connections can be automatically generated using the **Detect Mesh Connections** option available from the right-click context menu of the **Mesh Connections** or **Mesh Connection Group** folder. The **Tolerance Value**, pairing type, and other properties used for auto detection can be set in the Details view of the **Mesh Connection Group** folder under the **Auto Detection** category. Sheet thickness can also be used as a Tolerance Value.

     The automatic mesh connections feature is very helpful, but it can only detect edge-to-face connections. If you need to define edge-to-edge connections, you will need to define them manually. The feature can also find and create connections that you may not want. Always review the connections, or at least be aware that if problems arise, they may be due to automatically generated mesh connections.

   - For more control, or to control the engineering design, you may want to insert **Mesh Connection** objects manually.

     Highlight the **Mesh Connection Group** folder and select the **Manual Mesh Connection** option on the **Mesh Edit** context tab, or right-click the object and select **Insert** > **Manual Mesh Connection**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Manual Mesh Connection** option is also available when the top-level **Mesh/Mesh Edit** folder is selected. Selecting the option at this level adds a **Mesh Connection Group** object in addition to a **Mesh Connection** object.

You can also select one or more Contact Regions or the **Contacts** folder, right-click, and select **Create**>**Mesh Contact(s) or Connection(s)**. This option enables you to create Mesh Connection objects from Contact Regions. The application scopes the new Mesh Connection objects to the geometries of the Contact Region(s) and sets the tolerance to be equal to the trim tolerance of the contact region. The Mesh connections are added into a new Mesh Connection Group folder.

2. In the Details view specify **Primary Geometry** and **Secondary Geometry**.

   - "Primary" indicates the topology that will be captured after the operation is complete. That is, it is the topology to which the secondary topologies in the connection are projected.
   - "Secondary" indicates the topology that will be pinched out during the operation. That is, it is the topology that is projected and merged with the primary.

   The primary geometry can be one or more faces or edges. The secondary geometry can only be one or more edges or vertices. When specifying faces, the annotation is displayed on both sides of the faces.

   Note:

   Mesh connections support common imprints, which involve multiple secondaries connected at the same location to a common primary. See Common Imprints and Mesh Connections (p. 491).

3. In the Details view specify **Tolerance**. The **Tolerance** here has a similar meaning to the **Tolerance Value** global connection setting, and is represented as a transparent sphere. See Tolerances Used in Mesh Connections (p. 489) for details about **Tolerance** and how it relates to the **Snap Tolerance** described below.

4. For edge-to-face mesh connections only, in the Details view specify **Snap to Boundary** and **Snap Type**. When **Snap to Boundary** is **Yes** (the default) and the distance from a secondary edge to the closest mesh boundary of the primary face is within the specified snap to boundary tolerance, nodes from the secondary edge are projected onto the boundary of the primary face. The joined edge will be on the primary face along with other edges on the primary face that fall within the defined pinch control tolerance. See Pinch Control (p. 340) for details.

   Snap Type appears only when the value of **Snap to Boundary** is **Yes**.

   - If **Snap Type** is set to **Manual Tolerance** (the default), a **Snap Tolerance** field appears where you may enter a numerical value greater than 0. By default, the **Snap Tolerance** is set equal to the pinch tolerance but it can be overridden here. See Tolerances Used in Mesh Connections (p. 489) for details about **Snap Tolerance** and how it relates to the **Tolerance** described above.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If **Snap Type** is set to **Element Size Factor**, a **Primary Element Size Factor** field appears where you may enter a numerical value greater than 0. The value entered should be a factor of the local element size of the primary topology.

5. Highlight the **Mesh Edit** folder and choose **Generate** (right-click and choose from context menu). The surface bodies are displayed and show the mesh connections.

6. If necessary, review the mesh connections:

   a. Select one or more **Mesh Connection** or **Mesh Connection Group** objects, right-click, and select **Create Named Selections**.

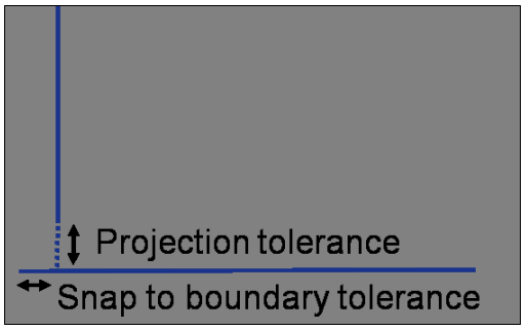
      A named selection is created for each mesh connection you selected. If you selected a mesh connection group, a named selection is created for each mesh connection within the group. Each named selection is automatically given the same name as the mesh connection from which you created it.

   b. Click a named selection to view the mesh for the mesh connection.

Tolerances Used in Mesh Connections

You can set two separate tolerances to define mesh connections. Setting appropriate tolerances is often critical to obtaining high quality mesh that adequately represents the geometry you want to capture.

- **Tolerance** – Projection tolerance to close gaps between bodies.
- **Snap Tolerance** – Snap to boundary tolerance to sew up mesh at the connection (applicable to edge-to-face mesh connections only).



The **Tolerance** value is used to find which bodies should be connected to which other bodies. Setting a larger **Tolerance** connects more bodies together, while setting it smaller may cause some connections to be missed. For this reason, you might want to set this to a larger value than needed. Setting a smaller value can avoid problems in automatic mesh connection creation, but can also result in other problems because the tolerance used in meshing is inherited from automatic mesh connection detection settings.

Using a Large Tolerance Value

For a large assembly for which you do not want to define mesh connections manually, automatic mesh connection detection provides many benefits. Setting a large **Tolerance** value to find connections

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

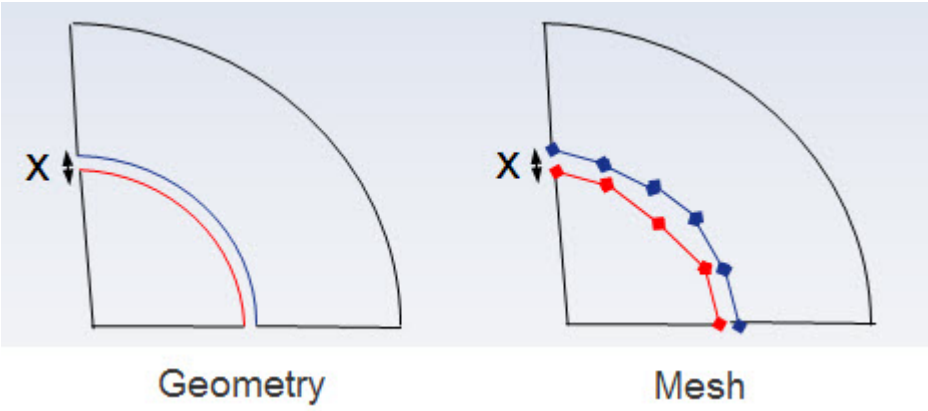
yields more connections, which provides a higher level of comfort that the model is fully constrained. However, larger values can be problematic for the following reasons:

- When more automatic mesh connections are created, more duplicates can be created and the mesher decides ultimately which connections to create. In general, making these decisions yourself is a better approach.
- The **Snap Tolerance** defaults to the same value as the **Tolerance**. If the value of **Tolerance** is too large for **Snap Tolerance**, the mesher may be too aggressive in pinching out mesh at the connection, and hence the mesh quality and feature capturing may suffer.

Using a Small Tolerance Value

When mesh connections are generated automatically, the **Tolerance** is used on the *geometry* edges and faces to determine which entities should be connected. However, the connections themselves are not generated until meshing occurs. Because the connections are performed on nodes and elements of the mesh rather than on the geometry, the tolerances do not translate exactly.

For example, in the case below, you would want to set a **Tolerance** that is slightly larger than the gap in the geometry. If the gap is defined as *x* and the tolerance is set to *x*, automatic mesh connection detection could find the connection, but the meshing process may result in mesh that is only partially connected.



Tips for Setting Tolerances

As detailed above, setting the correct tolerance can be very important, and in some cases may require some speculation and/or experimentation. The following tips may help:

- You can adjust the **Tolerance** used to generate automatic mesh connections after the connections are found. Sometimes it is a good idea to use one **Tolerance** value to find the mesh connections, select all the mesh connections, and then reduce or increase the **Tolerance** later.
- Having **Snap to Boundary** turned on and using a **Snap Tolerance** are not always advisable. It depends on the model and the features you want to capture.

Mesh Sizing and Mesh Connections

Mesh size has an effect on the quality and feature capture of a mesh connection as follows:

- Mesh size always affects the base mesh, as features are only captured relative to mesh size.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- During mesh connection processing, the base mesh is adjusted according to the common imprint/location. In cases where there is a large projection or a large difference in mesh sizes between the primary entity and the secondary entity, the common edge between bodies can become jagged. Also, as local smoothing takes place, there can be some problems in transition of element sizes. You can often use one of the following strategies to fix the problem:
  - Use more similar sizes between source and target.
  - Improve the tolerance used by mesh connections (either for projection, or for snapping to boundary).
  - Adjust the geometry's topology so that the base mesh is more accommodating to the mesh connection.

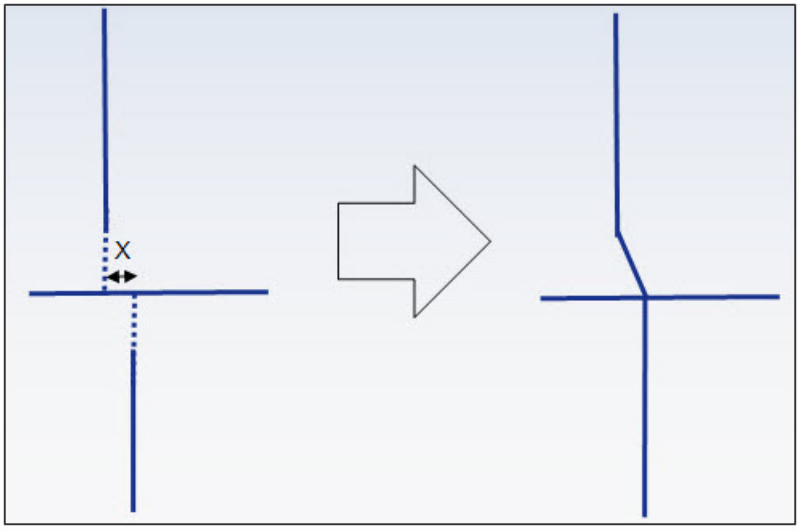
See Common Imprints and Mesh Connections (p. 491).

Common Imprints and Mesh Connections

The tolerance for common imprints comes from the minimum element size in the footprint mesh, which is the horizontal plate in the example below. Common imprints are made if the gap between imprints is smaller than or equal to 5 percent of the element size in the connection region. For this reason, setting the mesh size appropriately is important to control whether the imprints will be common or not.

For example, in the case shown below, if you want a common imprint, the minimum element size is set to **Yes**) should be >*x*.

In this case, you could scope local face mesh sizing on the horizontal plate to control the sizing.



Mesh Connections for Selected Bodies

You can select a geometric entity and lookup the mesh connection object in the tree outline. To find the relevant mesh connection object:

- Right-click a geometric entity, and then click **Go To** > **Mesh Connections for Selected Bodies**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh Connections Common to Selected Bodies

You can select a pair of geometric entities and lookup the shared **Mesh Connection** object in the tree outline. To find a relevant mesh connection object:

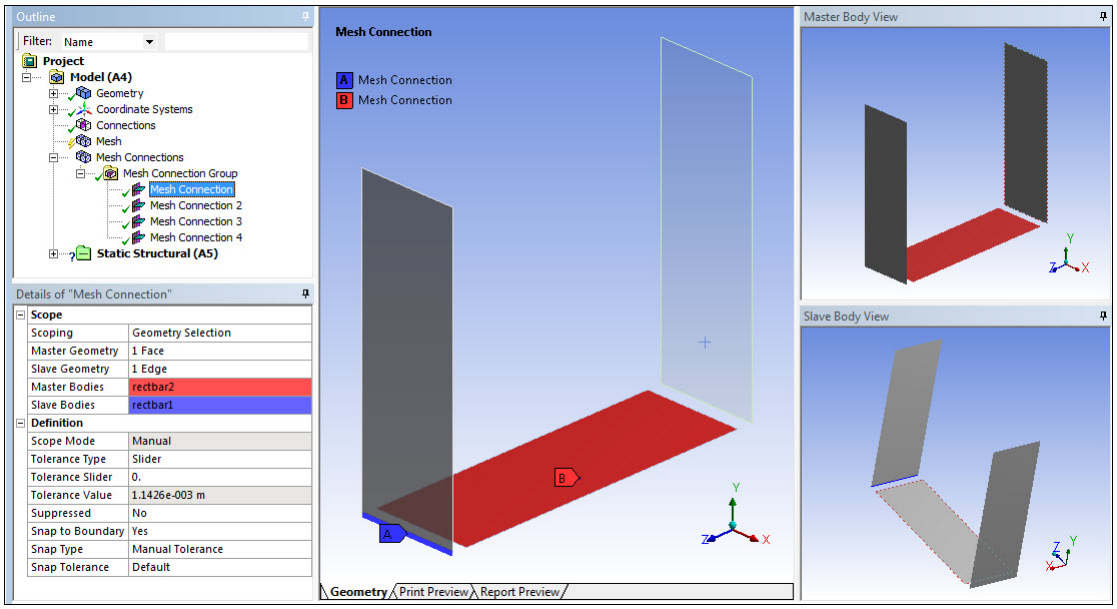
- Select the appropriate pair, and then click **Go To** > **Mesh Connections Common to Selected Bodies**.

This option can be helpful for finding spurious mesh connections, in which case duplicates can be removed.

Displaying Multiple Views of Mesh Connections

Use the **Body Views** button on the **Mesh Edit** Context tab to display parts in separate auxiliary windows.

For closer inspection of mesh connections, you can use the **Show Mesh** option on the Display Context tab along with **Body Views** and the **Sync Views** toggle button. When the Body Views button is engaged, any manipulation of the model in the Geometry window will also be reflected in both auxiliary windows. The **Body Views** toggle button enables you to display parts in separate windows and the **Sync Views** toggle button, when activated, causes any change in the Geometry window to also be reflected in the auxiliary windows.



Merging Mesh Connections

Mesh connections can be merged by selecting the desired objects, right-clicking, and selecting the Merge Selected Mesh Connections option. During the process, the application deletes the original objects and creates a new Mesh Connection object with a combined scoping.

Mesh Connections can only be merged under the following criteria:

- The mesh connections are not already connected.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The primary and secondary geometry must have similar topology for the connections being merged. For example, if you are merging two connections and the first has a face for the primary geometry and the second has an edge for the primary geometry, the connections cannot be merged. If both primaries are faces and both secondaries are edges, the connections can be merged.

When mesh connections are merged, the new mesh connections contain the merged set of entities as primaries and secondaries.

Note:

Be aware that the merge operation process can create undesired connections. This can create a connection that is not appropriate for mesh generation.

Diagnosing Failed Mesh Connections

The state of each mesh connection is displayed in the Tree Outline. For a description of the various states, see Understanding Mesh Connection and Contact Match States (p. 584).

General Failures

In the event of a general mesh connection failure, the following approach is recommended:

1. Select an ignored or failed mesh connection shown in the tree and look at **Control Messages** in the Details View.

   Note:

   You can use the **Filter** to identify **Mesh Connection** objects that are **State>Ignored**. However, if a mesh connection is in an "error" state, it cannot be filtered in the tree.

2. Click **Yes, Click to Display** to display related error messages.
3. Right-click the error messages:
   a. If a message provides "Problematic Geometry" information:
      i. Select the message, right-click, and select **Show Problematic Geometry** from the context menu.

         This action highlights the geometry in the Geometry window that is responsible for the message.

         Note:

         Any error message that is related to a specific mesh connection will be associated with the secondary geometry in the connection.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ii. Select the problematic bodies, right-click, and select **Go To > Mesh Connections for Selected Bodies**.

This action highlights all mesh connections attached to the problematic geometry.

iii. Review the tolerances and mesh sizes associated with the highlighted connections.

b. If a message provides "Go to Body" information:

i. Select the message, right-click, and select **Go to Body** from the context menu.

This action highlights the object in the Details view that is responsible for the message.

ii. Review the tolerances and mesh sizes associated with the highlighted body or bodies.

This action highlights all mesh connections attached to the problematic geometry.

Mesh Connection Failure

If you receive an error or warning message about one or more mesh connections:

1. Highlight and review the message.
2. Right-click the message and select the option **Go To Object**. The corresponding **Mesh Connection** object that is at issue becomes active in the tree.
3. Verify that all of the associated properties are properly defined.
4. Right-click the message and select the option **Show Problematic Geometry**. The corresponding **Primary**/**Secondary** geometry that is at issue becomes highlighted in the **Graphics** window.
5. Verify that the all of the associated geometries are properly defined.

Failures Due to Defeaturing from MultiZone Quad/Tri Meshing and/or Pinch Controls

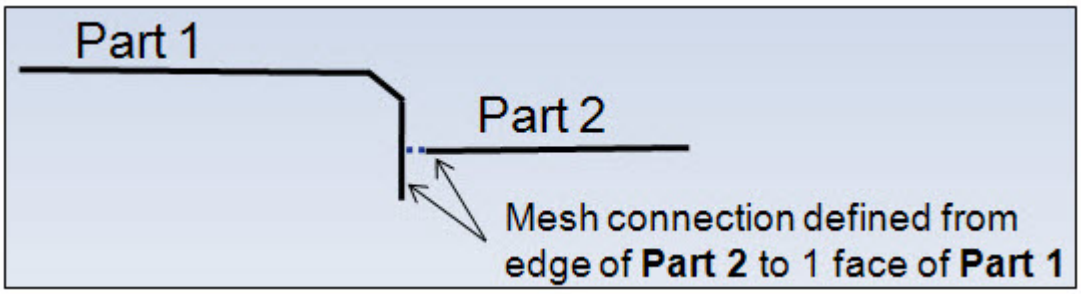
Due to the patch independent nature of the MultiZone Quad/Tri (p. 284) mesh method, a connection may fail because the mesh is associated with some face of the body but not with the face that is involved in the connection. This type of mesh connection failure, which may also occur when pinch controls are defined, is the result of the part mesh being significantly defeatured prior to mesh connection generation. To avoid mesh connection failures when using **MultiZone Quad/Tri** and/or pinch controls, use one of the following approaches:

- Use virtual topology (p. 545) to merge the faces of interest with the adjacent faces to create large patches, and then apply mesh connections to the patches.
- Protect small faces in mesh connections by defining Named Selections.

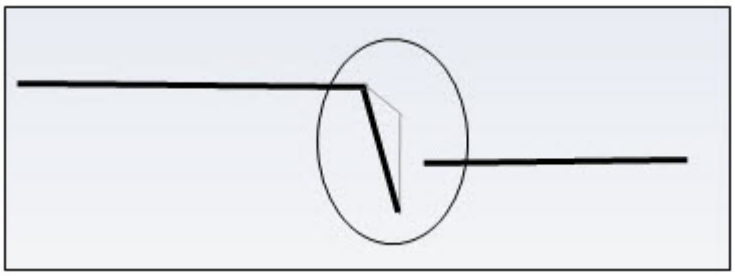
The software does not automatically extend the connection region because doing so may lose the engineering intent of the model.

For example, consider the two parts shown below.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you are using the **MultiZone Quad/Tri** mesh method or pinch controls, the part mesh may look like the one shown below. Notice that one face has been defeatured out.



In this case:

- If the defeatured face is the one defined in the mesh connection, the connection will fail.
- If the other face is the one defined in the mesh connection, the connection will succeed.
- If you include both faces in the mesh connection, the connection will succeed.

Since you cannot always control which face is defeatured, the most robust and recommended approach is to include both faces in the mesh connection.

Points to Remember

- After model assembly, you cannot generate new mesh connections in a mesh that already has mesh connections. Mesh connections only work with a model assembly if the mesh is unconnected in the upstream systems. Because the base meshes from the upstream systems are not available to the assembled model, you must regenerate the mesh to use mesh connections.
- The mesh must be up-to-date before you can generate mesh connections.

  If the mesh is not up-to-date, then the base mesh will be regenerated when you generate the mesh connections.
- Although the tolerance used for finding mesh connections and for generating mesh connections may be the same value, the tolerance itself has slightly different meanings in the two operations. When finding mesh connections, the tolerance is used to identify pairs of geometry edges or face(s)/edge(s). When generating mesh connections, the tolerance is used in pinching together the edge mesh or edge/face mesh. Since the geometry consists of NURBS, and the mesh consists of linear edges, the same tolerance may mean something slightly different in the two operations.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, consider a geometry that consists of two cylindrical sheet parts that share an interface constructed from the same circle. Also consider that you are finding mesh connections with a tolerance of 0.0. In this case, the mesh connection is easily found because the two edges are exactly the same. However, when the mesh connection is being formed, some segments of the edge may fail to be pinched together if the mesh spacing of the two parts is different and therefore the tolerance of the edge mesh is different. Also see Tolerances Used in Mesh Connections (p. 489).

- For a higher order element, a midside node along the connection between a secondary and a primary is located at the midpoint between its end nodes, instead of being projected onto the geometry.
- Although mesh connections do not alter the geometry, their effects can be previewed and toggled using the Display Context tab.
- For Error Limits (p. 127), mesh connections support the **Standard Mechanical** option only.
- If you define a mesh connection on topology to which a match control (p. 334), Face Meshing control (p. 314), or inflation control (global (p. 161) or local (p. 345)) is already applied, the mesh connection may alter the mesh, which in turn may eliminate or disable the match, mapped face meshing, or inflation control.
- Mesh connections cannot be mixed with refinement (p. 313) or post inflation (p. 170) controls.
- A mesh connection scoped to geometries (for the primary and the secondary) that lie on the same face are ignored by the mesher, and, as a result, no mesh connection is generated.
- Refer to Clearing Generated Data (p. 541) for information about using the **Clear Generated Data** option on parts and bodies that have been joined by mesh connections.
- Refer to Using the Mesh Worksheet to Create a Selective Meshing History (p. 448) for information about how mesh connection operations are processed by the **Mesh** worksheet.
- Mesh connections are not supported for external mesh models.
- Mesh connections are not supported between solid bodies and sheet bodies in a multibody part, or between sheet bodies and line bodies in a multibody part.

Contact Matches

Contact matches enable you to match mesh nodes between topologically disconnected solids within a specified tolerance. They are an alternate option to imprinting faces (for example, by using the DesignModeler application). Geometry tolerances are typically tighter than the tolerances used by contact matches, which can lead to problems in obtaining conformal mesh. In these scenarios, contact matches provide a more robust option.

Similar to mesh connections, contact matches are performed on mesh nodes. Contact matches can only be face-to-face between solid bodies.

Contact matches are a post-mesh operation, performed after the base mesh has been generated. The base mesh is then stored so that if you change a contact match, only local re-meshing is required to clean up the neighboring mesh. Likewise, if you make any changes to the base mesh, the contact matches must be re-generated.

Considerations for Contact Matches

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

How Mesh Size Affects Contact Matches
How Tolerances Affect Contact Matches
Applying Contact Matches
Displaying Multiple Views of Contact Matches
Troubleshooting Failed Contact Matches

Considerations for Contact Matches

You should be aware of the following points regarding contact matches:

- After model assembly, you cannot generate new contact matches in a mesh that already has contact matches.

  Contact matches only work with a model assembly if the mesh is unconnected in the upstream systems. Because the base meshes from the upstream systems are not available to the assembled model, you must regenerate the mesh to use contact matches.

- The mesh must be up-to-date before you can generate contact matches.
- Contact matches are only supported for the patch conforming mesh method.

  Contact matches are not supported for the following mesh methods:

  – Mixed order meshing (p. 202)

- For a higher order element, a midside node along the connection between a secondary and a primary is located at the midpoint between its end nodes, instead of being projected onto the geometry.
- For Error Limits (p. 127), contact matches support the **Standard Mechanical** option only.
- If you define a contact match on a topology to which a match control (p. 334), Face Meshing control (p. 314), or inflation control (global (p. 161) or local (p. 345)) is already applied, the contact match may alter the mesh, which in turn may eliminate or disable the match, mapped face meshing, or inflation control.
- Contact matches cannot be mixed with refinement (p. 313) or post inflation (p. 170) controls.
- A contact match scoped to geometries (for the primary and the secondary) that lie on the same face are ignored by the mesher, and, as a result, no contact match is generated.
- Contact matches are not supported for external mesh models.

How Mesh Size Affects Contact Matches

Mesh size affects the quality and feature capture of a contact match as follows:

- Mesh size always affects the base mesh, as features are only captured relative to mesh size.
- During mesh contact match processing, the base mesh is adjusted according to the common imprint and location.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In cases where there is a large projection or a large difference in mesh sizes between the primary entity and the secondary entity, there could be problems in the transition of the mesh away from the contact match area. If the mesh size difference is too great, the contact match will not be generated. You can often use one of the following strategies to fix the problem:

  – Use more similar sizes between source and target.

  – Improve the tolerance used by contact matches.

  – Adjust the geometry's topology so that the base mesh is more accommodating to the contact match.

- The **Tolerance** also affects how the mesh is matched for these types of bodies. For more information, see How the Tolerance Affects Gaps and Boundaries (p. 499).

How Tolerances Affect Contact Matches

Setting appropriate tolerances is critical to obtaining high quality mesh that adequately represents the geometry you want to capture.

The **Tolerance** value is used to find which mesh nodes on a body should be matched to mesh nodes on another body. Setting a larger **Tolerance** matches more nodes, while setting it smaller may cause some nodes not to be matched. For this reason, you might want to set this to a larger value than needed. Setting a smaller value can avoid problems in automatic contact matching, but can also result in other problems because the tolerance used in meshing is inherited from automatic contact match detection settings.

Considerations for Using a Large Tolerance Value

For a large assembly for which you do not want to define contact matches manually, automatic mesh contact match detection provides many benefits. Setting a large **Tolerance** value to find contact matches yields more matches.

However, larger values can be problematic. When more automatic contact matches are created, more duplicates can be created, which can cause problems when attempting to match the mesh. In general, making these decisions yourself is a better approach.

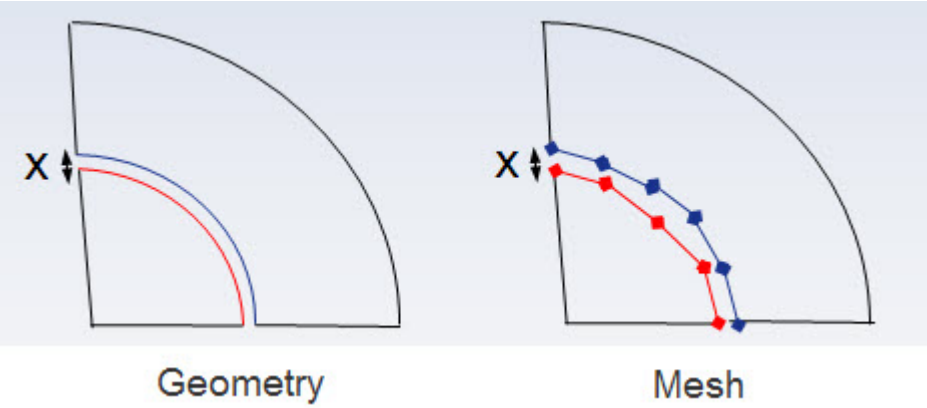
Considerations for Using a Small Tolerance Value

When contact matches are generated automatically, the **Tolerance** is used on the geometry edges and faces to determine which entities should be matched. However, the contact matches themselves are not generated until after the mesh has been generated. Because the contact matches are performed on nodes and elements of the mesh rather than on the geometry, the tolerances do not translate exactly.

For example, in the case below, you would want to set a **Tolerance** that is slightly larger than the gap in the geometry. If the gap is defined as *x*, and the tolerance is set to *x*, automatic mesh contact match detection could find the connection, but the meshing process may result in mesh that is only partially matched.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 196: Setting the Contact Match Tolerance

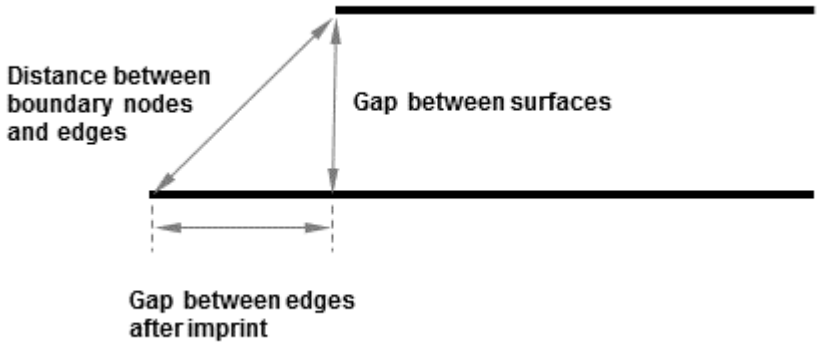


How the Tolerance Affects Gaps and Boundaries

The tolerance also controls how the mesh nodes are matched when there is a gap between the boundary edges of the "Primary" and "Secondary" bodies, or a gap between the bodies themselves, or both.

The following figure shows how the tolerance can be used to match mesh nodes between two solid bodies. In this example, there is a gap between the bodies, as well as a gap between the boundary edges.

Figure 197: Contact Match with Gaps Between "Primary" and "Secondary" Bodies



Assuming that the mesh size is not too large, the following table describes how the mesh will be matched when there are gaps between bodies and boundaries:

Table 4: Mesh Matching for Gaps

If...	Then...
There is a gap between the bodies only	The mesh is matched between the bodies as long as the gap between surfaces is within the specified tolerance. The gap will remain, but the mesh nodes will be matched.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If...	Then...
There is a gap between the boundary edges, but not between the bodies	This mesh is generated first, and the parts are meshed separately. The gap may be meshed depending on the mesh sizes being used and whether there is an imprint in the geometry. **Note:** If there is an imprint in the geometry and you want to remove the gap, you can remove it by inserting a pinch control prior to meshing. For more information, see Pinch Control (p. 340).
There is a gap between both the bodies and the boundary edges	The mesh is matched along the boundaries as long as the gap is within the specified tolerance, and the mesh size is less than or equal to the size of the gap. If the gap is not within the tolerance, or if the mesh size is too large, then the mesh is matched from the edge to the interior.

The mesh size also affects how the mesh is matched for these types of bodies. For more information, see How Mesh Size Affects Contact Matches (p. 497).

Applying Contact Matches

You can apply contact matches by using any of the following methods:

- Apply contact match controls from contact regions (p. 500)
- Automatically apply contact matches (p. 502)
- Manually apply contact matches (p. 503)

Applying Contact Match Controls from Contact Regions

1. In the Tree view, click the **Contacts** object and verify that, in the **Auto Detection** category, the **Tolerance Value** is set appropriately.

   This value controls the relative distance that the application will use to search for contact between parts. However, if the tolerance is set too high, then extraneous contact matches might be generated.

   For more information about auto detection tolerance, see Connection Group.

2. Select the contact regions for which you want to create contact matches by doing one of the following:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To create contact matches for one or more contact regions, select the Contact Regions, right-click, and select **Create** > **Mesh Contact(s) or Connection(s)**
- Or, to create contact matches for all of the contact regions, right-click the **Contacts** folder and select **Create** > **Mesh Contact(s) or Connection(s)**

The application scopes the new **Contact Match** objects to the contact region(s), and sets the tolerance to be equal to the trim tolerance of the contact region. The **Contact Match** objects are added into a new **Contact Match Group** folder.

3. In the Details view, specify the tolerance by setting the **Tolerance Type** and **Tolerance Value**.

   The tolerance here has a similar meaning to the **Tolerance Value** global connection setting, and is represented as a transparent sphere.

   Setting the correct tolerance can be very important, and in some cases may require some speculation and experimentation. You can adjust the tolerance after the contact matches are generated by selecting the contact matches and then changing the **Tolerance Value**.

   For details on setting the tolerance, see How Tolerances Affect Contact Matches (p. 498)

4. Generate the contact matches by doing one of the following:

To...	Do this...
Generate all contact matches	Right-click the **Mesh Edit** folder and select **Generate**.
Generate contact matches for a contact match group	Right-click the **Contact Match Group** object and select **Generate**.

   If the base mesh is out-of-date, it is regenerated. The nodes are matched between the primary and secondary geometries, and a message appears displaying the number of node pairs that were matched.

   If the normals between the primary and secondary faces are misaligned, some contact matches may not be generated.

5. If necessary, review the contact matches:

   a. Select one or more **Contact Match** or **Contact Match Group** objects, right-click, and select **Create Named Selections**.

      A named selection is created for each contact match you selected. If you selected a contact match group, a named selection is created for each contact match within the group. Each named selection is automatically given the same name as the contact match from which you created it.

   b. Click a named selection to view the mesh for the contact match.

      To better view the mesh on the contact region, click the **Wireframe** button on the **Graphics** toolbar.

6. If desired, merge the nodes by dragging the contact matches into a **Node Merge Group**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The mesh nodes are matched during the contact match operation, but they are not merged. If you want conformal mesh, you should merge the nodes.

For more information about merging nodes, see Node Merge (p. 508).

7. To convert the contact matches to geometry selections, select the contact regions, right-click, and select **Convert To** > **Geometry Selection**.

   Converting the contact matches is helpful if you want to delete the contact regions, but wish to retain the contact matches.

Automatically Applying Contact Matches

1. If necessary, insert a Contact Match Group folder (p. 485).
2. Select the **Contact Match Group** folder and, in the Details view, set the **Auto Detection** properties as needed.
3. Right-click the **Contact Match Group** folder, and then select **Detect Connections**.
4. In the Details view, verify the properties.

   Primary Geometry, Secondary Geometry

   "Primary" indicates the topology that will be captured after the operation is complete. That is, it is the topology to which the nodes in the secondary topologies are matched. The primary geometry can be one or more faces.

   "Secondary" indicates the topology that will be matched to the primary during the operation. The secondary geometry can be one or more faces.

   Tolerance

   The **Tolerance** here has a similar meaning to the **Tolerance Value** global connection setting, and is represented as a transparent sphere.

   Setting the correct tolerance can be very important, and in some cases may require some speculation and experimentation. You can adjust the tolerance after the contact matches are generated by selecting the contact matches and then changing the **Tolerance Value**.

   For details on setting the tolerance, see How Tolerances Affect Contact Matches (p. 498)

5. Generate the contact matches by doing one of the following:

To...	Do this...
Generate all contact matches	Right-click the **Mesh Edit** folder and select **Generate**.
Generate contact matches for a contact match group	Right-click the **Contact Match Group** object and select **Generate**.

If the base mesh is out-of-date, it is regenerated. The nodes are matched between the primary and secondary geometries, and a message appears displaying the number of node pairs that were matched.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The automatic contact match detection might detect contact matches that are undesirable. Therefore, if any contact matches fail, you should verify that the contact match is necessary before attempting to correct the error (p. 505).

If the normals between the primary and secondary faces are misaligned, some contact matches may not be generated.

6. If necessary, review the contact matches:

   a. Select one or more **Contact Match** or **Contact Match Group** objects, right-click, and select **Create Named Selections**.

      A named selection is created for each contact match you selected. If you selected a contact match group, a named selection is created for each contact match within the group. Each named selection is automatically given the same name as the contact match from which you created it.

   b. Click a named selection to view the mesh for the contact match.

      To better view the mesh on the contact region, click the **Wireframe** button on the **Graphics** toolbar.

7. If desired, merge the nodes by dragging the contact matches into a **Node Merge Group**.

   The mesh nodes are matched during the contact match operation, but they are not merged. If you want conformal mesh, you should merge the nodes.

   For more information about merging nodes, see Node Merge (p. 508).

Manually Applying Contact Matches

1. If necessary, insert a Contact Match Group folder (p. 485).
2. Do one of the following:

   - Right-click the **Contact Match Group** folder and select **Insert** > **Contact Match**.
   - Highlight the **Contact Match Group** folder, and then click the **Contact Match** option on the **Mesh Edit** toolbar.

3. In the Details view, verify the properties.

   Primary Geometry, Secondary Geometry

   "Primary" indicates the topology that will be captured after the operation is complete. That is, it is the topology to which the nodes in the secondary topologies are matched. The primary geometry can be one or more faces.

   "Secondary" indicates the topology that will be matched to the primary during the operation. The secondary geometry can be one or more faces.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tolerance

The **Tolerance** here has a similar meaning to the **Tolerance Value** global connection setting, and is represented as a transparent sphere.

Setting the correct tolerance can be very important, and in some cases may require some speculation and experimentation. You can adjust the tolerance after the contact matches are generated by selecting the contact matches and then changing the **Tolerance Value**.

For details on setting the tolerance, see How Tolerances Affect Contact Matches (p. 498)

4. Generate the contact matches by doing one of the following:

To...	Do this...
Generate all contact matches	Right-click the **Mesh Edit** folder and select **Generate**.
Generate contact matches for a contact match group	Right-click the **Contact Match Group** object and select **Generate**.

If the base mesh is out-of-date, it is regenerated. The nodes are matched between the primary and secondary geometries, and a message appears displaying the number of node pairs that were matched.

If the normals between the primary and secondary faces are misaligned, some contact matches may not be generated.

5. If necessary, review the contact matches:

   a. Select one or more **Contact Match** or **Contact Match Group** objects, right-click, and select **Create Named Selections**.

      A named selection is created for each contact match you selected. If you selected a contact match group, a named selection is created for each contact match within the group. Each named selection is automatically given the same name as the contact match from which you created it.

   b. Click a named selection to view the mesh for the contact match.

      To better view the mesh on the contact region, click the **Wireframe** button on the **Graphics** toolbar.

6. If desired, merge the nodes by dragging the contact matches into a **Node Merge Group**.

   The mesh nodes are matched during the contact match operation, but they are not merged. If you want conformal mesh, you should merge the nodes.

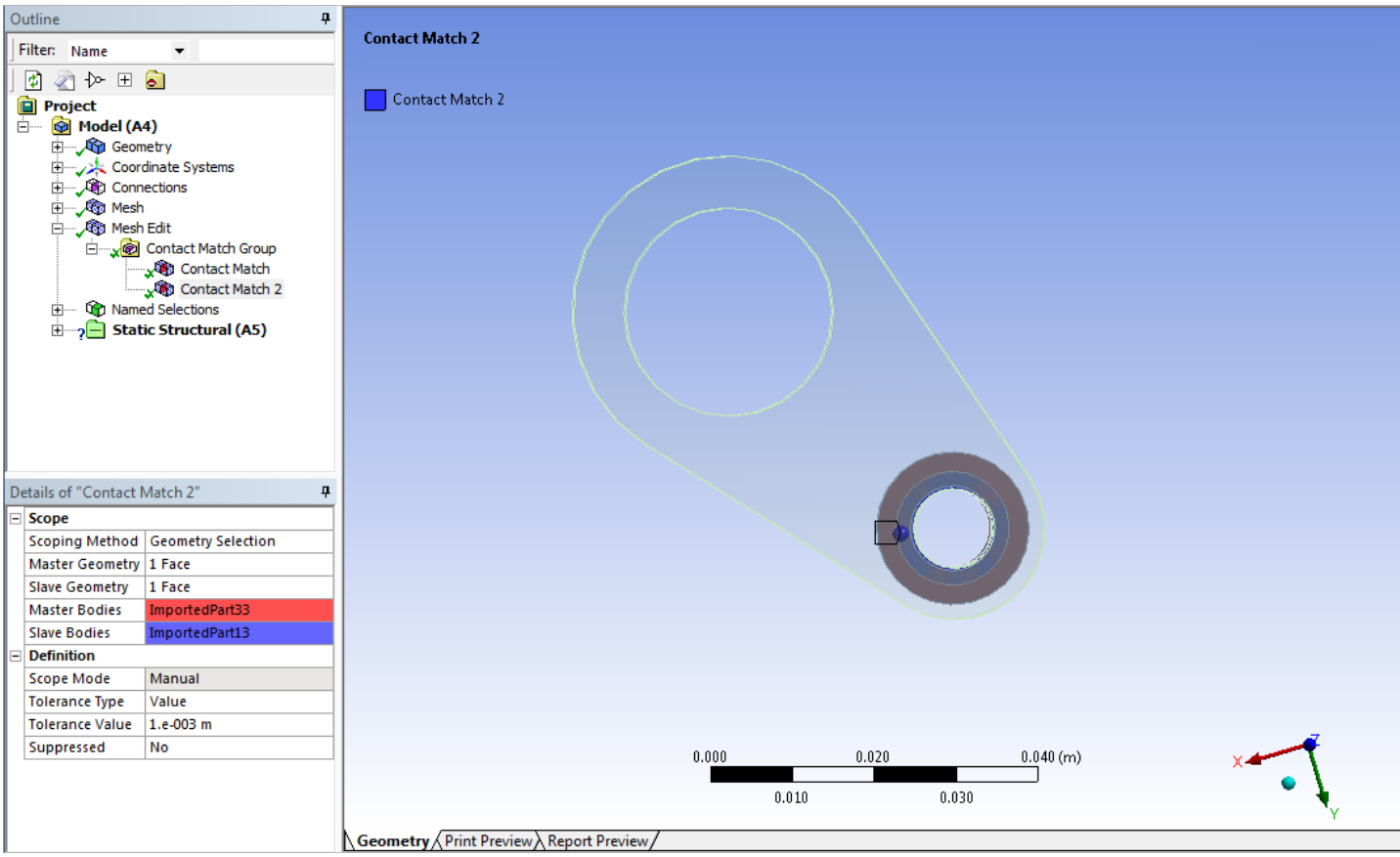
   For more information about merging nodes, see Node Merge (p. 508).

Displaying Multiple Views of Contact Matches

To more closely inspect a contact match, you can display the "Primary" and "Secondary" bodies in auxiliary windows next to the Geometry window.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 198: Viewing the "Primary" and "Secondary" Bodies in Auxiliary Windows



1. Select a contact match.
2. On the **Mesh Edit** toolbar, click the **Body Views** button.

   The "Primary" and "Secondary" bodies are displayed in auxiliary windows.
3. For closer inspection of contact matches, click the **Show Mesh** button on the Graphics Options toolbar.
4. If desired, synchronize the views between the Geometry window and the auxiliary windows by clicking the **Sync Views** button on the **Mesh Edit** toolbar.

   By synchronizing the views, any change in the Geometry window will be reflected in the auxiliary windows.

Troubleshooting Failed Contact Matches

Diagnosing Failed Contact Matches

If a contact match fails, the Contact Match object may be in the "Ignored" state in the Tree Outline ( ), or a warning or error message might be displayed.

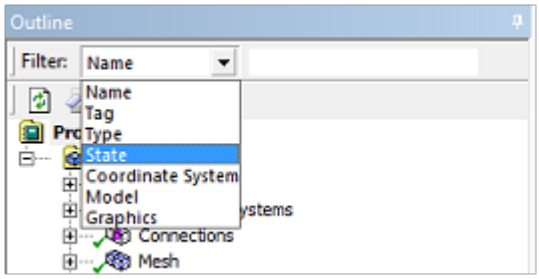
For a description of the various contact match states, see Understanding Mesh Connection and Contact Match States (p. 584).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To diagnose a failed contact match:

1. If necessary, use the **Filter** to identify any contact matches that are in the "Ignored" state.

   a. In the Outline, click the **Filter** drop-down menu and select **State**.

   b. In the adjacent drop-down menu, select **Ignored**.

2. If the contact match is in the "Ignored" state, then display the associated error message:

   a. In the Tree Outline, select the **Contact Match** object.

   b. In the Details view, click **Click to Display** to display the associated error message.

   You can use the **Filter** to identify contact matches that are in the "Ignored" state. However, if a contact match is in an error state, it cannot be filtered in the tree.

3. For each error message, take corrective action:

If the error message...	**Do this...**
Provides "Problematic Geometry" information	1. Right-click the error message and select **Show Problematic Geometry**. The geometry that is responsible for the message is highlighted in the Geometry window. Any error message that is related to a specific contact match will be associated with the secondary geometry. 2. Right-click the problematic bodies and select **Go To** > **Contact Matches for Selected Bodies**. All contact matches that are attached to the problematic geometry are highlighted. 3. Review the tolerances and mesh sizes associated with the highlighted contact matches.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the error message...	Do this...
	For more information, see Correcting Contact Match Errors (p. 507).
Provides "Go to Body "information	1. Right-click the error message and select **Go to Body**. The object that is responsible for the message is highlighted in the Details view. 2. Review the tolerances and mesh sizes associated with the highlighted body or bodies. For more information, see Correcting Contact Match Errors (p. 507). This action highlights all contact matches attached to the problematic geometry.
Provides "Go to Object" information	1. Right-click the message and select **Go To Object**. The corresponding **Contact Match** object that is at issue becomes active in the tree. 2. Verify that all of the associated properties are properly defined.

Correcting Contact Match Errors

Contact match errors may occur due to tolerance issues or mesh size differences. Automatic contact match detection (p. 502) can result in undesirable contact matches. Therefore, if you used this method to apply the contact matches, you should verify that the failed contact match is necessary before correcting the error.

1. If there is a tolerance error, verify that the **Tolerance Value** is correct.

   If the distance between the "Primary" and "Secondary" bodies is greater than the **Tolerance Value**, then increase the tolerance.

2. Create a named selection for the failed contact matches that should have the same mesh size:

   a. Select all of the failed contact matches, right-click, and select **Create Named Selection**.

      A named selection is created for each failed contact match.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Select the named selections that should have similar mesh size, right-click, and select **Merge Selected Named Selections**.

When choosing named selections, you should consider the location of the contact matches in the assembly, and the mesh size in the regions surrounding those contact matches.

3. Define the mesh size for the faces in the named selection:

a. Right-click the **Mesh** object and select **Insert** > **Sizing**.

b. In the Details view, set the **Scoping Method** to **Named Selection**, and then select the named selection that you created.

c. Define the mesh sizing by modifying the settings in the **Definition** group as needed.

For more information about the face sizing settings, see Descriptions of Local Sizing Control Options (p. 302).

4. Right-click the **Mesh** object and select **Generate**.

The base mesh is regenerated with the face sizing control.

5. Right-click the **Mesh Edit** object and select **Generate**.

The contact matches are regenerated.

Node Merge

Node Merge is a mesh editing tool that enables you to merge mesh nodes within a specified tolerance, making the mesh conformal across bodies, parts, and assemblies. Node merge can be used in conjunction with Node Move (p. 512) to remove large gaps in meshes without degrading mesh quality significantly. Node Merge can be performed on solid, sheet, and line bodies.

Scoping

Similar to Mesh Connections, Node merges are performed on mesh nodes. You must, however, explicitly specify the tolerance for node merges. Node merges can be face-to-face, face-to-edge, or edge-to-edge. Node Merge is a post-mesh operation, performed after the base mesh is generated. The base mesh is then stored so that if you change a node merge, only local re-meshing is required to clean up the neighboring mesh.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Requirements

For a Node merge to be successful, the mesh must be of similar size, and the number of mesh nodes to be merged on both topology entities need to be equal. If nodes on the primary and secondary do not have a 1–1 correspondence, then the resulting mesh after node merge can have holes.

Note:

Mesh Connections take priority over Node Merge operations. If nodes are connected by mesh connections, then the Node Merge operation will ignore the connection.

Application

Node merge operation supports two methods for connecting the mesh: *Automatic* and *Manual*. They both work similarly, with the exception that in manual node merge the node merge objects are explicitly populated in the tree, but are hidden in automatic mode. Automatic node merge is very useful when working on very large assemblies, as large number of node merge objects can slow down performance. Manual node merge, however, allows you to review individual node merge objects in detail.

A node merge operation is executed as a single operation on the base mesh of the whole model. If you have multiple node merge groups, executing **Generate** on a single group will merge only the nodes in that group, and will return the mesh to the state prior to the node merge operations on the other groups. To connect all of the node merge groups, execute **Generate** at the **Mesh Edit** folder level.

If you have applied any mesh connections or contact matches, you should generate them before applying node merges.

To automatically apply node merges:

1. Right-click the **Model** object in the Tree and choose **Mesh Edit**, or select the **Model** object and choose **Mesh Edit** from the **Model** toolbar.
2. Right-click the **Mesh Edit** object and choose **Insert**>**Node Merge Group**, or choose **Node Merge Group** from the **Mesh Edit** toolbar.
3. Select the **Node Merge Group** and set the **Method** to **Automatic Node Merge** in the Details view (this is the default setting).
4. Set the **Scoping Method** to **Geometry Selection** (the default) or to **Named Selection** and choose the body or bodies to search for node merges.
5. Modify the **Tolerance** settings in the Details view. The **Tolerance** value is used to find which bodies should be connected to which other bodies. For a discussion of Tolerance settings, see Tolerances Used in Mesh Connections (p. 489), although **Snap Boundaries** are not available for Node Merges.
   - **Tolerance Type**: Options include **Slider**, **Value**, and **Use Sheet Thickness**. Bodies in an assembly that were created in a CAD system may not have been placed precisely, resulting in small overlaps or gaps along the connections between bodies. You can account for any imprecision by specifying connection detection tolerance. This tolerance can be specified by value when

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the type is set to **Slider** and **Value**, or sheet thickness of surface bodies when the type is set to **Use Sheet Thickness**.

- **Tolerance Slider**: Appears if **Tolerance Type** is set to **Slider**. To tighten the connection detection, move the slider bar closer to +100 and to loosen the connection detection, move the slider bar closer to -100. A tighter tolerance means that the bodies have to be within a smaller region (of either gap or overlap) to be considered in connection. A looser tolerance will have the opposite effect. Be aware that as you adjust the tolerance, the number of connection pairs could increase or decrease.

- **Tolerance Value**: Appears if **Tolerance Type** is set to **Slider** or **Value**. This field is read-only if the **Tolerance Type** is set to **Slider** showing the actual tolerance value based on the slider setting. When the **Tolerance Type** is set to **Value**, you can provide an exact distance for the detection tolerance.

  After you provide a greater than zero value for the **Tolerance Value**, a circle appears around the current cursor location. The radius of the circle is a graphical indication of the current **Tolerance Value**. The circle moves with the cursor, and its radius will change when you change the **Tolerance Value** or the **Tolerance Slider**. The circle appropriately adjusts when the model is zoomed in or out.

- **Use Range**: Appears when the **Tolerance Type** property is set to **Slider** or **Value**. Options include **Yes** and **No** (default). If set to **Yes**, you will have the connection detection searches within a range from **Tolerance Value** to **Min Distance Value** inclusive.

- **Min Distance Percentage**: Appears if **Use Range** is set to **Yes**. This is the percentage of the Tolerance Value to determine the Min Distance Value. The default is 10 percent. You can move the slider to adjust the percentage between 1 and 100.

- **Min Distance Value**: Appears if **Use Range** is set to **Yes**. This is a read-only field that displays the value derived from: **Min Distance Value** = **Min Distance Percentage** * **Tolerance Value**/100.

- **Face/Face**: Options include **Yes** and **No** (default). Detects connection between the faces of different bodies. The maximum allowable difference in the normals for which contact is detected is 15 degrees.

- **Face/Edge**: Options include **Yes** and **No** (default). Detects connection between faces and edges of different bodies. Faces are designated as targets and edges are designated as contacts. Saying Yes exposes **Face Angle Tolerance** and **Edge Overlap Tolerance**.

- **Edge/Edge**: Options include **Yes** (default) and **No**. Detects connection between edges of different bodies.

- **Search Across**: This property enables automatic connection detection through the following options:

  – **Bodies** (default)

  – **Parts**: Between bodies of different parts, that is, not between bodies within the same multibody part.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Anywhere**: Detects any connections regardless of where the geometry lies, including different parts. However, if the connections are within the *same body*, this option finds only Face/Face connections, even if the **Face/Edge** setting is turned **On**.

- **Face Angle Tolerance**: Available only if **Face/Edge** is set to **Yes**. For faces that will be excluded from the proximity detection pair, this property defines the minimum angle between the primary face and secondary edge entity, above which the two face pairs will be ignored from proximity detection. The default value is **70°**.

- **Edge Overlap Tolerance**: Available only when **Face/Edge** is set to **Yes**. This tolerance value is the minimum percentage that an edge may overlap the face and is included as a valid proximity detection pair. The default value is **25%**.

6. Generate the node merge by doing one of the following:

To...	Do this...
Generate all node merges	Right-click **Mesh Edit** and choose **Generate**.
Generate node merges for a Node Merge Group	a. For each **Node Merge Group** that should not be generated, right-click the **Node Merge Group** and choose **Suppress**. b. Right-click the **Node Merge Group** and choose **Generate**.

To manually apply Node Merges using the **Node Merge Group**:

1. Right-click the **Model** object in the Tree and choose **Mesh Edit**, or select the **Model** object and choose **Mesh Edit** from the **Model** toolbar.

2. Right-click the **Mesh Edit** object and choose **Insert**>**Node Merge Group**, or choose **Node Merge Group** from the **Mesh Edit** toolbar.

3. Select the **Node Merge Group** and set the **Method** to **Manual Node Merge** in the Details view.

   An additional control, **Group By** is exposed in the Details view. Options for **Group By** include **None**, **Bodies**, and **Parts**, and **Faces**. This property allows you to group the automatically generated connections objects. Setting **Group By** to **Bodies** (default) or to **Parts** means that connection faces and edges that lie on the same bodies or same parts will be included into a single connection object. The **Faces** option is only available if the **Face/Face** or **Face/Edge** controls are set to **Yes**.

   Setting **Group By** to **None** means that the grouping of geometries that lie on the same bodies or same parts will *not* occur. Any connection objects generated will have only one entity scoped to each side (that is, one face or one edge). If there are a large number of source/target faces in a single region. Choosing **None** avoids excessive contact search times in the solver.

4. Right-click the **Node Merge Group** and select **Detect Connections**. The states of the **Node Merge** connection are denoted using the same symbols as are used for **Mesh Connections** (p. 584).

5. Generate the node merge by doing one of the following:

To...	Do this...
Generate all node merges	Right-click **Mesh Edit** and choose **Generate**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To...	Do this...
Generate node merges for a Node Merge Group	a. For each **Node Merge Group** that should not be generated, right-click the **Node Merge Group** and choose **Suppress**. b. Right-click the **Node Merge Group** and choose **Generate**.
Generate individual node merges	Select each node merge that you want to generate, right-click, and choose **Generate**.

To manually apply Node Merges individually:

1. Right-click the **Model** object in the Tree and choose **Mesh Edit**, or select the **Model** object and choose **Mesh Edit** from the **Model** toolbar.
2. Right-click the **Mesh Edit** object and choose **Insert**>**Node Merge**, or choose **Node Merge** from the **Mesh Edit** toolbar.

   The object is added as a child of a new **Node Merge Group**.

   Alternatively, you can select a **Node Merge Group** and set the **Method** to **Manual Node Merge** in the Details view. Then, right-click the **Node Merge Group** and choose **Insert**>**Node Merge**.
3. Set the **Scoping Method** to **Geometry Selection** (the default) or to **Named Selection** to determine how you will select the **Primary Geometry** and **Secondary Geometry**.
4. Select the **Node Merge** object and choose the **Primary** and **Secondary** Geometries:
   - "Primary" indicates the topology that will be captured after the operation is complete.
   - "Secondary" indicates the topology that will be pinched out during the operation.
5. Set the controls under **Definition** as desired.
6. Generate the node merge by doing one of the following:

To...	Do this...
Generate all node merges	Right-click **Mesh Edit** and choose **Generate**.
Generate the individual node merge	Right-click the node merge and choose **Generate**.

Note:

The mesher does not check the quality of the mesh after a Node Merge is performed. Therefore, you should perform a manual quality check any time you insert a Node Merge.

Node Move

The **Node Move** feature enables you to select and then manually move a specific node on the mesh to improve the local mesh quality.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Requirements

This feature has the following topological requirements:

- Node movement is restricted to the target part. That is, nodes cannot be moved outside of the part.
- Face/Edge nodes can be moved along the corresponding face/edge only.
- The application locks corner nodes to the underlying vertex.
- Moving a node on a solved analysis causes the solution data to become obsolete. However, the state does not become obsolete for other objects that depend on the mesh (for example, imported loading conditions, Element Orientations). If your node movement changes affect other objects, you will need to manually clear and refresh these objects to apply the new changes.
- The **Node Move** feature is disabled if you are working with Section Planes.

Application

The **Node Move** object is a child object of Mesh Edit. It is inserted into the tree by selecting the **Node Move** button on the Mesh Edit toolbar or by selecting the **Mesh Edit** object, right-clicking, and selecting **Insert**>**Node Move**.

To use this feature, you need to generate the mesh on your model. This can be done before or after you have inserted the object into the tree. Once generated, the node selection options are assigned automatically. The Select Type is set to Select Mesh and Vertex is the required picking tool. Moving the cursor across the mesh of your model displays the available mesh nodes. You may then select and move nodes.

In addition, once the object is placed in the tree, the **Node Move** toolbar displays.

Node Move toolbar options include the following:

Undo Last

Cancels the last node movement performed on the mesh. Operations that change the original mesh may make this option unavailable.

Undo All

Cancels all of the node movements that you have made to the mesh. Operations that change the original mesh may make this option unavailable.

Probe, Max, and Min

These are annotation options. Selecting the **Max** and/or **Min** buttons displays the maximum and minimum values for mesh criteria (Element Quality, Jacobian Ratio, etc.) that you have selected. The **Probe** feature is also criteria-based. You place a **Probe** on a point on the model to display an annotation on that point. **Probe** annotations show the mesh criterion-based value at the loc-

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ation of the cursor. When created, probe annotations do not trigger the database to be marked for the file needing to be saved (that is, you will not be prompted to save). Be sure to issue a save if you wish to retain these newly created probe annotations in the database. These options are not visible if the **Mesh** object **Display Style** property is set to the default setting, **Body Color**.

Edges Options

This drop-down menu provides options to change the display of your model, including:

- **No Wireframe**: displays a basic picture of the body.
- **Show Elements**: displays element outlines.

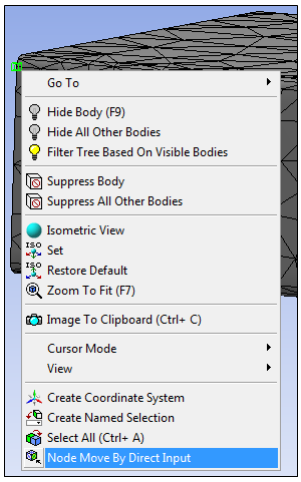
These options are the same options that are available on the Meshing Context Toolbar.

Free Mode

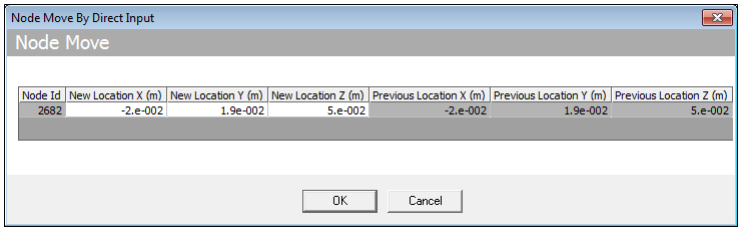
You can depress the **F4** key while you are moving a node to remove certain movement restrictions. This mode enables you to move an edge, vertex, or face node anywhere on a given face.

Direct Node Movement

You can also move a node by manually selecting the desired node, right-clicking, and then selecting the option **Node Move By Direct Input**.



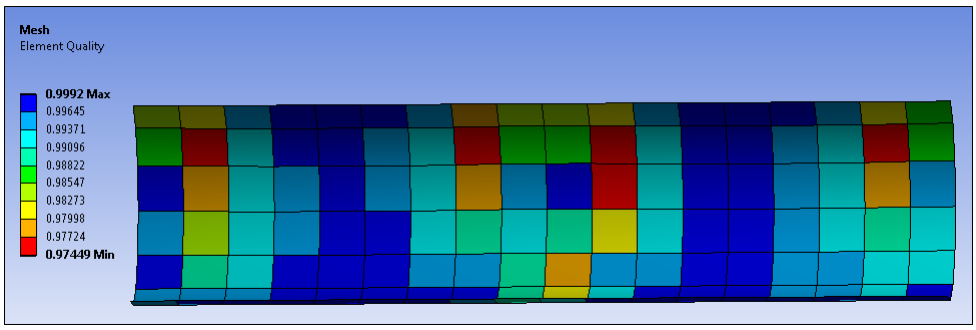
This option displays an entry window where you can manually change the X, Y, Z location of the node.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element Quality Display

To enhance the presentation of your selections and movements, make sure that you set the **Display Style** property of the Mesh to **Element Quality**. An example of this setting is provided below. The exact display is provided on the **Node Move** object (and the legend title is also **Mesh**).

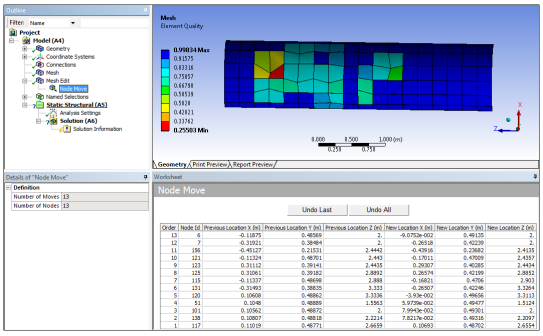


Worksheet

The **Worksheet** works in combination with the **Node Move** feature. The **Worksheet** records all of the node movements performed. In addition, and as illustrated below, the **Worksheet** provides information about the selection order of node movements, node numbers, coordinate-based location information (previous and new), as well as the options to undo the last movement or all of the movements that were made.

Note:

If you update your mesh (**Mesh** object>**Update**), the application maintains your movements in the **Worksheet** until the mesh is cleared (zero nodes) using the RMB option **Clear Generated Data** or you refresh data from the CAD source.



The illustration above also shows the **Worksheet** docked in the lower portion of the screen so that you can easily see all of your node movement information while also being able to see the model. Docking is possible with the docking tool shown here. This tool displays when you drag a window's title bar. Hovering the window over one of the blue arrows highlights the arrow. Releasing the mouse button docks the window in that screen location. It may also be useful to display the node numbers on your model using the display option **Node Number** available through Annotation Preferences.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

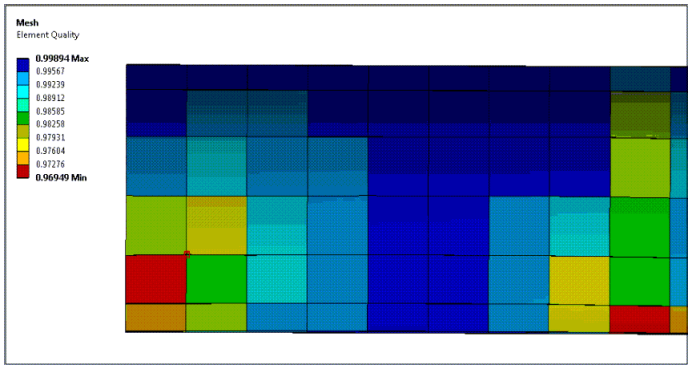
Note:

Node move operations are not persistent. The **Worksheet** view gives a history of what has been done based on the current Node IDs, but new Node IDs are created when the model is re-meshed. Therefore, the history recorded in the Worksheet is rendered out-of-date after re-meshing. For this reason, Node Move operations should be used sparingly, and only to fix small issues.

See the Windows Management Help section for additional information.

Example

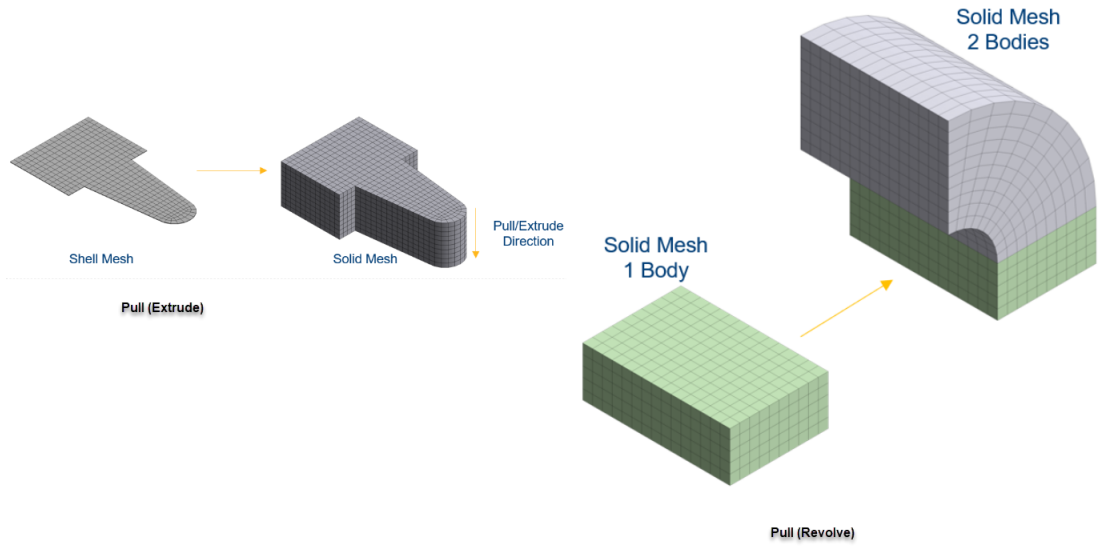
The following is an animated example of the use of the feature in tandem with the **Element Quality** display. View online if you are reading the PDF version of the help. Interface names and other components shown in the demos may differ from those in the released product.



Pull

Pull enables you to extrude or revolve element faces, to generate solid mesh. **Pull** can be applied directly to scoped element faces,or geometric faces or bodies (only shell body) from a surface or solid mesh. **Pull** generates volume of solid elements for the specified height or revolved volume from the source mesh along the specified coordinated system or the face normal vector. **Pull** allows you to create surface coating on solid bodies, element faces and geometric faces belonging to the solid bodies to generate shell mesh.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

Pull cannot be accessed through **Mesh** only system.

Pull creates linear elements if the base mesh is having linear elements and quadratic elements if the base mesh is quadratic. **Pull** mesh supports curvilinear mid nodes for quadratic elements.

On the **Tree** view under **Mesh** object, right-click **Pull > Generate Selected Pull** to create **Pull Part** under the **Geometry** tree. The **Pull** part created is independent of the contents of the **Pull** mesh. When you **Extrude** or **Revolve** object from mesh data, the base mesh consists of solid element faces or surface elements. Each connected set of elements forms a body. Each body has a top, bottom and side face. Two edges are created bounding the top and bottom face. These edges are called ring edges and they do not have vertices.

When you **Extrude** or **Revolve** object from CAD topology, bodies of the **Pull** part match the bodies of the base mesh. Bottom faces of the pull bodies match those of base mesh and top face match the corresponding face of the base mesh. A side face is created for each edge of the CAD topology. Edges and vertices of **Pull** bodies are created to form watertight solids.

The exceptions to the above rule are the following:

- When the mesh body having multiple parts are meshed using **Batch Connections** or **Automatic (PrimeMesh) Method**, a **Pull** part is created for a **Pull** feature.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When the **Pull** feature is Extrude up to target and the target is such that some of the base mesh must be removed to perform extrusion, you must apply the rules of extrusion of the mesh objects. Hence, the pull sides are not defined by the original CAD topology edges.

Note:

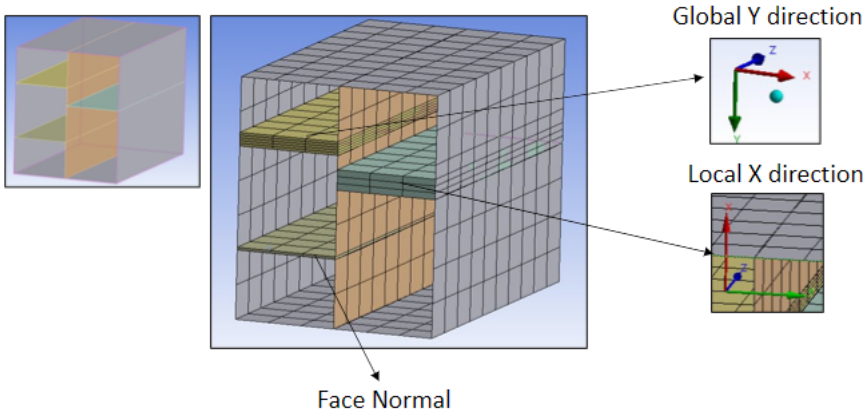
- For **Pull** defined by CAD topology, if the **Pull** mesh is cleared or CAD model is refreshed, all loads and boundary conditions behavior follow the one defined by the CAD topology entities.
- For **Pull** defined by mesh elements or element faces, if **Pull** mesh is cleared or CAD model is refreshed, all loads and boundary conditions need to be redefined.

Extrude

Extrude scopes shell bodies, elements belonging to shell bodies, geometric faces of shell or solid bodies, element faces of solid bodies.

To access **Extrude**,

1. Right-click **Mesh Object > Insert > Mesh Edit**.
2. Right-click **Mesh Edit > Insert > Pull > Extrude**.
3. Click **Pull (Extrude)** on the Tree view and click **Generate** to perform extrude mesh.



The Details view displays the **Pull (Extrude)** options :

Scope

Scoping Method: Allows you to scope the model based on your selection. There are two options. They are **Geometry Selection** and **Named Selection**.

Geometry Selection: Allows you to scope elements, element faces, geometric faces and sheet bodies for **Extrude**.

Named Selection: Allows you to select the element facets and elements belonging to sheet and solid bodies for **Extrude**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Definition

Method: Displays the selected method of **Pull**. This is a read-only field.

Extrude: Allows you to generate solid elements for specified number of layers and height

Height: Allows you to specify the total height for the volume of solid elements. **Height** allows only positive values.

Number of Layers: Allows you to specify the number of layers to be used for **Extrude**.

Extruded By: Allows you to specify the direction of extrusion. The available options are **Use Coordinate System**, **Face Normal** and **Face Normal (Reversed).**

- **Use Coordinate System**: Allows you to generate elements in the specified coordinate system. When you select **Use Coordinate System** option in **Extruded By**, the **Coordinate System** and **Use Coordinate System** fields appear. The **Coordinate System** allows you to select the **Global Coordinate System** by default. The **Use Coordinate System** allows you to select any of the XYZ coordinate axis.
- **Face Normal**: Allows you to generate elements along the face normal vector.
- **Face Normal Reversed**: Allows you to generate elements along the opposite direction of the face normal vector.

Extrude Upto: Allows you to generate elements up to the specified target. The target can be faces from solids or sheet bodies and multiple faces with sharing geometric edge.

Element Option: Allows you to select the element type for the solver to use. The available options are **Solid** and **Solid Shell**. The default value is **Solid**.

Suppressed: Allows you to suppress the selected entities. You can select **Yes** to suppress the selected entities and **No** to unsuppress the selected entities. The default value is **No**.

Part Properties

Material: Allows you to select the material of your choice. You can also select a different material from the parent body material.

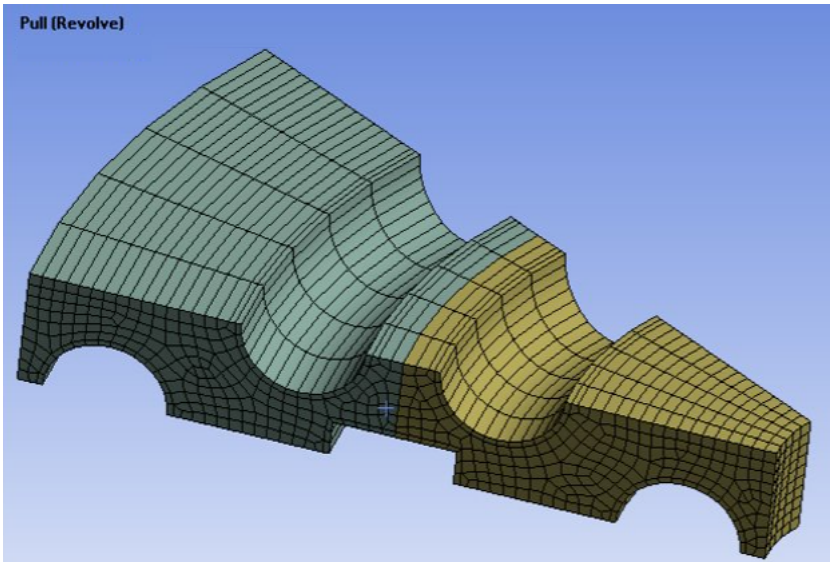
Revolve

Revolve scopes shell bodies, elements belonging to shell bodies, geometric faces of shell or solid bodies, element faces of solid bodies.

To access **Revolve**,

1. Right-click **Mesh Object > Insert > Mesh Edit**.
2. Right-click **Mesh Edit > Insert > Pull > Revolve**.
3. Click **Pull (Revolve)** on the Tree view and click **Generate** to generate pull elements along the revolution angle.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Details view displays the **Revolve** options:

Scope

Scoping Method: Allows you to scope the model based on your selection. There are two options. They are **Geometry Selection** and **Named Selection**.

Geometry Selection: Allows you to scope elements, element faces, geometric faces and sheet bodies for **Revolve**.

Named Selection: Allows you to select the element facets and elements belonging to sheet and solid bodies for **Revolve**.

Definition

Method: Displays the selected method of **Pull**. **Revolve** allows you to generate solid elements for the specified number of layers and revolution angle along the edge coordinate system.

Revolution Angle: Allows you to specify angle of revolution for the solid elements. Here, **Pull (Revolve)** automatically merges the nodes of the first layer and last layer for 360 degrees revolution. **Revolution Angle** allows only positive values.

Number of Layers: Allows you to specify the number of layers to be used for **Revolve**.

Coordinate System: Allows you to select the **Global Coordinate System** by default. This option supports only **Cartesian System**.

Axis of Revolution: Allows you to select the coordinate axis along which the selected elements can be revolved. The available options are **X Axis**, **Y Axis**, **Z Axis**.

Suppressed: Allows you to suppress the selected entities. You can select **Yes** to suppress the selected entities and **No** to unsuppress the selected entities. The default value is **No**.

Part Properties

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

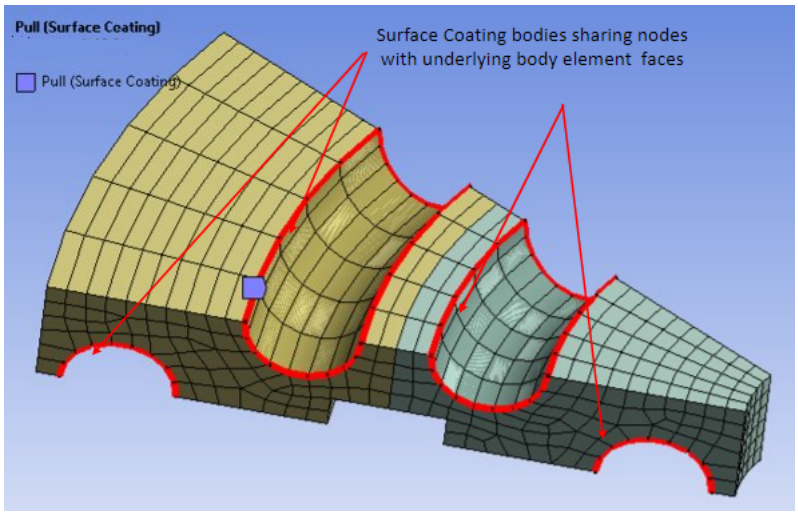
Material: Allows you to select the material of your choice. You can also select a different material from the parent body material.

Surface Coating

Surface Coating allows you to scope solid element faces. For mesh body, the bodies are created according to the bodies that own solid element faces. Each element face creates a topological face. Also, creates edges and vertices to form the topology. When you perform **Pull** on solid bodies with boundaries, bodies are created for each body that owns faces in the surface coating body and creates faces, edges, and vertices to match the topology of the base mesh.

To access **Surface Coating**,

1. Right-click **Mesh Object** > **Insert** > **Mesh Edit**.
2. Right-click **Mesh Edit** > **Insert** > **Pull** > **Surface Coating**.
3. Click **Pull (Surface Coating)** on the Tree view and click **Generate** to generate surface elements on the solid.



The Details view displays the **Surface Coating** options:

Scope

Scoping Method: Allows you to scope the model based on your selection. There are two options. They are **Geometry Selection** and **Named Selection**.

Geometry Selection: Allows you to scope element faces, geometric faces and solid bodies for **Surface Coating**.

Named Selection: Allows you to scope element faces, geometric faces and solid bodies for **Surface Coating**.

Definition

Method: Displays the selected method of **Pull**. **Surface Coating** allows you to create shell elements coating for the outer layer of the 3D objects. You can scope solid element faces, CAD faces and solid bodies through **Geometry Selection** and **Named Selection**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Line Coating: Allows you to produce line elements coating the edges of the 2D axisymmetric objects only. When you scope an edge using **Surface Coating**, the **Method** automatically changes to **Line Coating**. When you generate **Pull**, line element are created under the **Pull** part in the **Geometry** tree. **Pull (Surface Coating)** supports multiple surface coating on the same body or face and on element faces.

Suppressed: Allows you to suppress the selected entities. You can select **Yes** to suppress the selected entities and **No** to remove suppression from the selected entities. The default value is **No**.

Part Properties

Material: Allows you to select the material of your choice. You can also select a different material from the parent body material.

Nonlinear Effects: Choose whether to include nonlinear effects. Options include **Yes** (default) and **No**.

Thermal Strain Effects: Choose whether to include thermal strain effects. Options include **Yes** (default) and **No**.

Stiffness Option: Allows you to provide the stiffness behavior of the shell body created by surface coating. The available options are **Stress Evaluation Only**, **Membrane Only**, **Membrane and Bending**.

Thickness: Allows you to specify the thickness for surface coating. The **Thickness** option is available only when the **Stiffness Option** is set to **Membrane Only** or **Membrane and Bending**.

Limitations

- **Pull** cannot detect the intersection between the existing mesh and the created **Pull** or intersection between the two **Pull** objects. **Pull** can only detect self-intersection within a single **Pull**. Then, it provides an error message without creating **Pull** mesh.
- If **Pull** is scoped to two element faces, geometric faces or disconnected bodies from disjoint bodies in the same model for performing **Pull** operation, two **Pull** bodies are created in the generated **Pull** part under **Geometry** on the **Tree** view.
- When **Pull** is performed on a highly curved surface, the generated pulled elements may converge or intersect. Such elements are excluded from extrusion when **Extrude Upto** is set to **Yes**. When **Extrude Upto** is set to **No**, the **Pull** operation is aborted providing an error message.
- When a solid object is added through Construction Geometry after generating the **Pull** object, you cannot generate mesh on the solid body. You must insert and mesh all solid objects before generating the first **Pull** object.
- When you try to **Pull (Extrude)** using solid element faces, only single side face (perpendicular to extrude direction) is created instead of creating multiple faces as with the extruding topological entity.
- When you select multiple **Pull** objects for performing **Pull** operations, you do not have the option to generate pull. However, to generate multiple **Pull**, you can select the **Pull** objects then right-click **Mesh Edit > Generate**.
- When you suppress a **Pull** part under the **Geometry** tree, the **Pull** object under the **Mesh Edit** folder is not suppressed. It remains unchanged.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When you apply **Pull** on single bodies, **Pull** creates non conformal mesh. That is, nodes of the interfaces from parent body and pull body are not connected. You should define the manual contact to resolve the issue.
- When you have a **Pull** object generated, the **Node Merge** option is not available when you try to insert it by right-clicking **Mesh Edit > Insert > Node Merge**. You can now select **Beta Options** from **Tools > Options > Appearance** to access the **Node Merge** in **Pull**.
- **Pull** does not support cylindrical coordinate system.
- **Pull (Surface Coating)** does not support rigid bodies.
- **Refresh** is not supported when you scope mesh elements or element faces for **Pull** generation.
- For geometries created using the **Mesh Pull** feature and exported as a **Part Manager Database** file, the Ansys Mechanical does not currently support re-meshing the part using normal mesh methods when you import the .pmdb file into a new **Mechanical** session.
- When you perform **Pull** on a model having both solid and shell bodies, **Pull** scopes element faces of both solid and shell bodies without providing an error message.
- When same elements are scoped through two different Name Selection, multiple **Pull** can be used to generate the **Pull** for the same elements without providing any warning message.
- When you apply **Pull** on a model having multibody part with shared topology, it creates conformal mesh and shares topology between the bodies. Hence, if you do not want to share topology, you must transfer the model to Ansys Mechanical as multi-part assemblies.
- For **Extrude** and **Revolve**, **Pull** automatically suppresses the base body and hides it in the graphics when:
  - Sheet body is scoped
  - All the geometric faces of a sheet or solid body is scoped
  - All elements of a sheet body are scoped.
- When a single **Pull** is applied on the multipart assembly, the **Pull** creates non conformal mesh and may have self-intersection causing **Pull** failure. You may define separate **Pull** for each part of the multipart assembly for successful pull generation.
- **Line Coating** does not support the interface between the 2D and 3D surfaces.
- When you have meshed **Pull (Surface Coating)** objects on the **Tree** view and you insert a **Pull (Extrude)** or **Pull (Revolve)** object, the state change of **Pull (Extrude)** and **Pull (Revolve)** affect the state of **Pull (Surface Coating)**.
- **Pull** allows you to mesh only multibody parts with same order mesh. If mixed order mesh is scoped, then **Pull** generation gives error due to dropped nodes. To avoid this, **Pull** should have either linear or quadratic mesh as input and not both in a single **Pull** control.
- When a generated **Pull** in a system is transferred to another system and both the systems are updated and then if you suppress the **Pull** in the first system and update both the systems, the updated **Pull** in the first system is not transferred to the **Pull** in the second system.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When **Pull (Extrude, Revolve, Surface Coating)** is duplicated, the scoping of the original **Pull (Extrude, Revolve, Surface Coating)** is not retained for the duplicated **Pull (Surface Coating)**.
- When there are multiple **Pull (Surface coating)** scoped to same body under the **Mesh** object on the **Tree** view,
  - Clearing mesh on the first **Pull (Surface Coating)**, automatically clears the mesh on the second **Pull (Surface Coating)** and vice versa. Also, when this happens the **Pull (Surface Coating)** Part under the Geometry on the Tree view is not cleared for the second Pull.
  - Suppressing the second **Pull (Surface Coating)**, affects the state of first **Pull (Surface Coating)** and vice versa.
  - Generating a second **Pull (Surface Coating)**, clears and regenerates the first **Pull (Surface Coating)**.
- When you scope multibody parts such as solid and shell bodies sharing the edge which have mixed order mesh in **Pull**, self-intersections may occur when the **Pull** is generated along the face normal or face normal reversed. Hence, same order mesh should be scoped for the **Pull** and extrude direction should be specified using the coordinate system.
- When bodies scoped for **Pull** are suppressed, the **Pull** mesh goes out of state but the **Pull** part remains on the **Geometry** tree.

Common Display Features

This section describes some common display features:

Hiding or Suppressing Bodies

Hiding or Showing Faces

Creating Section Planes

Hiding or Suppressing Bodies

For a quick way to hide bodies (that is, turn body viewing off) or suppress bodies (that is, turn body viewing off and remove the bodies from further treatment in the analysis), select the bodies in the tree or in the **Geometry** window (choose the **Body** select mode, either from the toolbar or by a right-click in the **Geometry** window). Then right-click and choose **Hide Body** or **Suppress Body** from the context menu. Choose **Show Body**, **Show All Bodies**, **Unsuppress Body**, or **Unsuppress All Bodies** to reverse the states.

The following options are also available:

- **Hide All Other Bodies**: show only selected bodies.
- **Hide or Show**: contains menu options to hide specific body types. Based on the body types of your model, options include: **Solid Bodies**, **Surface Bodies**, and **Line Bodies**.
- **Invert Visibility**: inverts the visibility of hidden bodies versus those that are visible. When selected, all hidden bodies become visible and all visible bodies become hidden.
- **Suppress All Other Bodies**: enables you to unsuppress only selected bodies.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Invert Suppressed Body Set**: enables you to reverse the suppression state of all bodies (unsuppressed bodies become suppressed and suppressed bodies become unsuppressed).

Note:

- If another model level object, such as a Remote Point, Joint, or Contact Region, is scoped to a Body that becomes Suppressed, that object also becomes suppressed until it is re-scoped or the body is Unsuppressed.
- Results from hidden bodies are used in the formulation of the maximum and minimum values in the contour legend and in the Details View.
- Results from suppressed bodies are suppressed and are **not** used in the formulation of maximum and minimum values.
- When you suppress or unsuppress bodies, the state of the mesh remains the same.

Hiding or Showing Faces

You can hide selected faces on a model such that you are able to see inside the model. This feature is especially useful for bodies with interior cavities, such as engine blocks. To use the feature, first select faces on the model that you want to hide, then right-click anywhere in the **Geometry** window and choose **Hide Face(s)** in the context menu. This menu choice is only available if you have already selected faces.

Choose **Show Hidden Face(s)** from the context menu to restore the visibility of faces previously hidden using **Hide Face(s)**. The **Show Hidden Face(s)** menu choice is only available if there are hidden faces from choosing **Hide Face(s)**. It *cannot* be used to restore the visibility of faces previously hidden by setting **Visible** to **No** in the Details view of a Named Selection object.

Note:

The selected faces will appear hidden only when you view the geometry. The feature is not applicable to mesh displays or result displays.

Creating Section Planes

For viewing purposes, you can use the **Create Section Plane** option to slice the graphical image of your model based on a predefined coordinate system. For details, refer to Creating Coordinate-Based Section Planes in the Mechanical help.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Meshing: Ease of Use Features

The features described in this section are intended to assist you in meshing.

Updating the Mesh Cell State
Generating Mesh
Previewing Surface Mesh
Previewing Source and Target Mesh
Previewing Inflation
Previewing Weld Geometry
Previewing Weld Mesh
Previewing Quad Layer
Showing Program Controlled Inflation Surfaces
Showing Sweepable Bodies
Showing Problematic Geometry
Showing Problematic Location
Showing Problematic Elements
Showing Elements that Do Not Meet the Target Metric
Showing Removable Loops
Showing Suppressible Features
Displaying Element Normals on Mesh
Inspecting Large Meshes Using Named Selections
Generating Multiple Mesh Controls from a Template
Clearing Generated Data
Showing Missing Tessellations
Showing Mappable Faces
Grouping Mesh Objects By Type

Updating the Mesh Cell State

In contrast to the **Generate Mesh** (p. 528) feature (which only produces the mesh), the **Update** feature determines whether the geometry needs to be updated, refreshes the geometry if necessary, generates the mesh if necessary, and also writes the output data for any connected cells:

- The **Generate Mesh** feature is useful when you are investigating the impact of different settings on the mesh but you are not ready to export the mesh files.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Update** feature is useful if you make a connection in the Ansys Workbench Project Schematic from a Mesh cell to a system that requires a new type of output data (for example, if you make a connection from the Mesh cell to a Fluid Flow (CFX) or Fluid Flow (Fluent) analysis system). In such cases, the Mesh cell will go out-of-date. To bring the Mesh cell up-to-date, you can perform an **Update** on the Mesh cell from within the Project Schematic, or follow the procedure below from within the Meshing application.

To update the Mesh cell:

1. Select the Mesh object or any mesh control object.
2. Right-click to display the context menu.
3. Select **Update** in the menu.

Note:

As an alternative to steps 2 and 3, you can click the **Update** button on the **Mesh** toolbar.

Generating Mesh

The **Generate Mesh** operation uses all defined meshing controls as input to generate a mesh. **Generate Mesh** operates only on active objects, meaning that if bodies or controls are suppressed, they are ignored by the meshing operation. You can generate mesh on the entire (active) model, or selectively on (active) parts and/or bodies. This includes single body parts, multibody parts, individual bodies, or multiple selected bodies across different parts or within the same part.

Note:

- Selecting **Generate Mesh** generates a mesh based on the current mesh settings. It does not write the output data for any connected cells (downstream systems). **Generate Mesh** is useful when you are investigating the impact of different settings on the mesh but you are not ready to export the mesh files. Refer to Updating the Mesh Cell State (p. 527) for related information.
- Using selective meshing, you can selectively pick bodies and mesh them incrementally. After meshing a body, you can mesh the whole part or assembly or continue meshing individual bodies. Refer to Selective Meshing (p. 444) for additional information.

Monitoring the Meshing Process

The **Ansys Workbench Mesh Status** dialog box contains a **Highlight** check box that you can use to control whether the topology that is currently being processed by the mesher is highlighted in the **Geometry** window, which may help with troubleshooting (p. 580).

You can enable and disable the **Highlight** check box during the meshing process. Meshing performance should be similar regardless of whether topology highlighting is enabled, but it may be less distracting to disable it. If topology highlighting is enabled and you stop the meshing process, the highlighted topology is selected for you automatically.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This topology highlighting is not supported for the **Patch Independent Tetra** or **MultiZone** mesh methods is being used. For information about how to set the default for topology highlighting, refer to Meshing Options on the Options Dialog Box (p. 389).

Suppressing and Unsuppressing Bodies in a Model

When there is a combination of suppressed and unsuppressed (active) bodies in a model, the Meshing application meshes only the active bodies. This is true regardless of mesh method. In addition, all influence of the suppressed bodies on neighboring bodies and their meshes is suppressed. For example, if a size control is applied to a suppressed body, the size control will not affect that body, nor will it influence neighboring bodies (in general, if a size control is assigned to a suppressed body, that control is also suppressed unless it is also attached to other active bodies). Refer to Selective Meshing (p. 444) for additional information.

To generate the mesh for all active bodies:

1. Select the Mesh object or any mesh control object.
2. Right-click to display the context menu, or choose the **Mesh** drop-down menu from the toolbar.
3. Select **Generate Mesh** in the menu.

   All active bodies are meshed. If the model includes multiple parts, they are meshed in parallel. The **Ansys Workbench Mesh Status** dialog box appears, displaying the meshing progress and highlighting each entity as it is meshed.

   After the mesh has been generated, it is displayed when you select the **Mesh** object or the **Show Mesh** display option.
4. If necessary, stop the meshing process:

   a. In the **Ansys Workbench Mesh Status** dialog box, click **Stop**.

      To see which parts have been meshed, expand the **Geometry** object in the Tree Outline. A green status icon (✔) indicates that the part has been meshed.

   b. To restart the meshing process, right-click the Mesh object or any mesh control object and select **Update**.

      The meshing process resumes and meshes only the parts that have not yet been meshed.

To generate the mesh for individual active bodies:

1. Select the bodies by doing one of the following:

   - In the Tree Outline, select one or more Body objects.
   - Select one or more bodies in the Geometry window.

2. Right-click to display the context menu.
3. Select **Generate Mesh** in the menu.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The bodies that you selected are meshed. If you selected multiple parts, they are meshed in parallel. The **Ansys Workbench Mesh Status** dialog box appears, displaying the meshing progress and highlighting each entity as it is meshed.

After the mesh has been generated, it is displayed when you select the **Mesh** object or the **Show Mesh** display option.

4. If necessary, stop the meshing process:

   a. In the **Ansys Workbench Mesh Status** dialog box, click **Stop**.

      To see which parts have been meshed, expand the **Geometry** object in the Tree Outline. A green status icon (✔) indicates that the part has been meshed.

   b. To restart the meshing process, right-click the Mesh object or any mesh control object and select **Update**.

      The meshing process resumes and meshes only the parts that have not yet been meshed.

To generate the mesh for individual active parts:

1. Select the parts by doing one of the following:

   - In the Tree Outline, select one or more Part objects.
   - In the Geometry window, select one or more parts.

2. Right-click to display the context menu.

3. Select **Generate Mesh** in the menu.

   The parts that you selected are meshed. If you selected multiple parts, they are meshed in parallel. The **Ansys Workbench Mesh Status** dialog box appears, displaying the meshing progress and highlighting each entity as it is meshed.

   After the mesh has been generated, it is displayed when you select the **Mesh** object or the **Show Mesh** display option.

4. If necessary, stop the meshing process:

   a. In the **Ansys Workbench Mesh Status** dialog box, click **Stop**.

      To see which parts have been meshed, expand the **Geometry** object in the Tree Outline. A green status icon (✔) indicates that the part has been meshed.

   b. To restart the meshing process, right-click the Mesh object or any mesh control object and select **Update**.

      The meshing process resumes and meshes only the parts that have not yet been meshed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After successfully generating a mesh, you can view mesh statistics and mesh metric information that you can use to evaluate the mesh quality. For more information, see Statistics Group (p. 198) and Quality Group (p. 120).

To re-mesh:

1. Select the Mesh object.
2. Right-click to display the context menu and select **Clear Generated Data** in the menu.
3. Confirm that you want to clear the mesh by clicking the **Yes** button.
4. Right-click the **Mesh** object to display the context menu again and select **Generate Mesh** in the menu.

Note:

- The order of topological entities is not guaranteed during a CAD source refresh. In cases in which you mesh, refresh, and re-mesh, the mesher may not produce exactly the same mesh if the refresh caused the topological entities to be reordered. As a result of this re-ordering, the mesher meshes the entities in a different order as well, producing a slightly different result.
- When selected from the **Geometry** object in the Tree Outline, the **Generate Mesh** RMB menu option behaves slightly differently than when it is selected from the **Mesh** object in the Tree Outline. Refer to Selective Meshing (p. 444) for details.
- Refer to Meshing: Troubleshooting (p. 579) for tips and strategies for handling problems that may occur during meshing.

Previewing Surface Mesh

You can preview the surface mesh for all unsuppressed parts, individual unsuppressed parts, or individual unsuppressed bodies. This includes single body parts, multibody parts, individual bodies, or multiple selected bodies across different parts or within the same part. You can also export the previewed surface mesh file in Fluent format, as described in Exporting a Previewed Surface Mesh in Fluent Format (p. 533).

Note:

- This feature is not supported for the **Patch Independent** and **MultiZone** mesh method controls. It is also not supported for thin model sweeping (that is, use of the **Sweep** mesh method control with **Src/Trg Selection** set to **Manual Thin** or **Automatic Thin**) .
- When previewing surface mesh on bodies that are being meshed with **Sweep**, not all sizing information is used in the calculation. Previewing surface mesh on such bodies is a crude check to get a general idea whether the surface mesh is appropriate, but the full mesh may look different due to differences in sizing calculations when taking all meshing constraints into account.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Refer to Selective Meshing (p. 444) for general information about selective meshing and limitations related to using the **Preview Surface Mesh** feature with selective meshing.

To preview the surface mesh for all unsuppressed parts:

1. Select the Mesh object or any mesh control object.
2. Right-click to display the context menu, or choose the **Mesh** drop-down menu from the toolbar.
3. Select **Preview> Surface Mesh** in the context menu or **Preview Surface Mesh** in the drop-down menu. The surface mesh is displayed for the model when you select the **Mesh** object.

To preview the surface mesh for individual unsuppressed parts - from the object tree:

1. Select the Part objects.
2. Right-click to display the context menu.
3. Select **Preview> Surface Mesh** in the menu. The surface mesh is displayed for the parts when you select the **Mesh** object.

To preview the surface mesh for individual unsuppressed parts - from the Geometry window:

1. Select the Mesh object.
2. Select the parts in the **Geometry** window.
3. Right-click to display the context menu.
4. Select **Parts> Preview Surface Mesh** in the menu. The surface mesh is displayed for the parts when you select the **Mesh** object in the tree.

To preview the surface mesh for individual unsuppressed bodies - from the object tree:

1. Select the Body objects.
2. Right-click to display the context menu.
3. Select **Preview> Surface Mesh** in the menu. The surface mesh is displayed for the bodies when you select the **Mesh** object.

To preview the surface mesh for individual unsuppressed bodies - from the Geometry window:

1. Select the Mesh object.
2. Select the bodies in the **Geometry** window.
3. Right-click to display the context menu.
4. Select **Preview Surface Mesh On Selected Bodies** in the menu. The surface mesh is displayed for the bodies when you select the **Mesh** object in the tree.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After successfully previewing the surface mesh, you can view statistics about it. These statistics include mesh metric information that you can use to evaluate the mesh quality. For more information, see Statistics Group (p. 198).

Exporting a Previewed Surface Mesh in Fluent Format

Follow the steps below to export a previewed surface mesh in Fluent format:

To export a previewed surface mesh in Fluent format:

1. Follow the procedure to preview a surface mesh (p. 531).
2. Select **File> Export** from the main menu to export the surface mesh.
3. In the **Save As** dialog box, choose a directory and specify a file name for the file. Then choose **Fluent Input Files** from the **Save as type** drop-down menu and click **Save**.

   A `.msh` file suitable for import into Fluent will be created in the requested directory.

Previewing Source and Target Mesh

This feature allows you to preview the source and target meshes for scoped bodies. You can preview the source and target mesh on individual bodies or multiple selected bodies across different parts or within the same part. This feature applies only to a **Method** (p. 202) control set to the **Sweep** option.

Note:

- This feature is not supported for thin model sweeping (that is, use of the **Sweep** mesh method control with **Src/Trg Selection** set to **Manual Thin** or **Automatic Thin**).
- Refer to Selective Meshing (p. 444) for general information about selective meshing and limitations related to using the **Preview Source and Target Mesh** feature with selective meshing.

To preview the source and target mesh:

1. Scope the body or bodies of interest.
2. Right-click the Mesh object and insert a **Method** (p. 202) control.
3. In the Details View, set **Method** to the **Sweep** option.
4. Right-click the **Sweep Method** option in the tree to display the context menu, or choose the **Mesh** drop-down menu from the toolbar.
5. Select **Preview> Source and Target Mesh** in the context menu or **Preview Source and Target Mesh** in the drop-down menu. The source and target meshes are displayed when you select the **Mesh** object.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Previewing Inflation

The **Preview Inflation** feature helps you identify possible problems with inflation before you generate the mesh. You can preview inflation on single body parts, multibody parts, individual bodies, or multiple selected bodies across different parts or within the same part. You can also export the previewed inflation mesh file in Fluent format, as described in Exporting a Previewed Inflation Mesh in Fluent Format (p. 535).

Remember the following information when using the **Preview Inflation** feature:

- This feature is not supported for the **Patch Independent Tetra** and **MultiZone** mesh method controls.
- Match controls (p. 334) are not enforced when previewing inflation.
- In certain cases, **Preview Inflation** may return an invalid (p. 579) mesh. **Generate Mesh** (p. 528) may return a valid mesh with inflation, a valid mesh without inflation (because inflation failed), or result in a mesh failure (because tet meshing failed). For these reasons if **Generate Mesh** fails, using either the **Preview Surface Mesh** (p. 531) or **Preview Inflation** feature to locate the worst quality element is also likely to locate the cause of the mesh failure.
- When previewing inflation on bodies that are being meshed with **Sweep**, not all sizing information is used in the calculation. Previewing inflation on such bodies is a crude check to get a general idea whether inflation will work and/or if there will be problems with inflation prior to generating the full mesh, but the full mesh may look different due to differences in sizing calculations when taking all meshing constraints into account.
- This feature applies only when the **Inflation Algorithm** control is set to Pre (p. 172).
- After successfully previewing inflation, you can view statistics about it. These statistics include mesh metric information that you can use to evaluate the mesh quality. For more information, see Statistics Group (p. 198).
- Refer to Selective Meshing (p. 444) for general information about selective meshing and limitations related to using the **Preview Inflation** feature with selective meshing.

To preview inflation:

1. Apply inflation to the desired boundaries.
2. Select an object in the Tree Outline (the **Mesh** object, a mesh **Method** object, or an **Inflation** object) and right-click to display the context menu.
3. Select **Preview > Inflation** in the menu. In response, Workbench generates the inflation layers only and displays them in the **Geometry** window. You may need to click the **Mesh** object in the Tree Outline before you can view the inflation layers.

Figure 199: Previewed Inflation Mesh (p. 535) shows a model of an auto manifold to which inflation was applied. The **Preview Inflation** feature was selected, and the inflation layers were generated and displayed in the **Geometry** window.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 199: Previewed Inflation Mesh

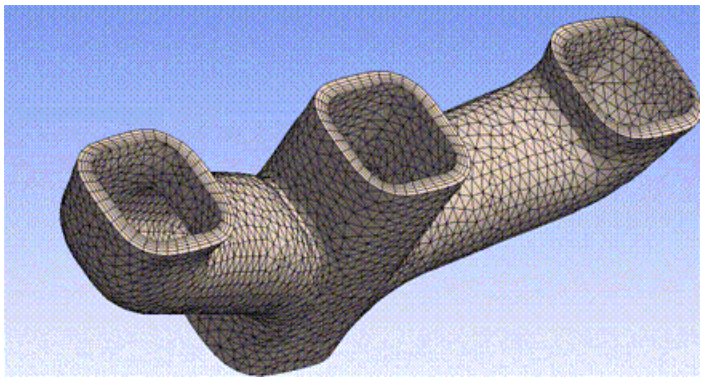
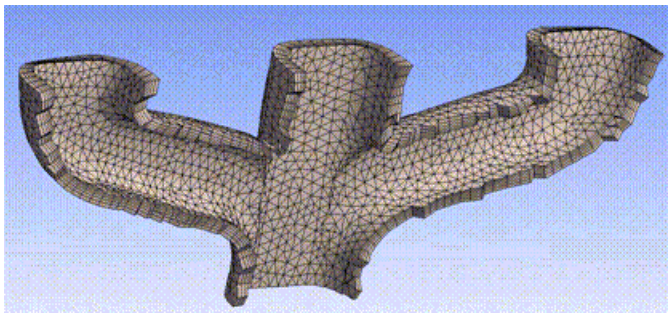


Figure 200: Section Plane View of Previewed Inflation Mesh (p. 535) shows a different view of the auto manifold model. For this view, a Section Plane was defined so that the inflation layers could be viewed internally.

Figure 200: Section Plane View of Previewed Inflation Mesh



Exporting a Previewed Inflation Mesh in Fluent Format

Follow the steps below to export a previewed inflation mesh in Fluent format:

To export a previewed inflation mesh in Fluent format:

1. Follow the procedure to preview inflation (p. 534).
2. Select **File> Export** from the main menu to export the inflation mesh.
3. In the **Save As** dialog box, choose a directory and specify a file name for the file. Then choose **Fluent Input Files** from the **Save as type** drop-down menu and click **Save**.

   A `.msh` file suitable for import into Fluent will be created in the requested directory.

Previewing Weld Geometry

The **Preview Weld Geometry** feature helps you to preview the weld faces before meshing. It also allows you to view the errors and warnings for weld before meshing. You can access **Preview Weld Geometry** from **Mesh** object and from each row of the weld worksheet.

To access **Preview Weld Geometry**,

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **Tree** view, right-click the **Mesh** object.
2. Select **Preview > Weld Geometry**.

Previewing Weld Mesh

The **Preview Weld Mesh** feature helps you to generate the weld mesh on the selected controls for preview. You can access **Preview Weld Mesh** from **Mesh** object and **Weld** control only. **Preview Weld Mesh** creates mesh only for the weld faces and its corresponding up and down faces. You can also right-click any column in the Worksheet to access **Preview Weld Mesh**.

To access **Preview Weld Mesh**,

1. In the **Tree** view, right-click the **Mesh** object.
2. Select **Preview > Weld Mesh**. The Weld Mesh is displayed on the weld faces and on its up and down faces.

Previewing Quad Layer

The **Preview Quad Layer** feature helps you to preview the quad layers before mesh generation. You can access **Preview Quad Layer** from **Mesh** object and **Quad Layer** control only.

Note:

When you use **Preview > Quad Layer**, you cannot preview the applied edge sizing.

To access **Preview Quad Layer**,

1. In the **Tree** view, right-click the **Mesh** object.
2. Select **Preview > Quad Layer** to display the quad layer mesh created.

You can also right-click **Quad Layer** and click **Preview > Quad Layer** to preview the quad layer before mesh generation

Showing Program Controlled Inflation Surfaces

If you select **Program Controlled** as the value for the Use Automatic Inflation (p. 163) control, all surfaces in the model are selected to be inflation boundaries, except:

- Surfaces that are members of Named Selections groups
- Faces on non-enclosure bodies if an enclosure body is present
- Faces on bodies that have manual inflations on them
- Interfaces between bodies in a multibody part
- Faces used in contact

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Faces used in symmetry
- Faces on bodies being meshed with Hex Dominant or manual Sweep controls

To view the faces that have been selected for inflation:

1. Select the **Mesh** object in the Tree Outline.
2. Right-click to display the context menu.
3. Select **Show> Program Controlled Inflation Surfaces** in the context menu.

Showing Sweepable Bodies

You can display bodies that are "sweepable" according to the criteria detailed under Mesh Sweeping (p. 399).

To display sweepable bodies:

1. Select the Mesh object or any mesh control object.
2. Right-click to display the context menu.
3. Select **Show> Sweepable Bodies** in the menu. All sweepable bodies are displayed.

Showing Problematic Geometry

If problematic geometry causes meshing to fail, Workbench alerts you by:

- Displaying a **Problematic Geometry** message annotation in the **Geometry** window
- Displaying messages in the Messages window to describe the problem

For related information on viewing problematic geometry, refer to:

- Show Problematic Geometry in the DesignModeler help
- Meshing: Troubleshooting (p. 579)

Showing Problematic Location

If problematic location causes meshing to fail, Workbench alerts you by:

- Displaying a **Show Problematic Location** message annotation in the **Geometry** window
- Displaying messages in the Messages window to describe the problem

---

Note:

When the problematic location is inside the body, you can use wire frame mode to visualize that particular location.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For related information on viewing problematic location, refer to:

- Show Problematic Geometry in the DesignModeler help
- Meshing: Troubleshooting (p. 579)

Showing Problematic Elements

When you select mesh **Method** as **Hex Dominant** or **Physics Preference** as **Explicit**, a warning message is provided if internal pyramid elements are created while meshing. You can right-click the warning message and click **Show Problematic Elements** to create a named selection for all pyramid elements.

Showing Elements that Do Not Meet the Target Metric

If Check Mesh Quality (p. 121) is set to **Yes, Errors and Warnings**, and the generated mesh contains elements that do not meet the specified target metric, a warning message is displayed in the Messages window to indicate the problem.

You can create a Named Selection for these elements by right-clicking the Messages field and selecting **Show Elements** in the context menu. A Named Selection named **Warning Elements** will be created.

Refer to Quality Group (p. 120) for related information.

Showing Removable Loops

You can use the **Show Removable Loops** feature prior to meshing to view loops that will be removed according to the criteria defined by the global **Sheet Loop Removal** (p. 197) and **Loop Removal Tolerance** (p. 197) controls.

This feature applies only to sheet models, and it responds only to the settings of the global loop removal controls. For example, if you set loop removal controls locally and your model contains loops that will be removed according to your local criteria, the **Show Removable Loops** feature will return nothing if the global **Sheet Loop Removal** control is off (set to **No**).

To view removable loops:

1. Select the **Mesh** object in the Tree Outline.
2. Right-click to display the context menu.
3. Select **Show> Removable Loops** in the context menu.

Note:

This feature is not supported for the **MultiZone Quad/Tri** (p. 284) mesh method.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Showing Suppressible Features

You can use the **Show Suppressible Features** to preview the faces that are to be suppressed, before applying the feature suppress control.

To view Suppressible Feature:

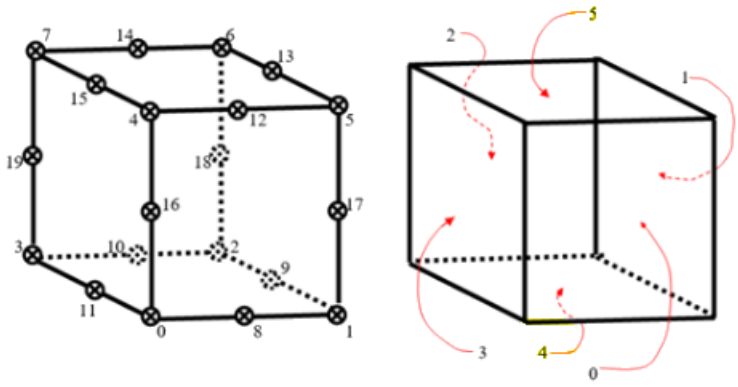
1. Right-click the **Mesh** object in the Tree Outline.
2. Select **Show > Suppressible Features** in the context menu.

Displaying Element Normals on Mesh

Display Element Normals on Mesh enables you to display a normal vector for each element. The normal vector orientation ensures the accurate representation and analysis of the Multiphysics effects within the cell structure. The default value is **No**. The available options are **Yes** and **No**. You can access **Display Element Normals on Mesh** from **File** > **Options** > **Graphics** > **Default Graphics Options**.

Hexahedral Element

For hexahedral element, you should pay attention to the connectivity, to ensure the correct orientation of the through-thickness direction. The mesher establish the orientation direction of hex elements considering the topological face selection as a reference point to determine the lower portion of the reoriented hex element. The shared internal faces are not considered for determining the orientation as the internal algorithm autonomously expands the hex layer for re-orientation, starting from the anchor face and progressing inwards.



Here,

nodes 0 to 3 define the lower surface.

nodes 4 to 7 define the upper surface.

face index 4 represents the bottom face.

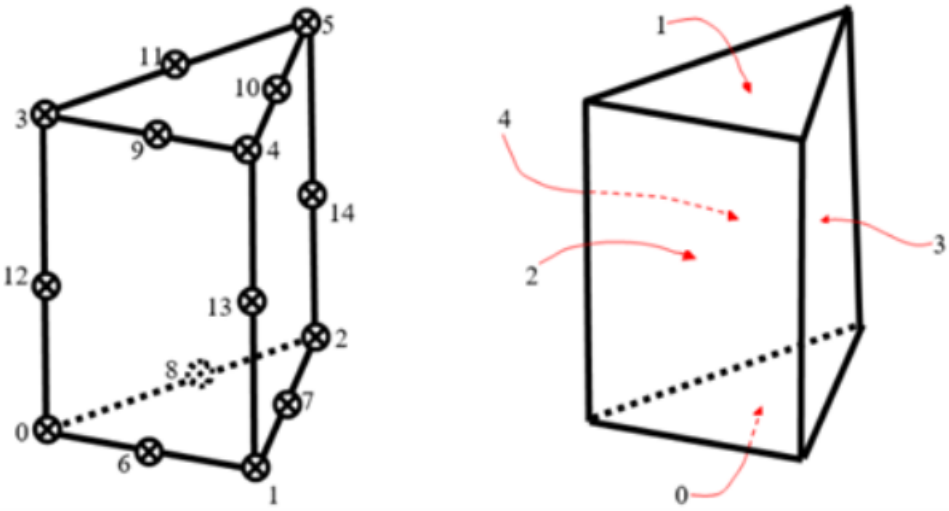
face index 5 represents the top face.

The normal to face index 5 defines the normal of the hex element.

Wedge Element

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For a wedge element, the connectivity of nodes defines the element's orientation.



Here,

nodes 0 to 2 define the lower surface.

Nodes 3 to 5 define the upper surface.

face index 0 represents the bottom face.

face index 1 represents the top face.

The normal to face index 1 defines the normal of the wedge element.

Note:

Hexahedral and wedge elements only support normal.

Inspecting Large Meshes Using Named Selections

You can use Named Selections to inspect only a portion of the total mesh. Although this feature is available regardless of mesh size, it is most beneficial when working with a large mesh (greater than 5 - 10 million nodes). For details, refer to Displaying Named Selections in the Mechanical help.

Generating Multiple Mesh Controls from a Template

The Object Generator enables you to make one or more copies of a template object, scoping each to a different piece of geometry. When defining mesh controls, you can use the Object Generator to make copies of a template mesh control, which may reduce the necessity to manually define multiple related mesh controls. For details, refer to Generating Multiple Objects from a Template Object in the Mechanical help.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clearing Generated Data

You can clear generated data from the database using a right-mouse click menu item. You can either clear all mesh and results data (if applicable) from a model, or clear the mesh data on a selected part or body.

Note:

- When you clear the mesh, the status of the part or body will indicate that it is not meshed.
- When used on parts and bodies that have been joined by mesh connections (p. 485), the **Clear Generated Data** option works as follows, where the "base" mesh, which is stored in a temporary file, is the mesh in its unsewn (pre-joined) state:
  - If a base mesh is available, the mesh is reverted to the base mesh and the requested parts/bodies are cleared.
  - If no base mesh is available, the entire mesh is cleared and a warning message is issued. Reasons the base mesh may not be available include situations in which you have deleted your temporary files, exported a .mechdat file for someone else to use, or moved your project database to a different computer.
  - All mesh connections in the model, including those not associated with the selected body or part, are cleared.
  - Because a Node Move cannot be undone, **Clear Generated Data** is not available from the **Mesh Edit** context menu when the **Mesh Edit** object has only **Node Move** objects as children.

To clear all mesh and results data from a model - from the object tree:

1. Select the **Mesh** object in the Tree Outline.
2. Right-click to display the context menu.
3. Select **Clear Generated Data** in the context menu.
4. When asked whether you want to clear the data, click **Yes**.

To clear the mesh data from the selected part or body - from the object tree:

1. Select the part or body in the Tree Outline.
2. Right-click to display the context menu.
3. Select **Clear Generated Data** in the context menu.
4. When asked whether you want to clear the mesh, click **Yes**.

To clear the mesh data from the selected body - from the Geometry window:

1. Select the Mesh object.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select the bodies in the **Geometry** window.
3. Right-click to display the context menu.
4. Select **Clear Generated Data On Selected Bodies** in the context menu.
5. When asked whether you want to clear the mesh, click **Yes**.

To clear the mesh data from the selected part - from the Geometry window:

1. Select the Mesh object.
2. Select the part in the **Geometry** window.
3. Right-click to display the context menu.
4. Select **Parts> Clear Generated Data** in the context menu.
5. When asked whether you want to clear the mesh, click **Yes**.

Showing Missing Tessellations

Geometry with missing facets can lead to incorrect geometry representation by the mesher. Using the **Show Missing Tessellations** feature, you can highlight geometry with missing facets, which will allow you to detect and resolve problems prior to mesh generation.

Note:

This feature is available only for the **Patch Independent Tetra** (p. 210) mesh method.

To highlight missing tessellations:

1. Select the **Geometry** object in the Tree Outline.
2. Right-click the **Geometry** object or in the **Geometry** window to display the context menu.
3. Select **Show Missing Tessellations** in the context menu.

If a face without tessellations is found:

- The boundary of the face will be highlighted in the **Geometry** window.
- A warning message will be displayed in the **Messages** window advising you to use the **Show Problematic Geometry** (p. 537) context menu option to locate the problem areas.

Showing Mappable Faces

You can display faces that are "mappable" according to the criteria detailed under Face Meshing Control (p. 314).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To display mappable faces:

1. Select the Mesh object or any mesh control object.
2. Right-click to display the context menu.
3. Select **Show> Mappable Faces** in the menu. All mappable faces are highlighted in the **Geometry** window.
4. Right-click the Mesh object or mesh control object and select **Insert> Face Meshing**. In the Details View, the **Geometry** field shows the number of faces that are mappable and therefore were selected.

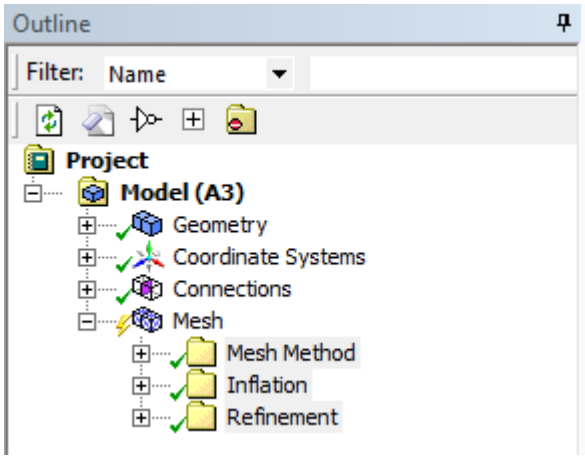
By default, **Definition>Mapped Mesh** is set to **Yes**, which means the faces in the **Face Meshing** control are mapped by default.

To edit the selected set of faces, click the **Geometry** field in the Details View to activate it. Then in the **Geometry** window, select the mappable faces that you want to use in the **Face Meshing** control. Click **Apply** in the Details View to complete your selection. Then proceed with the procedure described in Face Meshing Control (p. 314).

Grouping Mesh Objects By Type

If you have many different types of mesh objects, you can use the **Group All Similar Children** option to organize the objects into folders based on their type. In the following figure, the mesh methods, inflation control, and refinement have been grouped into separate folders.

Figure 201: Mesh Objects Grouped By Type



For more information, see Group Tree Objects.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Meshing: Virtual Topology

The following sections cover these topics:

Introduction

Creating and Managing Virtual Cells

Creating and Managing Virtual Split Edges

Creating and Managing Virtual Split Faces

Creating and Managing Virtual Hard Vertices

Common Virtual Topology Operations

Common Virtual Topology Features

Introduction

Uses of virtual topology include:

- Before performing analysis of a CAD model, you may want to group faces/edges together to form virtual cells. In such cases, virtual topology can aid you in reducing the number of elements in the model, simplifying small features out of the model, and simplifying load abstraction.
- You can split a face to create two virtual faces, or split an edge to create two virtual edges. For example, in the case of a rectangular face in which a single edge on one side of the face corresponds to two edges on the opposite side of the face, you can split the single edge so that node alignment across the face can have similar spacing.
- When needed, you can create virtual hard vertices to facilitate split face operations.
- Virtual topology can be helpful for handling fillets for **MultiZone** and sweep meshing. See Using Virtual Topology to Handle Fillets in MultiZone Problems (p. 439).

A CAD Model has two parts:

1. **Topology:** The connectivity of a CAD model, meaning: vertices are connected to edges, which are connected to faces, which are connected to volumes. Each one of these entities is referred to as a cell.
2. **Geometry:** The geometry of the CAD model is the underlying mathematical definition of the aforementioned cells.

A virtual cell in the Mechanical application or the Meshing application modifies the topology of only the local copy in the Mechanical application or Meshing application. Your original CAD model remains unchanged. New faceted geometry is also created with virtual topology. However, the mesher may project the nodes back to the original geometry where applicable.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can use Virtual Topology to simplify the geometry to aid in meshing. There are several ways to simplify the topology using either automatic, manual, or a combination of automatic and manual approaches. The best approach to use is generally based on the meshing objectives (number of elements desired) and the cleanliness of the CAD model.

Virtual cells are often created to:

- Reduce the element count (increase the mesh size).
- Fix the topology of bodies that are not sweepable, to have mappable faces so that the bodies are now sweepable.
- Fix meshing problems.

To reduce the element count, it is often a good idea to first use **Automatic** or **Repair** operations to reduce the number of faces/edges, and then use **Manual** virtual topology operations to ensure important topology is respected and/or features that could create mesh quality problems are removed.

To fix the topology of bodies make them sweepable. **Automatic** and/or **Repair** operations might be helpful, but it is often very dependent on the geometry. **Manual** virtual topology operations give you more control and are often the better approach.

Meshing problems sometimes occur because of the topology. When a meshing failure occurs it generally points to the offending topology. Using **Manual** virtual topologies is a good way to fix such problems.

For more information about the Automatic and Manual approaches to creating virtual cells, see:

- Creating Virtual Cells Manually (p. 549)
- Creating Virtual Cells Automatically Using Automatic Mode (p. 552)
- Creating Virtual Cells Automatically Using Repair Mode (p. 556)

Note:

- There are geometric limitations to creating virtual cells, including those related to cells that would have too much curvature, or other limitations in trying to represent a group of faces by a single face.
- The tesselation of models from CATIA4 may not be appropriate for virtual topology, which could prevent the creation of virtual cells for these models.

Creating and Managing Virtual Cells

Notes on Virtual Cell Creation

- Scoped objects except other virtual cells may need to be relinked to the new virtual cells when that virtual cell is composed of entities in the scoped object. When the virtual cell is deleted the object may need to be rescoped to the original entities.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- All scoped objects except for mesh controls and other virtual cells will be protected during automatic virtual cell generation. This will allow users to load their models and run auto virtual topology without deleting loads. All faces within a protected object may be merged with faces in the same protected object and not in any other protected object.
- If **Generate on Update** is set to **Yes** and you update the geometry, all **Virtual Cell** objects that were created automatically will be deleted and recreated based on the new geometry. Any loads that were attached to geometry within the deleted **Virtual Cell** objects will need to be reattached to the new geometry.
- If **Generate on Update** is set to **No** and you update the geometry, all **Virtual Cell** objects that were created automatically should remain persistent barring major topology changes of the model being updated. Reapplication of loads may not be necessary.
- For any virtual cells that were generated, you can choose whether the nodes will be projected back to the underlying geometry (in contrast to the virtual cell's faceted geometry). To do so, select the desired virtual cell and use the **Virtual Topology Properties** dialog (p. 573) to set the **Project to Underlying Geometry** option to **Yes** or **No**. The default is **Yes** for analytical geometry. Otherwise, the default is **No**. Computational expense increases when the option is set to **Yes**. Use these guidelines:
  - **Yes** is recommended if:
    - → You are using virtual topology to fine tune the topology of your mesh and need precise control from the Size Function (p. 105).
    - → Your geometry is valid and you want the mesh to capture it accurately. Here, there may be a slight impact on performance.
  - **No** is recommended if:
    - → You are using virtual topology to simplify bad/corrupt geometry or topology.
    - → You are trying to grossly defeature a model (for example, remove bosses, serial numbers and so on).

Virtual Cell Logic and Usage

The Virtual Topology feature is a cell dependency operation where existing virtual cells can be used to create new virtual cells both manually and automatically. Within the cell hierarchy, a virtual cell depends on another virtual cell if the latter is used to create the former.

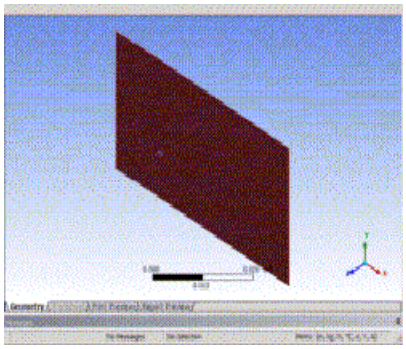
Virtual Face Dependency

A virtual face depends on a virtual edge if during the face creation the virtual edge is used to create virtual face loops. A virtual face depends on another virtual face if the latter was used to create the former.

The following image represents the Merge Face Edges in the off position.

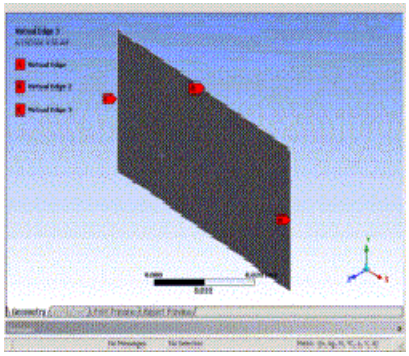
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 202: Merge Face Edges Off



The following image represents the Merge Face Edges in the on position.

Figure 203: Merge Face Edges On



Virtual Edge Dependency

A virtual edge depends on a virtual face if during the edge creation an owner loop belongs to the said face. A virtual edge depends on another virtual edge if the latter was used to create the former.

Note:

- If a virtual edge was created from a virtual split edge, you cannot delete the virtual split edge without first deleting the virtual edge. Conversely, if a virtual split edge was created from a virtual edge, you cannot delete the virtual edge without first deleting the virtual split edge.

  A warning message appears in the **Messages** window for each failed deletion. To highlight the geometry that is responsible for a message, select the message, right-click, and select **Show Problematic Geometry** from the context menu.

- In addition, if the virtual edge belongs to a virtual face, the virtual face will not be deleted either.

  A warning message appears in the **Messages** window for the virtual face, and you can use the **Show Problematic Geometry** option to highlight the face.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can manually designate faces and edges for inclusion into a virtual cell, or you can have the Mechanical application or the Meshing application automatically create virtual cells based on settings that you specify. You can use the **Automatic** mode to globally reduce the number of faces and edges where possible, or use **Repair** to focus more closely on problematic faces and edges. The geometry under a virtual cell is represented by the underlying cell's graphic resolution.

Note:

- There are geometric limitations to creating virtual cells, including those related to cells that would have too much curvature, or other limitations in trying to represent a group of cells by a single cell.
- The tessellation of models from CATIA4 may not be appropriate for virtual topology, which could prevent the creation of virtual cells for these models.

Creating Virtual Cells Manually

1. Insert a Virtual Topology object in the tree.
2. Choose the face (**Ctrl**+**F**) or edge (**Ctrl**+**E**) selection filter, and then pick one or more faces or one or more edges that you want to include in the virtual cell(s).

   You can use the **Close Vertices** button to identify tightly clustered vertices that might need to be merged.
3. Create the Virtual Cell object(s) by doing one of the following:
   - Choose **Merge Cells** on the **Virtual Topology** context toolbar. You can also use Ctrl+M to merge either the faces after selecting them, or the common edge between faces.
   - Click the right mouse button on the Virtual Topology object and select **Insert> Virtual Cell** from the context menu.
   - Click the right mouse button in the **Geometry** window and select **Insert> Virtual Cell** from the context menu.

   From the selected set of faces or edges, the software creates the virtual cell(s). During this process, adjacent selected entities are grouped appropriately to form virtual cell(s), while any single selected entity (that is, one that is selected but is not adjacent to any other selected entity) forms its own virtual cell. An error message appears in the **Messages** window for each subset of failed topologies. To highlight the geometry that is responsible for a message, select the message, right-click, and select **Show Problematic Geometry** from the context menu. Refer to the examples below.

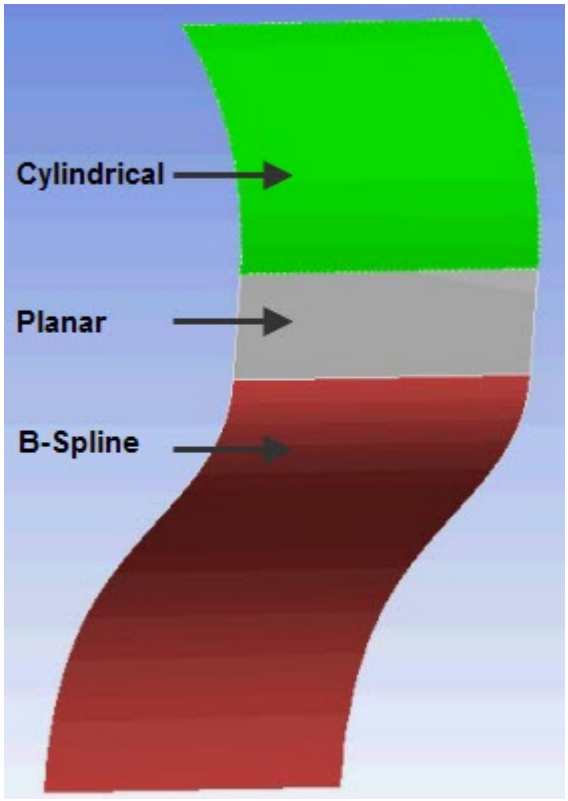
Note:

- A virtual cell cannot be created on a single edge that is straight or enclosed with no vertices.
- A virtual cell cannot be created on a single face that is cylindrical or planar. For example, if you select either the top face or middle face in the figure below and try to create a vir-

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tual cell, no virtual cell will be created. However, selecting the bottom face will result in creation of a virtual cell.

Figure 204: Single Face Virtual Cell Limitations



Examples of Virtual Cell Formation

In the example shown in Figure 205: Formation of Virtual Faces (p. 551), suppose that you select faces A, B, E, and F and then select **Insert> Virtual Cell**. As a result, virtual faces **AB** and **EF** will be formed, as shown in Figure 206: Virtual Faces After Operation (p. 551).

The edges would be handled according to the **Merge Face Edges** setting. For example, if **Merge Face Edges** is set to **Yes**, virtual edges **ac** and **km** also will be formed.

As another example, using the same starting point, if you select faces A, E, and F and then select **Insert> Virtual Cell**, virtual faces **A** and **EF** will be formed. In this case, if **Merge Face Edges** is set to **Yes**, virtual edge **km** also will be formed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 205: Formation of Virtual Faces

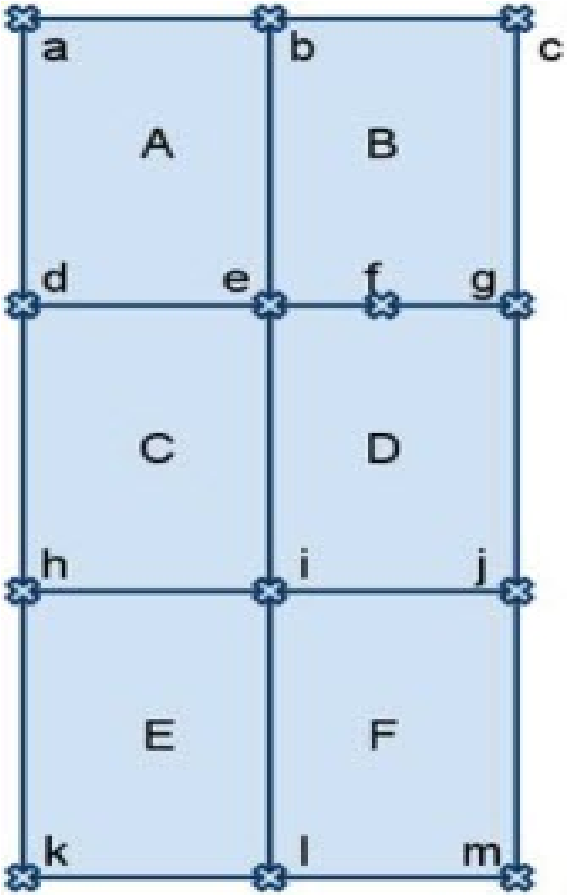
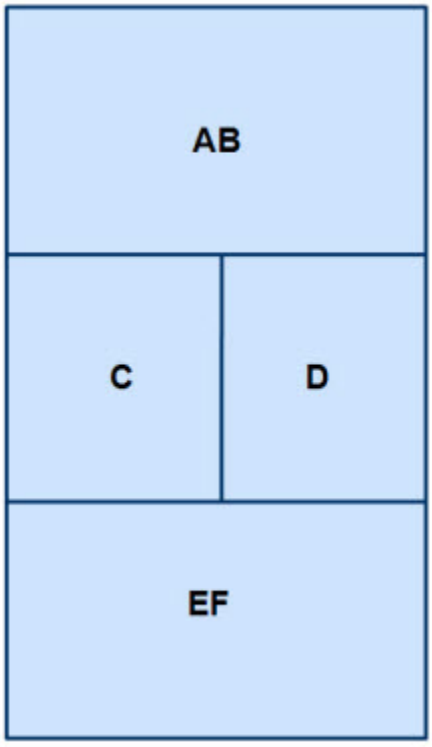


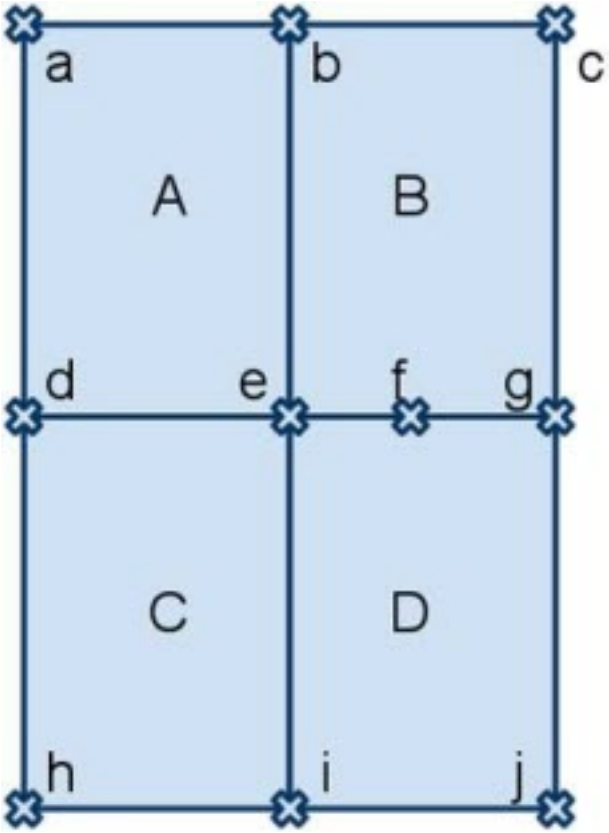
Figure 206: Virtual Faces After Operation



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the example shown in Figure 207: Formation of Virtual Edges (p. 552), suppose that you select all edges and then select **Insert> Virtual Cell**. In this case, only virtual edge **eg** will be formed because only those virtual edges that can be formed without forcing face merges will be created.

Figure 207: Formation of Virtual Edges



Creating Virtual Cells Automatically Using Automatic Mode

1. Insert a Virtual Topology object in the tree.
2. In the Details View, ensure that **Method** is set to **Automatic**.
3. Make adjustments as needed to any of the following settings in the Details View:
   - **Behavior** – Determines how aggressively the face(s) and edge(s) are merged. The choices are **Low**, **Medium**, **High**, **Edges Only**, and **Custom**. The **Edges Only** setting will merge only edges. The **Custom** setting exposes **Custom** properties (**Curvature** and **Feature Angles**) and **Advanced Custom** properties (**Aspect Ratio**, **Contact Angle**, **Edge Angle**, and **Shared Boundary Ratio**). These properties enable you to set parameters that control the creation of automatic Virtual Topologies.

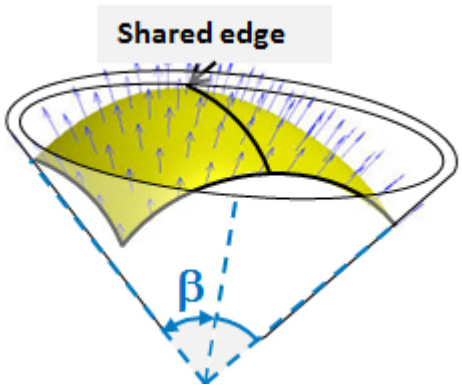
     Note:

     Setting any of the Custom or Advanced Custom Properties to **-1** resets the value of that property back to its default.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

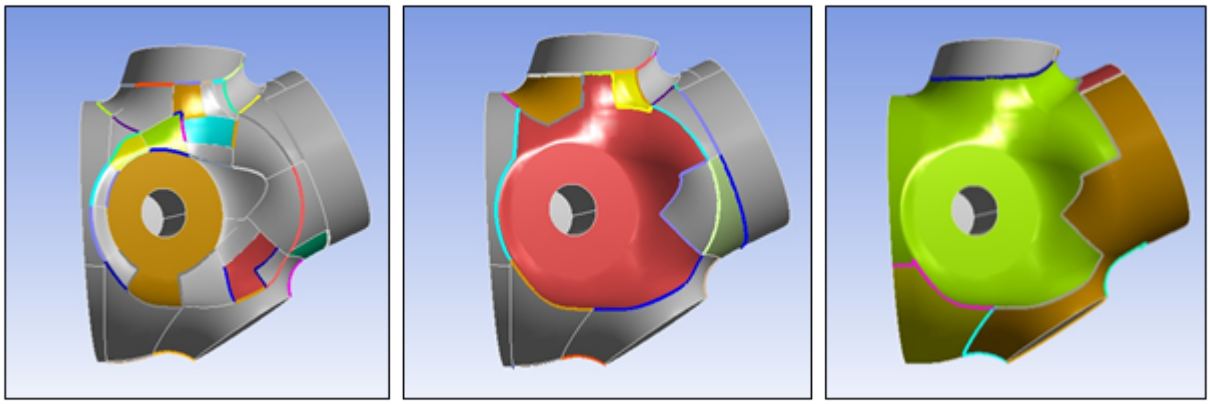
- **Gauss Curvature Angle** – Represents the flatness of the resultant face. If Angle β is greater than the Gauss Curvature Angle, the faces will stay separate. Increasing the Curvature Angle causes a greater number of faces to be grouped into fewer, larger faces, which might mean that there are fewer resulting Virtual Topologies.

Figure 208: Gauss Curvature Angle



The **Gauss Curvature Angle** can range from 0-180 degrees. The default setting is 60 degrees. The following figure shows the results of setting the **Gauss Curvature Angle** to 25, 60, and 120 degrees respectively.

Figure 209: Curvature Angle at 25, 60, and 120 degrees



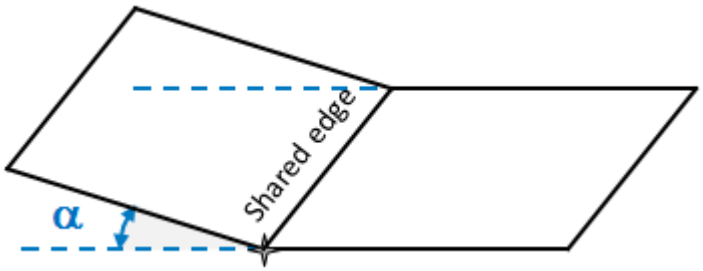
Note:

For best results, set the angle between 20 and 120 degrees. If the angle is set below this range, few faces will be merged. Setting the value above this range could result in problems with the mesh. Setting the value above this range could also result in large clusters of faces that ultimately fail to merge into a virtual face.

- **Feature Angle** – Sets the minimum angle between faces at a common edge. If Angle α, as shown in the following figure, is greater than the **Feature Angle**, the faces will stay separate. Faces are merged as the **Feature Angle** increases.

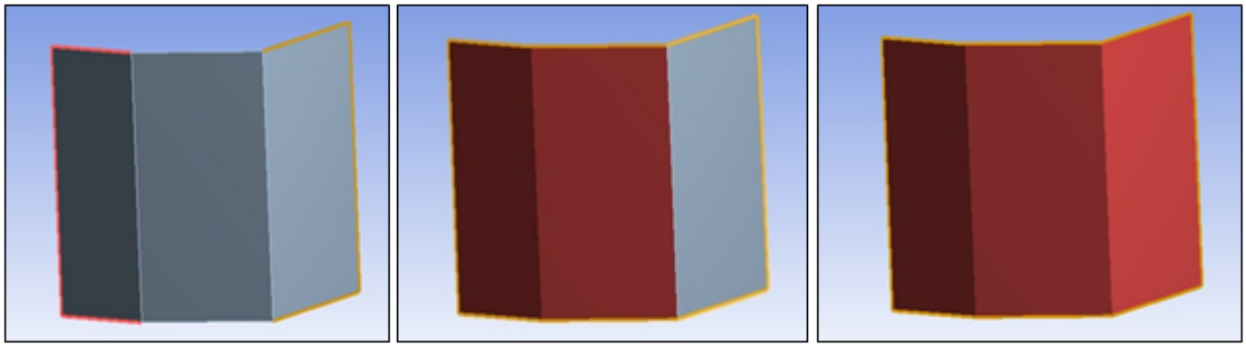
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 210: Feature Angle



The **Feature Angle** can range from 0 to 180 degrees. The default setting is 40 degrees. The following figure shows the results of setting the **Feature Angle** to 20, 40, and 80 degrees respectively.

Figure 211: Feature Angle at 20, 40, and 80 Degrees

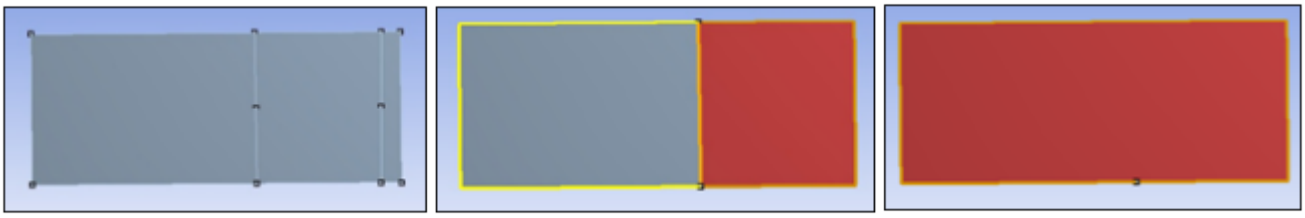


Note:

For best results, set the angle between 30 and 90 degrees. If the angle is set below this range, few faces will be merged. If the value is set above this range, it could result in large clusters of faces that ultimately fail to merge into a virtual face.

- **Aspect Ratio** – The ratio between the area of the face group to the square of the shared boundary length between the face and the Face Group. This setting controls how faces are grouped. Here, increasing the Aspect Ratio causes a greater number of faces to be grouped into fewer, larger faces. The Aspect Ratio can range from 0 to 1. The default setting is 1. The following figure shows the results of setting the Aspect Ratio to 0.2, 0.5, and 0.9, respectively.

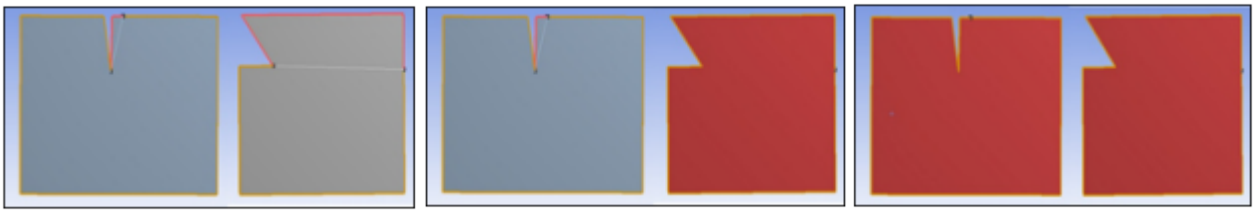
Figure 212: Aspect Ratio at 0.2, 0.5, and 0.9



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

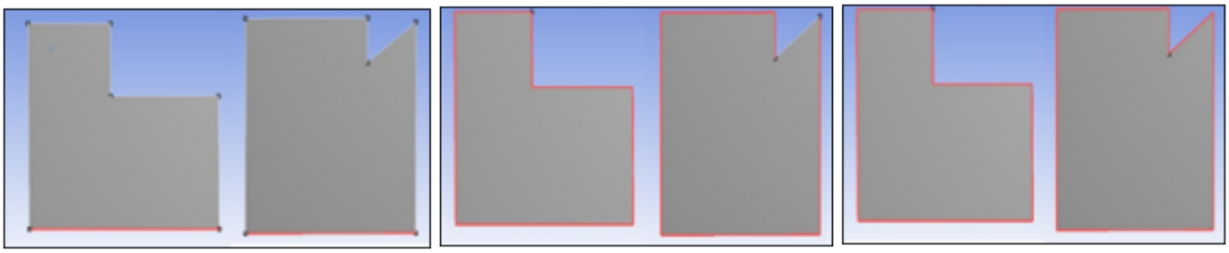
- **Contact Angle** – The angle introduced at common vertices between faces. This setting prevents complicated boundaries when grouping faces (for example, angles introduced at contact points between faces). Increasing the Contact Angle causes a greater number of faces to be grouped into fewer, larger faces. The angle can range from 0 to 360 degrees. The default setting is 360 degrees. The following figure shows the results of setting the **Contact Angle** to 270 degrees, 330 degrees, and 355 degrees, respectively.

  Figure 213: Contact Angle at 270, 330, and 355 degrees



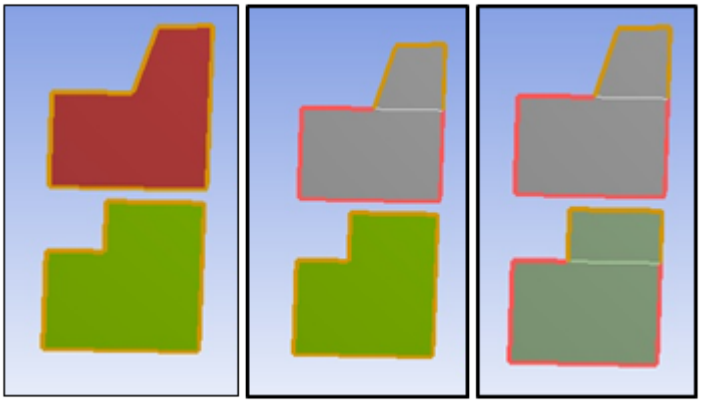
- **Edge Angle** – The feature angle between edges at their common vertex. Increasing the Edge Angle causes a greater number of edges to be grouped together. The angle can range from 0 to 180 degrees. The default setting for the Edge Angle is 80 degrees. The following figure shows the results of setting the **Edge Angle** to 80 degrees, 100 degrees, and 150 degrees, respectively.

  Figure 214: Edge Angle at 80, 100, and 150 degrees



- **Shared Boundary Ratio** – The ratio of the length of the common boundary to the length of the smallest perimeter. Increasing the Shared Boundary Ratio causes the number of grouped faces decreases. The ratio can range from 0 to 0.5. The default setting for the ratio is 0. The following figure shows the results of setting the **Shared Boundary Ratio** to 0.2, 0.3, and 0.4, respectively.

  Figure 215: Shared Boundary Ratio at 0.2, 0.3, and 0.4



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Make adjustments to the **Advanced** (p. 572) settings in the Details view.

5. Do one of the following:

   - Click right mouse button on the **Virtual Topology** object and choose **Generate Virtual Cells** from the context menu. Virtual cells are automatically created for each region that meets the criteria established by the settings in step 2.

     These virtual cells remain valid after a geometry update.

   - Choose the face (**Ctrl**+**F**) or body selection filter and pick two or more faces or two or more bodies. Click right mouse button on the **Virtual Topology** object and choose **Generate Virtual Cells on Selected Entities** from the context menu. From the selected set of faces or bodies, the software groups adjacent entities appropriately and automatically creates virtual cells for each region that meets the criteria established by the settings in step 3.

     These virtual cells become invalid after a geometry update.

Creating Virtual Cells Automatically Using Repair Mode

1. Insert a Virtual Topology object in the tree.

2. In the Details View change the **Method** to **Repair**.

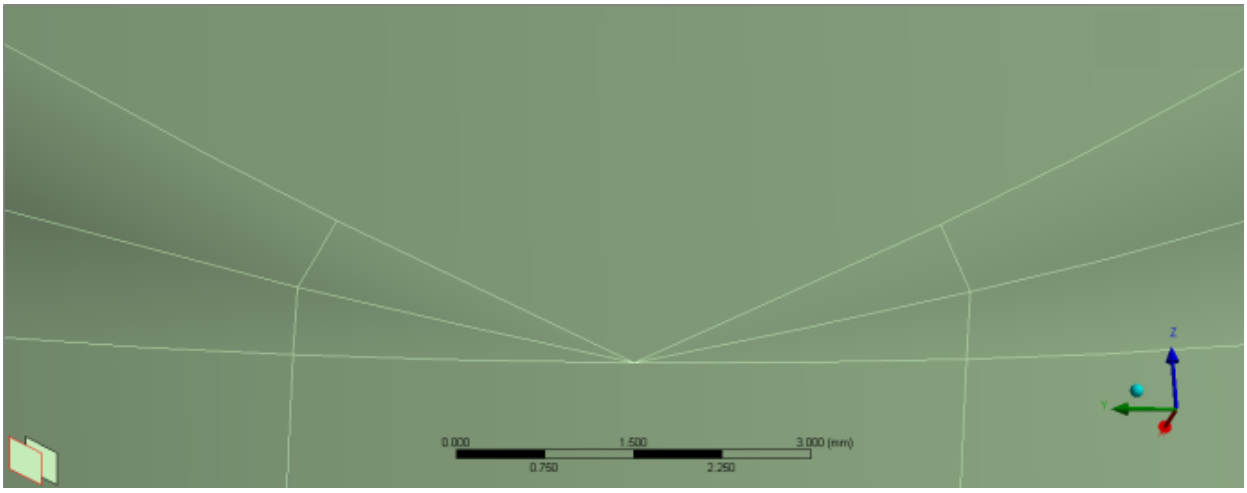
3. Make adjustments as needed to any of the following settings in the Details View:

   - **Behavior** – Determines the type of repair to be performed. The choices are **Repair All**, **Repair Small Edges**, **Repair Slivers**, and **Repair Small Faces**.

     – The **Repair Small Edges** setting tries to remove all the small edges, satisfying the **Repair Settings** criteria displayed in the Details View. The small edges are removed either by merging their attached faces or by merging the small edge with an adjacent edge. The **Repair Small Edges** setting exposes the **Max Edge Length** and **Min Edge Length** settings. The edges with lengths between **Max Edge Length** and **Min Edge Length** are repaired.

     The following example, the first figure shows small edges between faces.

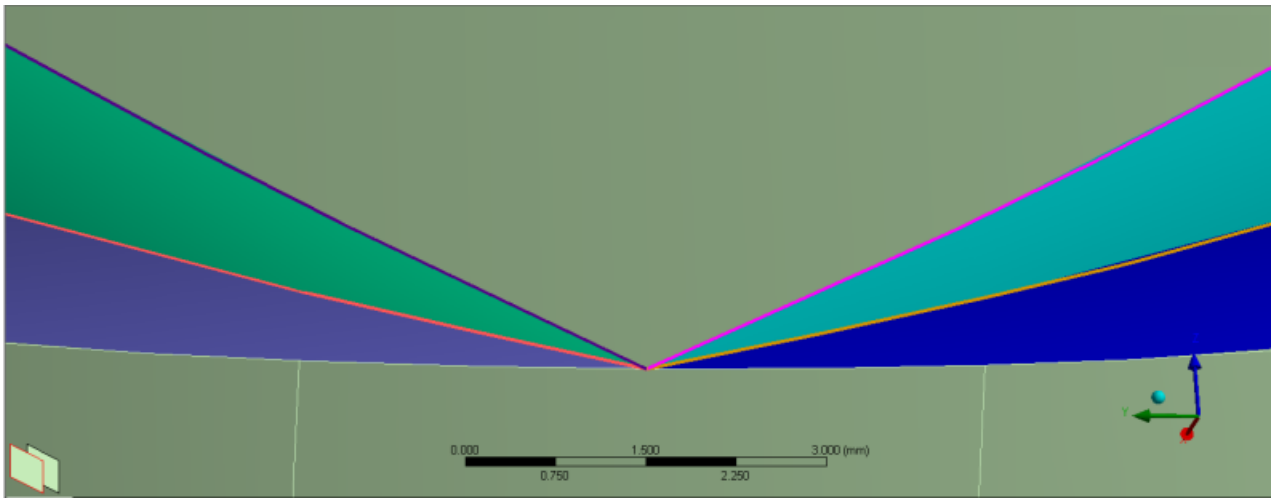
     Figure 216: Small Edges Between Faces

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

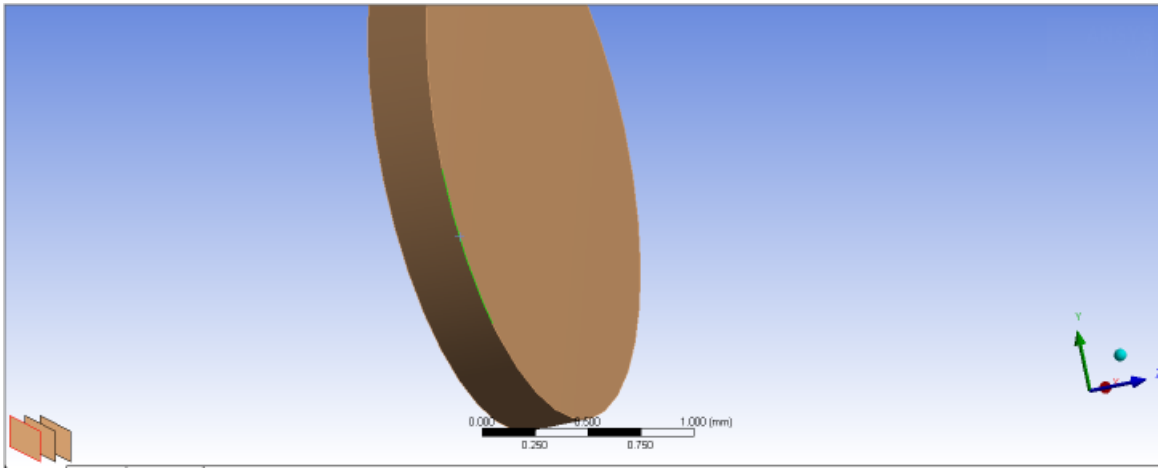
After **Small Edge Repair** (as shown in the following figure) with **Max Edge Length** set to 0.5 mm and **Min Edge Length** set to 0 mm, the small edges are removed by the faces being merged.

Figure 217: Small Edges Removed



In the following figure, small edges are attached to the same faces.

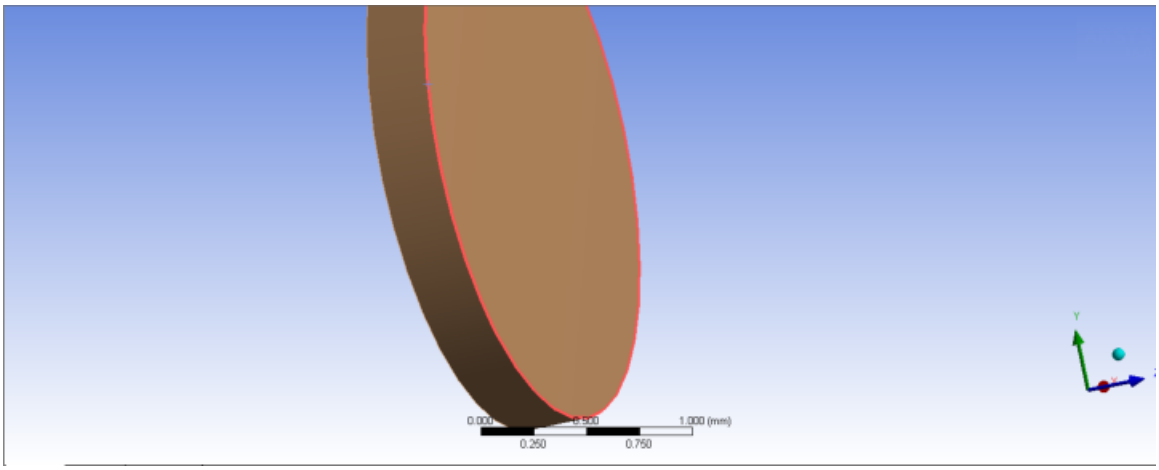
Figure 218: Small Edges Attached to the Same Faces



After **Small Edge Repair** with **Max Edge Length** set to 0.9 mm and **Min Edge Length** set to 0 mm, the small edges are merged.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

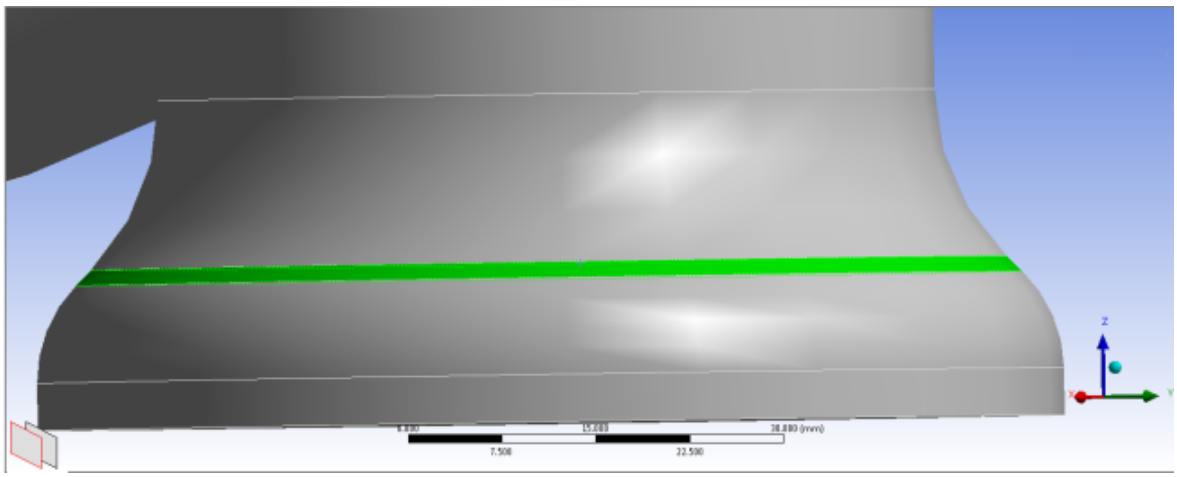
Figure 219: Small Edge Repair with Edge Merge



– The **Repair Slivers** option tries to remove all the slivers, satisfying the **Repair Settings** criteria shown in the Details View. The slivers are removed by merging the sliver face with another adjacent face. The **Repair Slivers** option exposes the **Max Sliver Width** and **Min Sliver Width** options. The sliver faces with width between **Max Sliver Width** and **Min Sliver Width** are repaired.

  The following figure shows a geometry with a sliver face.

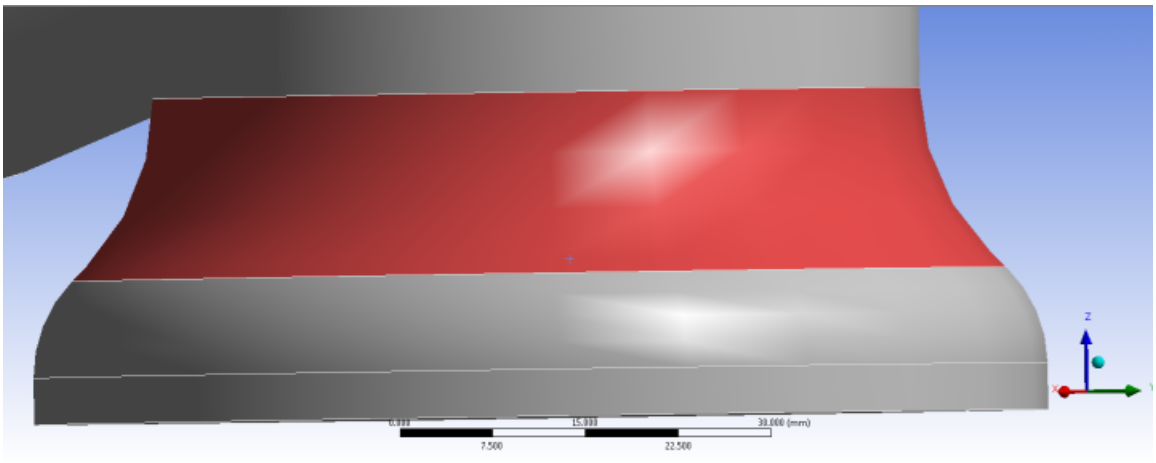
Figure 220: Sliver Face



  After **Sliver Repair** with **Max Sliver Length** set to 2 mm and **Min Sliver Length** to 0 mm, the sliver face is merged with an adjacent face.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 221: Sliver Repair

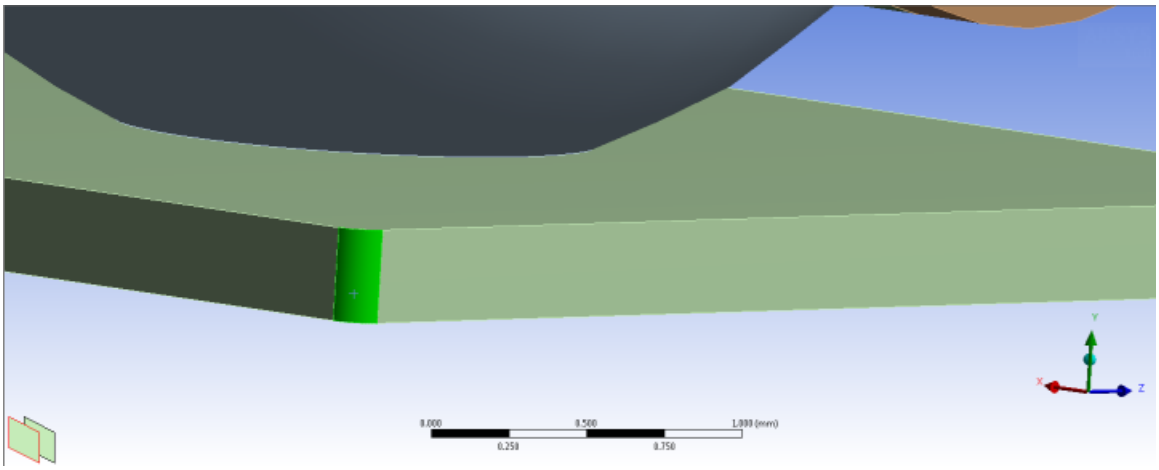


– The **Repair Small Faces** option tries to remove all the small faces satisfying the **Repair Settings** criteria shown in the Details View. The small faces are removed by merging the small face with another adjacent face.

  The **Repair Small Faces** option exposes the **Max Face Area** and **Min Face Area** options. The faces with area between **Max Face Area** and **Min Face Area** are repaired.

  The following figure shows a small face.

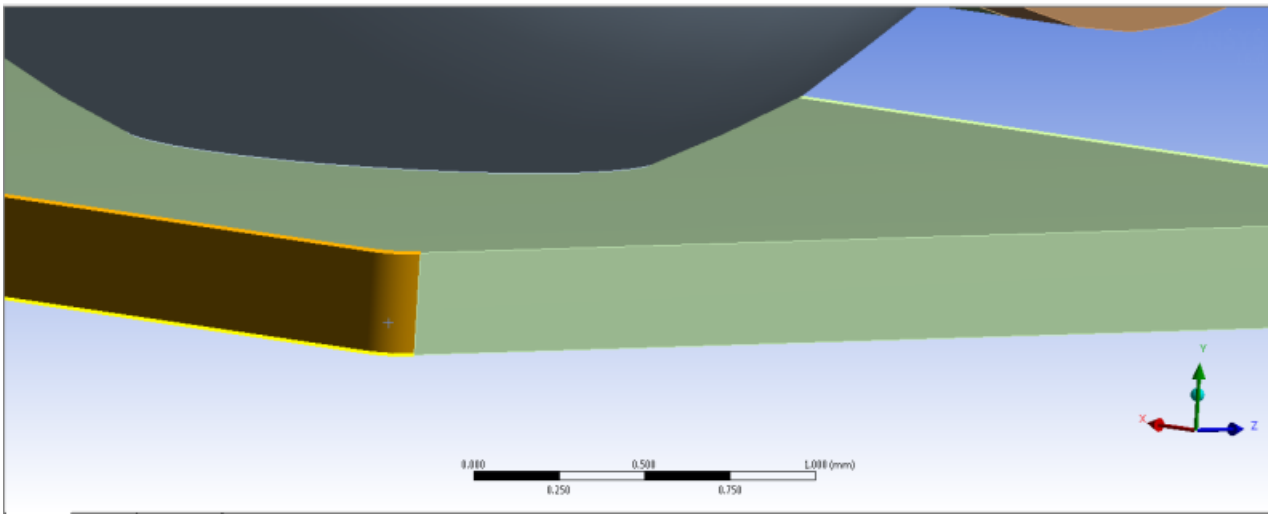
Figure 222: Small Face



After small face repair with **Max Face Area** set to 0.04 mm^2 and **Min Face Area** set to 0 mm^2, the small face is merged with an adjacent face.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 223: Small Face Repair



– The **Repair All** setting tries to remove all of the small edges, sliver faces, and small faces that satisfy the **Repair Settings** criteria shown in the Details View.

Repair Settings– Exposes settings like **Max Edge Length** and **Min Edge Length**, depending on the **Behavior** settings. The default values for different **Repair Settings** are:

→ **Max Edge Length**: Default is set to **½ Min Size** (available under **Sizing** in mesh when **Size Function** is turned ON)

→ **Min Edge Length**: Default is **0**, or no lower limit

→ **Max Face Area**: Default is set to (½ Min Size)^2.

→ **Min Face Area**: Default is 0, or no lower limit

→ **Max Sliver Width**: Default is set to ½ Min Size

→ **Min Sliver Width**: Default is 0, or no lower limit

Note:

Setting any of the Repair Settings to **-1** resets the value of that setting back to its default.

---

4. Make adjustments to the Advanced (p. 572) settings in the Details view.
5. Do one of the following:

   • Click right mouse button on the **Virtual Topology** object and choose **Generate Virtual Cells** from the context menu. Virtual cells are automatically created for each region that meets the criteria established by the settings in step 2.

   These virtual cells remain valid after a geometry update.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Choose the face (**Ctrl**+**F**) or body selection filter and pick two or more faces or two or more bodies. Click right mouse button on the **Virtual Topology** object and choose **Generate Virtual Cells on Selected Entities** from the context menu. From the selected set of faces or bodies, the software groups adjacent entities appropriately and automatically creates virtual cells for each region that meets the criteria established by the settings in step 3.

  These virtual cells become invalid after a geometry update.

Creating and Managing Virtual Split Edges

When preparing geometry for meshing, it may be advantageous to split an edge into two virtual edges separated by a vertex. For example, in the case of a rectangular face in which a single edge on one side of the face corresponds to two edges on the opposite side, you can split the single edge so that node alignment across the face can have similar spacing. This can be achieved by creating **Virtual Split Edge** objects. You can also modify the split interactively by dragging the newly created vertex along the length of the original edge. This alters the split location.

Splitting an Edge

1. Insert a Virtual Topology object in the tree.
2. Choose the edge selection filter (**Ctrl**+**E**) and then in the **Geometry** window, pick the edge that you want to split. The selected edge can be either a "real" edge or a previously-defined virtual edge.

   Note:

   To simplify specification of the split location, when picking the edge to split you should position your cursor at the point on the edge where you want the split to occur. Then select **Virtual Split Edge at +** as described in step 3.

3. Create the Virtual Split Edge using either of these methods:

   - To define the split location according to your cursor location on the edge, right-click in the **Geometry** window and select **Insert> Virtual Split Edge at +** from the context menu, or choose **Split Edge at +** on the **Virtual Topology** context toolbar.
   - To define the split without specifying the location, select the edge you want to split, right-click in the **Geometry** window, and select **Insert> Virtual Split Edge** from the context menu, or choose **Split Edge** on the **Virtual Topology** context toolbar. By default the split ratio will be set to 0.5, but you can change it later by using the **Virtual Topology Properties** dialog (p. 573).

     Note:

     If the software cannot create the split, the error message “Unable to split the edge at selected location or with given split ratio” will appear and the split will not be created.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. In step 3, if you created the **Virtual Split Edge** object by selecting **Virtual Split Edge at +**, the split ratio is determined automatically by the software. If you created it by selecting **Virtual Split Edge**, you can either accept the default of 0.5 or specify a different split ratio by right-clicking and selecting **Edit Selected Virtual Entity Properties...**. Then edit the **Split Ratio** field on the **Virtual Topology Properties** dialog.

   The split ratio defines the location of the split by specifying the ratio between the distance from the start point of the edge to the split location and the overall length of the edge. Specify a value from 0 to 1. For example, a value of 0.5 will split the edge into two edges of equal length. A value of 0.75 will split the edge into two edges where the first edge is three quarters of the length of the original edge, and the second edge is only one quarter of the length of the original edge. A value of 0 or 1 is valid only if the selected edge is a closed edge. In the case of a closed edge, such as a circle, the edge will be split into two edges, and the new vertex will be placed along one of the new edges.

   You can change the location of the split at any time by accessing the **Virtual Topology Properties** (p. 573) dialog and editing the **Split Ratio** field, or by modifying the edge split interactively.

   To modify an edge split interactively, select any portion of the original edge or its vertex split location in the **Geometry** view. Then, while pressing **F4** on the keyboard, drag the mouse along the length of the edge to redistribute the split ratio. The display indicates the initial 3-D location of the split, together with a preview of its new location and split ratio.

   When you change the split ratio of an edge split that is attached to a face split, both the edge split and the face split are adjusted accordingly.

Virtual Split Edge Dependency

Existing virtual edges and/or existing virtual split edges can be used to create new virtual split edges. Within the split edge hierarchy, a virtual split edge depends on a virtual edge if the latter is used to create the former. Similarly, a virtual split edge depends on another virtual split edge if the latter is used to create the former.

Note:

If a virtual edge was created from a virtual split edge, you cannot delete the virtual split edge without first deleting the virtual edge. Conversely, if a virtual split edge was created from a virtual edge, you cannot delete the virtual edge without first deleting the virtual split edge.

A warning message appears in the **Messages** window for each failed deletion. To highlight the geometry that is responsible for a message, select the message, right-click, and select **Show Problematic Geometry** from the context menu.

Locking Locations of Dependent Virtual Split Edges

In cases involving virtual split edge dependency, you can choose whether the locations of dependent splits are locked or unlocked when the location of the parent split is modified. If unlocked, the location of the dependent split moves to maintain its defined split ratio when the parent is modified. If locked, the location of the dependent split persists when the parent is modified. Regardless of the setting, the

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

location of the dependent split will move if preserving its location would invalidate the split (see Figure 227: Overridden Locked Dependent Splits (p. 564) for an example).

Note:

For parametric updates, all virtual split edges are treated as unlocked.

1. Highlight the Virtual Topology object in the tree.

Note:

Locking must be set globally on the **Virtual Topology** object. It cannot be set locally for individual **Virtual Split Edge** objects.

2. In the Details View, set the **Lock position of dependent edge splits** option to **Yes** (default) or **No**.

The following figures illustrate locking. Figure 224: Original Virtual Split Edge with Dependent Virtual Split Edge (p. 563) shows an example in which a split at .5 was defined on original edge **ab**, creating two new edges—**ac** and **cb**. A second split at .5 was then defined on edge **cb**, creating two new edges—**cd** and **db**.

Figure 224: Original Virtual Split Edge with Dependent Virtual Split Edge

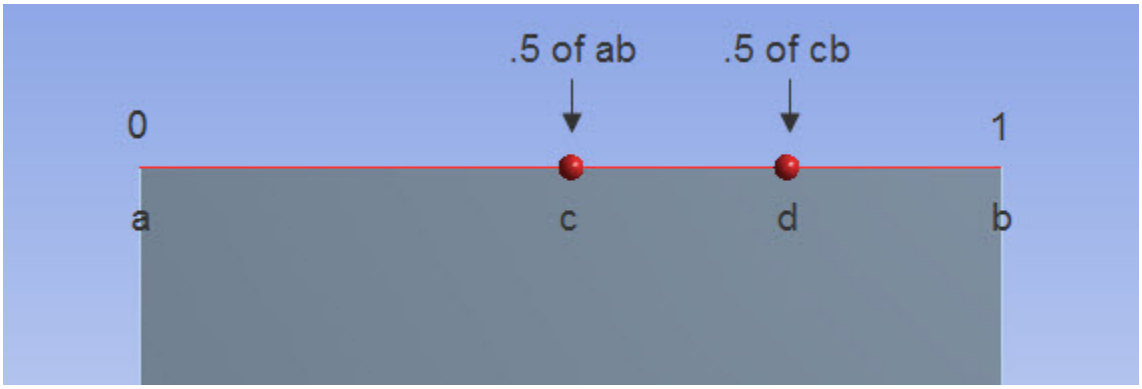


Figure 225: Unlocked Dependent Splits (p. 564) shows the expected behavior when **Lock position of dependent edge splits** is set to **No**, and the split located at point **c** is changed from .5 to .1. Notice that point **d** has moved to maintain its defined split ratio at .5 of **cb**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 225: Unlocked Dependent Splits

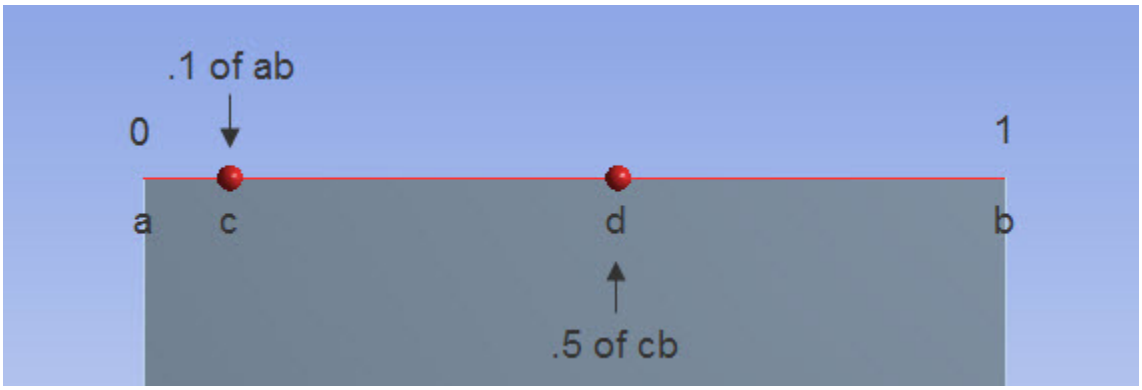
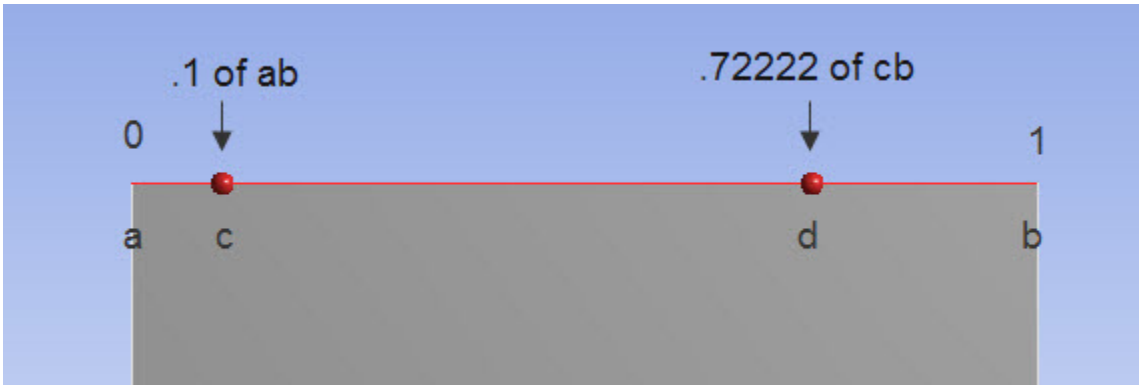


Figure 226: Locked Dependent Splits (p. 564) shows the expected behavior when **Lock position of dependent edge splits** is set to **Yes**, and the split located at point **c** is changed from .5 to .1. Notice that in this case point **d** has not moved, and the split ratio is now .72222 of **cb**.

Figure 226: Locked Dependent Splits



In Figure 227: Overridden Locked Dependent Splits (p. 564), the split located at point **c** was changed from .5 to .9. Even with locking set to **Yes**, in this case point **d** was moved because preserving its location would have invalidated the split (that is, if its original location had been preserved, point **d** would no longer be located on edge **cb**).

Figure 227: Overridden Locked Dependent Splits

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes on Virtual Split Edges

- When the **Sweep** mesh method (p. 235) is used, the guiding edges must have consistent lengths to obtain a regular mesh in the sweep direction. You can define virtual split edges to achieve consistent lengths for these edges.
- If your **Geometry** view is configured to display **Shaded Exterior and Edges**, you can conveniently review virtual split edges by clicking the **Virtual Topology** group in the Tree Outline. In this mode, split edges will be highlighted in two different colors (automatically assigned) to draw attention to the splits.
- You cannot use the virtual split edge feature to split edges belonging to line bodies.

Creating and Managing Virtual Split Faces

When preparing geometry for meshing, it may be advantageous to split a face. This can be achieved by creating **Virtual Split Face** objects, which allow you to split a face along two vertices to create 1 to N virtual faces. The selected vertices must be located on the face that you want to split.

Splitting a Face

1. Insert a Virtual Topology object in the tree.
2. Choose the vertex selection filter and then in the **Geometry** window, pick two vertices on the face that you want to split.

   Note:

   - The vertices must be attached to the same face and in the same part.
   - Optionally, you can create one or more Virtual Hard Vertex objects to facilitate the split face operation. **Virtual Hard Vertex** objects allow you to define a hard point according to your cursor location on a face, and then use that hard point in the split face operation. Refer to Creating and Managing Virtual Hard Vertices (p. 568).
   - To see all vertices in the **Geometry** window, including any virtual hard vertices, make sure that the **Show Vertices** option is enabled.

3. Create the Virtual Split Face by doing one of the following:
   - Choose **Split Face at Vertices** on the **Virtual Topology** context toolbar.
   - Click right mouse button on the Virtual Topology object and select **Insert> Virtual Split Face at Vertices** from the context menu.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click right mouse button in the **Geometry** window and select **Insert> Virtual Split Face at Vertices** from the context menu.

Note:

- If the software cannot create the split, the error message "Unable to split face at selected vertices, please ensure vertices are attached to the same face and in the same part" will appear and the split will not be created.
- To change the location of a virtual hard vertex interactively, select it in the **Geometry** view. Then, while pressing **F4** on the keyboard, drag the mouse to move the vertex to the desired location. You cannot move the vertex beyond the face on which it was created. The display indicates the initial 3-D location of the vertex, together with a preview of its new location.

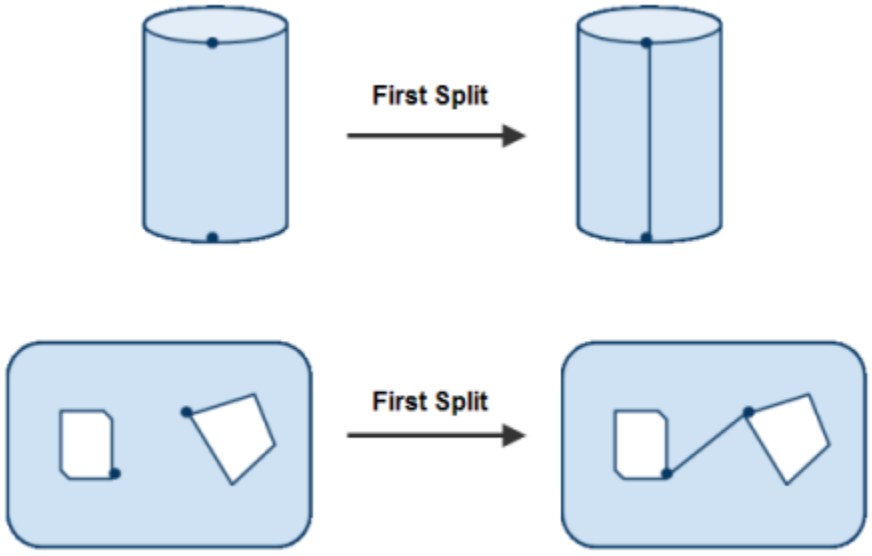
  When you change the location of a virtual hard vertex, the corresponding face split is adjusted accordingly.

  You cannot use the **Virtual Topology Properties** dialog to change the location of a virtual hard vertex.

Special Considerations for Virtual Split Faces

Certain types of faces cannot be split by a single split operation; they require two splits. For example, refer to the cylindrical face in the figure below. As a result of the first split, a seam edge is created and the face is no longer a periodic face. To actually split the cylindrical face, you then must select two more vertices and split the face a second time.

Figure 228: Types of Faces Requiring Two Virtual Split Face Operations

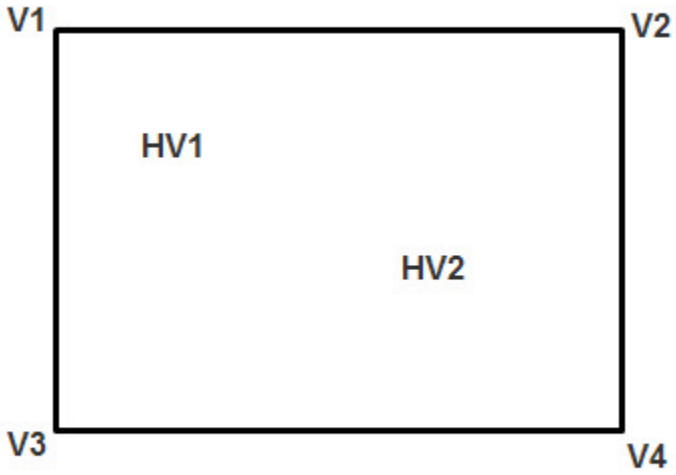


Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To split a face along more than two vertices you must perform a series of split face operations using two vertices at a time. For example, refer to the figure below, where V1, V2, V3, and V4 are vertices, and HV1 and HV2 are virtual hard vertices. Splitting the face through V1, HV1, HV2, and V4 would require three split operations:

1. Select V1 and HV1 and split the face.
2. Select HV1 and HV2 and split the face.
3. Select HV2 and V4 and split the face.

Figure 229: Splits Requiring a Series of Virtual Split Face Operations



Virtual Split Face Dependency

Existing virtual faces and/or existing virtual split faces can be used to create new virtual split faces. Within the split face hierarchy, a virtual split face depends on a virtual face if the latter is used to create the former. Similarly, a virtual split face depends on another virtual split face if the latter is used to create the former.

Faces can be split by selecting vertices of virtual split edges, providing the vertices meet the requirements described in the procedure above.

Note:

If a virtual face was created from a virtual split face, you cannot delete the virtual split face without first deleting the virtual face. Conversely, if a virtual split face was created from a virtual face, you cannot delete the virtual face without first deleting the virtual split face.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If a virtual split face was created from vertices of virtual split edges, you cannot delete the virtual split edges without first deleting the virtual split face.

If a virtual split face was created from virtual hard vertices, you cannot delete the virtual hard vertices without first deleting the virtual split face.

A warning message appears in the **Messages** window for each failed deletion. To highlight the geometry that is responsible for a message, select the message, right-click, and select **Show Problematic Geometry** from the context menu.

Creating and Managing Virtual Hard Vertices

You can create one or more Virtual Hard Vertex objects to facilitate a split face operation (p. 565). **Virtual Hard Vertex** objects allow you to define a hard point according to your cursor location on a face, and then use that hard point in the split face operation.

Creating a Virtual Hard Vertex

1. To create a virtual hard vertex (hard point), choose the face selection filter (**Ctrl**+**F**) and then in the **Geometry** window, pick the face that you want to split. Position your cursor on the face where you want the hard point to be located, left-click, and do one of the following:

   - Right-click in the **Geometry** window and select **Insert> Virtual Hard Vertex at +** from the context menu.

   - Choose **Hard Vertex at +** on the **Virtual Topology** context toolbar.

Hanging Edges Resulting From Split Faces

It is important to understand the effect hard vertices and hanging edges can have on meshing. If a face is not completely split, the partial face split is treated as a hanging edge and can create constraints on the mesher. If a virtual hard vertex is created but is not used for a split (or it comes in from the geometry), it can also create constraints on the mesher.

Common Virtual Topology Operations

This section describes common virtual topology operations.

Importing Legacy Models

Upon import of a legacy model into release 2025 R1, suppressed virtual topology entities will be deleted. This includes any virtual topology entities that were suppressed manually (for example, by right-clicking on the virtual topology entity in the Tree Outline and selecting **Suppress** from the context menu), but it does not include virtual topology entities that are suppressed because the body containing them is suppressed. If entities are deleted, a warning message will be issued advising you to import the model into an earlier release, unsuppress the affected entities, and save the model for use in release 2025 R1.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Handling of Geometry Changes That Result in Incomplete Virtual Entities

The Virtual Topology object that appears in the Tree Outline represents all definitions of virtual face or virtual edge groups, and all definitions of virtual split edges, virtual split faces, and virtual hard vertices within a model. Individual objects for these virtual entities do not appear in the Tree. If a geometry operation invalidates a virtual entity, refreshing the geometry does not cause the **Virtual Topology** object in the Tree Outline to become underdefined. For example, if you include a fillet and one neighboring face in the creation of a virtual cell, but later remove the fillet from the CAD model and refresh the geometry, that individual virtual cell will become underdefined (as it only includes the one neighboring face), but it will not be deleted, and there will be no change in the Tree Outline. If in a later operation, the fillet is re-added to the CAD model and refreshed, the virtual cell will be restored. When a virtual entity becomes underdefined due to a geometry operation, a message is issued. You might be able to view partial Virtual Entities by right-clicking on the **Show Problematic Geometry** message.

Underdefined Virtual Topologies are not displayed in Virtual Topology graphics, nor are they included in Virtual Topology statistics.

Note:

If you suppress a part or all bodies in a multibody part in the DesignModeler application, and refresh the geometry in the Meshing application, any virtual topology that had been defined on those bodies will be removed. The virtual topology will not be removed if only some bodies within a multibody part are suppressed in DesignModeler.

Using Named Selections with Virtual Topology

If you are using Named Selections with virtual topology and you subsequently modify the virtual topology, you must manually update the Named Selections. For example, if you create a Named Selection to define local inflation and then define virtual topology on that Named Selection, you must update the Named Selection before generating the mesh or the inflation will not be defined correctly. This limitation does not always occur if you perform a similar operation using the DesignModeler application. For example, after you perform a merge operation in DesignModeler, the software may be able to relink the Named Selection to the topology automatically when the geometry is refreshed.

Cycling Through Virtual Entities in the Geometry Window

You can use the ← and → buttons on the **Virtual Topology** context toolbar to cycle through virtual topology entities in the sequence in which they were created and display them in the **Geometry** window.

1. In the **Geometry** window, select a virtual entity.
2. On the **Virtual Topology** context toolbar, click ←or →.

Remember the following information when using ← and →:

- ← and → are grayed out until at least one virtual entity has been defined.
- If no virtual entities are selected, clicking →displays the first virtual entity in the sequence and clicking ← displays the last virtual entity in the sequence.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If the currently selected virtual entity is the last in the sequence, clicking → displays the first virtual entity in the sequence. If the currently selected virtual entity is the first in the sequence, clicking ← displays the last virtual entity in the sequence.
- If using → when multiple virtual entities are selected, the entity that has the highest ID (based on the order in which the entities were created) is considered to be the current selection, and clicking → displays the entity that follows it.
- If using ← when multiple virtual entities are selected, the entity that has the lowest ID (based on the order in which the entities were created) is considered to be the current selection, and clicking ← displays the entity that precedes it.
- If the current multiple selection contains no virtual entities, ← and → work as though there are no selections (That is, clicking → displays the first virtual entity in the sequence and clicking ← displays the last virtual entity in the sequence). If the current multiple selection contains a mixture of virtual entities and non-virtual entities, the non-virtual entities are ignored.
- If a split is selected, both edges or all faces of the split are displayed.
- If any virtual entities are deleted or merged, the sequence is adjusted automatically. For example, the following behaviors occur if you create 14 virtual entities and then perform these actions in order:

  1. If you select the fifth virtual entity and click →, the sixth virtual entity is displayed.
  2. Click ←, and the fifth virtual entity is displayed.
  3. Delete the sixth virtual entity, select the fifth virtual entity, and click →. The seventh virtual entity is displayed. This occurs because the sixth virtual entity was deleted.
  4. Merge the seventh and eighth virtual entities, which creates a fifteenth virtual entity.
  5. Select the fifth virtual entity and click →. The ninth virtual entity is displayed. This occurs because the sixth virtual entity was deleted and the seventh and eighth were merged to form a new entity.

Deleting All Virtual Entities

You can use the **Delete All Virtual Entities** option to delete all virtual topology entities from a model at one time, regardless of type.

1. Highlight the **Virtual Topology** object in the Tree Outline or select any virtual topology entity in the **Geometry** window.
2. Right-click and select **Delete All Virtual Entities**.
3. Answer **Yes** at the prompt.

Deleting All Virtual Cells

You can use the **Delete All Virtual Cells** option to delete all virtual topology cells (the virtual topology group itself, including any virtual split edges, virtual split faces, or virtual hard vertices, will not be deleted). This option is available only when the **Virtual Topology** object in the Tree Outline is highlighted

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and virtual cells, as well as virtual split edges, virtual split faces, and/or virtual hard vertices exist. Otherwise, use the **Delete All Virtual Entities** option.

1. Highlight the **Virtual Topology** object in the Tree Outline.
2. Right-click the **Virtual Topology** object in the Tree Outline or in the **Geometry** window. Select **Delete All Virtual Cells**.
3. Click **Yes** at the prompt.

Deleting All Virtual Split Edges

You can use the **Delete All Virtual Split Edges** option to delete all virtual split edges (the virtual topology group itself, including any virtual cells, virtual split faces, or virtual hard vertices, will not be deleted). This option is available only when the **Virtual Topology** object in the Tree Outline is highlighted and virtual split edges, as well as virtual split faces and/or virtual hard vertices exist. Otherwise, use the **Delete All Virtual Entities** option.

1. Highlight the **Virtual Topology** object in the Tree Outline.
2. Right-click the **Virtual Topology** object in the Tree Outline, or in the **Geometry** window. Select **Delete All Virtual Split Edges**.
3. Answer **Yes** at the prompt.

Deleting All Virtual Split Faces

You can use the **Delete All Virtual Split Faces** option to delete all virtual split faces (the virtual topology group itself, including any virtual cells, virtual split edges, or virtual hard vertices, will not be deleted). This option is available only when the **Virtual Topology** object in the Tree Outline is highlighted and virtual split faces, as well as virtual split edges and/or virtual hard vertices exist. Otherwise, use the **Delete All Virtual Entities** option.

1. Highlight the **Virtual Topology** object in the Tree Outline.
2. Right-click the **Virtual Topology** object in the Tree Outline, or in the **Geometry** window. Select **Delete All Virtual Split Faces**.
3. Click **Yes** at the prompt.

Deleting All Virtual Hard Vertices

You can use the **Delete All Virtual Hard Vertices** option to delete all virtual hard vertices (the virtual topology group itself, including any virtual cells, virtual split faces, or virtual split edges, will not be deleted). This option is available only when the **Virtual Topology** object in the Tree Outline is highlighted and virtual hard vertices, as well as virtual split faces and/or virtual split edges exist. Otherwise, use the **Delete All Virtual Entities** option.

1. Highlight the **Virtual Topology** object in the Tree Outline.
2. Right-click the **Virtual Topology** object in the Tree Outline or in the **Geometry** window. Select **Delete All Virtual Hard Vertices**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **Yes** at the prompt.

Deleting Selected Virtual Entities

You can use the **Delete Selected Virtual Entities (And Dependents)** option to delete selected virtual topology entities, along with any dependents if applicable.

To use a right mouse button click:

1. Highlight any object in the Tree Outline (for example, the **Geometry** or **Mesh** object).
2. In the **Geometry** window, select the virtual entities that you want to delete.
3. Right-click and select **Delete Selected Virtual Entities (And Dependents)**.
4. Answer **Yes** at the prompt.

To use the **Virtual Topology** context toolbar:

1. Highlight the **Virtual Topology** object in the Tree Outline.
2. In the **Geometry** window, select the virtual entities that you want to delete.
3. Choose **Delete** on the **Virtual Topology** context toolbar.
4. Click **Yes** at the prompt.

Also see:

- Creating and Managing Virtual Cells (p. 546)
- Creating and Managing Virtual Split Edges (p. 561)
- Creating and Managing Virtual Split Faces (p. 565)
- Creating and Managing Virtual Hard Vertices (p. 568)

Common Virtual Topology Features

This section describes features that are common to all types of virtual topology.

Setting Advanced Properties

The following Advanced settings are available in the Details View for **Automatic** and **Repair** modes.

- **Generate on Update** – Sets whether you want to include the settings in this Details View when you update the geometry.
- **Simplify Faces** – Removes hard edges and hard vertices from the selection.
  - **Yes**: If the **Simplify Faces** property is turned on, the program removes hard edges and hard vertices.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **No**: If the **Simplify Faces** property is turned off, faces are not simplified.

- **Merge Face Edges** – The property is relevant only during the virtual face creation process. It applies only to manually-created virtual faces and can be modified at any time, but the modification will have no effect on previously-created virtual faces.

  - **Yes**: If the **Merge Face Edges** property is turned on, the program will attempt to merge bounding edges of a newly manually-created virtual face and create virtual edges. The criterion used to merge edges is based on the **Behavior** setting.

  - **No**: If the **Merge Face Edges** property is turned off, only a virtual face will be created out of selected faces.

- **Lock position of dependent edge splits** – See Locking Locations of Dependent Virtual Split Edges (p. 562).

- **Virtual Faces**, **Virtual Edges**, **Virtual Split Edges**, **Virtual Split Faces**, **Virtual Hard Vertices**, **Total Virtual Entities** – Read-only indications of corresponding counts in the model. See Viewing Virtual Topology Statistics (p. 576).

Using the Virtual Topology Properties Dialog to Edit Properties

You can use the **Virtual Topology Properties** dialog to edit the properties of selected virtual topology entities.

To use a right mouse button click:

1. Highlight any object in the Tree Outline (for example, the **Geometry** or **Mesh** object).
2. In the **Geometry** window, select the virtual entities whose properties you want to edit.
3. Right-click and select **Edit Selected Virtual Entity Properties...**.
4. Make the desired changes in the **Virtual Topology Properties** dialog.
5. To apply any changes and/or exit, press **Enter** on your keyboard or click **X** on the dialog.

To use the **Virtual Topology** context toolbar:

1. Highlight the **Virtual Topology** object in the Tree Outline.
2. In the **Geometry** window, select the virtual entities whose properties you want to edit.
3. Choose **Edit** on the **Virtual Topology** context toolbar.
4. Make the desired changes in the **Virtual Topology Properties** dialog.
5. To apply any changes and/or exit, press **Enter** on your keyboard or click **X** on the dialog.

Remember the following information when using the **Virtual Topology Properties** dialog:

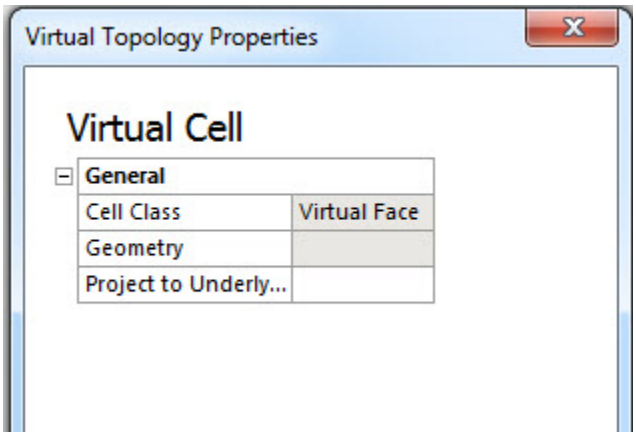
- If all selected virtual entities have the same value for a particular property, that value appears in the **Virtual Topology Properties** dialog. Otherwise, the value for that property is blank.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Fields that are grayed out are read-only.
- The changes you make in the **Virtual Topology Properties** dialog will be applied to all selected virtual entities.
- If you change a split location, the graphic in the **Geometry** window will be redrawn.

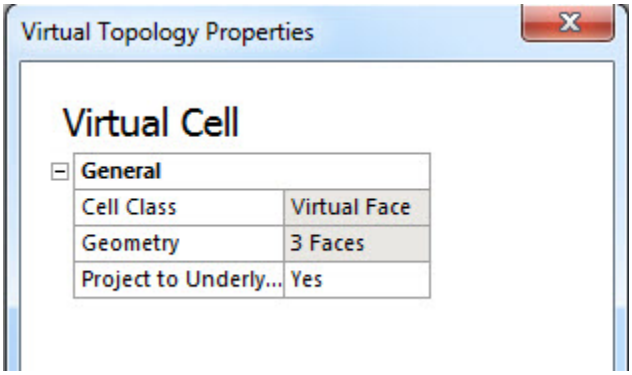
Consider the examples below. For the first example, two virtual faces were selected. One virtual face was composed of five faces, and its **Project to Underlying Geometry** option was set to **No**. The other virtual face was composed of three faces, and its **Project to Underlying Geometry** option was set to **Yes**.

Figure 230: Virtual Topology Properties Dialog: Example 1



In the example below, two virtual faces were selected. In this case each virtual face was composed of three faces, and **Project to Underlying Geometry** was set to **Yes** for both virtual faces.

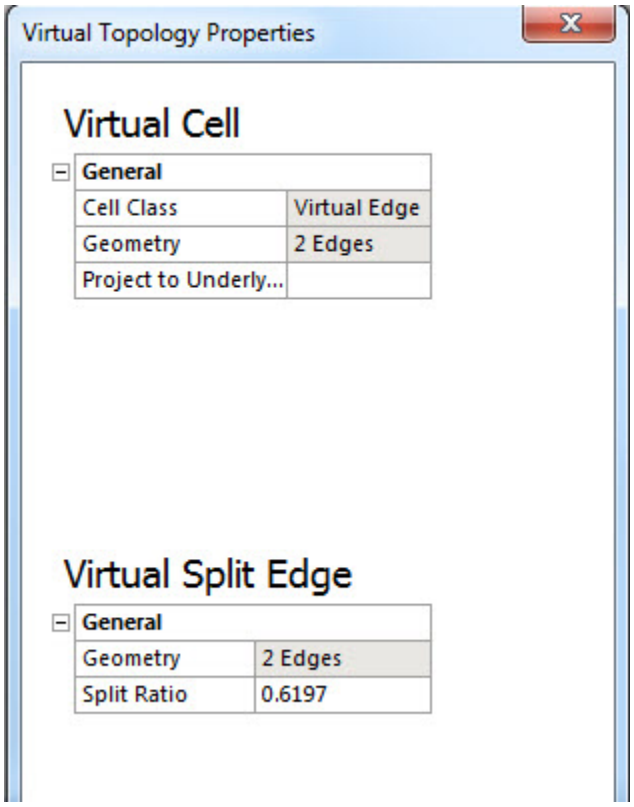
Figure 231: Virtual Topology Properties Dialog: Example 2



In the example below, two virtual edges and one virtual split edge were selected. Both virtual edges were composed of two edges, but **Project to Underlying Geometry** was set to **Yes** for one virtual edge and to **No** for the other.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

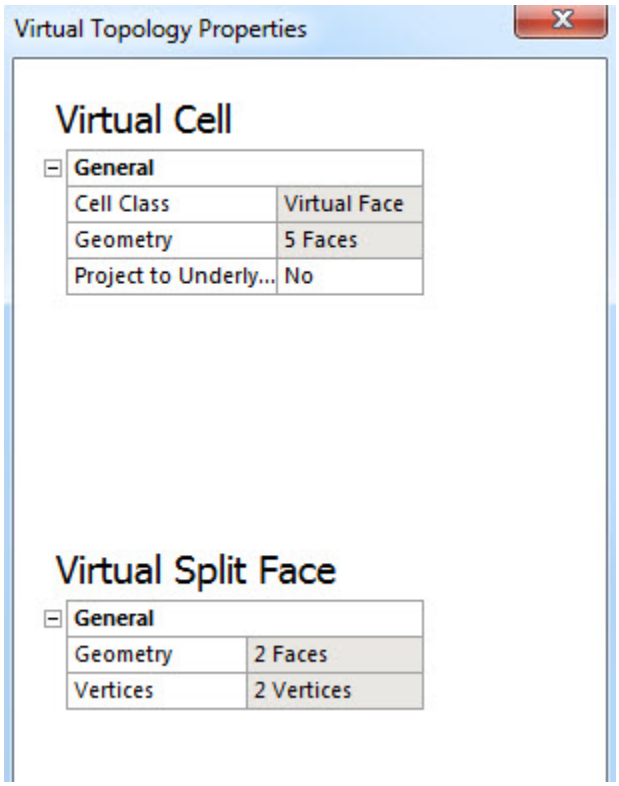
Figure 232: Virtual Topology Properties Dialog: Example 3



In the example below, one virtual face and one virtual split face were selected. The virtual face was composed of five faces, and its **Project to Underlying Geometry** option was set to **No**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 233: Virtual Topology Properties Dialog: Example 4

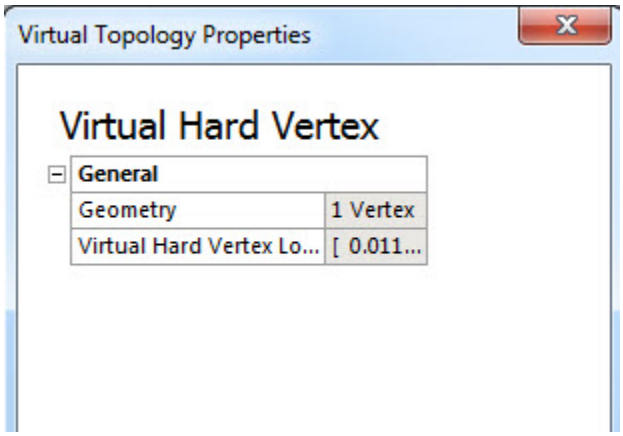


In the example below, one virtual hard vertex was selected.

Note:

Although you cannot use the dialog to modify the **Virtual Hard Vertex Location**, you can do so interactively using the **F4** key. See Creating and Managing Virtual Split Faces (p. 565) for details.

Figure 234: Virtual Topology Properties Dialog: Example 5



Viewing Virtual Topology Statistics

The Details View of the **Virtual Topology** object provides counts of the following:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Virtual faces
- Virtual edges
- Virtual split edges
- Virtual split faces
- Virtual hard vertices
- Total virtual entities

Remember the following information when viewing virtual topology statistics:

- Statistics are updated whenever a geometry or virtual topology change occurs.
- If any body is suppressed, the virtual topology entities on that body are not counted.
- If you create a virtual topology entity and then use it to create another virtual topology entity, the former exists in the background in a suppressed state and is not counted.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Meshing: Troubleshooting

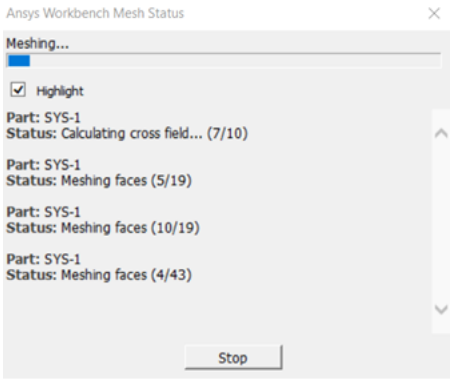
This section is intended to provide you with tips and strategies for avoiding and handling problems that may occur when using the Meshing application. Topics include:

- Meshing Process (p. 580)
- Recommended First Course of Action for Meshing Failures (p. 580)
- Understanding Messaging (p. 582)
- Identifying Poor Quality Mesh (p. 580)
- Using Named Selection Diagnostics to Find Meshing Issues (p. 596)
- Tips for Using Virtual Topology (p. 597)
- Tips for Successful Meshing Using Local Controls (p. 597)
- Handling Patch Conforming Tetrahedral, Hex Dominant, Quad Dominant, and All Triangle Meshing Failures (p. 589)
- Handling General MultiZone Meshing Failures (p. 593)
- Handling General Sweep Meshing Failures (p. 592)
- Handling Failures due to Protected Topology (p. 585)
- Handling Patch Independent Tet Meshing Failures (p. 588)
- Understanding States (p. 583)
- Shape Checks and Meshing Failures (p. 584)
- Handling Selective Meshing Failures (p. 585)
- Handling Thin Sweep Meshing Failures (p. 593)
- Handling Failed Mesh Connections (p. 596)
- Handling Failed Contact Matches (p. 596)
- Meshing Complicated Models (p. 598)
- Using a Localized Operating System on Linux (p. 600)
- Using Lustre Parallel File Systems on Linux (p. 600)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Meshing Process

When you generate mesh, the mesh is generated as a separate process per part. If there are multiple parts, each part can be meshed in parallel. While meshing, the status of meshing is displayed in the bottom left-hand side of the screen where an interrupt button is also available. When you double-click on the mesh status, Ansys WorkBench Mesh Status window displays the detailed information on the meshing progress on parallel cores.



Other information, such as the amount of memory used can be found by using a task manager.

While the mesher is working at certain points it will highlight topology (edges, faces, bodies). If the mesher gets stuck for a long time on a particular topology, you should inspect the highlighted topology and possibly merge it with another topology using virtual topology, adjust the mesh sizes in that area or return to a geometry tool to investigate and fix the problematic location.

For details about topology highlighting during the meshing process, refer to Generating Mesh (p. 528). For information about how to set the default for topology highlighting, refer to Meshing Options on the Options Dialog Box (p. 389).

Identifying Poor Quality Mesh

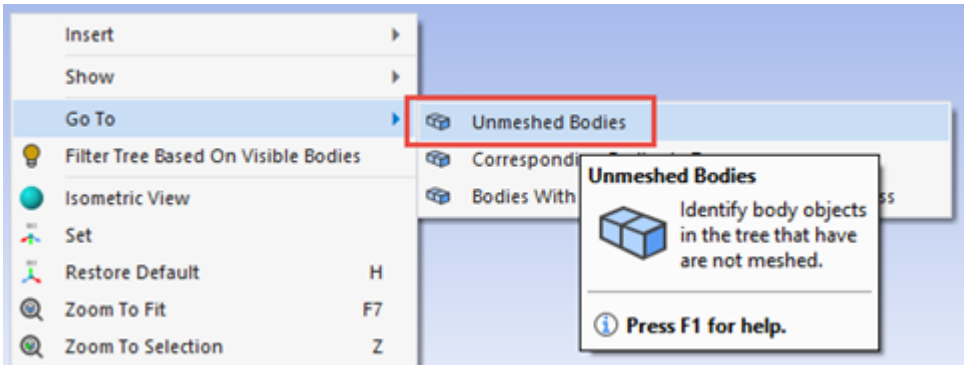
The following approaches are recommended to improve the mesh quality and obtain a valid mesh:

- Set the Display Style to identify quality by color.
- Use the **Mesh Metric** (p. 133) option to view information about a number of quality statistics and set **Min/Max** values to find poor quality elements.
- Use the Worksheet as your **Scoping Method** and use worksheet data to define criterion-based measurements.

Recommended First Course of Action for Meshing Failures

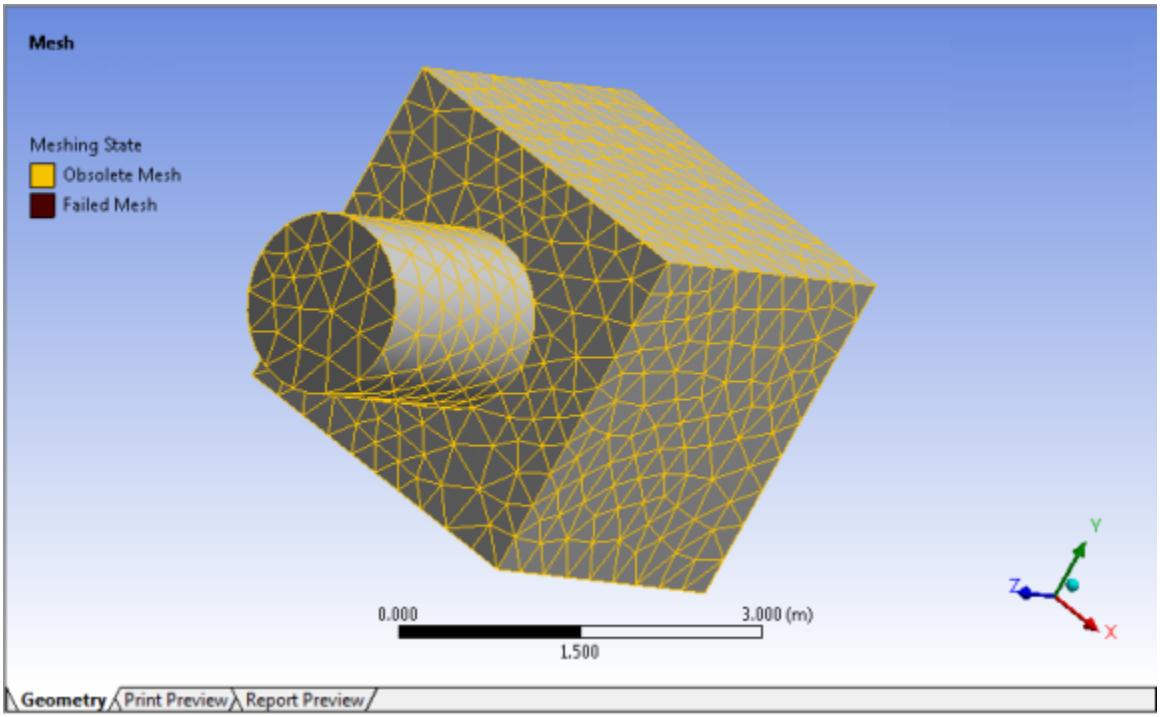
If your mesh generation fails, it may be a partial or complete meshing failure. Your first course of action should be to examine any messages that the mesher returns to the **Messages** window. The messages include hints that explain why the meshing completely or partially failed. In some cases, you can right-click the message and select **Show Problematic Geometry** to highlight any entities associated with the message in the **Geometry** window and see what the failed mesh looks like. If the geometry contains many bodies and only a few have failed to mesh, to isolate the bodies that failed to mesh, right-click the Geometry window and click **Go To** > **Unmeshed Bodies**. Other bodies can then be hidden.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



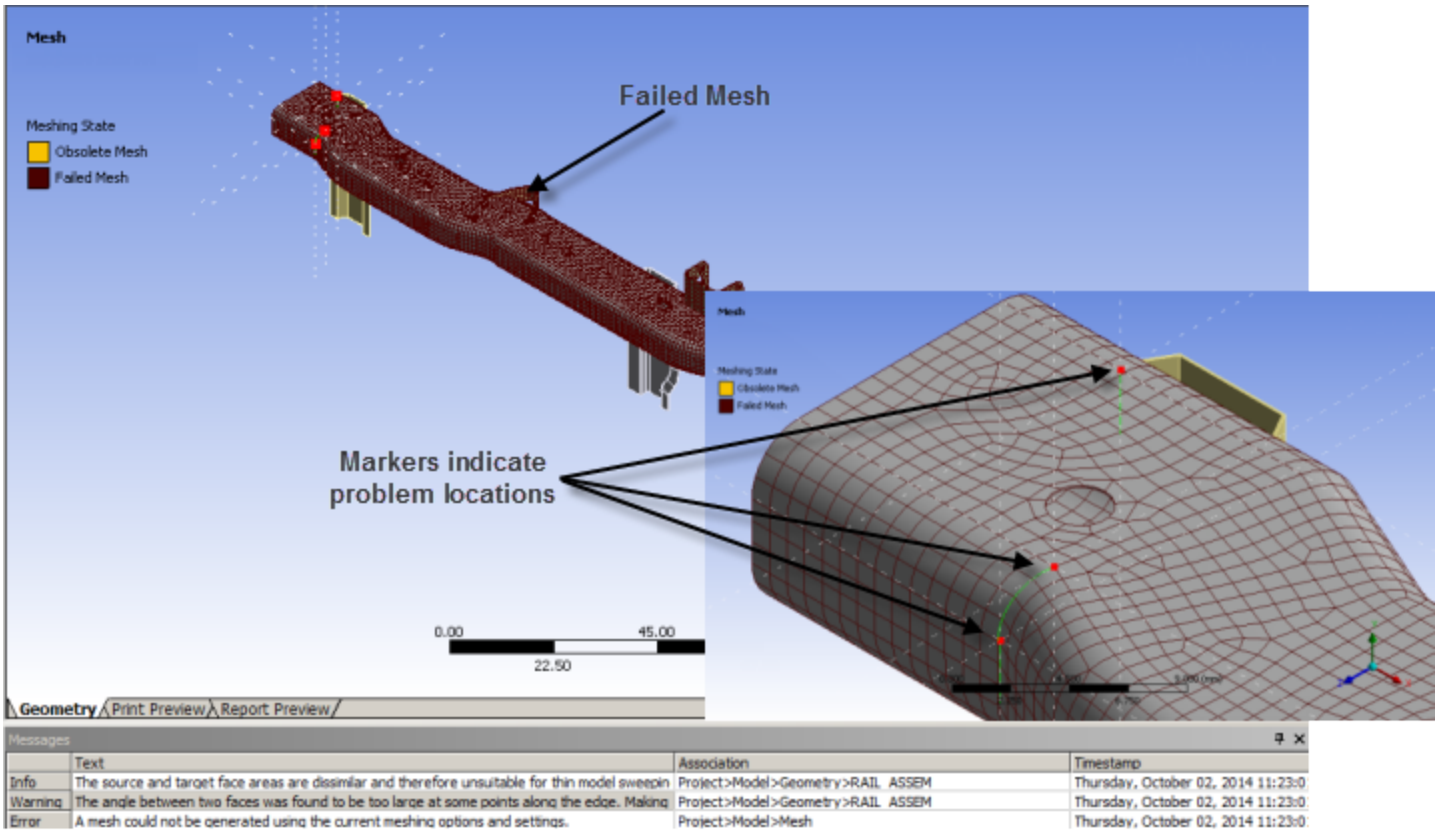
The mesher helps you to visually identify obsolete or failed meshes. As shown in the figures below, failed meshes are shaded in maroon and obsolete meshes are colored yellow. The approximate location the meshing failure is identified by a convergence of white lines. If this is not clear, color by connectivity can be enabled to identify the point at which unmeshed face edges will be marked as red (single connectivity) and if you switch to wireframe mode these locations can be seen more clearly.

Figure 235: Obsolete Mesh



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 236: Failed Mesh



If the entities are very small, you can refer to the status bar at the bottom of the window to view statistics related to the entities. Then create a Named Selection to retain information about the problematic entities. Continue reading below for more information about messaging.

Understanding Messaging

The **Messages** window prompts you with feedback concerning meshing operations. Every message returned from the mesher is not necessarily an error. Messages come in three forms:

- **Error**: Requires action from you to obtain a mesh.
- **Warning**: Guides you in best practices or alternative courses of action. A warning does not require you to fix any problems, but may give an indication of a subsequent related error.
- **Information**: Helps you make better decisions related to meshing your model or provides you with information about the approach the mesher used to mesh your model.

Once messages are displayed, you can:

- Double-click a message to display its contents in a pop-up dialog box.
- Highlight a message and then press the key combination **Ctrl+C** to copy its contents to the clipboard.
- Press the **Delete** key to remove a selected message from the window.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select one or more messages and then use the right mouse button click to display the following context menu options:
  - **Go To Object** - Selects the object in the Tree Outline that is responsible for the message.
  - **Show Problematic Geometry** - Highlights the geometry in the **Geometry** window that is responsible for the message. This option is not always available.
  - **Show Intersecting Surface Mesh** - Creates two named selection groups. One named selection group has the failed body, and the other group contains the intersecting elements displayed by the **Show Intersecting Surface Mesh**.
  - **Show Message** - Displays the selected message in a pop-up dialog box.
  - **Copy** - Copies the selected messages to the clipboard.
  - **Delete** - Removes the selected messages.
  - **Refresh** - Refreshes the contents of the **Messages** window as you edit objects in the Tree Outline.

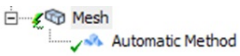
Understanding States

Sometimes the mesher returns an invalid mesh. Refer to the state of the body in the Tree Outline to determine whether a body was meshed:

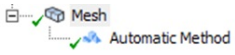
- A check mark denotes a body in a fully defined but unmeshed state.
- A check mark with a hash denotes a meshed body.
- A question mark denotes a body that needs more information before it can be sent to the solver.

When your model contains an active unmeshed body, the **Mesh** object in the Tree Outline is preceded by a lightning bolt to indicate a body is out-of-date and requires action:

When your model is in the process of being meshed, the Mesh object is preceded by a green lightening bolt to indicate that the meshing is in progress:



When your model is fully meshed (all bodies are in a meshed state), the **Mesh** object is preceded by a check mark to indicate that the meshing data is fully defined and ready for the next stage of your analysis (that is, an update in the Meshing application or a solve in the Mechanical application):



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Understanding Mesh Connection and Contact Match States

Mesh connections and contact matches use the following symbols to denote mesh state:

- A check mark denotes a connection or match that is fully defined and ready to be connected or matched.
- A check mark with a hash denotes a successful connection or match.
- A question mark denotes a connection or match that is invalid that needs more information before it can be connected or matched.
- A circle with a bar in it denotes a connection or match that is ignored. Check the Details view of the mesh connection or contact match for the reason.
- A lightning bolt indicates that the connection or match is out-of-date and requires action.
- An 'X' denotes a suppressed connection or match. Unsuppressing the connection or match requires it to be reconnected or rematched even if it was previously connected or matched.

Shape Checks and Meshing Failures

Meshing may fail if the mesh quality does not meet the criterion of the defined shape checks (p. 127). The following approaches are recommended to improve the mesh quality and obtain a valid mesh:

1. To identify faces that do not meet the shape checking criteria, right-click the warning message and select **Show Problematic Geometry**.
2. Use a different shape check setting.

   Some shape checks (p. 127) have a stricter set of criterion than others. By using a different shape check setting a mesh might be generated, and the mesh metrics bar graph (p. 133) can be used to find the mesh violating the stricter shape checks. In this way, locating the problem is the first step to fixing it.

   Note:

   You can turn off most shape checks altogether by setting Check Mesh Quality (p. 121) to **No**.

3. Use the methods described in Identifying Poor Quality Mesh (p. 580) to determine the quality of the mesh.
4. Use the **Preview Surface Mesh** (p. 531) and/or **Preview Inflation** (p. 534) features.

   With this approach the boundary mesh is generated even if the mesh would violate the defined shape checks. Once the previewed mesh is generated, use the mesh metrics bar graph (p. 133) to determine the location of bad quality elements. Generally, fixing the bad quality surface mesh is the best way to fix the volume mesh because bad quality mesh is usually a result of the geometry

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

over-constraining the mesh topology. Using defeaturing controls (such as Loop Removal (p. 196) and Mesh Defeaturing (p. 110)), pinch (p. 187) controls, and virtual topologies (p. 545) are all good strategies to remove geometry features that may cause problems for the meshing algorithms.

Note:

Not all mesh methods support the use of **Preview Surface Mesh** (p. 531) and **Preview Inflation** (p. 534).

For additional information about the shape checking acceptance criterion used by Ansys Workbench, refer to Ansys Workbench and Mechanical APDL Application Meshing Differences (p. 87).

Handling Selective Meshing Failures

Selective meshing (p. 444) may lead to unexpected results in cases where a mesh control change affects only one body. This may in turn lead to sweep mesh failure because the source and target meshes no longer align or the resultant change makes a body unsweepable. If desired, you can set the **Allow Selective Meshing** (p. 389) option to **No** to disable selective meshing and allow the mesh control changes to ripple through the entire part.

Selective meshing is not persistent for a geometry update or re-mesh operation. However, you can use the **Mesh** worksheet to create a selective meshing history so that your meshing steps can be repeated in the desired sequence. Otherwise, you may need to go through your body meshing steps manually if the single mesh update does not satisfy your meshing requirements. Refer to Using the Mesh Worksheet to Create a Selective Meshing History (p. 448) for details.

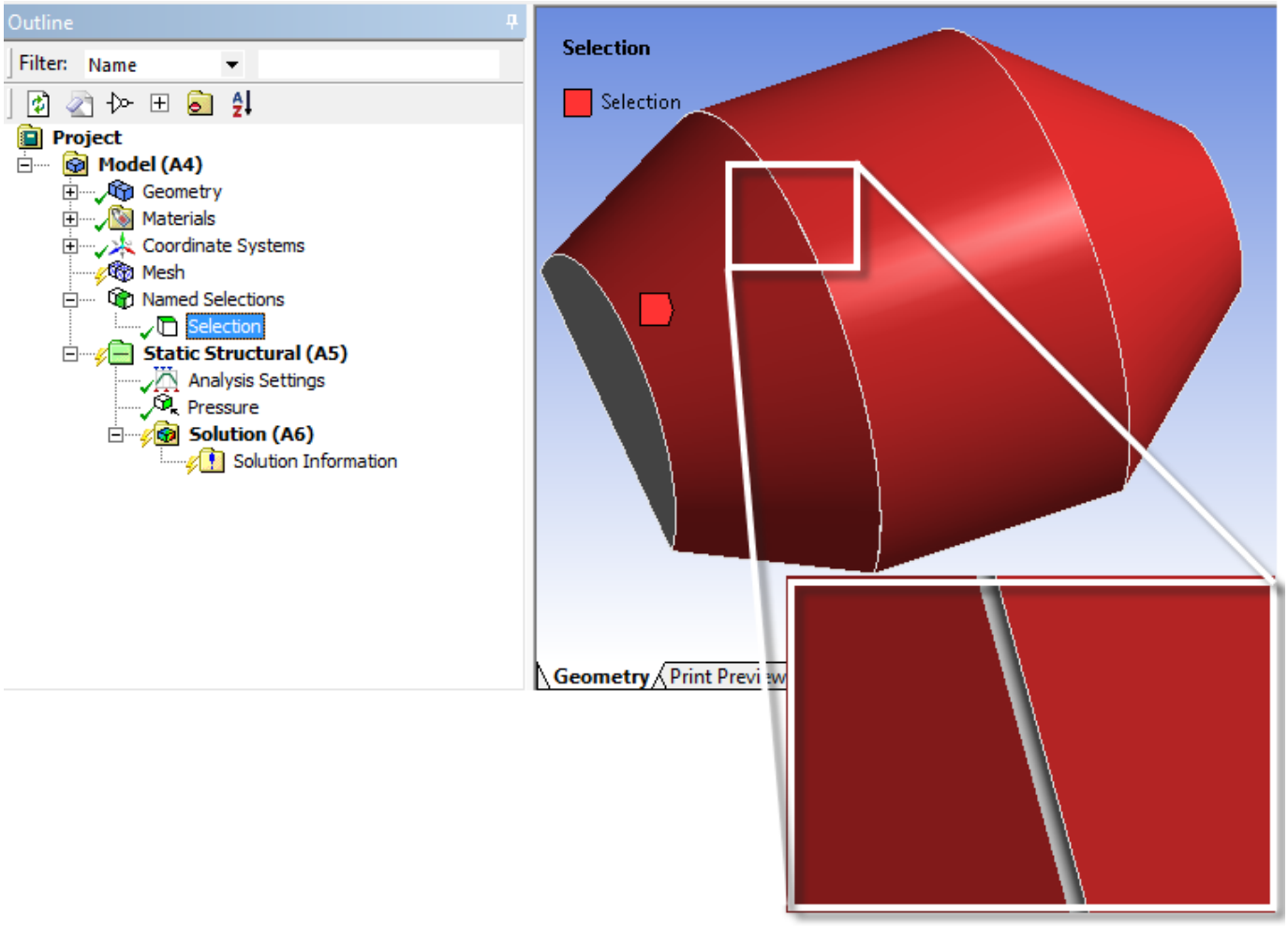
Handling Failures due to Protected Topology

Improper set up of hard protected topology (p. 184) may result in meshing failures if the mesher cannot return a reasonable mesh while also respecting the topology.

In the example, a sliver face is missing from the Named Selection defined. The **Protected** option is set to **Yes**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

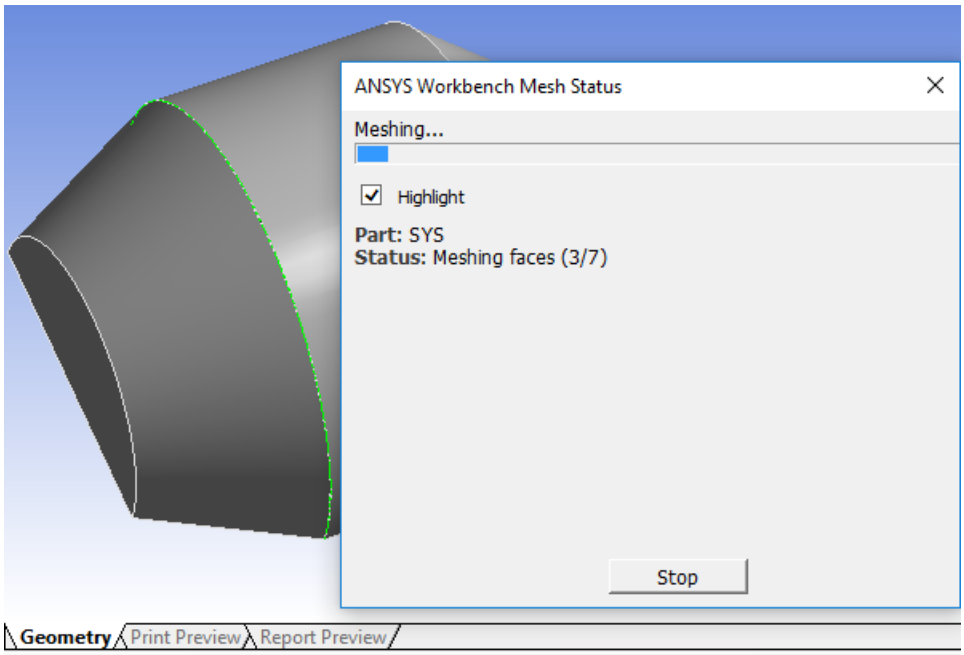
Figure 237: Example with Missing Face



The missing face could lead to problems during meshing since boundaries of hard protected topologies receive special treatment to ensure the mesh is properly associated. The special protection ensures the outer boundary is captured accurately, however, it could have a negative impact on meshing since the mesher is forced to capture the sliver surface. With the **Highlight** option enabled, the problematic face is highlighted during the meshing process (Figure 238: Problematic Topology Highlighted During Meshing (p. 587)). If a face is highlighted for a long period of time it can indicate a problem meshing the face. In this case, the problem comes from the protected topology.

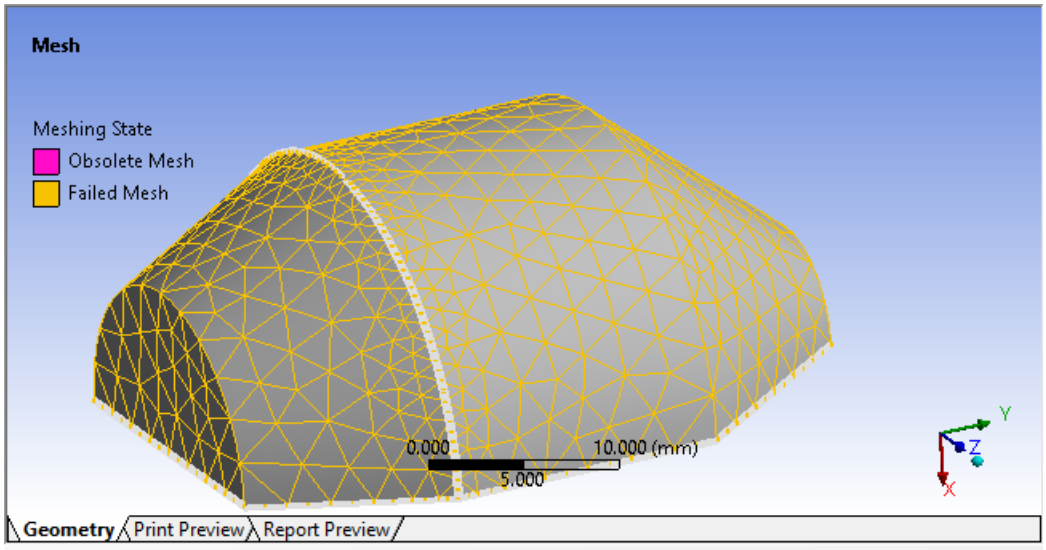
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 238: Problematic Topology Highlighted During Meshing



The missing face could also lead to other meshing failures. For example, the sizing applied may be such that the mesher cannot return a reasonable mesh while also respecting the topology. In this case, the mesher may fail in face meshing while trying to protect the boundaries of the Named Selection. The faces that fail to mesh are indicated in the message window (use the right mouse button to select the error message to show the problematic geometry).

Figure 239: Failed Surface Mesh Due to Protected Topology



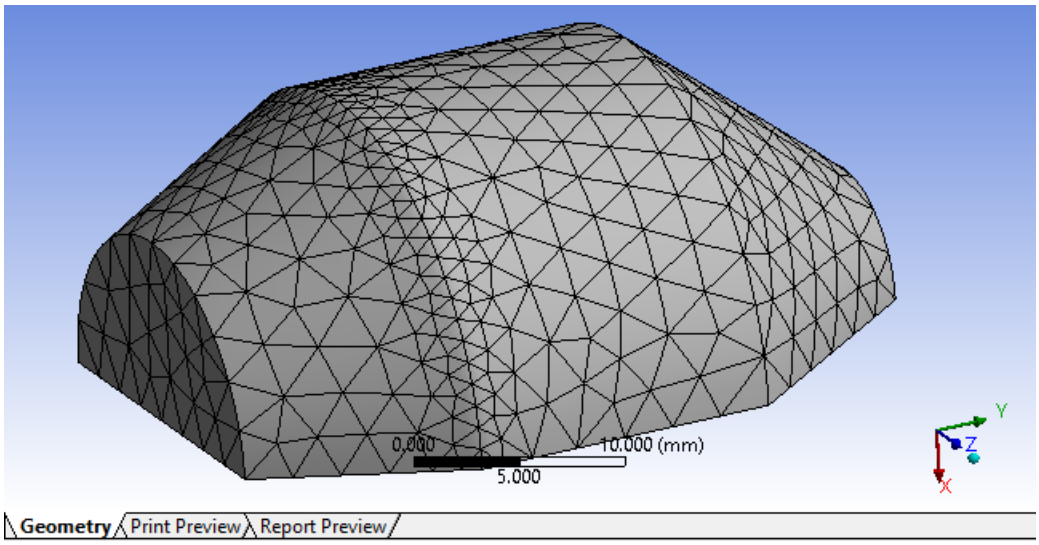
Messages

	Text	Association
Error	One or more entities failed to mesh. The mesh of the bodies containing these entities may not l	Project>Model>
Error	One or more surfaces cannot be meshed with acceptable quality. Try using a different element	Project>Model>
Error	A mesh could not be generated using the current meshing options and settings.	Project>Model>

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Examine the model to locate objects scoped to either the sliver face or the neighboring faces of the sliver face (Named Selection). By including the sliver face in the Named Selection definition, the mesh can be generated while respecting the sizing and the topology.

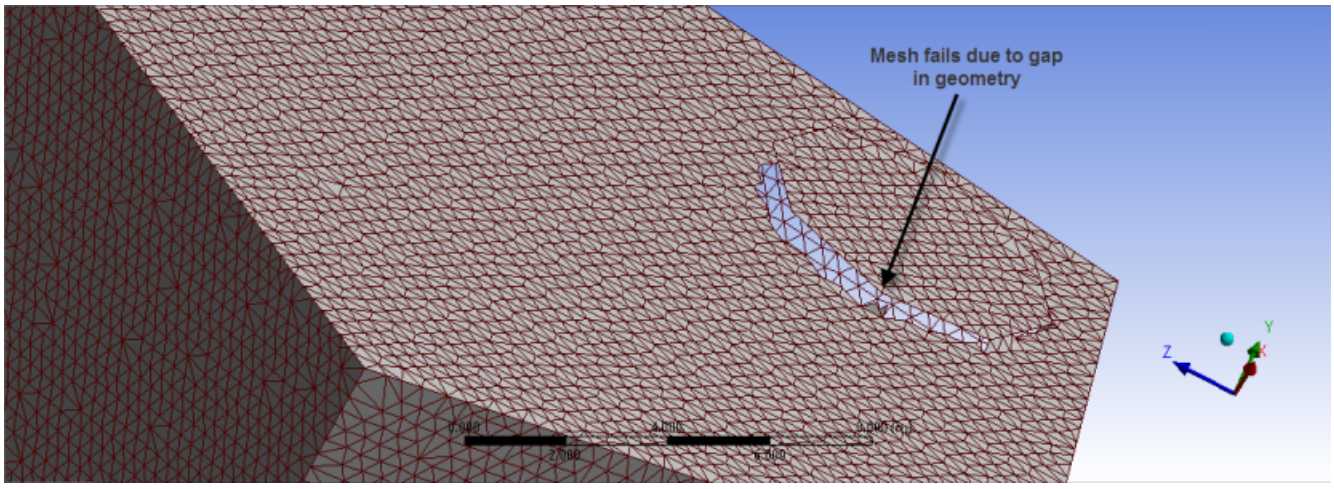
Figure 240: Mesh Respecting Protected Topology



Handling Patch Independent Tet Meshing Failures

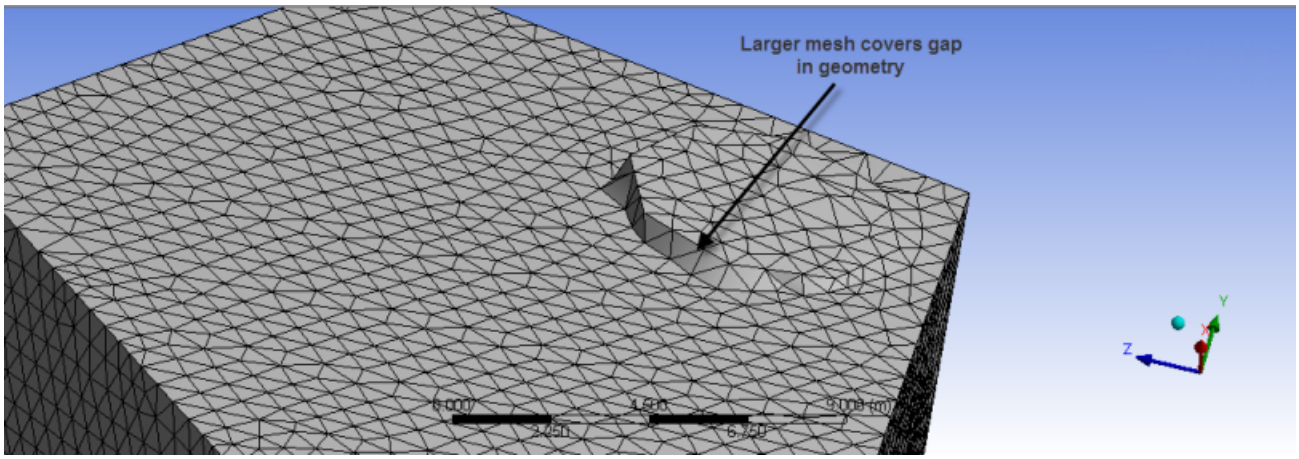
If there are gaps in your geometry and Patch Independent tet meshing fails, the mesh size may be set smaller than the size of the gaps in the geometry. In such cases, try adjusting the size of the mesh in those regions so they are larger than the geometry gap size.

Figure 241: Patch Independent Tet Mesh Failure Due to Geometry Gap



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 242: Patch Independent Tet Mesh Failure Corrected with Larger Mesh

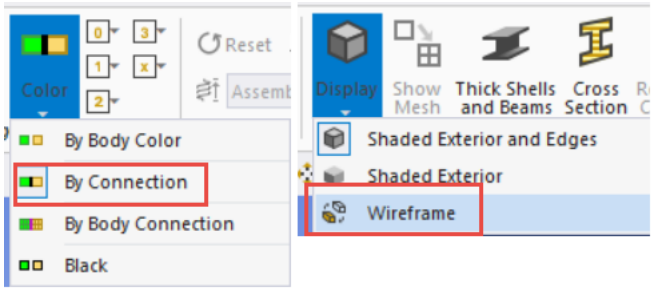


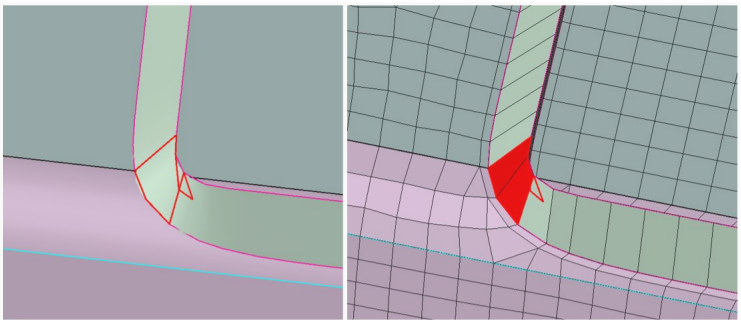
Handling Patch Conforming Tetrahedral, Hex Dominant, Quad Dominant, and All Triangle Meshing Failures

Some mesh failures are due to an inappropriate defeature size (either default or user input) or dirty geometry. Use the following guidelines to determine which issue is the cause of the failure:

1. If a message provides "Problematic Geometry" information, use it to determine which portions of the model fail. Often, one or more faces fail to mesh.

2. Use **Diagnostic** and **Visualization** Tools to help you find the problematic locations. Diagnostics for intersecting faces can point out issues where corrupted surface mesh is causing failure and can be linked to geometry corruption or defeaturing tolerances causing edge meshing issues. Switching on **Edge Coloring** > **By Connection** along with Wireframe display can highlight unmeshed faces very clearly with red edges. Diagnostics for Free Mesh Edges will also highlight these areas with Named Selections.





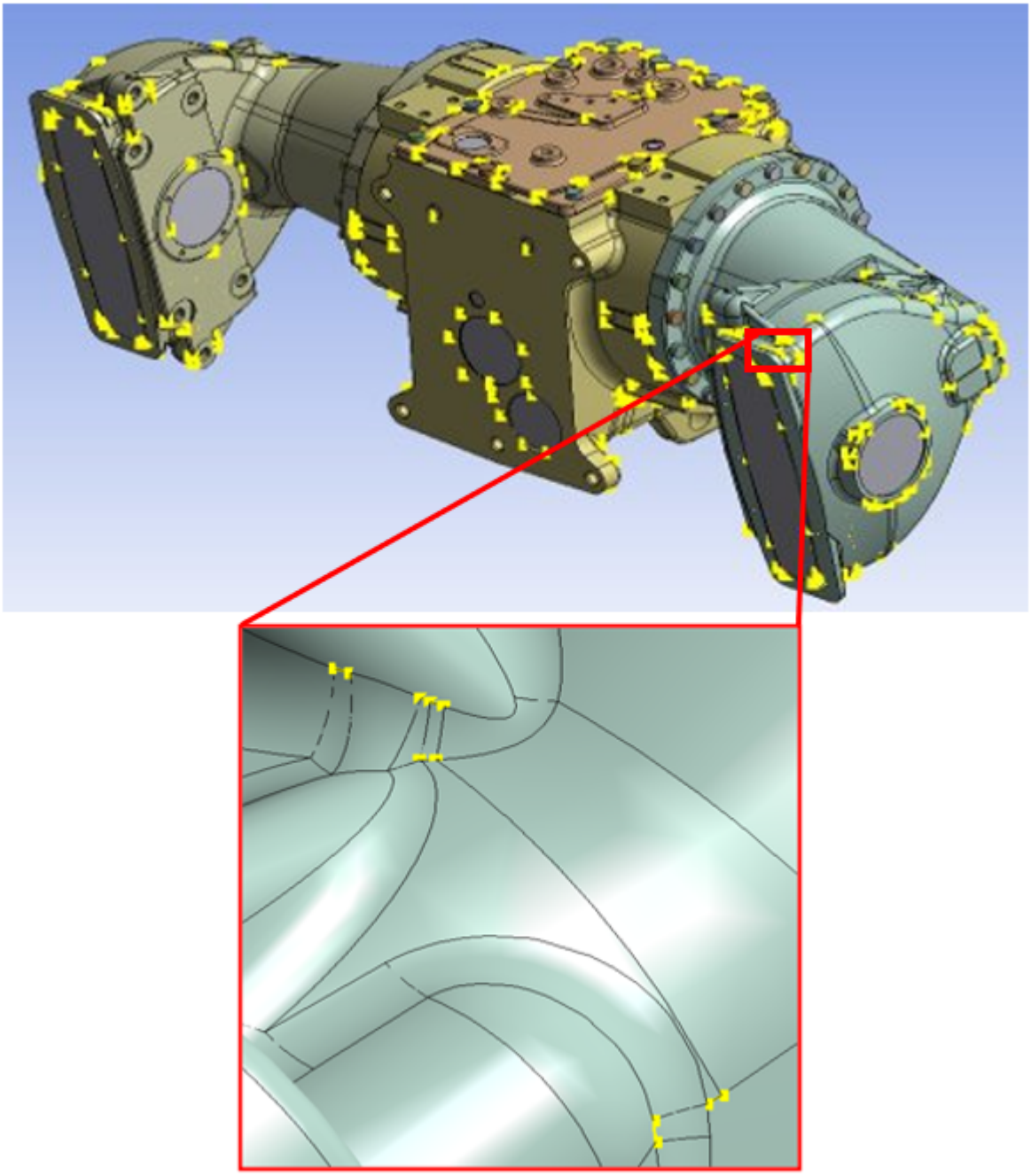
3. If a face fails to mesh, check whether the geometry looks problematic. Problems may arise if very sharp or thin faces exist or if faces are corrupted (the graphics may look strange/wrapped). If so, some options used to address the problem are as follows:

   - Add local size controls to reduce mesh size.
   - Add local size controls to modify defeaturing tolerance.
   - Merge faces locally using Virtual Topology.
   - Add a local Pinch control.
   - Modify the geometry in the problematic location using a geometry tool.

4. If a face fails to mesh but does not appear problematic at first inspection, check to see if the edges attached to that face may be problematic. For example:

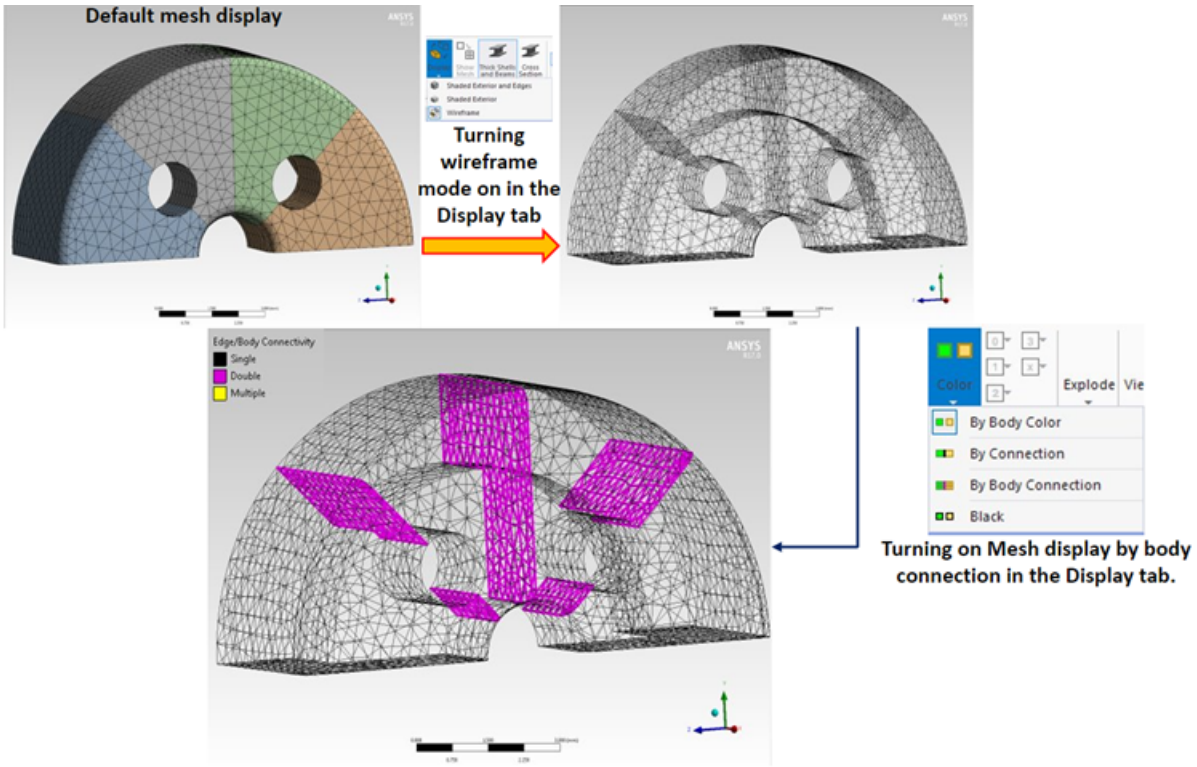
   - Turn on the **Show Vertices** and **Close Vertices** options to see if any edge is significantly faceted (that is, it has many edge splits in comparison to mesh size), or if there are any unexpected clusters of vertices. The mesher will try to respect geometric vertices, so unnecessary vertices can lead to complications in meshing that may be avoidable. Use of virtual topologies (p. 545), pinch (p. 187) controls and so on, to handle in these situations.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- For a multibody part, turn on the **Edge Coloring > By Body Connection** option to see if the edge connectivity is unusual between bodies.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Handling General Sweep Meshing Failures

In the event of a sweep mesh failure, the following approaches are recommended:

1. Check for contradicting edge sizings.
2. For **Src/Trg Selection**, use a manual setting instead of automatic.
3. Check side faces to see if they are mappable. Use of virtual topologies (p. 545) can help make bodies sweepable:
   - Turn on the **Show Vertices** and **Close Vertices** options to see if any edges have unnecessary splits. Extra edge splits can make faces that appear to be mappable more difficult to map. Virtual edges (p. 546) can help in these cases.
   - Use of virtual face splits (p. 565) can help make faces more mappable, as can use of the mapped face mesh control (p. 314) and its advanced options.
4. Turn on the **Edge Coloring > By Connection** option to see if the edge connectivity is unusual. In some cases, the geometry connectivity may not be as expected, and this may create problems during meshing. These problems could be fixed in the DesignModeler application, the CAD package, or possibly through the use of virtual topologies (p. 545).
5. For a multibody part, turn on the **Edge Coloring > By Body Connection** option to see if the edge connectivity is unusual between bodies.

For detailed information about the requirements and characteristics of sweep meshing, refer to Mesh Sweeping (p. 399).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For additional information, refer to Figure : Strategies for Avoiding Stretched Elements in the Mechanical APDL help.

Handling Thin Sweep Meshing Failures

In the event of a thin sweep mesh failure, first refer to Thin Model Sweeping (p. 406) for detailed information about the requirements and characteristics of thin sweep meshing.

The **Preview Source and Target Mesh** and **Preview Surface Mesh** features do not support the thin model sweeper. Thus, if a failure occurs, you must use the feedback in the **Messages** window to determine the problem:

- If **Src/Trg Selection** is set to **Automatic Thin**, determine whether the correct source/target faces are being used. You can review the source/target faces by right-clicking the appropriate message. If incorrect source/target faces are being used, select the **Manual Thin** option on the sweep method and pick the correct faces manually.
- In many cases, the messages tell you to use virtual topology (p. 545) to merge an edge/face. Thin sweeping requires one division through the thickness for multibody parts. Side edges must connect directly from source to target to comply with this rule.
- If a message tells you the target faces are not meshed, try swapping the source/target faces.
- In cases where a thin sweep body and a general sweep body are neighbors, the general sweep body has higher priority and is meshed first. The general sweep operation may place nodes on the side area/edge of the thin sweep body. If you receive a message describing this situation, you must apply additional mesh controls to prevent it.
- If both source and target areas are meshed, thin sweep will fail. In this case, you must find some way to eliminate the situation.
- Thin sweep may issue warning messages stating that the source you picked was swapped with the target, and that some controls on the target faces were ignored as a result. These types of warning messages are for your reference only.

Handling General MultiZone Meshing Failures

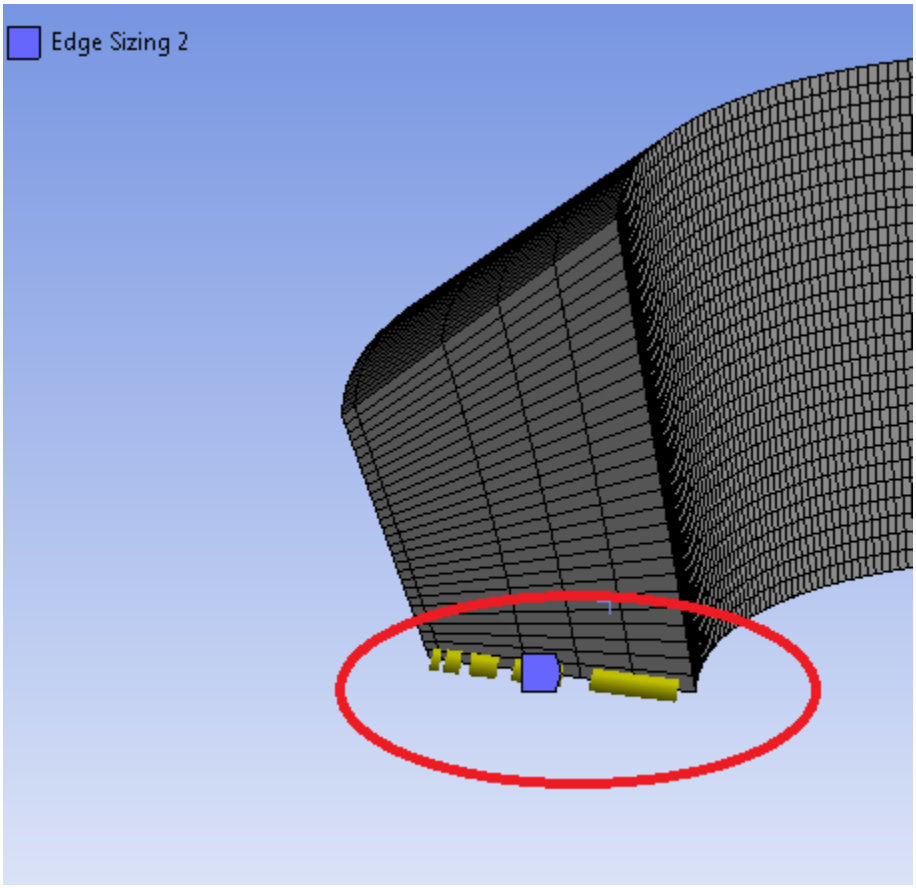
In the event of a **MultiZone** mesh failure, the following approaches are recommended:

1. If using automatic source face selection, try using manual source face selection (or vice versa). For manual source face selection, ensure that all sources and targets are selected. Refer to Using MultiZone (p. 423) for more information.
2. Ensure that all side faces are mappable. Refer to MultiZone Face Mappability Guidelines (p. 432) for more information. Use of virtual topologies can help make bodies sweepable:
   - Turn on the **Show Vertices** and **Close Vertices** options to see if any edges have unnecessary splits. Extra edge splits can make faces that appear to be mappable more difficult to map. Virtual edges (p. 546) can help in these cases.
   - Use of virtual face splits (p. 565) can help make faces more mappable, as can use of the mapped face mesh control (p. 314).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

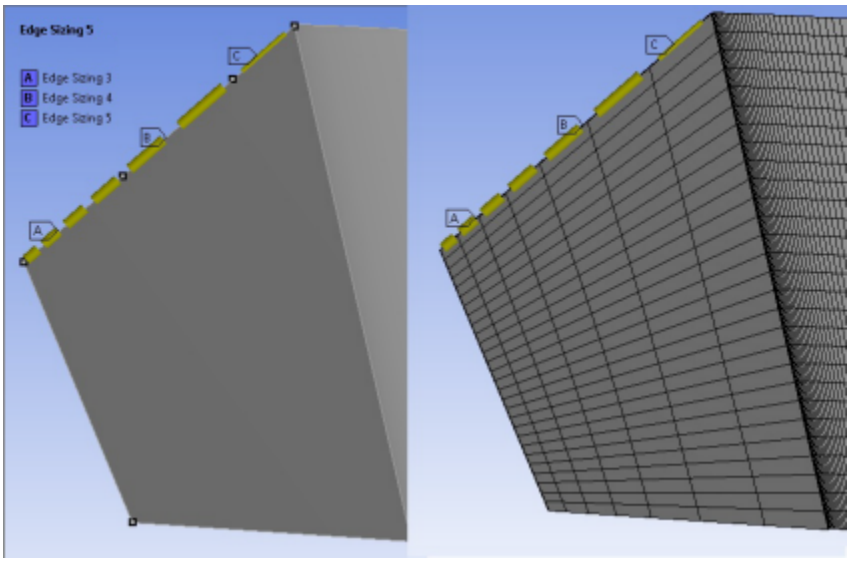
3. If MultiZone doesn't respect edge biasing, as shown in Figure 243: Edge Biasing Not Respected by MultiZone (p. 594) below, it may be because the opposite edge is split. To work around this, perform the edge biasing on the opposite edges to get a better edge distribution, as shown in Figure 244: Edge Biasing Respected by MultiZone (p. 595).

Figure 243: Edge Biasing Not Respected by MultiZone



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 244: Edge Biasing Respected by MultiZone



Note:

When a curve with bigeometric distribution is split, the curve is split into GEO1 and GEO2 starting at the split point.

4. Turn on the **Edge Coloring > By Connection** option to see if the edge connectivity is unusual. In some cases, the geometry connectivity may not be as expected, and this may create problems during meshing. These problems could be fixed in the DesignModeler application, the CAD package, or possibly through the use of virtual topologies (p. 545).
5. For a multibody part, turn on the **Edge Coloring > By Body Connection** option to see if the edge connectivity is unusual between bodies.

For detailed information about the requirements and characteristics of **MultiZone**, refer to MultiZone Meshing (p. 419).

Handling Multizone Thin Sweep Failures

When the mesh fails without providing any useful message, you should try following:

1. Set sweep thickness slightly higher than the maximum thickness of the body.
2. Select source or target faces (try to select source and target faces completely, partial selection may return partial mesh).

Handling Multizone Cart Sweep Failures

When the mesh fails while performing Cart Sweep, you should try the following:

- Select the source face.
- Adjust cartesian size, you need to be careful about the sizes while performing Cart Sweep. For some cases, it works only with particular size.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Do not apply too many size parameters, it makes Cart Sweep overconstrained and mesh fails.
- Ensure that there are no drafts in the model, Cart Sweep cannot handle drafts.

Handling Failed Mesh Connections

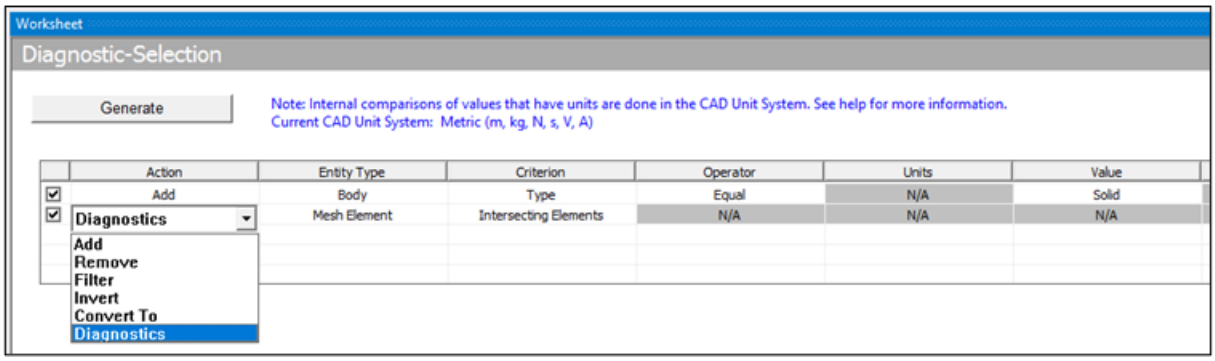
In the event of a mesh connection failure, refer to Diagnosing Failed Mesh Connections (p. 493).

Handling Failed Contact Matches

In the event of a contact match failure, refer to Troubleshooting Failed Contact Matches (p. 505).

Using Named Selection Diagnostics to Find Meshing Issues

The **Diagnostics Action** allows you to create Named Selections for the selected bodies which will highlight some key issues with the mesh that may need your attention to make it analysis-ready. Before adding a **Diagnostics** row to the worksheet, rows should be added to select relevant bodies to be used in the diagnostic checks as shown in the example below where all solid bodies are selected in the first row.



Several Criteria exist in the **Diagnostics Action** and depending on the **Entity Type** used, their effects areas follows:

Entity Type	Criterion	Usage
Face	Defeatured	Shows where geometry Faces have been fully defeatured during meshing.
Edge	Defeatured	Shows where geometry Edges have been fully defeatured during meshing.
Edge	Partially Defeatured	Shows where geometry Edges have been partially defeatured during meshing.
Vertex	Defeatured	Shows where geometry Vertices have been fully defeatured during meshing.
Mesh Node	Free Mesh Edges	Shows mesh nodes at edges which are open. That is, if volume meshing has failed due to unmeshed faces it will highlight these areas.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh Element	Intersecting Elements	Shows where mesh elements are intersecting causing meshing failure. Useful for troubleshooting failed-to-mesh bodies. The fixes can be as follows: • Add local size controls to reduce mesh size. • Add local size controls to modify defeaturing tolerance. • Merge faces locally using Virtual Topology. • Modify the geometry in the problematic location using a geometry tool.
Mesh Element	Body Interference Elements	Shows mesh elements which are intersecting due to bodies overlapping in 3D space. Elements at the surface intersection are selected. Geometry tools may be needed to repair such areas if overlaps are deemed too large.
Mesh Element	Sharp Angle Elements	Shows mesh elements which are flat due to geometry faces coming together at a tangent or sharp angle. Useful to diagnose geometry problems causing bad quality elements. Such problems may need to be fixed in a geometry editor by addition of a chamfer or round. For example, using Pull in Discovery or Discovery Modeling.
Mesh Element	Free Mesh Edges	Shows mesh nodes at edges which are open. That is, if volume meshing has failed due to unmeshed faces it will highlight these areas.
Element Face	Intersecting Elements	Shows where mesh elements are intersecting causing meshing failure. Useful for troubleshooting failed-to-mesh bodies.
Element Face	Body Interference Elements	Shows element faces which are intersecting due to bodies overlapping in 3D space. Elements at the surface intersection are selected. Geometry tools may be needed to repair such areas if overlaps are deemed too large.

When the **Diagnostics** worksheet tool has been used to generate element based named selection, you can right-click the named selection to access the **Select Mesh Clusters in Group** and use **Walk** in the **Selection** tab to graphically traverse through the mesh elements and mesh element clusters.

Tips for Using Virtual Topology

Virtual topology (p. 545) surfaces made up of two loops are not automatically mappable. For a faceted surface made up of two loops to be map meshed, a mapped **Face Meshing** control must be scoped to it or it must be a side area of a general sweep body.

Tips for Successful Meshing Using Local Controls

Mesh Based Defeaturing will pinch the together edges, if within the specified tolerance. But, large values can cause problems in some locations, especially in models with mixed scales. Local controls allow you to apply defeaturing locally by selecting problematic locations identified by error messages after meshing and:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- adding a body or face sizing with a larger (or smaller) defeaturing tolerance.
- adding a pinch control where you chooses the primary and secondary edge to focus the defeaturing to a specific pair of edges.

Meshing Complicated Models

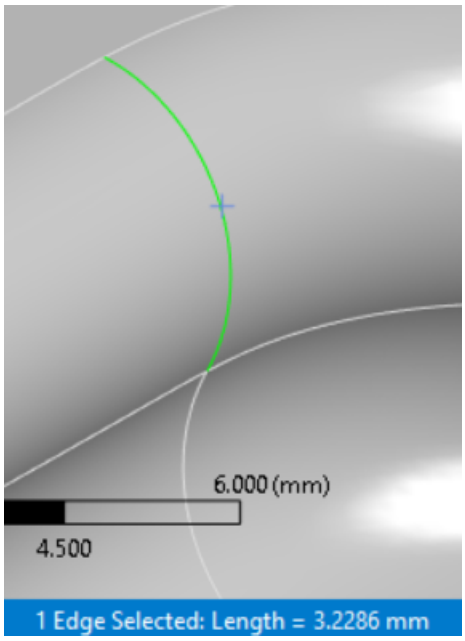
Meshing a complicated model may require special attention and experimentation. In such cases, the following strategies and guidelines are recommended for obtaining a successful mesh:

1. Analyze the model to determine its complexity:
   - Identify the small features that you do (and do not) want to retain.
   - Consider the model's size and its relationship to the element size transitions that are appropriate for the mesh. A smoother transition from the fine element size to the coarse element size will result in a larger number of elements, which should be considered (especially if the model is quite large). Coarser transitions will result in a smaller number of elements. You must determine what is acceptable.
   - Refer to the value in the **Minimum Edge Length** (p. 120) field.

Sizing	
Use Adaptive Sizing	No
Growth Rate	Default (1.2)
Mesh Defeaturing	Yes
Defeature Size	Default (2.e-002 mm)
Capture Curvature	No
Capture Proximity	No
Bounding Box Diagonal	883.51 mm
Average Surface Area	947.83 mm^2
Minimum Edge Length	3.1955e-002 mm
Quality	

     This field provides a read-only indication of the minimum edge length in the model.
   - Think about the element size that you expect to obtain, especially the desired *minimum* element size. To help you determine the desired size, in the **Geometry** window, select the edges of small features that you want to retain and refer to the status bar for feedback about the selections.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Perform a low-effort mesh evaluation by using appropriate sizes as determined from #1, but without controls such as inflation, match mesh controls, and so on, that add constraints to the mesher. Also, try to start with a coarser mesh size and refinement in later steps.

   - If the mesh is successful, examine it to see whether the mesh size and transition rates are acceptable. In most cases, you will need to make some adjustments to obtain the desired results.

   - If the mesh fails, examine any messages that the mesher returned to the **Messages** window, as described elsewhere in this Troubleshooting (p. 579) section.

3. Adjust settings to retain desired small features:

   - In many cases, small features are either small holes or channels in the model and are associated with high curvature. For this reason, using the **Curvature-based sizing** (p. 107) is a good strategy for retaining these features.

   - Be careful when using **Proximity-based sizing** (p. 107). If the value of **Minimum Edge Length** (p. 120) is too small, using Proximity-based sizing may lead to meshing problems.

4. Adjust settings to defeature (remove) unimportant small features:

   - The Meshing application automatically defeatures small features according to the specified Defeature Size (p. 111). Refer to the **Minimum Edge Length** (p. 120) value to help determine which small features will be defeatured automatically.

   - For solid models, **Defeature Size** is set to 50% of the value of **Curvature Min Size** (p. 113) by default. If you set a larger **Defeature Size**, you must also set a larger **Curvature Min Size** because the defeature size cannot be as large as the minimum element size.

5. Check the quality with the **Quality Histogram** and using **Display** style to view contours and max/min values of metrics such as **Skewness**, **Element Quality** and **Aspect Ratio**. Adjust the mesh settings to achieve the desired quality.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Continue making adjustments until your results are satisfactory. Try adjusting controls such as face sizing (p. 300), edge sizing (p. 300), transition rate (p. 112), and smoothing (p. 133). You may also want to experiment with virtual topology (p. 545).

Using a Localized Operating System on Linux

If you are using a localized operating system (such as French or German) on Linux, you must perform additional steps in order for the Meshing application to recognize the correct numerical format. Refer to the platform details section of the Ansys, Inc. Linux Installation Guide for details.

Using Lustre Parallel File Systems on Linux

Meshing application projects created prior to Release 16.0 need to be updated before they can be used on Lustre parallel file systems on Linux. To do this:

1. Load the project into Release 16 software on a system that does not use a Lustre parallel file system.
2. Perform an operation that changes each model in the Meshing application (for example, hide and then show a part). If systems share the same model, the change must be done for only one of the systems.
3. Save the project.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Meshing: Mesh Workflows

Mesh Workflows offer an alternative paradigm in Ansys Mechanical to enable mesh generation from complicated CAD geometries. When you use CAD geometries in simulation, it requires clean up and modifications (for example, hole filling, overlap resolution).

Mesh Workflows allow you to create specific meshing procedures like wrap, pull, extrude, and more, using **Ansys PrimeMesh** technology. **Mesh Workflows** have a built-in execution logic that provides an easy-to-use and customizable solution for complex meshing tasks.

Mesh Workflows topics include:

Mesh Workflow Concepts
Mesh Workflows Introduction
Mesh Workflows Context Tab
Mesh Workflow
Mesh Workflow Types
Mesh Workflow Steps
Mesh Workflow Controls
Mesh Workflow Outcomes
Property Worksheet
Mesh Workflow States
Mesh Workflow Domain Browser

Mesh Workflow Concepts

Mesh Workflows define a new meshing framework exposing advanced meshing capabilities to **Ansys Mechanical** for dealing with complex CAD geometries and meshing procedures. In **Mesh Workflows**, a meshing procedure is modelled as a sequence of **Steps** (a workflow) that can be parameterized and interconnected through the concepts of **Controls** and **Outcomes**.

Controls define the inputs for each workflow step, exposing the underlying algorithm parameters and defining the input scope for the step.

Outcomes expose the output data for each workflow step, expose information about the underlying algorithm execution, such as failure information and the generated mesh data as output scopes.

Mesh Workflow Steps operate on a new computational domain called the **PrimeMesh** model, based on the **PrimeMesh** meshing kernel, which exposes new meshing capabilities to **Ansys Mechanical**.

PrimeMesh Model

The **PrimeMesh** model eases mesh generation in the following ways:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Provides a lightweight simulation geometry that operates independently of the CAD constraints.
- Allows geometry modifications and manipulations that are often difficult to achieve with CAD systems.
- Operates differently from CAD-based systems and allows the creation of volumes without a CAD body.
- Uses wrapper approaches or surface and edge-based tools to create closed volumes for volume meshing, eliminating the need for boolean operations, which often fail with complex geometry.

A **PrimeMesh** model can be initialized by transferring the desired geometry from **Mechanical** to the **Mesh Workflows**, using the exposed Input object.

In the **Mesh Workflows**, all entities under a **Mechanical** part, including solid bodies, sheet bodies, line bodies, surfaces, curves, and vertices get converted to corresponding **PrimeMesh** topological entities.

PrimeMesh topological entities are simplified, faceted versions of the original geometry and the mesher keeps the original detailed geometry as a reference.

In the **PrimeMesh** Model, you can perform various operations to interact with the **PrimeMesh** topology, allowing flexible changes that are difficult to achieve in a CAD system. These operations include creating tolerant edge-based connections and merging complex surfaces or edges without losing the original shape.

When all **Mesh Workflow Steps** are completed, the resulting **PrimeMesh** model can be transferred to the **Mechanical Mesh** by completing the **Output** object. On completion, the underlying **PrimeMesh** model is translated to the corresponding **Mechanical** entities as a set of geometry bodies, mesh parts and named selections.

Note:

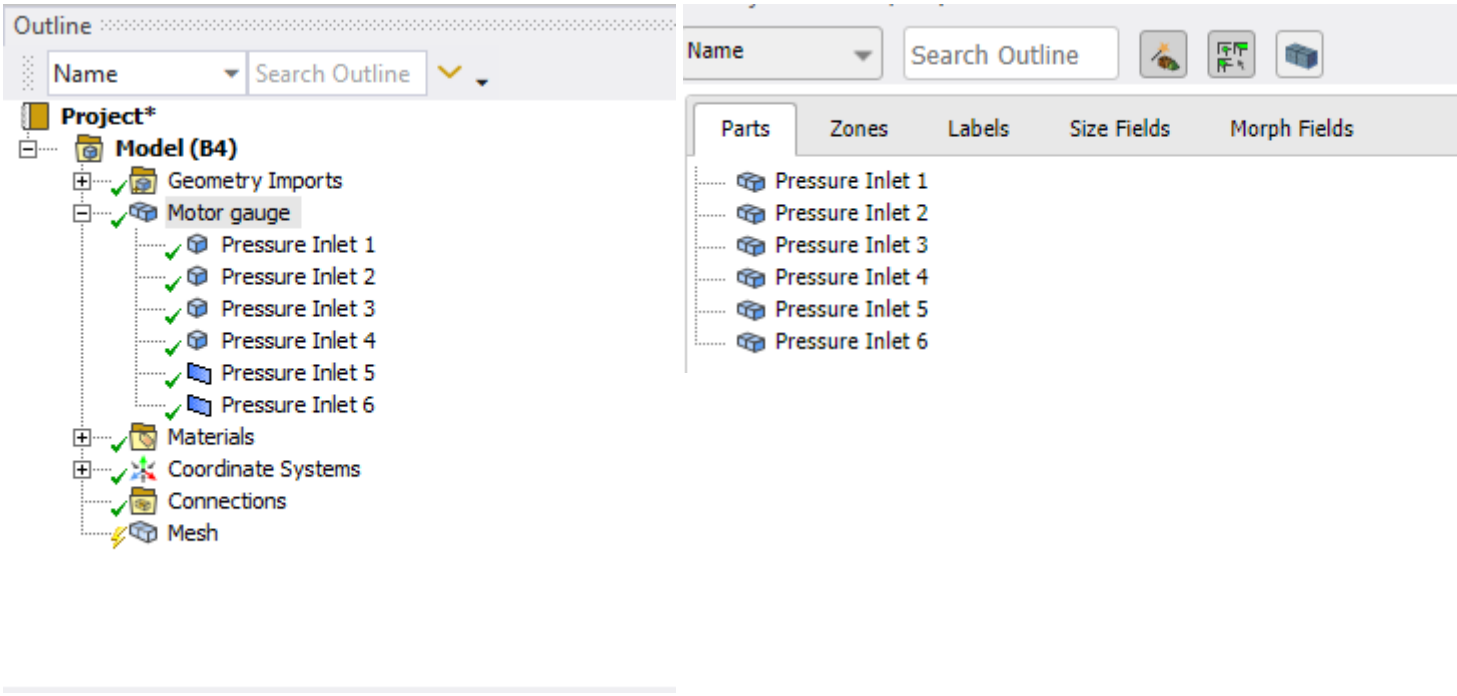
Each **Mesh Workflow** operates on a separate **PrimeMesh** model, as no data is shared between multiple workflows.

The **PrimeMesh** model computational domain has:

- **Parts**: Refers to the topology and mesh data of the model. When **PrimeMesh** consumes a **Mechanical** Part, it forms a **PrimeMesh** Part for each component.

  The following image shows the multibody part in **Mechanical** and **PrimeMesh**:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

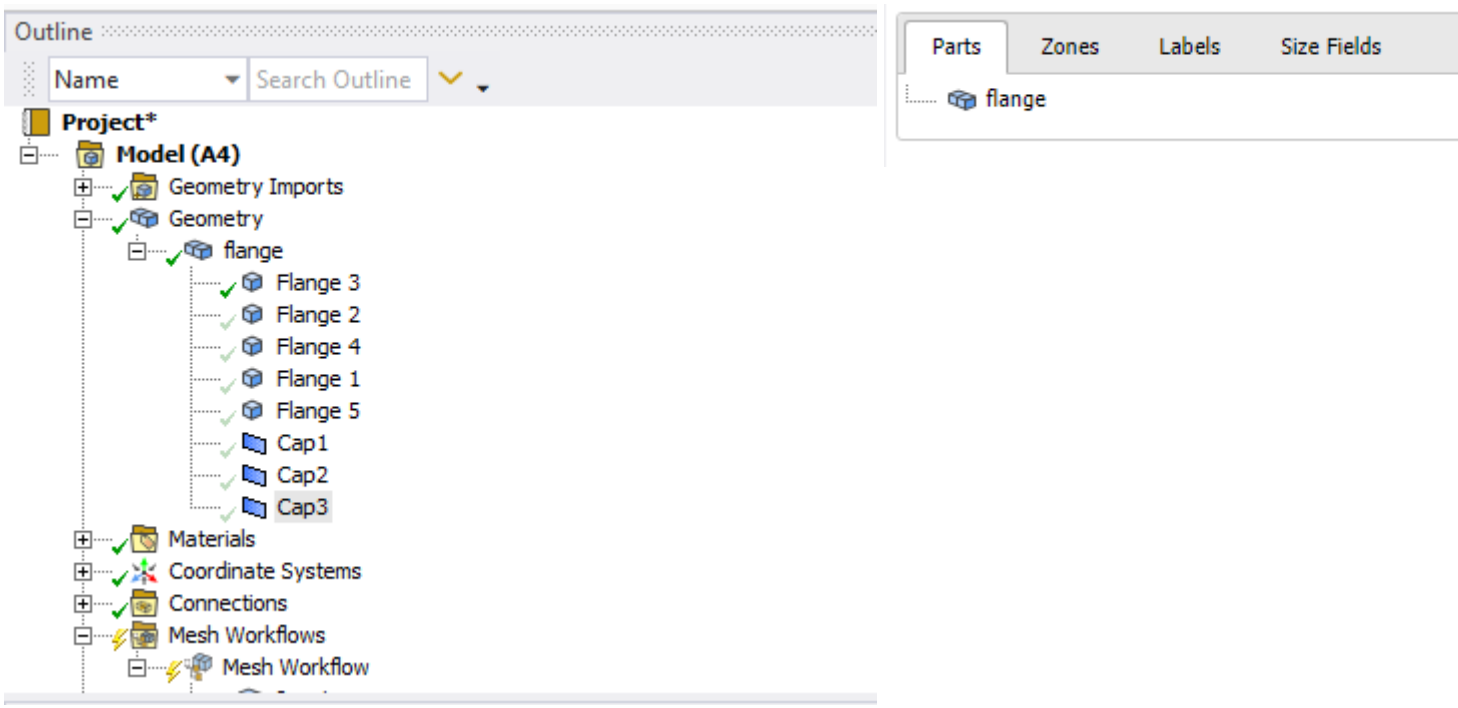


Note:

PrimeMesh Parts are independent of each other, and do not share any data with other Parts.

The following image shows the multibody part with shared topology in **Mechanical** and **PrimeMesh**:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



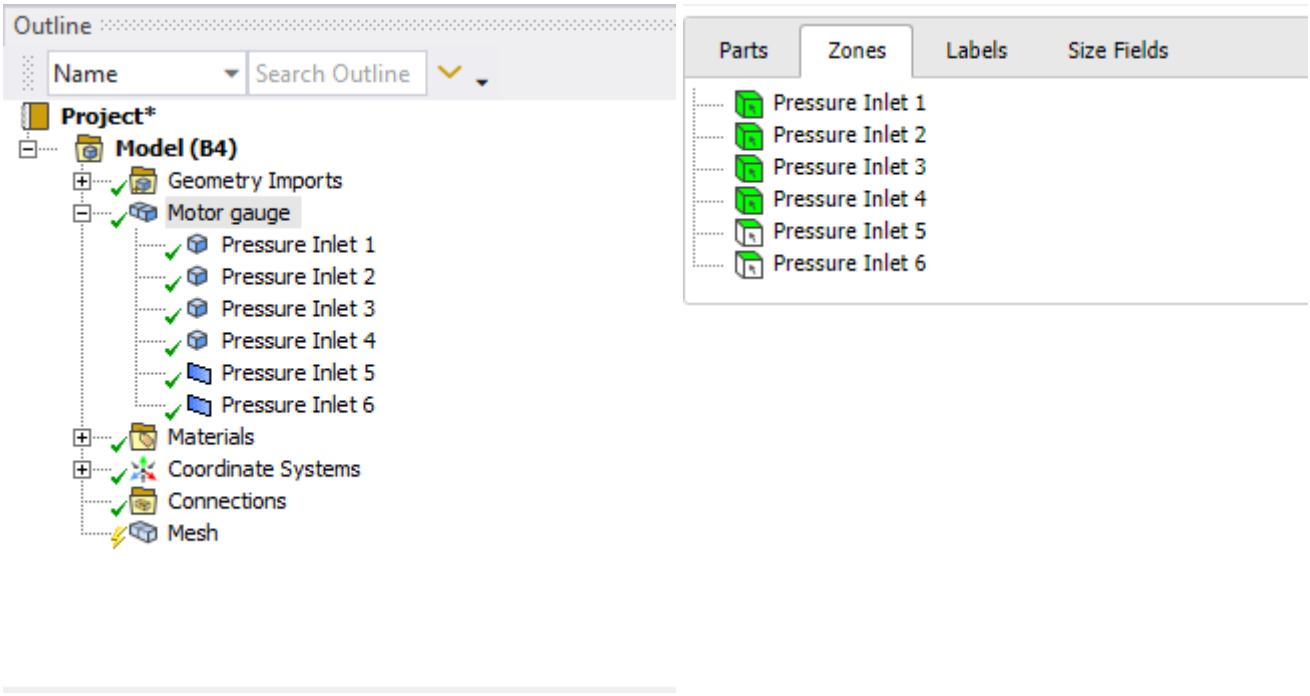
Note:

In multibody parts, bodies with shared topology in **Mechanical** create a single part in **PrimeMesh**.

For example, in the **Wrap** Step, you can delete the original part from the CAD in **Mesh Workflow** after creating a representation better suited for simulation. In some cases, this approach saves significant time and effort compared to cleaning up CAD and creating closed volumes by connecting and refining surface topologies. When meshing the **PrimeMesh** model **Parts**, **PrimeMesh** needs to collect the generated mesh entities to propagate them to output back into Mechanical entities, such as geometric entities like solid or sheet bodies, or **Named Selection** groups of Faces or Bodies.

- **Zones**: Represents a group of complete topological or mesh entities that define a complete partitioning of a part. Each zone has the entities with same dimension such as **Volume Zones** (3D), **Face Zones**(2D) and **Edge Zones** (1D), and has one-to-many mapping like one entity can belong to a single zone but one zone can consist of multiple entities. For its non-overlapping property, **Zones** can be used for assigning material property and boundary condition. The following image shows the multibody part in **Mechanical** transfers in to zones in **PrimeMesh**:

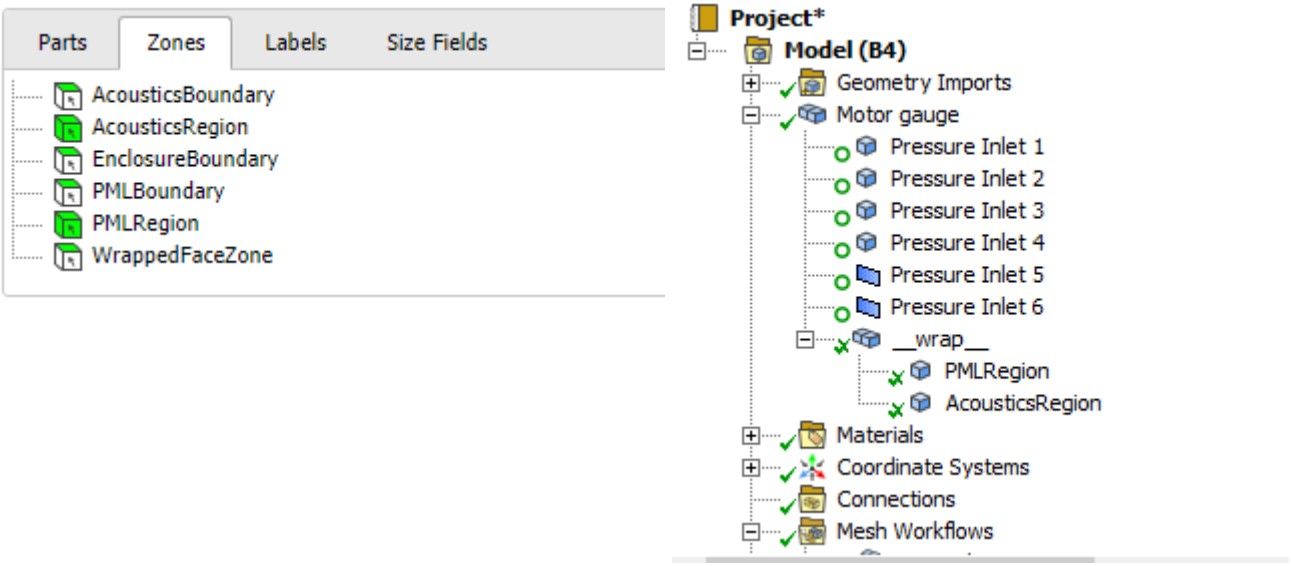
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

Mesh entities create a unique partition of the domain, so they do not overlap. For example, mesh elements (like hexes or tets) cannot appear in multiple **Volume Zones**, and mesh faces (like quads or tris) cannot appear in multiple **Face Zones**

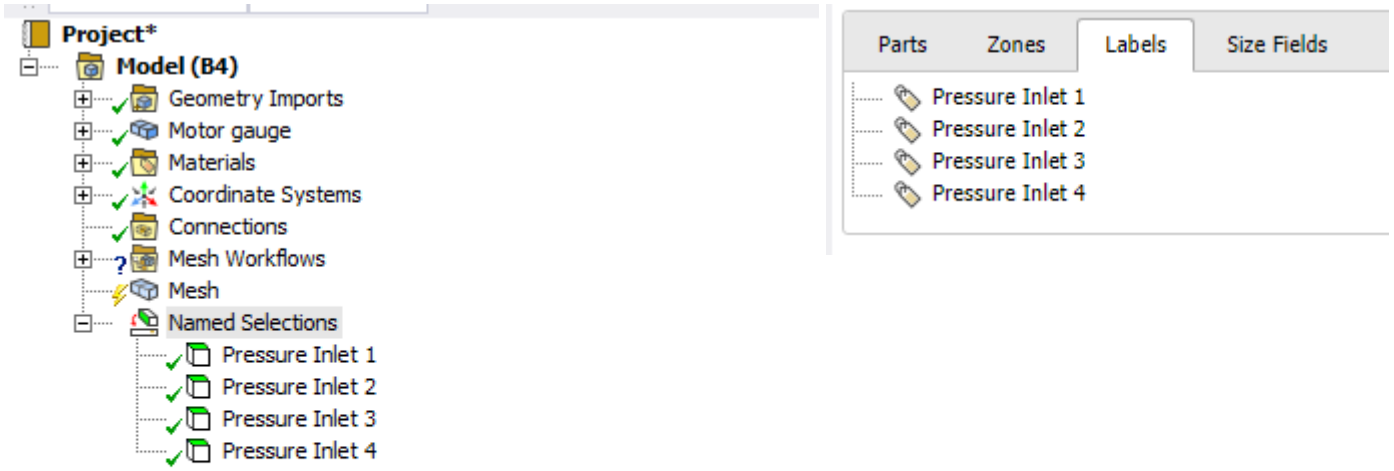
The following image shows the volume zones in **PrimeMesh** transfer to solid bodies in **Mechanical**:



- **Labels**: Represents a named grouping of entities. You can apply these labels for meshing operations and to determine which surfaces appear during the transfer to the Mesh Folder.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following image shows how the **Named Selections** in **Mechanical** transferred to **Labels** in **PrimeMesh**:



Mechanical	PrimeMesh	Description and Differences
Parts	**Parts**	Compared to **Mechanical** Parts which are consisting of geometry bodies, **PrimeMesh** Parts can either consist of topology entities created from imported CAD parts or contain mesh entities created from faceted geometries and/or imported meshes.
Solid Body	**Volume Zone**	Each solid body becomes a topological volume in the **PrimeMesh** model once initialized. You can define **Volume Zones** with multiple topological volumes for property assignment. **Volume Zones** transfer to **Mechanical** as solid bodies.
Sheet Body	**Face Zone**	**Face Zone** is a collection of topological faces or a group of interconnected mesh faces. **Face Zones** transfer to **Mechanical** as surface bodies.
Line Body	**Edge Zone**	**Edge Zone** is a collection of topological edges or a group of interconnected mesh edges. **Edge Zones** transfer to **Mechanical** as line bodies.
Named selections	**Labels**	**Labels** are names associated to entities. Compared to **Named selections** in **Mechanical**, **Labels** do not have a state and can appear or disappear at any time. At mesh workflow initialization time, **Named Selections** are propagated as **Labels** in the **PrimeMesh** model.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

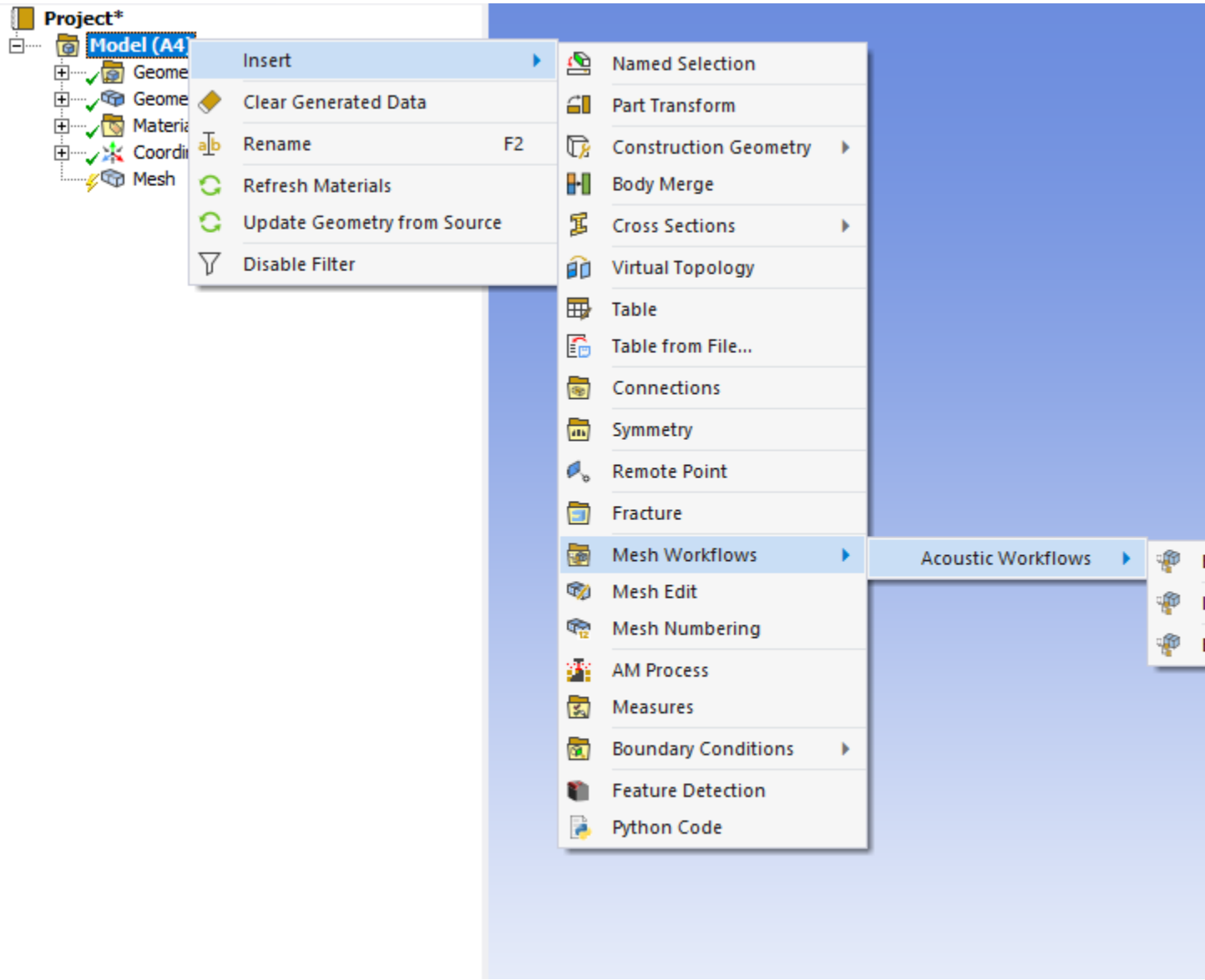
Mesh Workflows Introduction

Mesh Workflows offer a workflow-based approach for mesh generation of specific needs and support the use of predefined templates which can be customized as needed. Each **Mesh Workflow** consists of a series of steps which define the mesh generated, and each specific step in the workflow is based on a generic operation of a certain type. The **Controls** of an operation define what-to-do and how-to-do and provide a flexible composition of meshing algorithms and the **Outcomes** capture the results of an operation.

To access **Mesh Workflows**,

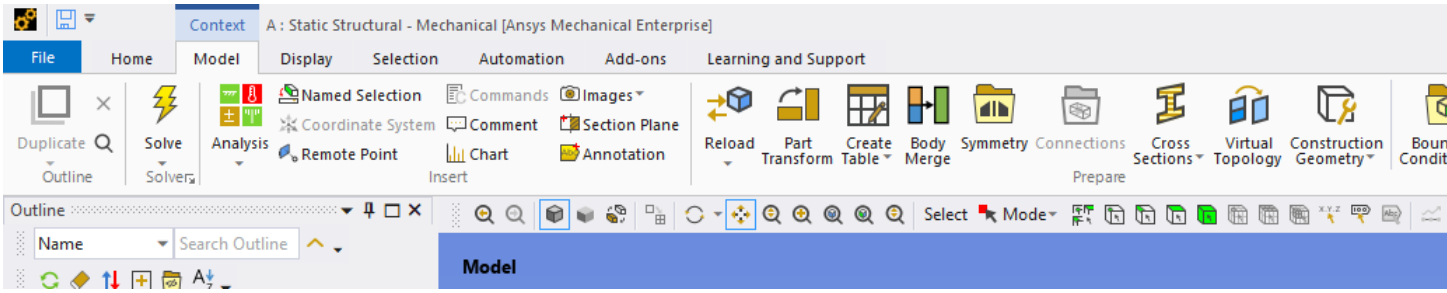
1. On the **Tree** Outline, right-click **Model** and click **Insert > Acoustic Workflows > Mesh Workflows**.

2. Select **Mesh Workflow Type** as per your requirement.



or

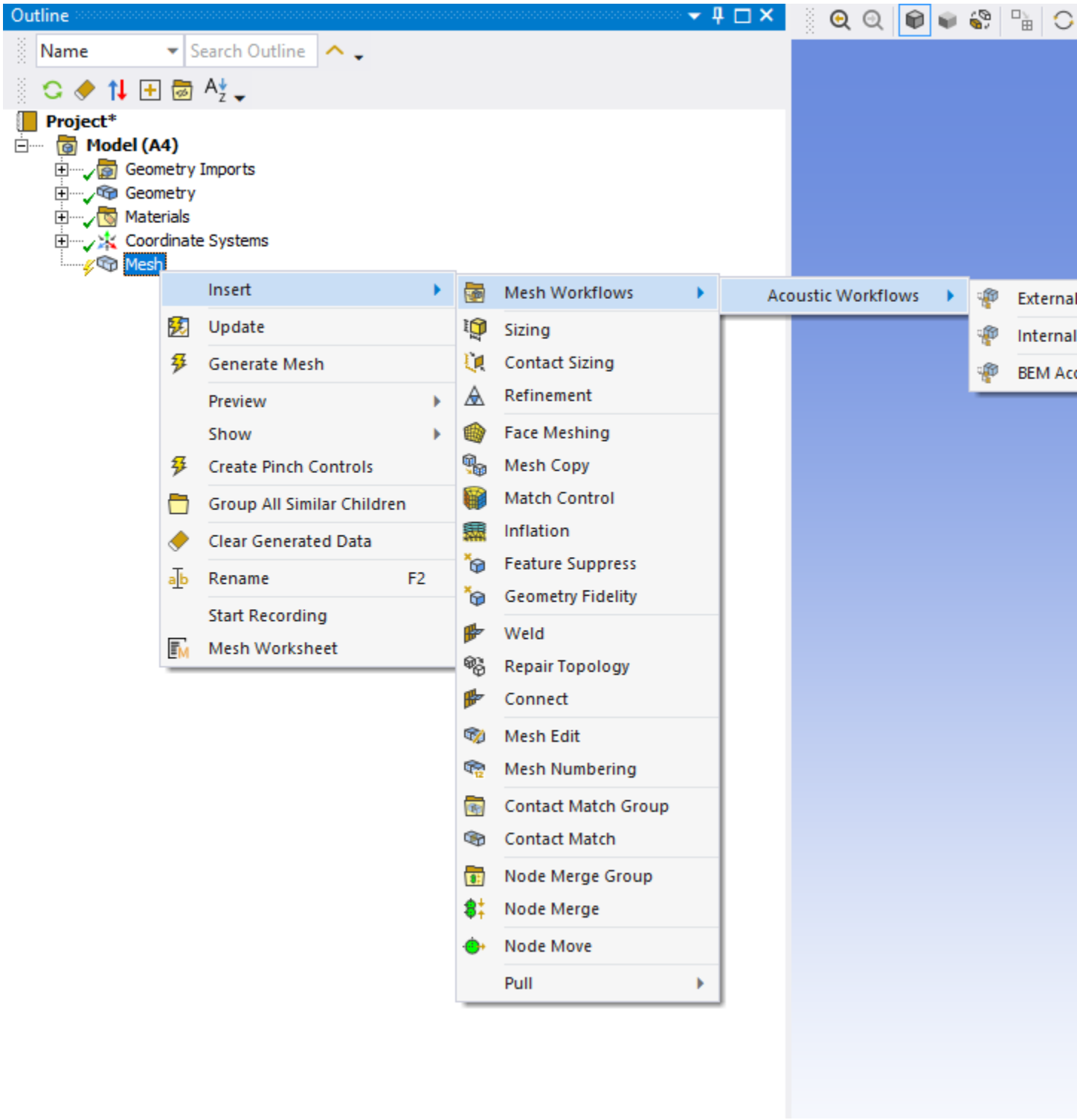
1. On the **Tree** Outline, click **Model**.
2. In the **Model Context** tab, under the **Mesh** group, click **Mesh Workflows**, click **Acoustic Workflows** and select the **Mesh Workflow Type**.



or

1. On the **Tree** Outline, right-click **Mesh** and click **Insert > Acoustic Workflows > Mesh Workflows**.
2. Select the **Mesh Workflow Type** as per your requirement.

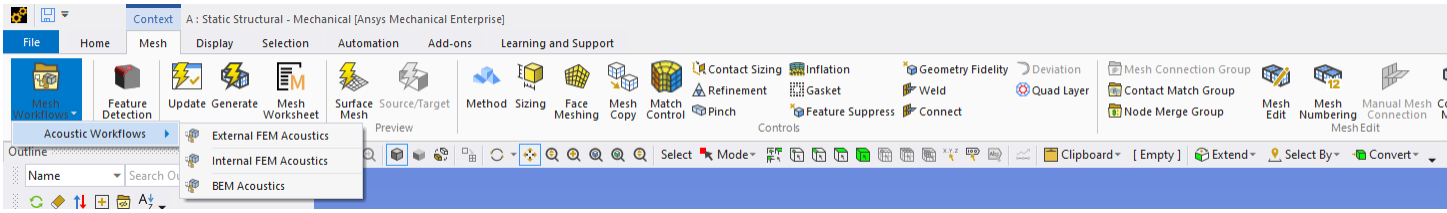
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



or

1. On the **Tree** Outline, click **Mesh**.
2. In the **Mesh Context** tab, under the **Mesh Workflow** group, click **Mesh Workflows**, click **Acoustic Workflows** and select the desired **Mesh Workflow Type**.

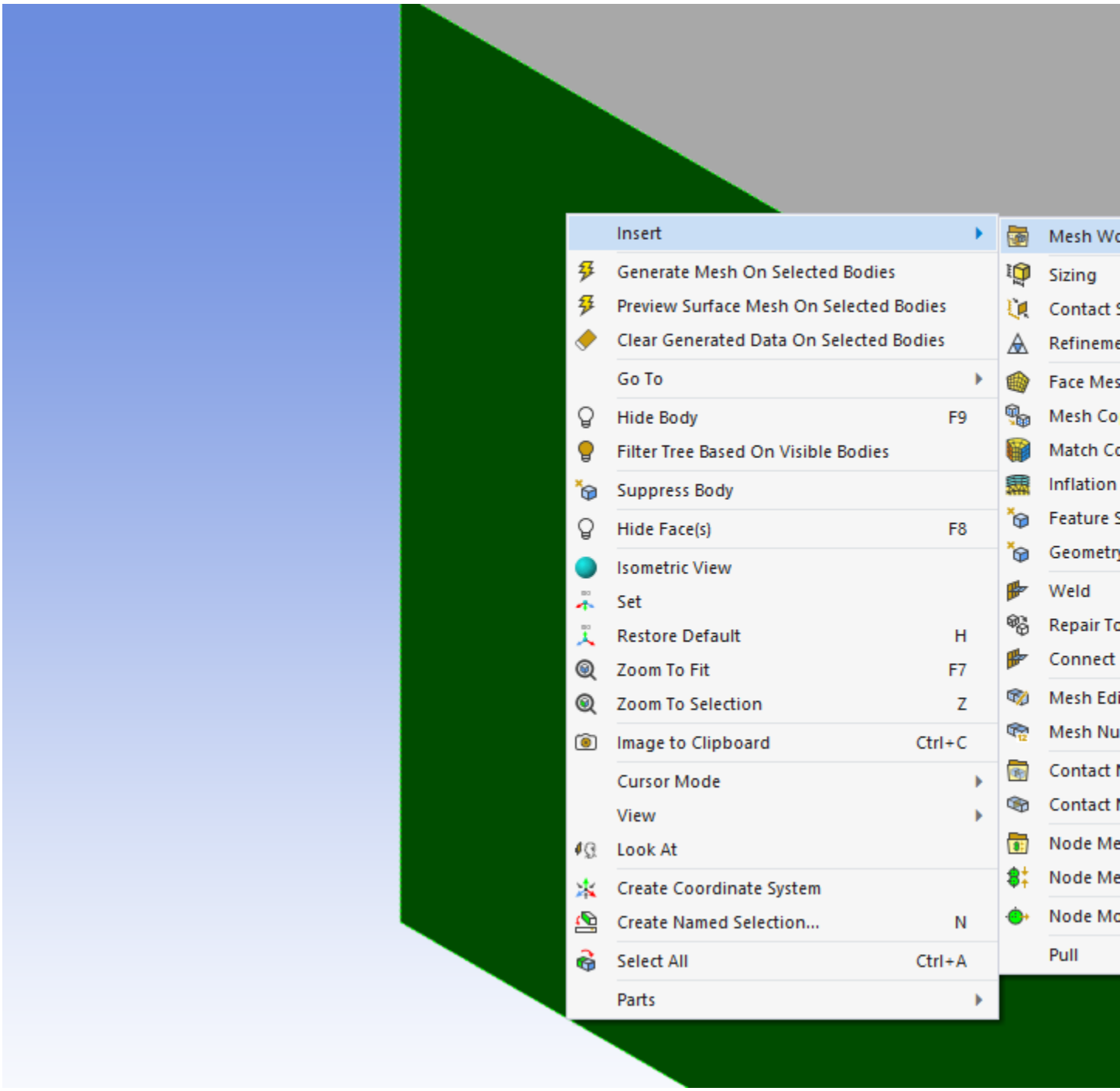
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



or

1. Right-click the **Geometry** window, click **Insert > Mesh Workflows > Acoustic Workflows**.
2. Select **Mesh Workflow Type** as per your requirement.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When you select a **Mesh Workflow Type**, a **Mesh Workflows** object is created on the tree. When you click the **Mesh Workflows** object on the **Tree**, the **Mesh Workflows Details** view and the **Mesh Workflows Context** (p. 612) tab are displayed.

Mesh Workflows object has a **Mesh Workflow** child-object of the selected **Mesh Workflow Type**. **Mesh Workflow** child-object has a predefined workflow based on the selected **Mesh Workflow Type**. You can select the created **Mesh Workflow** child-object to display the Details view of the corresponding **Mesh Workflow** (p. 613) child-object and its associated **Mesh Workflow Property Worksheet** (p. 676).

The **Mesh Workflows Details** view has the following options:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Definition

- **Active Workflow Group**: Displays the active workflow type used for generating mesh.

> **Note:**
>
> **Mesh** object parameters in the Details view do not have any impact on the **Mesh Workflow** parameters.

Right-click options available in **Mesh Workflows** are:

- **Insert**: Allows you to insert the desired mesh workflow type.
- **Delete**: Allows you to delete the respective mesh workflows.
- **Clear Generated Data**: Allows you to reset the generated data for the selected mesh workflow.
- **Clear Output Data**: Allows you to clear the output data after completing the mesh workflow. **Clear Output Data** is available only after completing the mesh workflow.
- **Generate Mesh Workflows**: Allows you to generate the selected mesh workflow.

Mesh Workflows Context Tab

The **Mesh Workflow Context** tab allows you to:

- Generate a workflow from defined steps.
- Initialize, import, execute and export the workflows.
- Visualize the workflow properties using the worksheet.

The **Mesh Workflows Context** tab has the following options:

Mesh Workflows

- **Generate Mesh Workflows**: Allows you to generate mesh workflows.
- **Initialize Workflow**: Allows you to initialize mesh workflow, transferring selected input geometry to the mesh workflow operations. **Named Selections** scoped to the input geometry translate into corresponding **Labels** after initializing the workflow.
- **Import Workflow**: Allows you to import workflow steps from a file. **Import Workflow** is available only after deleting steps and clearing data from the current workflow.
- **Execute Step**: Executes the next step of specific **Mesh Workflow**. **Execute Step** is available only when **Input** is scoped. If the mesh workflow is not initialized yet, **Execute Step** initializes the workflow before executing the step.
- **Execute All Steps**: Executes all step of specific **Mesh Workflow**. **Execute All Steps** is available only when **Input** is scoped. If the mesh workflow is not initialized yet, **Execute All Steps** initializes the workflow before executing the step.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Export Workflows**: Export the workflow steps as a template to a .emx file.
- **Complete Workflow**: Allows you to complete the workflow by transferring generated data back to new mechanical geometry parts along with the corresponding part meshes. You cannot edit the specified mesh workflow type after completing the workflow. **Complete Workflow** is available only when workflow steps are fully executed. When you click **Complete Workflow**, the status bar displays the progress of the workflow. You can click the progress bar displayed on the status bar to open the **Ansys Workbench Mesh Status** window to view the transferring of generated data. To interrupt a running workflow, In **Ansys Workbench Mesh Status** window, you can use **Stop** to interrupt the generated workflow data transfer to the new mechanical geometry. **Complete Workflow** is available only when workflow steps are fully executed.

Views

- **Property Worksheet**: Helps you view the published properties of each control involved in the mesh workflows, provides an easy visualization of the driving input options for mesh workflows and to edit individual controls. For information about the **Worksheet**, refer to Property Worksheet (p. 676).
- **Domain Browser**: Allows you to view the parts, zones, or labels involved in the mesh workflows and to edit individual controls in mesh workflows. For information about the **Domain Browser**, refer to Mesh Workflow Domain Browser (p. 682).

Mesh Workflow

Mesh Workflows offer a framework for creating simulation meshes through a predefined or customized workflow composed of individual executable steps. The predefined workflows serve as meshing templates for targeted industrial applications. **Mesh Workflow Types** (p. 618) offer predefined steps and values for creating acoustic mesh. The predefined **Mesh Workflow Types** are:

- **External FEM Acoustics**
- **Internal FEM Acoustics**
- **BEM Acoustics**

Note:

Mesh Workflow child-object has rules embedded for data consistency. Hence, you are not allowed to delete or modify some **Steps**.

Mesh Workflow child-object **Details** view has the following options:

General

- **Workflow Type**: Displays the selected mesh workflow type.

Definition

- **Length Unit**: Provides the length unit of the input model. When you use **Initialize Workflow** to transfer the input geometry, mesh workflow derives the **Length Unit** from the transferred input.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Steps File Location**: Displays the file location from which you load the steps using **Import Workflow**.

Mesh Workflow object has the following child-objects:

- **Input** (p. 615)
- **Steps** (p. 615)
- **Output** (p. 617)

Right-click options available in **Mesh Workflow** Object are:

- **Rename**: Allows you to rename the specified mesh workflow object.
- **Delete**: Allows you to delete the specified mesh workflow object.
- **Clear Generated Data**: Allows you to clear generated data for the specified mesh workflow object.
- **Clear Output Data**: Allows you to clear the output data after completing the specified mesh workflow object. **Clear Output Data** is available only after completing the specified mesh workflow object.
- **Execute Step**: Initializes the workflow and executes the next step of the specified mesh workflow. **Execute Step** is available only when you initialize mesh workflow.
- **Execute All Steps**: Initializes the workflow and execute all steps of the specified mesh workflow. **Execute All Steps** is available only when you initialize mesh workflow.
- **Generate Mesh Workflows**: Allows you to generate the specified mesh workflow object.

Behavior of Mesh Workflow Generated Bodies

Mesh Workflow generated bodies have the following special behaviors:

- When you right-click **Geometry** object and apply **Freeze Mesh on all Parts** on mesh workflow generated bodies, it only freezes geometry bodies that are not mesh workflow generated.
- **Mesh Workflow** generated bodies behave like frozen bodies because the **Mesh Workflows** completely handle their meshing and the local meshing methods cannot control **Mesh Workflows**.
- When you right-click **Mesh** object and click **Clear Generated Data**, clears only the mesh that does not come from a mesh workflow.
- When you right-click **Mesh** object and click **Clear All Generated Data**, clears output data from the mesh workflow and the local mesh.
- **Mesh Workflow** generated bodies in the **Geometry** window may displays both input geometry and the mesh workflow generated geometry. Hence, you may have to hide the input geometry in the **Geometry** window to select the Mesh Workflow generated bodies.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Input

Input allows you to select the geometry parts used for initializing the mesh workflow. **Input** supports the transfer of geometry from **imported** bodies. You cannot transfer internally created bodies (for example, pull bodies) or bodies from frozen parts as input.

Note:

: When you scope bodies with shared topology, the selection is automatically extended to the entire multi-body part.

Right-click options available in the **Input** are:

- **Initialize Workflow**: Allows you to initialize the mesh workflow by transferring selected input geometry to the mesh workflow operations. **Named Selections** scoped to the input geometry translate into corresponding **Labels** after initializing the workflow. **Initialize Workflow** is available only when you scope a geometry to **Input** .
- **Rename**: Allows you to rename the **Input**.

The **Input Details** view has the following options:

Scope

- **Scoping Method**: Allows you to scope the input. The available options are **Geometry Selection** and **Named Selection**. The default is **Geometry Selection**.
  - **Geometry Selection**: Allows you to scope the geometry bodies. When you select **Geometry Selection**, **Geometry** allows you to select the bodies from the **Geometry** window.
  - **Named Selection**: Allows you to scope the available name selections.

Steps

Steps provide an ordered sequence of mesh workflow operations for the selected mesh workflow type. Each **Step** represents controls and outputs defined for the specific operation. A set of control data defines each **Control** based on its specific operation control type. **Outcomes** are predefined to collect desired outputs after running the operation. The major purpose of **Outcomes**, besides monitoring results, is also to collect output information which can be used as input for a subsequent operation.

Right-click options available in the **Steps** are:

- **Delete Settings**: Allows you to delete Mesh Setting in the **Steps Details** view. The available option is:
  - **Acoustic Settings**: Allows you delete the acoustic Mesh Setting in the **Steps Details** view.
- **Add Settings**: Allows you to add Mesh Setting in the **Steps Details** view. The available option is:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Acoustic Settings**: Allows you add the acoustic Mesh Setting in the **Steps Details** view.

- **Propagate Settings**: Transfer mesh settings when you update some sizes or number of layers associated with individual steps in the **Mesh Workflows**. **Propagate Settings** is available only when **Analysis Frequency** is greater than **0.0 Hz**. The available option is:

  - **Acoustic Settings**: Transfer acoustic mesh settings when you update some sizes or number of layers associated with individual steps in the **Mesh Workflows**.

- **Rename**: Allows you to rename the **Steps**.

- **Insert**: Allows you to insert mesh workflow steps. You can only **Insert** a mesh workflow step if the **Output** node is not completed.

- **Delete Steps and Clear Data**: Allows you to delete all steps and clear data in the workflow. When you **Delete Steps and Clear Data** for the mesh workflow object, the input node remains defined and you can select the **Mesh Workflow Type** for the **Mesh Workflow**.

- **Execute Step**: Executes the next step of specific **Mesh Workflow**. **Execute Step** is available only when **Input** is scoped. If the mesh workflow is not initialized yet, **Execute Step** initializes the workflow before executing the step.

- **Execute All Steps**: Executes all step of specific **Mesh Workflow**. **Execute All Steps** is available only when Input is scoped. If the mesh workflow is not initialized yet, **Execute All Steps** initializes the workflow before executing the step.

Steps Details view has the following options:

General

- **Number of Cores**: Allows you to provide the maximum number of cores required for the mesh workflow execution. The default value is **8**. You can parametrize **Number of Cores**.

Mesh Settings

- **Analysis Frequency**: Allows you to set the frequency for the input model, which is used to derive the **Mesh Size**. The default value is **0.0 Hz**.

  For setting **Analysis Frequency**, you should enter a value greater than **0.0 Hz**. You can parametrize **Analysis Frequency**.

- **Quadratic Elements**: Allows you to define to use the quadratic elements for the mesh or not. The default value is **No**.

- **Speed of Sound**: Allows you to set the speed of sound, which is used to derive the **Mesh Size**. The default value is **343.0 m/s**. You can parametrize **Speed of Sound**.

- **Mesh Size**: Displays the mesh size for the mesh workflow. Mesh Size is a read-only value calculated based on the below formula:

  wavelength = C/F

  C = Speed of Sound, and

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

F = Analysis Frequency

For linear elements,

Mesh Size = **wavelength/12**

For quadratic elements,

Mesh Size = **wavelength/6**

- **Number of Acoustic Layers**: Displays the number of acoustic layers for the acoustic region.

  Layers created around the acoustic region during the **Extrude Acoustic Region** step are called Acoustic Layers. **Number of Layers** in the **Extrude Acoustic Region** step is equal to the **Number of Acoustic Layers**.

  When **Quadratic Elements** is set to **No** the default value for **Number of Acoustic Layers** is **2**.

  When **Quadratic Elements**is set to **Yes** the default value for **Number of Acoustic Layers** is **1**.

- **Number of PMLs/IPMLs**: Displays the number of PML/IPML layers for the PML/IPMLs region.

  Layers created around the PML/IPML during the **Extrude PML/IPML** Region step are called PML/IPML. **Number of Layers** in the **Extrude PMLs/IPMLs** Region step is equal to the **Number of PMLs/IPMLs**.

  When **Quadratic Elements** is set to **No** the default value for **Number of PMLs/IPMLs** is **4**.

  When **Quadratic Elements** is set to **Yes** the default value for **Number of PMLs/IPMLs** is **2**.

Output

Output controls the data transfer from generated **Mesh Workflow** data to **Ansys Mechanical** geometry parts along with its associated mesh with type entities based on zones.

Right-click options available in **Output** are:

- **Complete Workflow**: Allows you to complete the workflow by transferring generated data back to new mechanical geometry parts along with the corresponding part meshes. You cannot edit the specified mesh workflow type after completing the workflow.**Complete Workflow** is available only when workflow steps are fully executed. When you click **Complete Workflow**, the status bar displays the progress of the workflow. You can click the progress bar displayed on the status bar to open the **Ansys Workbench Mesh Status** window to view the transferring of generated data. To interrupt a running workflow, In **Ansys Workbench Mesh Status** window, you can use **Stop** to interrupt the generated workflow data transfer to the new mechanical geometry. **Complete Workflow** is available only when workflow steps are fully executed.
- **Rename**: Allows you to rename the output.
- **Clear Output Data**: Allows you to clear the output data in the workflow. **Clear Output Data** is available only if the mesh workflow is completed.

The **Output Details** view has the following options:

Definition

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Data Transfer Type**: Create the new geometries based on the data transfer type. The available option is:
  - **By Zones**: Allows you to create geometrical entities and named selections from zones.
- **Process Entities of Type**: Sets the type of entities dimension based on zones. The available options are:
  - **Volume Zones**: Allows you to process the volume zones and transfer it as solid bodies in **Geometry**. The default value for **External FEM Acoustics** and **Internal FEM Acoustics** is **Volume Zones**.
  - **Face Zones**: Allows you to process the face zones and transfer it as surface bodies in **Geometry**. The default value for **BEM Acoustics** is **Face Zones**.

Mesh Workflow Types

Mesh Workflow Types have predefined workflow templates to perform specific mesh operations that can be customized as per your requirement. Each **Mesh Workflow Type** has **Steps** to perform specific operation using **Controls** and **Outcomes**. **Mesh Workflow Types** available are:

- **External FEM Acoustics** (p. 619)
- **Internal FEM Acoustics** (p. 621)
- **BEM Acoustics** (p. 623)

Sizing Recommendations for Acoustic Workflows

In **Acoustics Mesh Workflows**, the number of elements required within a wavelength depends on the desired frequency and required accuracy. You may need at least two or more elements per wavelength to capture the acoustic behaviour efficiently.

The number of elements per wavelength is calculated as:

$$N = \left(\frac{c}{n}\right) e$$

where,

N is the Number of Elements per Wavelength

c is the Propagation speed of sound

n is the Frequency of Interest

e is the Element Size

Hence, if you want two elements (N=2) per wavelength,you need to set the element size as:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$e = \frac{c}{2n}$$

The speed of sound depends on the medium of transmission. The speed of sound is 343 m/s in air at 20°C and 1481 m/s in water at 20°C.

Guidelines related to Acoustics domain size and Acoustics mesh size to be followed while working with Acoustic workflows are below:

- Domain size should be equal to wavelength (Without PML/IPML). Domain size can be smaller with PML/IPML.
- Mesh size is dependent on speed of sound, Wavelength and Number of Elements in the Wavelength.
- Recommend you to consider the following number of elements in acoustic domain:
  - Number of lower order elements should be greater than or equal to 12 in acoustic domains.
  - Number of higher order elements should be greater than or equal to 6.
  - Number of PML/IPML elements should be greater than or equal to 4.
  - Add buffer elements greater than or equal 2 that are required as a part of acoustic domain.

  For example, if you are working on sound propagation for at 5000 Hz frequency in Air domain with speed of sound as 343000 mm/s.

  Acoustics Domain Size = 343000/5000 = 68.6 mm.

  For Lower order mesh, Element Size for Acoustics and PML/IPML Mesh = 68.6/12 = 5.7167 mm.

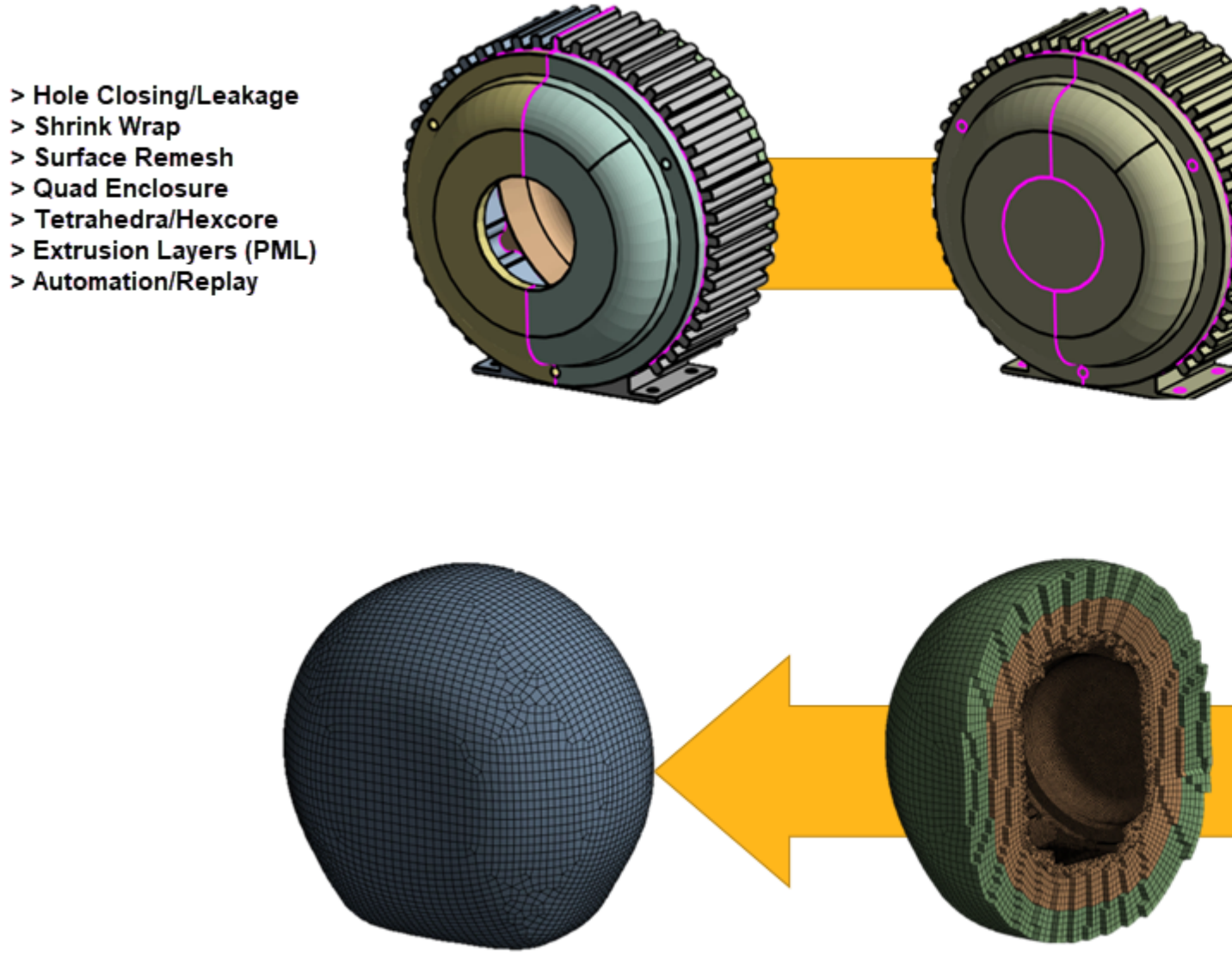
  For Higher order mesh, Element Size for Acoustics and PML/IPML Mesh = 68.6/6 = 11.4333 mm.

External FEM Acoustics

External FEM Acoustics workflow creates external acoustic sound radiation domain to predict the propagation of radiated sound in the near field and far field due to vibration of the external surface of the structural assembly or flow source port of the fluid flow.

External FEM Acoustics workflow can create different domain shape as per your requirement. The available options are **Convex Irregular Shape Enclosure** (Irregular Body Fitted Convex), **External Part Enclosure** (custom domain shape through imported CAD file), **Spherical Enclosure**, **Hemispherical Enclosure**. All mesh domains have the option to create near field acoustic domain mesh which is enough for radiation or infinite element boundary condition. You also have the option to add Perfectly Matched Layers (PML/IPML) to define PML boundary condition.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When you select **Mesh Workflows** as **External FEM Acoustics**, **Mesh Workflow** loads a predefined template with **Steps** and **Outcomes**. **Mesh Workflow** performs the **Steps** through **Controls** and **Outcomes** to achieve the desired mesh for the **External FEM Acoustics**.

External FEM Acoustics workflow has the following steps:

- **Fill Holes** (p. 625): Fill the holes defined in the Hole Filling step. The operation type available for **Fill Holes** is Fill Holes (p. 625).

> **Note:**
>
> You can add or delete **Fill Holes** (p. 625) step as per your requirement.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Wrap Parts** (p. 626): Wraps the selected parts. The operation type available for **Wrap Parts** is Wrap (p. 626).
- **Improve Wrap Mesh** (p. 627): Improves the surface mesh. The operation type available for **Improve Wrap Mesh** is Mesh Surface (p. 627).
- **Create Enclosure** (p. 627): Creates the enclosure around the specified scope. The operation type available for **Create Enclosure** is Create Enclosure (p. 627).
- **Mesh Volume** (p. 628): Creates the volumetric mesh. The operation type available for **Mesh Volume** is Mesh Volume (p. 628).
- **Improve Volume Mesh** (p. 628):Improves the volumetric mesh generated in the **Mesh Volume** step. The operation type available for **Improve Volume Mesh** is Improve Volume Mesh (p. 628).
- **Extrude Acoustic Region** (p. 629): Generates layers of prismatic elements from the input surface to a specified height. The generated volume may have layers of constant size hexahedral or triangular prism elements depending on the input surface. The operation type available for **Extrude Acoustic Region** is Extrude (p. 629).
- **Extrude PML/IPML Region** (p. 629): Generates layers of prismatic elements from the acoustic region to a specified height. The generated volume may have layers of constant size hexahedral or triangular prism elements depending on the input surface. The operation type available for **Extrude PML/IPML Region** is Extrude (p. 629).

  Note:

  You can add or delete **Extrude PML/IPML Region** as per your requirement. You can create PML/IPML layers extruding the external surface layer based on the **Number of Layers** and **Per Layer Height** you specify.

- **Create Acoustic Regions** (p. 629): Creates the topology from the mesh only model. The operation type available for **Create Acoustic Regions** is Create Topology (p. 629).
- **Merge Acoustic Regions** (p. 629): Merges the volume. The operation type available for **Merge Acoustic Regions** is Merge Volumes (p. 629).
- **Assign Physics Properties** (p. 630): Defines material properties and thickness on the scoped parts or zones. The operation type available for **Assign Physics Properties** is Manage Zone Properties (p. 630).

Points to Remember

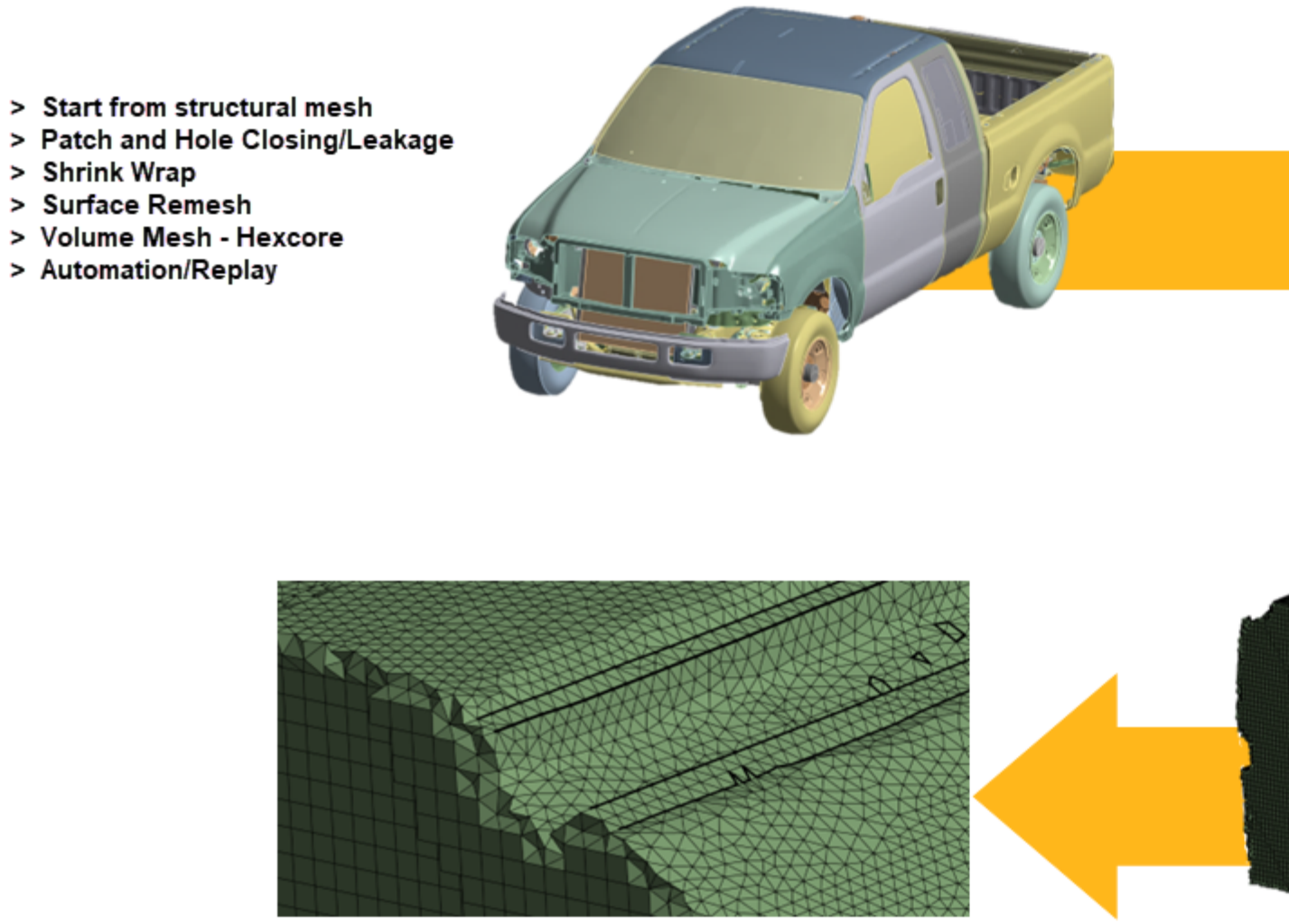
Sizing Recommendations for Acoustic Workflows (p. 618)

Internal FEM Acoustics

Internal FEM Acoustics workflow creates acoustic domain for internal cavity acoustic wave propagation and transmission. The workflow contains steps to close the openings (holes) on the external surface and create a watertight acoustic cavity. **Internal FEM Acoustics** workflow meshes the surface cavity with triangular elements followed by volumetric meshing adjacent to the surface of

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the cavity. You can generate hybrid mesh using hexcore for volumetric meshing. This helps to reduce the total mesh count if needed.



Note:

Internal FEM Acoustics recommends a minimum of 12 elements per wavelength for FEM mesh having linear elements and a minimum 6 elements per wavelength of sonic speed of medium for quadratic elements.

When you select **Mesh Workflows** as **Internal FEM Acoustics**, **Mesh Workflow** loads a predefined template with **Steps** and **Outcomes**. **Mesh Workflow** performs these **Steps** through **Controls** and **Outcomes** to achieve the desired mesh for the **Internal FEM Acoustics**.

Internal FEM Acoustics workflow has the following steps:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Fill Holes** (p. 625): Fill the holes defined in the Hole Filling step. The operation type available for **Fill Holes** is Fill Holes (p. 625).

  Note:

  You can add or delete **Fill Holes** (p. 625) step as per your requirement.

- **Wrap Parts** (p. 626): Wraps the selected parts. The operation type available for **Wrap Parts** is Wrap (p. 626).
- **Improve Wrap Mesh** (p. 627): Improves the surface mesh. The operation type available for **Improve Wrap Mesh** is Mesh Surface (p. 627).
- **Mesh Volume** (p. 628): Creates the volumetric mesh. The operation type available for **Mesh Volume** is Mesh Volume (p. 628).
- **Improve Volume Mesh** (p. 628):Improves the volumetric mesh generated in the **Mesh Volume** step. The operation type available for **Improve Volume Mesh** is Improve Volume Mesh (p. 628).
- **Create Acoustic Regions** (p. 629): Creates the topology from the mesh only model. The operation type available for **Create Acoustic Regions** is Create Topology (p. 629).
- **Assign Physics Properties** (p. 630): Defines material properties and thickness on the scoped parts or zones. The operation type available for **Assign Physics Properties** is Manage Zone Properties (p. 630).

Points to Remember

Sizing Recommendations for Acoustic Workflows (p. 618)

BEM Acoustics

BEM (Boundary element method) Acoustics requires only surface mesh of the source body for both internal cavity sound propagation or transmission, and for external sound propagation for radiation. You can either close openings (holes) to create watertight acoustic cavity for internal sound propagation or does not close openings for external sound radiation. You can use **Wrap** to create the surface mesh and remesh the surface with quadrilateral elements to improve surface mesh quality and reduce the mesh count. For **External BEM Acoustics**, a microphone grid surface of hemispherical shape can be created for results postprocessing using **Create Enclosure** step.

Note:

BEM Acoustics mesh has linear element by default. Hence, **BEM Acoustics** recommends a minimum of 8 elements per wavelength of sonic speed of medium.

When you select **Mesh Workflows** as **BEM Acoustics**, **Mesh Workflow** loads a predefined template with **Steps** and **Outcomes**. **Mesh Workflow** performs these **Steps** through **Controls** and **Outcomes** to achieve the desired mesh for the **BEM Acoustics**.

BEM Acoustics workflow has the following steps:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Fill Holes** (p. 625): Fill the holes defined in the Hole Filling step. The operation type available for **Fill Holes** is Fill Holes (p. 625).

  Note:

  You can add or delete **Fill Holes** (p. 625) step as per your requirement.

- **Wrap Parts** (p. 626): Wraps the selected parts. The operation type available for **Wrap Parts** is Wrap (p. 626).
- **Improve Wrap Mesh** (p. 627): Improves the surface mesh. The operation type available for **Improve Wrap Mesh** is Mesh Surface (p. 627).
- **Quad Based Remesh** (p. 627): Remesh the surface to convert the tri-based mesh into a quad-dominant mesh. The operation type available for **Quad Based Remesh** is **Mesh Surface**.

  Quad Based Remesh uses **Constant Size Surface Mesher** (p. 638) for remeshing.
- **Create Enclosure** (p. 627): Creates the enclosure around the specified scope. The operation type available for **Create Enclosure** is Create Enclosure (p. 627).
- **Create Acoustic Regions** (p. 629): Creates the topology from the mesh only model. The operation type available for **Create Acoustic Regions** is Create Topology (p. 629).
- **Assign Physics Properties** (p. 630): Defines material properties and thickness on the scoped parts or zones. The operation type available for **Assign Physics Properties** is Manage Zone Properties (p. 630).

Points to Remember

Sizing Recommendations for Acoustic Workflows (p. 618)

Mesh Workflow Steps

Mesh Workflow Steps perform a set of sequential operations for the selected **Mesh Workflow Type**. Each **Step** object has specific control types defined to perform specific operations. When you select a **Step** in the **Tree** outline, the **Details** view for the respective **Step** has the following options:

General

- **Operation Type**: Displays the operation type for the selected mesh workflow step.
- **Create Checkpoint**: Creates checkpoint for the control and saves the current operation when **Create Checkpoint** is **Yes**.
- **Max Number of Warnings**: Allows you to provide the maximum number of warnings that can be displayed for the selected operation type. The default value is **5**.

Right-click options for each **Mesh Workflow Steps** are as follows:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Insert > Control**: Allows you to add control types available for the selected operation. Some controls are predefined for the selected mesh workflow operation. You can also add and delete the control types as per your requirements. For more details, see **Mesh Workflow Controls** (p. 632).
- **Inset > Outcome**: Allows you to add outcomes available for the selected operation. You can also scope **Outcome** as input for other mesh workflow operations. For more details, see **Mesh Workflow Outcomes** (p. 672).
- **Insert Step Before**: Allows you to add operation with predefined controls before the selected **Step**. You can right-click the selected **Step** and click **Insert Step Before > Operation** to add the operation before the selected **Step** in the **Mesh Workflow**. The operations available are specific to selected **Step** depending on the selected **Mesh Workflow Type**.
- **Insert Step After**: Allows you to add operation with defined controls after the selected **Step**. You can right-click the selected **Step** and click **Insert Step After > Operation** to add the operation after the selected Step in the mesh workflow. The operations available are specific to selected **Step** depending on the selected **Mesh Workflow Type**.
- **Duplicate Step**: Allows you to duplicate the operation and add the duplicate with same controls at the end of mesh workflow steps. **Duplicate Step** follows the same constraints as **Insert > Step**, depending on the selected **Mesh Workflow Type**.
- **Duplicate Step Before**: Allows you to duplicate the operation and add the duplicate with same controls before the selected Step. **Duplicate Step Before** follows the same constraints as **Insert Step Before > Step**, depending on the selected **Mesh Workflow Type**.
- **Duplicate Step After**: Allows you to duplicate the operation and add the duplicate with same controls after the selected Step. **Duplicate Step** After follows the same constraints as **Insert Step After > Step**, depending on the selected **Mesh Workflow Type**.
- **Delete**: Allows you to delete the mesh workflow operation. You can delete an operation only when the mesh workflow is not complete.
- **Rename**: Allows you to rename the mesh workflow operation.
- **Revert To Step**: Allows you to revert to the current operation or step when checkpoint is enabled. When you perform **Revert To Step** after completing the workflow, the output generated is automatically cleared and reverted to the selected step.

Fill Holes

The **Fill Holes** operation allows you to fill the holes in the model. You can use **Hole filling** controls to define the holes to be filled. You can view the scoped entities in the Geometry window, which is highlighted in blue color.

Note:

All hole edges should ideally be scoped in one or more **Named Selections**, which helps to scope those edges by the corresponding labels in the **Hole filling** control. When you scope disconnected edges or edges, **Fill Holes** might fail to fill all holes and creates a FailingEdges label for diagnosis and provides an error message.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Fill Holes** operation have the following controls:

- **Hole Filling** (p. 633): Creates patch surfaces to close holes in the geometry. **Hole Filling** can be performed only on continuous edges.
- **Checkpoint** (p. 659): Provides a revert option that allows you to revert the **Fill Holes** operation.

The **Fill Holes** operation has the following outcome:

- **Scope** (p. 673): Scopes the created labels of the **Fill Holes** operation.

Note:

You can insert one or more **Hole Filling** (p. 633) control.

Wrap

The **Wrap** operation allows you to wrap the selected parts. **Wrap** allows you to extract a closed watertight surface used to create a volume mesh from geometry where the inputs

- are not connected
- have holes, leaks, or gaps
- have small features that need to be ignored or stepped over

The **Wrap** operation uses an appropriate material point to identify the relevant surfaces of the selected objects. You can create a watertight faceted representation from the boundaries between the regions of interest (identified through material points) and all other regions. The nodes on the faceted representation are then projected back to the input geometry, resulting in a wrapper surface closely representing the input geometry.

The **Wrap** operation allows you to wrap the input scope using the following controls:

- **Constant Size Wrapper** (p. 635): Wraps the input scope with the same size throughout the model.
- **Custom Names Wrapper** (p. 637): Creates entities with the names you provide.
- **Material Point** (p. 637): Defines fluid regions or seal regions depending on whether the type is Include or Exclude.
- **Leak Detection** (p. 638): Visualizes the leakages in the input scope so that you can close it.
- **Checkpoint** (p. 659): Provides a revert option that allows you to revert the **Wrap** operation.

The **Wrap** operation has specific outcomes that can be added as per your needs. You can right-click **Wrap**, click **Insert** and select the required scope to scope it as outcome. The outcome can be used as input scope in the consecutive operations. The available outcomes are:

- **Scope** (p. 673): Scopes the created parts of the **Wrap** operation.
- **Face Zone Scope** (p. 674): Scopes the created face zones in the **Wrap** operation.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Volume Zone Scope** (p. 674): Scopes the created volume zones in the **Wrap** operation.

Mesh Surface

The **Mesh Surface** operation allows you to improve the surface mesh. **Mesh Surface** operation allows you to mesh the surface using the constant size to improve the surface mesh quality.

The **Mesh Surface** operation has the following controls:

- **Constant Size Surface Mesher** (p. 638): Creates uniform mesh along the surface. Surface mesher uses triangular or quadrilateral mesh elements of the same size to create a mesh.
- **Wrapper Specific Surface Mesher** (p. 640): Creates surface meshing using sizes specific to wrapper and improves the surface mesh quality.
- **Checkpoint** (p. 659): Provides a revert option that allows you to revert the Mesh surface operation.

Convert To allows you to change from one control to another control in the operation. You can right-click a **Mesh Surface** control, click **Convert To** and select the respective control to which you want to convert the selected control in the operation.

Create Enclosure

The **Create Enclosure** operation creates the enclosure around the specified scope. The **Operation Type** is **Create Enclosure**.

The **Create Enclosure** operation has the following controls:

- **External Part Enclosure** (p. 641): Creates an external enclosure around the specified input scope using an existing part specified as external scope.
- **Spherical Enclosure** (p. 643): Creates a spherical enclosure from the input scope automatically with the provided parameters.
- **Hemispherical Enclosure** (p. 645): Creates a hemispherical enclosure from the input scope automatically with the provided parameters.
- **Convex Irregular Shape Enclosure** (p. 647): Creates a convex enclosure from the input scope automatically. The convex enclosure adapts to the input scope shape to minimize the internal enclosed volume using the provided parameters.
- **HemiConvex Irregular Shape Enclosure**: (p. 649) Creates a hemiconvex enclosure of the input scope. The hemiconvex enclosure adapts to the input scope shape to minimize the internal enclosed volume using the provided parameters.
- **Custom Names** (p. 652): Creates enclosures or extrusions with the provided name.

Convert To allows you to change from one control to another control in the operation. You can right-click a **Create Enclosure** control, click **Convert To** and select the respective control to which you want to convert the selected control in the operation.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Create Enclosure** operation has specific outcomes which can be added as per your needs. You can right-click **Create Enclosure**, click **Insert** and select the required scope to scope it as outcome. The outcome can be used as input scope in the consecutive operations. The available outcomes are:

- **Scope** (p. 673): Scopes the created zones of the **Create Enclosure** operation.
- **Internal Enclosure Scope** (p. 674): Scopes the internal enclosure created in the **Create Enclosure** operation.
- **External Enclosure Scope** (p. 675): Scopes the external enclosure created in the **Create Enclosure** operation.
- **Face Zone Scope** (p. 674): Scopes the created face zones in the **Create Enclosure** operation.
- **Volume Zone Scope** (p. 674): Scopes the created volume zones in the **Create Enclosure** operation.
- **Internal Volume Zone Scope** (p. 676): Scopes the internal volume zones created in the **Create Enclosure** operation.

Mesh Volume

The **Mesh Volume** operation creates volumetric mesh. You can use **Constant Size Volume Mesher** or **Size Filed Volume Mesher** controls to generate the volume mesh.

The **Mesh Volume** operation has the following controls:

- **Constant Size Volume Mesher** (p. 652): Creates a volume mesh of uniform size on the entire volume.
- **Size Field Volume Mesher** (p. 668): Mesh the model with the sizing defined in the size field provided as input.
- **Material Point** (p. 637): Defines fluid regions or seal regions depending on whether the type is Include or Exclude.
- **Checkpoint** (p. 659): Provides a revert option that allows you to revert the **Improve Volume Mesh** operation.

The **Mesh Volume** operation have the following outcome:

- **Scope** (p. 673): Scopes the created zone of the **Mesh Volume** operation.

Improve Volume Mesh

The **Improve Volume Mesh** operation allows you to improve the volumetric mesh generated in the **Mesh Volume** step.

The **Improve Volume Mesh** operation has the following control types:

- **Volume Mesh Improvement** (p. 654): Improves the quality of the volume mesh with specified quality metrics using the provided input options.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Second Order Conversion** (p. 655): Improves the quality of the volume mesh with specified quality metrics using the provided input options.
- **Checkpoint** (p. 659): Provides a revert option that allows you to revert the **Improve Volume Mesh** operation.

Extrude

The **Extrude** operation generates layers of prismatic elements from the input surface to a specified height. The generated volume may have layers of constant size hexahedral or triangular prism elements depending on the input surface.

Note:

Extrude does not support inputs with topology.

The **Extrude** operation has the following controls:

- **Extrusion** (p. 655): Creates extruded layers from the input surface for a specified number of layers with specified height.
- **Custom Names** (p. 652): Creates enclosures or extrusions with the provided name.
- **Checkpoint** (p. 659) Provides a revert option that allows you to revert the **Extrude** operation.

The **Extrude** operation has specific outcomes which can be added as per your needs. You can right-click **Extrude** operation, click **Insert** and select the required scope that you need as outcome in the consecutive operations. The available outcomes are:

- **Extrusion Start Scope** (p. 675): Scopes the base face used for extruding.
- **Extrusion End Scope** (p. 675): Scopes the end face created after extrusion.
- **Face Zone Scope** (p. 674): Scopes the created face zones in the **Extrude** operation.
- **Volume Zone Scope** (p. 674): Scopes the created volume zones in the **Extrude** operation.

Create Topology

The **Create Topology** operation allows you to create the topology from the mesh only model. Scoped entities for topology creation can be defined in **Topology Creation** control.

The **Create Topology** operation has the following controls:

- **Topology Creation** (p. 657): Defines the scope to create the topology from the mesh.
- **Checkpoint** (p. 659): Provides a revert option that allows you to revert the **Create Topology** operation.

Merge Volumes

The **Merge Volumes** operation allows you to merge the volumes.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Merge Volumes** operation has the following controls:

- **Volumes Merging** (p. 657): Merges all the volumes that share the input faces defined on the scope into a single volume.
- **Checkpoint** (p. 659): Provides a revert option that allows you to revert the **Merge Volumes** operation.

The **Merge Volumes** operation has specific outcomes which can be added as per your needs. You can right-click **Merge Volumes**, click **Insert** and select the required scope that you need as outcome in the consecutive operations. The available outcome is:

- **Volume Zone Scope** (p. 674): Allows you to scope the created volume zones in the selected operation.

Manage Zone Properties

The **Manage Zone Properties** operation allow you to define material properties and thickness on the scoped parts or zones. On completing the Workflow, the defined properties on the scoped zones or parts are assigned to the geometry parts or bodies.

Note:

The **Manage Zone Properties** operation is not a mandatory step for completing the mesh workflow. You can also assign properties to the newly created geometry after completing the workflow.

The **Manage Zone Properties** operation have the following controls:

- **Zone Material Assignment** (p. 658): Defines the material assigned to zones.
- **Zone Thickness Assignment** (p. 658): Defines the thickness assigned to zones.
- **Checkpoint** (p. 659): Provides a revert option that allows you to revert the **Manage Zone Properties** operation.

Patch Holes

The **Patch Holes** operation allows you to create patching surfaces for regions identified using the **Material Point** and **Hole Patching** control properties.

The **Patch Holes** operation has the following controls:

- **Hole Patching** (p. 659): Allows you to patch holes in the model to close the volume defined by the provided Include and Exclude material points in the **Material Point** control.
- **Material Point** (p. 637): Defines fluid regions or seal regions depending on whether the type is Include or Exclude.
- **Checkpoint** (p. 659) Provides a revert option that allows you to revert the **Patch Holes** operation.

The **Patch Holes** operation has the following outcome:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Scope** (p. 673): Scopes the created labels of the **Patch Holes** operation.

Read Mesh

The **Read Mesh** operation allows you to read or append the mesh from a supported file (.pmdat,.msh,.cdb) to the mesh workflow model. The **Operation Type** is **Read Mesh**.

Note:

You can only insert **Read Mesh** before the first **Step** in a predefined **Mesh Workflow**.

The **Read Mesh** operation has the following controls:

- **Mesh Reading** (p. 660): Read a mesh into the mesh workflow model.
- **Checkpoint** (p. 659): Provides a revert option that allows you to revert the **Read Mesh** operation.

Write Mesh

The **Write Mesh** operation allows you to write the mesh created during the mesh workflow operation in the model. The **Write Mesh** operation supports only .pmdat, .msh, .cdb file extension. The **Operation Type** is **Write Mesh**.

Note:

You can only insert **Write Mesh** after the last **Step** in a predefined **Mesh Workflow**.

The **Write Mesh** operation has the following controls:

- **Mesh Writing** (p. 660): Writes and save the mesh generated in the mesh workflow for the selected step.
- **Checkpoint** (p. 659): Provides a revert option that allows you to revert the **Write Mesh** operation.

Create Size Field

Create Size Field step allows you to create size field for the mesh workflow model. The created size field depends on the defined size controls and the selected scope. Always, the smaller size take precedence over the larger size.

Create Size Field has the following controls:

- **Checkpoint** (p. 659)
- **Custom Names** (p. 660)
- **Constant Sizing** (p. 661)
- **Curvature Sizing** (p. 662)
- **Proximity Sizing** (p. 664)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Body of Influence Sizing** (p. 665)

Create Size Field step has specific outcomes which can be added as per your needs. You can right-click **Create Size Field**, click **Insert** and select the required scope that you need as outcome in the consecutive operations. The available outcome is:

- **Size Field Name** (p. 676)

Mesh Workflow Controls

Mesh Workflow Controls determine the execution of **Mesh Workflow Steps** based on the set of defined controls. **Mesh Workflow Controls** allow you to provide inputs for executing the respective **Mesh Workflow Steps**. Each **Mesh Workflow Step** has one or more specific controls that are mandatory or optional. When you define a **Mesh Workflow Step**, the mandatory controls are automatically added. You should define the optional controls for the **Step** as per your requirement.

Right-click options available in **Mesh Controls** are:

- **Rename**: Allows you to rename the mesh workflow controls.
- **Delete**: Allows you to delete the mesh workflow controls.

Each **Control** has the following options in its **Details** view:

General

- **Control Type**: Displays the selected control type.

Scope

- **Define By**: Allows you to define the input to the selected control. The available options are **Value** and **Outcome**.
  - **Value**: Allows you to set manually the value of the **Scoping Method** and **Scoping Pattern**.
  - **Outcome**: Allows you to select the existing scoped outcomes from the previous steps as input.
- **Scoping Method**: Allows you to select the entities for the selected control. The available options are:
  - **Part**: Allows you to select Parts for defining the scope of the control.
  - **Label**: Allows you to select Labels for defining the scope of the control.
  - **Zone**: Allows you to select Zones for defining the scope of the control.
- **Scoping Pattern**: Allows you to specify the name pattern to get the selected **Scoping Method** with the provided name pattern. **Scoping Pattern** supports the**Regular Expression**. The following patterns are supported:

Expression	Description	Example

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

.*	Matches any number of any characters including none.	`can.*` matches all strings beginning with "can". can, cane, candle, cancel, and so on `.*can`matches all strings ending with "can". Glucan, decan, scan, and so on.
?	Matches exactly one character.	`?an` matches "can," "tan," "man," "ran," and so on.
[...]	Matches one character from the characters given in brackets. A pair of characters separated by a hyphen denotes a range of characters to be matched.	`[ct]an` matches "can" or "tan." `Can[0-5]` matches "Can0," "Can1," "Can2," "Can3," "Can4," and "Can5" `Can[^2-3]` matches "Can1" but does not match "Can2" or "Can3."
(?!...)	Negative lookahead. Matches any string that does not have the provided string.	`(?!Can).*` matches any string that does not begin with "Can". `(?!Can.*|bin.*).*` matches any string that does not begin with "Can" or bin. `(?!.*can$).*` matches all strings that do not end with "can".
|	Separates multiple names while scoping. Separates multiple expressions to match.	`Can|Decan|scan` matches "Can," any string starting with "Decan," and any string containing "scan." `Can|hole.*|.*pipe` matches "Can," any string starting with "hole," and any string ending with "pipe."

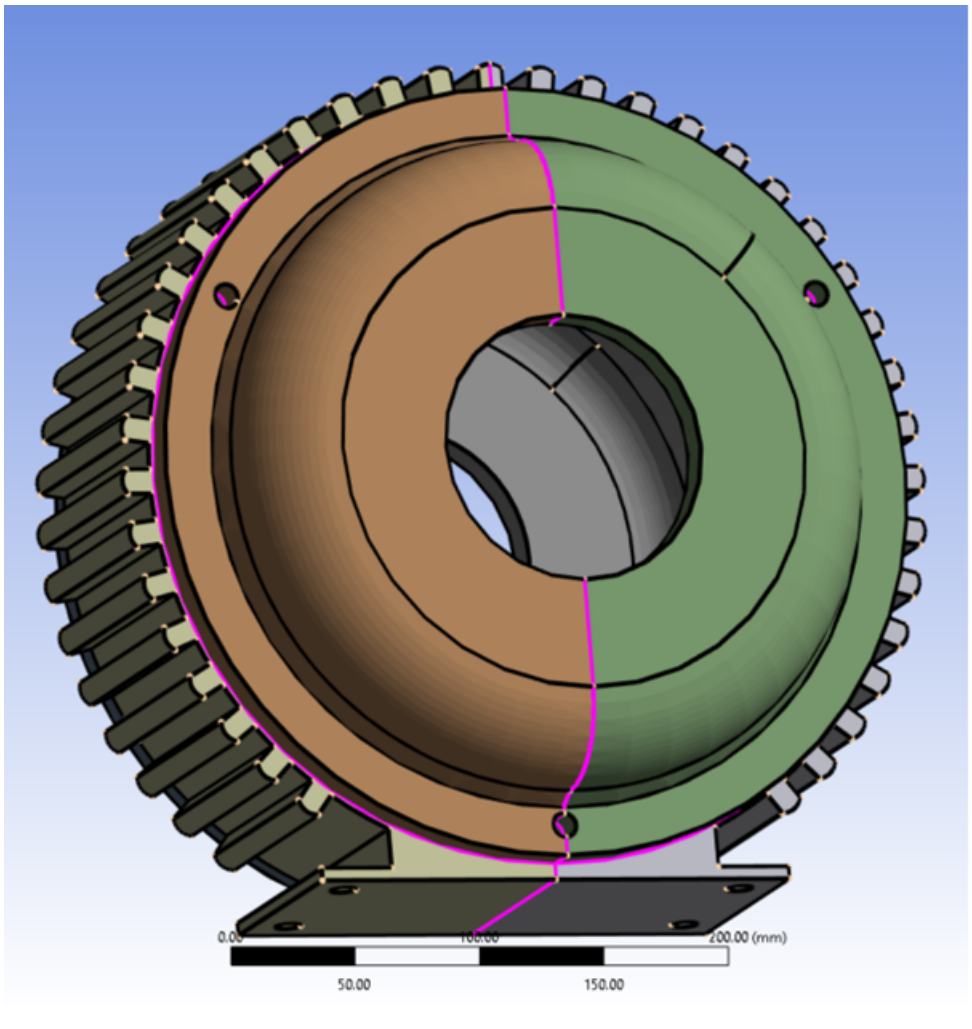
Note:

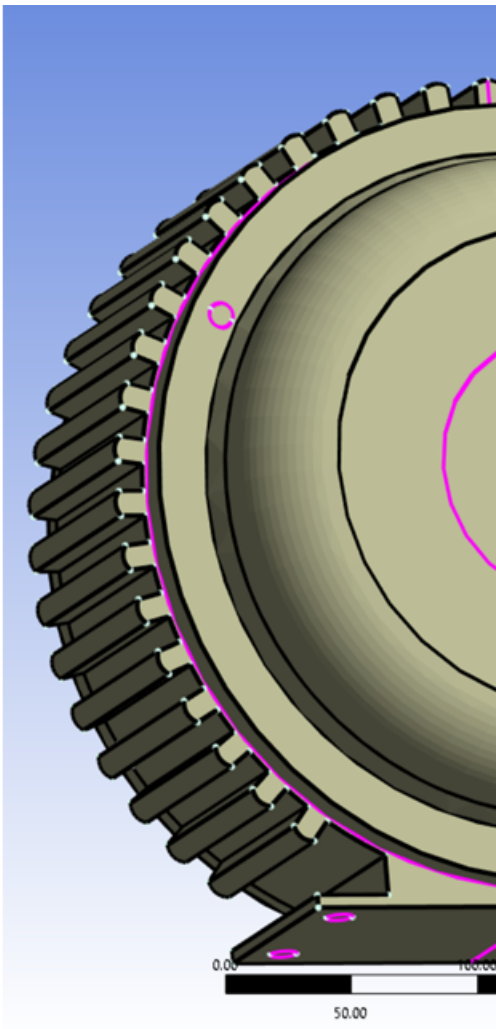
- You should always use **** character with special characters like **{},[],(),^,$,.,|,*,+,?,** . For more information, refer Regular Expression.
- You should be cautious while reading names in right-to-left scripts. When applying matching patterns for such names, you should consider the implicit reading direction while building the pattern.

Hole Filling

Hole Filling allows you to create patch surfaces to close holes in the surface meshes. **Hole Filling** can be performed only on continuous edges.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Hole Filling Details view has the following options:

General

- **Control Type** (p. ?)

Scope

- **Scoping Method** (p. ?)
- **Scoping Pattern** (p. ?)

Definition

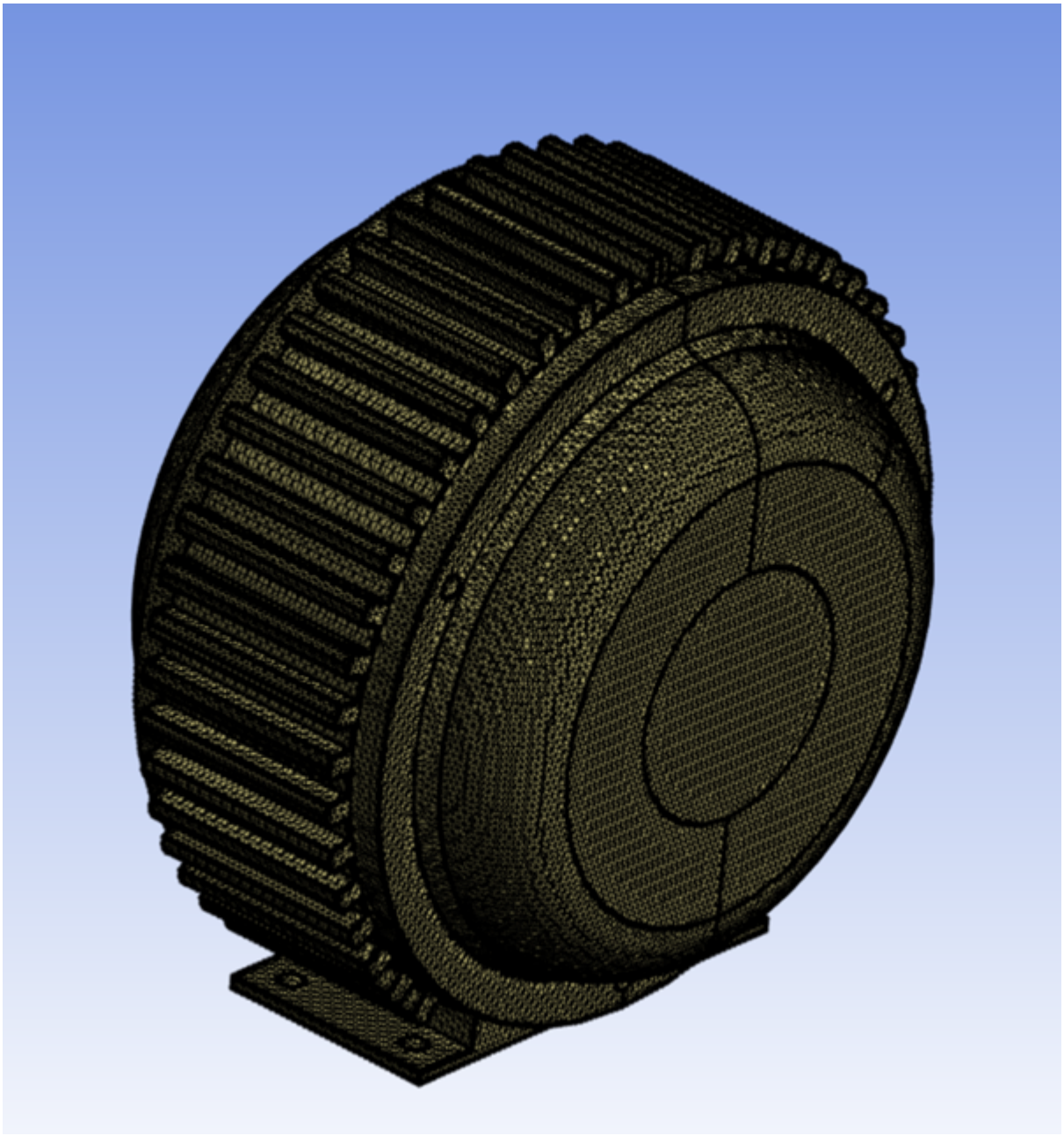
- **Topology Connection**: Allows you to create a connected topology for the newly created faces when **Topology Connection** is **Yes**. The default value is **No**. When **Topology Connection** is **Yes**, hole edges should be continuous to perform fill holes operation.
- **Part Name**: Allows you to provide the name of the created part where new faces are defined.
- **Face Zone Name**: Allows you to name the created face zone.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Label Name**: Allows you to label the created face zone with the specified name.

Constant Size Wrapper

Constant Size Wrapper wraps the input scope with the same size throughout the model.



Constant Size Wrapper Details view has the following options:

General

- **Control Type** (p. ?)

Scope

- **Scoping Method** (p. ?)

Only **Part** can be selected for Constant Size Wrapping.

- **Scoping Pattern** (p. ?)

Definition

- **Define By**: Allows you to define the element size based on value or settings. The available options are:
  - **Value**: Defines the element size based on the provided value.
- **Settings**: Defines the element size based on the settings under Mesh Settings in the Steps Details view.
- **Element Size**: Provides the minimum feature element size of the input scope to be considered for wrap operation. When **Define By** is **Value**, you can specify the element size for wrapping. When **Define By** is **Settings**, displays the element size calculated based on the provided **Mesh Settings** in the **Steps Details** view. The **Element Size** is read-only. You can click on the right corner of the option and click **Publish** to publish element size to the **Property Worksheet**. You can parameterize the **Element Size** only when **Defined By** is **Value**.
- **Mesh Type**: Allows you to select the mesh type. When **Mesh Type** is **Triangles**, generates triangular mesh. When **Mesh Type** is **Quadrilaterals**, generates quad mesh. The default value is **Triangles**.
- **Live Region Type**: Allows you to select the region type to extract the wrap region. The available options are:
  - **Material Point**: Allows you to define the volume of the model being wrapped. The material point is set locally along the X, Y and Z coordinates.
  - **External**: Allows you to wrap the surface from the external region. For **External FEM Acoustics** and **BEM Acoustics** workflows, the default value is **External**.
- **Delete Input Scope**: Allows you to delete the input source of the model after wrapping when **Delete Input Scope** is **Yes**.
- **Exclude Enclosure**: Allows you to exclude the external region and wrap inner parts only for the box in box models when **Exclude Enclosure** is **Yes**. The default value is **No**.
- **Face Zone By Part**: Creates a face zone for each part of the input scope when **Face Zone By Part** is **Yes**. The created face zones are converted to named selections in **Mechanical** after completing the mesh workflow. When **Face Zone By Part** is **No**, creates a single face zone for the whole input scope. Hence, only a single named selection is available for **Mechanical** after completing the mesh workflow. The default value is **No**.
- **Reverse Surface Orientation**: Allows you to reverse the orientation of the created wrapper surfaces. The default value is **No**. When **Reverse Surface Orientation** is **No**, the orientation is same as the position of the **Live** material point for the wrapper. When **Reverse Surface Orientation** is **Yes**, the orientation is opposite to the position of the **Live** material point for wrapper. **BEM Acoustics** uses **Reverse Surface Orientation** where you need to wrap the external and solve the internal field.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Custom Names Wrapper

Custom Names Wrapper control creates entities with the names you provide. **Custom Names Wrapper Details** view has the following options:

General

- **Control Type** (p. ?)

Definition

- **Part Name**: Allows you to provide name for the part that the wrapper creates enclosing the volume.
- **Volume Zone Name**: Allows you to provide name for the volume zone of the created part.
- **Face Zone Name**: Allows you to provide name for the face zone forming the boundary of the enclosed volume.
- **Edge Label Name**: Allows you to label the created feature edges with the provided names.

Material Point

Material Point defines fluid regions or seal regions depending on whether the type is **Include** or **Exclude**. A 3D coordinate provides the position of the material point along the X, Y and Z coordinates. You can parametrize **X Coordinate**, **Y Coordinate** and **Z Coordinate**. **Material Point Details** view has the following options:

General

- **Control Type** (p. ?)

Definition

- **Define By**: Allows you to define the material point. The available options are:
  - **Location**: Allows you to specify the location coordinates of the material point. The available options are:
    - → **X Coordinate**: Allows you to specify the X coordinate of the material point.
    - → **Y Coordinate**: Allows you to specify the Y coordinate of the material point.
    - → **Z Coordinate**: Allows you to specify the Z coordinate of the material point.

    You can parametrize the X,Y,Z coordinates.
  - **Coordinate System**: Allows you to select the defined coordinate systems to define material points. You can click to select from the available list of defined Coordinate Systems.
- **Material Point Type**: Allows you to select the type of material point. The available options are **Include** and **Exclude**. The default value is **Include**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Include**: Defines material points inside the wrapped volume.

– **Exclude**: Defines material points outside the wrapped volume.

Leak Detection

Leak Detection allows you to visualize the leakages in the input scope so that you can close it. **Leak Detection Details** view has the following options:

General

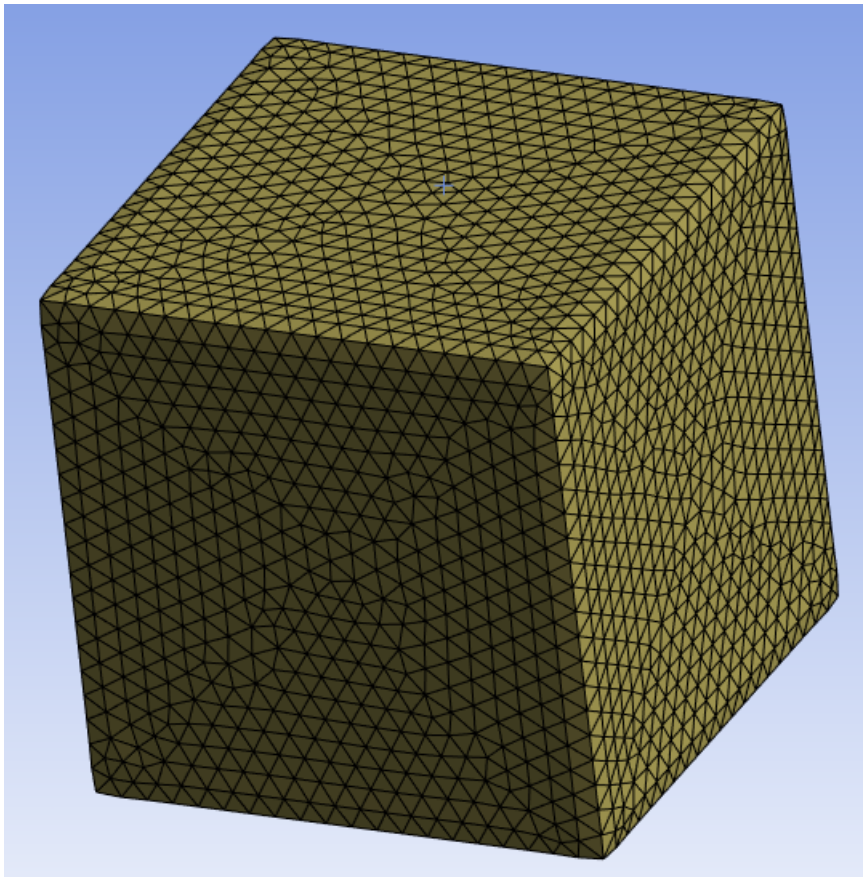
- **Control Type** (p. ?)

Definition

- **Max Hole Size**: Allows you to provide the maximum hole size to prevent leakage. You can click on the right corner of the option and click **Publish** to publish **Max Hole Size** to the **Property Worksheet**. You can parameterize **Max Hole Size**.

Constant Size Surface Mesher

Constant Size Surface Mesher control creates a surface mesh on the scoped set of entities consisting of triangular or quadrilateral mesh elements of same size.



Constant Size Surface Mesher Details view has the following options:

General

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Control Type** (p. ?): Displays the selected control type.

Scope

- **Define By** (p. ?): Allows you to define the input to the selected control. The available options are:
  - **Value**: Allows you to set manually the value of the **Scoping Method** and **Scoping Pattern**.
  - **Outcome**: Allows you to select the existing scoped outcomes from the previous steps as input.

  and .
- **Scoping Method** (p. ?): Allows you to select the entities for the selected control. The available options are:
  - **Part**: Allows you to select Parts for defining the scope of the control
  - **Label**: Allows you to select Labels for defining the scope of the control.
  - **Zone**: Allows you to select Zones for defining the scope of the control.
- **Scoping Pattern** (p. ?): Allows you to specify the name pattern to get the selected **Scoping Method**. **Scoping Pattern** supports **Regular Expression**.

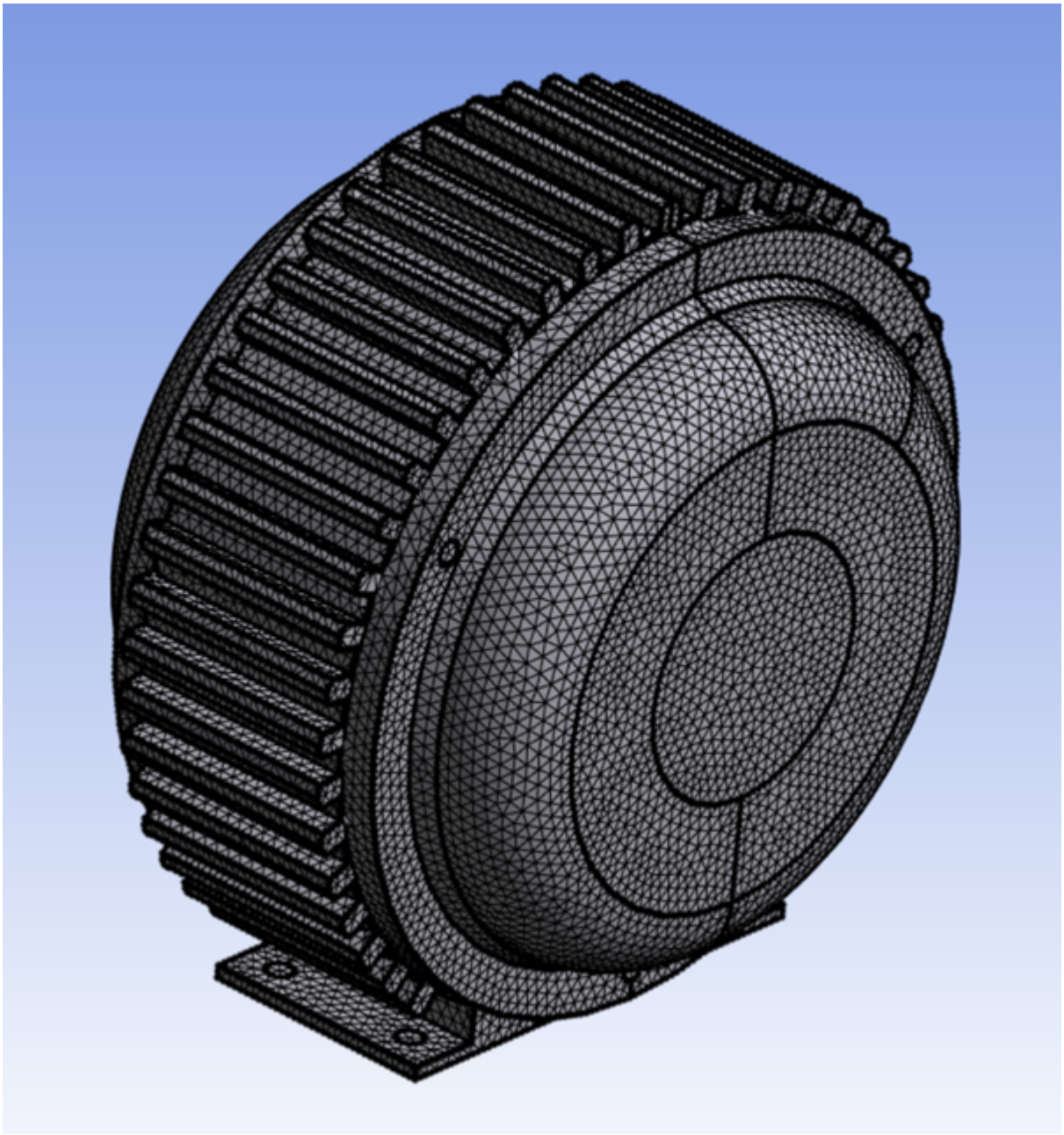
Definition

- **Define By**: Allows you to define the element size based on value or settings. The available options are:
  - **Value**: Defines the element size based on the provided value.
  - **Settings**: Defines the element size based on the settings under **Mesh Settings** in the **Steps Details** view.
- **Element Size**: Provides the element size for surface meshing. When **Define By** is **Value**, you can specify the element size for surface meshing. When **Define By** is **Settings**, displays the element size calculated based on the provided **Mesh Settings** in the **Steps Details** view. The **Element Size** is read-only. You can click on the right corner of the option and click **Publish** to publish **Element Size** to the **Property Worksheet**. You can parameterize **Element Size** only when **Define By** is **Value**.
- **Mesh Type**: Allows you to select the type of mesh you want to generate. The default value is **Triangles**. The available options are:
  - **Triangles**: Creates mesh with triangular elements.
  - **Quadrilateral**: Creates mesh with quadrilateral elements.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Wrapper Specific Surface Mesher

Wrapper Specific Surface Mesher provides post wrapping improvement operation using surface meshing based on the mesh settings and surface quality measures. For example, you can improve the wrapper generated surface by remeshing based on the current size field and target skewness. You should use **Wrapper Specific Surface Mesher** only after wrapping.



Wrapper Specific Surface Mesher Details view has the following options:

General

- **Control Type** (p. ?): Displays the selected control type.

Scope

- **Define By** (p. ?)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Scoping Method** (p. ?)

  Only **Part** can be selected for surface meshing.

- **Scoping Pattern** (p. ?)

Definition

- **Define By**: Allows you to define the element size based on value or settings. The available options are:
  - **Value**: Defines the element size based on the provided value.
  - **Settings**: Defines the element size based on the settings under **Mesh Settings** in the **Steps Details** view.
- **Element Size**: Provides the element size. When **Define By** is **Value**, you can specify the element size for surface meshing. When **Define By** is **Settings**, displays the element size calculated based on the provided **Mesh Settings** in the **Steps Details** view. The **Element Size** is read-only. You can click on the right corner of the option and click **Publish** to publish **Element Size** to the **Property Worksheet**. You can parameterize **Element Size** only when **Defined By** is **Value**.

  Note:

  If you apply size fields for the **Wrapper Specific Surface Mesher**, the **Max Size** specified in the size field takes precedence over the **Element Size**.

- **Target Skewness**: Allows you to provide the target skewness to improve surface mesh of wrapper part. The Target Skewness value ranges from **0** (low quality) to **1** (high quality). The default value is **0.6**. You can click on the right corner of the option and click **Publish** to add **Target Skewness** to the **Property Worksheet**. You can parameterize **Target Skewness**.
- **Mesh Type**: Allows you to select the type of mesh you want to generate. The default value is **Triangles**. The available options are:
  - **Triangles**: Creates mesh with triangular elements.
  - **Quadrilateral**: Creates mesh with quadrilateral elements.

External Part Enclosure

External Part Enclosure creates an external enclosure around the specified input scope using an existing part specified as external scope. **External Part Enclosure Details** view has the following options:

General

- **Control Type** (p. ?)

Scope

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Define By**: Allows you to define the input to the selected control. The available options are **Value** and **Outcome**.
  - **Value**: Allows you to set manually the value of the **Inner Source Scoping Method** and **Inner Source Scoping Pattern**.
  - **Outcome**: Allows you to select the existing scoped outcomes from the previous steps as input.
- **Inner Source Scoping Method**: Allows you to select the **Part**, **Label** or **Zone** from the inner source. The available options are:
  - **Part**: Allows you to select parts for the inner source scoping.
  - **Label**: Allows you to select labels for the inner source scoping.
  - **Zone**: Allows you to select the zones for the inner source scoping.
- **Inner Source Scoping Pattern**: Allows you to specify the pattern to match the **Part**, **Label** or **Zone** based on the selected **Inner Source Scoping Method**. **Inner Source Scoping Pattern** supports all **regular expressions** (p. ?).
- **Define By**: Allows you to define the input to the selected control. The available options are **Value** and **Outcome**.
  - **Value**: Allows you to set manually the value of the **External Source Scoping Method** and **External Source Scoping Pattern**.
  - **Outcome**: Allows you to select the existing scoped outcomes from the previous steps as input.
- **External Source Scoping Method**: Allows you to select the **Part** from the external source. The available options are:
  - **Part**: Allows you to select parts of the external source.

  Note:

  External Scoping Method is not available when **Auto External Scope** is set to **Yes**.

- **External Source Scoping Pattern**: Allows you to specify the pattern to match the **Part** based on the selected **External Scoping Method**. **External Source Scoping Pattern** supports all **regular expressions** (p. ?).

Definition

- **Auto External Scope**: Derives the external scope for the enclosure from the input model automatically when **Auto External Scope** is **Yes**. The default value is **No**.
- **Define By**: Allows you to define the element size based on value or settings. The available options are:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

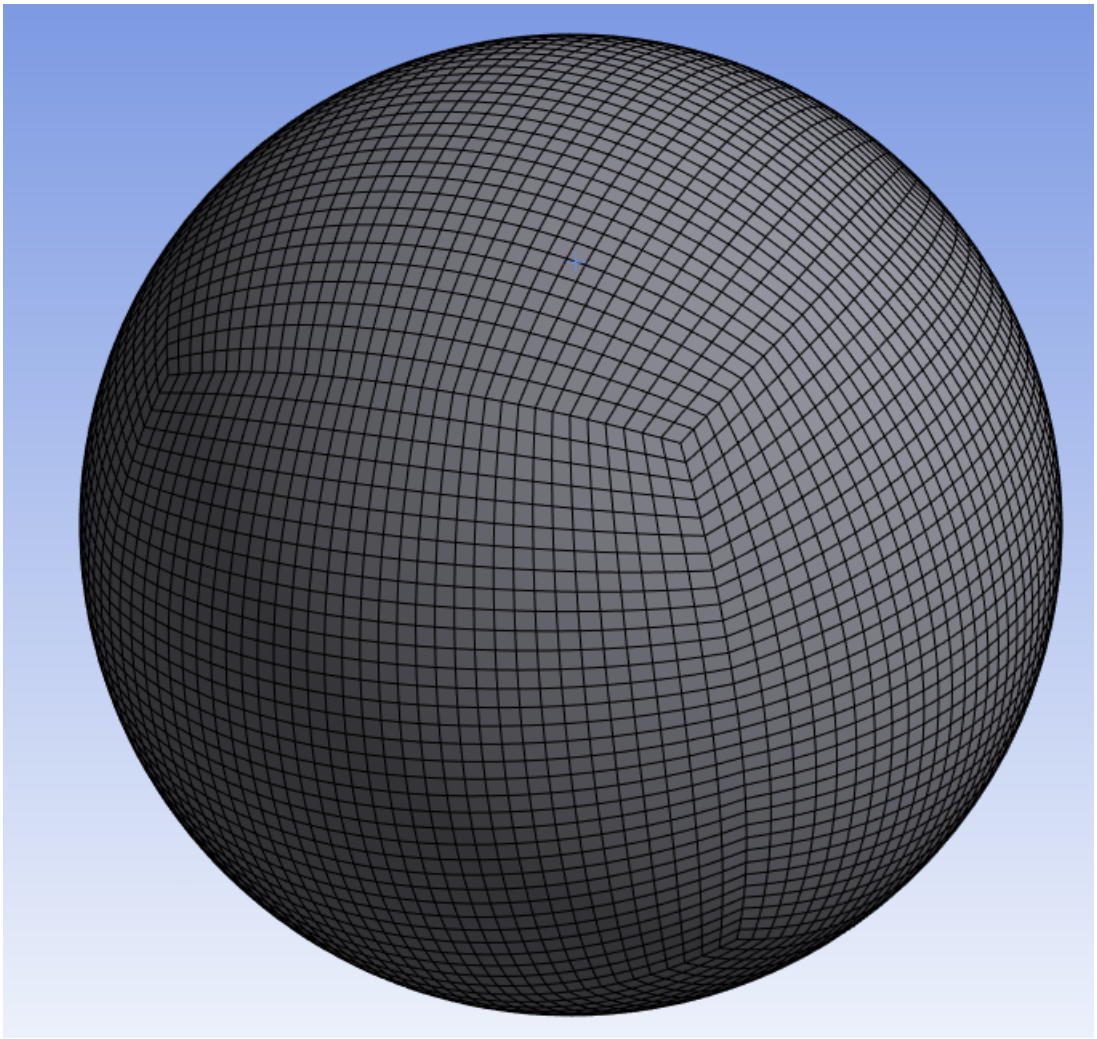
  - **Value**: Defines the element size based on the provided value.
  - **Settings**: Defines the element size based on the settings under **Mesh Settings** in the **Steps Details** view.

- **Element Size**: Provides the element size. When **Define By** is **Value**, you can specify the element size. When **Define By** is **Settings**, displays the element size calculated based on the provided **Mesh Settings** in the **Steps Details** view. The **Element Size** is read-only. You can click on the right corner of the option and click **Publish** to publish **Element Size** to the **Property Worksheet**. You can parameterize **Element Size** only when **Defined By** is **Value**.
- **Mesh Type**: Allows you to provide the type of mesh. The default value is **Quadrilaterals**. The available options are:
  - **Triangles**: Creates mesh with triangular element.
  - **Quadrilaterals**: Creates mesh with quadrilateral elements.
- **Smoothing Iterations**: Allows you to provide the number of smoothing iterations required. The default value is **0**. You can click on the right corner of the option and click **Publish** to publish **Smoothing Iterations** to the **Property Worksheet**.
- **Smoothing Preserve Volume**: Allows you to preserve the volume after smoothing when **Smoothing Preserve Volume** is **Yes**. The default value is **No**.

Spherical Enclosure

Spherical Enclosure control automatically creates a spherical enclosure for the input scope with the provided parameters.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Spherical Enclosure Details view has the following options:

General

- **Control Type** (p. ?)

Scope

- **Define By** (p. ?)
- **Scoping Method** (p. ?)

  Only **Part** can be selected for creating spherical enclosure.

- **Scoping Pattern** (p. ?)

Definition

- **Define By**: Allows you to define the element size based on value or settings. The available options are:

  – **Value**: Defines the element size based on the provided value.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Settings**: Defines the element size based on the settings under **Mesh Settings** in the **Steps Details** view.

- **Element Size**: Allows you to provide the element size. When **Define By** is **Value**, you can specify the element size. When **Define By** is **Settings**, displays the element size calculated based on the provided **Mesh Settings** in the **Steps Details** view. The **Element Size** is read-only. You can click on the right corner of the option and click **Publish** to publish **Element Size** to the **Property Worksheet**. You can parameterize **Element Size** only when **Defined By** is **Value**.

- **Center Modality**: Allows you to define the center of the spherical enclosure. The default value is **Minimal**. The available options are:

  - **Minimal**: Uses the center of the minimal enclosure sphere.

  - **Centered**: Uses the center of the bounding box of the model.

  - **User Defined**: Allows you to define the center using the provided location coordinate (X, Y, Z).

- **Mesh Type**: Allows you to provide the type of mesh. The default value is **Quadrilaterals**. The available options are:

  - **Triangles**: Creates mesh with triangular elements.

  - **Quadrilaterals**: Creates mesh with quadrilateral elements.

- **Minimum Absolute Radius**: Allows you to provide the minimum radius for the spherical enclosure. The default value is **0.0**. You can click on the right corner of the option and click **Publish** to publish **Minimum Absolute Radius** to the **Property Worksheet**. You can parameterize **Minimum Absolute Radius**.

- **Minimum Number of Layers**: Allows you to provide the minimum number of layers between the model and the enclosure. You can parametrize **Minimum Number of Layers**. The default value is **2**. You can click on the right corner of the option and click **Publish** to publish **Minimum Number of Layers** to the **Property Worksheet**.

- **Percentage Increment**: Allows you to specify the minimal percentage increment of the enclosure dimensions with respect to the model. The default value is **1.0**. You can click on the right corner of the option and click **Publish** to publish **Percentage Increment** to the **Property Worksheet**.

Hemispherical Enclosure

Hemispherical Enclosure control automatically creates a hemispherical enclosure for the input scope with the provided parameters. **Hemispherical Enclosure Details** view has the following options:

General

- **Control Type** (p. ?)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scope

- **Define By** (p. ?)
- **Scoping Method** (p. ?)

  Only **Part** can be selected for creating hemispherical enclosure.

- **Scoping Pattern** (p. ?)

Definition

- **Define By**: Allows you to define the element size based on value or settings. The available options are:
  - **Value**: Defines the element size based on the provided value.
  - **Settings**: Defines the element size based on the settings under **Mesh Settings** in the **Steps Details** view.
- **Element Size**: Allows you to provide the element size. When **Define By** is **Value**, you can specify the element size. When **Define By** is **Settings**, displays the element size calculated based on the provided **Mesh Settings** in the **Steps Details** view. The **Element Size** is read-only. You can click on the right corner of the option and click **Publish** to publish **Element Size** to the **Property Worksheet**.
- **Center Modality**: Allows you to define the center of the hemispherical enclosure. The default value is **Centered**. The available options are:
  - **Centered**: Uses the center of the input scope projected on the plane defined along the orientation.
  - **User Defined**: Allows you to define the center using the provided location coordinate (X, Y, Z). The available options are :
    - → **X Coordinate**: Allows you to provide the location coordinate along X-axis.
    - → **Y Coordinate**: Allows you to provide the location coordinate along Y-axis.
    - → **Z Coordinate**: Allows you to provide the location coordinate along Z-axis.
- **Orientation Modality**: Allows you to define the orientation of the hemisphere. The default value is **Plus Z.** The available options are **Plus X**, **Plus Y**, **Plus Z**, **Minus X**, **Minus Y**, **Minus Z**.
- **Plane Distance**: Allows you to provide the distance from the base of the hemisphere to the model.
- **Open Closure**: Allows you to delete the base of the hemisphere while creating an open enclosure when **Open Closure** is **Yes**. The default value is **No**.
- **Delete Colliding Faces**: Allows you to delete any model face colliding with hemisphere base when **Delete Colliding Faces** is **Yes**. The default value is **No**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mesh Type**: Allows you to provide the type of mesh. The default value is **Quadrilaterals**. The available options are:
  - **Triangles**: Creates mesh with triangular elements.
  - **Quadrilaterals**: Creates mesh with quadrilateral elements.

  Note:

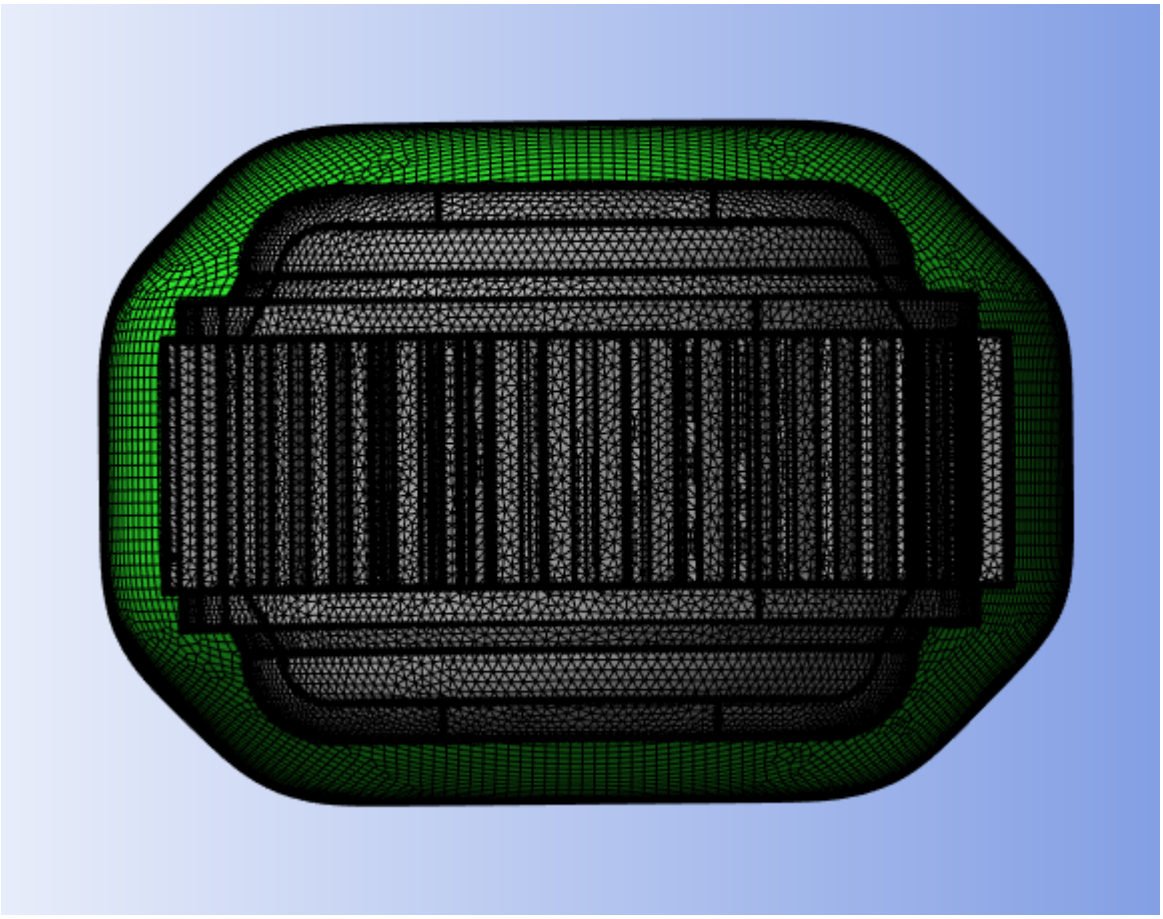
  Hemispherical Enclosure creates quadrilateral mesh for the hemisphere and triangular mesh for the base.

- **Minimum Absolute Radius**: Allows you to provide the minimum radius for the hemispherical enclosure. The default value is **0.0**. You can click on the right corner of the option and click **Publish** to publish **Minimum Absolute Radius** to the **Property Worksheet**. You can parameterize **Minimum Absolute Radius**.
- **Minimum Number of Layers**: Allows you to provide the minimum number of layers between the model and the enclosure. You can parametrize **Minimum Number of Layers**. The default value is **2**. You can click on the right corner of the option and click **Publish** to publish **Minimum Number of Layers** to the **Property Worksheet**.
- **Percentage Increment**: Allows you to specify the minimal percentage increment of the enclosure dimensions with respect to the model. The default value is **1.0**. You can click on the right corner of the option and click **Publish** to publish **Percentage Increment** to the **Property Worksheet**.

Convex Irregular Shape Enclosure

Convex Irregular Shape Enclosure control automatically creates a convex enclosure of the input scope. The convex enclosure adapts to the input scope shape to minimize the internal enclosed volume using the provided parameters.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Convex Irregular Shape Enclosure Details view has the following options:

General

- **Control Type** (p. ?)

Scope

- **Define By** (p. ?)
- **Scoping Method** (p. ?)

  Only **Part** can be selected for creating **Convex Irregular Shape Enclosure**.

- **Scoping Pattern** (p. ?)

Definition

- **Define By**: Allows you to define the element size based on value or settings. The available options are:
  - **Value**: Defines the element size based on the provided value.
  - **Settings**: Defines the element size based on the settings under **Mesh Settings** in the **Steps Details** view.

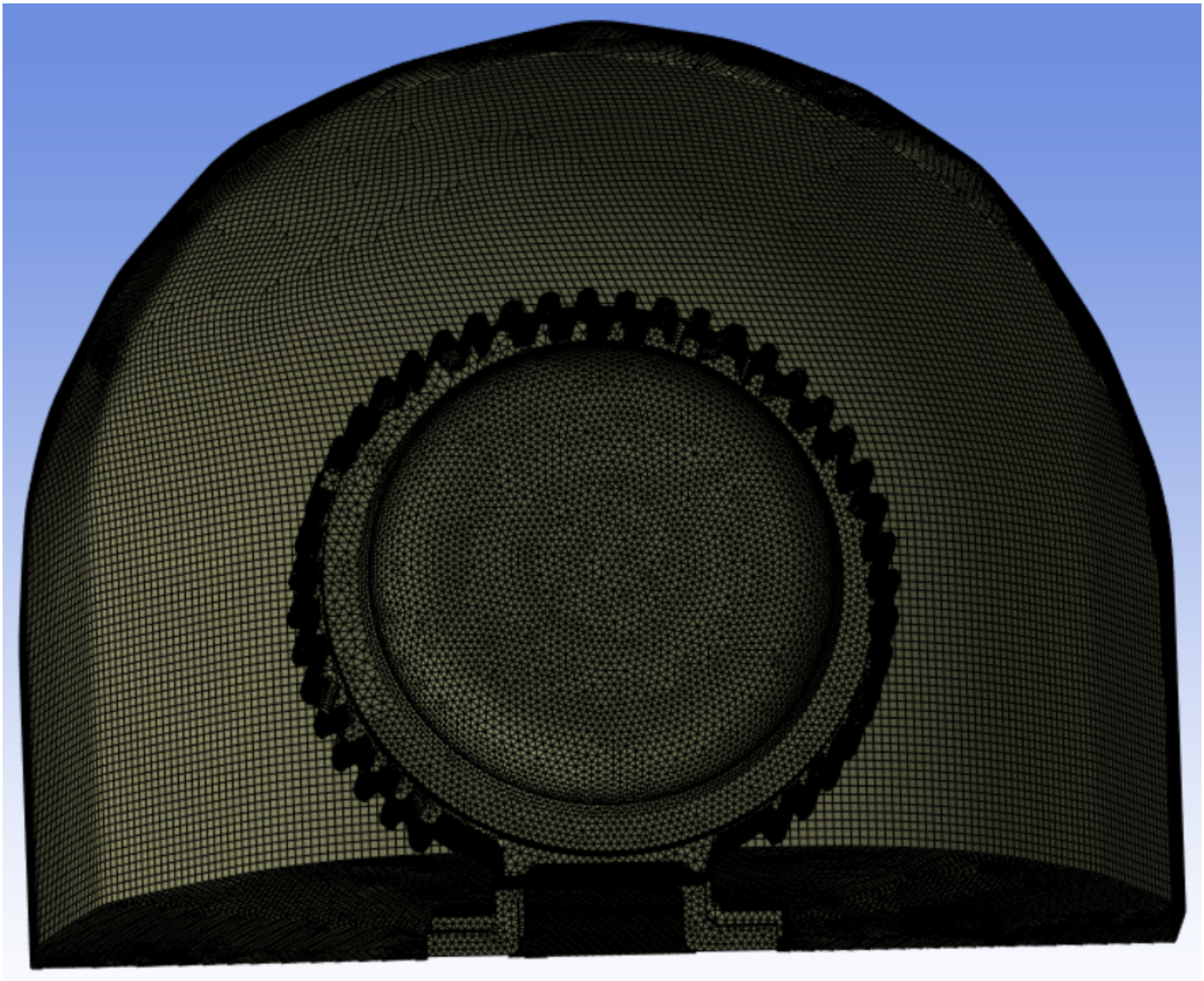
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Element Size**: Provides the element size. When **Define By** is **Value**, you can specify the element size. When **Define By** is **Settings**, displays the element size calculated based on the provided **Mesh Settings** in the **Steps Details** view. The **Element Size** is read-only. You can click on the right corner of the option and click **Publish** to publish **Element Size** to the **Property Worksheet**. You can parametrize **Element Size** only when **Defined By** is **Value**.
- **Mesh Type**: Allows you to select the type of mesh to be used for meshing. The default value is **Quadrilaterals**. The available options are:
  - **Triangles**: Creates mesh with triangular elements.
  - **Quadrilaterals**: Creates mesh with quadrilateral elements.
- **Scale Factor**: Allows you to define minimal increase in size of the convex enclosure with respect to the input scope. You can click on the right corner of the option and click **Publish** to publish **Scale Factor** to the **Property Worksheet**. The default value is **0.0**.
- **Distance**: Allows you to specify the absolute distance to change the enclosure distance from the model. The default value is **0.0 mm**.
- **Number of Layers**: Allows you to specify the minimal number of volumetric layers to be created between the model and the enclosure. The default value is **2**. You can click on the right corner of the option and click **Publish** to publish **Number of Layers** to the **Property Worksheet**.
- **Smoothing Iterations**: Allows you to provide the number of smoothing iterations required. The default value is **5**. You can click on the right corner of the option and click **Publish** to publish **Smoothing Iterations** to the **Property Worksheet**.
- **Smoothing Preserve Volume**: Allows you to preserve the volume after smoothing when **Smoothing Preserve Volume** is **Yes**. The default value is **No.**

HemiConvex Irregular Shape Enclosure

HemiConvex Irregular Shape Enclosure control automatically creates a hemiconvex enclosure of the input scope. The hemiconvex enclosure adapts to the input scope shape to minimize the internal enclosed volume using the provided parameters.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



HemiConvex Irregular Shape Enclosure Details view has the following options:

General

- **Control Type** (p. ?): Displays the selected control type.

Scope

- **Define By** (p. ?)
- **Scoping Method** (p. ?)

  Only **Part** can be selected for creating **HemiConvex Irregular Shape Enclosure**.

- **Scoping Pattern** (p. ?)

Definition

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Define By**: Allows you to define the element size based on **Value** or **Settings**. The available options are:
  - **Value**: Defines the element size based on the provided value.
  - **Settings**: Defines the element size based on the settings under **Mesh Settings** in the **Steps Details** view.
- **Element Size**: Provides the element size. When **Define By** is **Value**, you can specify the element size. When **Define By** is **Settings**, displays the element size calculated based on the provided **Mesh Settings** in the **Steps Details** view. The **Element Size** is read-only. You can click on the right corner of the option and click **Publish** to publish **Element Size** to the **Property Worksheet**.You can parameterize **Element Size** only when **Defined By** is **Value**.
- **Orientation Modality**: Allows you to define the orientation of the hemiconvex irregular shape enclosure. The default value is **Plus Z**. The available options are **Plus X**, **Plus Y**, **Plus Z**, **Minus X**, **Minus Y**, **Minus Z**.
- **Plane Distance**: Allows you to provide the distance from the base of the hemiconvex irregular shape enclosure to the model. The default value is **0.0 mm**. You can click on the right corner of the option and click **Publish** to publish **Plane Distance** to the **Property Worksheet**.
- **Open Enclosure**: Allows you to delete the base of the hemiconvex irregular shape enclosure while creating an open enclosure when **Open Enclosure** is **Yes**. The default value is **No**.
- **Delete Colliding Faces**: Allows you to delete any model face colliding with hemiconvex irregular enclosure base when **Delete Colliding Faces** is **Yes**. The default value is **Yes**.
- **Mesh Type**: Allows you to select the type of mesh to be used for meshing. The default value is **Quadrilaterals**. The available options are:
  - **Triangles:**: Creates mesh with triangular elements.
  - **Quadrilaterals**: Creates mesh with quadrilateral elements.
- **Scale Factor**: Allows you to define minimal increase in size of the hemiconvex enclosure with respect to the input scope. The default value is **1.0**. You can click on the right corner of the option and click **Publish** to publish **Scale Factor** to the **Property Worksheet**.
- **Distance**: Allows you to specify the minimal absolute distance to change the hemiconvex enclosure distance from the model. The default value is **0.0 mm**. You can click on the right corner of the option and click **Publish** to publish **Distance** to the **Property Worksheet**.
- **Number of Layers**: Allows you to specify the minimal number of volumetric layers to be created between the model and the enclosure. The default value is **2**. You can click on the right corner of the option and click **Publish** to publish **Number of Layers** to the **Property Worksheet**.
- **Smoothing Iterations**: Allows you to provide the number of smoothing iterations required. The default value is **5**. You can click on the right corner of the option and click **Publish** to publish **Smoothing Iterations** to the **Property Worksheet**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Smoothing Preserve Volume**: Allows you to preserve the volume after smoothing when **Smoothing Preserve Volume** is **Yes**. The default value is **No**.

Custom Names

Custom Names create enclosures or extrusions with the provided name. **Custom Names Details** view has the following options:

General

- **Control Type** (p. ?)

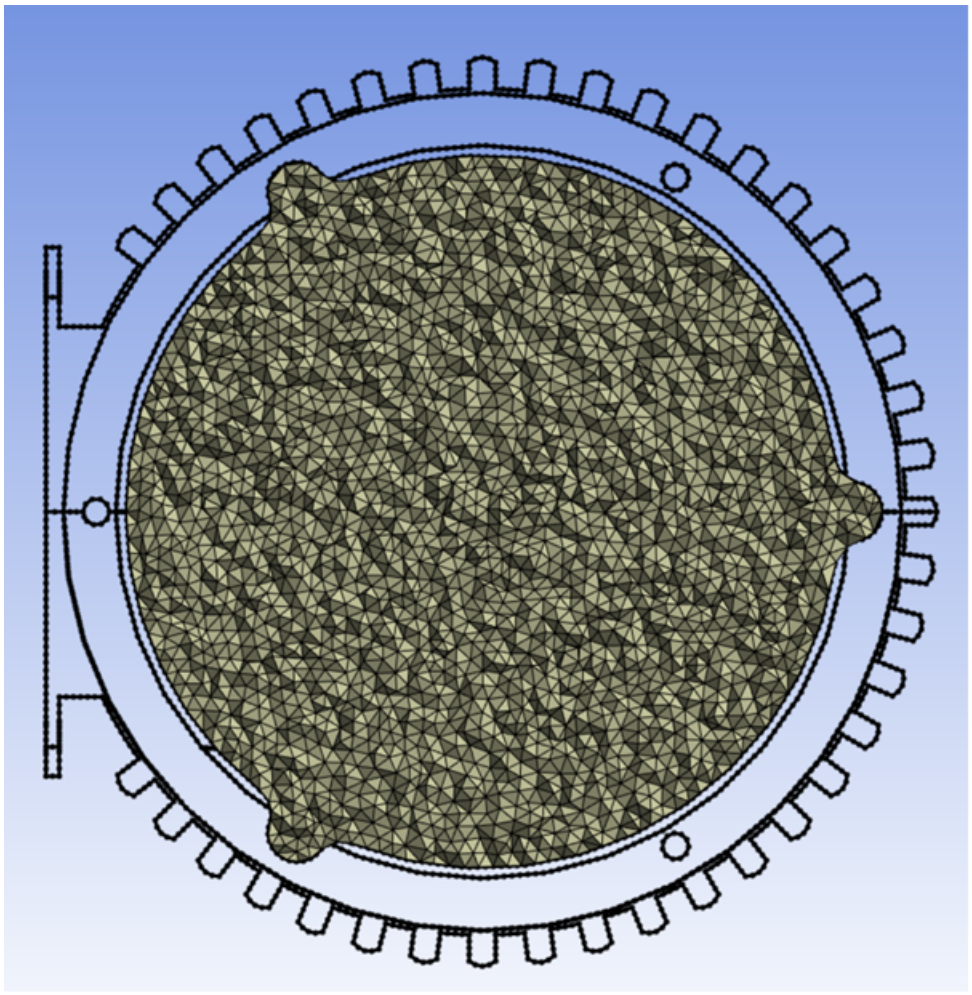
Definition

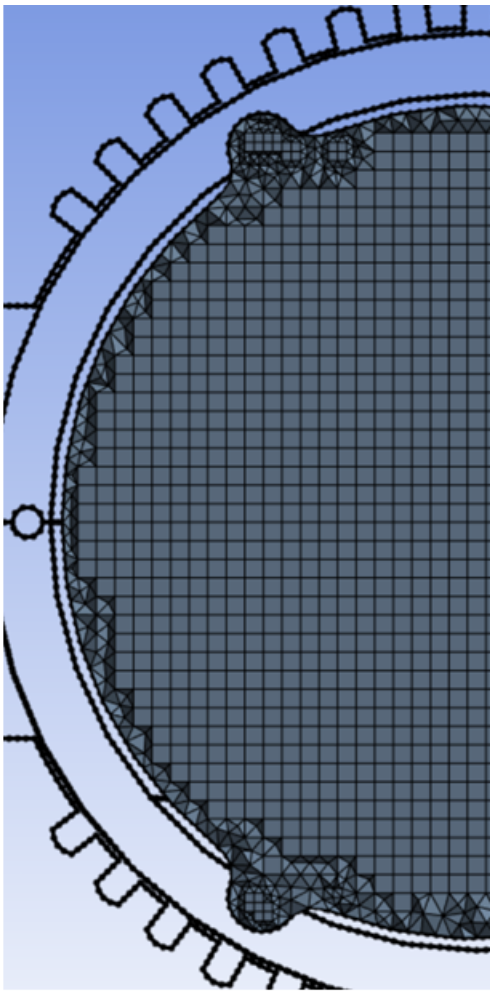
- **Internal Label Name**: Allows you to provide the name of the label attached to internal entities.
- **External Label Name**: Allows you to provide the name of the label attached to external entities.
- **Volume Zone Name**: Allows you to name the volume zone.
- **Face Zone Names**: Allows you to name the face zone.

Constant Size Volume Mesher

Constant Size Volume Mesher control creates a volume mesh of uniform size on the entire volume.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Constant Size Volume Mesher Details view has the following options:

General

- **Control Type** (p. ?)

Scope

- **Define By** (p. ?)
- **Scoping Method** (p. ?)
- **Scoping Pattern** (p. ?)

Definition

- **Define By**: Allows you to define the element size based on value or settings. The available options are:
  - **Value**: Defines the element size based on the provided value.
  - **Settings**: Defines the element size based on the settings under **Mesh Settings** in the **Steps Details** view.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Element Size**: Provides the element size for volume meshing. When **Define By** is **Value**, you can specify the element size for volume meshing. When **Define By** is **Settings**, displays the element size calculated based on the provided **Mesh Settings** in the **Steps Details** view. The **Element Size** is read-only. You can click on the right corner of the option and click **Publish** to publish **Element Size** to the **Property Worksheet**. You can parameterize **Element Size** only when **Defined By** is **Value**.
- **Mesh Type**: Allows you to provide the type of mesh. The default value is **Tetrahedrons**. The available options are **Tetrahedrons** and **HexCore**.
  - **Tetrahedrons**: Creates tetrahedral mesh.
  - **HexCore**: Creates hexcore mesh. When you select **HexCore**, **HexCore Relative Tet Layer** allows you to specify the number of tet layer elements created between the boundary and hex cells. The default value is **0.25**.
- **Remesh**: Allows you to remesh the entire volume when **Remesh** is **Yes**. The default is **No**.
- **Label Name**: Allows you to provide the name for the created volume. The name is added to **Labels** tab in the **Domain Browser**.

Volume Mesh Improvement

Volume Mesh Improvement control improves the quality of the volume mesh with specified quality metrics using the provided input options. **Volume Mesh Improvement Details** view has the following options:

General

- **Control Type** (p. ?)

Scope

- **Define By** (p. ?)
- **Scoping Method** (p. ?)
- **Scoping Pattern** (p. ?)

Definition

- **Target Skewness**: Allows you to provide the target skewness to achieve a desired quality mesh. The **Target Skewness** value ranges from **0** (low quality) to **1** (high quality). The default value is **0.9**. You can click on the right corner of the option and click **Publish** to add **Target Skewness** to the **Property Worksheet**. You can parameterize **Target Skewness**.
- **Target Dihedral Angle**: Allows you to provide the dihedral angle between volume mesh element faces. The default value is **120 degrees**. You can parametrize the **Target Dihedral Angle**. You can click on the right corner of the option and click **Publish** to add **Target Dihedral Angle** to the **Property Worksheet**. You can parameterize **Target Dihedral Angle**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Number of Attempts**: Allow you to provide the maximum number of attempts for volume improvement. The default value is **5**. You can click on the right corner of the option and click **Publish** to add **Number of Attempts** to the **Property Worksheet**. You can parameterize **Number of Attempts**.

Second Order Conversion

Second Order Conversion control converts linear surface mesh elements or volumetric mesh elements to quadratic elements. **Second Order Conversion** has the following options:

General

- **Control Type** (p. ?)

Scope

- **Define By** (p. ?)
- **Scoping Method** (p. ?)
- **Scoping Pattern** (p. ?)

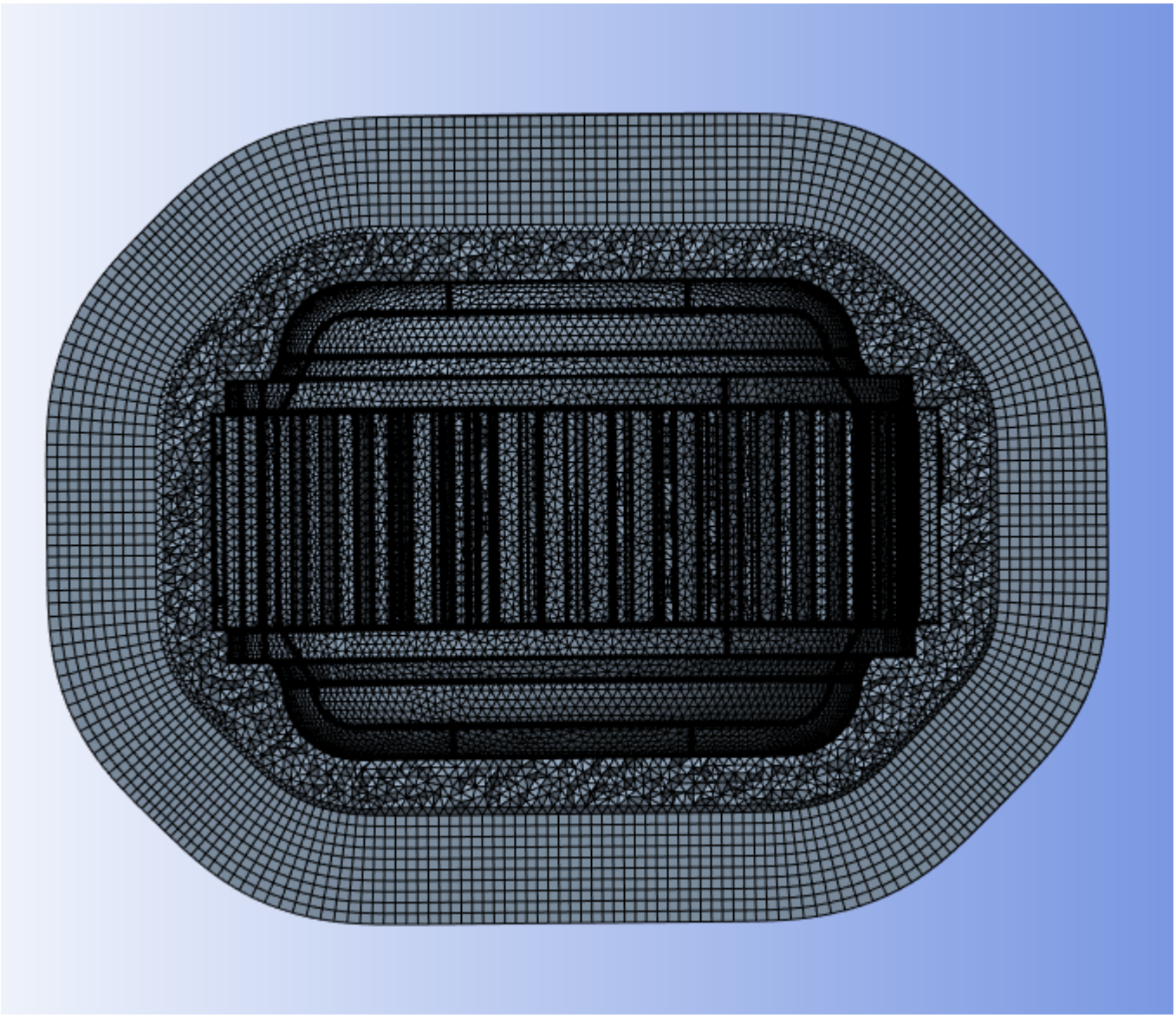
Definition

- **Define By**: Allows you to create quadratic elements based on value or settings. The available options are:
  - **Value**: Allows you to create quadratic elements based on the selected **Create Quadratic Elements** option.
  - **Settings**: Allows you to create quadratic elements based on the defined settings under **Mesh Settings** in the **Steps Details** view.
- **Create Quadratic Elements**: Allows you to create elements with mid-side nodes when **Create Quadratic Elements** is **Yes**. The default value is **No**.

Extrusion

Extrusion control creates extruded layers from the input surface for a specified number of layers with specified height.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Extrusion Details view has the following options:

General

- **Control Type** (p. ?)

Scope

- **Define By** (p. ?)
- **Scoping Method** (p. ?)
- **Scoping Pattern** (p. ?)

Definition

- **Define By**: Allows you to perform extrusion based on value or settings. The available options are:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Value**: Performs extrusion based on **Per Layer Height** and **Number of Layers**.
  - **Settings**: Performs extrusion based on the settings under **Mesh Settings** in the **Steps Details** view.

- **Per Layer Height**: Allows you to specify the height of each layer of solid elements. The default value is **10 mm**. You can click on the right corner of the option and click **Publish** to publish **Per Layer Height** to the **Property Worksheet**. You can parameterize **Per Layer Height** only when **Defined By** is **Value**.
- **Number of Layers**: Allows you to specify the number of layers to be used for extruding the model. The default value is **2**. You can click on the right corner of the option and click **Publish** to publish **Number of Layers** to the **Property Worksheet**. You can parameterize **Number of Layers** only when **Defined By** is **Value**.

Topology Creation

Topology Creation control allows you to define the scope to create the topology from the mesh. **Topology Creation Details** view has the following options:

General

- **Control Type** (p. ?)

Scope

- **Define By** (p. ?)
- **Scoping Method** (p. ?)
- **Scoping Pattern** (p. ?)

Definition

Delete Empty Volumes: Allows you to delete the volumes that do not have boundary nodes enclosing the same in the created topology when **Delete Empty Volumes** is **Yes**. The default value is **Yes**.

Volumes Merging

Volumes Merging control merges all the volumes that share the input faces defined on the scope into a single volume. **Volumes Merging Details** view has the following options:

General

- **Control Type** (p. ?)

Scope

- **Define By** (p. ?)
- **Scoping Method** (p. ?)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Scoping Pattern** (p. ?)

Zone Material Assignment

The **Zone Material Assignment** control allows you to define the material assigned to the zones. On completing the **Output** (p. 617), the material assigned to a zone are transferred to the created geometry parts and bodies accordingly. This allows you to have a fully defined geometry upon completion of the workflow which is mandatory for automatic and persistent updates.

The **Zone Material Assignment** control **Details** view has the following options:

General

- **Control Type** (p. ?)

Scope

- **Define By** (p. ?)
- **Scoping Method** (p. ?)
- **Scoping Pattern** (p. ?)

Definition

- **Assignment**: Allows you to select the desired material from the **Engineering Data Materials Selection** window. You can click to open the **Engineering Data Materials** window to select the required material.

Zone Thickness Assignment

The **Zone Thickness Assignment** control allows you to define the thickness assigned to zones. On completing the **Output** (p. 617), assigns the thickness of the scoped zones to the newly created geometry parts and bodies accordingly. This allows you to have a fully defined geometry on completing the workflow, which is mandatory for automatic and persistent updates.

The **Zone Thickness Assignment** control **Details** view has the following options:

General

- **Control Type** (p. ?)

Scope

- **Define By** (p. ?)
- **Scoping Method** (p. ?)
- **Scoping Pattern** (p. ?)

Definition

- **Thickness**: Allows you to specify the thickness of the input scope.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Checkpoint

Checkpoint provides a revert option that allows you to revert the operation. You can add **Checkpoint** for every operation if you want to save the state before executing the operation. After execution, the saved state can be restored using **Revert To Step**and the operation can be re-executed.

General

- **Control Type** (p. ?)

Definition

- **Create Checkpoint**: Creates the checkpoint and allows you to revert to the previous step when **Create Checkpoint** is **Yes**. The default value is **Yes**.

Note:

You can add only one **Checkpoint** control for each step.

Hole Patching

Hole Patching allows you to patch holes in the model to close the volume defined by the provided **Include** and **Exclude** material points in the **Material Point** control.

Note:

You should have at least one **Include** material point to perform hole patching.

Hole Patching Details view has the following options:

General

- **Control Type** (p. ?)

Scope

- **Scoping Method** (p. ?)
- **Scoping Pattern** (p. ?)
- **Exclude Scoping Method**: Allows you to specify the entities to be excluded while performing hole patching.
- **Exclude Scoping Pattern**: Allows you to specify the name pattern to be excluded from the selected **Scoping Method** while performing hole patching.

Definition

- **Element Size**: Allows you to specify the minimum feature element size of the input scope to be considered for patching holes. You can click on the right corner of the option and click

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Publish to publish **Element Size** to the **Property Worksheet**. You can parameterize **Element Size**.

- **Part Name**: Allows you to provide the name for the created part after patching holes.
- **Face Zone Name**: Allows you to provide the name for the created face zones after patching holes.
- **Label Name**: Allows you to name the label attached to the patched hole faces.
- **Max Hole Size**: Allows you to specify the maximum hole size to be considered for patching.

Mesh Reading

Mesh Reading allows you to read a mesh into the mesh workflow model. The read mesh can be either appended to the already existing mesh or overwrite it. **Mesh Reading** supports .pmdat, .msh, .cdb file extension for reading the mesh.**Mesh Reading Details** view has the following options:

General

- **Control Type** (p. ?)

Definition

- **File Name**: Allows you to provide path of the file to be read.
- **Append Data**: Allows you to append the mesh data from the file to the mesh workflow model
- **Repair Model**: Allows you to repair the mesh workflow model after reading the mesh data, creating missing zones and volumes if required.

Mesh Writing

Mesh Writing allows you to write and save the mesh generated in the mesh workflow for the selected step. **Mesh Writing** supports .pmdat, .msh, .cdb file extension to write your mesh.

Mesh Writing Details view has the following options:

General

- **Control Type** (p. ?)

Definition

- **File Name**: Allows you to provide name and path to write the file.

Custom Names(Sizing)

Custom Names control creates size field with the provided name. **Custom Names Details** view has the following options:

General

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

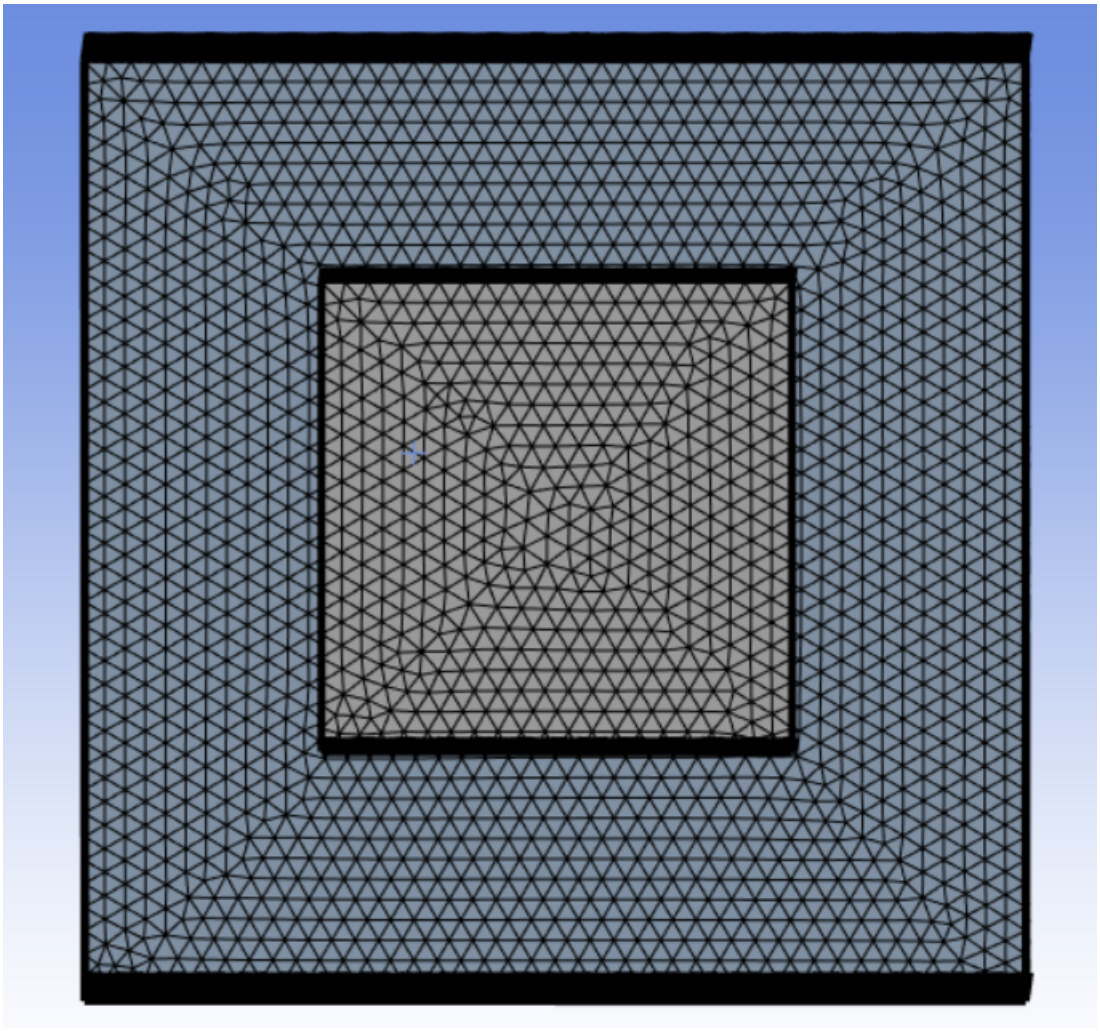
- **Control Type** (p. ?): Displays the selected control type.

Definition

- **Name**: Allows you to provide the name for the created size field.

Constant Sizing

Constant Sizing control allows you to set the maximum size on the selected scope based on the other applied size controls on the scope. If no size controls are applied on the selected scope, a uniform size is provided throughout the mesh. That is, constant sizing is ignored on the selected scope, if the any other applied size controls on the scope specify a smaller size.



Constant Sizing Details view has the following options:

General

- **Control Type** (p. ?): Displays the selected control type.

Scope

- **Scoping Method** (p. ?): Scopes **Part**, **Zone** or **Label** in the model.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Scoping Pattern** (p. ?): Scopes the **Part**, **Zone** or **Label** with the provided pattern.

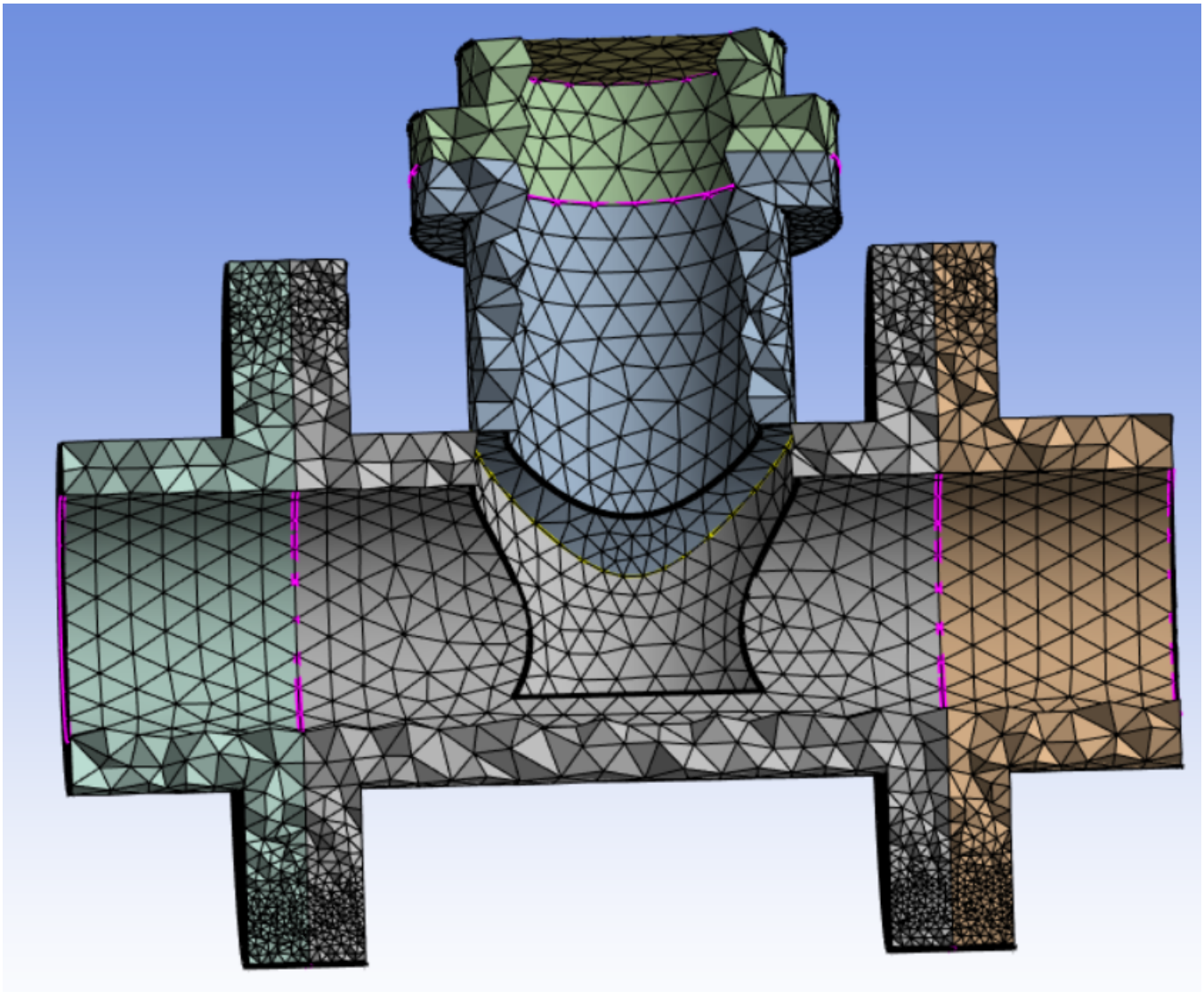
Definition

- **Define By**: Allows you to scope the operation based on your selection. The available options are:
  - **Value**: Allows you to define the maximum size based on the provided element size.
  - **Setting**: Allows you to define the maximum size based on the defined acoustic settings.
- **Element Size**: Provides the maximum element size for surface meshing. You can click on the right corner of the option and click **Publish** to publish **Element Size** to the **Property Worksheet**.
- **Growth Rate**: Allows you to specify the increase in element edge length with each succeeding layer of elements. The default value is **1.2**.

Curvature Sizing

Curvature Sizing control allows you to refine the surface mesh to capture the underlying curve and surface curvature.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Curvature Sizing Details view has the following options:

General

- **Control Type** (p. ?): Displays the selected control type.

Scope

- **Scoping Method** (p. ?): Allows you to select the entities for the selected control. The available options are:
  - **Part**: Allows you to select Parts for defining the scope of the control.
  - **Label**: Allows you to select Labels for defining the scope of the control.
  - **Zone**: Allows you to select Zones for defining the scope of the control.
- **Scoping Pattern** (p. ?): Allows you to specify the name pattern to get the selected **Scoping Method**. **Scoping Pattern** supports **Regular Expression**.

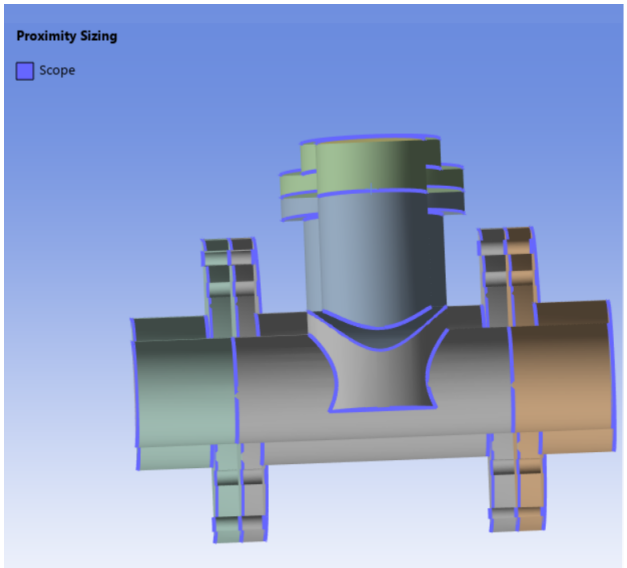
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

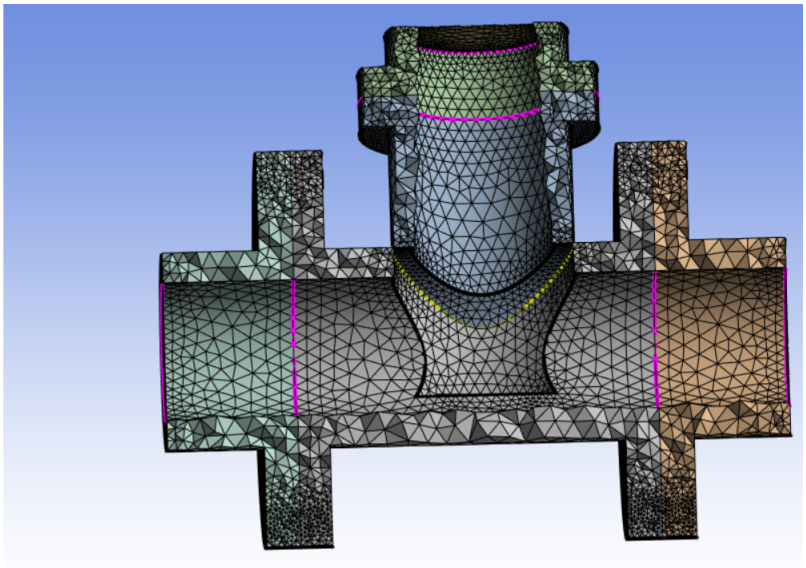
Definition

- **Define By**: Allows you to scope the operation based on your selection. The available options are:
  - **Value**: Allows you to define the maximum size based on the provided element size.
  - **Setting**: Allows you to define the maximum size based on the defined acoustic settings.
- **Project On Geometry**: Allows you to project the curvature created for size field on the underlying geometry of the model when **Project On Geometry** is **Yes**. The default value is **Yes**.
- **Growth Rate**: Allows you to specify the increase in element edge length with each succeeding layer of elements. The default value is **1.2**.
- **Min Size**: Allows you to specify a minimum size to be used for curvature sizing calculation.
- **Max Size**: Allows you to specify a maximum size to be used for curvature sizing calculation.
- **Normal Angle**: Allows you to specify the maximum allowable angle at which one element edge is allowed to span for the specified curvature. The default value is **18 degrees**.

Proximity Sizing

Proximity Sizing control allows you to compute edge and face sizes in gaps using the specified minimum number of element layers.





Proximity Sizing Details view has the following options:

General

- **Control Type** (p. ?): Displays the selected control type.

Scope

- **Scoping Method** (p. ?): Allows you to select the entities for the selected control. The available options are:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Part**: Allows you to select Parts for defining the scope of the control.
  - **Label**: Allows you to select Labels for defining the scope of the control.
  - **Zone**: Allows you to select Zones for defining the scope of the control.

- **Scoping Pattern** (p. ?): Allows you to specify the name pattern to get the selected **Scoping Method**. **Scoping Pattern** supports **Regular Expression**.

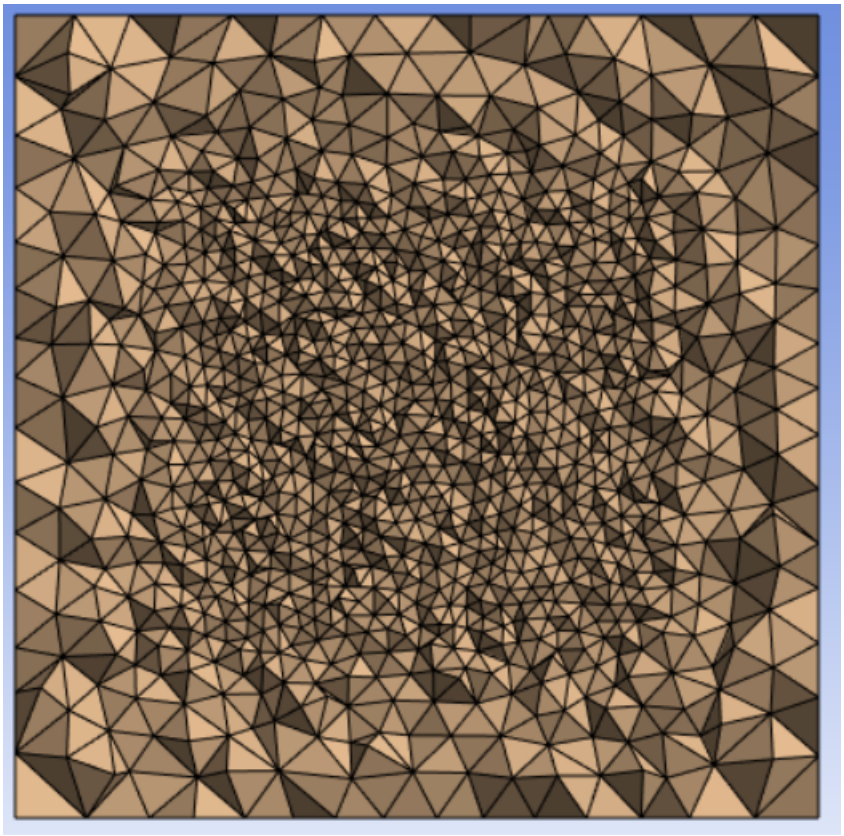
Definition

- **Define By**: Allows you to scope the operation based on your selection. The available options are:
  - **Value**: Allows you to define the maximum size based on the provided element size.
  - **Setting**: Allows you to define the maximum size based on the defined acoustic settings.
- **Growth Rate**: Allows you to specify the increase in element edge length with each succeeding layer of elements. The default value is **1.2**.
- **Min Size**: Allows you to specify a minimum size to be used for proximity sizing calculation.
- **Max Size**: Allows you to specify a maximum size to be used for proximity sizing calculation.
- **Element Per Gap**: Allows you to specify the minimum number of layers of elements to be generated in the gap. The default value is **3**.
- **Ignore Self Proximity**: Ignores the proximity between two faces in the same face zone when **Ignore Self Proximity** is **Yes**. The default value is **Yes**.
- **Ignore Orientation**: Ignores the face normal orientation while calculating the proximity when **Ignore Orientation** is **Yes**. The default value is **No**.

Body of Influence Sizing

Body of Influence Sizing control allows you to apply mesh sizing based on the specified element size and controls the mesh density based on neighboring bodies of influence.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Body of Influence Sizing Details view has the following options:

General

- **Control Type** (p. ?): Displays the selected control type.

Scope

- **Scoping Method** (p. ?): Allows you to select the entities for the selected control. The available options are:
  - **Part**: Allows you to select Parts for defining the scope of the control.
  - **Label**: Allows you to select Labels for defining the scope of the control.
  - **Zone**: Allows you to select Zones for defining the scope of the control.
- **Scoping Pattern** (p. ?): Allows you to specify the name pattern to get the selected **Scoping Method**. **Scoping Pattern** supports **Regular Expression**.

Definition

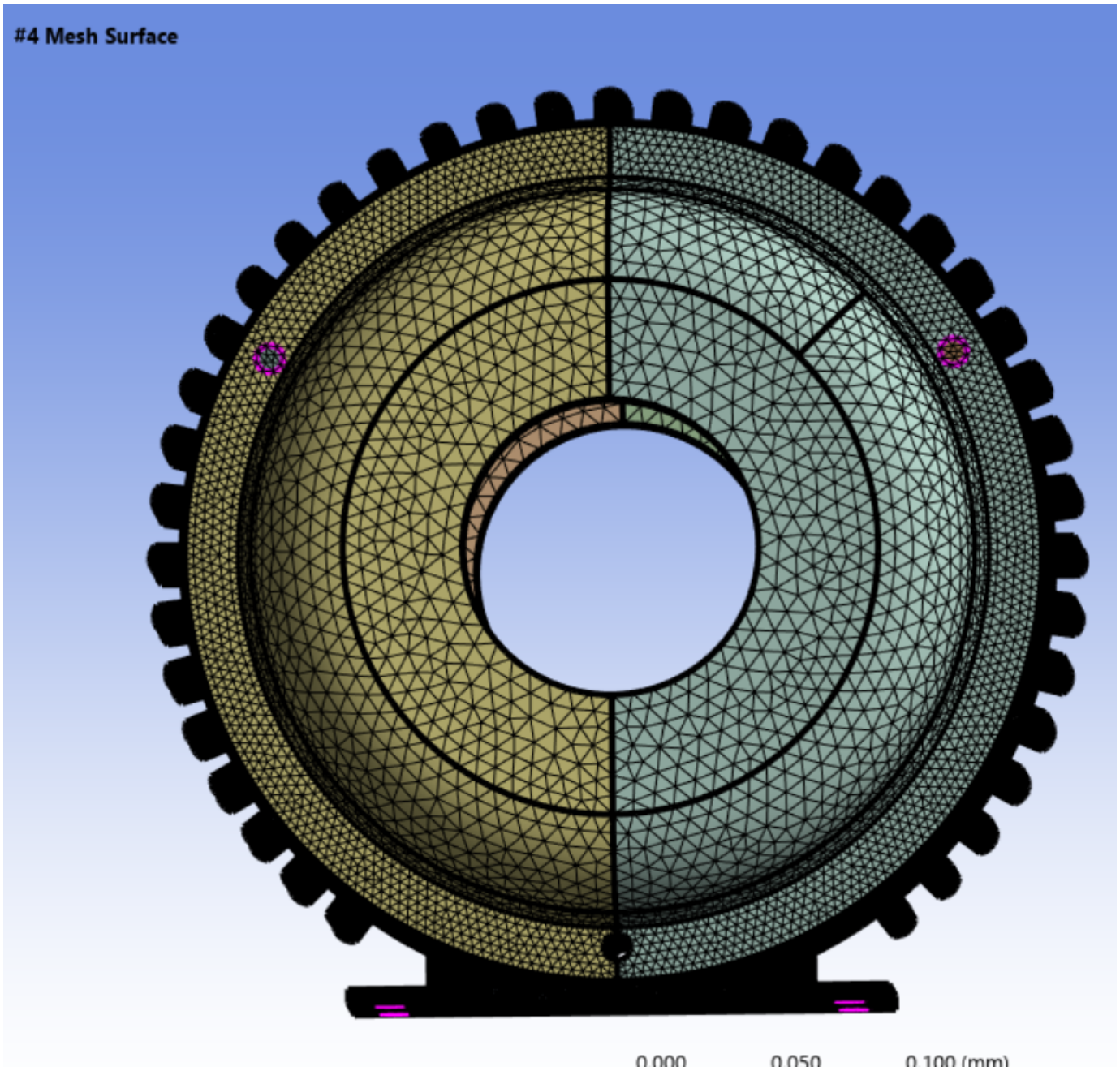
- **Define By**: Allows you to scope the operation based on your selection. The available options are:
  - **Value**: Allows you to define the maximum size based on the provided element size.
  - **Settings**: Allows you to define the maximum size based on the defined acoustic settings.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Element Size**: Allows you to provide the element size for the selected scope. You can click on the right corner of the option and click **Publish** to publish **Element Size** to the **Property Worksheet**.
- **Growth Rate**: Allows you to specify the increase in element edge length with each succeeding layer of elements. The default value is **1.2**.

Size Field Surface Mesher

Size Field Surface Mesher control allows you to mesh the surface of the model with the sizing defined in the size field provided as input.



Size Field Surface Mesher Details view has the following options:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General

- **Control Type** (p. ?): Displays the selected control type.

Scope

- **Define By** (p. ?): Allows you to define the input scope of the Size Field Surface Mesher control.
- **Scoping Method** (p. ?): Allows you to scope Part, Label or Zone as input for the **Size Field Surface Mesher** control.
- **Scoping Pattern** (p. ?): Allows you to specify the name pattern to get the selected **Scoping Method**. **Scoping Pattern** supports **Regular Expression**.

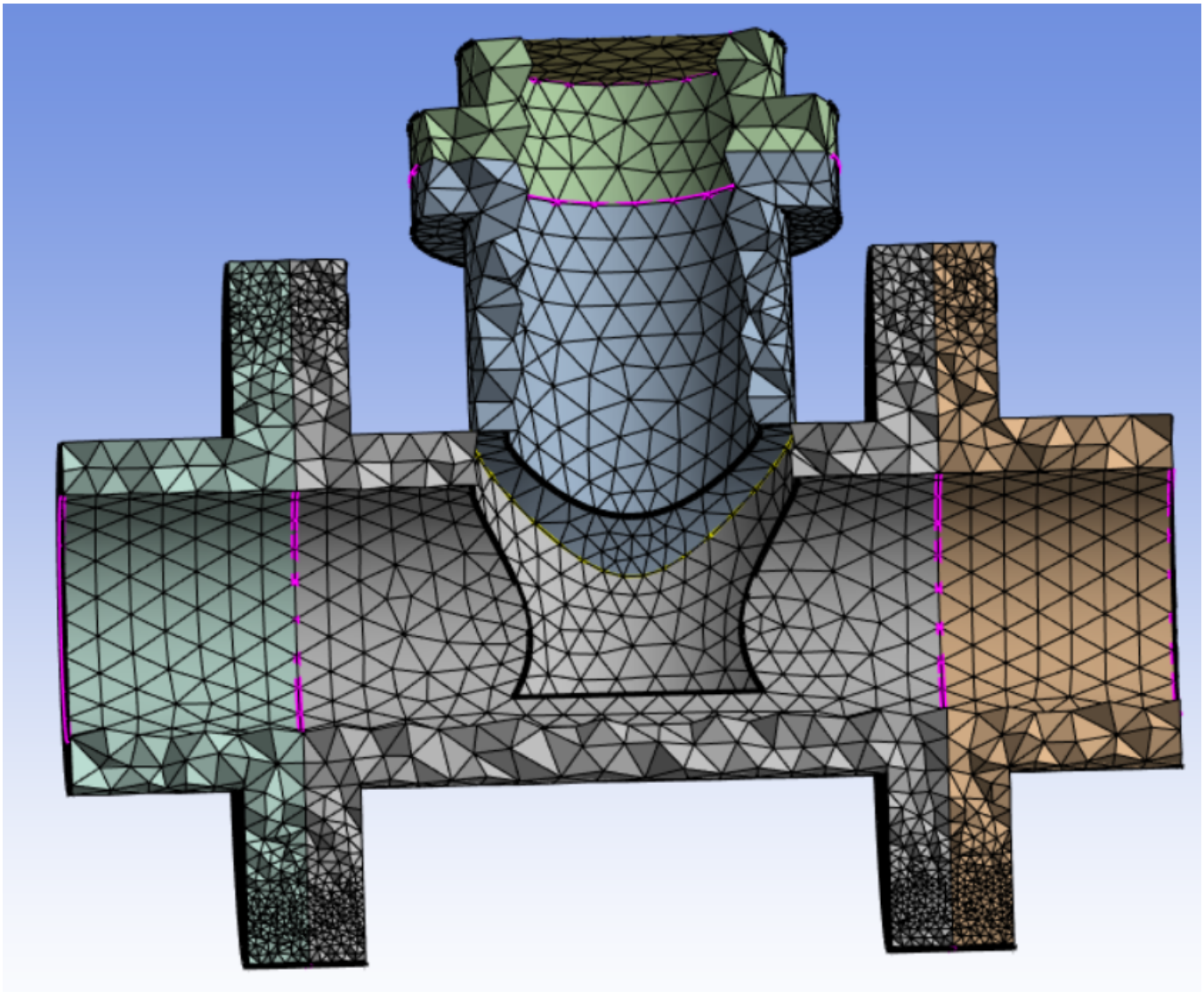
Definition

- **Define Size Field By**: Allows you to define the size field name pattern. The available options are:
  - **Value**: Allows you to provide the size field name pattern manually.
  - **Outcome**: Allows you to select an existing size field outcome from a previous step.
- **Size Field Name Pattern**: Allows you to specify the name pattern of size fields to be activated for the surface meshing.
- **Mesh Type**: Allows you to select the type of mesh you want to generate. The default value is **Quadrilateral**. The available options are:
  - **Triangle**: Creates mesh with triangular elements.
  - **Quadrilateral**: Creates mesh with quadrilateral elements.
- **Project On Geometry**: Allows you to project the created mesh on the underlying geometry when **Project On Geometry** is **Yes**. The default value is **Yes**.
- **Retain Existing Mesh**: Allows you to retain the mesh on the already meshed topofaces while remeshing when **Retain Existing Mesh** is **Yes**. The default value is **No**.

Size Field Volume Mesher

Size Field Volume Mesher allows you to mesh the model with the sizing defined in the size field provided as input.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Size Field Volume Mesher Details view has the following options:

General

- **Control Type** (p. ?): Displays the selected control type.

Scope

- **Define By** (p. ?): Allows you to define the input scope of the **Size Field Volume Mesher** control.
- **Scoping Method** (p. ?): Allows you to scope Part, Label or Zone as input for the **Size Field Volume Mesher** control.
- **Scoping Pattern** (p. ?): Allows you to specify the name pattern to get the selected **Scoping Method**. **Scoping Pattern** supports **Regular Expression**.

Definition

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

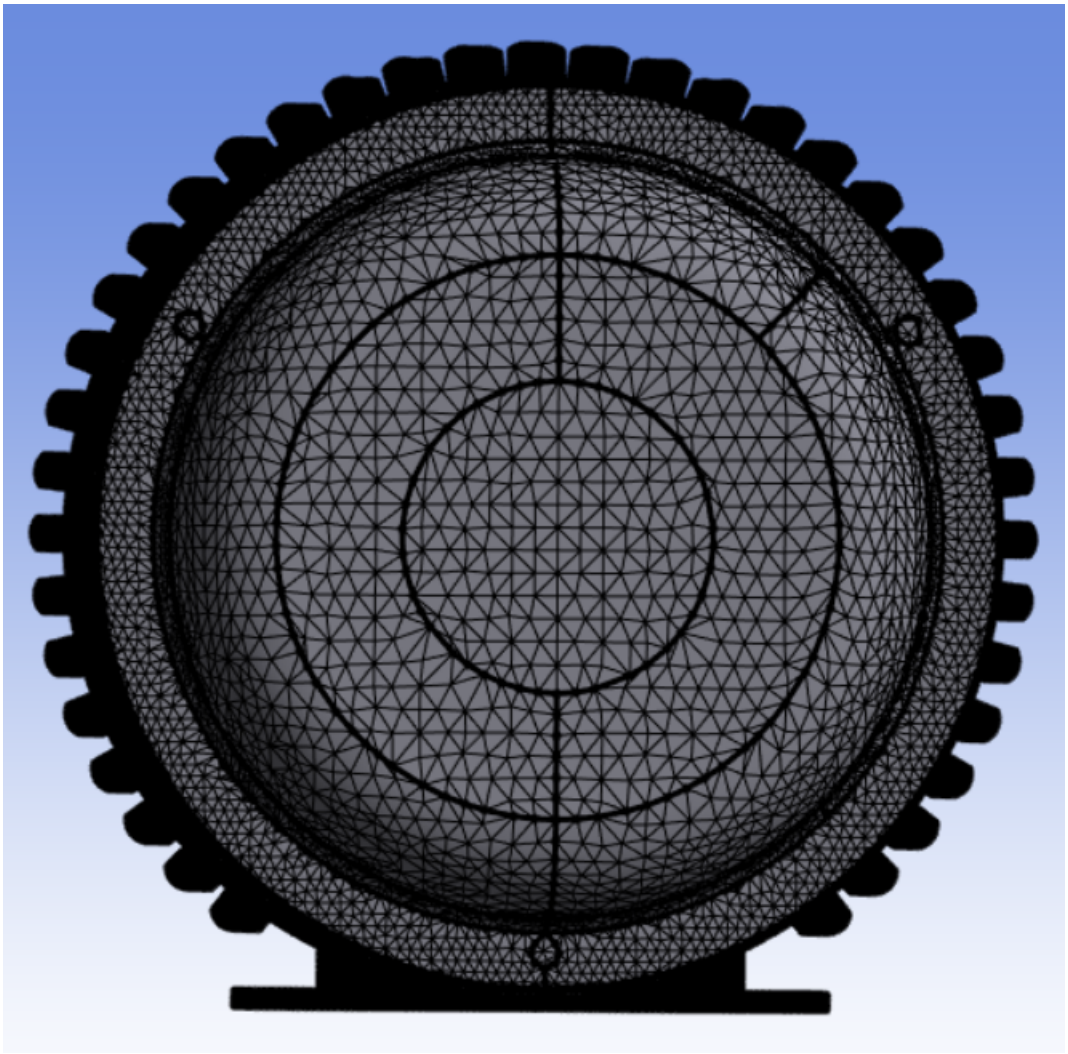
- **Define Size Field By**: Allows you to define the size field name pattern. The available options are:
  - **Value**: Allows you to provide the size field name pattern manually.
  - **Outcome**: Allows you to select an existing size field outcome from a previous step.
- **Size Field Name Pattern**: Allows you to provide name for the created size field.
- **Label Name**: Allows you to name the created volume. The created name is available as label under **Labels** tab in the **Domain Browser**.
- **Mesh Type**: Allows you to provide the type of mesh. The default value is **Tetrahedrons**. The available options are:
  - **Tetrahedrons**: Creates tetrahedral mesh.
  - **HexCore**: Creates hexcore mesh. When you select **HexCore**, **HexCore Relative Tet Layer** allows you to specify the number of tet layer elements created between the boundary and hex cells. The default value is **0.25**.
- **Growth Rate**: Allows you to specify the growth rate of the size field. The default value is **1.5**.

  You can click on the right corner of the option and click **Publish** to publish **Growth Rate** to the **Property Worksheet**. You can parameterize **Growth Rate**.
- **Remesh**: Allows you to remesh the entire volume when **Remesh** is **Yes**. The default is **No**.

Size Field Wrapper

Size Field Wrapper control allows you to wrap the model with the sizing defined in the size field provided as input.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Size Field Wrapper Details view has the following options:

General

- **Control Type** (p. ?): Displays the selected control type.

Scope

- **Scoping Method** (p. ?): Allows you to scope **Part** as input for the **Size Field Wrapper** control
- **Scoping Pattern** (p. ?): Allows you to specify the name pattern to get the selected **Scoping Method**. **Scoping Pattern** supports **Regular Expression**.
- **Target Scoping Method**: Allows you to merge the newly created part of the wrapper to the existing part automatically. Only **Part** can be selected for target scoping in **Size Field Wrapper**.
- **Target Scoping Pattern**: Allows you to specify the name pattern to get the selected target **Scoping Method**. For more information, refer **Scoping Pattern** (p. ?).

Definition

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Size Field Name Pattern**: Allows you to specify the name pattern of size fields to be activated for the wrapper operation.
- **Live Region Type**: Allows you to select the region type to extract the wrap region. The available options are:
  - **Material Point**: Allows you to define the volume of the model being wrapped. The material point is set locally along with X, Y and Z coordinates. For **Internal FEM Acoustics**, the default value is **Material Point**.
  - **External**: Allows you to wrap the surface from the external region. For **External FEM Acoustics** and **BEM Acoustics** workflows, the default value is **External**.
- **Delete Input Scope**: Allows you to delete the input source of the model after wrapping when **Delete Input Scope** is **Yes**. The default value is **Yes**.
- **Exclude Enclosure**: Allows you to exclude the external region and wrap inner parts only for the box in box models when **Exclude Enclosure** is **Yes**. The default value is **No**.
- **Face Zone By Part**: Creates a face zone for each part of the input scope when **Face Zone By Part** is **Yes**. The created face zones are converted to named selections in Mechanical after completing the mesh workflow. When **Face Zone By Part** is **No**, creates a single face zone for the whole input scope. Hence, only a single named selection is available for Mechanical after completing the mesh workflow. The default value is **No**.
- **Reverse Surface Orientation**: Allows you to reverse the orientation of the created face zonelets. The default value is **No**. When **Reverse Surface Orientation** is **No**, the orientation is same as the position of the **Live** material point for the wrapper. When **Reverse Surface Orientation** is **Yes**, the orientation is opposite to the position of the **Live** material point for wrapper. **BEM Acoustics** uses **Reverse Surface Orientation** where you need to wrap the external and solve the internal field.

Mesh Workflow Outcomes

Outcomes provide the results of the successfully executed operation. **Outcomes** act as read-only outputs automatically filled after the execution of the operation, based on the applied controls, which you can use to monitor the results of the operation and can be set as input to the following **Mesh Workflow Steps**. The **Outcomes** created can be scoped as input to controls in the next **Mesh Workflow Steps**. When you want an **Outcome** to be scoped as input to controls in the following operation, you can right-click the **Step**, click **Insert** and select the **Outcomes** specific to the operation. The **Outcomes** added for an operation can be scoped in the consecutive operations setting.

Outcome Details view has the following options:

General

- **Outcome Type**: Allows you to select the outcome type. The available options are:
  - **Scope** (p. 673)
  - **Failure Info** (p. 673)
  - **Face Zone Scope** (p. 674)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Volume Zone Scope** (p. 674)

– **Internal Enclosure Scope** (p. 674)

– **External Enclosure Scope** (p. 675)

– **Extrusion Start Scope** (p. 675)

– **Extrusion End Scope** (p. 675)

– **Size Field Name** (p. 676)

Scope

Scope outcome allows you to scope the created entities of the operation. The created outcome can be used to define the scope of the consecutive operations. Right-click the **Step** and click **Insert > Scope** to add scope as an outcome. **Scope Details** view has the following options:

General

- **Outcome Type:** Displays the selected outcome type.

Scope

- **Scoping Method**: Displays the scoping method for the outcome type.
- **Scoping Pattern**: Displays the scoping pattern for the outcome type.

Definition

- **Label Edges**: Provides label to identify the edges created during the operation.

Failure Info

Failure Info provides information about the failures that occurred in the respective mesh workflow.

Note:

Failure Info Outcome is automatically added to its operation upon failure. You cannot add **Failure Info** as **Outcome** manually.

Failure Info Details view has the following options:

General

- **Outcome Type**: Displays the selected outcome type.

Definition

- **Error Text**: Provides you the error messages for the failed mesh workflow operation.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Face Zone Scope

Face Zone Scope allows you to create an outcome, scoping created face zones in the selected control. Right-click the **Step** and click **Insert > Face Zone Scope** to add face zone as an outcome make it available for scoping in the subsequent operations.

Face Zone Scope Details view has the following options:

General

- **Outcome Type** (p. ?)

Scope

- **Scoping Method** (p. ?)
- **Scoping Pattern** (p. ?)

Volume Zone Scope

Volume Zone Scope allows you scope the created volume zones in the selected operation. Right-click the **Step** and click **Insert > Volume Zone Scope** to add volume zones as an outcome for the subsequent operations. **Volume Zone Scope Details** view has the following options:

General

- **Outcome Type** (p. ?)

Scope

- **Scoping Method** (p. ?)
- **Scoping Pattern** (p. ?)

Internal Enclosure Scope

Internal Enclosure Scope allows you to scope the internal enclosure created in the selected operation. Right-click the **Step** and click **Insert > Internal Enclosure Scope** to add internal enclosure as an outcome and make it available for scoping in the subsequent operations. **Internal Enclosure Scope Details** view has the following options:

General

- **Outcome Type** (p. ?)

Scope

- **Scoping Method** (p. ?)
- **Scoping Pattern** (p. ?)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

External Enclosure Scope

External Enclosure Scope allows you to scope the external enclosure created in the selected operation. Right-click the **Step**, click **Insert > External Enclosure Scope** to add external enclosure as an outcome and make it available for scoping in the subsequent operations. **External Enclosure Scope Details** view has the following options:

General

- **Outcome Type** (p. ?)

Scope

- **Scoping Method** (p. ?)
- **Scoping Pattern** (p. ?)

Extrusion Start Scope

Extrusion Start Scope allows you to scope the base face used for extruding. Right-click the **Step** and click **Insert > Extrusion Start Scope** to add extrusion start scope as an outcome and make it available for scoping in the subsequent operations. **Extrusion Start Scope Details** view has the following options:

General

- **Outcome Type** (p. ?)

Scope

- **Scoping Method** (p. ?)
- **Scoping Pattern** (p. ?)

Extrusion End Scope

Extrusion End Scope allows you to scope the end face created after extrusion. Right-click the **Step** and click **Insert > Extrusion End Scope** to add extrusion end as an outcome and make it available for scoping in the subsequent operations. **Extrusion End Scope Details** view has the following options:

General

- **Outcome Type** (p. ?)

Scope

- **Scoping Method** (p. ?)
- **Scoping Pattern** (p. ?)

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Internal Volume Zone Scope

Internal Volume Zone Scope allows you to scope the internal volume zones created in the selected control. Right-click the **Step** and click **Insert > Internal Volume Zone Scope** to add internal volume zone as an outcome and make it available for scoping in subsequent operations.

Internal Volume Zone Scope Details view has the following options:

General

- **Outcome Type** (p. ?)

Scope

- **Scoping Method** (p. ?)
- **Scoping Pattern** (p. ?)

Size Field Name

Size Field Name outcome allows you scope the name of the created size field. Right-click the **Step**, click **Insert > Size Field Name** to add **Size Field Name** as an outcome and make it available for scoping in the subsequent operations.

Size Field Name Details view has the following options:

General

- **Outcome Type** (p. ?): Displays the selected outcome type.

Definition

- **Size Field Name Pattern**: Provides the name of the created size field and adds the name under **Size Fields** tab in the **Domain Browser**.

Property Worksheet

Propety Worksheet allows you to view the published properties of each control involved in the mesh workflows, provides an easy visualization of the driving input options for mesh workflows and to edit individual controls.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Association	Property	Value
1	Steps	Analysis Frequency	1.5 Hz
2	#1 Fill Holes > Hole Filling	Scoping Pattern	hole.*
3	#2 Wrap Parts > Constant Size Wrapper	Element Size	19.056 m
4	#2 Wrap Parts > Material Point	X Coordinate	N/A
5	#2 Wrap Parts > Material Point	Y Coordinate	N/A
6	#2 Wrap Parts > Material Point	Z Coordinate	N/A
7	#3 Improve Wrap Mesh > Wrapper Specific Surface Mesher	Element Size	19.056 m
8	#4 Create Enclosure > Convex Irregular Shape Enclosure	Element Size	19.056 m
9	#4 Create Enclosure > Convex Irregular Shape Enclosure	Number of Layers	2
10	#5 Mesh Volume > Constant Size Volume Mesher	Element Size	19.056 m
11	#6 Improve Volume Mesh > Second Order Conversion	Create Quadratic Elements	No
12	#7 Extrude Acoustic Region > Extrusion	Per Layer Height	19.056 m
13	#7 Extrude Acoustic Region > Extrusion	Number of Layers	2
14	#8 Extrude PML Region > Extrusion	Per Layer Height	19.056 m
15	#8 Extrude PML Region > Extrusion	Number of Layers	4

Go to page: 1 Show rows: 20 1-15 of 15

Property Worksheet has the following options:

- **Publish Properties**: Allows you to add the unpublished properties for the selected controls or selected steps on the tree outline. You can select the check box of a respective step or control and click **Publish** to add it to the **Property Worksheet**.
- **Association**: Displays the path of the step, control or outcome that uses the published property.
- **Property**: Displays the options published in the **Details** view as **Property** in the **Property Worksheet**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Value**: Displays the values of the property. You can edit the value for the property from the worksheet for each control without accessing the individual control type.

  Note:

  You can hover over the **Property Worksheet** options and click the dropdown to sort or filter the option in the worksheet as per your requirement. **Value** does not have the sorting and filtering options. To reload or refresh the **Property Worksheet**, right-click anywhere in the worksheet and click **Reload**.

- **Go to page**: Allows you to navigate to the specified page.
- **Show rows**: Allows you to specify the number of rows to be displayed on a page.
- : Allows you to navigate to the previous page in the **Property Worksheet**.
- : Allows you to navigate to the next page in the **Property Worksheet**.

Right-click options available in the **Property Worksheet** are:

- **Unpublish**: Removes the selected row from the **Property Worksheet**.
- **Go To Object**: Allows you to view the selected control on the tree outline.

Mesh Workflow States

Mesh Workflow States enable you to understand the states of workflows, steps and other related **Mesh Workflows** objects. **Mesh Workflow States** help you to handle the problems occurring while working with **Mesh Workflows**.

Mesh Workflows object, its child-objects and controls have states that are defined based on the data propagated in the mesh workflow. The States propagate up in the object hierarchy. The parent object gets the most relevant child object state. This helps you to navigate to the issues quickly and resolve it.

Mesh Workflows objects and controls have the following states:

- A check mark ✓ denotes that a control is fully defined, or an operation has been executed successfully.
- A check mark with a hash ✓ denotes a step is completed and is reversible after execution.
- A question mark ? denotes that the required input is not available. You may need to:
  - define the scope
  - set parameters

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  – wait for an Outcome from the previous operation

- A lightning bolt denotes that an operation is ready to be executed or that an Outcome is not ready yet.
- A red exclamation mark denotes that the operation is blocked due to an error during execution.

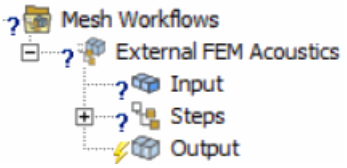
Mesh Workflow objects and **Controls** states with examples are as follows:

Symbol	Status	Description
Mesh Workflows	Requires input	Indicates that the Mesh Workflows requires input and is waiting for outcomes from the operations.
Input	Fully defined or executed successfully	Indicates that the input is fully defined and initialized the workflow successfully.
Input	Ready to be executed or Outcome is not ready	Indicates that the input is scoped and is ready for initializing workflow
#1 Fill Holes	Completed the operation and the operation is reversible	Indicates the Fill Holes operation is successfully executed and you can revert the operation.
#2 Wrap Parts	ocked the operation due to an error	Indicates that the wrap operation is blocked during execution due to an error.

Refer Understanding the States of Mesh Workflows (p. 679) for details about how the Mesh Workflow states evolve while setting up and executing a workflow.

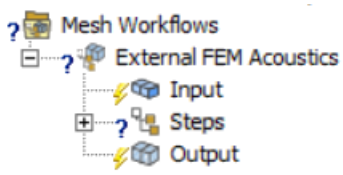
Understanding the States of Mesh Workflows

When your **Mesh Workflows** are inserted from a predefined workflow type or have an under-defined input, the **Mesh Workflows** and the **Input** objects are preceded by a question mark to indicate the mesh workflow is under-defined and requires input data:



When you have not initialized **Mesh Workflow** (that is, input geometry is scoped or defined but not transferred), **Mesh Workflows** is preceded by a question mark and the **Input** object is preceded by a lightning bolt to indicate the mesh workflow requires action:

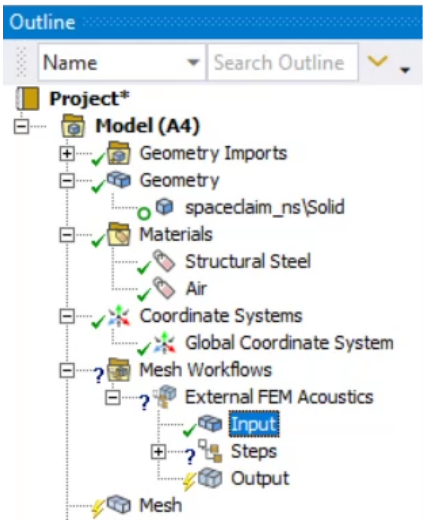
Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



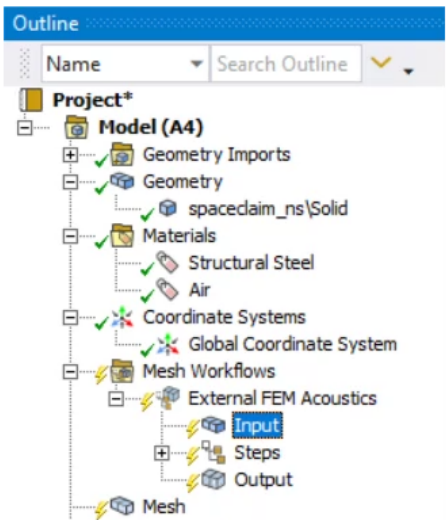
Note:

When you scope the input geometry and initialize the workflow, the original body is marked as inactive using a circle symbol under the **Geometry** object as shown below. You are not allowed to scope this body for meshing using other **Mesh Methods** or **Mesh Workflow** objects. The circle symbol also indicates an inactive body that does not participate in other mesh workflows.

Initialized Workflow



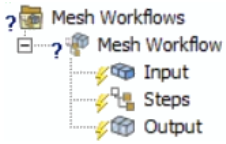
Cleared (Generated) Workflow (Data)



You may view the following changes:

- **Suppressed** in the Details view becomes read-only and parameterization applied on bodies are removed.
- **Geometry** window does not display the inactive bodies when you select any of the bodies under **Geometry** object in the **Tree** Outline.

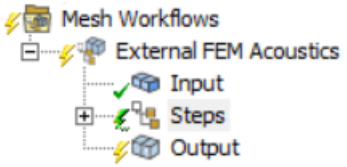
When your mesh workflow contains no **Mesh Workflow Steps**, the **Mesh Workflows** and its child-objects are preceded by a question mark to indicate the **Mesh Workflow Type** is under-defined, whereas the **Input** object is preceded by a lightning bolt to indicate the input geometry is scoped or defined but not transferred:



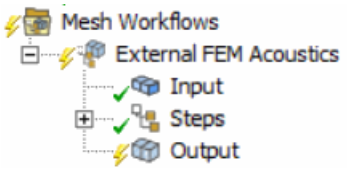
When your mesh workflow is initialized, the **Input** object is preceded by a check mark to indicate the input geometry is fully transferred. Additionally, when some of the workflow steps are in the process

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

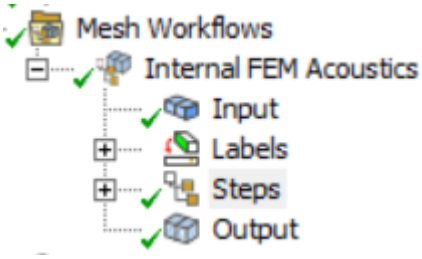
of being executed, the **Steps** object is preceded by a green lightening bolt to indicate that the steps execution is in progress:



When your mesh workflow is initialized, the **Input** object is preceded by a check mark to indicate the input geometry is fully transferred. Additionally, when your workflow steps are fully executed, the **Steps** object is preceded by a check mark to indicate that the workflow generated data is ready for output:



When your mesh workflow is completed, the **Mesh Workflows** and all its child-objects including **Output** object are preceded by a check mark to indicate the workflow generated data is fully transferred to **Mesh** and **Geometry** object respectively.



Note:

A new body is created under the **Geometry** object after generating the **Output**, the created body might be preceded by a question mark to indicate the geometry is under-defined (for instance, there is no materials assigned to the zone properties).

Understanding the States of Mesh Workflow Steps

Mesh Workflow Steps use the following icons to denote step state:

- A check mark ✓ denotes a executed step which has checkpoint defined and you can revert to the step after executing workflow.
- A check mark with a hash ✓ denotes a step that is reversible after executed.
- A question mark ? denotes a step that is blocked that needs input from previous step's outcome or user-defined values.
- A lightning bolt indicates that the step is ready to be executed.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A red exclamation mark denotes that the operation is blocked due to an error during execution.

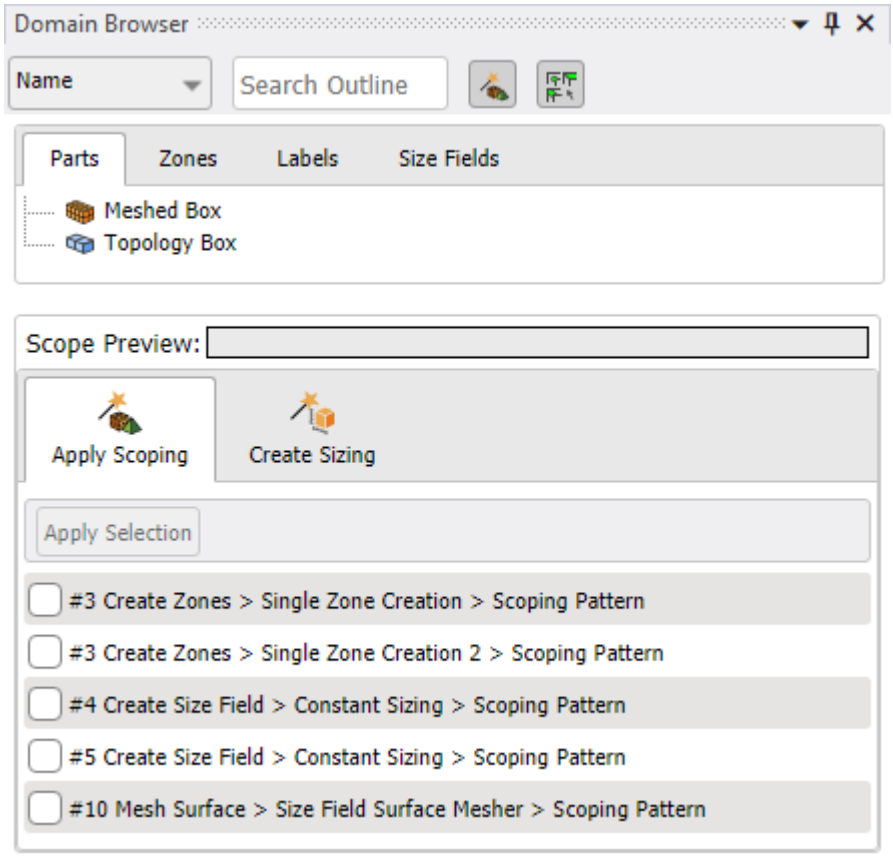
Mesh Workflow Domain Browser

Domain Browser allows you to view the parts, zones, labels or size fields involved in the mesh workflows and to edit individual controls in mesh workflows.

Domain Browser has the following options:

- **Name**: Allows you to search based on the names of parts, zones, labels or size fields.
- **Search Outline**: Allows you filter the entities in the mesh workflow by matching your search criteria. **Search Outline** uses the same regular expression supported in controls.
- : Allows you to enable the wizards to edit and configure the mesh workflow. For information related to **Show Wizards**, refer to Show Wizards (p. 684).
- : Displays the selected entities in the **Geometry** window.
- **Parts**: Displays the parts in the mesh workflows.

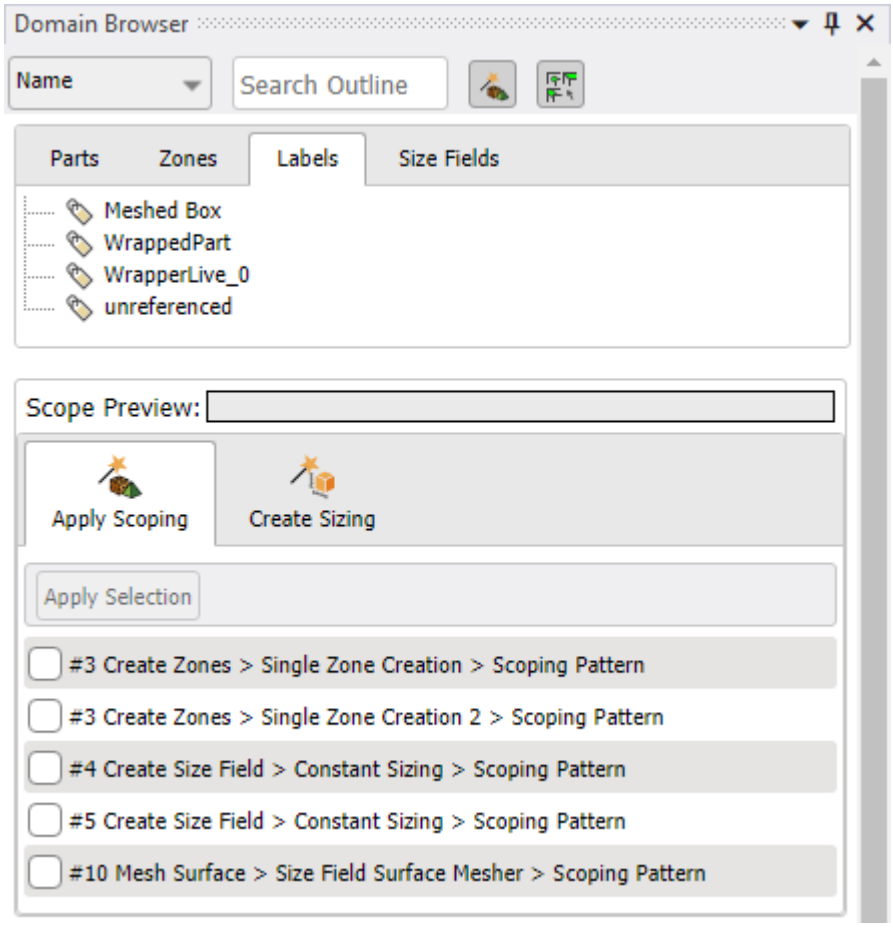
  In **Domain Browser**, **Parts** tab have the following icons:



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - : Denotes the parts with topology.
  - : Denotes the parts without topology.
- **Zones**: Displays the zones in the mesh workflows.
  - : Denotes the edge zones in the **Mesh Workflow**.
  - : Denotes the face zones in the **Mesh Workflow**.
  - : Denotes the volume zones in the **Mesh Workflow**.
- **Labels**: Displays the labels in the mesh workflows. **Labels** are groups of entities that you use in various mesh workflow operations. When you initialize a workflow, the named selections in the imported geometry become **Labels** in the **Mesh Workflow**.

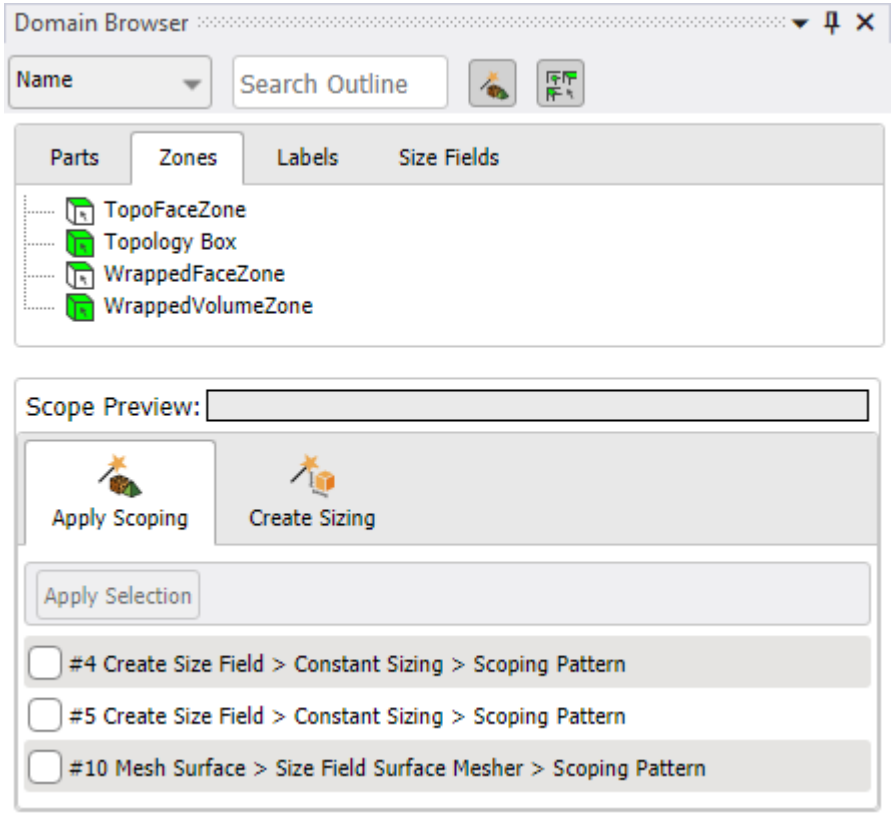
  In **Domain Browser**, **Labels** tab have the following icons:



  - : Denotes the labels in the **Mesh Workflow**.
- **Size Fields**: Displays the size fields in the mesh workflows.

  In **Domain Browser**, **Size Fields** tab have the following icons:

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



– : Denotes the size fields in the **Mesh Workflow**.

when you click parts, zones, labels and size fields right-click options available are:

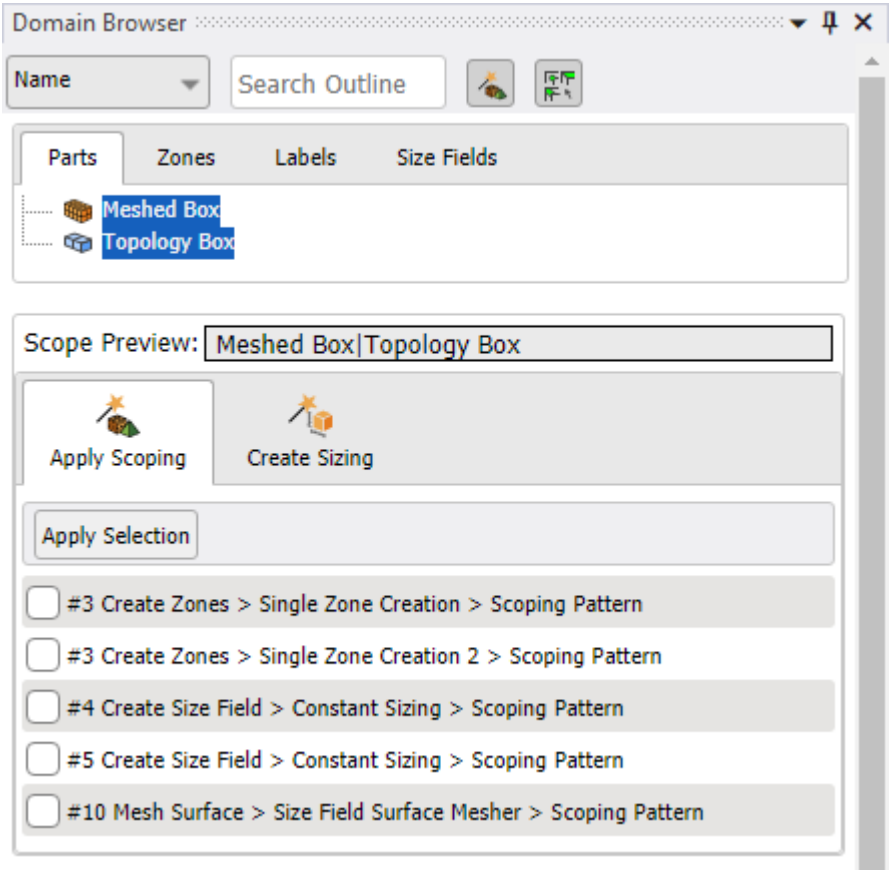
- **Select All**: Allows you to select all the entities.
- **Unselect All**: Allows you to unselect all the entities.

Show Wizards

Show Wizards ( ) displays the wizards in the **Domain Browser**. **Domain Browser** has the following wizards:

- **Apply Selection** (p. 685): Allows you to scope the entities for scoping patterns displayed in the **Domain Browser**.
- **Create Sizing** (p. 686): Allows you to create size field controls for existing **Create Sizing** operations for the mesh workflow model.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Wizards have the following option:

- **Scope Preview**: Previews the current pattern that is used by the wizards. If you select some entities (parts, zones, labels or size fields) in the **Domain Browser**, the **Scope Preview** shows a pattern that matches all the selected entities. If you do not select any entities, the **Scope Preview** shows the pattern defined in the **Search Outline** to match the current **Domain Browser** entity filter.

Apply Scoping

Scope the parts, zones, labels or size fields for scoping patterns and size fields for the size field name patterns available in the **Domain Browser**.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

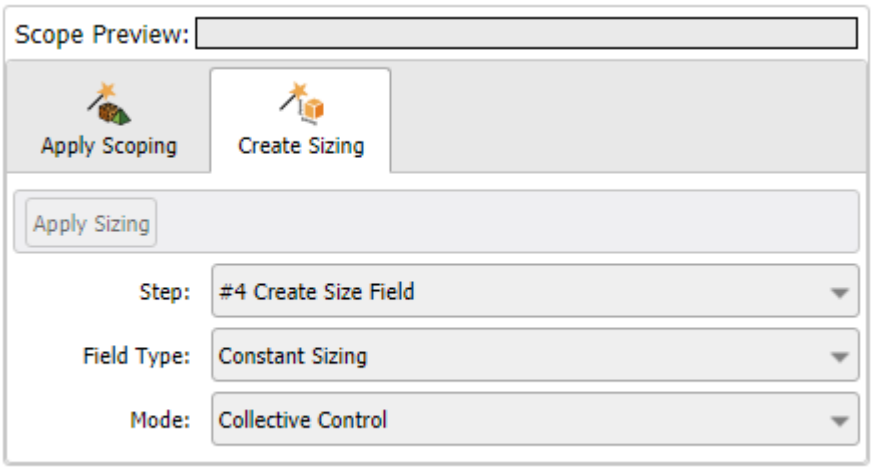
Apply Scoping has the following option:

- **Apply Selection**: Allows you to select the parts, zones, labels or size fields from **Domain Browser** for the scoping pattern. **Apply Selection** is available only when you select parts, zones, labels or size fields in the **Domain Browser**.

  To scope entities in **Domain Browser**, select the entities in **Domain Browser**, select the check box below the **Apply Selection** for the corresponding scoping patterns, and then click **Apply Selection** to scope the entities to the selected patterns.

Create Sizing

Create size field controls for the existing **Create Size Field** operations for the mesh workflow model. **Create Sizing** is available only when a **Create Size Field** operation is available in the **Mesh Workflow**.



Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create Sizing has the following options:

- **Apply Sizing**: Allows you to select the parts, zones or labels for the size field. **Apply Sizing** is available only when you select parts, zones or label in the **Domain Browser**.
- **Step**: Displays the available **Create Size Field** steps in the mesh workflow.
- **Field Type**: Allows you to select the control type for the **Create Size Field** operation. The available control types are:
  - **Constant Sizing**: Allows you to set the maximum size on the selected scope based on the other applied size controls on the scope.
  - **Curvature Sizing**: Allows you to refine the surface mesh to capture the underlying curve and surface curvature.
  - **Proximity Sizing**: Allows you to compute edge and face sizes in gaps using the specified minimum number of element layers.
  - **Body of Influence Sizing**: Allows you to apply mesh sizing based on the specified element.
- **Mode**: Allows you to select the mode of creating size field for the mesh workflow. The available options are:
  - **Collective Control**: Creates the single size field control type for the selected parts, zones or labels.
  - **Individual Control**: Creates the separate size field control type for selected parts, zones or label. The format for naming a single control is <part name> (<control type>).

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Index

Symbols

0-thickness walls, 470

A

accessing Meshing functionality, 35
activation of Mesh worksheet steps, 448
ADF file format, 389
advanced global meshing controls, 180
algorithms - tetrahedral meshing algorithms , 207
allow nodes to be moved off boundary - sizing global meshing control, 117
Allow Selective Meshing, 389
Ansys Fluent
  input file format, 389
Ansys Fluent Meshing
  exporting faceted geometry to, 42
arbitrary match control
  description, 338
ASCII file format
  Fluent input file format, 389
aspect ratio (base/height) - inflation global meshing control, 170
aspect ratio for quadrilaterals
  mesh metric, 141-142
aspect ratio for triangles
  mesh metric, 140
assemblies of parts vs. multibody parts, 21
assembly - initial size seed meshing control option, 113
automatic inflation - inflation global meshing control, 163
automatic meshing, 89
automatic method - in method control, 206
automatic virtual topology, 546
auxiliary meshing tools - listed and described, 483
average surface area - sizing global meshing control , 120
axis sweeping, 235

B

baffle meshing, 470
batch mode, 84
bias (sweeping) - in method control , 235
bias factor - in local sizing control, 302
bias type - in local sizing control, 302
binary file format
  Fluent input file format, 389
body of influence - local mesh sizing tool option, 302
bounding box diagonal - sizing global meshing control , 119

C

CAD instances, 466
Cartesian
  Additive Manufacturing, 261
  Body Fitted, 261
Cartesian option - in method control, 261
CFD/fluids meshing strategies, 33
CFD/fluids meshing with tetrahedrons, 345
CFL condition, 155
CFX-Mesh method
  replacing, 40
CGNS
  file format, 389
  version, 389
CGNS format export, 42
CGNS Version, 389
characteristic length
  mesh metric, 155
check mesh quality global meshing control, 121
checking overlapping contact regions - procedure, 79
clearing generated data, 541
collision avoidance - inflation global meshing controls, 174
component system
  Mesh, 35
conformal meshing, 21
  mesh method interoperability, 21
contact
  sizing - mesh control tool, 311
contact meshing, 462
contact sizing
  description, 311
coordinate system
  creating section planes, 525
Courant-Friedrichs-Lewy condition, 155
Create Section Plane, 525
curvature min size - sizing global meshing control, 113
curvature normal angle - local mesh sizing tool option, 302
curvature normal angle - sizing global meshing control, 114
CutCell Cartesian meshing
  missing tessellations, 542
  orthogonal quality, 153
cyclic match control
  description, 336

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

D

Default Method, 389
Default Physics Preference, 389
defaults global meshing controls, 96
Defeature Size, 284
defeaturing, 110
  mesh, 110
determination of physics, analysis, and solver settings, 39
dockable worksheet, 448
dynamic defaults - sizing, 104

E

ease of use meshing features - listed, 527
edge behavior - in local sizing control , 302
edge bias - in local sizing control, 302
edge splits
  using F4 to modify, 561
  virtual, 561
element option
  in sweep method mesh control, 235
element order
  default global control setting, 101
  method control setting, 202
element order - default global meshing control, 101
element quality
  finding worst quality elements, 133
  mesh metric, 139
element shape
  meshing according to, 20
element size - default global meshing control, 102
element size - local mesh sizing tool option, 302
elements - statistics global meshing control, 199
elements that do not meet target metric, 538
enable washers - sizing global meshing control, 117
error limits global meshing control , 127
exporting - procedure
  previewed inflation mesh, 535
  previewed surface mesh, 533
exporting faceted geometry
  to Ansys Fluent Meshing, 42
exporting meshes
  Ansys Fluent mesh format, 42
  Ansys ICEM CFD format, 42
  CGNS format, 42
  checking overlapping contact regions, 79
  Mesh Application File format, 42
  overlapping contact regions, 79
  overlapping Named Selections, 79
  Polyflow Classic format, 42
    Pmeshes, 63
  resolving overlapping contact regions, 79
extended ICEM CFD meshing, 83
Extra Retries For Assembly, 389

F

F4
  modifying virtual topology splits, 561, 565
face meshing
  description, 314
face splits
  using F4 to modify, 565
  virtual, 565
File Format, 389
Fill Small Holes, 284
fillet ratio - inflation global meshing controls , 178
first aspect ratio - inflation global meshing control, 170
first layer height - inflation global meshing control, 169
Fluent
  input file format, 389
fluent export format global meshing control, 100
fluent export preview surface mesh global meshing control, 100
Fluent mesh export, 42
fluids meshing strategies, 33
fluids meshing with tetrahedrons, 345
Format of Input File (*.msh), 389
Free Mesh Type, MultiZone
  Hexa Core, 242
  Hexa Dominant, 242
  Not Allowed, 242
  Tetra, 242

G

gasket mesh control
  object reference, 349
gasket mesh control object reference, 349
generating mesh - procedure, 528
generating mesh controls from a template, 201, 540
generation of contact elements, 483
Geometry window, 36
global meshing settings - listed and defined, 93
Graphics window, 36
growth rate - inflation global meshing control , 169
growth rate - local mesh sizing tool option, 302
growth rate - sizing global meshing control, 110
growth rate type - inflation global meshing controls , 177

H

hard divisions - in local sizing control, 302
hard entities, 468

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

hard points
using in virtual face splits, 565, 568
HDF5 file format, 389
height of washer - sizing global meshing control, 117
hex dominant option - in method control, 233
histogram of mesh metrics, 133
history
meshing steps, 448

I

ICEM CFD
batch mode, 84
extended meshing, 83
interactive meshing, 83
interactive mode, 84
writing ICEM CFD files, 84
ICEM CFD mesh export, 42
importing meshes
Ansys CFX, 40
Ansys Fluent, 40
Ansys ICEM CFD, 40
ANSYS Polyflow Classic, 40
imprinting
classifications for MultiZone, 427
effects on meshing, 21
matched vs. non-matched, 21
incremental meshing, 444
incremental meshing of bodies, 528
inflation advanced options - inflation global meshing controls, 174
inflation algorithm - inflation global meshing control, 170
inflation controls
with all triangles mesher, 453
with MultiZone, 453
with patch conforming mesher, 453
with patch independent mesher, 453
with quadrilateral dominant mesher, 453
with sweeper, 453
Inflation element type - inflation global meshing control, 174
inflation global meshing controls, 161
inflation mesh control tool, 345
inflation option - inflation global meshing control , 166
Inflation options on the Options dialog box, 389
initial size seed - sizing global meshing control , 113
inspecting large meshes, 540
instances, 466
interactions - mesh control, 475
interactive ICEM CFD meshing, 83
interactive mode, 84
interoperability
mesh method, 21

J

Jacobian ratio
mesh metric, 142

L

launching the Meshing application, 35
layer compression - inflation global meshing controls, 174
legacy data, 40
linear elements, 101
local meshing settings - listed and defined, 201
loop removal global meshing controls, 196
loop removal tolerance - loop removal global meshing control, 197
loops
removing, 538
showing removable loops, 538

M

manual creation of virtual cells, 546
Mapped Mesh Type, MultiZone
Hexa, 242
Hexa/Prism, 242
Prism, 242
match control
arbitrary, 338
cyclic, 336
description, 334
match meshing - and symmetry, 462
matched vs. non-matched imprinting, 21
max size - sizing global meshing control, 110
maximum angle - inflation global meshing controls , 178
maximum corner angle
mesh metric, 150
maximum layers - inflation global meshing control , 169
maximum thickness - inflation global meshing control , 169
Merge Edges Bounding Manually Created Faces, 389
merge face edges - automatic virtual topology, 546
Mesh Application File format export, 42
mesh connection, 484
mesh control interaction tables, 475
mesh control tools
description, 201
instances, 466
patterns, 466
precedence, 201

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

renaming, 484
mesh copy
description, 331
mesh data
clearing, 541
mesh defeaturing global meshing controls, 110
mesh edit, 484, 508, 512
snap to boundary, 484
mesh match, 484
mesh method interoperability, 21
mesh metric
histogram, 133
mesh metric - quality global meshing control
aspect ratio for quadrilaterals, 141-142
aspect ratio for triangles, 140
characteristic length, 155
element quality, 139
Jacobian ratio, 142
maximum corner angle, 150
orthogonality quality, 153
parallel deviation, 149
skewness, 151
warping factor, 147
mesh metric - statistics global meshing control , 133
Mesh Metrics bar graph, 133
mesh numbering, 484
mesh objects
grouping by type, 543
mesh quality workflow, 120
mesh refinement, 461
mesh sweeping
general sweeping, 399
thin model sweeping, 406
Mesh system, 35
meshing
algorithms - tetrahedral, 207
assembly level vs. part/body level, 20
automatic, 89
auxiliary tools, 483
by element shape, 20
capabilities in Workbench, 19
CFD/fluids, 345
checking overlapping contact regions - procedure, 79
clearing generated data, 541
conformal meshing, 21
control interactions, 475
controls, 89
determination of physics, analysis, and solver settings, 39
ease of use features - listed, 527
exporting - procedure
previewed inflation mesh, 535
previewed surface mesh, 533
fluids, 345
generating mesh - procedure, 528
generating mesh controls from a template, 201, 540
global controls - listed and defined, 93
grouping meshing objects, 543
implementation in Workbench, 19
importing meshes, 40
inspecting large meshes, 540
local mesh controls, 201
loop removal, 538
mesh sweeping
general sweeping, 399
thin model sweeping, 406
Meshing application - basic workflow, 27
Meshing application - basic workflow for CFD, 28
Meshing application - basic workflow for combining CFD/fluids and structural meshing, 31
Meshing application - basic workflow for Fluids, 28
Meshing application - description, 27
Meshing application - exporting meshes, 42
Meshing application - launching, 35
Meshing application - working with legacy mesh data, 40
Meshing application interface, 36
Named Selections, 77
non-conformal meshing, 21
options
export, 389
inflation, 389
meshing, 389
overview, 389
sizing, 389
virtual topology, 389
overlapping contact regions, 79
overlapping Named Selections, 79
overview, 19
parameters, 87
previewing inflation - procedure, 534
previewing source and target mesh - procedure, 533
previewing surface mesh - procedure, 531
repairing geometry in overlapping Named Selections, 79
replacing a Mesh system with a Mechanical Model system, 36
resolving overlapping contact regions - procedure, 79
restarting the mesher, 528
selective, 444, 528

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

showing elements that do not meet target metric - procedure, 538
showing geometry in overlapping Named Selections - procedure, 79
showing inflation surfaces - procedure, 536
showing mappable faces - procedure, 542
showing missing tessellations, 542
showing problematic geometry - procedure, 537
showing removable loops, 538
showing sweepable bodies - procedure, 537
sizing options, 89
specialized - 0-thickness walls, 470
specialized - baffle meshing, 470
specialized - CAD instance, 466
specialized - contact meshing, 462
specialized - hard entities, 468
specialized - inflation controls, 453
specialized - listed, 399
specialized - match meshing and the symmetry folder, 462
specialized - mesh refinement, 461
specialized - mixed order meshing, 461
specialized - MultiZone, 419
specialized - non-manifold faces, 470
specialized - pyramid transitions, 462
specialized - rigid body contact meshing, 463
specialized - rigid body meshing, 463
specialized - spot weld, 468
specialized - sweeping, 399
specialized - thin solid meshing, 466
specialized - winding body meshing, 462
specialized - wire body meshing, 462
stopping the mesher, 528
troubleshooting, 579
types - listed, 20
updating the Mesh cell state - procedure, 527
Workbench vs. Mechanical APDL, 87
workflow, 19
workflow - basic meshing, 27
workflow - CFD meshing, 28
workflow - combining CFD/fluids and structural meshing, 31
workflow - Fluids meshing, 28
workflows, 601
Meshing application
basic meshing workflow, 27
basic workflow, 27
basic workflow for CFD, 28
basic workflow for fluids, 28
combining CFD/fluids meshing and structural meshing, 31
description, 27
determination of physics, analysis, and solver settings, 39
interface overview, 36
strategies for CFD/fluids meshing, 33
meshing support for hard entities, 468
meshing support for pattern instances, 466
meshing support for spot welds, 468
method mesh control tool
description, 202
interactions - MultiZone quad/tri, 475
midside nodes
mixed order meshing, 203
minimum edge length - sizing global meshing control , 120
mixed order meshing, 203, 461
model assembly global meshing controls, 199
multibody parts vs. assemblies of parts, 21
MultiZone, 419
Free Mesh Type
Hexa Core, 242
Hexa Dominant, 242
Not Allowed, 242
Tetra, 242
imprinting classifications, 427
Mapped Mesh Type
Hexa, 242
Hexa/Prism, 242
Prism, 242
MultiZone option - in method control, 242
MultiZone quad/tri mesh control interactions, 475
MultiZone quad/tri option - in method control, 284
MultiZone Sweep Sizing Behavior
options, 389

N

Named Selections
and regions for CFX applications, 77
exporting faceted geometry to Fluent Meshing, 42
inspecting large meshes, 540
program controlled inflation and, 164
sending to solver, 78
node merge, 508
node move, 512
nodes - statistics global meshing control, 198
non-conformal meshing, 21
non-manifold faces, 470
num cells across gap - sizing global meshing control , 115
Number of CPUs for Meshing Methods, 389
Number of CPUs for Parallel Part Meshing, 389

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

number of layers - inflation global meshing control , 169

O

Object Generator
  using to generate mesh controls, 201, 540
Options dialog box
  export, 389
  inflation, 389
  meshing, 389
  overview, 389
  sizing, 389
  virtual topology, 389
orthogonal quality
  mesh metric, 153
overriding of mesh control tools, 201

P

parallel deviation
  mesh metric, 149
Parameter Workspace
  Meshing application and, 87
parameters
  Meshing application and, 87
part - initial size seed meshing control option, 113
patch conforming tetrahedral meshing algorithm , 208
patch independent tetrahedral meshing algorithm , 210
  missing tessellations, 542
patterns, 466
persistence
  meshing steps, 448
physics preference default global meshing control , 96
pinch control
  defining pinch control automation, 187
  defining pinch controls locally, 340
  defining pinch controls manually, 340
  generate pinch on refresh - pinch global meshing control , 192
  pre, 187, 340
  snap to boundary, 340
pinch controls
  post, 484
pinch global meshing controls, 187
pinch tolerance - pinch global meshing control , 192
Pmeshes, 63
Polyflow Classic format export, 42
  Pmeshes, 63
Polyflow Classic import, 40
polyflow export unit global meshing control, 100
post inflation - inflation global meshing control, 170
post pinch controls, 484
pre inflation - inflation global meshing control, 170
precedence of mesh control tools, 201
previewing inflation - procedure, 534
previewing source and target mesh - procedure, 533
previewing surface mesh - procedure, 531
problematic geometry, 537
program controlled inflation, 164
projecting nodes to underlying geometry, 546
proximity min size - sizing global meshing control, 115
proximity size function sources - sizing global meshing control, 115
pyramid transitions, 462

Q

quadratic elements, 101
quadrilateral dominant option - in method control , 282
quality global meshing controls, 120
quality metrics, 133

R

recording
  selective meshing steps, 448
refinement, 461
  description, 313
removing loops, 538
renaming mesh control tool, 484
repairing overlapping Named Selections, 79
replacing a Mesh system with a Mechanical Model system, 36
replacing CFX-Mesh method, 40
resolving overlapping contact regions - procedure, 79
rigid body behavior - advanced global meshing control, 182
rigid body contact meshing, 463
rigid body meshing, 463

S

section planes
  creating, 525
selective meshing, 444, 528
  adding steps manually, 448
  history, 448
  Mesh worksheet, 448
  recording steps, 448
Send to Solver, 78
sheet loop removal - loop removal global meshing control, 197
showing elements - procedure, 538
showing geometry in overlapping Named Selections - procedure, 79

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

showing inflation surfaces - procedure, 536
showing mappable faces - procedure, 542
showing missing tessellations, 542
showing problematic geometry - procedure, 537
showing removable loops, 538
showing sweepable bodies - procedure, 537
sizing
  description, 296
  specifying options, 109
sizing global meshing control, 105
  capture curvature, 107
  capture proximity, 107
  influence of, 89
  uniform, 108
sizing global meshing controls, 103
skewness
  mesh metric, 151
smoothing - sizing global meshing control, 133
smoothing iterations - inflation global meshing controls, 180
Snap to Boundary, 340
snap to boundary, 484
Snap Type, 340
soft divisions - in local sizing control , 302
solver preference default global meshing control , 99
span angle center - sizing global meshing control , 112
specialized meshing, 399
sphere of influence - local mesh sizing tool option, 302
splitting
  edges, 561
  faces, 565
spot welds, 468
stair stepping - inflation global meshing controls, 174
statistics
  quality, 133
  virtual topology, 576
statistics global meshing controls, 198
straight sided elements - advanced global meshing control, 180
strategies for CFD/fluids meshing, 33
Sweep Element Size, MultiZone, 242
sweep option - in method control, 235
sweeping - mesh
  general sweeping, 399
  thin model sweeping, 406
symmetry - and match meshing, 462
system
  Mesh, 35

T

target Jacobian ratio - default global meshing control, 131
target quality - default global meshing control, 130
target skewness - default global meshing control, 131
template
  using to generate mesh controls, 201, 540
tessellations
  missing, 542
tetrahedral meshing algorithms, 207
Tetrahedrons meshing
  missing tessellations, 542
  orthogonal quality, 153
thin model sweeping, 235
thin solid meshing, 466
topology checking control, 183
transition - sizing global meshing control, 112
transition ratio - inflation global meshing control , 168
triangle option - in method control , 284
troubleshooting, 579

U

Unmeshable Areas, 389
updating the Mesh cell state - procedure, 527
use automatic inflation - inflation global meshing control, 163
use post smoothing - inflation global meshing controls, 179
use sheet thickness for pinch - pinch global meshing control , 191

V

Verbose Messages from Meshing, 389
virtual cell
  creation, 546
  projecting to underlying geometry, 546
virtual hard vertex
  creation, 565, 568
  using F4 to modify, 565, 568
virtual split edge
  creation, 561
  using F4 to modify, 561
virtual split face
  creation, 565
  using F4 to modify, 565
virtual topology
  automatic mode, 546
  common features, 572
  common operations, 568
  overview, 545
  repair mode, 546

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

statistics, 576
Virtual Topology Properties dialog, 573
Virtual Topology Properties dialog, 573

W

warning limits global meshing control , 127
warping factor
mesh metric, 147
winding body meshing, 462
wire body meshing, 462
Workbench
meshing capabilities, 19
Workbench meshing vs. Mechanical APDL meshing, 87
workflows , 27
worksheet
selective meshing, 448
step activation and deactivation, 448
worst quality elements, 133
Write ICEM CFD Files, 284
writing ICEM CFD files, 84

Release 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.