

Discovery

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

Version 2026 R1
March 2026

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

Contents

1: About the Ansys Discovery Application 34
1.1. Installation and Licensing Considerations 34
1.2. Quality Assurance Services 34
1.3. Functional Differences Between Simulation Stages 35
1.3.1. Structural Physics 35
1.3.2. Fluid Flow Physics 36
1.3.3. Solid Thermal Physics 37
1.3.4. Physics Combinations and Calculation Types 37
1.3.5. Fidelity Control 38
1.3.6. Optimization Functionality 38
1.3.7. Monitoring Functionality 38
1.3.8. Results Display 40
1.4. Terms to Know 41
1.5. The SID: A Visual Summary 44
1.5.1. SID Applied Physics 45
1.5.2. SID Progress Status 46
1.6. Overview of Discovery Application User Help 47
1.6.1. In-product Guidance 47
1.6.1.1. What's New 48
1.6.1.2. AnsysGPT™ 48
1.6.1.3. Interactive Help 48
1.6.1.4. Interactive Tours 50
1.6.1.5. Search 52
1.6.1.6. Contextual Feedback 53
1.6.2. Online Resources 54
1.6.2.1. Accessing Online Help 54
1.6.2.2. Installing the Local Documentation Package 54
1.6.3. Did You Know? 55
1.6.3.1. Running the Discovery Application from the Command Line 55
1.6.3.2. Shortcuts 56
1.6.3.3. User Tips 58
2: File Operations 59
2.1. Saving Projects 59
2.2. Creating and Opening Projects 59
2.3. Transfer Discovery Application Data to Another Application 59
2.3.1. Discovery Application File Transfer Considerations 60
2.4. Supported File Types for Import and Export 62
2.5. Importing an Autodesk Fusion Simulation Project 64
2.6. Generating a Project Report 65
2.6.1. Adding Notes 66

2.7. Updating Geometry From A New CAD File....66
2.8. Geometry Transfer....66
2.9. Components and Assemblies....67
2.9.1. Working With Components....68
2.10. Running the Discovery Application from the Command Line....68
3: Discovery Application Settings....71
3.1. General....71
3.2. Navigation....73
3.3. Licensing....73
3.4. Units and Display Precision....74
3.5. Physics....76
3.6. Results....81
3.7. Detailing....83
3.8. Sheet Metal....87
3.9. Advanced....92
3.10. Remote Services....93
3.11. File....94
3.11.1. General Settings for File Import and Export....95
3.11.2. Settings for Optimizing File Import and Export....96
3.12. Customize....105
3.13. Add-Ins....106
3.14. Extensions....106
3.15. About Discovery....106
4: Selecting....107
4.1. Select Tool and Modes....107
4.2. Using the Select Tool....107
4.3. Using the Selection Context Menu....110
4.4. Clearing a Selection....110
4.5. Reverting Your Selection....111
4.6. Selecting by Rolling the Mouse Wheel....111
4.7. Select Tool Guide....111
4.8. Filtering the Selection....111
4.9. Using Box Mode....113
4.9.1. Selecting Objects by Drawing a Box....113
4.9.2. Changing Your Selection....113
4.9.3. Box Selection Examples....114
4.10. Using Lasso Mode....114
4.10.1. Selecting Objects by Drawing a Freeform Shape....115
4.10.2. Lasso Selection Example....115
4.11. Using Polygon Mode....115
4.11.1. Selecting Objects Using a Polygon....116
4.11.2. Polygon Selection Example....116
4.12. Using Paint Mode....116
4.12.1. Selecting Objects by Painting....116

4.13. Using Boundary Mode....117
4.13.1. Selecting Objects Using a Boundary....117
4.13.2. Boundary Selection Example....118
4.14. Select Components Mode....118
4.15. Advanced Selection....118
4.15.1. Named Selections and Queries....119
4.15.1.1. Creating a Named Selection....119
4.15.1.2. Overlapping Named Selections....119
4.15.1.3. Modifying Named Selections....119
4.15.1.4. Using the Query Editor....120
4.15.2. Power Selecting Objects....121
4.15.2.1. Selection Categories....122
4.15.3. Layers....124
4.15.3.1. Working With Layers....125
4.15.4. Opening Properties....125
4.15.4.1. Modifying Object Properties....125
4.15.4.2. Modifying Analysis Properties....126
4.15.4.3. Creating Custom Component Properties....126
5: Designing....127
5.1. Clipboard....127
5.1.1. Pasting an Object into the Graphics Window....127
5.1.2. Pasting an Object onto a Face....128
5.1.3. Pasting a Round onto Another Edge....128
5.1.4. Cutting an Object....128
5.1.5. Copying an Object....128
5.1.6. Deleting an Object....128
5.1.7. Style Painter....129
5.1.7.1. Painting Display Properties from One Object to Another....129
5.2. Sketching....129
5.2.1. Using the Sketch Tools....130
5.2.2. Editing a Sketch....131
5.2.3. Toggling Between a Curve and a Construction Curve....131
5.2.4. Mirroring in Sketch Mode....131
5.2.5. Sketch Group....132
5.2.6. Sketch Tool Guides and Options....133
5.2.7. Sketch Grid Tool Cluster....134
5.2.8. The Sketch Plane....135
5.2.8.1. Displaying a Sketch Plane....135
5.2.8.2. Selecting a New Location for the Sketch Plane....135
5.2.8.3. Moving the Sketch Plane....135
5.2.8.4. Viewing the Sketch Plane Head-on....135
5.2.8.5. Switching to the Last Sketch Plane....135
5.2.8.6. Using Alt+Select in Sketch Mode....136
5.2.9. Move Grid Tool....136

5.2.9.1. Moving the Sketch Grid....136
5.2.9.2. Moving the Center of the Sketch Grid....137
5.2.10. Sketching with Constraints....137
5.2.10.1. Saving and Retaining Sketches....137
5.2.10.2. Adding Layout Curves to an Imported Datum....138
5.2.10.3. Constraint Types....138
5.2.10.4. Autoconstrain Tool....142
5.2.10.5. Object States....143
5.2.10.6. Creating Multiple Sketches with Constraints....144
5.2.10.7. Creating a New Sketching Plane....144
5.2.10.8. Viewing Constraints....144
5.2.10.9. Deleting Constraints....145
5.2.10.10. Using Mirror with Sketch Constraints....145
5.2.10.11. Referencing Existing Geometry....146
5.2.11. Sketch Tools....146
5.2.11.1. Line Tool....146
5.2.11.2. Rectangle Tool....151
5.2.11.3. Ellipse Tool....154
5.2.11.4. Circle Tool....155
5.2.11.5. Arcs....158
5.2.11.6. Polygon Tool....162
5.2.11.7. Splines....164
5.2.11.8. Point Tool....170
5.2.11.9. Trimming and Modifying....171
5.2.11.10. Face Curve Tool....179
5.2.11.11. Project to Sketch Tool....180
5.2.12. Working with Sketch Curves in 3D....182
5.2.12.1. Using Other Tools to Create and Edit Sketch Curves in 3D Mode....183
5.2.12.2. Sketch Curves in 3D Examples....183
5.2.13. Editing Sketches....183
5.2.13.1. Editing a Sketch....184
5.2.13.2. Editing the Length and Angle Dimensions of a Line....184
5.2.13.3. Editing the Distance from Another Point....184
5.2.13.4. Changing the Properties of a Sketch Object....184
5.2.13.5. Helpful Tool Guides....185
5.2.13.6. Editing Sketches Examples....185
5.2.14. Copying a Sketch....186
5.2.15. Layout Sketches....186
5.2.15.1. Creating a Layout....186
5.2.15.2. Converting a Sketch to a Layout....187
5.2.15.3. Importing a 2D AutoCAD DXF or DWG File as a Layout....187
5.2.15.4. Editing a Layout....187
5.2.16. Moving in Two Dimensions....187
5.2.16.1. Moving Sketch Entities....187

5.3. Design Modes....188
5.3.1. Switching Design Modes....188
5.3.2. Flipping the View While in Section Mode....189
5.3.3. Mode Ribbon Group....189
5.3.4. Design Mode Examples....189
5.3.5. Editing in Section Mode....189
5.3.5.1. Editing in Cross-Section....190
5.3.5.2. Section Tool Options....190
5.3.5.3. Section Tool Examples....191
5.4. Editing....191
5.4.1. Edit Ribbon Group....192
5.4.2. Pulling with the Pull Tool....193
5.4.2.1. Creating and Editing a Solid....193
5.4.2.2. Pull Tool Guides....197
5.4.2.3. Pull Tool Options....198
5.4.2.4. Pull Tool Examples....199
5.4.2.5. Offsetting or Extruding Faces....202
5.4.2.6. Extending or Extruding Surface Edges....204
5.4.2.7. Rounding Edges....206
5.4.2.8. Rounding Between Faces and Surfaces....211
5.4.2.9. Modifying Rounds....212
5.4.2.10. Chamfering Edges....216
5.4.2.11. Extruding Edges....221
5.4.2.12. Pivoting Edges....223
5.4.2.13. Revolving Faces....224
5.4.2.14. Revolving Edges....226
5.4.2.15. Revolving Helices....228
5.4.2.16. Sweeping....228
5.4.2.17. Drafting Faces....234
5.4.2.18. Creating Slots....237
5.4.2.19. Scaling Solids and Surfaces....240
5.4.2.20. Copying Edges and Faces....242
5.4.2.21. Pulling with the Select tool....243
5.4.2.22. Pivoting an Edge with the Select Tool....244
5.4.2.23. Parameterizing Measurements Through Pull....244
5.4.3. Move Tool....245
5.4.3.1. Moving Objects....246
5.4.3.2. Move Tool Guides....250
5.4.3.3. Move Tool Options....251
5.4.3.4. The Move Handle....252
5.4.3.5. Moving Symmetrically....254
5.4.3.6. Creating Patterns....256
5.4.3.7. Pivoting and Pulling Solids....265
5.4.3.8. Moving a Solid or Surface with the Select Tool....265

5.4.3.9. Moving with the Fulcrum Tool Guide....271
5.4.3.10. Move Grid Tool....273
5.4.3.11. Moving Protrusions and Depressions....273
5.4.3.12. Parameterizing Measurements Through Move....274
5.4.4. Fill Tool....274
5.4.4.1. Using the Fill Tool....275
5.4.4.2. Filling a Region....285
5.4.4.3. Filling Sketch or Layout Lines....285
5.4.4.4. Fill Tool Guides....285
5.4.4.5. Fill Tool Options....286
5.4.4.6. Fill Tool Examples....287
5.4.4.7. Removing Rounds....289
5.4.5. Blend Tool....293
5.4.5.1. Creating a Blend....293
5.4.5.2. Blend Tool Options....294
5.4.5.3. Blending Between Faces....296
5.4.5.4. Blending Between Two or More Edges....299
5.4.5.5. Blending Between Multiple Points....305
5.4.6. Replace Tool....306
5.4.6.1. Replacing a Face....306
5.4.6.2. Replacing Multiple Targets....306
5.4.6.3. Replacing a Target with Multiple Sources....306
5.4.6.4. Replacing Multiple Targets with Multiple Sources....307
5.4.6.5. Replacing Faces Examples....307
5.4.6.6. Aligning or Simplifying Faces....308
5.4.6.7. Simplifying Curves....308
5.4.6.8. Replace Tool Example....308
5.4.6.9. Replace Tool Guides....308
5.4.7. Bend Tool....309
5.4.7.1. Bending a Line or Edge....309
5.4.8. Mathematical Expressions....309
5.5. Intersecting....310
5.5.1. Intersect Ribbon Group....311
5.5.2. Combine and Split Tools....311
5.5.2.1. Splitting Solids and Surfaces....311
5.5.2.2. Merging Solids and Surfaces....312
5.5.2.3. Making 3D Curves at the Intersection Between Faces....312
5.5.2.4. Combine Tool Guides....313
5.5.2.5. Combine Options....313
5.5.2.6. Combine Examples....315
5.5.2.7. Using Combine to Merge Solids and Surfaces....315
5.5.2.8. Merging Solids....315
5.5.2.9. Merging Surfaces....316
5.5.2.10. Merging Surface Protrusions....318

5.5.2.11. Capping a Surface with a Plane....318
5.5.2.12. Removing Material From a Solid....319
5.5.2.13. Splitting Bodies and Surfaces....319
5.5.2.14. Splitting a Surface....321
5.5.2.15. Splitting a Solid with a Surface, Plane, or Another Solid....322
5.5.3. Split Body Tool....323
5.5.3.1. Splitting a Body by One of its Faces....323
5.5.3.2. Splitting a Body Selectively....324
5.5.3.3. Split Body Tool Guides....324
5.5.3.4. Split Body Tool Options....325
5.5.3.5. Split Body Example....325
5.5.4. Splitting Faces or Edges....325
5.5.4.1. Creating an Edge on a Face Example....326
5.5.4.2. Split Face Tool Guides....329
5.5.4.3. Split Tool Options....330
5.5.5. Project Tool....330
5.5.5.1. Projecting the Edges of a Face, Surface, Sketch, or Note Text to a Solid....330
5.5.5.2. Projecting Points....331
5.5.5.3. Project Tool Guides....331
5.5.5.4. Project Tool Options....331
5.5.5.5. Projecting Examples....332
5.6. Datum Objects and Relationships....332
5.6.1. Inserting Planes....332
5.6.1.1. Inserting a Plane....333
5.6.1.2. Inserting a Plane Parallel to the Screen....334
5.6.1.3. Inserting a Plane Through the Mid-point Between Two Points....334
5.6.1.4. Clipping Your Design with a Plane....334
5.6.1.5. Changing the Appearance of a Plane....334
5.6.1.6. Inserting a Plane Example....335
5.6.2. Inserting Axes....335
5.6.2.1. Inserting an Axis....335
5.6.2.2. Inserting an Axis Through the Mid-point Between Two Points....336
5.6.2.3. Inserting an Axis Example....336
5.6.3. Inserting Points....336
5.6.4. Inserting Coordinate Systems....336
5.6.4.1. Inserting a Coordinate System....337
5.6.4.2. Inserting a Coordinate System Based on Selected Faces....337
5.6.4.3. Inserting a Coordinate System Using the Move Tool....337
5.6.5. Pinned Datums....339
5.6.5.1. Pinned Planes....339
5.6.5.2. Pinned Axes....339
5.6.5.3. Pinned Points....340
5.6.6. Inserting Temporary Points, Axes, and Planes....340
5.6.7. Linear Patterns....345

5.6.7.1. Creating a One- or Two-dimensional Linear Pattern....345
5.6.7.2. Creating a Pattern of a Pattern....346
5.6.7.3. Editing Linear Pattern Properties....347
5.6.8. Circular Patterns....350
5.6.8.1. Creating a One- or Two-dimensional Circular Pattern....350
5.6.8.2. Creating a Radial Circular Pattern....352
5.6.8.3. Editing Circular Pattern Properties....353
5.6.9. Fill Patterns....355
5.6.9.1. Creating a Fill Pattern....355
5.6.9.2. Grid Patterns....356
5.6.9.3. Offset Patterns....356
5.6.9.4. Skewed Patterns....356
5.6.9.5. Editing Fill Pattern Properties....357
5.6.10. Shelling a Solid....358
5.6.10.1. Creating a Shell....359
5.6.10.2. Creating a Closed Shell....359
5.6.10.3. Editing a Shell....359
5.6.10.4. Shell Tool Guides....359
5.6.10.5. Creating a Shell Example....359
5.6.11. Creating Offset Relationships....360
5.6.11.1. Creating an Offset Relationship....360
5.6.11.2. Offset Tool Guides....360
5.6.11.3. Creating Offset Relationship Example....360
5.6.12. Creating Mirror Relationships....361
5.6.12.1. Mirroring an Object....361
5.6.12.2. Removing a Mirror Face or Plane....361
5.6.12.3. Mirror Tool Options....362
5.6.12.4. Setting Up a Mirror Relationship....362
5.6.12.5. Finding Similar Faces on the Other Side of a Mirror....363
5.6.12.6. Mirror Tool Guides....363
5.6.12.7. Mirroring Examples....363
5.7. Bodies....365
5.7.1. Geometry from Equations....365
5.7.1.1. Surfaces from Equations....365
5.7.1.2. Curves from Equations....368
5.7.2. Creating Cylinders....371
5.7.2.1. Drawing a Cylinder....372
5.7.2.2. Creating a Swept Pipe....372
5.7.2.3. Cylinder Tool Options....372
5.7.2.4. Cylinder Examples....372
5.7.3. Creating Spheres....374
5.7.3.1. Creating a Sphere Over a Face or Edge....374
5.7.3.2. Sphere Tool Options....374
5.7.3.3. Sphere Examples....374

5.8. Dimensions....375
5.8.1. Dimensioning a Sketch Line During Creation....376
5.8.2. Dimensioning the Start, End or Middle Point of a Sketch Line from Another Point in Your Sketch....376
5.8.3. Dimensioning Existing Sketch Lines....377
5.8.4. Dimensioning While Moving or Pulling....377
5.8.5. Creating a Ruler Dimension....377
5.8.6. Creating an Angular Ruler Dimension....378
5.9. Detach....378
5.9.1. Detaching in 2D....378
5.9.2. Detaching in 3D....378
5.9.3. Detaching a Cylinder....378
5.10. Assemble....379
5.10.1. Tangent Tool....379
5.10.1.1. Defining a Tangent Condition....379
5.10.1.2. Doing It Faster....380
5.10.1.3. Defining an Offset Distance for a Tangent Condition....380
5.10.1.4. Making Objects Tangent without an Assembly Condition....380
5.10.1.5. Making Objects Tangent Tool Guides....380
5.10.2. Align Tool....380
5.10.2.1. Aligning Objects....381
5.10.2.2. Doing It Faster....381
5.10.2.3. Reversing the Sense of an Align Condition....381
5.10.2.4. Defining an Offset Distance for an Align Condition....382
5.10.2.5. Aligning Objects without an Assembly Condition....382
5.10.2.6. Align Objects Tool Guides....382
5.10.2.7. Aligning Objects Examples....382
5.10.3. Orient Tool....383
5.10.3.1. Defining an Orient Condition....383
5.10.3.2. Doing It Faster....384
5.10.3.3. Defining an Offset Angle for an Orient Condition....384
5.10.3.4. Orienting Objects without an Assembly Condition....384
5.10.3.5. Orienting Objects Tool Guides....384
5.10.3.6. Orienting Objects Example....384
5.10.4. Rigid Tool....385
5.10.4.1. Defining a Rigid Condition....385
5.10.5. Gear Tool....385
5.10.5.1. Defining a Gear Condition....385
5.10.5.2. Reversing the Sense of a Gear Condition....386
5.10.5.3. Reversing the Rotation Direction of a Gear Condition....386
5.10.5.4. Gear Tool Examples....386
5.10.6. Anchor Tool....387
5.10.6.1. Defining an Anchor Condition....387
5.11. Automate....387

5.11.1. History Tracking....387
5.11.1.1. Using History Tracking....388
5.11.1.2. History Tracking Options....389
6: Changing the Display....393
6.1. Clipboard....393
6.1.1. Pasting an Object into the Graphics Window....393
6.1.2. Pasting an Object onto a Face....393
6.1.3. Pasting a Round onto Another Edge....394
6.1.4. Cutting an Object....394
6.1.5. Copying an Object....394
6.1.6. Deleting an Object....394
6.1.7. Style Painter....394
6.1.7.1. Painting Display Properties from One Object to Another....395
6.2. Orient....395
6.2.1. Spin Tool....395
6.2.1.1. Spinning Your Design....396
6.2.1.2. Setting the Spin Center....396
6.2.1.3. Displaying the Spin Center Indicator....396
6.2.1.4. Orientation Shortcuts....396
6.2.2. Pan Tool....397
6.2.2.1. Panning Your Design....397
6.2.3. Zoom Tool....398
6.2.3.1. Zooming In and Out....398
6.2.3.2. Using the Zoom Legend....398
6.2.3.3. Zooming the Design to Fit the Design Window....399
6.2.3.4. Zooming into a Selected Area....399
6.2.3.5. Zooming In and Out a Preset Amount....399
6.2.4. Switching Between the Spin, Pan, and Zoom Commands....399
6.3. Style....399
6.3.1. Color Usage....399
6.3.1.1. Changing Object Transparency....400
6.3.2. Graphics Styles....401
6.3.3. Rendering Styles....403
6.3.3.1. Changing the Rendering Style of a Solid or Surface....404
6.3.4. Style Overrides....404
6.3.4.1. Overriding the Layer Color of a Solid, Surface, Face, or Curve....404
6.3.4.2. Overriding Edge Color....405
6.3.4.3. Removing Color Overrides and Returning the Object to the Layer Color and Transparency....405
6.3.5. Edge Styles....405
6.3.6. Layers....406
6.3.6.1. Working With Layers....406
6.4. Grid....407
6.4.1. Showing or Hiding the Sketch Grid....407

6.4.2. Determining How Geometry Above or Below the Grid is Displayed....407
6.4.3. Show Sketch Grid Examples....407
6.5. Display....408
6.6. Visibility....409
6.6.1. Hiding and Showing Objects....409
6.6.2. Clip with Volume Tool....410
6.6.2.1. Clipping a Design Interactively....411
6.6.2.2. Clipping a Design Based on a Selection....411
6.6.2.3. Clipping a Design Based on a Location....411
6.6.2.4. Clip Volume Tool Guides and Options....412
6.6.2.5. Clip with Volume Examples....412
6.6.3. Clip with Plane Tool....414
6.6.3.1. Clipping a Design with Planes....415
6.6.3.2. Reversing the Direction of a Clipped View....415
6.6.3.3. Combining Clips....415
6.6.3.4. Plane Clip Tool Options....416
7: Subdivision Surface Tools....417
7.1. Creating Subdivision Surfaces....417
7.2. Converting To and From Subdivision Surfaces....417
7.3. Converting Topology Optimization Results to Hybrid Models....419
7.4. Using the Warp Tool....421
7.5. Editing Subdivision Surfaces....421
7.6. Modifying Subdivision Surfaces....423
7.7. Viewing Subdivision Surfaces....426
8: Facets....428
8.1. Selecting Facets....428
8.1.1. Using the Expand Tool....428
8.1.2. Using the Shrink Tool....430
8.1.3. Using the Fill Tool....430
8.1.4. Using a Slider to Adjust Selection....431
8.1.5. Selecting Facet Edges....432
8.2. Faceted Body Cleanup....433
8.2.1. Automatically Cleaning up Faceted Bodies....434
8.2.1.1. Using the Check Facets Tool....434
8.2.1.2. Auto Fix Tool....434
8.2.2. Shrinkwrap Tool....435
8.2.2.1. Creating a Shrinkwrap....435
Shrinkwrap Tool Guides and Options....436
8.2.2.3. Shrinkwrap Examples....437
8.2.3. Holes Tool....439
8.2.3.1. Finding and Fixing Holes in Faceted Bodies....439
Holes Tool Guides and Options....440
8.2.3.3. Fixing Holes Examples....440
8.2.4. Intersections Tool....442

8.2.4.1. Checking for Intersections....443
8.2.4.2. Intersections Tool Guides....443
8.2.5. Fix Sharps Tool....443
8.2.5.1. Finding and Fixing Sharps....443
Fix Sharps Tool Guides and Options....444
8.2.6. Over-Connected Tool....445
8.2.6.1. Finding and Fixing Over-Connected Facets....445
8.2.6.2. Over-Connected Tool Guides....446
8.3. Organizing Faceted Bodies....446
8.3.1. Convert Tool....447
8.3.1.1. Converting a Solid Body to a Faceted Body....447
8.3.1.2. Convert Tool Options....447
8.3.2. Using the Separate Tool....449
8.3.2.1. Separate Tool Guides and Options....450
8.3.3. Using the Separate All Tool....450
8.3.4. Using the Join Tool....450
8.3.4.1. Join Tool Guides....451
8.4. Editing Faceted Surfaces....451
8.4.1. Using the Pull or Move Tool....451
8.5. Modifying Faceted Bodies....452
8.5.1. Merge Tool....453
8.5.1.1. Merging Faceted Bodies....453
8.5.1.2. Merge Tool Options....454
8.5.2. Subtract Tool....454
8.5.2.1. Subtracting Faceted Bodies....454
8.5.2.2. Subtract Tool Options....455
8.5.3. Split Tool....455
8.5.3.1. Splitting a Faceted Body....455
8.5.3.2. Split Tool Options....456
8.5.4. Intersect Tool....456
8.5.4.1. Intersecting Faceted Bodies....457
8.5.4.2. Intersect Tool Options....457
8.5.5. Shell Tool....457
8.5.5.1. Shelling a Faceted Body....458
8.5.5.2. Shelling and Adding Infill Using Basic Shapes....458
8.5.5.3. Shell Tool Guides and Options....460
8.5.5.4. Basic Infill Shapes....461
8.5.6. Scale Tool....468
8.5.6.1. Scaling a Faceted Body....468
8.5.6.2. Scale Tool Guides and Options....469
8.5.7. Thicken Tool....469
8.5.7.1. Thickening a Surface Faceted Body....469
8.5.7.2. Thicken Tool Options....470
8.6. Adjusting Facets....470

8.6.1. Smooth Tool....470
8.6.1.1. Smoothing Facets or a Faceted Body....470
8.6.1.2. Smooth Tool Options....471
8.6.2. Reduce Tool....473
8.6.2.1. Reducing the Number of Facets....473
8.6.2.2. Reduce Tool Options....473
8.6.3. Regularize Tool....474
8.6.3.1. Regularizing Selected Facets....474
8.6.3.2. Regularize Tool Options....475
8.6.3.3. Regularize Tool Examples....475
8.7. Inspecting Faceted Bodies....476
8.7.1. Overhangs Tool....476
8.7.1.1. Checking a Body for Overhangs....476
Overhangs Tool Guides and Options....477
8.7.1.3. Overhangs Tool Examples....477
8.7.1.4. Converting the Highlighted Overhang(s) to a Selection....478
8.7.1.5. Minimizing the Overhang....478
8.7.2. Thickness Tool....478
8.7.2.1. Checking Faceted Body Thickness....479
Thickness Tool Guides and Options....479
8.7.2.3. Thickness Examples....480
8.7.3. Cavity Tool....480
8.7.3.1. Checking a Body for Cavities....480
8.7.3.2. Checking for Cavity Examples....480
8.7.3.3. Cavity Tool Options....482
8.8. Reverse Engineering....482
8.8.1. Auto Skin Tool....482
8.8.1.1. Creating the CAD Geometry....482
8.8.1.2. Creating the SubD Hybrid Model....486
8.8.1.3. Auto Skin Tool Guides and Options....486
8.8.2. Skin Surface Tool....487
8.8.2.1. Sketching a Surface Patch....487
8.8.2.2. Skin Surface Tool Guides and Options....488
8.8.2.3. Using Box-select....489
8.8.2.4. Editing Curvature in a Sketch....489
8.8.2.5. Periodic Surfaces....490
8.8.2.6. "Stitching" Surfaces Examples....492
8.8.3. Extract Curves Tool....495
8.8.3.1. Creating Edges with Extract Curves....495
8.8.3.2. Extract Curves Tool Guides and Options....495
8.8.4. Fit Spline Tool....496
8.8.4.1. Fitting a Spline Surface on a Mesh....496
8.8.5. Orient Facets Tool....496
8.8.5.1. Orienting a Faceted Body....496

9: Detailing....499
9.1. Dimension Tool....499
9.1.1. Creating a Dimension Annotation....499
9.1.2. Creating Dimension Annotations Examples....500
9.1.3. Dimensioning Between Virtual Points....501
9.1.3.1. Dimensioning to a Virtual Sharp....502
9.1.3.2. Dimensioning to a Mid-plane Line....502
9.1.3.3. Examples....503
9.2. Note Tool....504
9.2.1. Creating a Note....504
9.2.2. Creating a Note Field....505
9.2.3. Copying a Note....505
9.2.4. Editing a Note....506
9.2.5. Fitting Text Within a Note Box....507
9.2.6. Adjusting the Text Width....508
9.2.7. Adjusting Line and Character Spacing....508
9.2.8. Aligning Multiple Notes....508
9.2.9. Attaching and Detaching Notes to Other Notes....509
9.2.10. Displaying the Text of an Annotation and Hiding the Plane....509
9.2.11. Creating Circular Notes....509
9.2.11.1. Creating a Circular Note....510
9.3. Adding Hyperlinks to Annotations....511
9.3.1. Adding a Hyperlink to a Note....511
9.3.2. Editing a Hyperlink....511
9.3.3. Opening a Hyperlink....511
9.3.4. Removing a Hyperlink....511
9.4. Bill of Materials (BOMs) Tool....512
9.4.1. Adding a Bill of Materials to a Document....512
9.4.2. Editing a Bill of Materials....513
9.4.3. Adding Columns to a BOM Table....514
9.4.4. Exporting a Bill of Materials....514
9.4.5. Using a BOM Template....514
9.4.6. Sorting a Bill of Materials....514
9.4.7. Grouping Rows in a Bill of Materials....515
9.5. Balloon Tool....516
9.5.1. Adding BOM Balloon Notes....517
9.5.2. Automatically Adding BOM Balloon Notes to a View....517
9.6. Note Leader Tool....517
9.6.1. Creating a Note Leader....517
9.6.2. Snapping a Leader to 45° Increments....518
9.6.3. Underlining Note Text with a Leader....518
9.6.4. Inserting an All-Around Symbol....519
9.6.5. Changing a Note Leader's Segments....519
9.6.6. Changing a Note Leader's Properties....519

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

9.6.7. Attaching a Note Leader to a Virtual Sharp....519
9.6.8. Note Leader Examples....519
9.7. Bolt Circle Tool....520
9.7.1. Creating a Bolt Circle....520
9.8. Table Tool....521
9.8.1. Adding a Table....521
9.8.2. Table Editing Operations....521
9.8.3. Moving or Rotating a Table....521
9.8.4. Deleting Text Within a Table....522
9.8.5. Deleting a Table....522
9.8.6. Modifying Table Properties....522
9.8.7. Copying Table Contents....522
9.9. Update Annotations Reading Direction Tool....522
9.9.1. Reorienting Annotations....522
9.10. Center Line Tool....523
9.10.1. Adding a Center Mark or Center Line....523
9.10.2. Center Marks and Lines Examples....523
9.11. Detail Dimensions Tool....523
9.11.1. Scaling 3D Annotation Dimensions....524
9.12. Engineering Notebook....524
9.12.1. Creating an Engineering Notebook....524
9.12.2. Sheet Setup....525
9.12.3. Using the Drawing Scale....525
9.12.4. Rendering View Styles....525
9.12.5. Orienting Views....526
9.13. Engineering Notebook Views....526
9.13.1. Mastering Views....527
9.13.2. General View Tool....527
9.13.3. Projected View Tool....527
9.13.4. Cross Section View Tool....528
9.13.4.1. Cross Section View Tool Guides and Options....529
9.13.5. Detail View Tool....530
9.13.5.1. Detail View Tool Options....531
9.13.6. Broken View Tool....531
9.13.6.1. Broken View Tool Options....532
9.13.7. Broken-out Section View Tool....532
9.13.7.1. Broken-out Section View Tool Options....533
10: Measuring and Analyzing....534
10.1. Quick Measurements....534
10.2. Inspect....535
10.2.1. Measure Tool....535
10.2.1.1. Viewing a Measurement....535
10.2.1.2. Measure Tool Guides....536
10.2.2. Mass Properties Tool....536

10.2.2.1. Viewing Mass Properties of an Object....536
10.2.2.2. Measuring the Projected Area of an Object or Objects....537
10.2.2.3. Measuring the Properties of a Planar Section....537
10.2.2.4. Selecting a Coordinate System....537
10.2.2.5. Measuring a Projected Area....537
10.2.2.6. Mass Properties Tool Guides....538
10.2.3. Environmental Properties Tool....538
10.2.3.1. Displaying Environmental Properties....538
10.2.3.2. Environmental Properties Tool Guides and Options....539
10.3. Interference....539
10.3.1. Displaying the Edges of Intersection....539
10.3.2. Displaying Volumes of Intersection....539
10.3.3. Interference Tool Guides and Options....540
10.4. Quality....540
10.4.1. Normal Tool....541
10.4.1.1. Displaying Normal Directions for Faces....541
10.4.1.2. Normal Tool Options....541
10.4.1.3. Normal Tool Examples....541
10.4.2. Grid Tool....542
10.4.2.1. Viewing a Face or Surface Grid....542
10.4.2.2. Grid Tool Options....542
10.4.2.3. Displaying a Face Grid Examples....542
10.4.3. Curvature Tool....543
10.4.3.1. Displaying Edge Curvature....543
10.4.3.2. Displaying Face Curvature....543
10.4.3.3. Curvature Tool Options....544
10.4.3.4. Curvature Tool Examples....544
10.4.4. Dihedral Tool....544
10.4.4.1. Displaying a Dihedral Graph....545
10.4.4.2. Dihedral Tool Options....545
10.4.5. Draft Tool....545
10.4.5.1. Displaying a Draft Angle....545
10.4.5.2. Draft Tool Guides and Options....546
10.4.5.3. Draft Tool Examples....546
10.4.6. Stripes Tool....547
10.4.6.1. Displaying Stripes....547
10.4.6.2. Stripes Tool Options....548
10.4.6.3. Stripes Tool Examples....548
10.4.7. Deviation Tool....548
10.4.7.1. Displaying Deviation....548
10.4.7.2. Deviation Tool Guides and Options....549
10.4.8. Control Points Tool....550
10.4.8.1. Displaying Control Points....550
11: Repairing Problems....551

11.1. Navigate Through Issues....551
11.1.1. Navigating Identified Issues....551
11.2. Solidify....552
11.2.1. Stitch Tool....552
11.2.1.1. Stitching Faces Together....552
11.2.1.2. Stitch Tool Guides and Options....553
11.2.2. Gaps Tool....553
11.2.2.1. Removing Gaps....554
11.2.2.2. Gaps Tool Guides and Options....554
11.2.3. Missing Faces Tool....555
11.2.3.1. Finding Missing Faces....555
11.2.3.2. Missing Faces Tool Guides and Options....556
11.3. Fix....557
11.3.1. Split Edges Tool....557
11.3.1.1. Consolidating Split Edges....557
11.3.1.2. Split Edges Tool Guides and Options....558
11.3.2. Extra Edges Tool....559
11.3.2.1. Removing Edges....559
11.3.2.2. Extra Edges Tool Guides....560
11.3.3. Duplicates Tool....560
11.3.3.1. Finding and Fixing Duplicate Faces....560
11.3.3.2. Duplicates Tool Guides and Options....560
11.4. Fix Curves....561
11.4.1. Fit Curves Tool....561
11.4.1.1. Fixing and Replacing Small Curves....561
11.4.1.2. Fit Curves Tool Guides....562
11.4.2. Curve Gaps Tool....562
11.4.2.1. Fixing Curve Gaps....563
11.4.2.2. Curve Gaps Tool Guides....563
11.4.3. Duplicate Curves Tool....563
11.4.3.1. Fixing Duplicate Curves....564
11.4.3.2. Duplicate Curves Tool Guides....564
11.4.4. Small Curves Tool....564
11.4.4.1. Finding Small Curves....564
11.4.4.2. Small Curves Tool Guides....564
11.5. Adjust....565
11.5.1. Merge Faces Tool....565
11.5.1.1. Merging Neighboring Faces....566
11.5.1.2. Merge Faces Tool Guides....566
11.5.2. Small Faces Tool....566
11.5.2.1. Removing Small and Sliver Faces....566
11.5.2.2. Small Faces Tool Guides and Options....567
11.5.3. Simplify Tool....568
11.5.3.1. Simplifying a Design....568

11.5.3.2. Simplify Tool Guides....568
11.5.4. Inexact Edges Tool....569
11.5.4.1. Fixing Inexact Edges....569
11.5.4.2. Inexact Edges Tool Guides....569
11.6. Inspect....570
11.6.1. Check Geometry Tool....570
11.6.1.1. Check Geometry Tool Options....570
12: Preparing Designs for Analysis....572
12.1. Beams and Shells....572
12.1.1. Beams....572
12.1.1.1. Assigning an Existing Beam Profile to a Line or Edge....573
12.1.1.2. Assigning a Beam Profile Using a Vertex Pair....575
12.1.1.3. Assigning a Beam Profile Using a Vertex Chain....576
12.1.1.4. Extracting a Beam....576
12.1.1.5. Changing a Beam Orientation....576
12.1.1.6. Connecting a Beam....577
12.1.1.7. Split Beams....578
12.1.2. Midsurface Tool....579
12.1.2.1. Making a Midsurface Body from a Regular Open Surface Body....580
12.1.2.2. Creating Midsurface Faces by Selecting Offset Pairs....580
12.1.2.3. Creating Midsurface Faces Based on an Offset Range....581
12.1.2.4. Midsurface Tool Guides and Options....581
12.1.2.5. Midsurfaces Examples....582
12.1.3. Extend Tool....583
12.1.3.1. Extending or Trimming Adjacent Surfaces....584
12.1.3.2. Creating a Weld Body Between Surfaces....584
12.1.3.3. Extend Tool Guides and Options....586
12.1.3.4. Extending/Trimming Examples....587
12.2. Bolts....587
12.2.1. Defining Bolts....588
12.2.2. Bolt Tool Options....588
12.3. Fillet Welds....589
12.3.1. Defining Fillet Welds....591
12.3.2. Fillet Welds Tool Guides and Options....591
12.4. Changing the Prepare Tab Display....592
12.5. Generate....592
12.5.1. Volume Extract Tool....593
12.5.1.1. Creating a Volume Body....593
12.5.1.2. Creating a Facet-Based Tolerant Volume Body....594
12.5.1.3. Volume Extract Tool Guides and Options....594
12.5.1.4. Extracting Volume Examples....595
12.5.2. Enclosure Tool....597
12.5.2.1. Creating an Enclosure....597
12.5.2.2. Enclosure Tool Guides and Options....598

12.5.2.3. Enclosures Examples....599
12.5.3. Split by Plane Tool....599
12.5.3.1. Splitting Objects by Plane....600
12.5.3.2. Split by Plane Tool Guides and Options....600
12.5.4. Imprint Tool....601
12.5.4.1. Imprinting a Part....601
12.5.4.2. Imprint Tool Guides and Options....602
12.5.5. Wrap Tool....604
12.5.5.1. Wrapping Geometry and Text....604
12.5.5.2. Wrap Tool Guides and Options....605
12.5.5.3. Wrapping Examples....605
12.6. Remove....606
12.6.1. Interference Tool....607
12.6.1.1. Removing Interference....607
12.6.1.2. Interference Tool Guides and Options....608
12.6.2. Suppression Tool....608
12.6.2.1. Using the Suppression Tool....608
12.6.2.2. Suppression Tool Guides....609
12.6.3. Rounds Tool....609
12.6.3.1. Removing Rounds....610
12.6.3.2. Rounds Tool Guides and Options....611
12.6.3.3. Removing Rounds Examples....611
12.6.4. Faces Tool....612
12.6.4.1. Faces Tool Guides....612
12.6.5. Short Edges Tool....612
12.6.5.1. Short Edges Tool Guides and Options....613
12.7. Identify....613
12.7.1. Detect Small Bodies Tool....614
12.7.1.1. Finding Small Bodies....614
12.7.1.2. Small Bodies Tool Options....614
12.7.2. Detect Surface Bodies Tool....615
12.7.2.1. Finding Surface Bodies....615
12.7.2.2. Surface Bodies Tool Options....615
12.7.3. Detect Logos Tool....615
12.7.3.1. Finding Logos....615
12.7.3.2. Logos Tool Options....616
12.7.4. Detect Threads Tool....616
12.7.4.1. Finding Threads....616
12.7.4.2. Threads Tool Options....616
12.7.5. Detect Holes Tool....617
12.7.5.1. Finding Holes....617
12.7.5.2. Holes Tool Options....617
12.7.6. Detect Bad Faces Tool....617
12.7.6.1. Finding Bad Faces....618

12.7.6.2. Bad Faces Tool Options....618
12.7.7. Detect Sliver Faces Tool....618
12.7.7.1. Finding Sliver Faces....619
12.7.7.2. Sliver Faces Tool Options....619
12.7.8. Detect Clearance Tool....619
12.7.8.1. Finding Clearance....619
12.7.8.2. Clearance Tool Options....620
12.7.9. Detect Sweepable Bodies....620
12.7.9.1. Finding Sweepable Bodies....620
12.7.9.2. Sweepable Bodies Tool Options....621
12.7.10. Detect Fasteners Tool....621
12.7.10.1. Finding Fasteners....621
12.7.10.2. Fasteners Tool Options....622
12.8. Shared Topology....622
12.8.1. Navigating Through Shareable Topology....623
12.8.2. Pre-check Tool....623
12.8.2.1. Checking the Topology in Preparation for Sharing....623
12.8.2.2. Precheck Tool Options....624
12.8.3. Share Tool....624
12.8.3.1. Using the Share Topology Tool....625
12.8.3.2. Manually Selecting Topologies to Share....626
12.8.3.3. Share Tool Guides and Options....628
12.8.4. Unshare Tool....629
12.8.4.1. Using the Unshare Tool....629
12.8.4.2. Unshare Tool Guides and Options....630
12.8.5. Share Topology Review Tool....630
12.8.5.1. Using the Share Topology Review Tool....630
12.8.5.2. Share Topology Review Tool Guides and Options....631
12.8.5.3. Share Topology Display Options....631
12.9. Ansys Transfer....633
12.9.1. Fluent Workflow Options....634
13: Icepak....635
13.1. Identify Objects Tool....635
13.1.1. Converting Bodies to Icepak Objects....635
13.1.2. Identify Objects Tool Guides and Options....636
13.2. Icepak Simplify Tool....637
13.2.1. Simplifying Bodies....637
13.2.2. Icepak Simplify Tool Examples....638
13.3. Opening Tool....641
13.3.1. Creating an Opening Surface Body....641
13.4. Fan Tool....642
13.4.1. Creating a Fan....642
13.4.2. Creating a Fan From a Hub and a Case....643
13.4.3. Fan Properties....644

13.5. Grille Tool....644
13.5.1. Creating a Rectangular Grille....645
13.5.2. Creating a Circular Grille....646
13.5.3. Creating a Polygonal Grille....647
13.5.4. Creating an Automatic Grille....649
13.6. Enclosure Tool....650
13.6.1. Creating an Enclosure From a Box-Shaped Object....650
13.6.2. Enclosure Tool Options....651
14: Sheet Metal....654
14.1. Creating and Editing Sheet Metal....654
14.1.1. Highlighting Objects in a Sheet Metal Design....654
14.2. Working with Sheet Metal....655
14.3. Importing Sheet Metal Designs....655
14.3.1. Convert Tool....655
14.3.1.1. Converting a Solid to Sheet Metal....656
14.3.1.2. Convert Tool Guides....657
14.3.2. Square-Up Tool....657
14.3.2.1. Squaring up Sheet Metal Geometry....658
14.3.3. Identify Tool....658
14.3.3.1. Identifying Sheet Metal Geometry....658
14.3.3.2. Identify Tool Guides....659
14.3.3.3. Identify Tool Examples....659
14.4. Sketching Sheet Metal Parts....660
14.4.1. Line Tool....661
14.4.1.1. Sketching a Line in Sheet Metal....662
14.4.1.2. Line Tool Options....663
14.4.2. Tangent Line Tool....663
14.4.2.1. Sketching a Tangent Line in Sheet Metal....664
14.4.2.2. Tangent Line Tool Options....664
14.4.3. Tangent Arc Tool....665
14.4.3.1. Sketching a Tangent Arc in Sheet Metal....665
14.4.3.2. Tangent Arc Tool Options....665
14.4.4. Rectangle Tool....666
14.4.4.1. Sketching a Rectangle in Sheet Metal....666
14.4.4.2. Rectangle Tool Options....667
14.4.5. Three-Point Rectangle Tool....668
14.4.5.1. Sketching a Three-Point Rectangle in Sheet Metal....668
14.4.5.2. Three-Point Rectangle Tool Options....668
14.4.6. Ellipse Tool....669
14.4.6.1. Sketching an Ellipse in Sheet Metal....669
14.4.6.2. Ellipse Tool Options....670
14.4.7. Sweep Arc Tool....670
14.4.7.1. Sketching a Sweep Arc in Sheet Metal....671
14.4.7.2. Sweep Arc Tool Options....671

14.4.8. Three-Point Circle Tool....672
14.4.8.1. Sketching a Three-Point Circle in Sheet Metal....672
14.4.8.2. Three-Point Circle Tool Options....672
14.4.9. Circle Tool....673
14.4.9.1. Sketching a Circle in Sheet Metal....674
14.4.9.2. Circle Tool Options....674
14.4.10. Polygon Tool....675
14.4.10.1. Sketching a Polygon in Sheet Metal....675
14.4.10.2. Polygon Tool Options....675
14.4.11. Spline Tool....676
14.4.11.1. Sketching a Spline in Sheet Metal....676
14.4.11.2. Spline Tool Options....677
14.4.12. Three-Point Arc Tool....678
14.4.12.1. Sketching a Three-Point Arc in Sheet Metal....678
14.4.12.2. Three-Point Arc Tool Options....678
14.5. Editing Sheet Metal Parts....679
14.5.1. Selecting Sheet Metal Objects....679
14.5.2. Sheet Metal Pull Tool....680
14.5.2.1. Pulling a Sheet Metal Part....680
14.5.2.2. Creating Panel Bends....681
14.5.2.3. Creating Walls....682
14.5.2.4. Creating Joggles....682
14.5.2.5. Pulling a Sheet Metal Parts Examples....683
14.5.2.6. Pulling a Partial Wall....685
14.5.2.7. Pulling a Wall Between Two Parallel Walls....685
14.5.2.8. Controlling the Bend Angle of a Pull....685
14.5.2.9. Creating a Round or Chamfer....686
14.5.2.10. Creating a Jog or S Junction....686
14.5.2.11. Sheet Metal Pull Tool Guides and Options....686
14.5.3. Sheet Metal Move Tool....687
14.5.3.1. Moving an Edge Relief....688
14.5.3.2. Moving a Wall Between Joggles....688
14.5.3.3. Sheet Metal Move Tool Example....688
14.5.3.4. Rotating with the Move Tool....688
14.6. Modifying Sheet Metal Parts....690
14.6.1. Junction Tool....690
14.6.1.1. Changing a Junction on a Sheet Metal Design....690
14.6.1.2. Removing a Junction....691
14.6.1.3. Reversing an Overlap Junction....691
14.6.1.4. Creating a Partial Flange Junction....691
14.6.1.5. Junction Tool Options....692
14.6.2. Edge Relief Tool....693
14.6.2.1. Changing an Edge Relief Type and Dimensions....694
14.6.3. Corner Relief Tool....694

14.6.3.1. Adding a Corner Relief....695
14.6.3.2. Changing the Type of a Corner Relief....695
14.6.3.3. Two-Dimensional (2D) and Three-Dimensional (3D) Corner Reliefs....696
14.6.3.4. Two-Dimensional (2D) and Three-Dimensional (3D) Corner Relief Options....696
14.6.3.5. Examples of Corner Relief Styles....697
14.6.3.6. Corner Relief Added to a Shelled Part Converted to Sheet Metal....698
14.6.3.7. Changing a Corner Relief Type Back to Default....698
14.6.3.8. Creating a Smooth Corner Relief Between Equal Bends....698
14.6.3.9. Watertight Corner on a No-Overlap Junction....698
14.6.3.10. Watertight Corner on a Full Overlap Junction....698
14.6.3.11. Watertight Corner on a Partial Overlap Junction....699
14.6.3.12. Converting a Watertight Corner to a Corner with Reliefs....700
14.6.3.13. Removing a Corner Relief....701
14.6.4. Notch Tool....701
14.6.4.1. Creating a Notch....701
14.6.4.2. Modifying a Notch....701
14.6.4.3. Removing a Notch but Leaving the Geometry....702
14.6.4.4. Removing a Notch by Filling....702
14.6.5. Hem Tool....702
14.6.5.1. Creating a Hem Using the Hem Tool....702
14.6.5.2. Creating a Hem Using the Pull Tool....703
14.6.5.3. Creating a Partial Edge Hem....703
14.6.5.4. Creating a Hem on Multiple Edges....704
14.6.5.5. Creating a Hem Using the Move Tool....704
14.6.5.6. Modifying a Hem....705
14.6.5.7. Removing a Hem Object but Leaving the Geometry....705
14.6.6. Miter Tool....705
14.6.6.1. Creating a Miter....706
14.6.6.2. Modifying a Miter....706
14.7. Creating Sheet Metal Parts....706
14.7.1. Split Tool....706
14.7.1.1. Creating a Partial Split or Rip....707
14.7.1.2. Splitting Using an Imprinted Edge or Sketch Curve....707
14.7.1.3. Split Tool Guides....707
14.7.2. Bead Tool....707
14.7.2.1. Adding a Bead to Your Design....708
14.7.2.2. Bead Tool Options....708
14.7.2.3. Bead Tool Examples....708
14.7.3. Bend Tool....709
14.7.3.1. Bending Along a Line....709
14.7.3.2. Bending in Multiple Components at Once....710
14.7.3.3. Reversing Bend Direction....710
14.7.3.4. Removing Bend Segments....710
14.7.3.5. Cutting Holes in Flattened Conical Bends....711

14.7.3.6. Adding Bend Steps....711
14.7.3.7. Creating a Joggle....712
14.7.3.8. Creating a Joggle from Existing Bends....712
14.7.3.9. Creating a Bead Using the Bend Tool....713
14.7.3.10. Using the Bend Tool Guides....713
14.7.3.11. Using the Cross Break Option....714
14.7.3.12. Bend Tool Guides and Options....715
14.7.4. Marker Tool....715
14.7.4.1. Creating a Marker....716
14.7.4.2. Moving a Marker....716
14.7.5. Double Wall Tool....716
14.7.5.1. Creating a Double Wall....717
14.7.5.2. Double Wall Tool Examples....717
14.7.6. Gusset Tool....717
14.7.6.1. Creating a Gusset....718
14.7.6.2. Moving a Gusset....718
14.7.7. Tab Tool....718
14.7.7.1. Creating a Tab....719
14.7.7.2. Creating a Single Tab....719
14.7.7.3. Tab Tool Options....720
14.7.7.4. Tab Tool Examples....721
14.7.8. Hinge Tool....722
14.7.8.1. Creating a Sheet Metal Hinge....722
14.7.8.2. Hinge Tool Options....722
14.7.9. Flange Tool....723
14.7.9.1. Creating a Flange....724
14.7.9.2. Flange Tool Options....724
14.7.10. Engraved or Cut-out Text in Sheet Metal....724
14.7.10.1. Adding Engraved or Cut-out Text to Your Design....724
14.7.10.2. Converting a Sheet Metal Note to a Regular Annotation Note....725
14.7.11. Patterns in Sheet Metal....725
14.7.11.1. Making a Lightweight Pattern....725
14.7.11.2. Converting a Lightweight Pattern to a Normal Pattern....726
14.8. Unfolding Sheet Metal Components....726
14.8.1. Unfolding a Component....726
14.8.2. Unfolding a Panel Bend....728
14.8.3. Unfolding Sheet Metal Component Examples....729
14.8.4. Creating a View of an Unfolded Part in a Drawing Sheet....729
14.8.5. Adding Bend Dimensions to an Unfolded View....730
14.8.6. Exporting an Unfolded Part as .dxf....730
14.8.7. Calculating Unfolded Lengths....730
14.8.8. Calculating Bend Allowances....731
14.8.9. Using a Bend Deduction Table....732
14.8.9.1. Comma-Separated Value File....734

14.8.9.2. Keyword Description Parameters for Bend Tables....735
14.9. Changing Sheet Metal Part Properties....737
14.9.1. Exporting Sheet Metal Part Properties....738
14.9.2. Changing Part Properties Example....739
14.10. Correcting a Sheet Metal Component....739
14.10.1. Identifying Problems When Unfolding Parts....740
15: Materials....741
15.1. Material Properties....741
15.1.1. Tabular Material Properties....744
15.2. Environmental Properties....744
15.3. Changing the Appearance of Materials....744
15.4. User-Defined Material Libraries....745
15.5. Accessing Additional Granta MI Materials....745
15.6. Compressible Flows....746
16: Scripting....748
16.1. Script Editor....748
16.1.1. Inserting a Selection....748
16.1.2. Recording Actions for a Script....748
16.1.3. Writing a Script....749
16.1.4. Publishing Scripts to the Ribbon....751
16.1.5. Embedding Scripts....752
16.1.6. Debug versus Run....752
16.1.7. Using the DiscoveryInputHelper....753
16.1.8. Known Limitations....754
17: Extension Builder....756
17.1. User Interface....756
17.1.1. Extension Builder Toolbar....757
17.1.1.1. Adding a Reference Script....757
17.1.1.2. Adding a Button....757
17.1.1.3. Adding Tool Options....757
17.1.1.4. Previewing an Extension....757
17.1.1.5. Publishing an Extension....757
17.1.2. Extension Panel....757
17.1.2.1. Editing Nodes....758
17.1.2.2. Deleting Nodes....758
17.1.2.3. Adding Buttons to Groups....758
17.1.2.4. Adding Tool Options....758
17.1.2.5. Adding Tool Options Sections....758
17.1.2.6. Drag and Drop Options....758
17.1.3. Properties Panel....759
17.1.3.1. Editing Properties....759
17.1.4. Script Editor....759
17.1.4.1. Managing Script Tabs....759
17.1.5. Console Window....759

17.2. Tools....759
17.2.1. Adding Tools....759
17.2.2. Managing Tree Objects....760
17.2.3. Tool Options....760
17.2.3.1. Adding Tool Options from Toolbar....761
17.2.3.2. Adding Tool Options from Tree....761
17.2.3.3. Adding Text box Value With Units....761
17.2.4. Tool Options Sections....761
17.2.4.1. Adding Tool Options Sections....761
17.2.5. Copy Script Reference....762
17.2.5.1. Copying a Script Reference....762
17.3. API....762
17.4. Intellisense....763
17.5. Preview....763
17.5.1. Previewing an Extension....763
17.5.2. Error Handling....763
17.6. Save and Publish....763
17.6.1. Saving your Extension....763
17.6.2. Publishing your Extension....763
17.7. Help....764
17.7.1. Adding Help....764
17.7.2. Adding Tool Tips....764
17.8. Examples....764
17.8.1. Example 1: Creating a Box....764
17.8.2. Example 2: Creating a Box or Sphere....765
17.8.3. Example 3: Creating a Heat Sink....767
17.8.3.1. Example 3: Working with Foundation....767
17.8.3.2. Example 3: Working with Heat Sink Options....768
17.8.3.3. Example 3: Working with Materials....768
18: Physics Conditions....770
18.1. QuickScope....770
18.1.1. Including Bodies with QuickScope....771
18.1.2. QuickScope Tool Guides and Options....771
18.2. Physics on Faceted Geometry....772
18.3. General Physics Conditions....773
18.3.1. Gravity....773
18.3.2. Symmetry....773
18.4. Structural Conditions....774
18.4.1. Support....774
18.4.1.1. Setting a Support....774
18.4.1.2. Support Tool Guides and Options....775
18.4.2. Force....775
18.4.2.1. Specifying a Distributed Force Load....776
18.4.2.2. Specifying a Torque Force Load....776

18.4.2.3. Force Tool Guides and Options....777
18.4.3. Pressure....778
18.4.3.1. Specifying a Pressure Load....778
18.4.4. Displacement....778
18.4.4.1. Specifying a Displacement Constraint....778
18.4.4.2. Displacement Tool Guides and Options....779
18.4.5. Moment....780
18.4.5.1. Specifying a Moment....780
18.4.5.2. Moment Tool Guides and Options....780
18.4.6. Mass....781
18.4.6.1. Specifying a Mass....781
18.4.6.2. Mass Tool Guides and Options....781
18.4.7. Bearing Load....782
18.4.7.1. Specifying a Bearing Load....782
18.4.7.2. Bearing Load Tool Options....783
18.4.8. Bolt Preload....783
18.4.8.1. Specifying a Bolt Preload in Explore....784
18.4.8.2. Specifying a Bolt Preload in Refine....784
18.4.8.3. Bolt Preload Tool Options....785
18.4.9. Acceleration....785
18.4.9.1. Setting an Acceleration Constraint....785
18.4.9.2. Acceleration Tool Guides and Options....786
18.4.10. Velocity....786
18.4.10.1. Specifying a Velocity Constraint....786
18.4.10.2. Velocity Tool Guides and Options....787
18.5. Structural Connections....787
18.5.1. Contacts....788
18.5.1.1. Defining Contacts....788
18.5.1.2. Contact Tool Guides and Options....790
18.5.2. Automatically Generated Contacts....792
18.5.2.1. Selecting and Reviewing Automatically Generated Contacts....792
18.5.2.2. Converting Contacts in Contact Review....792
18.5.2.3. Reverting Contacts to Default Type....793
18.5.2.4. Modifying the Default Contact Behavior....793
18.5.2.5. Splitting Contacts and Contact Groups....794
18.5.2.6. Default Contacts Tool Guides and Options....795
18.5.3. Joints....797
18.5.3.1. Setting Up a Joint....797
18.5.3.2. Joints Tool Guides and Types....797
18.5.4. Springs....798
18.5.4.1. Specifying a Longitudinal Spring....798
18.5.4.2. Specifying a Torsional Spring....799
18.5.4.3. Springs Tool Guides and Options....799
18.5.5. Connection Assessment....800

18.5.5.1. Setting up Connection Assessment....800
18.5.5.2. Using Connection Assessment....800
18.5.5.3. Connection Assessment Tool Guides and Options....801
18.6. Fluid Flow Conditions....802
18.6.1. Inlets and Outlets....802
18.6.1.1. Setting Up Inlets and Outlets....804
18.6.1.2. Inlet and Outlet Tools....805
18.6.2. Intake and Exhaust Fans....807
18.6.2.1. Fan Curves....807
18.6.2.2. Adding a Fan....807
18.6.2.3. Loading a Pre-Defined Fan Model....808
18.6.2.4. Specifying the Curve for a Fan....808
18.6.2.5. Fan Tools....808
18.6.3. Walls....809
18.6.3.1. Setting Up a Wall Condition....809
18.6.3.2. Thermal Conditions for Walls....810
18.6.3.3. Internal Radiation....811
18.6.3.4. Wall Types and Options....812
18.6.4. Porous Conditions....813
18.6.4.1. Setting Up a Porous Condition....813
18.6.4.2. Deriving the Porous Coefficients Based on Experimental Pressure and Velocity Data....814
18.6.4.3. Specifying Bidirectional and Orthotropic Porous Media....815
18.6.4.4. Porous Tool Guides and Options....817
18.6.5. Rotating Fluid Zones....818
18.6.5.1. Setting Up a Rotating Fluid Zone....819
18.6.6. Internal Fans....819
18.6.6.1. Fan Curves....819
18.6.6.2. Adding an Internal Fan....820
18.6.6.3. Loading a Pre-Defined Fan Model....821
18.6.6.4. Specifying the Curve for a Fan....821
18.6.6.5. Internal Fan Tool Guides and Options....822
18.6.7. Internal and External Flow....822
18.6.7.1. Extracting an Internal Flow Volume....824
18.6.7.2. Creating an External Flow Volume....824
18.6.7.3. Internal Flow Tool Guides....825
18.6.7.4. External Flow Tool Options....825
18.6.8. Fluid-Fluid Interfaces....825
18.6.8.1. Reviewing Fluid-Fluid Interfaces....825
18.6.8.2. Fluid-Fluid Interfaces Tool Guides and Options....826
18.6.8.3. Defining Virtual Walls....827
18.7. Solid Thermal Conditions....828
18.7.1. Temperature....828
18.7.1.1. Specifying a Temperature Condition....828

18.7.1.2. Temperature Tool Guides and Options....828
18.7.2. Convection....829
18.7.2.1. Specifying a Convection Condition....829
18.7.2.2. Convection Tool Guides and Options....829
18.7.3. Heat....830
18.7.3.1. Specifying a Heat Condition....830
18.7.3.2. Heat Tool Guides and Options....830
18.7.4. Insulated....831
18.7.4.1. Specifying an Insulated Condition....831
18.7.4.2. Insulated Tool Guides and Options....831
18.8. Fluid-Solid Heat Transfer....831
18.8.1. Modeling Fluid-Solid Heat Transfer....833
18.8.2. Reviewing Fluid-Solid Interfaces....834
18.8.3. Fluid-Solid Interfaces Tool Guides and Options....836
18.9. Electromagnetics Conditions....837
18.9.1. Creating a Circuit Port....838
18.9.2. Creating a Mode Port....838
18.9.3. Electromagnetics Tools and Options....838
18.9.4. Electromagnetics Best Practices....838
18.10. Electric Conditions....839
18.10.1. Creating a Voltage or Current Source....839
19: Simulation Options....841
19.1. Changing Simulation Options Default Settings....843
19.2. Calculation Types....844
19.3. Structural Linear and Non-Linear Modeling....845
19.4. Modal Structural Simulations....846
19.5. Topology Optimization....846
19.5.1. Optimizing for Strength or Response to Free Vibration....847
19.5.2. Optimizing for a Specified Frequency....851
19.5.3. Optimizing to Minimize Stress....854
19.5.4. Optimizing to Minimize Volume....858
19.5.5. Topology Optimization Goals and Tool Options....862
19.5.6. Manufacturing Constraints....864
19.5.6.1. Setting the Pull Direction....865
19.5.6.2. Specifying the Table Direction....865
19.5.6.3. Specifying Overhang Prevention....866
19.5.6.4. Setting the Protected Depth....867
19.6. Random Vibration Analysis....867
19.6.1. Defining a Random Vibration Analysis....868
19.7. Structural Solution Progression....868
19.8. Fluid Flow LiveGX Solver....868
19.9. Fluid Flow Solver....869
19.10. Fluid Flow Turbulence Modeling....870
19.10.1. Turbulence Model Options....871

19.11. Fluid Flow Meshing Methods....872
19.12. Mass Flow-Weighted Average for Monitors....873
19.13. Fluid Flow Convergence Criteria....873
19.14. Electromagnetics Settings and Convergence Criteria....874
19.15. Simulation Mesh Quality....875
19.16. Troubleshooting Stalled Solutions....875
20: Simulation Optimization....877
20.1. Optimizing Performance with Solution Fidelity....877
20.1.1. Global Solution Fidelity....877
20.1.1.1. Curvature Size Function....878
20.1.1.2. Proximity Size Function....878
20.1.1.3. Curvature and Proximity Size Function....879
20.1.1.4. Fixed Size Function....881
20.1.2. Local Fidelity....881
20.1.2.1. Body of Influence....882
20.1.3. Fidelity Auto Generation for Electromagnetics....882
20.1.4. Specifying Automatic Mesh Refinement for Topology Optimization....883
20.2. Designing with Parameters in Defined Variations....883
20.2.1. Defined Variations Parameters....884
20.2.1.1. Geometry Available as Parameters....884
20.2.1.2. Physics Available as Parameters....884
20.2.1.3. Topology Optimization Options Available as Parameters....885
20.2.1.4. Script Parameters....886
20.2.2. Defined Variations Table....886
20.2.2.1. Defined Variations Generator....887
20.2.2.2. Defined Variations Chart....888
20.2.3. Defined Variations Test Cases and Generator....888
20.2.4. Defined Variations and Ansys Cloud (Explore)....888
20.2.4.1. Activating Cloud Capability....889
20.2.4.2. Remote Solving of Defined Variations (Explore)....889
20.3. Sensitivity Analysis and Predicting Quality Results....890
20.3.1. Running a Sensitivity Analysis....890
21: Mesh Evaluation....891
21.1. Monitoring Mesh Statistics....891
21.2. Monitoring Mesh Quality....891
21.2.1. Element Quality....892
21.2.2. Aspect Ratio....892
21.2.3. Jacobian Ratio....894
21.2.4. Warping Factor....895
21.2.5. Parallel Deviation....897
21.2.6. Maximum Corner Angle....898
21.2.7. Skewness....898
21.2.8. Orthogonal Quality....900
21.3. Mesh Metrics....902

21.3.1. Viewing Mesh Metrics for the Simulation 903
21.4. Mesh Quality Best Practices....903
22: Results....906
22.1. Displaying Results....906
22.1.1. Displaying Vectors....907
22.1.2. Displaying Contours....907
22.1.3. Displaying Iso-Surfaces....908
22.1.4. Displaying Streamlines....909
22.1.5. Displaying Particles....910
22.1.6. Displaying Direction Fields....911
22.1.7. Displaying Near-Field Patterns....911
22.1.8. Displaying Far-Field Patterns....911
22.1.9. Results Legend....912
22.2. Working with Results....913
22.2.1. Inserting a Cut Plane....913
22.2.2. Displaying Deformations....914
22.2.3. Animating Results....914
22.2.4. Restarting the Solver....914
22.2.5. Adding the Optimized Body to the Model....914
22.2.6. Displaying Charts....914
22.2.7. Displaying the Mesh with Local Fidelity for Fluids....914
22.2.8. Generating a Mesh....915
22.3. Results Tools....915
22.4. Monitoring of Results....916
22.4.1. Results Variables....917
22.4.2. Setting Up Monitors....922
22.4.3. Computing Differences in Monitored Locations 923
22.4.4. Modifying and Plotting Monitors....923
22.4.5. Charts....924
22.4.5.1. Setting Up Structural Charts....924
22.4.5.2. Setting Up PSD Charts for Random Vibration Analysis....924
22.4.5.3. Setting Up Electromagnetic Charts....925
22.4.5.4. Modifying and Plotting Charts....925
22.4.5.5. Available Charts....925
22.4.6. Tabular Results....926
22.4.6.1. Available Tabular Results....926
22.5. Saved Scenes....927
22.5.1. Physics Conditions Available as Default Saved Scenes....927
22.5.2. Saving or Exporting Saved Scenes, Snapshots and Animations....928
22.5.2.1. Exporting Scenes to 2D or 3D Image Files....928
22.5.2.2. Creating and Exporting Saved Scenes Snapshots to ZIP....929
22.5.2.3. Saving Scene Animations to MP4....929
22.5.3. Automatic Capture of Scenes for Viewing....929
22.5.3.1. Capturing Scenes Manually 929

22.5.4. Results Comparison Dashboard.....930
22.5.4.1. Comparing Snapshots Results.....930
22.5.4.2. Results Comparison Dashboard Options.....930
23.1.1. Copyright and Trademark Information.....932

1: About the Ansys Discovery Application

The Ansys® Discovery application provides a single, immersive and interactive workspace for creating your model, exploring the simulation design, and analyzing the solution. You can create and modify geometry using direct modeling technology, define simulations, and interact with results in real-time.

The Discovery application uses an intuitive approach to simulate structural, fluid flow, thermal designs or electromagnetics designs. You can:

- Get started with samples, tours and tutorials
- Optimize topology
- Monitor solutions
- Create, iterate and evaluate defined variations
- Access relevant and interactive context-sensitive help

The application offers three stages depending on what you want to do. You can move between stages at any time to access functionality that may be limited to a given stage.

- Focus on direct 3D geometry modeling in the **Model** stage without the distraction of simulation controls or results.
- Define your simulation in **Explore** and/or **Refine** stages.
  - **Explore** stage gives you quick insights by leveraging fault-tolerant GPU meshing and solution. Define a simulation, change the geometry model or physics setup and results are updated in real time.
  - **Refine** stage provides you with design validation by leveraging body-fitted CPU meshing and either GPU or CPU solutions using Ansys® LiveGX and Ansys® Mechanical™ solvers.

This section contains the following topics:

1.1. Installation and Licensing Considerations

The Discovery application requires certain prerequisites for your installation computer. For details, see **Installation Prerequisites**.

The application uses a Pro, Premium, Enterprise (PPE) licensing scheme. For details, see **Product Licensing**.

1.2. Quality Assurance Services

You can verify that your Ansys programs are running properly on your computer and operating system by using different levels of service for verifying and validating the program under Ansys Quality Assurance Services. Such testing can benefit you when your system undergoes changes, such as an upgrade to the operating system or the installation of different libraries.

To enable Quality Assurance Services for the Discovery application, set the following environment variable:

```
ENABLE_ANSYS_QA_SERVICES261 = 1
```

When Quality Assurance Services is enabled:

- A message appears during application startup to indicate the Quality Assurance Services Agreement is supported. To dismiss the message for future runs, select the **Do not show this message again** check box.
- A similar statement appears in the **File Menu** > **Settings** > **About Discovery** dialog box.

1.3. Functional Differences Between Simulation Stages

There are a few differences in capability and behavior between the **Explore** and **Refine** stages. These differences are explored in this section.

This section contains the following topics:

> **Note:** Electromagnetics conditions do not require any additional license. Structural, thermal, and fluid simulation in the **Refine** stage require an additional license, except when the LiveGX solver is being used for fluid simulation.

1.3.1. Structural Physics

The differences between **Explore** and **Refine** for structural conditions are as follows.

Condition	**Explore**	**Refine**
Acceleration	✓	✓
Bearing load	✓	✓
Bolt, preload	✓	✓
Contacts, bonded	✓	✓
Contacts, excluded	✓	✓
Contacts, sliding	✓*	✓
Displacement, combined	✓*	✓
Displacement, rotation	✓*	✓
Displacement, translation	✓***	✓
Force, distributed	✓	✓
Force, torque	✓	✓
Joint, fixed	✓	✓
Joint, hinged	✓	✓
Joint, spherical	✓	✓
Mass	✓	✓
Moment	✓	✓
Pressure	✓	✓
Support, ball	✓	✓
Support, combined	✓	✓
Support, displaced	✓	✓
Support, fixed	✓	✓
Support, hinged	✓	✓

Condition	Explore	Refine
Support, sliding	✓	✓
Velocity	✓	✓
* Friction coefficient values for contacts are approximated in the Explore stage, but separation is allowed.		
⁑ Stiffness behavior for remote displacements is handled differently in Explore and Refine modes.. In the Explore stage, the application uses a rigid behavior. However, the application uses a deformable behavior in the Refine stage.		
⁂ The Refine stage always uses a displacement condition. The Explore stage uses a displacement condition if the face is aligned with the global coordinate system but uses a rigid joint otherwise.		
▯ The Explore stage uses a rigid formulation of the joint, while the Refine stage uses a deformable formulation of the joint.		
▯ The Refine stage always uses a frictionless support. The Explore stage uses a frictionless support if the face is planar and a rigid joint otherwise.		
▯ The Explore stage uses a rigid joint, while the Refine stage uses a deformable joint.		

1.3.2. Fluid Flow Physics

The differences between **Explore** and **Refine** for fluid flow conditions are as follows.

Condition	Explore	Refine
Flow, external (Quick Start)	✓	
Flow, internal (Quick Start)	✓	✓
Fluid-solid interfaces	✓⁑	✓
Fluid-fluid interfaces	✓	✓
Virtual wall	✓	✓
Inlet, pressure	✓	✓
Inlet, mass flow	✓	✓
Inlet, swirling flow	✓	✓
Inlet, velocity	✓⁂	✓⁑
Outlet, mass flow rate	✓	✓
Outlet, pressure	✓	✓
Outlet, velocity	✓	✓
2D Intake fan	✓	✓
2D Exhaust fan	✓	✓
Wall, free slip	✓	✓
Wall, no slip	✓	✓

Condition	Explore	Refine
Wall, rotating	✓	✓
Wall, stationary	✓	✓
Porous	✓	✓
Rotating fluid zone	✓	✓
3D Internal fan	✓	✓
* Idealized as an insulated no-slip wall.		
⁑ Sets Cartesian components of velocity according to some average direction of the face.		
⁂ Implements a normal velocity at all points on the face.		

1.3.3. Solid Thermal Physics

There are no differences between **Explore** and **Refine** for solid thermal physics.

1.3.4. Physics Combinations and Calculation Types

The differences between **Explore** and **Refine** for available physics combinations and calculation types are as follows.

Functionality	Explore	Refine
Structural stress	Static*	Static
Structural with thermal stress	Static*	Static
Structural stress with solid thermal temperature distribution	Static*/Steady state	Static/Steady state
Solid thermal temperature distribution	Steady state, Time-dependent ⁑	Steady state
Modal	Up to 20 modes	Up to 20 modes
Free-modal	Up to 20 modes	Up to 20 modes
Pre-stressed modal	Up to 20 modes	Up to 20 modes
Random vibration	Spectrum Analysis	Spectrum Analysis
Topology optimization	Static Structural, Modal (all types)	N/A
Fluid flow (isothermal)	Steady state, Time-dependent⁑	Steady state
Fluid flow with thermal effects	Steady state, Time-dependent⁑	Steady state
Fluid-solid heat transfer	Steady state, Time-dependent⁑	Steady state
Fluid-Solid heat transfer with electric conduction	Steady state, Time-dependent⁑	Steady state
Electric conduction	Steady state	Steady state
Electric conduction with solid thermal	Steady state	Steady state
Electromagnetic	N/A	Frequency Reponse (FDTD)

Functionality	Explore	Refine
* Linear solutions only		
‡ Constant condition inputs		

1.3.5. Fidelity Control

The differences between **Explore** and **Refine** for fidelity control are as follows:

Functionality	Explore	Refine
Global fidelity	✓**	✓
Local fidelity	✓	✓
Auto Generate fidelity (Electromagnetics only)		✓
Size Preview	✓	✓*
Resolution. Available for all simulations except Electromagnetics. **Note:** For fluids, sharp-edge capturing needs to be turned off to view the resolution.	✓	
* Available for any local fidelity in the Refine stage.		
** Using multipliers greater than 1 for topology optimization solves is not recommended, as it could lead to excessive solve times.		

1.3.6. Optimization Functionality

The available options for optimization between **Explore** and **Refine** are as follows:

Functionality	Explore	Refine
Topology optimization	✓	
Defined variations	✓	✓

1.3.7. Monitoring Functionality

The differences between **Explore** and **Refine** for monitoring functionality are as follows.

Depending on your geometry selection and calculation type, you may see the following differences.

	Explore	Refine
Point monitors	All physics	Fluid flow, Structural
Line monitors	All physics	Fluid flow

	Explore	Refine
Surface bodies	All physics	
Plane monitors	All physics	Fluid flow
Detailed monitor charts	Time-dependent fluid flow, solid thermal, topology optimization, and some fluids variables for a steady-state fluid flow	Steady-state fluid flow

Note: All physics in the table above does not include Electromagnetics conditions.

Physics Variables

Functionality	Explore	Refine
Pressure, static	Fluid*	Fluid⁂
Pressure, total	Fluid*	Fluid⁂
Fan Pressure Rise	Fluid	Fluid
Velocity	Fluid*	Fluid⁂
Vortices Lambda 2	Fluid*	Fluid⁂
Force	Fluid*	Fluid⁂
Flow Uniformity	Fluid*	Fluid⁂
Wall Yplus	Fluid	Fluid
Temperature	Fluid thermal*, Solid thermal*, Electric thermal	Fluid thermal⁂, Solid thermal, Electric thermal
Heat flux	Solid thermal*, Electric thermal	Fluid thermal⁂, Solid thermal, Electric thermal
Heat flow	Solid thermal*, Electric thermal	Fluid thermal⁂, Solid thermal, Electric thermal
Pressure drop⁂	Fluid	Fluid
Volume	Fluid thermal*, Solid thermal*, Structural thermal	Fluid thermal*, Solid thermal*, Structural thermal
Volume flow	Fluid	Fluid
Mass	Fluid thermal*, Solid thermal*, Structural thermal	Fluid thermal*, Solid thermal*, Structural thermal
Mass flow	Fluid	Fluid
Factor of safety⁂	Structural	Structural
Displacement	Structural	Structural
Stress	Structural	Structural

Functionality	Explore	Refine
Von Mises stress	Structural	Structural
Principal stress	Structural	Structural
Strain (elastic)	Structural	Structural
Principal strain (elastic)	Structural	Structural
Force reaction	Structural, Random Vibration	Structural
Moment reaction	Structural, Random Vibration	
Contact reaction		Structural
Contact stress (pressure)	Structural	Structural
Contact stress (frictional)		Structural
Joint force		Structural
Joint moment		Structural
Bolt Axial Stress	Structural	Structural
Bolt Shear Force	Structural	Structural
Bolt Torsional Moment	Structural	Structural
Frequency	Modal, Free modal	Modal, Free modal, Pre-stress modal
Voltage, current density, Joule Heat	Electric	Electric
All structural result variables are also available in structural thermal, and all fluid result variables in fluid-thermal.		
* For time-dependent solutions, these also report per time point detailed monitors.		
⁑ For steady-state solutions, these also report per iteration detailed monitors.		
⁂ Created by default, these cannot be user defined.		

1.3.8. Results Display

The differences between **Explore** and **Refine** for displaying results are as follows. All structural result variables are also available in structural thermal, and all fluid result variables in fluid thermal.

Functionality	Explore	Refine
Contours	✓, Surface display priority: Inner, Outer, Highest Value or Lowest Value, Probe results	✓, Surface display priority: Outer, Highest Value or Lowest Value, Probe results
Deformation	Available only for surface contours of the displacement variable	✓* Available for iso-surfaces and vectors
Directional field	Available only for velocity magnitude, Probe results	✓⁑
Iso-surface	✓, Probe results	✓, Probe results

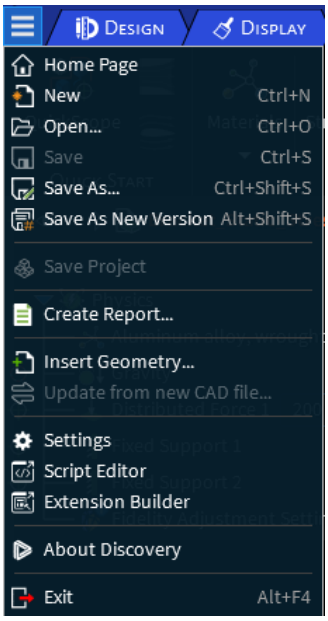
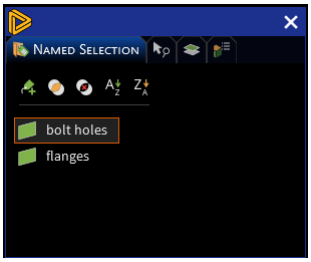
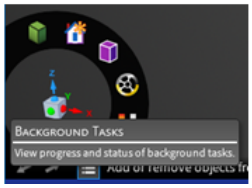
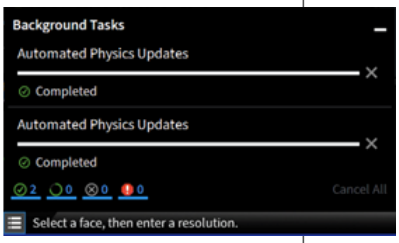
Functionality	Explore	Refine
Optimized body, add	✓	
Particle tracks	✓, Probe results	✓⁂
Reset	✓	
Streamlines	✓ ⁂, Probe results	✓⁂ ⁂, Ribbon
Vectors	✓, Probe results	✓⁂
Show mesh	Available for Fluid flow simulations	✓
Clip to user-defined range	✓	✓
Cut planes	✓	✓
Show Animation	✓	✓
Charts	✓	✓
* Available for all variables and plots except contours of highest value, lowest value and body plots. Not available for Fluids.		
⁂ Available for all components of any vector quantity in the direction of flow.		
⁂ In both **Explore** and **Refine**, streamlines and particles will always run from emitter to outlet using a single emitter. Multiple emitters can be added, up to 10 total.		

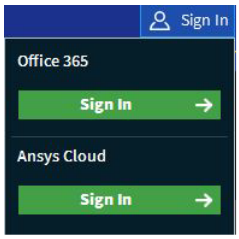
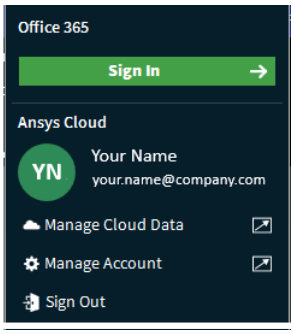
1.4. Terms to Know

Terminology specific to the Discovery application, what it looks like and where you'll find it.

Term Description	Control	Model	Explore	Refine
The simulation information display, or **SID**, shows a visual summary of current status, physics type, messages, and other information about your simulation. See The SID: A Visual Summary on page 44 for a detailed description.			✓	✓
The heads-up display, or **HUD**, puts tools in easy reach. Sub-tools and tool guides are on the left, and value inputs and options are on the right. Drag or to reposition and pin the HUD or click to return HUD to original position.		✓	✓	✓

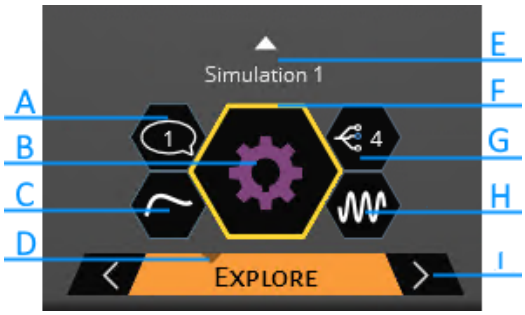
The **Results arc** provides different types of displays for viewing simulation results. Clicking the centered green **solve** circle will run a simulation on your model.			✓	✓
The **View arc** contains controls for altering the view of your design by changing the orientation angle or the appearance of solids by using different rendering or graphics style.		✓	✓	✓
The **stage navigator** switches between different stages by clicking the left or right arrows.		Design and modify the geometry.	Define a simulation and view instant results.	Define more complex physics and produce higher fidelity solutions.
A **halo** or **semi-halo** is a shortcut control to access common functionality found elsewhere in the interface.		✓	✓	✓
With **fidelity control**, you can increase the fidelity for a higher-precision solution, or decrease it for a quicker one.			✓	✓
Monitors are the most common numerical outputs for your simulation.			✓	✓

Manage your files and modify settings.		✓	✓	✓
Selection controls include access to **Named selection**, **Power selection**, **Properties**, parent/child selector, **Selection filter**, and **Revert selection**.	 	✓	✓	✓
The **status bar** provides instructions for using tools and hosts the **Background Tasks** controller.	  	✓	✓	✓
Press **F1** to open context-sensitive help in an interactive overlay, or click **Help** to open additional help options.		✓	✓	✓

Sign in includes access to Microsoft 365 and Ansys Cloud.	 	✓	✓	✓

1.5. The SID: A Visual Summary

The Simulation Information Display (SID) displays information about your simulation visually so you can quickly review. A cluster of controls centrally located at the base of the workspace, it includes:

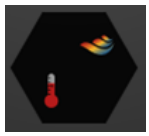


Label	Term	Description
A	Notification hex	Displays messages and notifications about your simulation.
B	Applied physics	Visual summary of the applied physics in your simulation. At a glance, you can see the type of physics used and whether multi-physics conditions are applied.
C	Calculation type hex	Displays the types of calculations in the simulation: • static or steady-state calculation: used to determine long-term behavior. • time-dependent calculation: used to follow the behavior over time. • frequency-response (FDTD): a time domain method used to solve for electric and magnetic fields as a function of time. • electrical conduction.
D	Fidelity control	Control the fidelity of your solution by sliding it to the right for higher precision, or to the left for a quick solution. Use the Spacebar on the keyboard to display and/or enter the fidelity value.
E	Simulation manager	Click to display options or rename the current simulation.
F	Current status indicator	Real-time information about the status and progress of your simulation. You can track the progress of the update by observing the color change in the SID outline.

Label	Term	Description
G	Defined variations hex	Click to display the defined variations table, with details such as dimensions and some physics conditions.
H	Modality hex	Displays a modal analysis simulation, used to model vibration characteristics.
I	Stage navigator	Switch back and forth from one stage to the next.

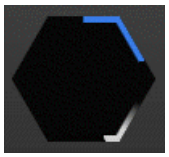
1.5.1. SID Applied Physics

The Simulation Information Display (SID) provides you with a summary of the applied physics in your simulation. At a glance, you can see the type of physics used and whether multi-physics conditions are applied. Here are some of the different physics conditions represented in the SID. Look at the physics tree to see details of the conditions that are used in this simulation.

	This is a structural simulation.		This is a fluid flow simulation.
	This is a structural simulation including effects of any temperature changes in the structural computation. These effects are induced by a change from the assembled temperature to a constant operating temperature.		This is a thermal fluid flow simulation. A temperature distribution is applied.
	This is a structural simulation with solid thermal conditions. The temperature distribution in solid bodies is computed along with the structural results.		This is a fluid flow simulation with solid thermal conditions applied to a face or body.
	This is a solid thermal simulation. A temperature distribution is computed in solid bodies due to the applied thermal conditions and the subsequent flow of heat.		This is a compressible flow simulation. The ideal gas law with variable density is used for all gaseous materials.
	This is an electromagnetics/electrical simulation.		This is an electrical simulation with solid thermal conditions applied to a face or body. The temperature distribution in solid bodies is computed along with the electrical results.

1.5.2. SID Progress Status

The Simulation Information Display (SID) provides you with real-time information about the status and progress of your simulation. You can track the progress of the update by observing the color change in the SID outline. Here are all the different states of the SID.

	Your simulation setup is incomplete or requires a higher Discovery application license.		Your model is being updated. Wait for the model update to finish before solving. You may also see this during file operations.
	Your model is being updated. Wait for the update to finish before completing your physics setup. You may also see this during file operations.		Your model is being updated. Wait for the update to finish before solving or making further revisions to the setup. You may also see this during file operations.
	Your simulation setup is incorrect, or something is missing. You cannot solve yet.		Your simulation is ready to solve.
	Your simulation results are out of date. Solve again.		The simulation is running. There are no results to display yet.
	The simulation is running; results are continually being updated.		The simulation is paused.
	The simulation is complete.		The simulation is running. In **Explore**, the results are continually being updated. In **Refine**, you can view your results when the update is complete.
	The simulation was interrupted.		The simulation is in the process of interrupting.
	A high-resolution result is being generated.		

1.6. Overview of Discovery Application User Help

The Discoveryapplication's user help provides you with the necessary information and assistance to effectively use the product. Help comes in various forms and formats, including in-product guidance, HTML, PDF, and more. The table below summarizes the different types of product assistance resources available in the application.

Help Resource	In-product	HTML	PDF
What's New	✓		
Interactive Tours on page 50	✓		
Interactive Help on page 48	✓		
Help Tab	✓		
Contextual Feedback on page 53	✓		
Release Notes		✓	✓
Discovery Application Online Help		✓	✓
Installation Guide		✓	✓
Migrating to the Ansys Discovery Application for Geometry Preparation			✓
Extensibility and Add-ins Developer Guide		✓	✓
3D Design Verification Manual		✓	✓
Technology Showcase: Example Problems		✓	✓
Beta Features Documentation		✓	✓

This section contains the following topics:

1.6.1. In-product Guidance

The application provides in-product guidance to assist you in your tasks. From broad, global settings to contextually relevant procedures, in-product guidance meets you where you are and delivers content for the task underway.

When you first launch the product, a **Welcome** window confirms some basic default selections. You can revisit this at any time on the **Learn** tab from **Home** page under **File Menu**.

Learn also contains:

- a **Getting Started** presentation to familiarize you with the different components in the product
- a **What's New** panel highlighting new features in each release
- a **Learn** page containing additional help resources

Content specific to technical topics or task completion includes:

- **Interactive Help** (also known as overlay help), which provides context-sensitive help and guidance
- **Interactive Tours**, which allow you to explore workflows interactively
- **Help** tab, which provides access to content available on the Innovation Space as well as several online resources

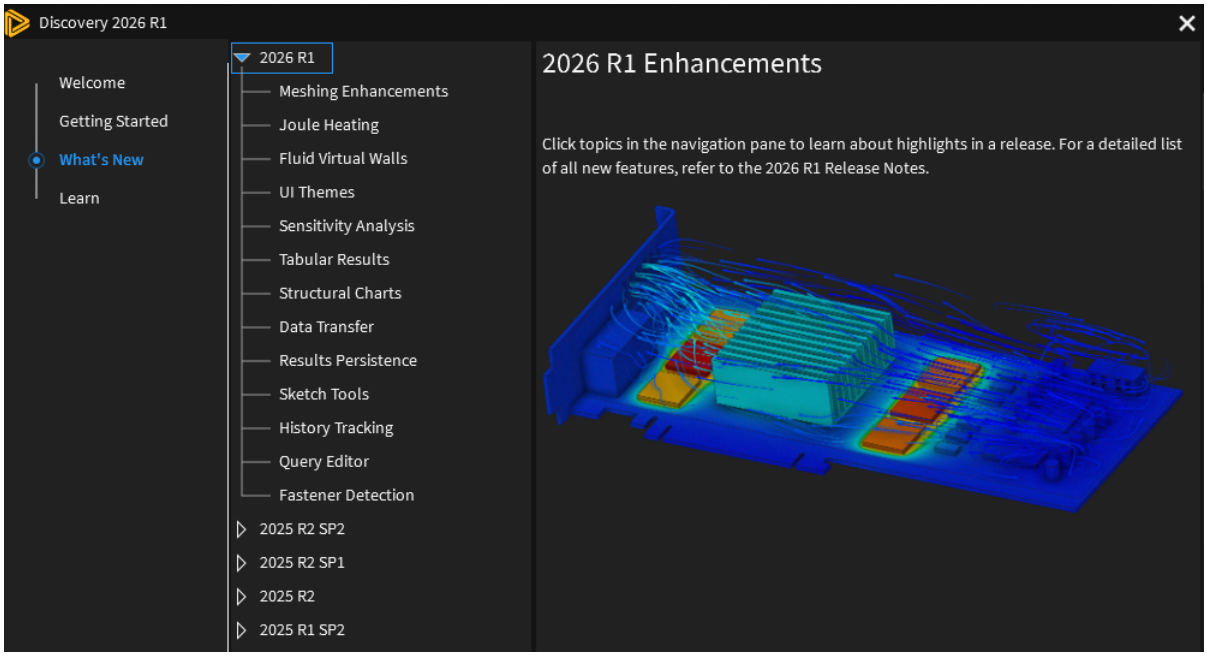
These features, along with Search, tooltips, status bar messages, and SID notifications, provide you with a comprehensive in-product guidance experience.

This section contains the following topics:

1.6.1.1. What's New

A **What's New** panel appears when you launch the Discovery application. Click **Learn** on the start screen, then **What's New**. The panel highlights some of the new features being introduced in any given release. See the **Release Notes** for a full list of new features for any given release.

You can re-visit the **What's New** panel anytime by navigating to **File menu** > **Home Page** > **What's New**.



1.6.1.2. AnsysGPT™

In the **Help** tab, click **Innovation Space** to open a window in your application where you can access several resources that will help you find answers to your questions. With an **AnsysGPT AI+** license increment, you can use **AnsysGPT™**, which will take your query and produce answers based on expert knowledge of the product. It draws upon information from knowledge articles, courses, forums, and other online help sources.

For more information about this feature, see **Ansys Engineering Copilot**.

1.6.1.3. Interactive Help

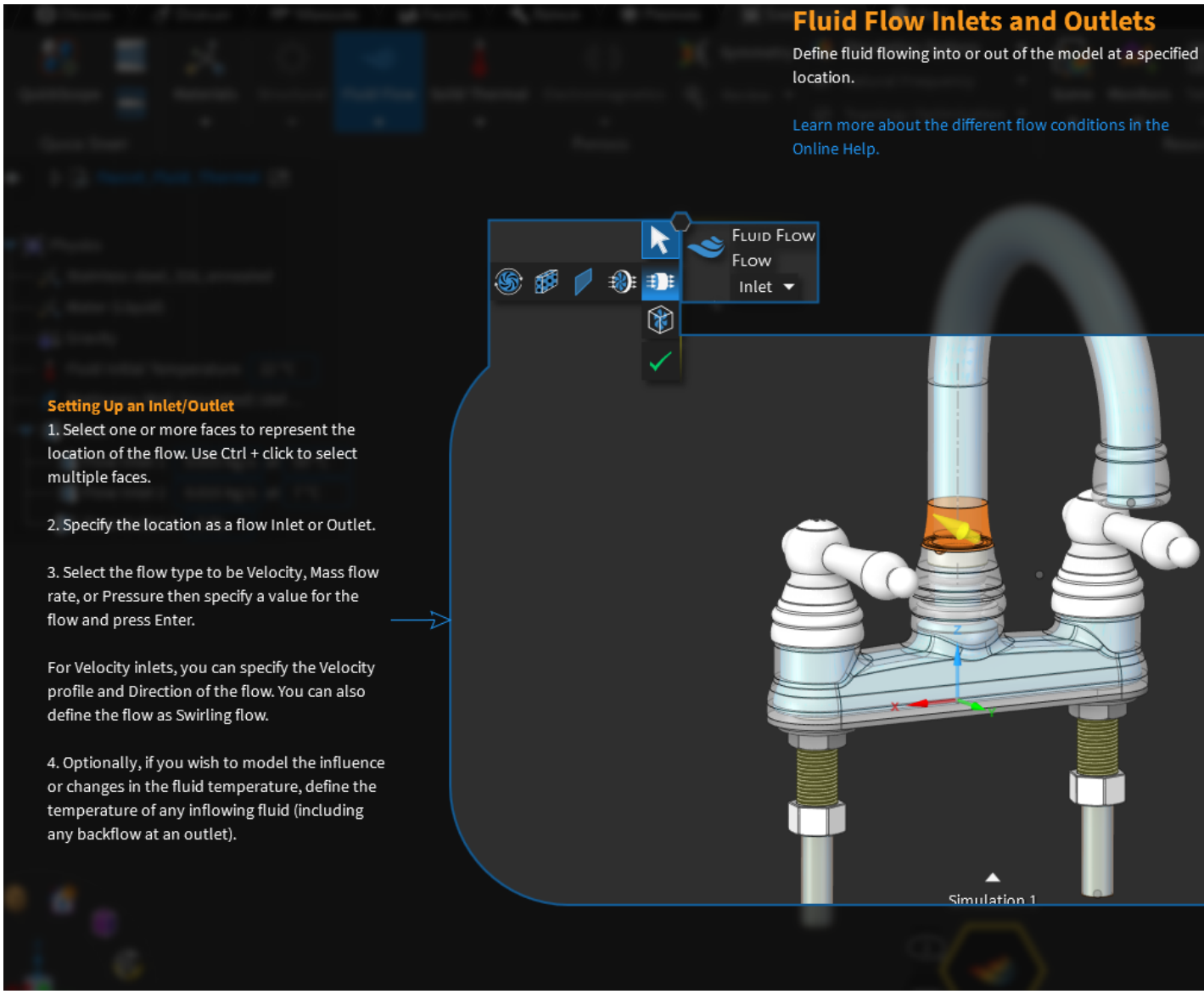
Discovery Help provides in-product contextual help and guidance, also known as overlay help. To access context-sensitive help, select **Help** located in the top right-hand corner of the Discovery application window.

To learn about a UI element, press F1 while hovering over it. For guidance on using your active tool, press F1 while in any empty space, or click **Help**. You can interact with the model and UI elements while the **Help** provides contextual guidance.

While in the overlay help, use the navigation buttons  to navigate between topics and select the corresponding UI elements. To return to the home page, click  and to exit Help, click  .

> **Note:** You can use F1 as a shortcut key to enter and exit **Help**. Pressing the Esc key will exit the current tool as well as the Help.

The interactive help contains links to other topics in the product or links to richer content in the online help.



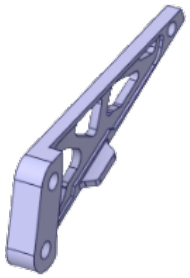
You can explore a variety of workflows interactively by following the steps outlined in Discovery's Interactive Tours.

1.6.1.4. Interactive Tours

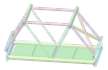
The Discovery application's user experience is a feature-driven intuitive interface that delivers relevant cues and guidance during the simulation workflow to avoid or resolve difficulties. Feature-focused, interactive tours invite you to explore the various stages and different physics types using relevant sample projects.

You can access the interactive tours anytime by navigating to **File menu** > **Home Page** > **Interactive Tours**.

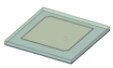
> **Note:** Interactive tours may exhibit performance and functional limitations depending on your licensing.

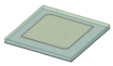
	The **Getting Started** tour showcases universal elements, user settings, tools, and shortcuts, and offers tips and best practices. The tour explores physics setup, simulation and results monitoring, and display. It is available for all license levels.

Model Stage Tours

Concept	**Functionality**	**Part Design** 	**Model Preparation** 	**Beams and Shells** 
Modeling	Fix geometry		✓	
	Extract interior volume		✓	
	Sketch	✓		
	Add constraints	✓		
	Add dimensions	✓		
	Extract beams and shells			✓
	Extend surfaces to nearby parts			✓
	Assign beam profiles			✓
Selecting	Select multiple objects	✓		✓
	Find similar objects (Power Select)	✓		
	Pull multiple objects	✓		
Transfer	To Ansys Workbench		✓	

Explore and Refine Stage Tours

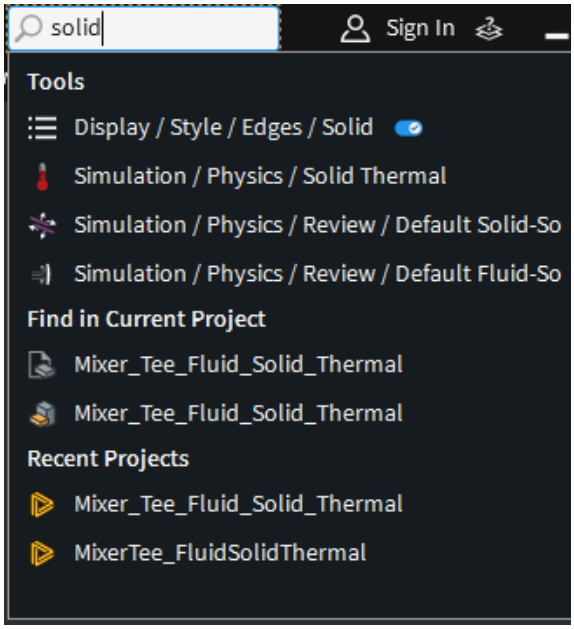
Concept	Functionality	Antenna Design 	Static Structural 	Fluid Flow 	Solid Thermal 	Topology Optimizing 	Contact Review 	Fluid-Solid Heat Transfer 
Physics type				 *steady-state*			 *natural frequency*	 
Stage		***Refine***	***Explore***	***Explore, Refine***	***Explore***	***Explore***	***Explore***	***Explore, Refine***
Modeling	Change geometry		✓		✓	✓		
	Extract interior volume			✓				
	Add dimensions				✓			
	Smooth faceted body					✓		
Selecting	Create named selections				✓			
	Edit objects in the tree			✓	✓	✓	✓	
	Select multiple objects	✓	✓		✓	✓	✓	
	Find similar objects (Power Select)				✓			
	Pull multiple objects				✓			
Physics	Assign/change materials	✓		✓	✓			
	Create multiple simulations				✓	✓		
	Perform higher fidelity solution	✓		✓				✓
	Work with contact pairs						✓	
Physics conditions	Define/modify flow conditions			✓				✓
	Define/modify structural constraints		✓		✓	✓	✓	
	Define thermal conditions			✓	✓			✓
	Define electromagnetics conditions	✓						

Concept	Functionality	Antenna Design 	Static Structural 	Fluid Flow 	Solid Thermal 	Topology Optimizing 	Contact Review 	Fluid-Solid Heat Transfer 
Optimizing	Optimize topology					✓		
	Optimize with solution fidelity						✓	
	Setup charts	✓						
	Define variations				✓			
Results	Include deformation		✓			✓		
	Display contours		✓	✓	✓	✓	✓	✓
	Display vectors			✓				✓
	Display streamlines			✓				
	Display far-field results	✓						
	Display near-field results	✓						
	Display mesh	✓						
	Include cut plane	✓		✓				
	Monitor results	✓		✓	✓		✓	✓
Transfer	To HFSS	✓						

1.6.1.5. Search

Search for tools in the active application, objects in the current project, and project files in your recent projects list. Start typing in the search box to see all the possible tools, objects, and projects that match the name in the search box.

- Search results display a path that lists the tab and group of the tool. Select the item in the search results list. This will activate the tool and automatically switch to the corresponding tab of that tool.
- Search for any object in the current project. Selecting the item in the search results list highlights the item in the tree and in the model.
- Search for a project by entering its name. Any projects listed under your recent projects containing any part of the name search will appear in the list. Selecting the project will load the file.



1.6.1.6. Contextual Feedback

The Discovery application provides you with many contextual cues to help you complete your task. With features such as tooltips, a status bar, and a simulation information display, you can easily access detailed information about your model and simulation to make informed decisions about your next steps.

Tooltips

Tooltips not only provide you with information about the user interface element as your cursor hovers over it, but it can also change depending on the state of your model. Tooltips in the ribbon typically contain more detailed information than tooltips that appear for elements in the graphics window.

To turn off the display of tooltips, go to **File Menu > Settings > General > User Interface** and click **Show overlay tooltips**.

Status Bar

The status bar provides useful information and guidance about your model and simulation. The left side of the status bar may display operational messages, guidance on your next action when actively in a tool, and in some cases the estimated time remaining for the simulation to solve. The right side of the status bar displays the type of geometry and number of objects selected in your model. When solving your simulation using a GPU solver, you can also see the GPU memory usage. The information in the status bar changes as you interact with your model.

Simulation Information Display (SID)

The SID shows the state and setup of the current simulation. Applied physics, defined variations, notification messages, calculation type, and much more are included in the SID.

The middle portion of the SID provides a visual summary of the physics used in your simulation. Around the SID's border is a progress status indicator, providing real-time information about the status and progress of your simulation. On the left side of the SID, the notification hex displays messages and notifications about your simulation. It will also guide you to successfully complete your tasks. The left side also shows the calculation type hex where you can see if your simulation is steady-state, time-dependent, or otherwise. The right side of the SID shows the defined

variations hex with information about physics and geometric parameters. The modality hex is also on the right, and it shows a modal analysis simulation.

1.6.2. Online Resources

The **Help** tab provides links to many online resources. You can access the following resources:

- Online Courses
- Ansys Learning Hub (subscription only)
- Ansys Education Resources
- Online Help
- **3D Design Verification Manual**
- **Technology Showcase: Example Problems**

This section contains the following topics:

1.6.2.1. Accessing Online Help

There are a few easy ways to find the online help quickly:

- Access it from within the application:
  - Launch the Discovery application and navigate to **File Menu > Home Page > Learn > Learn**. Click **Online Help**.
  - Launch the Discovery application and press **F1** or **Help** on the top right of the application window. In the Discovery Application Help description, click the Online Help link. You can also use the topic tree on the top right of the application window and select Online Help.
  - When you need specific help about an active tool in the Discovery application, press **F1**. The in-product help topic will contain a link to the online help where you can obtain richer content relevant to your task.
- Pin the Ansys Help to your Start menu or taskbar. From the **Start** menu, expand the Ansys folder and right-click **Ansys Help**. Select **Pin to Start** or **More > Pin to taskbar**.

1.6.2.2. Installing the Local Documentation Package

Viewing the online help on the Internet is recommended, as the help content on the Internet is routinely updated. If you do not have Internet access, or if you would like a local copy of the online help on your system, you can download an installable version of our product documentation from the Ansys Download Center.

1. Sign in to your account. If you don't have an account, you are prompted to create one.
2. From the list of **Downloads** on the left, click **Product Documentation**.
3. Under **Installable Help**, click **Local Help Windows** to begin the download.
4. From the Windows File Explorer **Downloads** folder, extract `ANSYSLOCALHELP_<release>_WINX64.zip` to a convenient location on your computer.
5. In the folder of extracted files, double-click `setup.exe`.
6. When the installer appears, choose **I agree to the terms** and click **Next**.
7. Confirm the installation directory and click **Next**.
8. Confirm the selected platforms and products (in this case, **Ansys, Inc. Documentation Only**) and click Next.
9. When the installation is complete, click **Exit**.
10. From the Windows **Start** menu, select **Ansys Help <release>** to launch the Ansys Help Viewer.

11. (Optional) To enable help calls directly from Discovery application products to the installed local documentation, select **Ansys Help Configuration <release>** from the Windows **Start** menu. Then specify the path to the installed help files and click **Save**.

You may need to restart your machine for changes to take effect.

1.6.3. Did You Know?

This section contains the following topics:

1.6.3.1. Running the Discovery Application from the Command Line

The Discovery application supports command-line arguments that you can include when launching the application from a command line or terminal prompt. All command-line arguments are case-insensitive.

Command-line syntax:

```
Discovery.exe <options>
```

> **Tip:** It is good practice to put quotation marks around the path to the solver executable and around the full path to the project. This ensures that spaces in the path or project are not an issue.

The following command-line arguments are available.

Option	Description	Value
`--exit-after-script`	Specifies the application should exit when script (defined by the `--run-script` option) is finished.	
`-h, --help`	Shows usage information.	
`--hidden`	Hides main window at start-up.	
`-l, --locale`	Specifies the country code (for example, `fr` for French) of the language to be used in the UI.	Country codes: Chinese Simplified - `zh-Hans` Chinese Traditional - `zh-Hant` English - `en` French - `fr` German - `de` Japanese - `ja` Example: `-l de` to use German in the UI.
`--mipassword`	The password used to connect to a Granta MI materials database.	String
`--miport`	The port used to connect to a Granta MI materials database.	Integer
`--miusername`	The username used to connect to a Granta MI materials database.	String

Option	Description	Value
`--no-addins`	Prevents loading of addins.	
`-o, --open`	Project to open at start-up.	File path
`-p, --license-prefs`	The preferred license to use.	String
`--run-script`	Script file to run.	File path
`--script-api`	Script API version when used with the `--run-script` option.	Integer
`--script-args`	Script arguments as a semicolon-separated list.	Example: `--script-args "arg1;arg2;arg3"`
`--script-async`	Boolean value indicating whether to run script asynchronously.	
`--script-throw`	Throws an error on exception while running the script file.	
`--skip-hardware-compatibility-checks`	Skips all hardware compatibility checks. Use with caution: disabling checks may cause the application to become unstable or crash.	
`--temp`	Overrides the temporary directory used by the application.	Directory

1.6.3.2. Shortcuts

You can create custom keyboard shortcuts for tools in the ribbon, view arc, results arc, and the sketch grid tool cluster in the graphics window. Although some tools already have predefined shortcuts, others do not, and you can customize your experience by changing or adding shortcuts.

Shortcut settings and the **Shortcut Buddy** are found in **File menu** > **Settings** > **Customize** > **Shortcuts**.

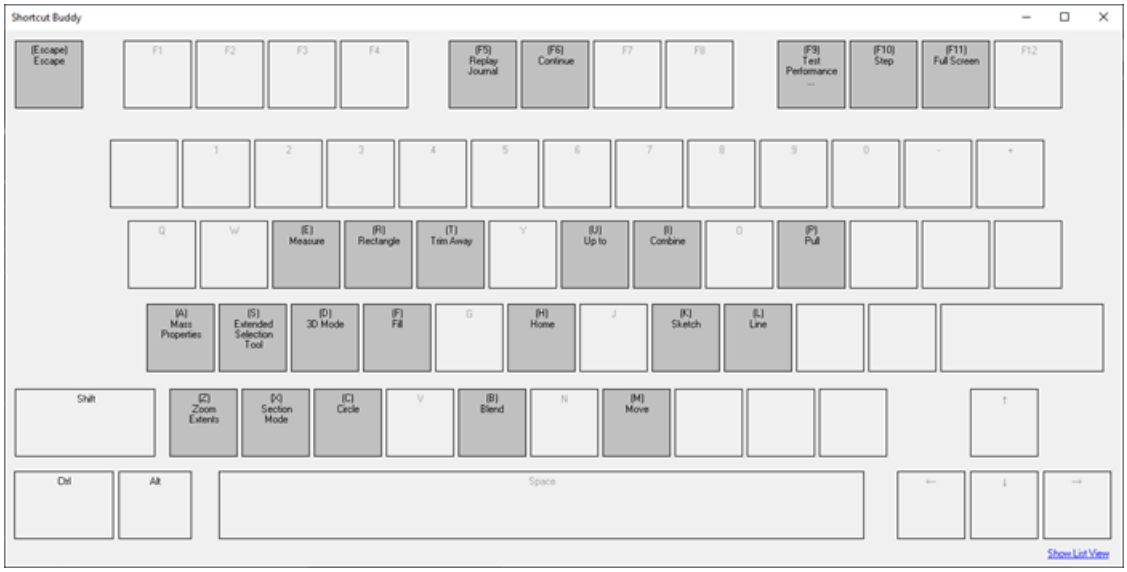
Predefined Shortcuts

There are many predefined shortcuts in the Discovery application. A reference of those predefined mappings is available under **Shortcut Buddy** in the settings panel and can be viewed either as list or keyboard mapping.

List View



Keyboard View



1.6.3.2.1. Creating Custom Keyboard Shortcuts

The Discovery application allows you to create custom keyboard shortcuts for tools in the ribbon, view arc, results arc, and the sketch grid tool cluster in the graphics window. Some tools already have a predefined shortcut, such as **Pull (P)**, while others like **Split Body** do not.

To create a custom shortcut:

1. Go to **File menu** > **Settings** > **Customize** > **Shortcuts**.

2. Click **Add Shortcut**.
3. Select the tool for which you want to create a shortcut from the **Command** dropdown menu.

   If the tool you select has no predefined shortcut, then the application will display **No shortcut** in the **Current shortcut** field. If the tool you select already has a shortcut, then **Current shortcut** will list the currently defined shortcut.
4. Specify a single character (number, letter, and most other keyboard strokes) in the **Enter new shortcut** field to represent the tool. You can include modifiers, such as **Alt**, **Ctrl**, **Shift**.

   > **Note:** **Alt+Ctrl** is the only modifier combination that is not supported.

   > **Note:** You cannot use Windows, **Tab**, **Esc** for shortcut commands.

   If the value you entered is used by another tool, the tool will be listed next to the **Used by** field.
5. Click **Assign** to create the custom shortcut.

   If you assigned shortcuts that were previously defined for other tools, you will see a message asking you to confirm your actions.

   A table is populated with all the custom shortcuts you create. You can continue to add more shortcuts, modify a shortcut in the table using **Edit Shortcut** and remove any shortcut from the table using **Delete Shortcut**.

   Your custom shortcuts will be saved in the application and available for use in new sessions.

   > **Note:** To obtain a list of available shortcuts in the application, click **Shortcut Buddy**. A table listing all shortcuts is displayed. At the bottom of the table, you can click **Show Keyboard View** to view the shortcuts in a different format.

1.6.3.3. User Tips

Short and informative tips are displayed on the bottom of the launch splash screen. These tips change randomly every time you launch and cover a wide range of topics.

2: File Operations

The Discovery application provides options for you to manage your files and settings. The sections that follow describe common file operations and how to customize your settings.

This section contains the following topics:

2.1. Saving Projects

You can save a new file or an existing project. Click **File > Save** to save your work, or click **File > Save As New Version** to save your work as a new file with a new version number.

Click **File > Save As** to save your work under a different file name or type, or to another location, the **Save As** dialog box appears.

- Your current directory is shown in the file path breadcrumb. You can jump directly to a parent directory by clicking one of the breadcrumb's path segments, even when browsing cloud locations. If a breadcrumb is too long, scroll arrows appear to let you scroll and see the hidden parts of the path. You can also use the mouse wheel.
- You can see the recently accessed file under **Recent Files**. This shows when the file was last opened, the path, and the file size.
- You can see who last saved the file and when, and the file size under **Sites**.
- You can see who shared the file and when, and the file size under **Shared With Me**.
- Right-click a recent file and select the **Open file location** button to navigate directly to the recent file's directory.
- You can see files by date or alphabetically. Files can be also viewed as tiles and lists.
- To search for a particular file start typing in the **Search** box. Stop typing briefly to see the tiles update.
- Exporting to SimAI is not available if a solve is paused in the Explore stage. The SimAI option generates a zip containing the `.vtp`, `.vtu`, and `.json` files. The `.json` file contains all the numeric meta data for boundary conditions and materials.

2.2. Creating and Opening Projects

You can create a new file or open an already existing project. Click **File > New** to create a new project.

Click **File > Open**. In the **Open** dialog box, view the applicable import settings for the file type. Specify the location and **File name** and click **Open**.

2.3. Transfer Discovery Application Data to Another Application

You can transfer Discovery application data to another Ansys application. For example, if you are a designer using the Discovery application, you may want to export your simulation and send it to an analyst for further analysis in the Mechanical application, the Fluent application, High-Frequency Structure Simulation (HFSS) application or Icepak application. You can set options that affect file import and export in **File Menu > Settings > File**.

To transfer Discovery application data to another Ansys application, go to the **Simulation** tab and select the one of the products in the **Transfer** group of the ribbon. Depending on the physics being simulated in the Discovery application file, the applicable product will be launched.

- If structural or solid thermal conditions exist, the Mechanical application will be launched with `.dscodat` files. This file is saved in the Discovery application's Projects folder (for example MechanicalExportFile.dscodat) or in Project directory (for example c:\users\username\Ansys Inc\Documents\Discovery Projects).
- If fluid or fluid-thermal conditions exist, the Fluent application will be launched with `.cas.h5` files. **Export to Fluent** is disabled for Fluid simulations that contain faceted bodies. When the **Export journal file** option is selected, a journal file is saved along with mesh and settings file and can be later imported in the Fluent application.
- If fluid, thermal or fluid-thermal, fluid-thermal with electric conduction conditions exist, the Icepak application will be launched with `.aedti` files.
- If electromagnetics conditions exist, the HFSS application will be launched with `.aedti` files.

You can also transfer your data without launching the Mechanical application, the Fluent application, HFSS application, or Icepak application. Select **Save As** from the **File** menu, and specify a file type. Select **Discovery Export Files** to save as `.dscodat` file or **Fluent CAS File** to save as `.cas.h5` file. Select **Electronics Desktop Import** to save as `.aedti` file. You can also select **Touchstone 2 Format** to save as `.ts` file, however export to Touchstone is only allowed for electromagnetics simulations with results. If you only select **Save As**, the file is saved as a `.dsco` file.

> **Note:** The Workbench application supports a smart update associativity transfer, which gives you the ability to link your Discovery model to the Mechanical application. Analyses, connections and physics conditions are transferred, which are then updated between Discovery and the Mechanical application. You can specify what data is exported in the **File** > **Workbench** tab of the **Settings** panel. This transfer is tightly tied with the Geometry Transfer. For more information on the smart update associativity transfer, see **Discovery**.

> **Note:** Geometry with large units and physics conditions, may not be transferred when attempting to transfer the Discovery application projects to the Mechanical application, the Fluent application or HFSS application.

2.3.1. Discovery Application File Transfer Considerations

There are differences between the Discovery application and other Ansys applications in areas such as model management, material properties, solvers, results, and so on. Awareness of these differences and their potential impact on data transfer enables you to transfer Discovery application files to other Ansys applications efficiently.

Manual Verification

After transferring data from the Discovery application to another Ansys application, you must perform manual verification to ensure the exported file is as you expected. When you transfer data from the Discovery application to the Mechanical application, you may also refer to the error log for information about data that did not transfer correctly.

Blocked and Unsupported Transfers

You need to be aware of these blocked and unsupported transfers:

- Transfer is available in both the **Explore** and **Refine** stages. However, transfer is blocked until the physics of the simulation can be determined. For example, at least one structural or one thermal physics condition must be defined for transfer to the Mechanical application to be possible.
- When installing the Discovery application as a component of another Ansys product package, you may choose between a *full* or *reduced* installation. A full installation provides full transfer to flagship capabilities, but a reduced installation provides limited transfer to flagship capabilities.
- Transfer is blocked unless the required license is present.

- Transfer is blocked when the Discovery application is operating in read-only mode. For details about read-only mode, see **Unlicensed and Underlicensed Behaviors**.
- If data does not transfer correctly, you may find more information about the failure in the error log. For example, this logging may occur if a physics condition fails to transfer. For transfers to the Mechanical application, see the ACT Log for details (**Extensions** > **View Log File**).
- User-defined names for physics conditions and material assignments in the Discovery application are not transferred to the Fluent application. Instead, they appear as default names. For example, in the Fluent application, they appear as flow_*n*, wall_*n*, material_*n*, and so on.
- Multiple Discovery application simulations are not supported for transfer in Mechanical application and the Fluent application. Only the current Discovery application session is transferred.
- No data related to parameters or defined variations is transferred.
- No mesh sizing information is transferred to either the Fluent application or the HFSS application.
- No results data is transferred.
- The External Flow tool doesn't require bodies to be subtracted from the Enclosure tool. Hence, when any external flow simulation modeling obstruction to flow is transferred to the Fluent application, the exported model does not contain the obstruction to flow features. However, if a fluid volume is created using the Enclosure tool, the obstruction flow features will be transferred to the Fluent application.
- Time-dependent fluid flow simulations, which are available only in the **Explore** stage, must be reset from steady state to time-dependent after they are transferred to the Fluent application.
- If you transfer a file to the Fluent application from the **Explore** stage, any solution fidelity refinements you set in the **Explore** stage are ignored. To be transferred to the Fluent application correctly, solution fidelity refinements must be set in the **Refine** stage before transferring the file from **Explore**.
- If you transfer a file to the Fluent application with internal radiation settings, a view factor file will not be included in the transfer. You must read or compute a view factor file in the Fluent application.
- When exporting a model comprising many faces with assigned monitors and results to the Fluent application, you may experience significant performance issues. The export process can be very slow or even stall.
- The Mechanical application will have a read-only mesh as part of the import. The mesh is transferred as read-only, but it can be modified in the Mechanical application to allow changes to the mesh and the transferred mesh sizes.
- Fluid-solid (conjugate) heat transfer simulations will not transfer the fluid or fluid thermal conditions when transferring to the Mechanical application. Instead, the thermal and/or static pressure result data is transferred as an imported load depending on the chosen physics transfer. For the result data to be included in the transfer, simulation results must be available, even if paused or stalled. For the **Refine** stage, only simulations solved using LiveGX can transfer the result data. Furthermore, the result data will only be exported for transfers from the same stage in which the simulation is solved.
- With a GLTF (*.*glb*) export, bolts will not be exported.

Other Considerations

Other information and behaviors that you need to be aware of include:

- Your version of the Discovery application and the version of the Ansys application that you want to transfer to must be the same (for example, both applications must be version 2026 R1).
- In the Discovery application, the moment condition is defined in relation to the local coordinate system. However, to transfer moment, the Discovery application converts and transfers it relative to the global coordinate system.
- Physics conditions that are suppressed in a Discovery application simulation are not transferred to the Mechanical application.
- For details about importing a Discovery application file into the Mechanical application, see **External Study Import**.

Smart Update Considerations

Considerations for the Workbench application's smart update associativity transfer include all other considerations, as well as:

- No mesh is transferred, even in the Refine stage.
- User-defined names for physics conditions are transferred to the Mechanical application.
- Multiple geometries imports are not supported. For example, geometry imports inside the Mechanical application imported on top of the Discovery application geometry will fail.

2.4. Supported File Types for Import and Export

You can open and insert files from other modeling applications, and you can save your designs into different file formats. If you insert a model while another is open, the material data from the second model is not imported.

For more detailed information than what is provided here, see the **CAD Integration** section of the product help or the Platform Support section of the Ansys Website, where you can view the CAD Support PDF for your release.

Import Support

Multiple files can be selected when you import a geometry. The application can import the file formats that appear in the table below.

File Type	File Extension	Reader / Plug-In	Support Notes
SpaceClaim	*.scdoc, *.scdocx	N/A	All versions
ACIS	*.sat, *.sab	Reader	Up to 2023
Ansys	*.agdb, *.pmdb, *.meshdat, *.mechdat	N/A	*.agdb up to 16, *.pmdb
Ansys Electronics Database	*.def	N/A	Supported
Ansys Stride files	*.stride	N/A	All versions
AutoCAD	*.dwg, *.dxf	Reader	Up to 2024
AutoCAD	*.dwg, *.dxf	Plug-In	2023, 2024
CATIA V4	*.model, *.exp	Reader	Up to 4.2.5
CATIA V5	*.CATPart, *.CATProduct, *.cgr	Reader	Up to 2025
CATIA V5 (CADNexus CAPRI CAE Gateway V4.20.0)	*.CATPart, *.CATProduct	Reader	2023, 2024,2025
CATIA V6	*.3dxml	Reader	Up to 2024x
CREO Elements / Direct Modeling	*.pkg, *.bdl, *.ses, *.sda, *.sdp, *.sdac, *.sdpc	Plug-In	20.6, 20.7
CREO Parametric	*.prt, *.xpr, *.asm, *.xas	Reader	Up to 12.0
CREO Parametric	*.prt, *.xpr, *.asm, *.xas	Plug-In	9.0, 10.0, 11.0
DesignSpark Mechanical	*.rsdoc, *.rsdocx	Reader	Supported
ECAD	*.idf, *.idb, *.emn	N/A	Up to 4.0
Fluent Mesh	*.tgf, *.msh	N/A	Supported
Fusion	*.f3d, *.f3z	Plug-In	2.0.15995 onward
IGES	*.igs, *.iges	Reader	Up to 5.3
Inventor	*.ipt, *.iam	Reader	Up to 2026

File Type	File Extension	Reader / Plug-In	Support Notes
Inventor	*.ipt, *.iam	Plug-In	2024, 2025
JT Open	*.jt	Reader	Up to 10.10
Monte Carlo N-Particle	*.mcnp	Reader	2024.2
NX	*.prt	Reader	Up to 2506
NX	*.prt	Plug-In	2406, 2412, 2506
OBJ	*.obj	N/A	Supported
OpenVDB	*.vdb	Reader	VDB 9+
OSDM	*.pkg, *.bdl, *.ses, *.sda, *.sdp, *.sdac, *.sdpc	Plug-In	20.4, 20.5
Other ECAD	*.anf, *.tgz, *.xml, *.cvg, *.gds, *.sf, *.strm	N/A	Supported
Parasolid	*.x_t, *.xmt_txt, *.x_b, *.xmt_bin	Reader	Up to 36.0
PDF Facets	*.pdf	N/A	Supported
PLY	*.ply	N/A	1.0
PNG	*.png	Reader	Supported
Point curve (text)	*.txt	N/A	Supported
Rhino	*.3dm	Reader	Up to 8
SketchUp	*.skp	Reader	Up to 2021
Solid Edge	*.par, *.psm, *.asm	Reader	Up to 2025
Solid Edge	*.par, *.psm, *.asm	Plug-In	2022, 2023
SOLIDWORKS	*.sldprt, *.sldasm	Reader	Up to 2025
SOLIDWORKS	*.sldprt, *.sldasm	Plug-In	2020, 2021, 2022
STEP	*.stp, *.step	Reader	Up to AP242
STL	*.stl	N/A	Supported
VRML	*.wrl	Reader	Supported

Export Support

The application can export to the file formats that appear in the table below.

File Type	File Extension	Support Notes
Discovery Export Files	*.dscodat	Supported
ANSYS Neutral Files	*.anf	Supported
ANSYS Stride Files	*.stride	Supported
ANSYS Part Manager Files	*.pmdb	Supported
ACIS	*.sat, *.sab	5.0

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

File Type	File Extension	Support Notes
AutoCAD	*.dwg, *.dxf	Up to 2018
Electronics Desktop Import File	*.aedti	Supported
Fluent CAS File	*.cas.h5	Supported
Fluent Journal File	*.jou	Supported
Fluent Meshing faceted geometry	*.tgf	Supported
FM Database	*.fmd	Supported
GLTF	*.glb	Supported
Icepak Project	*.icepakmodel	Supported
IGES	*.igs, *.iges	Up to 5.3
JT Open Facets	*.jt	Up to 10.10
JT Open Geometry	*.jt	Up to 10.10
MCNP	*.mcnp	Supported
OBJ	*.obj	Supported
Parasolid	*.x_t, *.xmt_txt, *.x_b, *.xmt_bin	Up to 36
PDF2D Ansys Engineering Notebook	*.pdf	Supported
PLY	*.ply	1.0
PNG	*.png	Supported
Rhino	*.3dm	Up to 7.0
STEP	*.stp, *.step	Up to 242
STL	*.stl	All
Touchstone 2 Format	*.ts	Supported
VRML	*.wrl	Supported
Visualization Toolkit File (VTK)	*.*.vtp, *.vtu	Supported

2.5. Importing an Autodesk Fusion Simulation Project

You can import an Autodesk Fusion Simulation project into the Discovery application. The import includes all geometry and simulation settings from Autodesk Fusion Simulation studies. When a Fusion design study is imported, the Discovery application creates a new simulation for each load case, including all loads, constraints, and materials for the load case. The design can then be further simulated and modified from within the Discovery application.

You can either import a Fusion Simulation data file (*.sdz) or launch the import directly from Fusion. Both methods require a separately installed Discovery application Addin, which can be downloaded and installed from the Ansys Discovery Downloads page on the Customer Portal. For information on how to export SDZ files or directly launch the Discovery application from Fusion, refer to the following Autodesk Help pages:

- Create a Fusion Simulation data file for import into Ansys
- Export a created design study directly into Ansys

To import an SDZ file into the Discovery application, select **File** > **Import** > **Import Autodesk Simulation File** and choose the SDZ file.

The following sections list supported and unsupported features and concerns when an Autodesk Fusion simulation is imported into the Discovery application.

Supported Study and Constraint Types

The Discovery application supports the import of the following study types from Fusion Simulation:

- Generative Design
- Linear Stress
- Modal
- Thermal
- Thermal Stress

The Discovery application will import almost all Geometry, Loads, Constraints, and Materials from the Fusion Simulation Study. Unsupported features are listed below.

Unsupported Study Types

The following study types are not supported during the import from Fusion Simulation:

- Buckling
- Nonlinear
- Explicit
- Shape Optimization

> **Note:** The Ansys Mechanical application does support these simulation study types and can be used instead.

Additional Limitations

The following are known limitations that result from simulation capability differences between the Discovery application and Fusion Simulation:

- Hydrostatic pressure loads are not currently supported. These are created as 0 pressure loads with a warning.
- Angular Global Acceleration and Angular Velocity loads are not currently supported. These are ignored with a warning.
- Edge-based contact is not currently supported. Edge contact locations are ignored with a warning.
- Thermal Radiation loads are not currently supported in the Discovery application. These are ignored with a warning.
- Bolt Connector pre-stress and idealization are handled significantly different in the Discovery application and Fusion. Bolt Connector objects are created in the Discovery application, but the location is approximate and will not produce valid simulation results. You must recreate explicit bolt geometry or delete the bolt objects to proceed with simulation. Appropriate warning messages are given.
- The Discovery application only supports isotropic, constant property material definitions. Any temperature dependent, non-linear or hyper-elastic material definitions are ignored. Warning messages are given.

2.6. Generating a Project Report

Create a project report (web page or Word document) showing details about your project. Project reports include, but are not limited to:

- project summary
- geometry elements
- simulation runs with associated
  - materials
  - physics conditions
  - connections
  - defined variations
  - monitor results
  - saved scenes results
  - notes

To create a report, select **Create Report...** in the **File** dropdown to open the **Save Report As** dialog box. Type a name in the **File name** field, specify the location and format (web page or Word document) and click **Save**.

A report is written to the specified file and opens automatically in either your default web browser or Microsoft Word.

2.6.1. Adding Notes

You can add notes to most objects in the model and physics trees, as well as to the monitors list. Notes support plain text and hyperlinks, and are included in the project report.

- Add note: Right-click a tree object to open the context menu and select **Add Note**.
- Delete note: Right-click to open the context menu and select Delete Note or click the delete icon in the note.

The noted object is outlined and a yellow corner triangle denotes the presence of the note.

2.7. Updating Geometry From A New CAD File

You can update your geometry if it has been modified in another CAD system.

Any previously assigned physics conditions will attempt to be automatically reassigned after replacement if **Auto-Update Assignment** is enabled. Review the simulation setup to ensure the conditions are appropriate before proceeding.

To update the geometry from a new CAD file, click **File > Update From New CAD File...**. In the **Open** dialog box, specify the location. Type a name in the **File name** field and click **Open**.

The geometry is updated with the selected file.

2.8. Geometry Transfer

The **Geometry Transfer** group in the **Prepare** tab contains tools for transferring geometry to Ansys applications. The following information can be transferred: named selections, materials and driving dimensions. Transfer tools give you the ability to transfer the geometry model to flagship products, with parametric updates. So, rather than only a one-way export, the model is parametrically updated between the Discovery application and the flagship products.

> **Note:** The Workbench application supports a smart update associativity transfer, which gives you the ability to link your Discovery application model to the Mechanical application. Analyses, connections and physics conditions are transferred, which are then updated between the Discovery application and the Mechanical

application. You can specify what data is exported in the **File** > **Workbench** tab of the **Settings** panel. For more information on the smart update associativity transfer, see **Discovery**.

When a geometry is transferred from the Discovery application to the Workbench application, a connection is established. If the Workbench application is closed, the Discovery application is also closed out. the Workbench application prompts you to save your work before closing.

Note: The Discovery application will attempt to transfer all objects in the model. However, it is up to the downstream applications to accept them.

The Geometry Transfer ribbon group contains the following tools:

- **Update Parameters** to update parameter values in the Workbench application parameter set.
- **Workbench** to start Ansys Workbench with a geometry system.
- **Mechanical** to start Ansys Mechanical with the geometry imported.
- **Fluent** options in the dropdown to start the Ansys Fluent application. Select the appropriate options in the **Fluent Launcher**.
  - Select **Watertight Geometry Workflow** to transfer the geometry to the Ansys Fluent application with the Watertight Geometry workflow selected. See the Ansys Fluent User's Guide for more information about Using the Watertight Geometry Workflow.
  - Select **Custom New Workflow** to transfer the geometry to the Ansys Fluent application. See the Ansys Fluent User's Guide for more information about Customizing Workflows.
  - Select **Export PMDB** to save Ansys Part Manager Files (*.pmdb).
- **AEDT** options in the dropdown to start either **HFSS**, **Maxwell**, **Q3D Extractor**, **Icepak** or **Mechanical** transfers.

Note: There are differences in material properties between geometry and simulation transfer to HFSS as only material names are imported, while material properties are not imported.

2.9. Components and Assemblies

A component can be used to help organize your project. A component can have solids, surfaces, construction geometry or even other components inside them.

Components are shown in the model tree. All the objects are contained within a top-level design component which is created automatically when you first save your design. You can create any further components using the options in the context menu. Once you create a component, the icon in the tree changes to show that the top-level design is now an assembly.

- You can load multiple instances of a component. These are known as **dependent components**. Changes to one instance will change all instances.
- Copies of components that have been made independent of other copies are known as **independent components**. Changes to the independent copy will not affect the original and vice versa.

The context-menu contains options to work with the components--expand/collapse tree nodes, sort components alphabetically, creating, deleting, renaming components, etc. You can also export an internal component as a separate file.

2.9.1. Working With Components

You can use the options from the context-menu to do the following:

Flatten Assembly	Flatten the assembly hierarchy. Underlying sub-assemblies will be flattened to components in the tree. Any groups defined for the sub-assemblies will be deleted as well.
Collapse, **Collapse All**, **Expand All**	Collapse/Expand tree nodes to view the hierarchy.
Sort A - Z **Sort Z - A**	Sort components alphabetically for tree nodes with more than two child nodes.
New Component	Create a new component within the selected component.
Delete Empty Components	Delete any empty subcomponents within the component. If the component itself is empty, or contains only empty sub-components, it is also deleted.
Move to New Component	Create a component within the active component and move the selected object into it. If the object has a custom name, the new component will have the same name.
Move Each to New Component	Create a new component for each selected object within the active component and move the objects into those components. If the object has a custom name, the new component will have the same name.
Activate Component for Modeling	Activate the component to focus modifications and new object creation within the active component.
Open Component	Open the selected component in a new tab.
Import Component Groups and Parameters	Import groups and parameters set up for the component.
Make Independent	Make the component independent so that changes to the independent instance will not affect all other instances. The component is renamed in the tree.
Replace Component	Replace the selected components or assembly with another component or assembly. After replacement, the new component is internalized.
Replace All Instances of Component	Replace all instances of the selected components or assembly with another component or assembly. After replacement, the new components are internalized. **Note:** When a component is opened in a tab, only instances in the opened component will be replaced.
Export Component	Export the selected component as a separate file.

2.10. Running the Discovery Application from the Command Line

The Discovery application supports command-line arguments that you can include when launching the application from a command line or terminal prompt. All command-line arguments are case-insensitive.

Command-line syntax:

```
Discovery.exe <options>
```

> **Tip:** It is good practice to put quotation marks around the path to the solver executable and around the full path to the project. This ensures that spaces in the path or project are not an issue.

The following command-line arguments are available.

Option	Description	Value
`--exit-after-script`	Specifies the application should exit when script (defined by the `--run-script` option) is finished.	
`-h, --help`	Shows usage information.	
`--hidden`	Hides main window at start-up.	
`-l, --locale`	Specifies the country code (for example, `fr` for French) of the language to be used in the UI.	Country codes: Chinese Simplified - `zh-Hans` Chinese Traditional - `zh-Hant` English - `en` French - `fr` German - `de` Japanese - `ja` Example: `-l de` to use German in the UI.
`--mipassword`	The password used to connect to a Granta MI materials database.	String
`--miport`	The port used to connect to a Granta MI materials database.	Integer
`--miusername`	The username used to connect to a Granta MI materials database.	String
`--no-addins`	Prevents loading of addins.	
`-o, --open`	Project to open at start-up.	File path
`-p, --license-prefs`	The preferred license to use.	String
`--run-script`	Script file to run.	File path
`--script-api`	Script API version when used with the `--run-script` option.	Integer
`--script-args`	Script arguments as a semicolon-separated list.	Example: `--script-args "arg1;arg2;arg3"`
`--script-async`	Boolean value indicating whether to run script asynchronously.	
`--script-throw`	Throws an error on exception while running the script file.	

Option	Description	Value
`—skip-hardware-compatibility-checks`	Skips all hardware compatibility checks. Use with caution: disabling checks may cause the application to become unstable or crash.	
`--temp`	Overrides the temporary directory used by the application.	Directory

3: Discovery Application Settings

Set the startup options, interface, application performance (speed vs. graphics quality), and customize the tools that are displayed while you are working with your design.

To modify settings, select **File** > **Settings**.

This section contains the following topics:

3.1. General

This section contains the following topics:

Graphics Performance

Rendering quality	Select a value from 1 to 7, where 1 is the lowest quality and 7 is the highest quality. Select a low number to increase the speed of the application or select a high number to increase the quality of the graphics. Increasing graphics quality may lower the responsiveness to actions in the Design window. If you notice a delay when working with your design, modify this option to increase application speed. Select **Custom** to set your own values for **Surface tolerance**, **Normal tolerance**, and **Max edge length**. • **Surface tolerance** is the distance between **Facet Edges** and the actual surface. It is a measure of the distance between the facets and the real geometry. • **Normal tolerance** is the angle between adjacent **Facet Normals**. It is a measure of the smoothness of the faceting. • **Max edge length** sets a limit for **Facet size** and eliminates long thin faceting.
Optimize for Meshing	Adjusts the application's performance settings to make the meshing process faster and more efficient. **Optimize for Meshing** is unavailable when **Rendering quality** is set to **Custom**.
Recalculate Rendering	Use this option to manually update the 3D display in the graphics window after making custom changes. In some cases, modifications to the model may not be immediately reflected visually due to rendering performance optimizations. While the geometry remains accurate, the visual display may become misaligned or appear at reduced quality. Click **Recalculate Rendering** to refresh the view and restore proper visual alignment.
Anti-aliasing	The amount of smoothing applied to geometry rendering in the Design window.

Graphics performance mode (restart required)	Use this setting to improve overall performance and graphics, including smoother interactions with the model. This setting may also alleviate or resolve issues related to remote desktop (RDP) configuration and some initialization issues with configurations involving multiple GPUs where the application fails to launch due to more than one GPU supporting the application rendering. **Changes take effect after restarting the application.** • **Auto-detect** automatically selects the optimal mode based on your system hardware and drivers. • **Standard (best all-around)** provides balanced performance and visual quality. Suitable for typical workflows and systems. • **Enhanced (high frame rates)** prioritizes rendering speed for smoother interaction and higher frame rates. Recommended for high-end GPUs or performance-critical tasks.

User Interface

Show overlay tooltips	Select this option to display hints when you hover over tools, tool guides, and other icons. They briefly explain what will happen when you select the tool and provide some hints on how to use the tool.
Fade inactive overlays	Select **Fade inactive overlays** to hide unused overlays.
Dock sketch grid tool cluster	Select this option to dock the sketch grid tool cluster to the top and center of the window.
Default stage for new documents	**Explore** by default. Select this option to choose the stage you want to open new documents.
UI theme	**Dark** by default. Choose between the **Dark** and **Light** theme to modify the appearance of the UI. These themes prioritize the visual acuity of the model and results, standardize color use, and improve contrast. You can return to the legacy themes by choosing **Revert to legacy theme colors**. You can also enable and disable the **Solid background** setting. When enabled, a solid color without any gradient is applied to the graphics scene. Click the color picker to choose a different solid color from the palette.
Heads-up display size	**Comfortable** by default. Select this option to modify the display options. You can choose between **Compact**, **Wide** and **Comfortable**.
Reset heads-up display placement on tool change	Controls the placement of the HUD. When selected, the HUD is unpinned on tool change and returns to its default location based on the model or the selection. This option is **OFF** by default.

Appearance

Add datums to scene image	Select this option to include currently visible datums (planes, axes, origins) in the image when copying the scene from a window.
Add tool graphics to scene image	Select this option to include the tool elements in the image when copying the scene from a window.
Show arc centers	Select this option to display small crosses on the sketch grid at the centers of circles, ellipses, polygons, and arcs.

Make background transparent when saving images	Select this option to ensure a transparent background when images are saved.
Fill cross section	**ON** by default. Select this option to fill cross sections.
Crop scene image to non-empty area	Select this option to have the copied scene image automatically cropped to the non-empty area in the scene. When selected, you can specify a margin (in pixels) around the visible objects for all four margins: **Left**, **Right**, **Top** and **Bottom**.
Enhanced window shaded	Select this option to shade the faces in your design with shadows.
Reset Window Layout	This setting resets the application display into the original default state. Any windows that are undocked will be docked to their default positions.

Selection

Automatically calculate power selection results	Select this option to automatically display objects in the Selection panel. Objects similar to the selected object are automatically displayed.**ON** by default.
Hit radius	Enter the number of pixels your cursor can be from the object you want to select.
Power selection relative tolerance	Enter a percentage to change the relative tolerance used when searching in the Selection panel.

3.2. Navigation

Customize how you spin, pan, and zoom. You also can customize which mouse button/key-press combinations are used as shortcuts for these tools.

Mouse Mapping	The mouse-mapping options enable you to assign actions to mouse clicks and keyboard/mouse combinations. Choose the **Spin**, **Pan** and **Zoom** settings.
Zoom	Choose the **Zoom drag direction** and **Zoom with mouse wheel** settings
Spin	Choose the spin options to **Show spin center**, **Rotate about preselected object** and select the spin **Speed**.

> **Note:** You can also spin, pan, and zoom with the left mouse button by toggling the tools in the **Orient** group on the **Display** tab.

3.3. Licensing

Licensing Options

Discovery license	Select the preferred Discovery application license level: **Discovery Enterprise**, **Discovery Premium**, or **Discovery Pro**. For more information, see **Product Licensing**.

Check for lower license if requested is unavailable (restart required)	Select to check for a lower level license if the license requested by the **Discovery license** option is unavailable.
Alternate license (restart required)	Select an Alternate license for flagship products. For more information, see **Product Licensing**.
Check for Discovery Pro license prior to alternate license (restart required)	Select to check whether you have a Discovery application Pro license before checking for an alternate license. This option is applicable only when **Discovery license** is set to **Discovery Pro**, and an **Alternate license** preference is specified.

3.4. Units and Display Precision

The options selected affect how values are displayed including all input units (tree, HUD, defined variations) and all output (charts, monitors, legend, defined variations).

This section contains the following topics:

Model Units and Display Precision - All New Documents

Choose the units for:

- **Length scale** - Select **Standard**, **Small** or **Large**.

  The modeling precision for smallest sketched entities and smallest round size for standard units are given in the table below.

Entity	Modeling Precision
Smallest sketched line	0.0301 mm
Smallest sketched radius at 90 degrees	0.02772 mm
Smallest sketched radius at 180 degrees	0.015 mm
Smallest sketched radius at 270 degrees	0.0115 mm
Smallest circle radius	0.03 mm
Smallest 3D round size	0.1 mm

  The table below summarizes the model size box and precision values for various model sizes.

Model Size	Units	Size Box	Precision
Large	Kilometers, meters, miles, feet	+500 km to -500 km	$1.0e^{-5}$ m
Standard	Meters, Centimeters, Millimeters, feet and inches	+500 m to -500 m	$1.0e^{-8}$ m
Small	Micrometers, nanometers and mils	+0.5 m to -0.5 m	$1.0e^{-11}$ m

- **Model unit system** - Choose the units depending on the **Length scale**. Select **Metric** or **Imperial**.

Model Unit System	Units
Metric units	Millimeters, Centimeters, Meters, Micrometers, Kilometers, and Nanometers.

Imperial units	Inches, Feet, Feet and Inches, Miles and Mils.

- **Length** - Select the default units used for length.

Unit System	Units
Small units	Micrometers, Nanometers and Mils.
Large units	Kilometers, Meters, Miles and Feet.

- **Angle** - Select Degrees, Degrees/Minutes/Seconds, Degrees/Minutes, Radians.
- **Length decimal places** - If you use Imperial units, you can select Decimal/Fraction to display distance measurements that are less than 1 unit.
- Select **Show trailing zeros** option to display trailing zeros in decimal measurements.
- **Angle decimal places** - Select this option to display angle measurements that are less than 1 unit.
- Select **Show trailing zeros** option to display trailing zeros in decimal measurements.

Sketch Grid - All New Documents

Grid spacing	Enter the distance you want between the smallest grid lines.
Grid per block	Enter the number of smaller grid lines between larger, heavier grid lines.
Reset to Default	Remove user-defined settings and reset to default values.

Simulation Units and Display Precision - All New Documents

Simulation unit system	Select Metric or Imperial. Select **Show inputs with non-default inputs as entered** to turn off display units and show only input values.
Length	Select the default units used for length. Metric units include Millimeters (default), and Meters. Imperial units include Inches, and Feet.
Force	Select the default units used for force.
Mass	Select the default units used for mass.
Stress and pressure	Select the default units used for stress and pressure.
Temperature	Select the default units used for temperature.
Significant digits displayed	Select the number of significant digits to be displayed.

Note: Any changes to the **Simulation Units and Display Precision** settings apply only to new documents.

Any changes to modeling units from the status bar or **Settings> Units and Display Precision** do not affect these units.

3.5. Physics

Set the physics options for the simulation.

This section contains the following topics:

Gravity Defaults

By default, the Discovery application adds a gravity condition to your analysis to simulate self-weight, and/or buoyancy effects, for structural and fluid physics respectively. If neither of these are selected, then the gravity specification will have no effect on your simulation

Include self-weight	When enabled, self-weight is simulated in the specified direction of gravity. Including self-weight depends on the density of the material. If the self-weight is negligible in comparison with other forces, you can turn self-weight off by clearing the selection of **Include self-weight**. Turning on self-weight also means that the gravitational acceleration is used in combination with any added mass inputs.
Include buoyancy	When enabled, buoyancy is simulated within a fluid flow analysis. For fluids, buoyant flow is the most important effect of gravity. Because the force of gravity is proportional to fluid density, buoyant flow can arise whenever the fluid density varies due to the temperature or material specification. The addition of buoyancy simulates natural or free convection, where the fluid density varies in temperature using the Boussinesq approximation. If these effects are not important in your simulation (for example, where the velocity of the fluid is such that buoyancy has little impact), you can turn buoyancy off by clearing the selection of **Include buoyancy**.
Gravity direction	To specify the direction of the gravity, select the appropriate axis from the **Gravity direction** dropdown list.

Physics Defaults

Working temperature	Specify the **Working temperature** value and units.
Fluid-solid interface tolerance	Specify the value for **Fluid-solid interface tolerance**. If fluid and solid bodies are within this distance, a fluid-solid interface is created for fluid-solid heat transfer simulations. If any thin solid bodies (for example: pipes, insulation) are adjacent to a fluid, consider reducing this distance to not have duplicate interfaces. **Restart recommended.**
Fluid-solid interface face overlap allowance	Specify the default percentage of face overlap allowed during interface detection. **Restart recommended.**
Contact borderline for review tool	For Connection Assessment, set the range for connections that should be highlighted as borderline.
Max bolt hole coaxial offset	Specify a default tolerance for the **Max hole coaxial offset** setting. This option ensures the application detects bolts with misaligned geometry when the misalignment distance falls within the specified tolerance.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Material Defaults

Libraries and Locations	Specify which libraries to use, and their locations. Access additional materials by connecting to the Granta MI materials databases. To see the settings under **Granta MI Materials**, select **Granta MI materials database** under the **Material Defaults**.
Assignments	Select the default solid, fluid, conducting and dielectric material under **Assignments**. Restart the application or create a new simulation for the new defaults to take effect.

Scoping Defaults

Include imported bodies by default	• **Never**: no matter the size of their assembly, the parts will not be automatically included in the physics scope • **Always**: no matter the size of their assembly, all the parts will automatically be included in the physics scope • **Depending on number**. You can also select the slider to choose a number between 0 and 499.
Automatically include new bodies	Select **Automatically include new bodies** to include all parts into the simulation. **ON** by default and automatically includes the newly created bodies into the simulation scope.
Suggest QuickScope Automatically	Select **Suggest QuickScope automatically** to automatically add a new named selection based on the scoped bodies. **ON** by default.

Contact Review Defaults

Specify the default visibility of contact markers.

Show contact markers	You can show contact markers for: • **Up to maximum**: up to the specified maximum number of markers • **Always**: all contacts all of the time • **Selection only**: only on the selected geometry Changing this option is useful when you are trying to work with projects that have a large number of contacts.

Solver Settings

Specify the **Max number of processors** settings to use during solving.

Max number of processors	Choose between **Use solver default**, **Use maximum available** or **Specify number of cores**. For **Specify number of cores**, indicate the number of cores.
Fluids solver	**Program Controlled** is the default selection. This option applies the most appropriate pressure-based solver for the simulation. Choose **SIMPLE** for steady-state, incompressible simulations or **Coupled** for compressible or swirling flows.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Enable Fluent solver option	Off by default. Switch on for the additional option to choose between the Fluent and the LiveGX solvers from Fluid Simulation Options.
Transfer to Fluent (restart required)	Choose **Export cas.h5 file** to use this file format for transfer to the Fluent application. When the **Export journal file** option is selected, a journal file is saved along with mesh and settings file and can be later imported into the Fluent application.

Simulation Options Defaults (Refine)

The application provides default simulation options which you can override. The modified default values can be used for all future simulations. Any changes made to the default settings can be reverted to the original default values using the **Reset Simulation Options Defaults** button. Note that these options are only applicable in the **Refine** stage.

Structural	For structural simulations, you can modify the **Solution progression controls** settings. When set to **Automatic**, the application automatically adjusts the solution progression based on the physics inputs. If you choose to **Define specific values**, you must specify the **Initial number of substeps** that the solver will use initially. Specify the **Minimum number of substeps** and the **Maximum number of substeps** needed.

Fluid Flow	For fluid flow simulations, the LiveGX solver is the default solver. The LiveGX solver leverages GPU solver technology combined with body-fitted meshing to provide both fast and accurate fluid solutions. Select **LiveGX CPU mode (restart required)** if you want the application to use the CPU solver instead. The CPU solver allows you to run larger cases, but it will take more time. **Note:** If you switch on **Enable Fluent solver option** under **Solver Settings** (see table above), **Use LiveGX solver** is on by default. You can turn off **Use LiveGX solver** to set the Fluent solver as the default or leave **Use LiveGX solver** selected so that LiveGX remains as the default. To manage convergence, you can choose the stopping criteria to determine when the problem has converged by selecting one of the **Stop on (restart recommended)** options as the default: • **Engineering convergence**: The engineering quantities of pressure, velocity, and temperature are monitored for each flow condition, and temperature and pressure are monitored for each wall condition. When these monitored quantities have stabilized, the solution will be determined as complete. This option is typically useful if these values are the primary focus of the solution and are unaffected by local areas of turbulent flow. If convergence is not achieved, the solution will stop when the **Maximum number of iterations** are reached. • **Numerical convergence**: Convergence is determined by examining differences throughout the fluid volume. This is useful if you are interested in the flow within the volume. Enter the **Maximum number of iterations**, if convergence is not met after this number of iterations, then the simulation will stop. Enter the **Target numerical convergence** to specify the difference for residual measurements (except energy) allowed between iterations. If the difference is below this level, it will be considered as converged. Enter the **Target numerical convergence, energy** to specify the difference for energy allowed between iterations. If the difference is below this level, it will be considered as converged. • **Monitored value convergence**: This is similar to engineering convergence but will only use monitors of average values on the surfaces that you have defined. • **Solution stopping criterion**: Convergence is determined when the **Target stopping criteria** tolerance is met. A default value of 0.01 (or 1%) is used. When the percent change in solution is below that tolerance, the solution is converged. This is only applicable when using the LiveGX solver. When modeling turbulent flow, choose one of the turbulence models as the default **Modeling method**. Alternatively, if the flow is not turbulent, set the fluid flow modeling method to **Laminar**. Learn more about each of the turbulence options. The default **Meshing method** is **Tetrahedra mesh**. Depending on the complexity of your model and the solution accuracy you can change the default setting to **Polyhedra mesh**. Learn more about Fluid Flow Meshing Methods on page 872. Enter a **Reference pressure** to serve as the default value when calculating the force on a wall.

Simulation Options Defaults (Explore)

The application provides default simulation options which you can override. The modified default values can be used for all future simulations. Any changes made to the default settings can be reverted to the original default values using the **Reset Simulation Options Defaults** button. Note that these options are only applicable in the **Explore** stage.

Fluid Flow	Choose either **Turbulent k-omega SST** or **Smagorinsky (LES)** as the default turbulence **Modeling method**. Alternatively, if the flow is not turbulent, set the fluid flow modeling method to **Laminar**. Learn more about each of the turbulence options. Enter the **Target stopping criteria** to determine convergence. A default value of 0.01 (or 1%) is used. When the percent change in solution is below that tolerance, the solution is converged. **Capture sharp edges** is set to **Program controlled** by default. This allows the application to choose how to best capture sharp edges on your model. You can change the default setting to capture **All sharp edges**. This option attempts to capture all sharp edges in the mesh leading to increased accuracy. This can increase computation time and decrease solution stability. Otherwise, choose **None** to avoid capturing sharp edges. Select **Capture thin structures** to accurately represent mesh areas with thicknesses smaller than the local size. This creates anisotropic cells in these thin regions and reduces the total cell count. Stretched (anisotropic) cells are created along thin directions, which can help lower the cell count compared to isotropic refinement. If this option is not enabled, isotropic cells are generated and very thin regions are merged or overlooked, possibly leading to a loss of detail.

Mesh Metrics (Refine)

Structural	You can control the default mesh metrics monitored during the solve and specify limits for those metrics. For structural simulations, those include: • Element quality (**OFF** by default.) – lower limit (default 0.05) • Jacobian ratio (**ON** by default.) – upper limit (default 30) • Aspect ratio (**OFF** by default.) – upper limit (default 20) Any changes made to the these settings can be reverted to the original default values using the **Reset Mesh Quality Options Defaults** button.
Fluid Flow	You can control the default mesh metrics monitored during the solve and specify limits for those metrics. For fluids simulations, those include: • Orthogonal quality (**ON** by default.) – lower limit (default 0.1) • Skewness (**OFF** by default.) – upper limit (default 0.95) • Aspect ratio (**OFF** by default.) – upper limit (default 20) Any changes made to these settings can be reverted to the original default values using the **Reset Mesh Quality Options Defaults** button.

Global Fidelity Defaults

Fidelity multiplier (Explore)	Choose from: • **Standard** is the default recommended setting. • **Aggressive** captures more geometric details for higher fidelity results but also consumes more memory during the solve. • **Extreme** captures more geometric details for higher fidelity results but also consumes more memory during the solve. • **Custom** allows you to enter a multiplier. Numbers greater than 1 (the default) increase fidelity while numbers less than 1 decrease fidelity.

Workflow Options

Test cases for topology optimization in defined variations	**Test cases for topology optimization in defined variations** defines test cases in the Design Variations table rather than specifying multiple simulations. **ON** by default.
Background automation for physics	**Background automation for physics** improves performance of the application while resource-heavy physics computations are being performed in background. **ON** by default.
Show fluid-to-fluid interfaces	**Show fluid-to-fluid interfaces** allows you to control the detection distance and visually diagnose the overlapping interfaces or conversely check that the interface does exist where it is expected. Decreasing the detection distance enforces a tighter tolerance for detecting whether faces are touching or not. Using a smaller detection distance usually removes any spurious faces that may be overlapping with another condition or interface. **ON** by default.

3.6. Results

Set the results options for the simulation.

This section contains the following topics:

Display

Show modal stress results (re-solve required)	Select to display modal stress results while solving structural cases. **OFF** by default. If you turn this option on during a session, you will have to re-solve your simulation.
Color map	Allows you to switch between color maps: standard **Rainbow**, **Accessible Rainbow (Turbo)**, and **Blue-Gray-Red (Cool Warm)**. Accessible Rainbow and Blue-Gray-Red address color blindness ambiguity.

Full Results

Include Refine results	Select to save the full set of Refine results to the **.dsco** file when saving.

Include Explore results	Select to save Explore results to the **.dsco** file when saving. Two options are available: • **Visual results quality**: **High**, **Medium** (default), **Low** • **Include result variables**: **All** or **Essential** (default). Essential includes: ° Fluid: velocity, total pressure ° Static structural: displacement, Von Mises stress ° Topology optimization: displacement, Von Mises stress ° Fluid thermal: temperature, velocity, total pressure ° Solid thermal: temperature, heat flux ° Fluid-solid heat transfer: velocity, total pressure, temperature, heat flux ° Electric: Joule Heat, current density, electric potential ° Electric thermal: temperature, heat flux, current density, Joule Heat, electric potential ° Electric thermal fluid thermal: temperature, heat flux, velocity, total pressure, current density, Joule Heat, electric potential. **Note:** When saving Explore results, only the graphical results are stored. This lets you view and interact with graphical results. Any tasks that need solver data, such as evaluating new monitors, will require you to solve again. **Note:** When saving Explore results that show both the mesh and a cut plane, only the cut plane results will be retained. If you want to view the surface mesh, you can either turn off the cut plane or avoid saving the file with the cut plane displayed. To see the complete mesh after resuming, simply re-solve.

Saved Scene Results

Snapshots are images taken of each saved scene that can be used wherever you need images--in the Web Viewer or documents such as presentations or reports. The **Saved Scene Results** settings gives you control over default settings to customize your scenes.

Generate snapshots automatically	Generate the images while saving, solving and creating a saved scene.
Save snapshots to file	Automatically add snapshots to the project file. • **Exclude scenes older then # runs**: Specify a value for the maximum number of runs for which you want to include scenes when saving a model. Scenes older than the specified run number (with a maximum up to 10) will not be included. • **Include 3D scene snapshots**: When enabled, 3D scene snapshots are included in the saved model.

Generate default scenes	Generate default scenes for some physics conditions. You can override this setting for a document by selecting **Generate Default Scenes** in the **Scene** dropdown list. **OFF** by default. • **Scene view**: Set the default view orientation for scene capture. Options include **Home**, **Isometric**, **Trimetric**, **Right**, **Left**, **Top**, **Bottom**, **Front** and **Back**.
Use transparent thumbnail background	Generate thumbnail images with a transparent background.
Set image resolution for remote downloads	Set the default resolution for downloading remote solve images. Options include: • **Low (1280 x 720)** • **Medium (1920 x 1080)** • **High (2560 x 1440)** • **4K (3840x2160)**

Report Content

Specify the **Report Content** settings to define the default inclusion and exclusions of scenes in reports. You can override this setting in the **Save As** dialog when generating a report. This override is maintained until you exit the session or manually change the option back. New sessions default to this setting.

Default scene image resolution	This option determines the default resolution of the images when exporting scenes using the **Export 2D** scene commands in the ribbon. • Choose **Window size** if you want images to be the same height/width as the window when the command was invoked. • Choose **4K (3840x2160)** to generate an image 3840x2160 pixels in size.
Default scene inclusion, 2D	Select the default option for the 2D scene images included in the report. • Choose **All** if you want scene images generated in both the Explore and Refine stages to be included in the report. • Choose **Automatic** to include images from any new scenes. • Choose **None** to exclude any scene images in the report.

Monitor

Specify **Monitor location for fluid simulations** to be on **Individual faces** or on **Conditions and interfaces**. If you select **Individual faces**, results are calculated only on the selected faces. If you select **Conditions and interfaces**, results are calculated at the location where the condition is applied.

3.7. Detailing

Set the detailing options.

This section contains the following topics:

Application Options

New window font	You can select any font that is installed on your system to set it as the default font for all new windows. This font will be applied to new windows, whether they are created for old or new documents, as well as to new sessions.
Default drawing graphics style	The default graphics style used for new drawings.
Autoscale drawing sheets	Check this option to have the scale automatically determined based on model size. Uncheck the option to set a default custom scale.
Custom scale	If **Autoscale drawing sheets** is unchecked, you can set a default custom scale.
Create total cross sections	Check this option to have new sections created as **Total** sections. Uncheck the option to have new sections created as **Area** sections.
Default section fill style	Choose either **Hatch** or **None**.

Engineering Notebook Options - All New Documents

Predefined Defaults	To reset all options to predefined defaults, click one of the **Defaults** buttons to set all document options to default. • **Reset to ASME Default** • **Reset to ISO Default** • **Reset to JIS Default** • **Reset to GB Default.**

View Options

View projection	For view project, you can select: **First Angle** or **Third Angle**. • For **First Angle**, you label the view based on the direction you are looking. For instance, if you are looking at the front of an object, you see its back, so you would label that view as the "Front view." • For **Third Angle**, you label the view based on the object itself. So, if you are looking at the front of an object, that would be the "Front view."
Front view position	Choose the default front view position: • **Top left** to place the front view at the top left corner of the drawing sheet. This is the ISO standard. • **Top right** to place the front view at the top right corner of the drawing sheet. • **Bottom left** to place the front view at the bottom left corner of the drawing sheet. This is the ASME and JIS standard. • **Bottom right** to place the front view at the bottom right corner of the drawing sheet. If you use third angle projection, ISO standards also allow this position.
Section line arrow size	Specify the length of the arrow on section lines.
Section line length	Specify the length of section lines.
Detail view name text height ratio	Set the ratio of a character's height to width. For example, a value of 1.4 sets the character height to 140% of its width.
View note layout	Select **One Line** to display the detail name and scale on one line. Select **Two Lines** to display the scale below the detail name.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Detail name note prefix	Select how you want the detail label to appear on the drawing sheet.
View scale note prefix	Select how you want the scale label to appear on the drawing sheet.
Section name note prefix	Select how you want the section label to appear on the drawing sheet.
Detail view boundary note placement	Select how you want the detail name and scale information to be positioned relative to the detail boundary.
Section cut line extension distance	Specify the length.
Section line arrow direction	Choose either **Away from cut line** or **Toward cut line**.
Projected view arrow style	Select the style you want to use for the projected view arrowheads from the dropdown list.
Show projected view name	Check whether to show the projected view name.
Show projected view arrow	Check whether to show the projected view arrow.
Projected view name prefix	Specify a new prefix or choose the default value.
Show auxiliary view name	Check whether or not to show the auxiliary view name.
Show auxiliary view arrow	Check whether or not to show the auxiliary view arrow.
Auxiliary view name prefix	Specify a new prefix or choose the default value.
Projected/auxiliary view arrow length	Specify the length of the arrow on projected and auxiliary views.

Annotation Options

Text height	Specify the height of the annotation text.
Leader circle size	Specify the size of the circle that connects note leaders to faces.
Leader arrow length	Specify the length of the arrow on note leaders.
Leader arrow width	Specify the size of the arrowhead on note leaders.
Leader shoulder length	Specify the length of the line from the note text to the note leader arrow.
Leader textbox gap	Specify the size of the margin between the note text and the beginning of the note leader.
Center line extend	Specify the length that a center line will extend past the edge of the object.
Extension line gap	Specify the size of the margin between geometry and the end of dimension lines.
Extension line extent	Specify the length that the dimension lines cross each other.
Dimension line extent	Specify the length of the arrow that appears outside the dimension lines.
Dimension text offset	Specify the distance between the dimension text and its leader line.
Foreshortened radial dimension center size	Specify the size of the cross shown to represent the center.
Chamfer dimension style	Choose either **Linear** or **Normal to chamfer**.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Chamfer dimension text format	Choose between: • **X 45** • **C**
Arrow fill style	Select the style you want to use for the arrowheads on note leaders from the dropdown list.
Dimension arrow shape	Select the shape you want to use for the arrowheads on dimensions and leaders from the dropdown list.
Dimension text location	Select how you want to align the note leader text with the note leader line from the dropdown list
Note space	Select the default space for adding notes. The options available are **View space** (default) and **Model space**. Notes in the **Model space** are their true size and are not scaled based on the detail scale setting.
Virtual sharps rendering style	Select the symbol you want to use to indicate the virtual sharp.
Enforce dimension line	Check the box to use the European standard of keeping the dimension line when the extension lines are shown.
Show zero value for base ordinate dimension	Display the baseline dimension as a 0.
Tight gap between dimension line and text	Check the box to shrink the gap between the dimension text and witness lines.
Show common dimension line for ordinate dimensions	Display all dimensions along the same line.
Horizontal dimension text	Check the box to keep all annotation text oriented horizontally.
Rotate text	Check **Rotate text** if you want the dimensions to be displayed along the line instead of perpendicular to the line.
Override layer color for annotations	Check the box to set all annotation text to the color selected shown in the **Annotation color** control. If you choose to override the layer color, select the color used for the override from the **Annotation color** control.
Hide annotations behind model	Allow solid geometry to hide annotations that are behind the geometry.
Annotation color	Choose annotation color.

Linestyle Options

Thick lineweight	Specify the default width for thick lines.
Medium lineweight	Specify the default width for medium lines.
Thin lineweight	Specify the default width for thin lines.
Trim Back	The values in the **Trim Back** section control the line weight for side offset handles.

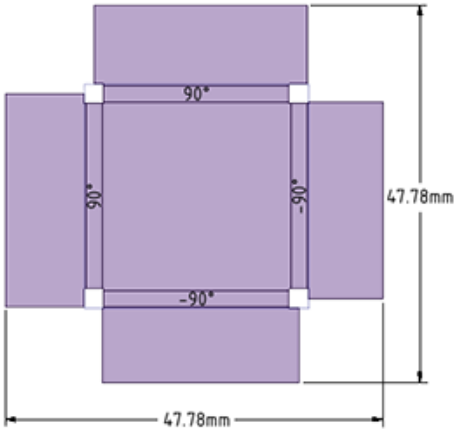
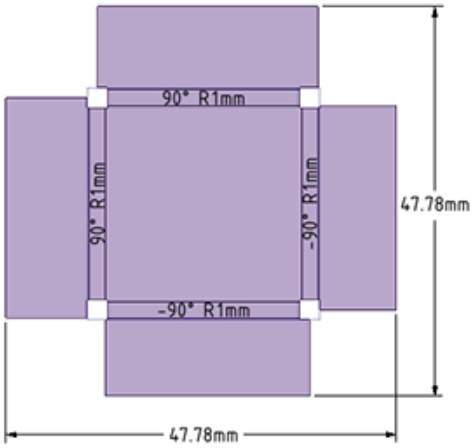
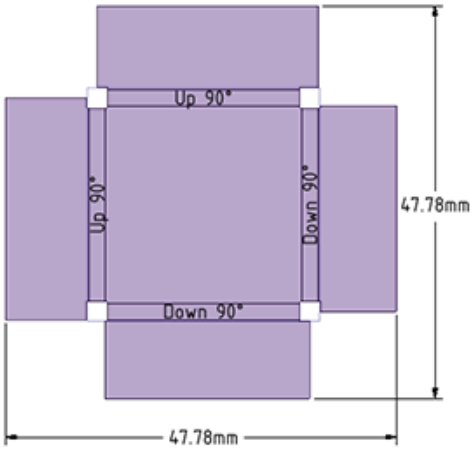
Type of Object	Select an object type from the list, then select the **Line style** and **Thickness** for that object. • Options for **Detail view clipping edges** apply to all clipping edges of partial views. You can also set a **Color** for the clipping edges. • You can also choose a **Color** for **Broken view clipping edges**. • Only a **Thickness** option is available for **Cross hatching**, **Area cross hatching border lines**, **Cross section arrow tips**, **Hidden edges**, **Visible edges**, **Broken-out section clipping edges** and **Projected view arrow tips**.

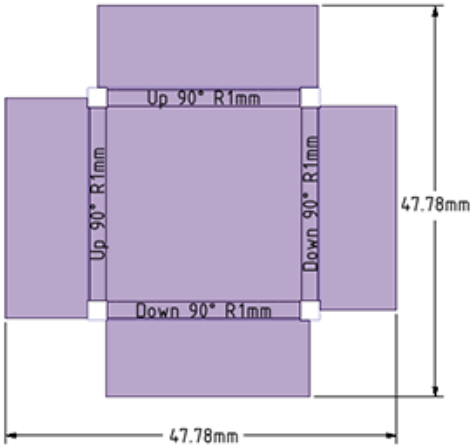
3.8. Sheet Metal

This section contains the following topics:

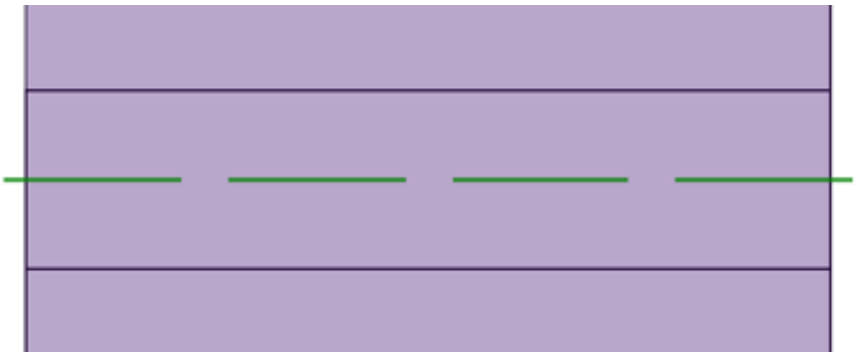
Basic Defaults

Thickness	Enter a value for the default wall thickness of sheet metal parts.
Bend Table	Specify an existing **Bend Table** as the default for new parts.
K-Factor type	Select **Constant** or **Variable**.
K-Factor (for R=T)	Enter a value for the K-factor. The K-factor is a value between .25 and .50 that is used to calculate the bend radius. K-factor is a percentage of the metal thickness and depends on factors such as the material and type of bending operation.
Bend radius	Enter a value for the default bend radius. The value is a **Thickness ratio** by default, but you can select **Value** to enter an absolute value.
Split face gap	Set the default gap created when you bend a split face, which is shown as C in the image below. 
Junction gap	Set the default gap created in Full, Partial, and No Overlap Junctions and Flanges. Use • **Thickness ratio**: changes the gap based on the thickness of the wall. • **Value**: specifies the exact width of the gap.
Junction overlap	Set the percentage of overlap in partial overlap junctions. The default is 50%. For individual partial overlap junctions, you can change the **Overlap Proportion** property in the sheet metal **Properties** window with the junction selected.

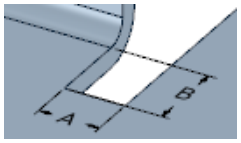
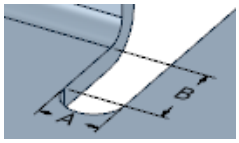
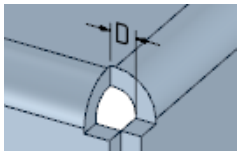
Bend annotations	Set the format for displaying bend annotations in the unfolded part. **+/- Angle**  **+/- Angle radius**  **Up/down angle**  **Up/down angle radius**



Note: When the bend radius is set to **Thickness ratio** and you change the sheet metal part thickness for the part, the value for the **Inner Radius** now changes in the sheet metal **Properties** window. Changing the part thickness property will not change all the bend radii, but the displayed bend radius changes to show that there is a problem for you to fix. You can fix them by selecting the desired bends and changing their properties manually.

Trim bend lines to part outline	Controls whether bend lines extend beyond the unfolded part outline. **OFF** by default. *Unchecked: the lines extend beyond the outline.*  *Checked: the lines are trimmed at the part outline.* 

Relief Defaults

Bend relief	**Type**	Choose **Square**, **Rounded**, or **Rip**. • To use a different relief for an individual bend, choose a **Type** from the **Edge Relief** menu in the ribbon. • To change the **Type** after a bend is created, select the **Relief** and change the type in the **Properties** window.
	Width and **Depth**	• **Thickness ratio**: changes the relief width and depth based on the thickness of the wall. • **Value**: specifies the exact width and depth of the bend relief. In the images below, A is the width and B is the depth of the bend relief. *Square relief*  *Rounded relief* 
Corner relief	**Type**	Select a relief **Type** from the dropdown menu. • **Filled** • **Smooth**: Diameter • **Square**: Width • **Circular**: Diameter • **Triangular**: Width and Height • **Rectangular**: Width and Height • **Oblong**: Diameter • **Diagonal**: Width • **Laser Edge**: Diameter • **Laser Symmetric**: Diameter • **Default** • **Punch**
	Depth	• **Thickness ratio**: changes the relief width and depth based on the thickness of the wall. • **Value**: specifies the exact width and depth of the bend relief. *D is the corner relief depth.* 
	2D Corner	Create a 2D corner relief.

Export

Round all sharp corners at DXF export	Select this option to round all the edges whose neighboring faces are not co-planar when you export a DXF file. Select this option to set the **Round radius** values: • **Thickness ratio**: changes the width and depth. • **Value**: specifies the exact width and depth.
Export bend extent lines at DXF export	Select this option to export the bend extent lines at DXF export. Bend lines can be trimmed to the part outline if **Trim bend lines to part outline** is checked in the **Basic Defaults** section. **OFF** by default.

Unfold Layers

Bend lines - up	Select a default color and line style. Line styles available include: • **Solid** • **Hidden** • **Dashed** • **Long dash dotted** • **Long dash double dotted** • **Dotted** • **Zigzag** • **Long dash dash** • **Long dash double dash**
Bend lines - down	
Forms	
Overall dimensions	Select a default color.
Bend dimensions	
Markers	

Colors

Select the default colors for the various types of sheet metal objects.	• **Sheet** • **Junction** • **Joggle** • **Edge relief** • **Corner relief** • **Tab** • **Miter** • **Form** • **Bead** • **Gusset** • **Hem** • **Notch** • **Hinge** • **Panel bend** **Note:** To distinguish between corner reliefs and edge reliefs, select two different colors. Colors on the model will appear more saturated than the colors shown here.
Reset Colors	Select to reset to default colors.

3.9. Advanced

This section contains the following topics:

Advanced Options

Modify how your design changes are displayed in the graphics window, enable script editor and reset the layout of the panels.

Ansys® Product Improvement Program	Participate in the **Ansys® Product Improvement Program** which enables Ansys, Inc. to collect and analyze anonymous usage data reported by Ansys software without affecting your work or product performance. Analyzing product usage data helps Ansys® to understand customer usage trends and patterns, interests, and quality or performance issues. The data enables Ansys® to develop or enhance product features that better address your needs.
Notify when new version available	Select this option to be notified of new versions. You can also manually check for updates using the **Check for updates** button. This setting is removed for customized branding (Speos, etc.).
Enable history tracking by default	Select for different default behavior.
Maximum undo steps	Enter the maximum number of steps that can be undone. **Note:** Restart required.
Language	Select one of the available languages from the dropdown. **Note:** Restart required.
Ignore problem area search limit in repair tools	The problem search area is capped at 250. Select this option to ignore this restriction, which allows the repair tools to find and repair all problem areas.
Default extrude behavior	Select a different default behavior for extrusion from the dropdown.
Top view direction	Set the default top direction of a session. **Z** is the up axis and **Y** is the down axis. Once you set it, you need to open a new file to see the change. This setting remains constant even if you sketch or import any model.
Enable advanced callbacks in the extension builder	Enable callback functionality for custom scripts. Two types of callbacks are supported: one triggered on every graphics window refresh and another triggered upon tool exit.
Animate changes to view projection	When switching between different views, this setting will animate the change for a smoother visual experience.

Compute GPU	When running on a multi-GPU hardware configuration, the dropdown list will display any CUDA-supported GPUs found on your system and available. • Default • [CUDA-support GPU] **Note:** Restart required. **Important:** When running on a multi-GPU hardware configuration and the Discovery application fails to launch, it may be due to a settings mismatch between the Windows OS GPU selection and the NVIDIA control panel GPU selection. Both of these settings must agree. Confirm and align the two settings: • In **System** > **Display** > **Graphics** > **Default graphics settings**, select GPU from the Default high performance GPU dropdown list. • In **NVIDIA** > **3D Settings** > **Manage 3D settings**, select the same GPU (as above) from the OpenGL rendering GPU dropdown list.
Temporary data	You can use the default data location or **Browse** to a new location. Specify a value for the number of days after which you want to **remove temporary data older than days (Restart required)**.
Diagnostic logs	You can use the default data location or **Browse** to a new location.
Save Diagnostics	On-demand saving of diagnostics logs.
Enable prerequisite checks	Select this option to automatically verify that all required components, resources, and conditions are available before running certain operations or workflows. This helps prevent errors by ensuring prerequisites are met before execution.
Reset All User Settings	Remove all user-defined settings and reset to default values.

Beta Options

Select each Beta feature you want to enable. You may need to restart the application for some Beta features.

For more information about Beta features available in this release, see *Discovery Beta Features Documentation* in the Ansys Help.

3.10. Remote Services

Set the options to use remote services.

General

Default authentication mode	Select from **Login interactively** or **Use Windows credentials**. **Note:** Restart required.
Synchronize data with cloud	Select to maintain GL transmission format binary file (GLB) synchronization between the local Discovery desktop application data and the cloud (Web Companion application). • **Off**: synchronize is off • **On save**: synchronize GLB on save

Ansys Cloud

Select **Remote solve in Explore** to solve remotely in the Ansys Cloud. Available only in the Explore stage.

Note: Restart required.

Microsoft 365

Identify and save your organization's Microsoft 365 **URL** server address for the Discovery application's Microsoft 365 cloud service.

Note: Restart required.

Ansys HPC Platform Services (HPS)

Identify and save your organization's HPS **URL** server address for the Discovery application's High-Performance Computing (HPC) Platform Service (HPS).

Note: Restart required.

3.11. File

You can use the Settings panel to set options that affect file import and export.

This section contains the following topics:

3.11.1. General Settings for File Import and Export

Import

Use Discovery color tones when importing	Select this option to use the Discovery application's color scheme for layers. This option is enabled by default to avoid fully saturated colors, on which highlighting is less visible. Faces of imported data will retain their original color tones if this option is turned off.
Automatically save imported documents	Select this option to immediately save an opened non-Discovery application file to a dsco file. If you do not select this option, then opened files are not saved until you save the design.
Improve imported data	Select this option to improve a file when it is opened or inserted. You can choose for the application to: • **Simplify geometry**: simplifies complex faces and curves into planes, cones, cylinders, etc., • **Remove splits**: removes split faces on import, • **Heal bodies and faces**: attempts to clean minor geometric irregularities on import, • **Stitch nearby surfaces together**: stitches surfaces on import. Improve effectiveness with **Find coincident surfaces** and specifying **Tolerance**. De-selecting this option imports the file without improvements so the file appears more quickly, but you may have to perform the improvements manually.
Check geometry	Select this option to check the project geometry.
Import hidden components and geometry	Select this option to open or insert hidden components within CATIA v5, Parasolid, Creo Parametric, SOLIDWORKS, and SketchUp files, and turn off their visibility in the model tree.
Use lightweight	Select this option to use lightweight geometry import, which uses less memory and allows for faster import times. Lightweight file formats are limited to STEP, CATIA, IGES, SOLIDWORKS, Solid Edge, Creo, NX, Inventor and Rhino. An assembly containing multiple bodies can be a mix of lightweight and heavyweight. You can convert a lightweight body to heavyweight by right-clicking the body in the tree or in the design window and selecting **Convert to Heavyweight** from the context menu. Most modeling tools will not work on lightweight bodies. Simulation is only available in the explore stage and will only work on single bodies. For full simulation capabilities included bodies must be converted to heavyweight. The **Use lightweight** option is supported only for reader imports. This differs from the SpaceClaim application, which supports lightweight geometry import for both reader import and plug-in import.
Objects to be imported	Select the options you want to import. Choose from **Free curves**, **Axes**, **Points**, **Coordinate systems**, **Planes** and **Object names**.

Import large file	When importing large files, configure the following options: • **Import empty components**: Choose whether to include empty components during import. • **Exclude small parts by volume**: Enter a volume threshold to exclude parts smaller than this value during import. • **Alert when file is larger than <value> gigabytes**: Choose to display a notification when the file size exceeds the specified value. The default is **1 GB**.
Named selections	Choose from **Allow All**, **Block All**, **Prefix**, or **Suffix**. When using **Prefix** or **Suffix**, enter the **Named selection key**.
Material properties	Select this option to import material properties. **ON** by default.

Export

Improve data on export	Select this option to clean up imprinted edges and split curves when you export data.
Export hidden components and geometry	Select this option to save components that have their visibility turned off in your design as hidden components when you save them as any other file type. This option is supported for ACIS, Rhino, and SAT/SAB formats.
Export object names	Select this option to export CATIA files with object names.
Split periodic faces	Select this option to break down a face to minimize translation error.

Dialog Appearance

Hide file thumbnails (restart required)	Select this option to hide the file thumbnails.

Microsoft 365 Options

Increment file version	Choose when to increment the file version when saving to Microsoft 365 locations. You can choose either **For each save (most up-to-date available for collaboration)** or **On close (minimize storage quota, best for large files)**.

3.11.2. Settings for Optimizing File Import and Export

You can set import and export options for the file types listed below. If you work frequently with any of these file types, you should set their file options to optimize the importing and exporting process for your needs. To modify file settings, access **File** > **Settings** > **File** and then select a file type.

This section contains the following topics:

ACIS

Import	**Convert ACIS bodies to Discovery model components**	Select this option to create components and use the name of the ACIS bodies for the components. If you deselect this option, then the ACIS structure is used in the model tree. **Split body name by the following characters** to insert the character between the part name and the body name, when the names are different. For example, with the character set to a period, the name would be *part.body*. If you deselect this option, then the component name of imported components and solids are the same in the model tree.

AutoCAD

Import	**Model Space units**	Select how to manipulate the unit of an imported AutoCAD file since these files do not have units saved in them. Select **Automatic** to recognize the units in which the file was saved and import the model in same unit scale but in current session units. For example, If the model contains a 1 inch line saved in AutoCAD and the current Discovery application session is in mm, then the line will be shown as 25.4 mm in the application. Select any of the specific units if the model was saved in AutoCAD in that unit. Then import the model in the current session units.
	Insert polyface mesh, 3D face and solid entities into 3D as	Select how to import these entities as 3D into the application.
	Insert polyface mesh, 3D face and solid entities onto drawing sheet as	Select how to import these entities onto drawings sheets.
	Insert hatch entities as	Select the behavior for hatch entities.
	Center imported data	This is useful for AutoCAD files that have models positioned outside of the application's modeling range (+/- 1km). The center of the bounding box for the imported data is moved to world origin coordinates (0, 0, 0). **ON** by default.
	Allow default code page conversion	This setting must be **ON** to support text entities and table names that require a code page conversion. The latest version of Teigha has removed the need for this but the current version of RealDWG still needs it.
	Associate dimensions to geometry	Import associated dimensions.
	Substitute font	Select a font to use when the original font if the DXF file is missing.
Export	**Version**	Select the AutoCAD version to set the format of exported designs.
	Save as black and white	Select this option to remove color information from exported designs.

	Associate dimensions to geometry	Select dimensions to associate with adjacent geometry.
	Include drawing format	**On** by default.
	Scale output 1:1	**Off** by default. Set the output of your AutoCAD files to a 1:1 ratio.
	Export solids as	Choose either **Solids** or **Hidden line curves**.

CATIA

Import	**Trim control points outside face boundaries**	Select this option to perform surface optimization on CATIA files when they are imported. **ON** by default.
	Map CATIA geometric sets to Discovery groups	Select this option to create Discovery application groups from CATIA Geometric Sets.
	Import CGR file as	Select this option to specify how to import the faceted data. Specify options for **Connected facets**, **Simple facets**, and **Solid/surface body**. When facets are connected, the imported model will share facet vertices and the imported bodies will be watertight as possible. Simple facets do not share any vertices in the imported body, between adjacent facets. Select **Merge faces** to seamlessly merge one or more faces. This reduces the overall model size.
	Expand assemblies	When this option is enabled, opening a CATIA file launches the Assembly Browser, from which you can choose to load the entire assembly, a sub-assembly, or a part.

CREO Parametric

Import	**Import quilts**	Select this option to import quilts, a collection of individual surface bodies.

ECAD

Import	**Layout Geometry**	Select this option to import the layout geometry.
	Layer Topology	Select this option to import the layer topology.
	Create solderballs as	If **Layer Topology** is selected, you can choose to **Create solderballs as** either **Bounding Box** or **Cylinders/Spheroids**.

Fluent

Export	**File**	Select this option to choose the file format between **Per design** or **Per component**.

FMD

Export	**Resolution**	Set the Resolution options to control the size and shape of mesh facets in areas with curvature: **Coarse** and **Custom** use preset values for Deviation and Angle. **Medium** and **Fine** enable the **Deviation** and **Angle** sliders so you can use your own specific values. Deviation is the distance between a chord drawn through a curved cross-section of the model and the curve's most distal point. The angle is the angle that the chord line makes with a line tangent to the cross-section curve.
	Facet maximum edge length	Enter a desired edge length. The value you specify is the maximum value for the edges, however smaller edge lengths may be created. The modeler attempts to meet your desired edge length setting, but in some design scenarios, may create an edge length that is less than what you specify as your facet maximum edge length setting, to best accommodate the design.

IGES

Import	**Import method**	• **Use facewise connection**: Select to stitch everything together and make a solid if possible. The import may fail if the model is corrupt. • **Import faces independently**: Select to import models as a collection of individual surfaces, skipping only the surfaces that are corrupt.
Export	**Export Discovery body as**	Select this option to export the body as: • **Manifold solid b-rep object (MSBO - Type 186)**: MSBO defines a manifold stitched body with all the necessary topology information like shells, faces, loops, edges and vertices. • **Trimmed surfaces (Type 144)**: the boundaries of these surfaces are represented as an unstitched body, by curve on a Parametric Surface Entity (Type 142).

JT Open

Import	**Import polylines**	Select this option to import polylines from JT facet data.
	Import data	Select this option to import **B-Rep** and **Facets** data.
	Import facets data as	Select this option to import facets data as **Connected facets**, **Simple facets** or **Solid/surface body**. When facets are connected, the imported model will share facet vertices and the imported bodies will be watertight as possible. Simple facets do not share any vertices in the imported body, between adjacent facets. Select **Merge faces** to seamlessly merge one or more faces. This reduces the overall model size.
Export	**Export current graphics**	Select this option to export the graphics.

	Resolution	Select this option to specify the export resolution: • **Coarse, Medium, Fine** to use the deviation and angle preset values. • **Custom** to use the **Deviation** and **Angle** sliders to set a custom resolution. Resolution indicates the number of sides of a polygon used to represent a circle. The deviation is the difference in distance between the circle's radius and the polygon's radius. The angle is the angle between the edge of the polygon and a tangent drawn through the circle at the same point. • **Use JTOpen toolkit configuration file** to import/export JT files, which allows you to control common translation parameters using a standard configuration file, the format of which is defined by JT Open Toolkit. Using the toolkit also ensures 100% data compatibility and allows you to have more control over the number of LODs (Levels of Detail) and tessellation parameters for each LOD in an exported JT file. • You can also select a **Configuration file** and specify its **length units**.
	Export part manufacturing information	Select this option to export part manufacturing information.
	Version	Select this option to specify a JT version.

MCNP

Export	**Surface entity tolerance**	Defines the minimum allowable distance between surface entities to consider them distinct.
	Minimum cell ID	Sets the starting index for assigning cell IDs.
	Minimum surface ID	Sets the starting index for assigning surface IDs.
	Minimum transform ID	Sets the starting index for assigning transform IDs.
	Enable logging	Enables output logging to the specified file.

OBJ

Import	**Units**	Select this option to choose a system of units. The file is imported using the same unit system in which it was originally created (which is set as **Automatic**).
Export	**Export current graphics**	Select this option to export the graphics.

	Resolution	Indicates the number of sides of a polygon used to represent a circle. The deviation is the difference in distance between the circle's radius and the polygon's radius. The angle is the angle between the edge of the polygon and a tangent drawn through the circle at the same point. Select options to specify the settings as: • **Coarse** and **Custom** to use the deviation and angle preset values. • **Medium** and **Fine** to use the **Deviation** and **Angle** sliders to set a custom resolution.
	Facet maximum edge length	Enter a value in current model units.

Parasolid

Export	**Version**	Select the Parasolid version to set the format of exported designs.
	Shallow assembly export	Select this option to create an extra instance node for every component in the assembly during export.

Rhino

Import	**Convert Rhino layers to Discovery components**	Select to import layers.
	Import facets data as	Options include: • **Connected facets**:When facets are connected, the imported model will share facet vertices and the imported bodies will be watertight as possible. • **Simple facets**:Simple facets do not share any vertices in the imported body, between adjacent facets. • **Solid/surface body** ◦ **Merge faces** to seamlessly merge one or more faces. This reduces the overall model size.
Export	**Convert Discovery components to Rhino layers**	Select to export layers.
	Version	Select this option to choose a Rhino version.

SketchUp

Import	**Import as**	Choose **Solid** or **Facets**. If **Facets**, enable **Simplify to surfaces** to cleanup geometry by fitting planes, cylinders, and cones.
Export	**Version**	Choose a SketchUp version.

Solid Edge

Import	**Expand Assemblies**	When this option is enabled, opening a *.asm file launches the Assembly Browser, from which you can choose to load the entire assembly, a sub-assembly, or a part.

SOLIDWORKS

Import	**Expand Assemblies**	When this option is enabled, opening a *.sldasm file launches the Assembly Browser, from which you can choose to load the entire assembly, a sub-assembly, or a part.

STEP

Import	**Import method**	Select this option to choose the import method: • **Use facewise connections** to attempt to stitch everything together and make a solid if possible. With this option selected, the import may fail if the model is corrupt. • **Import faces independently** to import models as a collection of individual surfaces, skipping only the surfaces that are corrupt.
Export	**Protocol**	Select this option to choose a protocol to use to format exported designs.

STL

Import	**Units**	Select this option to choose a system of units.
	Import as	Select this option to choose the import options. You can choose: • **Connected facets**: Create a lightweight mesh with full connectivity information between facets. • **Simple facets**: Create a lightweight mesh with no connectivity information between facets. • **Solid/surface body**: Convert STL triangles into full geometry. Select **Merge faces** to seamlessly merge one or more faces. This reduces the overall model size.
Export	**Format**	Select this option to choose the format between **Binary** (default) or **ASCII**.
	File	Select this option to choose the file format between **Per design**, **Per component** or **Per body**.
	Export current graphics	Select this option to export the graphics.

	Resolution	Set the Resolution options to control the size and shape of mesh facets in areas with curvature: **Coarse** and **Custom** use preset values for Deviation and Angle. **Medium** and **Fine** enable the **Deviation** and **Angle** sliders so you can use your own specific values. Deviation is the distance between a chord drawn through a curved cross-section of the model and the curve's most distal point. The angle is the angle that the chord line makes with a line tangent to the cross-section curve.
	Facet maximum edge length	Enter a desired edge length. The value you specify is the maximum value for the edges, however smaller edge lengths may be created. The modeler attempts to meet your desired edge length setting, but in some design scenarios, may create an edge length that is less than what you specify as your facet maximum edge length setting, to best accommodate the design.

TGF

Export	**Object**	Select this option to export the object: • **Per design** • **Per assembly** • **Per part** • **Per body**
	Facet resolution	Set the resolution options: • **Coarse** and **Custom** to use the deviation and angle preset values. • **Medium** and **Fine** to use the **Deviation** and **Angle** sliders to set the resolution of the TGF mesh. Deviation is the distance between a chord drawn through a curved cross-section of the model and the curve's most distal point. The angle is the angle that the chord line makes with a line tangent to the cross-section curve.
	Facet maximum edge length	Enter a desired edge length. The value you specify is the maximum value for the edges, however smaller edge lengths may be created. The modeler attempts to meet your desired edge length setting, but in some design scenarios, may create an edge length that is less than what you specify as your facet maximum edge length setting, to best accommodate the design.
	Extract feature edges	Enter the angle in **Feature angle**.

VRML

Export	**Export current graphics**	Select this option to export the graphics.

	Resolution	Set the resolution options: • **Coarse** and **Custom** to use the deviation and angle preset values. • **Medium** and **Fine** to use the **Deviation** and **Angle** sliders to set the resolution of the VRML mesh. Deviation is the distance between a chord drawn through a curved cross-section of the model and the curve's most distal point. The angle is the angle that the chord line makes with a line tangent to the cross-section curve.

Workbench

Import	**Basic Geometry Options**	Set basic geometry options for importing Workbench application files.
	Advanced Geometry Options	Set advanced geometry options for importing Workbench application files.

Export	**Export to Workbench**	The following options are supported for exporting to Workbench application:
		• Select how to **Merge Beams** during export to **Workbench**.
		  ◦ **Never**: No merging of beam bodies occurs. Each beam remains as a separate part in **Workbench**, regardless of connectivity or cross-section.
		  ◦ **Connected with same profile**: Merges beam bodies only if they are physically connected (share a node) and have the same cross-sectional profile. This reduces part count while preserving meaningful segmentation.
		  ◦ **Disconnected with same profile**: Merges all beam bodies that share the same cross-sectional profile, even if they are not physically connected. Use this to simplify the model further by grouping all identical-profile beams into single parts.
		• Select **Physics definition** to export physics conditions, solver settings, and other simulation setup data to downstream solvers.
		• Select **Material properties** to export material assignments and related simulation settings to downstream solvers. For more information, see Geometry Transfer on page 66.

	Always use Discovery's reader when possible	Set the **Always use Discovery's reader when possible** option depending on whether you want the application to ignore the Ansys CAD Configuration Manager's (CCM) settings when opening a document. File formats that can be imported fall into these categories: **1.** Formats that are *never* routed through Ansys Part Manager (IGES, STEP, STL, VRML, and so on). These formats always use the application's reader. **2.** Formats that *must* be read through Ansys Part Manager (CATIA, CREO Parametric, and so on). Workbench application preferences always apply to these formats. **3.** Formats that are readable by *both* Part Manager and the Discovery application's reader. Enabling the option forces any format from category 3 to be routed through the application's interoperability functions. When the option is disabled, each format's behavior is determined by its setting in the CCM.

3.12. Customize

Customize to show and hide tabs, create custom keyboard shortcuts for tools, and add customized interactive tours to the **Home Page**.

Ribbon Tabs

You can show and hide tabs in the ribbon. Changes you make take effect immediately.

Show	Select to expose the tab in the ribbon.
Hide	Select to hide the tab in the ribbon.
Reset	Select to return to the default tab order and exposure settings.

Click and drag a tab to change its order relative to the other tabs in the ribbon. Some tabs remain exposed, such as the **Design** and **Simulation** tabs, and cannot be hidden.

Shortcuts

You can create custom keyboard shortcuts for tools in the ribbon, view arc, results arc, and the sketch grid tool cluster in the graphics window.

Add Shortcut	Select to create a custom shortcut.
Edit Shortcut	Select to modify a shortcut.
Delete Shortcuts	Select to remove a shortcut.
Shortcut Buddy	Select to obtain a list of available shortcuts. A table listing all shortcuts is displayed. At the bottom of the table, you can click **Show Keyboard View** to view the shortcuts in a different format.

See Creating Custom Keyboard Shortcuts on page 57 for more information.

Interactive Tours

In the application, you can access several modeling and simulation interactive tours that will walk you through an entire workflow. In addition, you can load customized interactive tours that you or your organization have created. Browse to the folder that contains the customized interactive tour resources. Access the custom tour from the **File Menu > Home Page** under the **Custom Tours** tab.

> **Note:** The folder must contain an XML file and optionally a resx file (if the text strings are not in the XML). The folder can also include images in PNG and JPEG format and animations in GIF format. Additionally, geometry files must reside in the same folder if they are referenced in the XML.

> **Tip:** Copy an interactive tour's XML and corresponding resx files from the application's installation folder (~\Discovery\overlay\tutorials and Discovery\overlay\localization\tutorial_strings) to your newly created folder. Modify the XML and resx files as needed.

3.13. Add-Ins

Ansys-Discovery Connector, **Ansys Discovery 2026 R1 Integration** and **Electronics Database** are used for transferring data from the Discovery application to the Ansys Workbench application. For more information, see Geometry Transfer on page 66.

3.14. Extensions

Select 🗁 to select a .dstool to load at startup.

Once installed, enable or disable specified .dstools as needed.

> **Note:** Adding a new .dstool or enabling/disabling an existing .dstool requires a restart of the Discovery application.

Enable	Enable or disable specified .dstools as needed.
Location	Specified file location of the .dstool.

3.15. About Discovery

About Discovery provides copyright, software build and version, and (if applicable) Quality Assurance Services information for the Discovery application.

4: Selecting

You can select vertices, edges, planes, axes, faces, surfaces, rounds, bodies, solids, and components in 3D. In 2D, you can select points and lines. You can also select circle and ellipse centers, the midpoints of lines and edges, and the internal points and end points of splines.

You can select components and other objects in the design tree and use the Power Selection to select objects in the same part that are similar or related to the object currently selected.

Objects that cannot be selected are dimmed in the Design window.

4.1. Select Tool and Modes

You can click the arrow on the Select tool to use the following optional modes:

- Using Box: Click and hold the mouse button while drawing a box in the Design window. If you draw the box from left to right, all objects fully enclosed within the box will be selected. If you draw the box from right to left, all objects touching the box will be selected.
- Using Lasso: Click and hold the mouse button while drawing a freeform shape. All objects fully enclosed by the shape will be selected.
- Using Polygon: Click and move the mouse to draw a line, then click again to draw the next connecting line, so that you create a polygon shape around the area you want to select.
- Using Paint: Click and hold the mouse button while highlighting adjacent faces and edges. All edges and faces that you move the mouse over will be selected. Release the mouse button to finalize your selection.
- Using Boundary: Select faces or edges that define a boundary, click the **Select Seed** tool guide, and click any object within the boundary. All objects from that seed object to the boundary are selected.
- Select Components: In this selection mode, as you hover over geometry, when you are over an object that is in a component, that component will be pre-highlighted. This mode is set on a per window basis. It can be *On* in one window and *Off* in another.

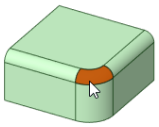
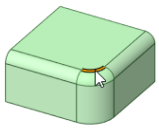
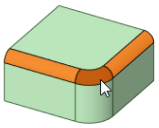
No matter which mode you use, the objects that will be selected are highlighted to preview your selection. You can use the Selection filter to control what gets selected. To select or deselect all available filters, select the **All** check box.

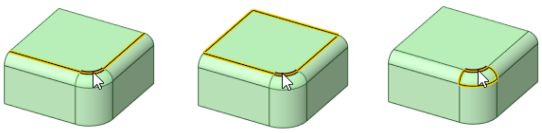
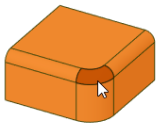
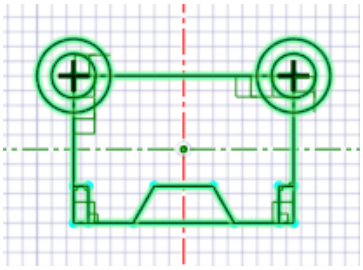
4.2. Using the Select Tool

1. Select the Select tool from the Edit ribbon group.
2. Hover over the vertices, edges, faces, bodies in the workspace to preview the selectable items in your design.
   If multiple objects occur at your cursor location, use the scroll wheel or arrow keys to preview each one.
   Click to select a vertex, edge, face, or body in 3D, or to select a line or point in 2D.
   The most commonly used selection methods are:
   - Click to select an object - a vertex, edge, face, or body in 3D, or to select a line or point in 2D.
   - Double-click to select an edge loop. (Double-click again to cycle through alternate loops.)
   - Triple-click to select a solid.

- Drag (or select **Using Box** from the Select tool menu) to create a selection box.
- Use **Ctrl+A** to select all similar objects, such as bodies, faces, edges, or points on the same solid or surface part. For example, if you have a sketch curve selected when you press Ctrl+A, then only sketch curves will be selected.
- Use **Ctrl**+click and **Shift**+click to add or remove items from the selection set.
  - Use **Ctrl**+click to add or remove one item from the selection set. Use **Shift**+click to add everything between your first click and the **Shift**+click to the selection.
  - **Ctrl** with box-selection toggles the selection, while **Shift** with box-selection adds to the selection.
  - You can add or remove items both in the workspace and on the Structure tree.
  - You can also press **Ctrl** and drag to add the items within the selection box to the selection.
- Use **Alt**+click to create a secondary selection set.

  Use the Alt key with any of the other selection techniques (double-click, triple-click, Ctrl, Shift) to create the second selection set. Alternate selections are shown in blue, and you can use them to guide the actions of the Pull and Move tools.

To select:	**Do this:**
Click to select an object.	• Click once to select a face:  • Click once to select an edge: 
Double-click to select connected faces:	Double-click to select and cycle through chains of similar, connected faces:  Double-click a face to select chains of rounds or tangent faces.

To select:	Do this:
All the edges around a face or closed loop	Double-click to select and cycle through chains of connected edges (edge loops):  When you repeatedly double-click outside surface edges, the selection cycles through open loop, tangent chain, and face loop, then repeats through that list. You can also right-click the edge or line and choose one of the loop options in the Select menu.
Contiguous edges or faces	Click one face or edge, then Shift+click another face or edge to select all the faces or edges between the two.
A solid or surface body	Triple-click to select the solid, or right-click the solid and select **Select > Body**. 
An entire sketch	Triple-click the sketch. 
A component	Right-click the component and select the **Select > Component** option. This option is available only when you right-click an object in the active component.
All objects of same type as selected object	Select an object and press **Ctrl+A.** to select all similar objects, such as bodies, faces, edges, or points on the same solid or surface part. For example, if you have a sketch curve selected when you press Ctrl+A, then only sketch curves will be selected.
All the objects in the active component (except layout surfaces)	Click **Select All** in the Select tool menu, or right-click and select **Select > Select All** from the context menu. The types of objects selected depend on whether you are in Sketch, Section, or 3D mode.
A lightweight component	Check the **Lightweight Components** box in the Options panel. Then right-click and select **Select > Component**.

To select:	Do this:
Objects that are behind other objects ("query selection")	Hold the **Ctrl** key and roll the mouse wheel.

Note:

- If any object was part of a group selection used to perform an action, the other parts of the group are highlighted when that object is selected. Click again to select the entire highlighted group.
- When selecting an object in a view on a drawing sheet, you can only select those objects that are on the cross-section plane, or that are within the boundary of a detail view. Box selecting in Sketch mode selects only sketch lines. Box-selecting in Section mode selects only section lines.

4.3. Using the Selection Context Menu

The **Select** context menu contains options for selection based on the object(s) selected. The available options include:

- Component
- Body
- Block (Meshing only)
- Component Blocks (Meshing only)
- Blocking (Meshing only)
- Inverse Selection
- Protrusion
- Depression
- Pattern Member
- All Pattern Members
- Pattern Edges
- Select Tangent Edge Loop
- Select Edge Loop One/Select Edge Loop Two
- Select All
- Clear Selection

4.4. Clearing a Selection

Click any empty space in the Design window or select **Clear Selection** from the **Select** tool menu.

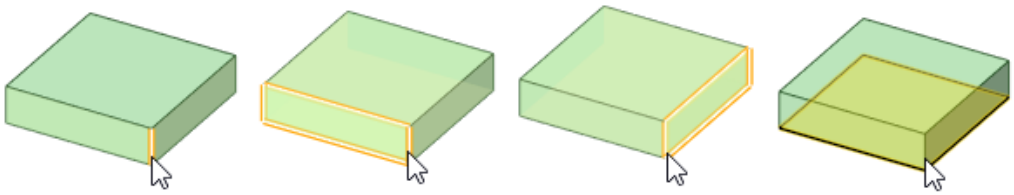
4.5. Reverting Your Selection

Click **Revert Selection** in the status bar to go back to your previous selection.

This tool is especially useful when you accidentally clear your selection or add the wrong object and want to go back to your previous selection. You can find this tool next to the **Selection filter** in the status bar.

4.6. Selecting by Rolling the Mouse Wheel

- **Coincident objects**: Objects often appear at the same location in 2D, such as a vertex and end point of a line that are located at the same point in space. When selecting, make sure you have selected the correct object by rolling the mouse wheel without moving the mouse.
- **Edge shared by two surfaces or solids**: Hovering over the edge shades the face that will be affected by an action to the edge if you select it. Roll the mouse wheel to switch between the two faces.
- **Vertex shared between two edges**: Hovering over a vertex shades the edge that will be affected by an action to the vertex if you select it. Roll the mouse wheel to switch between the edges.
- **Face of a solid when only the edge is displayed** (such as in a drawing sheet view): You can select the face by turning the mouse wheel. The edge becomes a slightly thicker line when the face is highlighted.
- **Edge loops**: If **Select edge loops using the mouse scroll wheel** is selected in the Advanced options, scrolling through edge loop choices pre-highlights the selection. Click the pre-highlighted edge loop to select it.



The up and down arrow keys work the same as the mouse wheel. Hover over the element you want to select, and press the up or down arrow keys to "scroll" through the possible selections. This is useful if you are on a laptop or when you use a mouse that does not have a scroll wheel.

For more information about selecting facets and the related tools, see Selecting Facets.

4.7. Select Tool Guide

The following tool guide is available:

	The **Select** tool guide is active by default. This tool lets you click, double-click, triple-click, Ctrl+click, Shift+click, and Alt+click to select items.

4.8. Filtering the Selection

Click the **Selection Filter** in the status bar to open the filter options. The options are:

- Smart
- All
  - Bodies

- Faces
- Edges
- Sketch Curves
- Annotations
    - Notes
    - Dimensions
    - Other
- Planes
- Axes
- Vertices
- Points
- Images
- Faceted Bodies
- Blocks (SCDM only)
    - Volumes
    - Faces
    - Edges
    - Vertices

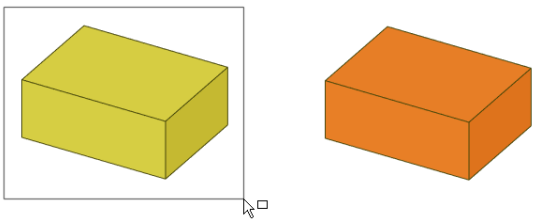
- Lightweight Components
- Transparent Objects

The **Smart** option is enabled by default and whenever you switch to another tool.

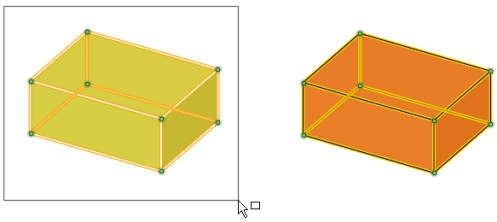
When **Smart** is enabled, all filters are checked and cannot be unchecked. Within the defined area, only the highest checked item in the filter list will be selected. This differs between Design Windows and Drawing Windows as follows.

- In a Design Window: Body, Face, Edge, Sketch Curve, Annotation, Plane, Axis, Point, ...
- In a Drawing Window: Annotation, Sketch Curve, Body, Face, Edge, Plane, Axis, Point, ...

The images below show only the body being selected in the Design Window when **Smart** is enabled.



Disable **Smart** to select all object types checked in the filter list. The images below show that all items are selected in the Design Window when **Smart** is disabled.



All is a shortcut for enabling/disabling all of the filter items.

When you switch to another tool, the filter selections are remembered but the **Smart** option is selected. If you deselect the **Smart** option, the previously selected options remain selected while the previously deselected options remain deselected.

The following Box selection options are available in the **Selection Filter**.

- **Automatic** selects faces and connected hidden faces
- **Visible** selects visible faces only
- **Through** selects visible and hidden faces

4.9. Using Box Mode

The **Using Box** mode is found in the menu under the **Select** tool in the **Edit** ribbon group on the Design tab, and also in the radial menu. You can access this menu while almost any tool is active. For example, if you want to change selection modes while the Pull tool is active, just click the arrow under Select. The Pull tool will remain active.

From the context menu, you can also choose **Select > Using Box**.

This mode enables you to draw a box in the Design window to select objects for most tools. As you draw a selection box, objects are highlighted (after a slight delay) to help you see exactly what will be selected.

Draw the box from left to right to select only the objects that are fully contained within the box. The box appears as a solid line.

Draw the box from right to left to select everything the box crosses, even if it isn't fully enclosed by the box. The box appears as a dashed line.

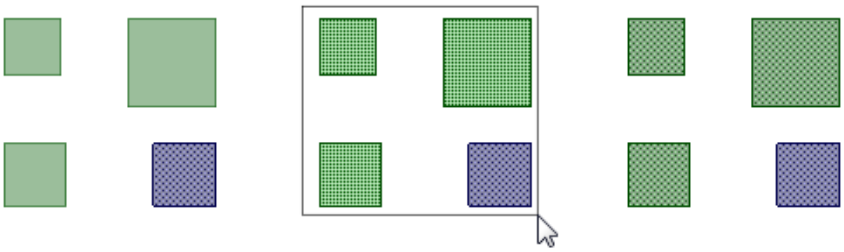
4.9.1. Selecting Objects by Drawing a Box

1. Click the arrow beneath the  **Select** tool and click **Using Box**.
2. (Optional) Use the **Selection Filter** in the status bar to control what will be selected.
3. Start drawing a rectangle in the Design window. Click and hold the left mouse button at one corner of the area you want to box-select.
4. Hold the mouse button and draw a rectangle in the Design window.
   - Draw the box from left to right to select only the objects that are fully contained within the box. The box appears as a solid line.
   - Draw the box from right to left to select everything in the box, even if it isn't fully enclosed by the box. The box appears as a dashed line.

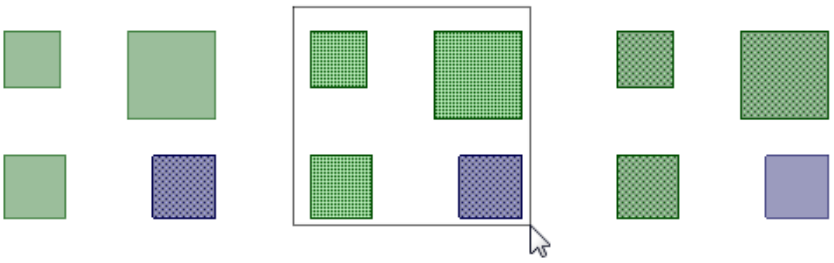
   The objects that will be selected are highlighted as you draw the box.
5. Release the mouse button to complete the box.
   The objects are selected when you release the mouse button.

4.9.2. Changing Your Selection

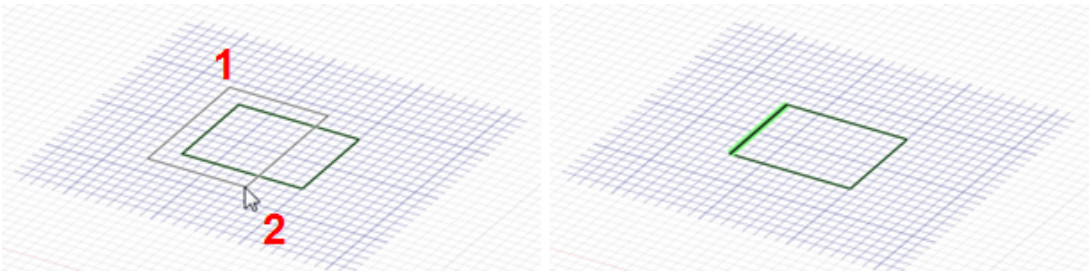
Holding **Shift** while box-selecting adds to the selection. In the first image, the blue box is selected. After holding **Shift** and box selecting, all of the boxes are selected:



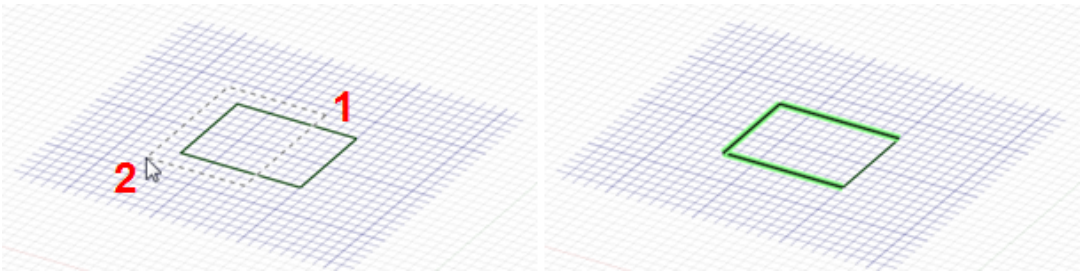
Holding **Ctrl** while box-selecting toggles the selection. In the first image, the blue box is selected. After holding **Ctrl** and box selecting, the blue box is deselected and the green boxes are selected:



4.9.3. Box Selection Examples



Box-selecting from left to right



Box-selecting from right to left

4.10. Using Lasso Mode

The **Using Lasso** mode is found in the menu under the **Select** tool in the Edit ribbon group on the Design tab, and also in the radial menu. You can access this menu while almost any tool is active. For example, if you want to change selection modes while the Pull tool is active, just click the arrow under Select. The Pull tool will remain active.

This select mode enables you to draw a free form shape in the Design window to select objects for most tools. As you draw, objects are highlighted (after a slight delay) to help you see exactly what will be selected.

4.10.1. Selecting Objects by Drawing a Freeform Shape

1. Click the arrow beneath the **Select** tool and click **Using Lasso**, or click and hold in the Design window to activate the radial menu and then select **Lasso**.
2. (Optional) Use the **Selection Filter** in the status bar to change the type of objects that will be selected.

   The **Smart** option is enabled by default and whenever you switch to another tool. Select **Smart** to select the highest checked item in the list that is found in the boxed, lassoed, or painted area. Drawing a box or freeform shape selects objects in the following order:

   - In a Design Window: Body, Face, Edge, Sketch, Annotation, Plane, Axis, Point
   - In a Drawing Window: Annotation, Sketch, Body, Face, Edge, Plane, Axis, Point

   Deselect **Smart** to select all the selected object types. Click to select or clear the types of objects you want to select.
3. Start drawing a shape in the Design window. Click and hold the left mouse button at the spot where you want to start drawing the shape.
4. Hold the mouse button and draw a shape in the Design window.

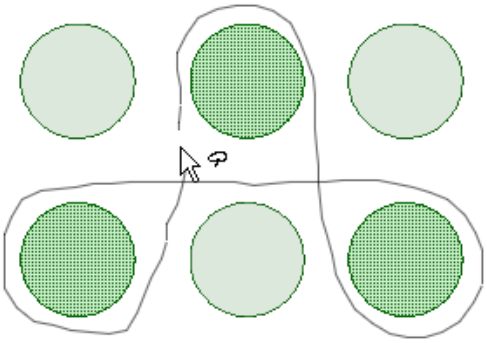
   You can make multiple loops, like drawing a figure 8.
5. Release the mouse button to complete the shape.

   The shape will be closed by a straight line between your starting and ending points when you release the mouse button.

   Everything except inserted images is selected when you release the mouse button.

> **Note:** You can activate Lasso mode by holding the Alt key while the Select tool is active. This hotkey only works when you are using the Select tool, and not when another tool is active.

4.10.2. Lasso Selection Example



Selecting three surfaces by drawing multiple loops

4.11. Using Polygon Mode

The **Using Polygon** mode is found in the menu under the Select tool in the Edit ribbon group on the Design tab, and under the Select tool in the Edit Group on the Sheet Metal tab. You can also right-click in the design window and

choose **Select > Using Polygon**. This selection mode enables you to select all objects within a polygon you create. Each time you click, a new starting point for the next line is created. You control the length and angle of each polygon line you create.

Use this mode to draw around a specific geometry or model feature you want to select, for example, a protrusion of other geometry you may want to clearly identify. Using a polygon shape for selection can apply when you are, for example, trying to select an irregular or angular shaped part, or assembly of parts.

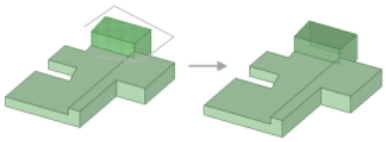
4.11.1. Selecting Objects Using a Polygon

1. Select the **Using Polygon** tool.
2. (Optional) Use the **Selection Filter** in the status bar to change the type of objects that will be selected.

   Deselect **Smart** to select all the selected object types. Click to select or clear the types of objects you want to select.
3. Click points to draw a polygon around the model feature that you want to select.

   As you draw the polygon lines, selected objects highlight as they are pre-selected.
4. Double-click or cross the polygon lines to finalize the selection.

4.11.2. Polygon Selection Example



Creating a polygon to select around a specific geometry.

4.12. Using Paint Mode

The **Using Paint** mode is found in the menu under the **Select** tool in the Edit ribbon group on the Design tab, and in the radial menu. You can access this menu while almost any tool is active. For example, if you want to change selection modes while the Pull tool is active, just click the arrow under Select. The Pull tool will remain active.

This select mode enables you to select faces, edges, and sketch curves in the Design window by "painting" faces and edges that are adjacent to each other. As you paint, objects are highlighted to help you see exactly what will be selected.

4.12.1. Selecting Objects by Painting

1. Click the arrow beneath the **Select** tool and click **Using Paint**, or click and hold in the Design window to activate the radial menu and then select **Paint**.
2. (Optional) Use the **Selection Filter** in the status bar to change the type of objects that will be selected.

   The **Smart** option is enabled by default and whenever you switch to another tool. Select **Smart** to select the highest checked item in the list that is found in the boxed, lassoed, or painted area. Drawing a box or freeform shape selects objects in the following order:

   - In a Design Window: Body, Face, Edge, Sketch, Annotation, Plane, Axis, Point

- In a Drawing Window: Annotation, Sketch, Body, Face, Edge, Plane, Axis, Point

Deselect **Smart** to select all the selected object types. Click to select or clear the types of objects you want to select.

3. Click the first object you want to select and hold the mouse button.
4. Drag the mouse onto adjacent faces or edges that you want to add to the selection.

   The objects that will be selected are highlighted as you move over them with the mouse.

   If you hold **Ctrl** you can move the mouse over an object without selecting it, or move the mouse over a selected object to deselect it.
5. Release the mouse button to complete the selection.

   The objects are selected when you release the mouse button.

4.13. Using Boundary Mode

The **Using Boundary** mode is found in the menu under the **Select** tool in the Design tab Edit group. This mode enables you to select all objects within a boundary.

4.13.1. Selecting Objects Using a Boundary

1. Select the **Using Boundary** tool from the **Select** dropdown menu in the Edit ribbon group on the Design tab.

   The **Select Bounds** tool guide is enabled by default.
2. (Optional) Use the **Selection Filter** in the status bar to change the type of objects that will be selected.

   Deselect **Smart** to select all the selected object types. Click to select or clear the types of objects you want to select.
3. Select the faces or edges you want to use to define a boundary.

   You cannot select a combination of faces and edges.

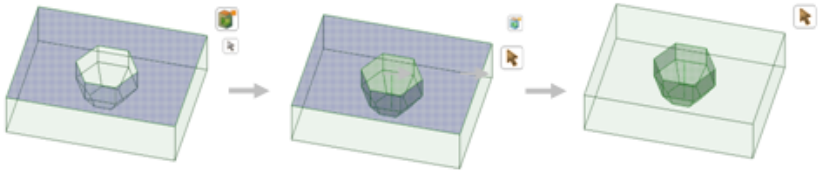
   The **Select Bounds** tool guide is sticky, so you do not have to hold the Ctrl key to add to your selection. You can click in white space to clear your selections.

   The objects remain selected as secondary references in case you need to add or subtract from the boundary.

   > **Note:** You can select bounds (Ctrl+alt+selection) and seeds (Ctrl+selections), then click the boundary tool under the **Selection** dropdown menu to perform the boundary selection operation without actually entering the Select tool. The **Select Bounds** tool guides does not display.
4. Click the **Select Seed** tool guide.
5. Click any object within the boundary to use as the seed object.

   You can select multiple seeds. All objects from the seed object to the boundary are pre-highlighted for selection.
6. Click the **Select** tool to finalize the selection.

4.13.2. Boundary Selection Example



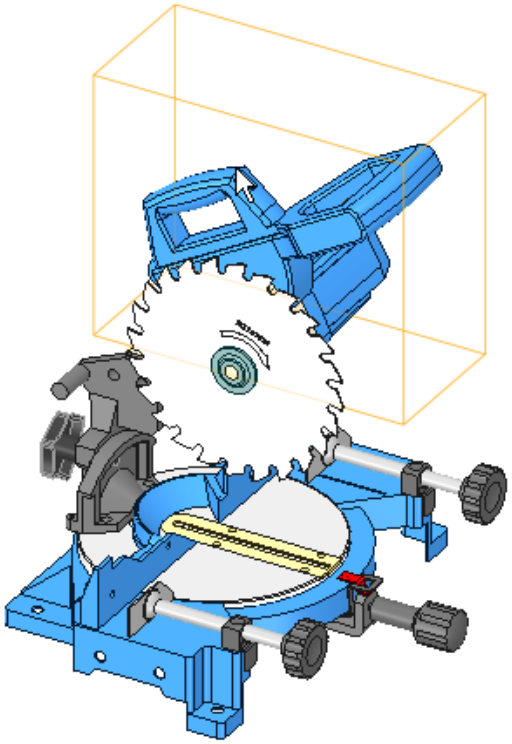
Select faces to define a boundary and then select the seed object. All objects from the seed object to the boundary are selected.

4.14. Select Components Mode

The **Select Components** mode is found in the menu under the **Select** tool in the Edit ribbon group on the Design tab. You can access this menu while almost any tool is active. For example, if you want to change selection modes while the Pull tool is active, just click the arrow under Select. The Pull tool will remain active.

In this selection mode, as you hover over geometry, when you are over an object that is in a component, that component will be pre-highlighted.

Pre-highlighting while box-selecting shows just the component bounding boxes. Faces, edges, etc. are not pre-highlighted.



4.15. Advanced Selection

The Discovery application provides you with advanced selection tools that allow you to create named selections and filter and select similar objects using power selection. You can use layers to group entities based on visual properties and review and modify properties of the model.

This section contains the following topics:

4.15.1. Named Selections and Queries

Named selections are used to group geometric entities together. Named selections can consist of primary selections, secondary selections (Alt-select), or a mix of both. These groups of geometric entities can be used for easier selection, to apply physics conditions, transfer to downstream applications, etc. You can also create queries for obtaining a selection set which can be used as a named selection.

The **Named Selection and Query** tab in the **Advanced Selection** window contains tools and options for creating and modifying named selections.

This section contains the following topics:

4.15.1.1. Creating a Named Selection

You can create and edit Named selections using the options in the **Named Selection and Query** tab in the **Advanced Selection** window.

This section contains the following topics:

To create a named selection:

1. Select any set of geometric entities in your model.
2. Click **Named Selection and Query** in the **Status bar** to open the advanced selection panel. In the **Named Selection and Query** tab, click **Create a named selection from the selected objects**, or use the shortcut **Ctrl+G**.
3. The new named selection appears in the named selections list with a default name. Right-click the default name to rename it.

   You can also sort the list of named selections alphabetically in ascending or descending order.
4. Hover over the named selection to highlight the entities in the named selection and click the named selection to select them.

4.15.1.2. Overlapping Named Selections

After creating your named selections, you can check to see if there are any overlapping selections in the list and you can repair them.

1. Click **Detect Overlaps** to detect overlapping selections.
   In top to bottom order, the application takes a named selection and checks if any of the entities are used in other named selections. Named selections are created for the overlapping selections.
2. Click **Remove Overlaps** to remove overlapping selections.
   In top to bottom order, the application takes a selection and checks if any of the entities are used in other named selections. If they are, the overlapping entities will be removed from the lower named selections. If no geometric entities exist in a named selection after going through all the named selections, the named selection will be deleted.

4.15.1.3. Modifying Named Selections

After creating a named selection, you can modify it by renaming, deleting, replacing, and adding to it. You can also separate, merge, subtract, and hide or show entities. To better organize your named selections you can group them into folders.

To modify the entities in a named selection, right-click a named selection in the list and choose one of the options in the context menu. The table below describes each of the options.

Option	Description
Separate	Select the named selection containing multiple geometric entities that you would like to separate. Each geometric entity will be placed in its own group. The new group names are based on the parent named selection. For example, "Group" will be separated into "Group.1", "Group.2", etc.
Merge	Select two or more named selections to merge. They will be merged into a single named selection, taking the name of the last named selection selected.
Add	Select any set of 3D entities. You may also select a named selection in the list to add to another named selection.
Subtract	Select any entity that is part of a named selection to subtract it from the named selection.
Replace	Select any set of 3D entities to replace a named selection. When replacing a named selection, you can only select one item in the **Named Selection** list to replace it with another named selection. However, you can select multiple entities in your geometry and use them to replace a named selection.
Create Folder	Select a set of named selections from the list and group them in a folder for easier access.
Ungroup	Select a named selection folder or an individual selection from a folder to ungroup them.
Delete	Select the named selection you would like to delete. If all the entities that comprise a named selection are removed from the design, the named selection is automatically deleted.
Rename	Select the named selection to change the default name. You can also double-click to rename it.
Hide	Hide entities in the selected named selection(s). Note that if the named selection contains only faces, then only faces on entities that are part of the named selection will be hidden.
Hide Others	Hide other named selection to focus on entities in the selected named selection(s).
Show	Show entities in previously hidden named selection(s).

4.15.1.4. Using the Query Editor

You can use the **Query Editor** to create queries for obtaining a selection set.

> **Important:** The **Query Editor** operates only on active components and visible bodies, hidden bodies are not processed by default. However, when a query is created on a Named Selection criterion using the equal to (=) operator and the scoped entity within the named selection is hidden, the entity will still be included in the query.

To create a query:

1. Click **New** to create a new query, or continue editing the current query.

2. Specify a name for the query, or retain the default name.
3. Enter data in the current row to define a selection set.

   As needed, enter data in subsequent rows to further refine the selection. Use **Preview** to view the current selection based on the data.

   Specify the following:

Option	Description
Action	Choose to **Add**, **Remove**, or **Filter** selections. • **Add**: Adds the selections defined in the current row to those in the previous row. • **Remove**: Removes the selections defined in the current row from those in the previous row. • **Filter**: Generates a subset of the selection defined in the previous row.
Type	Choose **Body**, **Face**, or **Edge** as the type of entities to select.
Criterion	Choose the criterion for selection based on the entity type selected.
Operator	Choose the appropriate operator: • = • != • < • <= • > • >=
Value	Specify the value based on the entity type and criterion selected. This could be a selection from the dropdown or a numerical or text value.

4. Click **Save** to save the query as a named selection in the **Named Selection and Query** tab of the Advanced Selection panel.

   If you make changes to a saved query, click **Cancel** to revert to the last saved state.
5. Hover over the named selection in the **Named Selection and Query** tab to highlight the entities in the named selection and click the named selection to select them.

> **Note:** Query-based selections are not automatically updated when the scope changes or new bodies are added to the Model tree. The selections that can be updated are indicated in the **Named Selection and Query** tab. Right-click and select **Update** to update the selection based on the changes.

To edit an existing query, double-click the query or right-click it in the **Named Selection and Query** tab and select **Edit** from the context menu to open the **Query Editor**.

> **Known Limitation:** The **Query Editor** is not available when History tracking is active.

4.15.2. Power Selecting Objects

Power Select allows you to select geometric entities in the same model that are similar or related to the entities currently selected. To search for similar objects:

1. Select any object.
2. Click **Power Selection** in the **Status bar** to open the advanced selection panel.
3. (Optional) In the **Power Selection** tab of the panel, select **Search all bodies** to search all bodies.
4. (Optional) Select a category from the dropdown list to show only those objects with the selected relationship.

   Select **All types** to display all the geometry related to your selection.

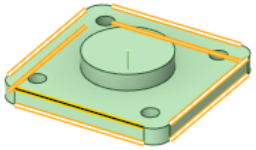
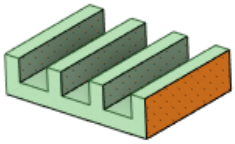
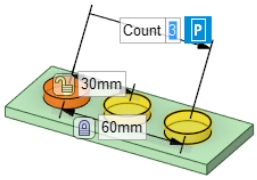
   After selecting objects with the selected relationship, you can also click **Refresh** to refresh the search results using the current selection and then select objects based on another applicable category.
5. (Optional) If you disable **Automatically calculate power selection results** in the **General - Selection** settings (**File Menu > Settings > General > Selection**) then you must click **Find** to search for related objects in your model. Otherwise, the search results are automatically displayed.
6. (Optional) Set the **Power selection relative tolerance** in the **General - Selection** settings to find items within a tolerance of the target size.
7. (Optional) For selections where you can select similar objects based on a range of values for the size (or area, volume, etc.), you can change the values for these parameters in the search results.
8. Click any number of the related objects in the list to add them to the current selection.

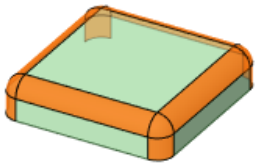
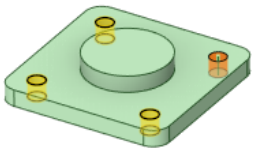
   You can **Ctrl+** select multiple objects.

4.15.2.1. Selection Categories

Selection categories are presented in the table below.

Category	Description	Example	Selection types
Beams	Beams based on profiles or length		Beams based on the profile of the currently selected beam. Beams with lengths <=, =, or >= the length of the currently selected beam.
Bodies	Bodies based on volume compared with the currently selected body.	This power selection category is especially useful when you need to find and delete multiple, small items in a design, such as fasteners.	All bodies with volume <=, =, or >= the volume of the currently selected body.
Coaxial faces	Objects with the same axis		Coaxial hole Coaxial protrusion Coaxial cylinder Coaxial surfaces
Components	All instances of the selected component.		Components

Category	Description	Example	Selection types
Edges	Edges that are like the currently selected edge		Edges with same length Edges with same length and same orientation Edges with same length in same face Surface edge loop Hard sheet metal edges
Features	Groups of faces that form defined regions on a model.		Protrusion Depression Inner faces Enclosed by
Midsurface Face Thickness	Midsurface faces with the same thickness (including zero)	Finding zero thickness faces and surface bodies allows you to verify that thickness has been assigned to all faces and surface bodies in the model.	Faces and Bodies
Offset faces	Faces that have an offset relationship defined		All offset baseline faces All offset faces Faces with same offset All coincident faces
Patterns	Pattern members, entire patterns, or recognized patterns		Pattern member All pattern members Recognized pattern **Note:** If the pattern does not have a pattern relationship in theDiscovery application, you must hold **Alt** and select the face that contains the pattern members in order to select a recognized pattern.

Category	Description	Example	Selection types
Rounds	Rounds and chamfers		Equal radius rounds All, Concave, Convex Rounds within a range of values, including equal or smaller radius rounds Equal or smaller-sized chamfers Variable radius round faces and chains of faces even if they are composed of a mix of constant and variable faces
Same Color	Faces and faceted bodies of the same color		Faces and faceted bodies
Same Curve Type	Curves of the same color, type, and length		Lines/curves that are the same type. For example, you can select an arc to find all other curves that are arcs.
Same Hole Dimensions	Standard Holes with the same Hole Dimensions		Standard Holes
Same size	Faces that have the same radius or area		Equal radius cylinder Equal radius hole Equal radius protrusion Faces with the same area Holes equal to or smaller than Surface holes equal to or smaller than
Suppressed Items			All bodies suppressed for physics

4.15.3. Layers

Layers can be thought of as a grouping mechanism based on visual characteristics. Visual characteristics include visibility, color and line style. Layers are managed in the Layers panel and accessed and modified with the **Layer** tool in the Display tab of the Style group.

Layers are especially useful when you want to hide annotation planes.

Sheet metal parts have a set of default layers when the part is in an unfolded state. They are:

- Dimension
- Bend Lines Up
- Bend Lines Down
- Bend Dimensions
- Forms

4.15.3.1. Working With Layers

Click **Layers** in the Style ribbon group of the Display tab to open **Layers**. Select a layer and right-click to access the context menu to do any of the following:

Sort A - Z **Sort Z - A**	Sort layers alphabetically.
Add Layer	Create a new layer. This layer becomes the active layer. Any newly created objects are automatically placed on the active layer.
Duplicate Layer	Duplicate a layer.
Activate Layer	Select a layer to be the active layer. Any newly created objects are automatically placed on the active layer.
Assign to Layer	Select one or more objects to assign to the layer.
Show	Show the layer.
Hide	Hide the layer.
Hide Others	Hide the other layers.
Select Objects	Select the object
Rename	Rename the selected layer. **Note:** Layer0 cannot be renamed.
Delete	Delete the selected layer. **Note:** Layer0 cannot be deleted.
Remove Empty Layers	Deletes any layers that are empty.

4.15.4. Opening Properties

When you click a component, surface, or solid within the design window or the model tree, its properties will appear in the **Properties** tab of the advanced selection panel. From there, you can adjust its values. To display object properties:

1. Click **Properties** in the **Status bar** to open the advanced selection panel.
2. Select an object in your model to see its properties displayed in the **Properties** tab of the advanced selection panel.

4.15.4.1. Modifying Object Properties

1. Select an object in the model tree or the design window.
2. Edit the properties values in the **Properties** tab.

4.15.4.2. Modifying Analysis Properties

The **Properties** tab lets you set shared topology on components when transferring to the Workbench application.

1. Select a top-level assembly component in the model tree.
2. In the **Properties** tab of the advanced selection panel, expand **Transfer Sharing**.
3. Open the dropdown menu and choose: **None**, **Group**, or **Share Topology**. If choosing **Share Topology**, you can select to **Merge**.

4.15.4.3. Creating Custom Component Properties

1. Select a component in the model tree.
2. Expand **Component Properties** in the **Properties** tab.
3. Click the **<New Property>** field to add a new component property.
4. Open the dropdown menu and choose: **Number**, **Boolean**, **Text**, or **Date** and choose an initial property value in the selected format.

5: Designing

The tools you use for 2D and 3D sketching and editing are found in Discovery's Design tab. With the design tools, you can sketch in 2D, generate and edit solids in 3D, and work with assemblies of solids.

In Discovery, there are three modes you can use to design: Sketch, Section, and 3D mode. You can switch between these modes at any time.

When creating designs, you will use the following tools most often:

Select	Use the **Select** tool to select 2D or 3D objects in your design for editing. You can select vertices, edges, axes, faces, surfaces, solids, and components in 3D. In 2D, you can select points and lines. You can also use this tool to change the properties of recognized or inferred objects.
Pull	Use the **Pull** tool to offset, extrude, revolve, sweep, draft, and blend faces; and to round or chamfer corner edges.
Move	Use the **Move** tool to move any single face, surface, solid, or component. The behavior of the Move tool changes based on what you have selected. If you select a face, you can pull or draft it. If you select a solid or surface, you can rotate or translate it.
Combine	Use the **Combine** tool to merge and split solids and surfaces.
Section Mode	Use **Section Mode** to create and edit designs by sketching on and editing any cross-section through the design.

If an operation runs more than three seconds, the Stop icon is activated in the status bar. Click **Stop** to cancel the current operation. You can also press and hold **Esc** to cancel out of any design action that is taking too long.

When using any tool guide, you can click an empty point in the Design window to clear the tool guide selection and return to the previous action within the tool.

This section contains the following topics:

5.1. Clipboard

Select a tool from the Clipboard ribbon group to cut, copy, or paste any 2D or 3D object. You can use these tools any time, even when you are designing with other 2D or 3D tools.

Objects are pasted into the active component. Objects that can be named (components, bodies, axes, or planes) will retain their names when pasted.

When you cut and paste a solid or surface, the object retains its color and face style.

This section contains the following topics:

5.1.1. Pasting an Object into the Graphics Window

1. Cut or copy the object.
2. Select the **Paste** tool from the Clipboard ribbon group.

   You can also press **Ctrl+V** or right-click in the graphics window and select **Paste** from the context menu.

   If you do not specify a location to paste, the copy is placed in its original location relative to the world origin.

When you paste an object and it is located outside of the visible area in the graphics window, the zoom is adjusted to make the object visible.

> **Note:** Face edges that are Copied and Pasted create Curves. In some cases, there may be a very small deviation between the curve and the underlying edge that was copied. Most cases involve spline edges because of the way spline curves are constructed. A small number of cases may occur where lines and arcs connect due how the connecting vertex is calculated.

5.1.2. Pasting an Object onto a Face

1. Select all the faces of the object.
2. (Optional) Alt+click the face of the object you want to attach to the face.
   The face is highlighted in blue.
3. Cut or copy the selection.
4. Click the face onto which you want to paste the object.
5. Paste the object.
   The object is pasted on the face at the location you clicked. The blue face is attached to the clicked face.

5.1.3. Pasting a Round onto Another Edge

1. Select the face of the round.
2. Cut or copy the face.
3. Click the edge onto which you want to paste the round.
4. Paste the round.

5.1.4. Cutting an Object

1. Select the object.
2. Select the **Cut** tool from the Clipboard ribbon group.
   You can also press **Ctrl+X** or right-click in the graphics window and select **Cut** from the context menu. If you cut the face of a solid, it is converted to a surface.

5.1.5. Copying an Object

1. Select the object.
2. Select the **Copy** tool from the Clipboard ribbon group.
   You can also press **Ctrl+C** or right-click in the graphics window and select **Copy** from the context menu.

5.1.6. Deleting an Object

1. Right-click the selected object (or set of objects).
2. Press **Delete**.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

If you want to delete something and fill the gap with neighboring geometry, or create faces in the gap, you should use the Fill tool.

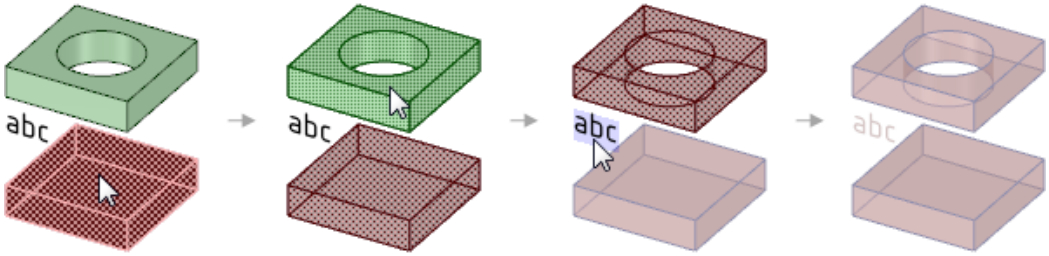
If you delete the face of a solid, it is converted to a surface.

5.1.7. Style Painter

Use the **Style Painter** tool to apply the display properties of one object to another object. The tool applies color and transparency intelligently from one object type to a different object type.

You can select a component in the model tree as the target for the **Style Painter** tool. The color and transparency are applied to all solids in a component. You cannot select a component as the source object, and only the solids in the target component are changed.

In the example below, the red part is semi-transparent and has blue edges. The style from the red block was painted to the green block and to the note. The color, transparency, and edge color of the red block was copied to the green block. Only the color of the red block was copied to the note, because a note doesn't have transparency or edge color.



5.1.7.1. Painting Display Properties from One Object to Another

1. Click **Style Painter** in the Clipboard group.
2. Select the source object that you want to copy the display properties from.
3. Select the target object that you want to copy the display properties to.

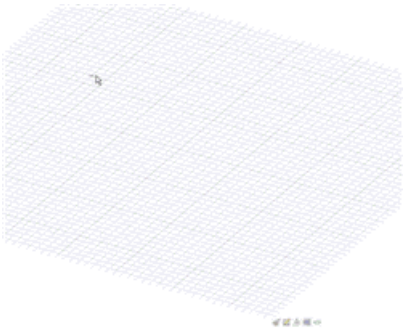
   The **Select Target** tool guide automatically becomes active so you can paint the properties of the source to as many targets as you like by simply clicking on them.

   If you can't select the object you want, such as an edge, try scrolling the mouse wheel over the object.
4. If you want to paint properties from a different source object, click the **Select Source** tool guide or hold **Ctrl** and select a different source object.

   The **Select Target** tool guide automatically becomes active again so you can click as many targets as you want.

5.2. Sketching

Use the **Sketch** tools to sketch shapes in 2D. When you exit the sketch, regions are formed by intersecting lines. These regions will become solids and lines become edges when you pull your sketch into 3D with the Pull tool. Even when pulled into 3D, a region can be decomposed back into its sketched lines for further editing as long as any remnant of the lines is still unused in 3D.



To use any of the sketch tools to sketch in 2D, you must first display the sketch grid. If you have a planar surface highlighted, and press a sketch tool shortcut (such as **L** for the **Line** tool), you can hover over planar surfaces in the design to highlight surfaces for the sketch grid. (Press **Esc** while in this state to return to the **Select** tool in 3D mode.) You can adjust the units and spacing of the grid, you can also fade the scene under the grid to enhance the visibility of your sketch.

You can lock the base dimension base point when sketching multiple objects. Locking a base point enables you to secure the dimensions of an object relative to that point, or, the dimensions of an object relative to any object you previously sketched. As you sketch, you can enter coordinates for each successive point relative to the previous point.

While you are sketching, you may need to orient your design. If you use the **Spin**, **Pan**, or **Zoom** tools to reorient the sketch, click the navigation tool again or press **Esc** to continue sketching where you left off.

When you copy and paste sketch objects, they are placed in their original location relative to the center of the grid. The objects will be highlighted when you paste, so you can move them to a different position, if required.

Sketched objects are added to the **Curves** folder in the model tree as you create them. If the list of sketch curves is long, then you will see **More Curves** in the list. Click **More Curves** to display the entire list.

This section contains the following topics:

5.2.1. Using the Sketch Tools

To use any of the sketch tools to sketch in 2D, you must first display the sketch grid. If you have a planar surface highlighted, you can hover over planar surfaces in the design to highlight surfaces for the sketch grid. In general, to use a sketch tool:

1. Click **Sketch Mode** in the Mode group on the Design tab.
2. Choose where you want to sketch.

   Hover over the planes and planar faces in your design to preview the location and orientation of the sketch grid.

   The sketch grid is placed automatically if you pre-select an object in the active component before you click **Sketch Mode**. If you pre-select an origin object, the sketch grid will be aligned with its X and Y axes and the sketch origin will be centered on the origin object. If you pre-select axes objects, the grid will run through the axes.

   If you previously selected a set of references that define a plane, the sketch grid is placed on the defined plane. The sketch grid mini-toolbar allows you to switch from sketching on one plane to sketching on another without leaving the sketch tools.

   a) If the sketch grid is currently displayed, click **Select New Sketch Plane** in the mini-toolbar, or right-click and select **Select New Sketch Plane** from the context menu.
   b) Hover over any existing geometry to display existing planes.
   c) Click to select the highlighted plane and display the sketch grid. Any vertices or edges on the plane are drawn in the current layer color and bolded.
   d) (Optional) Click **Plan View** in the mini-toolbar or in the Orient ribbon group to view the sketch grid head-on.

3. (Optional) Move or rotate the sketch grid.
   a) (Optional) Select any points, lines, or curves that you want to move with the sketch grid.
   b) Click **Move Grid** in the mini-toolbar.
   c) Use the Move handle to move or rotate the sketch grid. Highlighted curves will be moved with the grid.
   d) (Optional) Use the **Orient To Object** tool guide to orient the sketch grid to an object in the Design window. The object does not have to lie in the sketch plane.
4. Select any sketch tool from the **Sketch** ribbon group.

   You can draw points, lines, tangent lines, construction lines, rectangles, three-point rectangles, circles, three-point circles, ellipses, splines, tangent arcs, three-point arcs, sweep arcs, and polygons using the sketch tools. You can also create sketch lines by projecting the edges of 3D solids onto the sketch grid.

   Discovery's sketch tools also let you split, trim, and offset lines, as well as create corners and rounded corners.
5. Draw with the tool.

   While hovering, Discovery also provides extension lines when you are parallel to an edge or perpendicular to an end point. For certain drawing tools, it displays indicators of tangency, line midpoint, line end point, squares, and golden rectangles.

   All tools let you enter dimensions while sketching. In some tools, you can press **Shift** at a reference point to see dimensions from that point to the cursor.
6. When you are done sketching with the tool, you can:
   - Click another sketching tool.
   - Click **Return to 3D Mode** in the mini-toolbar or **3D Mode** in the Mode group in the ribbon bar to pull your sketch to 3D.
   - Press **Esc** or click the Select tool in the Edit ribbon group to edit the sketch.

5.2.2. Editing a Sketch

1. Click the **Select** tool. (You can also press **Esc** if you are in a sketching tool.)
2. Click and drag the line or point you want to edit.

   Alt+click and drag if you want to detach the line or point before moving it.

   Ctrl+click and drag to create a copy.

   Enter a value to dimension the move.

   You can also use the Move tool to edit a sketch.

5.2.3. Toggling Between a Curve and a Construction Curve

Right-click the curve and select **Construction On/Off**.

5.2.4. Mirroring in Sketch Mode

1. Draw a line or construction line.
2. Right-click the line and select **Set as Mirror Line from** the context menu.
3. Sketch on one side of the line to mirror the sketch on the other side.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

You can only mirror geometry drawn after you set the mirror line. If you want to mirror existing geometry, use the Mirror tool.

5.2.5. Sketch Group

The tools in the **Sketch** ribbon group are used to create sketch and construction curves.

Line	Use the **Line** tool to sketch lines in 2D.
Tangent Line	Use the **Tangent Line** tool to sketch lines tangent to any curves in your design.
Construction Line	Use the **Construction Line** tool to draw lines that help you create an accurate sketch. These lines become axes in 3D mode.
Rectangle	Use the **Rectangle** tool to draw a rectangle along the axes of the sketch grid.
Three-Point Rectangle	Use the **Three-Point Rectangle** tool to quickly sketch a rectangle at any angle in 2D.
Ellipse	Use the **Ellipse** tool to sketch an ellipse in 2D.
Circle	Use the **Circle** tool to sketch a circle in 2D when you know the location of the circle's center and radius, diameter, or a point on the circle's edge.
Three-Point Circle	Use the **Three-Point Circle** tool when you don't know the center of the circle, but you know where the edge of the circle must be. This tool works with any combination of free points, known points, or tangent attachments.
Polygon	Use the **Polygon** tool to sketch a polygon with up to 32 sides.
Tangent Arc	Use the **Tangent Arc** tool to sketch an arc tangent to any single curve or line in your design.
Three-Point Arc	Use the **Three-Point Arc** tool to create an arc by specifying its start and end points, and the radius or chord angle.
Sweep Arc	Use the **Sweep Arc** tool to create an arc with a known center and end points.
Spline	Use the **Spline** tool to sketch splines in 2D. A spline is a continuously curved line, without sharp boundaries (that is, without vertices).
Point	Use the **Point** tool to sketch points in 2D.
Face Curve	Use the **Face Curve** tool to sketch a curve on a face of a solid.

Create Rounded Corner	Use the **Create Rounded Corner** tool to trim back or connect two intersecting lines or arcs so that they meet with an arc tangent at both ends.
Offset Curve	Use the **Offset Curve** tool to create an offset of any line in the grid plane.
Project to Sketch	Use the **Project to Sketch** tool to project edges from a 3D object onto the sketch grid.
Create Corner	Use the **Create Corner** tool to trim back or extend two lines so that they meet at a corner.
Trim Away	Use the **Trim Away** tool to delete any line portion bounded by an intersection with a line or edge.
Split Curve	Use the **Split Curve** tool to split one line with another line or point.

The **Constraints** group contains the following tools to add constraints to sketches:

Dimension	Use the **Dimension** tool to create a dimension constraint.
Coincident	Use the **Coincident** tool to create a coincident constraint.
Midpoint	Use the **Midpoint** tool to create a mid-point constraint.
Concentric	Use the **Concentric** tool to create a concentric constraint.
Fixed	Use the **Fixed** tool to create a fixed constraint.
Horizontal	Use the **Horizontal** tool to create a horizontal constraint.
Vertical	Use the **Vertical** tool to create a vertical constraint.
Parallel	Use the **Parallel** tool to create a parallel constraint.
Perpendicular	Use the **Perpendicular** tool to create a perpendicular constraint.
Tangent	Use the **Tangent** tool to create a tangent constraint.
Equal Radius	Use the **Equal Radius** tool to create an equal radius constraint.
Equal Distance	Use the **Equal Distance** tool to create an equal distance constraint.
Symmetric	Use the **Symmetric** tool to create a symmetric constraint.
Equal Length	Use the **Equal Length** tool to create an equal length constraint.
Show Constraint Tips	Use the **Show Constraint Tips** tool to view the constraints applied.
Delete Constraints	Use the **Delete Constraints** tool to delete constraints.
Autoconstrain	Use the **Autoconstrain** tool to find the specified constraint types and automatically set up constraints.

The **End Sketch** group contains options to enable sketching in 3D and end sketch editing.

Enable 3D Sketch	Use **Enable 3D Sketch** to switch to sketching in 3D. See Working with Sketch Curves in 3D on page 182 for the sketch tools you can use in 3D mode.
End Sketch Editing	Use **End Sketch Editing** to complete sketch editing.

5.2.6. Sketch Tool Guides and Options

Sketch Tool Guides

The sketching tools have several tool guides that allow you to change the behavior of the tool. These guides are active when appropriate:

Select Reference Curve	Use **Select Reference Curve** to dimension a sketch based on an existing curve.
Move Dimension Base Point	Use **Move Dimension Base Point** to move the base point from your starting point to a different location. This is useful when you want to control the distance between your new sketch and existing object.
Change Dimension Reference Angle	Use **Change Dimension Reference Angle** to dimension a sketch based on a reference angle from a point on an existing object.

Sketch Tool Options

The following options are available:

Sketch Options	**Automatically Create Constraints**: When Constraint-based sketching is enabled, select this option to automatically generate constraints based on shapes created or on snaps. This option is enabled by default.
	Snap to grid: Select this option to turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units.
	Snap to angle: Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Dimension Options	**Cartesian dimensions**: Select a point in a sketch and then click this option to see Cartesian dimensions from the point. Cartesian dimensions show you the X and Y distances from the point you select. If you don't have a point selected, it shows you the X and Y distances from the origin.
	Polar dimensions: Select a point in a sketch and then click this option to see Polar dimensions from the point. Polar dimensions show you an angle and a distance from the point you select. If you don't have a point selected, it shows you the angle and distance from the origin.
	Lock base point: Select this option to dimension to a base point that is locked at a set distance or reference while sketching.
Mesh Fit Options	**Mesh Fit Options**: If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves.
	Fit curves: Disable this option if you do not want the system to fit curves through the points.
	Tolerance: Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
	Auto-merge: When enabled, the system will merge lines and arcs to form splines. Splines are displayed pink.

5.2.7. Sketch Grid Tool Cluster

While you are sketching, the Sketch Grid tool cluster provides quick access to the following actions:

Return to 3D Mode	Click **Return to 3D Mode** to switch to the Pull tool and pull your sketch into 3D. Any closed loops will form surfaces or faces. Intersecting lines will split faces.
Select New Sketch Plane	Click **Select New Sketch Plane** to select a new face to sketch on.
Move Grid	Click **Move Grid** to move or rotate the current sketch grid with the Move handle.
Plan View	Click **Plan View** for a head-on view of the sketch grid.

5.2.8. The Sketch Plane

The sketch grid indicates that you are performing actions in a 2D plane. Selection, sketching, creating layouts, adjusting blend planes, cross-section editing, and annotation all use the sketch grid. You can adjust the units and spacing of the grid, as well as how solids are displayed when the grid appears.

This section contains the following topics:

5.2.8.1. Displaying a Sketch Plane

1. Select **Sketch Mode** or any sketching tool.
2. Click a face, plane, axis, or drawing sheet.

   You can also select the combinations used to insert a plane to display a sketch grid at that location.

   If you do not see the grid, make sure that the **Show Sketch Grid** box is checked in the **Grid** ribbon group in the **Display** tab.

5.2.8.2. Selecting a New Location for the Sketch Plane

1. Click **Select New Sketch Plane** in the mini-toolbar, or right-click and select **Select New Sketch Plane** from the context menu.
2. Mouse over any existing geometry to display existing planes.
3. Click to select the highlighted plane and display the sketch grid.

5.2.8.3. Moving the Sketch Plane

1. (Optional) Select any points, lines, or curves that you want to move with the sketch grid.
2. Click **Move Grid** in the mini-toolbar.

   The Move handle is now placed on the end point of the last line or curve you drew. This makes drawing a trajectory in 3D space easier.
3. Use the Move handle to move or rotate the sketch grid.

5.2.8.4. Viewing the Sketch Plane Head-on

Click **Plan View** in the Sketch Grid tool cluster or in the Orient ribbon group to view the sketch grid head-on.

5.2.8.5. Switching to the Last Sketch Plane

Right-click the plane and select **Use Last Sketch Plane**.

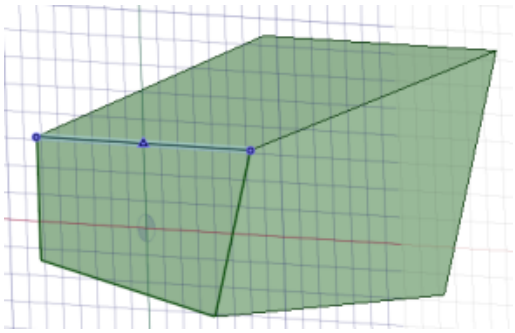
This command is in the context menu for all tools that can be used in both sketch and section modes, such as Pull.

All closed sketch curves in the current plane are converted to surfaces.

The last plane used for sketching on is set as active, and dynamic plane mode is ended when you use the **Use Last Sketch Plane** command.

5.2.8.6. Using Alt+Select in Sketch Mode

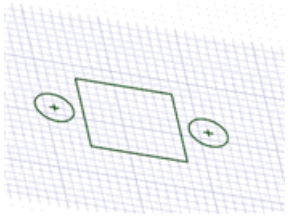
While working in **Sketch** mode, you can align a sketch plane to any linear reference that you **Alt+Select**. For example, on a block with angled planar faces, you can select a face and then **Alt+Select** an edge. The Sketch grid aligns to the Alt reference.



5.2.9. Move Grid Tool

Use the **Move Grid** tool to move the sketch grid. Make successive sketches by moving the grid after sketching closed line regions. These closed lines turn into regions when you move the grid.

The icon at the center of the sketch grid indicates the origin of the grid and moves with the grid as you move the grid.



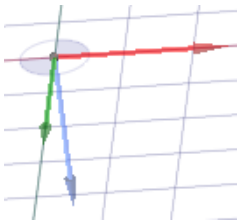
5.2.9.1. Moving the Sketch Grid

1. Click the **Move Grid** tool on the mini-toolbar or click the Move tool and check the **Move Grid** option.
2. (Optional) Select any sketch entities that you want to move along with the sketch grid.
3. Select a Move handle axis.
4. Drag along the axis of the Move handle to move or rotate the sketch grid.

   You can also right-click and select **Use Ruler Dimension**, and enter a value or press **Enter** to drag the grid. You can also use the standard Move tool guides when moving the grid.

   The Move handle can be moved around by dragging the center ball onto appropriate entities on the sketch grid.

5.2.9.2. Moving the Center of the Sketch Grid



1. Insert an origin.
2. Select an axis of the origin.
3. Switch to Sketch mode.

   You can also move the center of the sketch grid while sketching by using the Move Grid or Select New Sketch Plane tools in the Sketch mini-toolbar.

5.2.10. Sketching with Constraints

You can choose constraint-based sketching to reduce the degrees of freedom (place constraints) on the drawing objects in 2D drawings and designs. Adding constraints restricts the type and range of movement of the objects, and links objects such that certain editing tasks on one drawing object will cause linked objects to be modified. Use tools in the **Constraints** group in the **Design** tab to add constraints to sketches.

The tools in the **Constraints** group offer additional types of constraints which can be applied to any of the standard drawing objects (line, circle, arc, and so on) and are more robust.

This section contains the following topics:

5.2.10.1. Saving and Retaining Sketches

You can **Save** and **Open** models containing sketch **Constraints**. Designs that include **Constraints** will open in **Sketch** mode. Leaving **Sketch** mode will trigger the deletion of all constraint data. You need to save a copy of the sketch with constraints intact before solidifying if you intend to resume the design with constraints.

> **Note:** **Sketch constraints** are active only in 2D sketch mode. If you switch to 3D mode, existing constraint data is deleted. If you expect to edit constraints as part of your design workflow, you must save a copy of the sketch with constraints before switching to 3D mode.

You can also select the **Retain Sketch** option so that the sketch is not consumed by 3D geometry when sketching is complete.

- Sketch objects added after retaining the sketch are also retained.
- The sketch and sketch objects remain in the tree until you choose to delete them. To delete the sketch/sketch objects, select them in the tree and select **Delete** from the context menu.

After retaining the sketch, you can select the **Use for 3D** option to create regular curves and surfaces for closed sketch regions. The original sketch is retained for reuse even after you select **Use for 3D**.

Removing Interior Surfaces

Use the **Remove Interiors** option to remove interior surfaces that would be created for closed sketch regions within the sketch on switching to 3D mode. Select the **Sketching Plane** node in the model tree and enable **Remove Interiors** from the context menu.

5.2.10.2. Adding Layout Curves to an Imported Datum

When a .dwg or .dxf file is imported and constraint-based sketching is enabled, use the **Add Layout Curves** option to add layout curves to the imported datum. Select the datum in the model tree and enable **Add Layout Curves** from the context menu.

> **Note:** If you activate or switch to another sketch, you need to end the sketch before continuing to add layout curves.

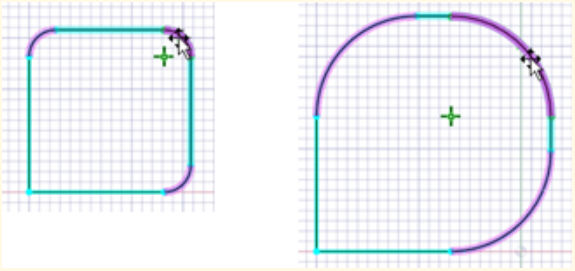
5.2.10.3. Constraint Types

The following **Constraints** can be added to your sketch:

- **Dimension**
- **Coincident**
- **Midpoint**
- **Concentric**
- **Fixed**
- **Horizontal**
- **Vertical**
- **Parallel**
- **Perpendicular**
- **Tangent**
- **Equal Radius**
- **Equal Distance**
- **Symmetric**
- **Equal Length**

> **Note:**
>
> - Most of these constraints require two selected drawing objects. The first selection is the target and will generally be moved to satisfy the constraint. The second is the anchor object and will generally remain in position.
>
> For the set up of constraints involving multiple steps in a particular sequence, the context-aware cursor visually displays your progress through the multi-step setup for the constraint. The constraint tool is indicated in the center surrounded by a segemented ring. Each segment represents a step in the workflow and is color coded.
>
> When the cursor is over a valid selection, the corresponding segment shows yellow. Invalid selections are shown in red. The segment turns green when the appropriate selection is made.
>
> When the entire segmented ring turns green, it indicates that the constraint is applied.

- **Fixed**, **Horizontal** and **Vertical** constraints require only one selected drawing object.
- **Concentric** and **Equal Radius** constraints propagate when modifying the curves.
- **Equal Radius** constraints are added automatically when you create multiple rounded corners using the **Ctrl** key and the **Create Rounded Corner** tool in the **Modify** group. All the rounds created this way maintain an **Equal Radius** constraint so that changing the size of one will change all of them.



5.2.10.3.1. Adding Dimension Constraints

1. Activate the **Dimension** constraint.
2. Select an individual sketch object, or two objects as required for certain types of dimensions. Move the cursor away from the object to position the dimension. The normal behavior for applying and locating dimensions is observed. However, when applied as a constraint, if you attempt to edit one object, other objects may be modified to maintain the dimension constraint.
   - You can use dimension constraints to modify the size of the selected object. After creating the dimension constraint, double-click to select the dimension and then modify the value.
   - Each dimension is automatically assigned a label. You can also specify labels of your choice.

     Note: A valid label should have an alphabet or an underscore (_) as the first character.

   - You can also define dimensions using basic mathematic expressions.
   - Dimensions may be positioned vertically, horizontally, or aligned with the object.
   - Angle dimensions are supported - select the first object and then hover over the second object until the angle dimension appears. Move the cursor to position the dimension.
   - Arc length dimensions are supported - hold the **Ctrl** key while selecting the arc to create an arc length dimension. Move the cursor to position the dimension.

5.2.10.3.2. Adding a Coincident Constraint

1. Activate the **Coincident** constraint.
2. Select the target drawing object to be moved.
3. Select the anchor drawing object.
   The target object will be made coincident to the anchor object.

5.2.10.3.3. Adding a Midpoint Constraint

1. Activate the **Midpoint** constraint.
2. Select the target drawing object to be moved.
   You can select a line or a point as the target object.
3. Select the anchor drawing object.
   For a target line selected, select a point as the anchor. For a target point selected, select a line as the anchor.
   The target object will be moved to satisfy the midpoint constraint.

5.2.10.3.4. Adding a Concentric Constraint

1. Activate the **Concentric** constraint.
2. Select the target drawing object to be moved.
3. Select the anchor drawing object.
   The target object will be made concentric to the anchor object.

5.2.10.3.5. Adding a Fixed Constraint

1. Activate the **Fixed** constraint.
2. Select the drawing object to apply the **Fixed** constraint.
   The **Fixed** constraint tool acts as a toggle. Click an object to apply, click a second time to remove the constraint.

5.2.10.3.6. Adding a Perpendicular Constraint

1. Activate the **Perpendicular** constraint.
2. Select the target drawing object to be moved.
3. Select the anchor drawing object.
   The target object will be moved to satisfy the perpendicular constraint.

5.2.10.3.7. Adding a Parallel Constraint

1. Activate the **Parallel** constraint.
2. Select the target drawing object to be moved.
3. Select the anchor drawing object.
   The target object will be moved to satisfy the parallel constraint.

5.2.10.3.8. Adding a Tangent Constraint

1. Activate the **Tangent** constraint.
2. Select the target drawing object to be moved.
   You can select a line or a circle as the target object.
3. Select the anchor drawing object.
   For a target line selected, select a circle as the anchor. For a target circle selected, select a line as the anchor.
   The target object will be moved to satisfy the tangent constraint.

5.2.10.3.9. Adding a Horizontal Constraint

1. Activate the **Horizontal** constraint.
2. Select the drawing object to apply the **Horizontal** constraint.

5.2.10.3.10. Adding a Vertical Constraint

1. Activate the **Vertical** constraint.
2. Select the drawing object to apply the **Vertical** constraint.

5.2.10.3.11. Adding an Equal Radius Constraint

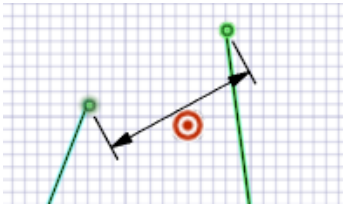
1. Activate the **Equal Radius** constraint.
2. Select the target drawing object to be changed.
3. Select the anchor drawing object.
   The target object will be modified to have the same radius as the anchor object.

5.2.10.3.12. Adding an Equal Distance Constraint

The **Equal Distance** constraint uses on-screen tool guides to determine the anchor and target distances, and to create the constraint. This is because more than one selection is used to create the constraint.

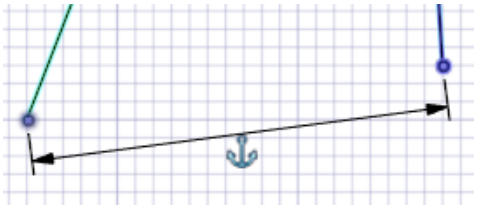
1. Activate the **Equal Distance** constraint.
2. With the **Target** tool guide active, select the target pair of drawing objects or geometries to define the first dimension.

   A dimension line with the target symbol is added to the design.

3. With the **Anchor** tool guide active, select the anchor pair of drawing objects or geometries.

   A dimension line with the anchor symbol is added to the design.

   The target objects will be modified to have the same distance between them as the anchor objects.

5.2.10.3.13. Adding a Symmetric Constraint

1. Activate the **Symmetric** constraint.

2. Select the **Make ends symmetric** option (enabled by default) to make the curve end points symmetric.
3. With the **Select Axis of Symmetry** tool guide active, select the axis about which the sketch objects are to be symmetric.
4. With the **Select Curves** tool guide active, select the pair of curves to be made symmetric.
   The target object will be made symmetric to the anchor object.

5.2.10.3.14. Adding an Equal Length Constraint

1. Activate the **Equal Length** constraint.
2. Select the target line.
3. Select the source line.
   The target line will be modified to have the same length as the anchor line.

5.2.10.4. Autoconstrain Tool

You can use the **Autoconstrain** tool to find the specified constraint types and automatically set up constraints.

Select **Autoconstrain** in the **Constraints** group.

1. Select the **Curve options** to allow points and/or axes to be considered for creating constraints.
2. Select the constraints to be automatically set up from the **Constraint Options** list. The following constraints can be set up:
   - Coincident constraint
   - Tangent constraint
   - Parallel constraint
   - Perpendicular constraint
   - Equal radius constraint
   - Concentric constraint

5.2.10.4.1. Autoconstrain Tool Guides and Options

Autoconstrain Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Problem	The **Select Problem** tool guide is active by default. This tool guide allows you to select sketch entities that are automatically found by the tool.
Select Geometry	The **Select Geometry** tool guide allows you to select entities that were not automatically found. Hold Ctrl to select multiple entities or box select in the design area.
Exclude Problem	The **Exclude Problem** tool guide allows you to exclude entities from selection.
Complete	The **Complete** tool guide adds constraints for the highlighted sketch entities.

Autoconstrain Tool Options

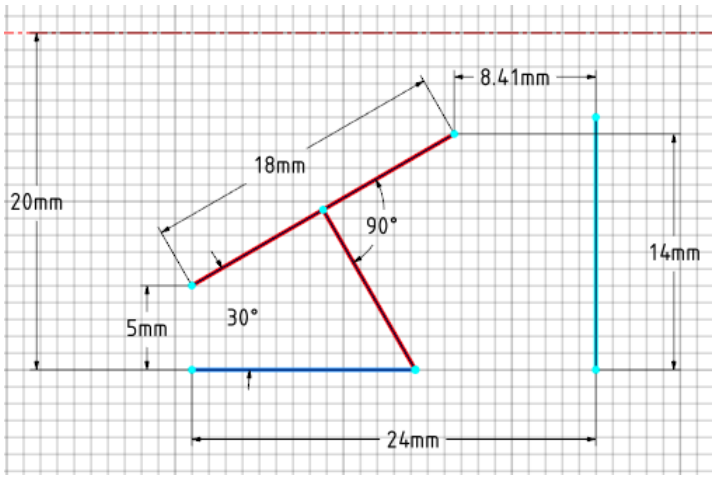
The following options are available:

Curve Options **Allow points** **Allow axes**	Select to allow points and/or axes to be considered for creating constraints.
Constraint Options	Select the constraints to be automatically set up. Coincident Tangent Parallel Perpendicular Equal radius Concentric

5.2.10.5. Object States

When using constraint-based sketching, you can make sketch objects under-defined, fully-defined, or over-defined.

Select **Show Constraint Colors** to view the sketch objects colored based on the object state. This option is enabled, by default.



- Under-defined objects have some degree of freedom and are highlighted cyan in the design window.
- Fully defined objects have no degrees of freedom and are highlighted green in the design window.
- Over-defined objects are highlighted red in the design window. This also applies to dimensionally over-constrained lines. One or more constraints must be removed before you can proceed with your design.

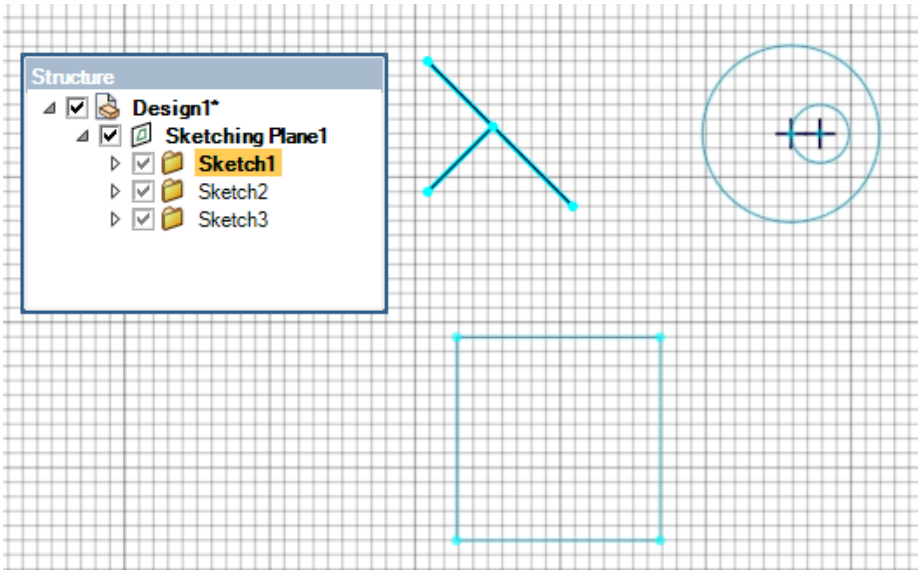
The sketch properties show the constraints needed to make the sketch objects fully defined.

> **Note:** Large changes on fully defined sketches can be problematic because the first dimension has to resize and then resolve all others. You should use several small changes to get to a certain size, instead of a large change that may fail.

For example, if you change a dimension by 30 mm in one go, the other dimensions may not be resolved correctly, resulting in an invalid dimension. However, if you change the dimension in 5 mm - 10 mm increments, the dimensions may be resolved correctly.

5.2.10.6. Creating Multiple Sketches with Constraints

You can create multiple sketches with constraints on the same sketching plane. The active Sketch node is shown in bold in the tree and the active sketch is highlighted in the design window.



To activate a different sketch, select the **Sketch** node in the model tree and select **Activate Sketch**.

Note: Constraints may be applied within sketches on the sketching plane.

5.2.10.7. Creating a New Sketching Plane

1. Click **Select New Sketch Plane** in the mini-toolbar, or right-click and select the **Select New Sketch Plane** option from the context menu.
2. Hover over any existing geometry to display existing planes.
3. Click to select the highlighted plane and display the sketch grid.

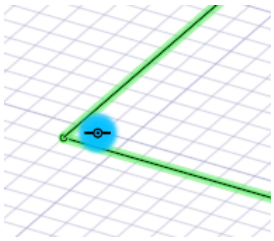
Existing sketches are solidified, and a new **Sketch** node is added to the **Sketching Plane** in the tree. The previous sketch remains active temporarily until you start sketching in the new Sketch.

Note: **Coplanar Sketch Constraints** can reside on different containing planes.

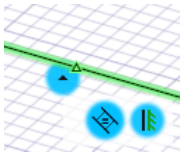
5.2.10.8. Viewing Constraints

Select **Show Constraint Tips** in the **Constraints** group to view the constraints applied.

Hover over the constraint tip to see the type of constraint. The geometries involved will be highlighted in green.



If multiple constraints are applied, click the dropdown to see all the constraints applied.



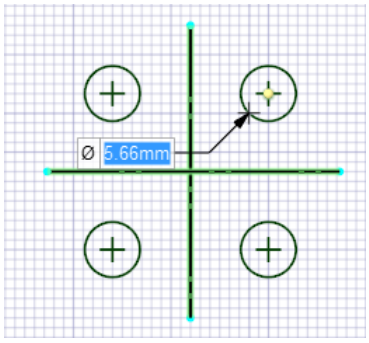
5.2.10.9. Deleting Constraints

You can delete constraints using the **Delete Constraint** option.

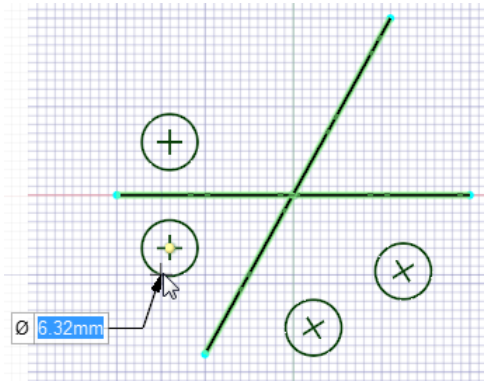
Select the constraint to be deleted. In case of multiple constraints, click the dropdown and then select the constraint to be deleted. Select **Delete** from the context menu.

5.2.10.10. Using Mirror with Sketch Constraints

1. With Constraint-based sketching active, sketch a line in the desired mirror position.
2. Use the right-click context menu, select **Set as Mirror Line** to set up the mirror.



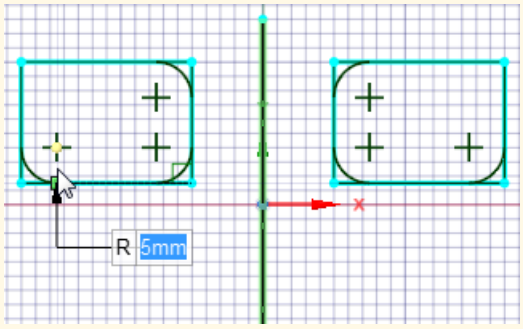
3. With multiple **Mirror Lines**, entities are mirrored about each mirror line whether they are perpendicular or not.



4. To stop creating mirror constraints, uncheck **Set as Mirror Line**.

Note:

- Splines are not supported with a **Mirror Line**, but the other **Sketch** tools are.
- A **Fixed constraint** is automatically applied to the mirror line to ensure more predictable dragging. It is possible to delete this constraint but dragging behavior may deteriorate.
- **Equal Radius** constraints are made automatically when creating multiple **Rounded Corners with a Mirror Line**.



- To delete **Mirror (Symmetry)** constraints individually, click an entity to open the mini toolbar. Individual constraints can be removed using the **Delete** dropdown menu.

Note: 1D Patterns and 2D Patterns are not supported for Constraint-based sketching.

5.2.10.11. Referencing Existing Geometry

It is possible to create constraints and dimensions to cross-sections and edges that lie in the sketch plane. Such cross sections and edges are treated as *reference curves* and will not be changed. Creating a sketch constraint or dimension requires at least one non-reference curve to allow for the constraint to be satisfiable.

5.2.11. Sketch Tools

This section contains the following topics:

5.2.11.1. Line Tool

Use the **Line** tool to sketch lines in 2D or to draw lines between points on objects in 3D. These lines will become edges when you pull your sketch into a solid with the **Pull** tool.

The sketch grid must be visible in the workspace before you can draw.



This section contains the following topics:

5.2.11.1.1. Drawing Lines on a Sketch Plane

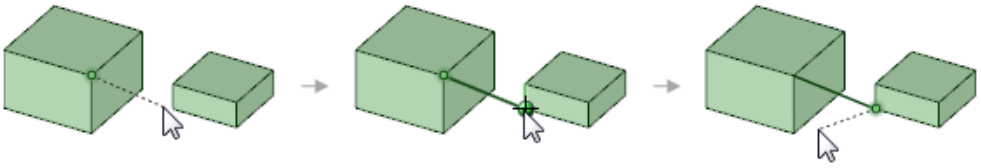
1. Click **Line** in the Sketch group or press **L**.
2. If a sketch plane is not active, click an object or objects in the Design window to make a sketch plane.
3. Click to set the first point of the line.
   You can dimension the points relative to other objects.
4. Continue selecting points to draw line segments.
   If you want any section of the line to be an arc, right-click and select **Switch to Arc**, then click to set the radius. Right-click and select **Switch to Line** to return to drawing straight lines between points. You must create at least one line segment before using this option.
5. End the line:
   - Double-click to set the end point of the line.
   - Right-click and select **Finish Line**.
   - Press **Esc**.
   - Connect the end point to the start point.
   - Click any tool (except the Clipboard and Orient tools).

> **Note:** Click and drag to draw one straight line.

5.2.11.1.2. Drawing Lines Between Points in 3D

1. Select **Enable 3D Sketch**.
2. Click **Line** in the Sketch group or press **L**.
3. (Optional) Select **Define line from center** in the Options panel to start drawing the line from its center instead of an end.
4. Click to set the first point of the line.
   Select any point on an object in your design.
5. Continue selecting points to draw line segments.
6. End the line:
   - Double-click to set the end point of the line.
   - Right-click and select **Finish Line**.
   - Press **Esc**.
   - Connect the end point to the start point.
   - Click any tool (except the Clipboard and Orient tools).

Sketching a line between two solids in 3D.



5.2.11.1.3. Using a Line as a Mirror

Right-click the line and select **Set as Mirror Line**.

5.2.11.1.4. Toggling Between a Line and a Construction Line

Right-click the line and select **Construction On/Off**.

5.2.11.1.5. Line Options

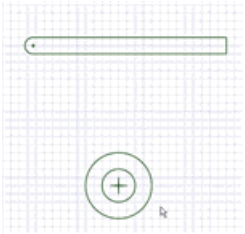
The following additional options are available in the Options panel:

Draw from center	Select this option to sketch a line from the center. Click to define the center of the line, then click again to set the length. You can also drag to draw the line.
Automatically create constraints	Select this sketch option to automatically generate constraints based on shapes created or on snaps. This option is enabled by default.
Snap to grid	Select this sketch option to turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units.
Snap to angle	Select this sketch option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Fit Curves	Disable this mesh option if you do not want the system to fit curves through the points.
Tolerance	This mesh option determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
Auto-merge	When this mesh option is enabled, the system will merge lines and arcs to form splines. Splines are displayed pink.

5.2.11.1.6. Tangent Line Tool

Use the **Tangent Line** tool to sketch lines tangent to any curves in your design.

The sketch grid must be visible in the workspace before you can draw.



This section contains the following topics:

5.2.11.1.6.1. Drawing a Tangent Line on a Sketch Plane

1. Click **Tangent Line** in the Sketch group.
   The tool is disabled if there are no curves or lines in the sketch plane.
2. Click the curve you want to draw tangent to.
3. As you move the mouse, the start point moves so that the line remains tangent to the curve.
   The tangency indicator shows you how the new line will be tangent with existing geometry if you click the current cursor location.
4. (Optional) Dimension the line with a length.
   You cannot dimension from another sketch object when creating a tangent line.
5. Click to set the end point of the line.
   If you move your mouse over another curve, the line snaps so that it is tangent to the second curve. Hold **Alt** to stop your cursor from snapping to curves.

The common sketch options are described in Sketch Tool Guides and Options on page 133.

5.2.11.1.6.2. Drawing Tangent Lines between Points in 3D

1. Select **Enable 3D Sketch**.
2. Click **Tangent Line** in the Sketch group.
3. Click to set the first point of the line.
   Select any point on an object in your design.
   The tangency indicator shows you how the new line will be tangent with existing geometry if you click the current cursor location.
4. Click to set the end point of the line.

5.2.11.1.6.3. Setting a Mirror Line with a Tangent Line

Right-click the line and select **Set as Mirror Line**.

5.2.11.1.6.4. Toggling Between a Tangent Line and a Construction Line

Right-click the line and select **Construction On/Off**.

5.2.11.1.6.5. Tangent Line Options

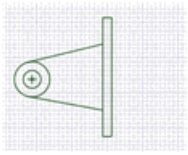
The following tool options are available:

Automatically create constraints	Select this sketch option to automatically generate constraints based on shapes created or on snaps. This option is enabled by default.
Snap to grid	Select this sketch option to turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units.
Snap to angle	Select this sketch option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Fit Curves	Disable this mesh option if you do not want the system to fit curves through the points.
Tolerance	This mesh option determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
Auto-merge	When this mesh option is enabled, the system will merge lines and arcs to form splines. Splines are displayed pink.

5.2.11.1.7. Construction Line Tool

Use the **Construction Line** tool to draw lines that help you create an accurate sketch. Construction lines become axes in 3D. They are also useful for creating mirrors.

The sketch grid must be visible in the workspace before you can draw.



This section contains the following topics:

5.2.11.1.7.1. Drawing a Construction Line

1. Select the **Construction Line** tool from the Sketch ribbon group.
2. (Optional) Dimension the first point relative to another sketch object.
3. Click to set the first point of the line.
4. (Optional) Dimension the line.
5. Click or press **Enger** to end the line.

   By default, the line is dimensioned to its start point, but you can dimension to another sketch object.

The common sketch options are described in Sketch Tool Guides and Options on page 133.

5.2.11.1.7.2. Using a Tangent Line as a Mirror

Right-click the line and select **Set as Mirror Line**.

5.2.11.1.7.3. Toggling Between a Line and a Construction Line

Right-click the line and select **Construction On/Off**.

5.2.11.1.7.4. Construction Line Tool Options

The following tool options are available:

Draw from center	Start your sketch from the center or midpoint.
Automatically create constraints	Select this sketch option to automatically generate constraints based on shapes created or on snaps. This option is enabled by default.
Snap to grid	Select this sketch option to turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units.
Snap to angle	Select this sketch option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Fit Curves	Disable this mesh option if you do not want the system to fit curves through the points.
Tolerance	This mesh option determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
Auto-merge	When this mesh option is enabled, the system will merge lines and arcs to form splines. Splines are displayed pink.

5.2.11.2. Rectangle Tool

Use the **Rectangle** tool in the Design tab or press **R** to draw a rectangle along the axes of the sketch grid. When you exit the sketch, any rectangles become surfaces, and the lines become edges of a rectangular solid when you pull your sketch into 3D with the **Pull** tool.

The sketch grid must be visible in the workspace before you can draw.

This section contains the following topics:

5.2.11.2.1. Drawing a Rectangle

1. Click **Rectangle** in the Sketch group or press **R**.
2. (Optional) Select **Define rectangle from center** in the Options panel to start drawing the rectangle by clicking the point for its center instead of a corner.
3. Click to set the first corner.

   Move your mouse over the sketch grid to preview the rectangle. Dashed lines appear when you create a square or golden rectangle.

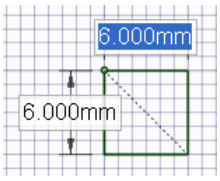
   You can dimension the points relative to other sketch objects.
4. Click to set the opposite corner of the rectangle.

5.2.11.2.2. Rectangle Options

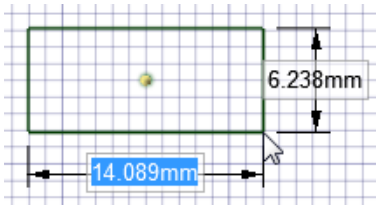
The following tool options are available:

Draw from center	Select this option to sketch a line from the center. Click to define the center of the line, then click again to set the length. You can also drag to draw the line.
Automatically create constraints	Select this sketch option to automatically generate constraints based on shapes created or on snaps. This option is enabled by default.
Snap to grid	Select this sketch option to turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units.
Snap to angle	Select this sketch option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Fit Curves	Disable this mesh option if you do not want the system to fit curves through the points.
Tolerance	This mesh option determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
Auto-merge	When this mesh option is enabled, the system will merge lines and arcs to form splines. Splines are displayed pink.

5.2.11.2.3. Sketch Rectangle Examples



Sketching a square



Sketching a rectangle from its center

5.2.11.2.4. Three-Point Rectangle Tool

Use the **Three-Point Rectangle** tool to quickly sketch a rectangle at any angle on the sketch plane. These lines will become the edges of a rectangular solid when you pull your sketch into 3D with the **Pull** tool.

The sketch grid must be visible in the workspace before you can draw.

This section contains the following topics:

5.2.11.2.4.1. Drawing a Three-Point Rectangle

1. Click **Three-Point Rectangle** in the Sketch group.
2. (Optional) Select **Define rectangle from center** in the Options panel to start drawing the rectangle by clicking the point for its center instead of a corner.
3. Click to set the first corner of the rectangle.

   Move your mouse over the sketch grid to preview the rectangle. Dashed lines appear when you create a square or golden rectangle.

   You can dimension the points relative to other sketch objects.
4. Click to set the length of the second side.

> **Note:** Click and drag to draw the first side, then click to set the length of the second side.

5.2.11.2.4.2. Three-Point Rectangle Options

The following additional options are available in the Options panel:

Draw from center	Select this option to sketch rectangles from their centers. Click to define the center of the rectangle, then click again to set the length of the sides. You can also drag to draw the rectangle. Alt+drag or Alt+click to return to the standard behavior when the option is selected.
Automatically create constraints	Select this sketch option to automatically generate constraints based on shapes created or on snaps. This option is enabled by default.

Snap to grid	Select this sketch option to turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units.
Snap to angle	Select this sketch option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Fit Curves	Disable this mesh option if you do not want the system to fit curves through the points.
Tolerance	This mesh option determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
Auto-merge	When this mesh option is enabled, the system will merge lines and arcs to form splines. Splines are displayed pink.

5.2.11.3. Ellipse Tool

Use the **Ellipse** tool to sketch an ellipse in 2D. The ellipse can become an elliptical solid or hole when you pull your sketch into 3D with the **Pull** tool. You can also sweep the ellipse in 3D, or rotate it.

The sketch grid must be visible in the workspace before you can draw.



This section contains the following topics:

5.2.11.3.1. Drawing an Ellipse

1. Click **Ellipse** in the Sketch group.
2. Click to set the center of the ellipse.
   You can dimension the points relative to other sketch objects.
3. Click to set the overall length and angular orientation of the first axis.
4. Click to set the length of the second axis.

The common sketch options are described in Sketch Tool Guides and Options on page 133.

5.2.11.3.2. Ellipse Tool Options

The following tool options are available:

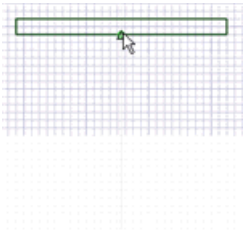
Automatically create constraints	Select this sketch option to automatically generate constraints based on shapes created or on snaps. This option is enabled by default.

Snap to grid	Select this sketch option to turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units.
Snap to angle	Select this sketch option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Fit Curves	Disable this mesh option if you do not want the system to fit curves through the points.
Tolerance	This mesh option determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
Auto-merge	When this mesh option is enabled, the system will merge lines and arcs to form splines. Splines are displayed pink.

5.2.11.4. Circle Tool

Use the **Circle** tool to sketch a circle in 2D when you know the location of the circle's center and a point on the circle's edge, or the radius or diameter. The circle can become a cylinder or hole when you pull it into 3D with the **Pull** tool, or a sphere or torus if you rotate or sweep it.

The sketch grid must be visible in the workspace before you can draw.



This section contains the following topics:

5.2.11.4.1. Drawing a Circle

1. Click **Circle** in the Sketch group or press **C**.
2. Click to set the circle's center.

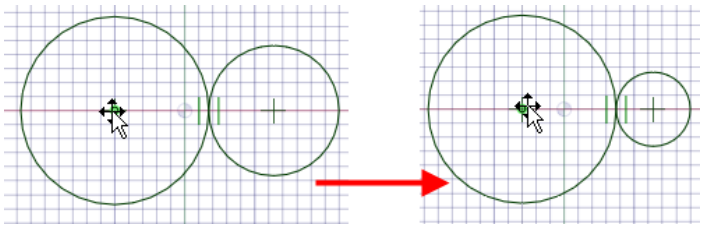
   You can dimension the points relative to other sketch objects.
3. Click to set the circle's diameter.
4. The circle will snap to existing sketches or determined circles and arcs in the plane of the sketch.

   If you sketch two circles that are tangent to each other, and then change the diameter of one circle by editing its dimension, tangency with the other circle is maintained.

   If you drag the center of a circle that is tangent to another circle, the radius of the other circle changes to maintain tangency.

The common sketch options are described in Sketch Tool Guides and Options on page 133.

5.2.11.4.2. Drawing Circle Example



Editing a tangent circle by dragging the circle's center with tangency is maintained.

5.2.11.4.3. Circle Tool Options

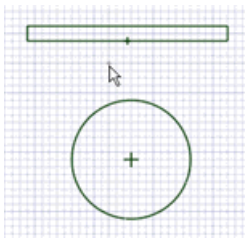
The following tool options are available:

Automatically create constraints	Select this sketch option to automatically generate constraints based on shapes created or on snaps. This option is enabled by default.
Snap to grid	Select this sketch option to turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units.
Snap to angle	Select this sketch option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Fit Curves	Disable this mesh option if you do not want the system to fit curves through the points.
Tolerance	This mesh option determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
Auto-merge	When this mesh option is enabled, the system will merge lines and arcs to form splines. Splines are displayed pink.

5.2.11.4.4. Three-Point Circle Tool

Use the **Three-Point Circle** tool when you don't know the center of the circle, but you know where the edge of the circle must be. This tool works with any combination of free points, known points, or tangent attachments. The circle will become a cylinder or hole when you pull it into 3D with the **Pull** tool. You can also rotate the circle about a line to make a sphere or torus.

The sketch grid must be visible in the workspace before you can draw.



This section contains the following topics:

5.2.11.4.4.1. Drawing a Three-Point Circle

1. Click **Three-Point Circle** in the Sketch group.
2. (Optional) Select **Three-point circle segment** in the Options panel to create an arc that is a segment of a three-point circle.
3. Click to set the first point on the circle's edge.

   If you click a curve or line, the circle will be drawn tangent to the curve or line, unless you click the midpoint or vertex.

   You can dimension the points relative to other sketch objects.
4. Click to set the second point on the circle's edge.

   If the circle disappears as you move your mouse over the sketch grid, then the cursor location cannot be included in any circle drawn through the first two points and the current point. If you click a curve or line, the circle will be drawn tangent to the curve or line unless you click the midpoint or vertex.

   If you sketch two circles that are tangent to each other, and then change the diameter of one circle by editing its dimension, tangency with the other circle is maintained.
5. Click to set the last point on the circle's edge.

   (Optional) For a **Three-point circle segment**, enter the diameter or click to set the final point.

   If you sketch two circles that are tangent to each other, and then change the diameter of one circle by editing its dimension, tangency with the other circle is maintained.

   If you drag the center of a circle that is tangent to another circle, the radius of the other circle changes to maintain tangency.

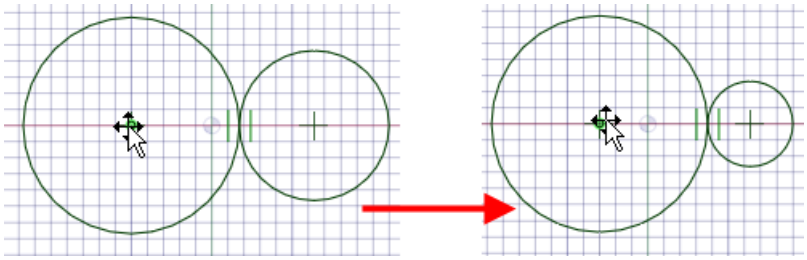
5.2.11.4.4.2. Three-Point Circle Options

The following additional options are available in the Options panel:

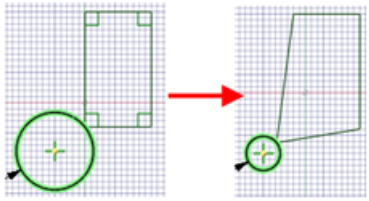
Draw arc	Create an arc that is a segment of a three-point circle.
Automatically create constraints	Select this sketch option to automatically generate constraints based on shapes created or on snaps. This option is enabled by default.
Snap to grid	Select this sketch option to turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units.
Snap to angle	Select this sketch option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Fit Curves	Disable this mesh option if you do not want the system to fit curves through the points.
Tolerance	This mesh option determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.

Auto-merge	When this mesh option is enabled, the system will merge lines and arcs to form splines. Splines are displayed pink.

5.2.11.4.4.3. Three-Point Circle Examples



Editing a tangent circle by dragging the circle's center, tangency is maintained



Dragging (with the Select tool) a three-point circle drawn through a rectangle's vertex maintains the connection

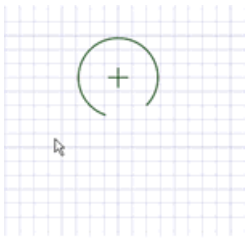
5.2.11.5. Arcs

This section contains the following topics:

5.2.11.5.1. Tangent Arc Tool

Use the **Tangent Arc** tool to sketch an arc that is tangent to a curve or line in your design. This arc will become an edge when you pull your sketch into 3D with the Pull tool.

The sketch grid must be visible in the workspace before you can draw.



This section contains the following topics:

5.2.11.5.1.1. Drawing a Tangent Arc

1. Click **Tangent Arc** in the Sketch group.
   The tool is disabled if there are no curves or lines in the sketch plane.

2. Select the line or curve you want to draw tangent to.

   This is usually the end point of a line, arc, or spline, but it can be on the line. If two lines share an end point, use the scroll wheel to set tangency to the other line.
3. (Optional) Dimension the radius and chord angle.
4. Click to set the end point of the arc.

   The arc cannot end on its start point to make a circle or end on the same line as its start point.

The common sketch options are described in Sketch Tool Guides and Options on page 133.

5.2.11.5.1.2. Drawing a Tangent Arc Between Points in 3D

1. Select **Enable 3D Sketch**.
2. Click **Tangent Arc** in the Sketch group.
3. Click to set the first point of the arc.

   Select any point on an object in your design.

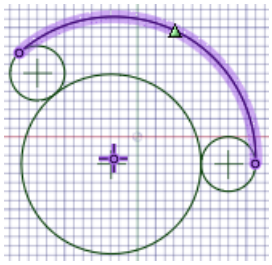
   The tangency indicator shows you how the new arc will be tangent with existing geometry if you select the current cursor location.
4. Click to set the end point of the arc.

5.2.11.5.1.3. Editing a Tangent Arc

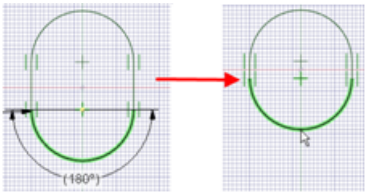
Drag the arc to edit it. When you drag a tangent arc, it increases its radius while maintaining its chord angle.

5.2.11.5.1.4. Tangent Arc Examples

Arc sketched tangent to two circles.



Editing a tangent arc maintains its connections.



5.2.11.5.1.5. Tangent Arc Options

The following tool options are available:

Automatically create constraints	Select this sketch option to automatically generate constraints based on shapes created or on snaps. This option is enabled by default.
Snap to grid	Select this sketch option to turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units.
Snap to angle	Select this sketch option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Fit Curves	Disable this mesh option if you do not want the system to fit curves through the points.
Tolerance	This mesh option determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
Auto-merge	When this mesh option is enabled, the system will merge lines and arcs to form splines. Splines are displayed pink.

5.2.11.5.2. Three-Point Arc Tool

Use the **Three-Point Arc** tool to create an arc by specifying its start and end points, and the radius or chord angle. This arc can be created tangent to another arc, line, or spline at its start. The arc will become an edge when you pull your sketch into 3D with the Pull tool. As an option, you can also create a three-point arc in 3D mode.

The sketch grid must be visible in the workspace before you can draw.

This section contains the following topics:

5.2.11.5.2.1. Drawing a Three-Point Arc

1. Click **Three-Point Arc** in the Sketch group.
2. Click to set the start point of the arc.

   If the start point is on another line, arc, or spline, then the arc will initially be drawn tangent to that line, arc, or spline.

   You can dimension the point relative to other sketch objects.
3. Click to set the end point of the arc.
4. Click to set the arc's radius.

   You can dimension the chord angle or radius.

   The arc will snap to tangent objects.

The common sketch options are described in Sketch Tool Guides and Options on page 133.

5.2.11.5.2.2. Three-Point Arc Options

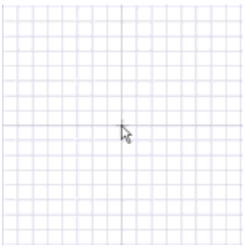
The following tool options are available:

Automatically create constraints	Select this sketch option to automatically generate constraints based on shapes created or on snaps. This option is enabled by default.
Snap to grid	Select this sketch option to turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units.
Snap to angle	Select this sketch option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Fit Curves	Disable this mesh option if you do not want the system to fit curves through the points.
Tolerance	This mesh option determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
Auto-merge	When this mesh option is enabled, the system will merge lines and arcs to form splines. Splines are displayed pink.

5.2.11.5.3. Swept Arc Tool

Use the **Sweep Arc** tool to create an arc with a known center and end points. Tangency is not a factor in the creation of this arc. The arc will become an edge when you pull your sketch into 3D with the **Pull** tool.

The sketch grid must be visible in the workspace before you can draw.



This section contains the following topics:

5.2.11.5.3.1. Drawing a Swept Arc

1. Click **Sweep Arc** in the Sketch group.
2. Click to set the center of the arc.
   You can dimension the point relative to other sketch objects.
3. Click to set the start point and radius of the sweep circle.
   You can dimension the chord angle or offset the angular dimension.
4. Click to set the end point of the arc.

The common sketch options are described in Sketch Tool Guides and Options on page 133.

5.2.11.5.3.2. Sweep Arc Options

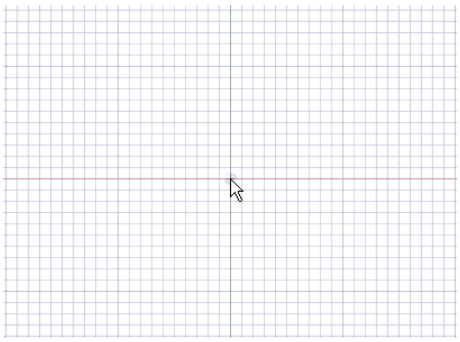
The following tool options are available:

Automatically create constraints	Select this sketch option to automatically generate constraints based on shapes created or on snaps. This option is enabled by default.
Snap to grid	Select this sketch option to turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units.
Snap to angle	Select this sketch option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Fit Curves	Disable this mesh option if you do not want the system to fit curves through the points.
Tolerance	This mesh option determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
Auto-merge	When this mesh option is enabled, the system will merge lines and arcs to form splines. Splines are displayed pink.

5.2.11.6. Polygon Tool

Use the **Polygon** tool to draw a polygon with between 3 and 64 sides. You can dimension the location of the axis, the length of the radius, the orientation angle, and set the number of sides as you sketch the polygon.

The sides of a sketched polygon maintain their relationship to each other. When you pull a polygon into 3D, faces with a polygon relationship are displayed with a pattern when you select the solid. Changing one face or edge affects all the faces in the relationship.

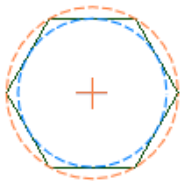


This section contains the following topics:

5.2.11.6.1. Drawing a Polygon

1. Click **Polygon** in the Sketch group.
2. (Optional) To dimension the polygon based on the diameter of a circle inscribed within the polygon, select the Internal radius option. Clear this option to dimension the polygon based on a circumscribed circle.

   The blue circle is inscribed within the polygon and the orange circle is circumscribed around it.



3. Click to set the center of the polygon.
   You can dimension the points relative to other sketch objects.
4. Drag the mouse to draw the polygon and change its orientation.
   The orientation is the polygon's angle relative to the X and Y axis.
   You can press **Tab** and type a number to change the diameter, orientation, or number of sides.
5. Click to complete the polygon.
   The sides of the polygon are all related, and act as one object. When pulled in 3D, the edges and faces of the polygonal solid will also maintain this relationship.
   Tip If you trim a polygon sketch with the Trim Away tool, you can drag the original sides of the polygon with the Select tool to recreate the polygon.

5.2.11.6.2. Setting the Number of Sides

Press **Tab** and type a number while you draw a polygon.

or

1. In Sketch mode, select the polygon with the Select tool.
2. Right-click the polygon and select **Properties**.
3. Enter a value for the **Number Of Sides** property.
   Polygons can have a minimum of 3 sides, and a maximum of 64 sides.

5.2.11.6.3. Removing the Polygon Relationship from the Faces of a Polygon Solid

Right-click a face of the polygon and select **Remove Association**.

Any changes you make to the face of the polygon will affect only that face.

5.2.11.6.4. Polygon Options

The following tool options are available:

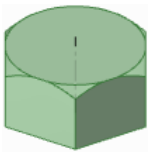
Internal radius	Dimension the polygon based on the diameter of a circle inscribed within the polygon.
Automatically create constraints	Select this sketch option to automatically generate constraints based on shapes created or on snaps. This option is enabled by default.
Snap to grid	Select this sketch option to turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units.

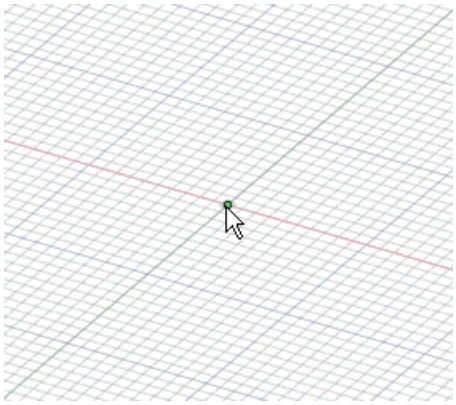
Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Snap to angle	Select this sketch option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Fit Curves	Disable this mesh option if you do not want the system to fit curves through the points.
Tolerance	This mesh option determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
Auto-merge	When this mesh option is enabled, the system will merge lines and arcs to form splines. Splines are displayed pink.

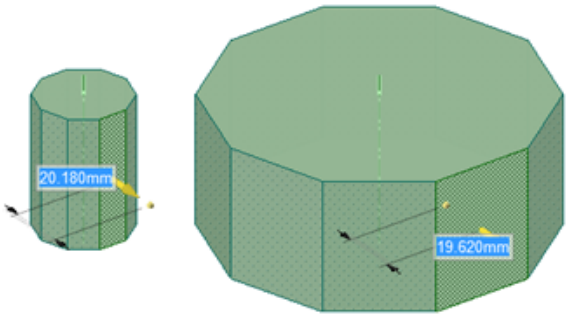
5.2.11.6.5. Polygon Examples

Hexagonal head nut.





A polygon pulled into a solid maintains the relationships between its sides. In this example, pulling one side pulls all sides of the polygon.



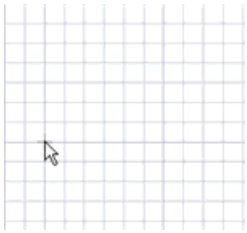
5.2.11.7. Splines

This section contains the following topics:

5.2.11.7.1. Spline Tool

Use the **Spline** tool to sketch splines in 2D or to draw splines between points on objects in 3D. A spline is a continuously curved line, without sharp boundaries (that is, without vertices). Splines can become edges when you pull your sketch into 3D with the **Pull** tool. Sweeping along a spline in 3D lets you create smooth, curvy shapes.

The sketch grid must be visible in the workspace before you can draw.



This section contains the following topics:

5.2.11.7.1.1. Drawing a Spline on a Sketch Plane

1. Click **Spline** in the Sketch group.
2. If a sketch plane is not active, click an object or objects in the Design window to make a sketch plane.
3. Click to set the first point of the spline.

   Hold **Alt** and click to set the point if you want it to be tangent with an adjacent sketch or edge. When highlighting the vertex (which is actually an end point of one of the intersecting edges which has influence at the time) scroll the mouse wheel to select other adjacent edges with which to set the tangent vector direction, as previewed by the green tangency symbol.
4. Click to set the next points of the spline.

   You can dimension spline points by entering the coordinate distance from the start point to each point, or dimension each point relative to another sketch object.
5. End the spline:
   - Double-click to set the end point of the spline.
   - Hold **Alt** and click to make the end point tangent with an adjacent sketch or edge.
   - Right-click and select **Finish Spline**.
   - Press **Esc**.
   - Connect the end point to the start point.
   - Click any other tool (except the Clipboard and Orient tools).

The common sketch options are described in Sketch Tool Guides and Options on page 133.

5.2.11.7.1.2. Drawing a Continuous Spline

1. Click **Spline** in the Sketch group.
2. Select **Draw continuous spline** from the Spline section of the Sketch options panel.
3. Click and drag to draw the spline.
4. Release to finish drawing.

5.2.11.7.1.3. Drawing a Spline Between Points in 3D

1. Select **Enable 3D Sketch**.
2. Click **Spline** in the Sketch group.
3. Click to set the first point of the spline.

   Select any point on an object in your design. Hold **Alt** and click to set the point if you want it to be tangent with an adjacent sketch or edge. When highlighting the vertex (which is actually an end point of one of the intersecting edges which has influence at the time) scroll the mouse wheel to select other adjacent edges with which to set the tangent vector direction, as previewed by the green tangency symbol.
4. Continue selecting points to draw spline points.
5. End the spline:
   - Double-click to set the end point of the line.
   - Hold **Alt** and click to make the end point tangent with an adjacent sketch or edge.
   - Right-click and select **Finish Line**.
   - Press **Esc**.
   - Connect the end point to the start point.
   - Click any tool (except the **Clipboard** and **Orient** tools).

5.2.11.7.1.4. Creating a Closed Spine

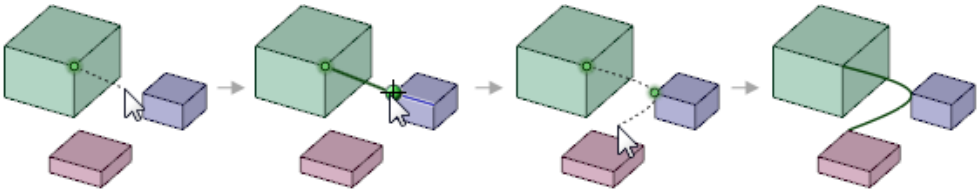
You can create a closed spline in the following ways:

- When drawing a spline, end it on the start point.
- When editing the spline, drag one end point on top of the other point.

Once you have created a closed spline, you cannot edit it into an open spline.

5.2.11.7.1.5. Sketch Spline Example

Sketching a spline between points in 3D.



5.2.11.7.1.6. Spline Options

The following tool options are available:

Automatically create constraints	Select this sketch option to automatically generate constraints based on shapes created or on snaps. This option is enabled by default.

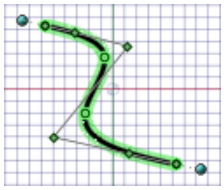
Snap to grid	Select this sketch option to turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units.
Snap to angle	Select this sketch option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Fit Curves	Disable this mesh option if you do not want the system to fit curves through the points.
Tolerance	This mesh option determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
Auto-merge	When this mesh option is enabled, the system will merge lines and arcs to form splines. Splines are displayed pink.

5.2.11.7.2. Editing a Spline

You can drag a spline, its spline points, or its control points while the Select tools is active. You can be in 2D or 3D mode.

Spline points are the points along the spine that define the position of its curves. Spline points are displayed as circles when you hover over a spline.

Control points are displayed outside of the spline, and define the curvature of the spline between points. Control points are displayed as diamonds connected by a dotted line when you select a spline.



This section contains the following topics:

5.2.11.7.2.1. Moving a Spline

1. Click the **Select** tool in the Edit group on the Design tab while in 2D mode.
2. Mouse over the spline to highlight the spline and display its defining points.
3. Click anywhere on the spline except for the defining points, and drag it to move it.

5.2.11.7.2.2. Editing a Spline

1. Click the **Select** tool in the Edit ribbon group while in 2D mode.
2. Mouse over the spline to highlight the spline and display its defining points.
3. Click and drag on any spline or control point to move it, leaving the other spline and control points fixed in space. You can box-select spline points.
4. (Optional) Edit the dimensions associated with that point.

5.2.11.7.2.3. Changing Spline End Points

1. Select the spline to display its end point control handle.

   The end point control handle is a light blue point at the end of a dotted line extending outwards from the end of the spline. In some cases, these end points may be located some distance away from your sketch. Zoom out from the sketch until you can see the end point control handle.

   If another line, arc, or spline shares that end point, the end point influence may snap into tangency with that sketch entity. To adjust the end point influence in this case, move the mouse a short distance away from the end point to display the end point control handle.

   After it is moved, the end point will snap to its original tangency.
2. (Optional) In 3D, hold **Alt** and select a planar or linear object that you want the spline to be tangent with. You can hold **Alt** and select a curve, edge or face if the curve, edge or face passes through the point. If you select a face, the tangency at the point where the point intersects the curve or surface is used. The default is the tangency direction with the least amount of curvature.
3. Drag the end point control handles to change the influence of that end point on the shape of the spline.

   The control handle will snap back to its initial tangent direction, which is indicated by a dotted line.

   The amount of the end point's influence can be controlled by dragging the end point control handle closer to or further away from the end point.

5.2.11.7.2.4. Adding Spline Points

1. Click the **Select** tool in the Edit group on the Design tab while in 2D mode.
2. Mouse over the spline to highlight the spline and display its defining points.
3. Right-click anywhere on the spline except on the defining points, and select **Add Knot** to create another spline point at that location.

5.2.11.7.2.5. Removing Spline Points

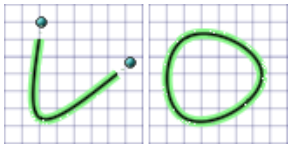
1. Click the **Select** tool in the Edit group on the Design tab while in 2D mode.
2. Click the spline to highlight it and display its defining points.
3. Right-click the point you want to remove and select **Remove Spline Point**.

   The spline adjusts to accommodate the removal of the point.

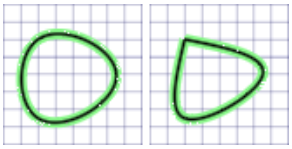
5.2.11.7.2.6. Changing the End Point Tangency of a Spline

1. Click the **Select** tool in the Edit group on the Design tab while in 2D mode.
2. Select the spline in the Design window.
3. Set the value for **Periodic** in the Properties panel:

   - **True**: If you change this property to True, the tangencies of the beginning and end of the spline curve will match to create a closed curve. If a spline curve is open and you set the property to True, the spline will be closed, as shown here:

- **False**: If you change this property to False, the end tangencies of a closed curve don't match. If a spline curve is closed and you set the property to False, the curve will become teardrop-shaped because the end points will no longer be tangent, as shown here:

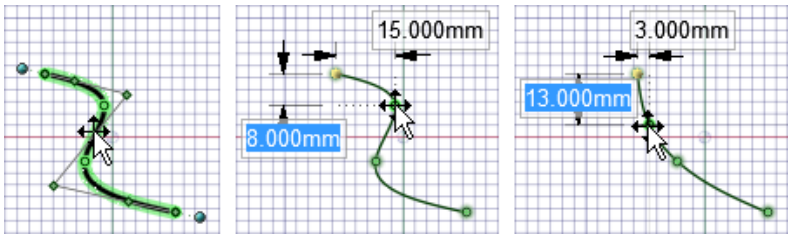


5.2.11.7.2.7. Extending a Spline by Pulling

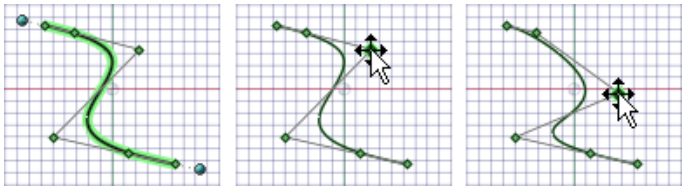
1. Click the **Pull** tool in the Edit group on the Design tab while in 3D mode.
2. Click an end point of the spline and drag in the direction of the Pull arrow.
   You can also click the end point, click the **Up To** tool guide, then click the object up to which you want to pull.

5.2.11.7.2.8. Editing Spline Examples

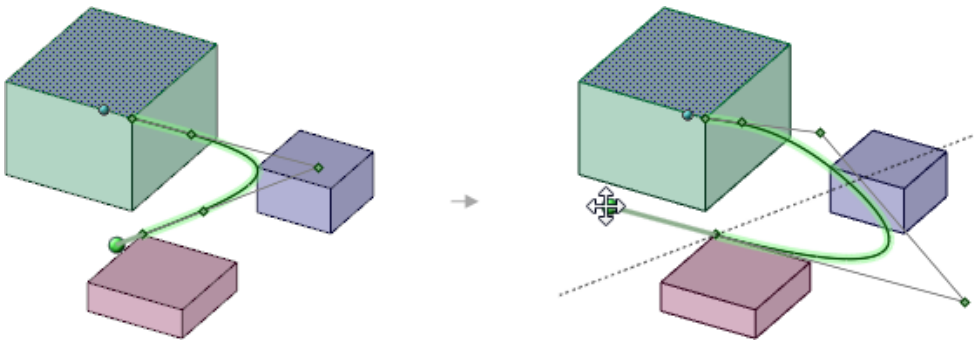
Dragging a spline point.



Dragging a control point.



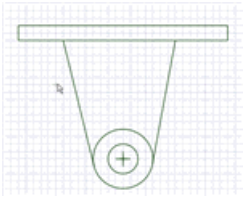
Dragging end point control handles in 3D.



5.2.11.8. Point Tool

Use the **Point** tool to sketch points in 2D and 3D. Points are useful as a dimensional reference, for splitting, and for creating a point on a line or curve through which you want to draw a three-point circle. You can also project vertices in 3D to create points in the sketch.

The sketch grid must be visible in the workspace before you can draw.



This section contains the following topics:

5.2.11.8.1. Drawing a Point

1. Click **Point** in the Sketch group on the Design tab.
2. Select a mode:
   - If a sketch plane is not active, press **Esc** to sketch in 3D or click an object or objects in the Design window to make a sketch plane.
   - If a sketch plane is active, click **3D Mode** in the Mode group or press **D** to sketch in 3D.
3. (Optional) Dimension the point relative to another object.
4. Click to place a point:
   - **On a face or surface**: click a location on a face or surface.
   - **On an edge or curve**: click a location on the edge or curve.
   - **On the midpoint of a straight line between two points in 2D**: hold **Alt** and **Shift** and select two points, then click the temporary point at the midpoint.
   - **On the midpoint of a straight line between two points in 3D**: select two points.
   - **Projected onto a face**: select a point and then a face to project onto.
   - **Projected onto an edge**: select a point and then an edge to project onto.
   - **In a UV direction within a face/surface**: select an existing point on a face and then a location on the face to specify the direction.
   - **Along an edge from an existing point on the edge**: select a point on an edge and a location along the edge.

   You can also use temporary points, axes, and planes to help you create points. When hovering over, or selecting on a point with the **Select** tool, the X, Y, Z locations of the point display in the status bar.

5.2.11.8.2. Point Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select	Use the **Select** tool guide to select a location for the point.

Select Direction	Use the **Select Direction** tool guide to change the direction of the dimension for the point.

Point Tool Options

The following tool options are available:

Automatically create constraints	Select this sketch option to automatically generate constraints based on shapes created or on snaps. This option is enabled by default.
Snap to grid	Select this sketch option to turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units.
Snap to angle	Select this sketch option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Fit Curves	Disable this mesh option if you do not want the system to fit curves through the points.
Tolerance	This mesh option determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
Auto-merge	When this mesh option is enabled, the system will merge lines and arcs to form splines. Splines are displayed pink.

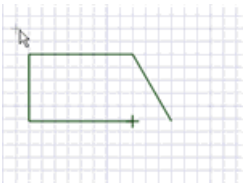
5.2.11.9. Trimming and Modifying

This section contains the following topics:

5.2.11.9.1. Create Rounded Corner Tool

Use the **Create Rounded Corner** tool to trim back or connect two intersecting lines or arcs so that they meet with an arc tangent at both ends. You can also use this tool with coplanar 3D lines, such as lines you created by blending.

The sketch grid must be visible in the workspace before you can create rounded corners, which are also known as fillets.



This section contains the following topics:

5.2.11.9.1.1. Connecting Two Lines with a Rounded Corner

1. Click **Create Rounded Corner** from the Sketch group on the Design tab.
2. Select a line or arc.

The first line or arc you select will be extended, if necessary.

3. Hover over another line to preview the round that will be created.

   If you hover over a line that is the edge of a solid, that edge will not change. Only a sketched line will change. The radius of the arc is previewed as you move the mouse. Specify a radius to dimension it.

4. Select an intersecting line to trim the lines, or a non-intersecting line to connect the lines.

   Where you click the intersecting line defines the radius of the tangent arc, or you can dimension the round (or chamfer) by entering its diameter.

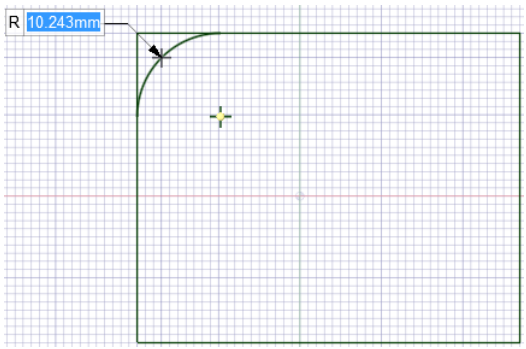
   If the lines intersect, click the part of the intersecting line you want to keep. If the lines do not intersect, you can click anywhere on the line to extend the first line to form a rounded corner. Circles are never trimmed.

For lines that meet at a corner, you can select and drag the vertex.

You can also select the tool first and then make your selection.

5.2.11.9.1.2. Creating Multiple Rounded Corners using Ctrl

1. Sketch some curves with several vertices (for example, a rectangle).
2. Enter the **Create Rounded Corner** tool.
3. Click a corner vertex and drag to preview the round.

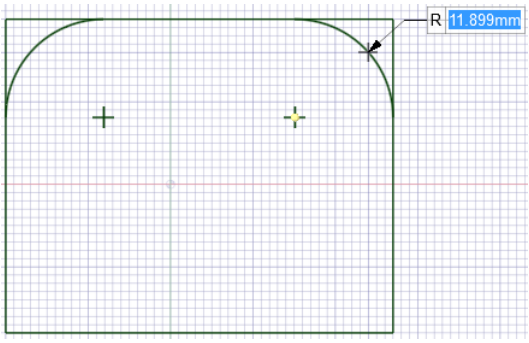


4. Press and hold the **Ctrl** key.

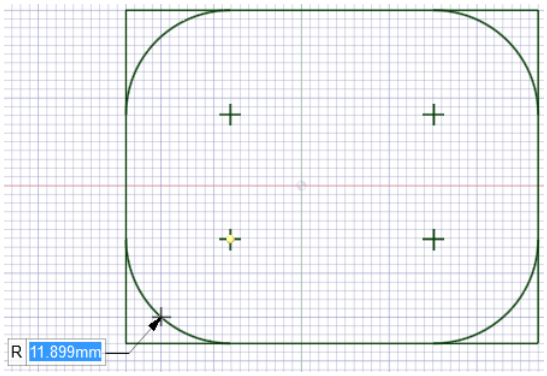
   The preview disappears.



5. While holding Ctrl, click another corner vertex. Then release the Ctrl key and move the mouse to preview BOTH rounds.



6. Press and hold the **Ctrl** key
   The preview disappears.
7. Repeat for the remaining corners.
8. On the last corner (meaning no more **Ctrl** key selection), you will see all rounds previewing simultaneously.
9. Click to place the rounds or specify a radius value.



5.2.11.9.1.3. Editing a Rounded Corner

Drag the rounded corner with the **Select** tool to change its radius. Tangency is maintained.

5.2.11.9.1.4. Create Rounded Corner Tool Options

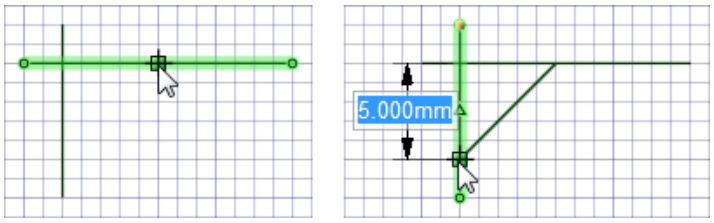
The following options are available with the Create Rounded Corner tool.

Chamfer	Create an equal-sided chamfer instead of an arc.
Trim	Leave the underlying lines instead of removing them.
Automatically create constraints	Select this sketch option to automatically generate constraints based on shapes created or on snaps. This option is enabled by default.
Snap to grid	Select this sketch option to turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units.
Snap to angle	Select this sketch option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.

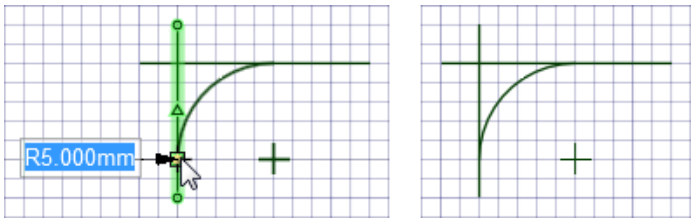
Fit Curves	Disable this mesh option if you do not want the system to fit curves through the points.
Tolerance	This mesh option determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
Auto-merge	When this mesh option is enabled, the system will merge lines and arcs to form splines. Splines are displayed pink.

5.2.11.9.1.5. Create Rounded Corner Examples

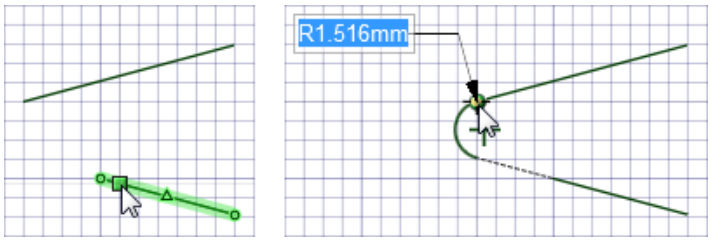
Rounding a corner with the Chamfer Mode option.



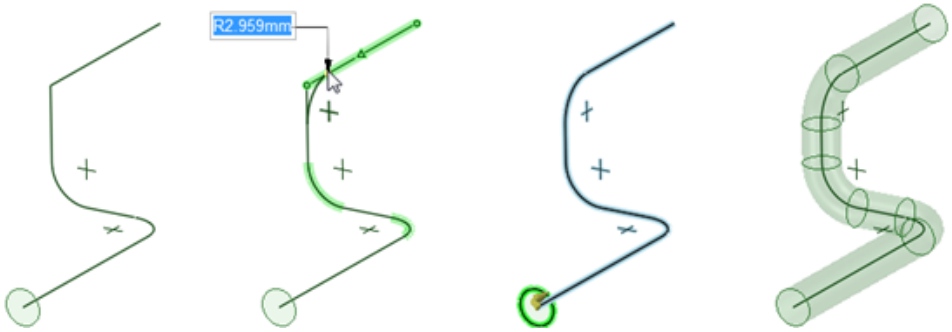
Rounding a corner with the Disable Trimming option.



Rounding a corner where the first line selected is extended.



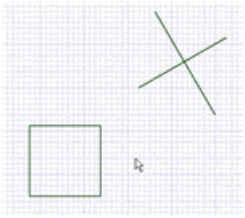
Rounding corners of 3D lines, then sweeping to create a pipe.



5.2.11.9.2. Create Corner Tool

Use the **Create Corner** tool to trim back or extend two lines so that they meet at a corner.

The sketch grid must be visible in the workspace before you can create corners.



This section contains the following topics:

5.2.11.9.2.1. Connecting Two Lines with a Corner

1. Click **Create Corner** in the Sketch group.
2. Click a line, arc, or spline.
3. Hover over another line to preview the corner that will be created.

   If you hover over a line that is the cross-section edge of a solid or a silhouette edge, it will not change. Only a sketched line will change.
4. Click an intersecting line to trim the lines, or a non-intersecting line to extend the lines.
5. If the lines intersect, click the part of the intersecting line you want to keep. If the lines do not intersect, you can click anywhere on the line to extend the lines to form a corner.

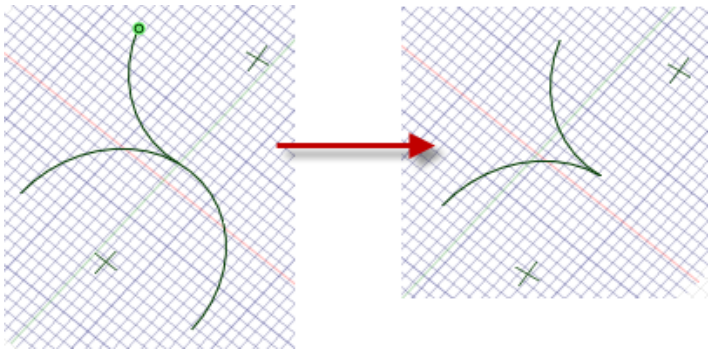
5.2.11.9.2.2. Create Corner Tool Options

The following tool option is available:

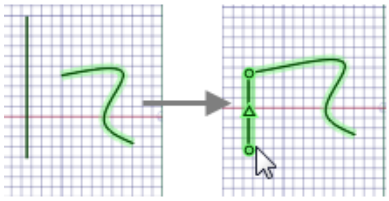
One side only	When more than one nonintersecting line is selected, only the first line will be trimmed or extended. If the two lines intersect, the first line is trimmed by the second line.

5.2.11.9.2.3. Create Corner Tool Examples

Creating a corner between arcs.



Extending a spline to the intersection with another sketch curve.



5.2.11.9.3. Trim Away Tool

Use the **Trim Away** tool to delete any line portion bounded by an intersection with a line or edge.

> **Note:** Line segments to be removed appear in red.

The sketch grid must be visible in the workspace before you can trim lines.

This section contains the following topics:

5.2.11.9.3.1. Deleting a Bounded Line

1. Click **Trim Away** in the Sketch group or press **T**.
2. Mouse over a line to preview the section that will be deleted.
3. Click to delete the highlighted line section.
   The portion of the line you clicked is deleted up to any intersection with another 2D line or edge of a solid.

5.2.11.9.3.2. Trim Away Options

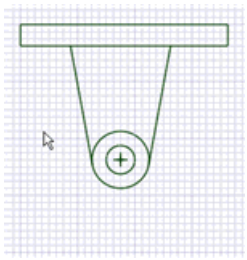
The following tool option is available:

Inverse trim	Trim curve segments other than those selected.

5.2.11.9.4. Offset Curve Tool

Use the **Offset Curve** tool to create an offset of any curve in the grid plane. If you are creating an offset spline, you will get the best results with an offset distance appropriate to the curvature of your spline.

The sketch grid must be visible in the workspace before you can create offset curves.



This section contains the following topics:

5.2.11.9.4.1. Offsetting a Curve

1. Click **Offset Curve** in the Sketch group on the Design tab.
2. Select the curves you want to offset.

   Standard curve selection techniques apply, so you can double-click to select connected loops of curves. Ctrl and Shift can also be used to extend or replace the selected items prior to offsetting.

   You can also use box, lasso, and paint select modes to select multiple sketch curves.

   If you select more than one curve, then they will be merged when they are offset.

   To clear the current selection, click an empty spot in the Design window.
3. Hover the mouse over a line or point and press **Shift** to create an offset from that curve or point.

   As you move the cursor away from the selected curve, you can see a preview of the offset result. The offset defaults to one grid square.
4. (Optional) Select an option to specify how intersecting offset curves should meet.
5. Enter a dimension or click to set the width of the offset.

5.2.11.9.4.2. Offset Curve Tool Options

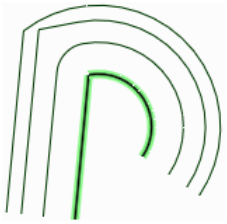
The following options are available in the **Offset Line** tool:

Offset both ways	Create two offset lines on either side of the selected line.
Close with corner	Close intersecting offset lines with a corner.
Close with arc	Close intersecting offset edges with an arc.
Close naturally	Close intersecting offset edges with a curve.
Automatically create constraints	Select this sketch option to automatically generate constraints based on shapes created or on snaps. This option is enabled by default.
Snap to grid	Select this sketch option to turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units.
Snap to angle	Select this sketch option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Fit Curves	Disable this mesh option if you do not want the system to fit curves through the points.
Tolerance	This mesh option determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated.
Auto-merge	When this mesh option is enabled, the system will merge lines and arcs to form splines. Splines are displayed pink.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

5.2.11.9.4.3. Offset Curve Tool Example

Highlighted line offset using each option. Shown from top to bottom: ***Close naturally****,* ***Close with corner****, and* ***Close with arc****.*



5.2.11.9.5. Split Curve Tool

Use the **Split Curve** tool to split one curve with another curve or point. You can also use a 3D object that intersects the curve, such as a plane or face. The segments of the split curve can then be selected and edited independently.

The sketch grid must be visible in the workspace before you can split curves.



This section contains the following topics:

5.2.11.9.5.1. Splitting a Curve

1. Click **Split Curve** in the Sketch group.
2. Click the curve you want to split.
3. Click a curve or point that intersects the curve you want to split. The curve will be split at the intersection.

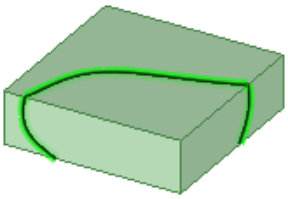
5.2.11.9.5.2. Split Curve Options

The following options are available:

Automatically create constraints	Select this sketch option to automatically generate constraints based on shapes created or on snaps. This option is enabled by default.
Snap to grid	Select this sketch option to turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units.
Snap to angle	Select this sketch option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.

5.2.11.10. Face Curve Tool

You can draw spline curves on a 3D face using the Face Curve tool. These curves follow the contour of a face. The face curve can be used like any other edge to modify the object.

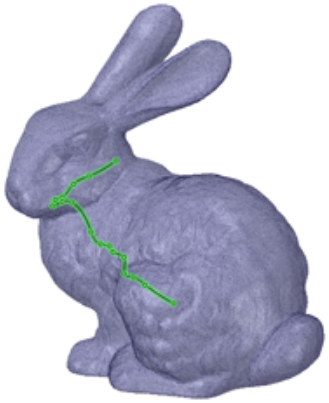


This section contains the following topics:

5.2.11.10.1. Inserting a Face Curve

1. Click **Face Curve** in the Insert group on the Design tab.
2. Click spots on the faces or edges of a part to draw the spline curve. Your spline can be drawn on one or more faces, but can only be drawn on one object.

   Face curves can also be sketched on mesh bodies by selecting facet points as spline points.



3. Double-click to end the curve or close the curve by clicking on your beginning point.

   Now you are in editing mode and can change the curve as desired. The curve will not be applied to your design until you complete the operation.
4. (Optional) Right-click a spot on the curve and select **Add Face Curve Point** to add a spline point.
5. (Optional) Right-click a spline point and select **Remove Face Curve Point** to delete the point.
6. (Optional) Drag points to change the curve.
7. (Optional) Set the value for **Periodic** in the Properties panel:
   - **True**: If you change this property to True, the tangencies of the beginning and end of the curve will match to create a closed curve. If a curve is open and you set the property to True, the spline will be closed.
   - **False**: If you change this property to False, the end tangencies of a closed curve don't match. If a curve is closed and you set the property to False, the curve will become teardrop-shaped because the end points will no longer be tangent.
8. Click the **Complete** tool guide or press **Enter** to apply the curve to your design.

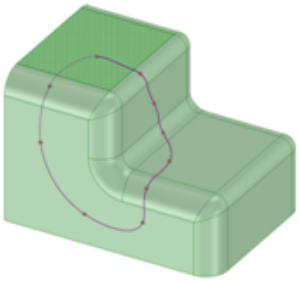
> **Tip:** You will get much more predictable results if the face to face boundaries you are drawing over are tangent. Click the edges (tangent or not) to place a point for more control over the intersection point at that boundary.

5.2.11.10.2. Face Curve Tool Guides

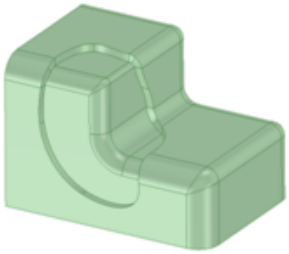
Complete Face Curve	The **Complete Face Curve** tool guide creates the face curve.

5.2.11.10.3. Face Curve Tool Examples

Sketching a face curve that is closed over tangentially-connected solid faces.



The result of offsetting the edges that were created as a result of the face curve above.



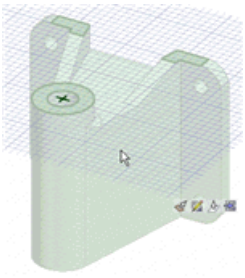
5.2.11.11. Project to Sketch Tool

Use the **Project to Sketch** tool to copy an edge or vertex of a solid to a 2D line or point that you can manipulate with the sketch tools. You can also project a note onto the sketch grid. Projecting an axis onto the sketch grid creates a construction line.

Except when explicitly selecting a new sketch plane, this is the only sketch tool that can select items that are not on the sketch grid. You can also select note text and project the text onto your sketch.

The sketch grid must be visible in the workspace before you can project edges onto it.

You can project a body onto a layout sketch plane on a drawing sheet.



This section contains the following topics:

5.2.11.11.1. Projecting an Edge, Vertex, or Note onto the Sketch Grid

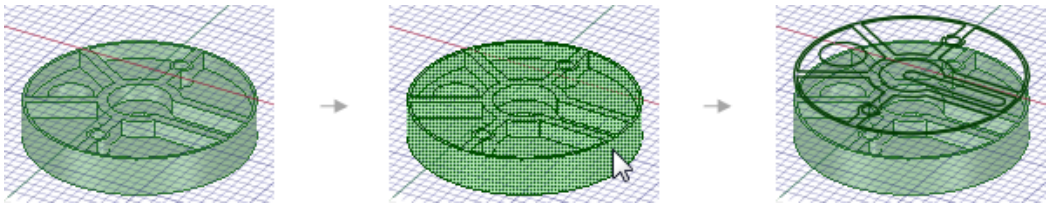
1. Click **Project to Sketch** in the Sketch group on the Design tab.
   Mouse over your design to highlight the objects available for projection.
2. Click the edge, edge chain, vertex, or note text you want to project into the sketch plane.
   Standard edge selection methods apply, so you can double-click, Ctrl+click, and Shift+click to create or modify your selection.
   You can also click and drag to draw a selection box enclosing the objects you want to project, including silhouette edges.
   The edge, vertex, or text appears as a sketched line to show that it is now a line or point in the sketch plane.
   Duplicate edges (that appear on top of each other) in the sketch are consolidated when you return to 3D mode.
   Tip If you have difficulty seeing a projected line, check that **Fade Scene Under Grid** is selected on the Display tab.

5.2.11.11.2. Projecting a Solid onto a Sketch Grid

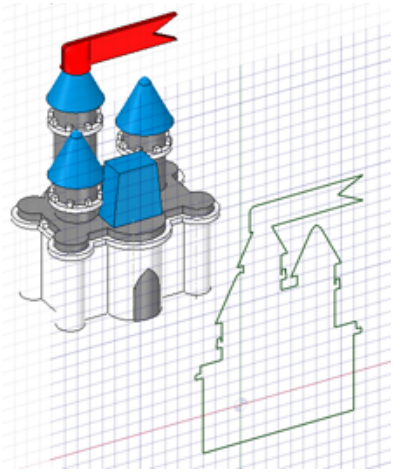
1. Click **Project to Sketch** in the **Sketch** group on the **Design** tab.
2. Click the **Select Bodies** tool guide if you want to select bodies in the design window.
   You do not need the tool guide to select bodies in the model tree.
3. Select one of the following options:
   - **All body edges** if you want all edges projected onto the sketch grid.
   - **Visible body edges** if you want only the visible, or silhouette, edges projected onto the sketch grid.
   - **Body outline** if you want to just project the body outline. Select a body in the Design window or model tree.

5.2.11.11.3. Project to Sketch Tool Examples

Projecting all body edges to the sketch grid.



Projecting the body outline to the sketch grid.



It may be necessary to use the tools in the **Fix Curves** group on the **Repair** tab if the sketch does not automatically close into a surface.

5.2.11.11.4. Project to Sketch Tool Guides and Options

Project to Sketch Tool Guides

The following tool guides are available:

Select	Select objects to project.
Select bodies	Select bodies to project.

Project to Sketch Tool Options

The following tool options are available:

All body edges	Project all body edges on to the sketch grid.
Visible body edges	Project only the visible or silhouette edges on to the sketch grid.
Layout Sketch	When selected, the created curves will be layout curves. These curves will not automatically fill or imprint on to faces when you return to 3D mode.

5.2.12. Working with Sketch Curves in 3D

Use **Enable 3D Sketch** to switch to sketching mode in 3D.

You can use the following sketch tools in 3D mode:

- Line
- Three-Point Circle
- Tangent Line
- Construction Line
- Spline

- Tangent Arc
- Three-Point Arc
- Point

You can use the following tools from the Sketch group to change curves in 3D mode:

Create Rounded Corner	Create Rounded Corner can be used on curves when they are coplanar.
Create Corner	Create Corner can also be used on coplanar curves.
Trim Away	Trim Away can be used to trim curves in 3D mode.
Split Curves	Split Curves can be used to split curves in 3D mode.

This section contains the following topics:

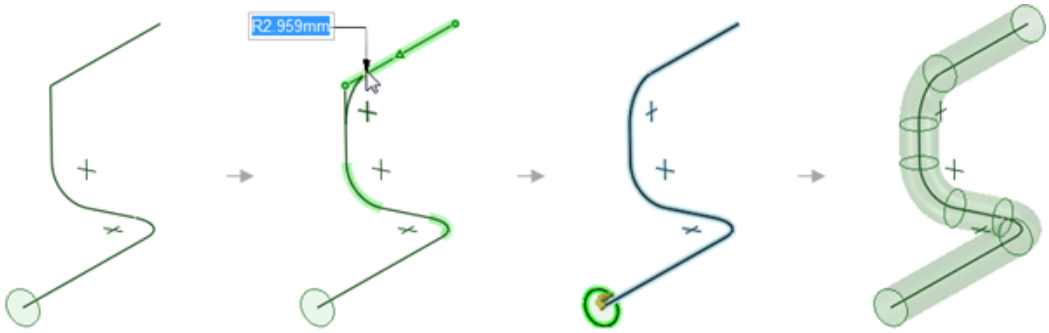
5.2.12.1. Using Other Tools to Create and Edit Sketch Curves in 3D Mode

There are several methods that can create or edit sketch curves in 3D mode:

- Use the Pull tool to pull a point into a line or sweep a point along an edge.
- Use the Move tool to translate or rotate curves in 3D mode.
- Use the Face Curve tool to sketch on a face or surface.
- You can use the Extend tool on the Prepare tab with sketch curves in 3D mode.
- You can change the radius of a circle or an arc directly in 3D by selecting it with the Select tool.

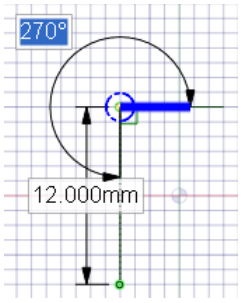
5.2.12.2. Sketch Curves in 3D Examples

Sketching a path and sweeping to create a pipe.



5.2.13. Editing Sketches

You can edit the dimensions of sketched lines, arcs, and points using the **Select** tool. Tangency is maintained when you edit a sketch.



This section contains the following topics:

5.2.13.1. Editing a Sketch

1. Click the **Select** tool. (You can also press **Esc** if you are in a sketching tool.)

   Select the Select Reference Curve tool guide to dimension from another object, similar to the Shift+hover functionality in other tools.
2. Click and drag the line or point you want to edit.

   Alt+click and drag if you want to detach the line or point before moving it.

   Ctrl+click and drag to create a copy.

   Enter a value to dimension the move.

5.2.13.2. Editing the Length and Angle Dimensions of a Line

1. Click the end point of a line or arc to edit the length or diameter and angle or chord angle dimensions of the line or arc.
2. Mouse over the opposite end point of the line or arc to display a blue circle.
3. Drag to move the reference point for the dimension's length.
4. Enter the dimension for the length from the end point of the line to the new reference point.
5. Mouse over the line showing the end of the angle reference to highlight it in blue.
6. Drag the blue line to move the reference for the angle dimension.

   If you move the angular reference 360°, the angle measurement is changed from clockwise to counterclockwise.
7. Enter the dimension to adjust the angle of the line from the new reference line.

5.2.13.3. Editing the Distance from Another Point

1. Click the midpoint of a line, the center of an arc, or any sketched point to display a dimension to a reference point.
2. Mouse over the reference point, then drag the blue circles to the location you want to dimension from.
3. Click to set the new reference location.

   To skew the X-Y lines, click the outer blue circle and enter the skew angle.

5.2.13.4. Changing the Properties of a Sketch Object

1. Click the **Select** tool. (You can also press **Esc** if you are in a sketching tool.)
2. Select the sketch object to display its properties in the Properties panel.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

You may need to adjust the width of the Properties panel or the columns within the properties panel. Place your mouse over the vertical column line and drag to widen the column. Place your mouse over the border of the panel and drag to widen the panel.

3. You can:
   - Select a new color from the Color property to color the object.
   - Switch between lines and construction lines by selecting a value from the Construction property.
   - Switch between straight lines and mirror lines by selecting a value from the Mirror property.
   - Enter the number of sides for a polygon in the Number of Sides property.

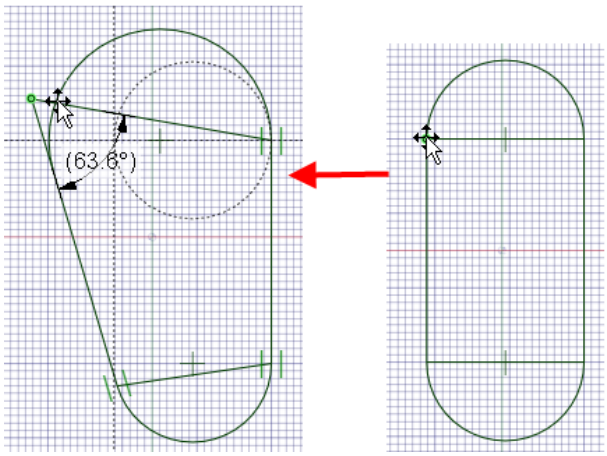
5.2.13.5. Helpful Tool Guides

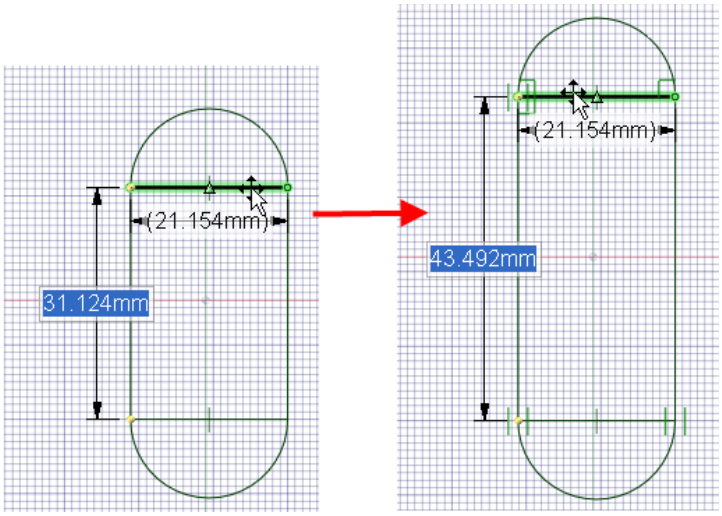
The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Reference Curve	Select the **Select Reference Curve** tool guide and click another sketch line to edit the dimensions from the selected point to that sketch line.
Move Dimension Base Point	Use the **Move Dimension Base Point** tool guide and click any point to edit the dimensions from the selected point to that point.
Change Dimension Reference Angle	Use the **Change Dimension Reference Angle** tool guide and click to set the angle reference for the selected point.

5.2.13.6. Editing Sketches Examples

Editing a racetrack by dragging internal lines. You can drag to resize, rotate, lengthen, or make one side larger than the other by dragging different lines or points in the sketch.





5.2.14. Copying a Sketch

You can copy a sketch line in Sketch mode with the **Select** tool.

Ctrl+drag (with the **Select** tool) while sketching to copy a sketch line.

Press the spacebar while Ctrl+dragging a sketch line to dimension the distance between the first line and the second, copied line.

You can also select a sketch line, copy it, and then select a point to paste the line onto the point.

5.2.15. Layout Sketches

You can sketch on a plane when you want to draw curves but have no immediate need to generate 3D objects. If you want to create a region that can be pulled into 3D right away, create a sketch instead.

You can think of a layout as a pencil drawing made on your design. If you try to pull layout lines to 3D, they do not behave the same way that sketched lines do. Closed lines are not converted to regions, so if you pull a layout line, it creates a surface, not a solid. When you are ready to use your layout to create geometry, project the layout lines to a sketch. Projecting a layout line to a sketch is like inking the line.

Layouts always appear on planes in the model tree.

You can use layers when working with layouts to help organize your design. For example, you can color individual lines on each layout, show or hide the lines, or put the bounding planes on a separate layer and turn that layer's visibility off to declutter your design. (When you import files, they will initially appear in one color.)

This section contains the following topics:

5.2.15.1. Creating a Layout

1. Insert a plane.
2. Right-click the plane in the model tree or graphics window and select **Activate Curve Container**.
   The icon on the model tree changes to reflect that the plane is a layout.
3. Sketch on the plane.

Alternatively, while sketching, enable **Layout Sketch** in the Sketch Tool Options to create the sketch curves as layout curves.

5.2.15.2. Converting a Sketch to a Layout

1. Select the sketch entities that you want to appear on the layout.
   - Select in the graphics window
   - Select in the model tree under the **Curves** folder
2. Click the **Create** > **Plane** tool.
   - A layout plane is drawn around the selected sketch entities
   - The curves are removed from the **Curves** folder in the **Design** tree
   - If all of model curves were converted, the **Curves** folder is removed from the **Design** tree
3. Right-click the plane in the model tree or graphics window and select **Activate Curve Container**.

5.2.15.3. Importing a 2D AutoCAD DXF or DWG File as a Layout

You can import a 2D AutoCAD file into your design in the following ways:

- Drag and drop the DXF or DWG file into the Design window to create a layout in the active component.
- Drag the file onto a plane in the model tree to place the drawing on that plane and convert it to a layout.

5.2.15.4. Editing a Layout

When constraint-based sketching is enabled (default), use the **Add Layout Curves** option to add layout curves to the imported datum. See Adding Layout Curves to an Imported Datum on page 138 for details.

When constraint-based sketching is disabled, right-click the plane in the model tree or graphics window and select **Activate Curve Container**.

A layout plane is drawn around the selected sketch entities.

5.2.16. Moving in Two Dimensions

Use the **Move** tool to move lines or points with the Move handle. When you move points or lines with this tool, they do not maintain their connections to other lines or points. If you want to maintain the connections in your sketch, use the Select tool to edit the sketch.

This section contains the following topics:

5.2.16.1. Moving Sketch Entities

1. Click **Move** in the Edit group on the Design tab.
2. Select the curve or curves that you want to move.
3. Select the following options:
   - **Move grid**: Select this option to move the sketch grid.
   - **Ruler**: Can be used to dimension the move. See the instructions below.

- **Create patterns**: Create a pattern of the selected objects. Pattern relationships of sketch curves and points are lost when you go into Section or 3D mode.
- **Maintain sketch connectivity**: Keeps the connections between a sketch curve and other curves that share its end points. If you deselect this option and move a sketch curve, the curve will move independent of other curves.
- Other options in the Options panel are not relevant when you move sketch entities.

4. (Optional) To change the anchor location of the Move handle
   - Drag the yellow center sphere on the Move handle.
   - Click the **Anchor** tool guide and select the face, edge, or vertex on which to place the Move handle.

     The yellow center sphere turns into a blue cube when the Move handle is anchored.
5. (Optional) To change the direction or trajectory for the move,
   - Drag one of the small balls on the rotational axis to reorient the Move handle, or dimension the orientation by typing the rotation angle *while you are dragging*, then pressing **Enter**.
   - You can also Alt+click a point or line, or click the **Move Direction** tool guide, then click a point or line, to orient one of the Move handle's axes toward that point or along that line.

     If you Alt+click a trajectory, you can move along the trajectory. Ctrl+Alt+click to add contiguous lines or edges to the trajectory.

     If you Alt+click a plane, the direction of movement is set perpendicular to the plane.
6. Click an axis and drag in that direction to move the selected object.

   A line extends from the Move handle axis to indicate the direction you selected for movement.
7. To dimension the move,
   a) Click **Ruler** in the Options panel.
   b) Click an edge or face to anchor the ruler.

      The ruler is oriented along the selected Move handle axis.
   c) Type a distance and press **Enter**.

5.3. Design Modes

In Discovery, there are three modes you can use to design: Sketch, Section, and 3D. These modes make it clear when you are working in 2D vs. 3D, and clarify the difference between working in cross-section (where you can alter 3D objects) and sketching (where you cannot alter 3D objects).

This section contains the following topics:

5.3.1. Switching Design Modes

1. Click a tool in the Mode ribbon group.

   You can also right-click an empty area of the Design window and select **Sketch Mode**, **Section Mode**, or **3D Mode** from the context menu, or press **K** (for Sketch mode), **X** (for Section mode), or **D** (for 3D mode).

   If you are sketching, you can also click **Return to 3D mode** in the Sketch plane mini-toolbar.
2. If you are switching from 3D mode to one of the other modes, select a plane to sketch on or create the cross-section.

When you select an origin object or its center and click Sketch mode, the sketch plane is aligned to the origin object's XY axis.

5.3.2. Flipping the View While in Section Mode

Right-click and select **Flip Viewing Direction** while section mode is active.

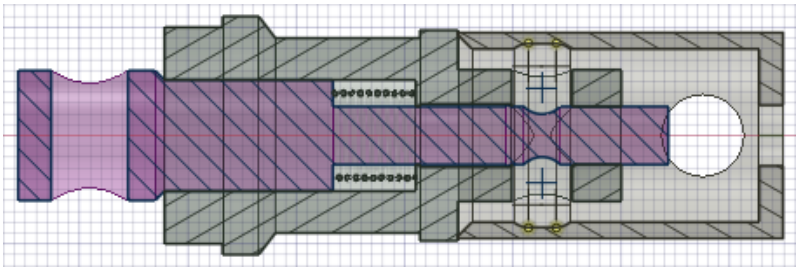
5.3.3. Mode Ribbon Group

The Mode ribbon group contains the following tools:

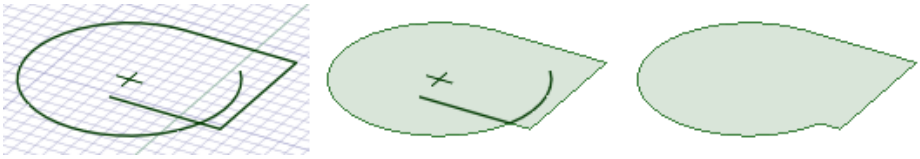
Sketch mode	**Sketch mode** displays the sketch grid, so you can use any of the sketch tools to sketch in 2D.
Section mode	**Section mode** lets you edit solids and surfaces by working with their edges and vertices in cross-section. You can also use all the sketch tools in Section mode to create and edit solids and surfaces in cross-section.
3D mode	**3D mode** lets you work directly with objects in 3D space.

5.3.4. Design Mode Examples

A design in section mode. The active component is displayed in the part color (magenta in this example) and the inactive components are displayed in gray.



Switching from Sketch mode to 3D mode automatically converts closed sketched curves into surface objects and the sketch curves become edges. Only the sketch curves that contribute to the closed surface are converted to edges. Leftover curves remain in the design as free curves.



5.3.5. Editing in Section Mode

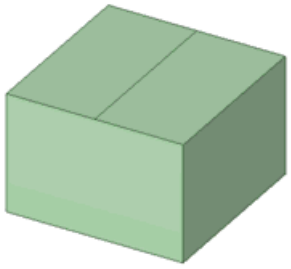
Use **Section** mode to edit solids by working with their edges and vertices in cross-section. You can modify faces, edges, planes, cylinders, rounds, and chamfers in section mode. You can edit solids and surface bodies.

In Section mode, lines represent faces and points (or vertices) represent edges.

For example, to rotate a face around an edge, select the line that represents the face, Alt+click the vertex that represents the edge, and pull. Moving a sketched line in Section mode does not move the solid it is sketched on. You must move a section line (a line that represents a face) to modify a solid in Section mode.

Hatching is used to show the intersection of the cross-section plane and a solid. Arc centers are shown as small cross marks. Hatching appears bolder inside faces to indicate what is shown in a cross-section view. (See Examples, below).

You can use the following tools: Select, Pull, Move, Combine, Split Body, Shell, Offset, Fill, and all sketch tools. Use the Select tool to edit spline faces (represented by a spline in cross-section). You can also cut, copy, and paste.You should clip the scene above the grid to enhance the visibility of the cross-section.



This section contains the following topics:

5.3.5.1. Editing in Cross-Section

1. Select or de-select options based on whether you want to maintain and view relationships while you edit in cross-section.
2. Select the face you want to use to create the cross-section plane, or select any faces, edges, or vertices that define a plane.

   If you are in a drawing sheet with cross-section views, you do not have to choose a face, as the plane of the drawing sheet is automatically used as the section plane.
3. (Optional) Move or rotate the cross-section grid and click the Section tool when you are finished.
4. Click and drag the edges and vertices of the cross-section to edit them.

   You can also bend edges with the **Bend** tool, and pull section points (edges) and section lines (faces) with the **Pull** tool.

   We recommend zooming into your design so that it is easier to select the correct entity. For example, if you are trying to select an edge, but your design appears very small in the Design window, it is possible to accidentally select a midpoint or end point of the edge instead. Moving the midpoint or end point of an edge will not be reflected by a mirrored entity.

5.3.5.2. Section Tool Options

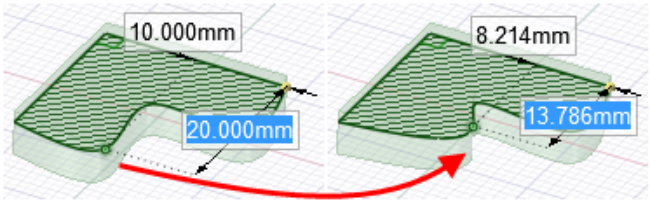
The following options are available in the Section tool:

Maintain Mirror	Select this option to maintain the influence of mirrors in your design while editing.
Maintain Offset	Select this option to maintain the influence of baselines in your design while editing. Baseline faces are shown as blue edges when viewed in cross-section.

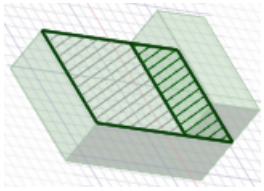
Curve Fitter Options	If the section plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves. • **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points. • **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated. • **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

5.3.5.3. Section Tool Examples

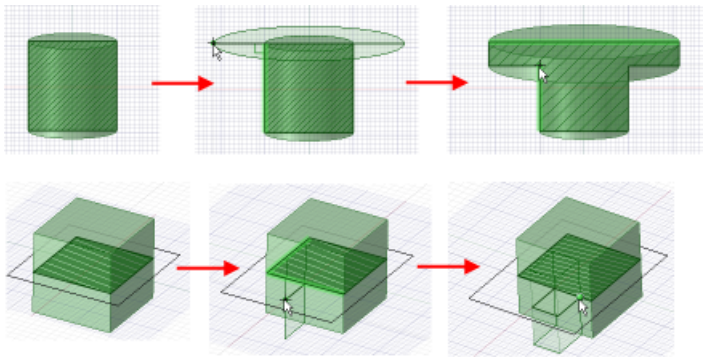
Moving a spline point in section mode.



Bold hatching indicates the hatching that would be shown on a drawing sheet cross-section view.



Extruding while sketching in Section mode.



5.4. Editing

Use the editing tools to create and edit 2D and 3D solids. You can select, pull, move (in 2D and 3D), edit in cross-section, fill (heal), bend lines and edges.

In Discovery, there is little need for a distinction between creating and editing. There is no hierarchical feature tree, so you have considerable freedom when designing. Create a box by pulling on a rectangular region. Edit the size of the box by pulling on one of its faces. Draw a rectangle to create a pull-able region. Draw a rectangle on a face to create a new face.

In general, you are either editing or creating with one of the main tools (Select, Pull, Move, Combine, or sketching and editing in Section mode), the many secondary tools, or inserting relationships between faces in the design (Shell, Offset, Mirror). Combining objects (intersecting, merging, cutting, etc.) is handled by the tools in the Intersect ribbon group.

Selection is integral to face and edge manipulation. You can extend selections with standard controls (double-click, Ctrl, Shift), by right-clicking and selecting from the Select menu, or using Discovery's power selection functions.

A quick note about solids and surfaces: Discovery always converts a closed set of surface faces into a solid. Similarly, sketched lines that clearly create regions on faces are replaced with real edges. The change in face transparency and edge lines reflects this transformation.

You can cut, copy, paste, and detach objects in most tools.

You can also nudge objects in most tools. Hold Ctrl+Alt and the up or down arrow to nudge an object. The up arrow increases the dimension and the down arrow decreases the dimension. If you nudge multiple times then click Undo, all the nudges will be reverted.

In Discovery, there are three modes you can use to edit your designs:

Sketch mode	**Sketch mode** displays the sketch grid, so you can use any of the sketch tools to sketch in 2D.
Section mode	**Section mode** lets you edit solids by working with their edges and vertices in cross-section.
3D mode	**3D mode** lets you work directly with objects in 3D space.

This section contains the following topics:

5.4.1. Edit Ribbon Group

The Edit ribbon group contains the following tools:

Select	Use the **Select** tool to select 2D or 3D objects in your design for editing. You can select vertices, edges, curves, planes, axes, faces, surfaces, rounds, solids, and components in 3D. In 2D, you can select points and lines. You can also select circle and ellipse centers, the midpoints of lines and edges, and the internal points and end points of splines. The Select tool can also be used to edit a sketch.
Pull	Use the Pull tool to offset, extrude, revolve, sweep, draft, scale, drape, and blend faces; use it to round, chamfer, or extrude edges.
Move	Use the Move tool to move one or more faces, surfaces, solids, or components. The behavior of the Move tool changes based on what you have selected.
Fill	Use the Fill tool to fill in the selected faces with the surrounding surface or solid. The region must be completely surrounded to be filled. You can also use the Fill tool to patch new surfaces through loops of edges.
Replace	Use the Replace tool to replace one face (or multiple faces) with another face (or faces). You can also use it to simplify a spline face that is very similar to a cylinder or plane, or to align a set of planar faces that are almost aligned.

5.4.2. Pulling with the Pull Tool

Use the **Pull** tool to offset, extrude, revolve, sweep, and draft faces. You can use it to round, chamfer, extrude, copy, or pivot edges. You can also drag a point with the **Pull** tool to draw a line on a sketch plane.

The **Pull** tool can also be used on facets in **Faceted Bodies**.

Pulling the apex of a cone changes its height. Pulling through the base plane will invert the cone. Pulling a loop of edges attached to a vertex will create conical faces at the corners when appropriate.

You can select a face, then pull, dragging anywhere to act, or you can click, drag, and release a highlighted face. In general, the result of a pull stays selected or highlighted after the pull operation.

The action of the Pull tool depends on which faces and edges you select to work with, and which faces, planes, or edges you select to drive the change. For example, if you choose to work with a face, then select an edge to "drive" the pull, the Pull tool infers that you want to pivot the face around that edge. When multiple actions can be inferred, you can use the tool guides to correct the Pull tool's inference. The Pull tool maintains any offset, mirror, pattern, or coaxial relationships.

When you pull a face, there are two main decisions you need to make. The first is to determine the direction you want to pull in. A default direction is offered to you, but it can be overridden using the **Direction** tool guide. The second is to determine what is going to happen at the edges of the face. By default, the edges of the face are determined by its neighbors, but you can override this behavior by including the edges in your Pull selection to create an extrusion. When you pull, connected chamfers are automatically removed and replaced.



Note: If you entered the **Design** tab with sheet metal features selected, the **Pull** tool will work as it does in Sheet metal. To work as usual, right-click the sheet metal part in the model tree and choose **Suspend Sheet Metal** in the context menu.

This section contains the following topics:

5.4.2.1. Creating and Editing a Solid

1. Click **Pull** in the **Edit** group on the **Design** tab or press **P**.

   Hover over faces and edges in your design to preview the natural Pull direction. If your mouse is over multiple faces or edges, use the scroll wheel to preview the Pull direction for each one.

2. Select the faces, edges, and points you want to pull to create 3D solids or surfaces.

   You can right-click in the Design window and select **Anchor Pull Handle**, then click to anchor the Pull handle on another object. This command is useful when you want to dimension a Pull from a different location than the center of a face.

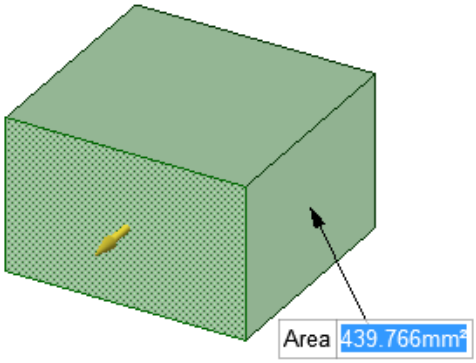
   If the Pull arrow appears gray instead of yellow, then the object is locked and can't be pulled unless you unlock it. To unlock an object, right-click the object in the model tree and select Unlock.

3. (Optional) Alt+click to select the face or edge that will drive the change.

You can also select the appropriate tool guide, then click the face, point, or edge. The driving edge or face is shown in blue.

Measurements can also be used to drive modifications:

a. Enter the **Pull** tool
b. Select a face, edge or vertex
c. Enter the **Measure** tool (shortcut is "e") and measure any single object or measure between two objects
d. Click in the **Measurement** results (hover over measurements to display purple box). Once selected, that single measurement will display on screen with arrows pointing to either object chosen for measurement.
e. Click in the highlighted dimension box and modify the value for a one-time adjustment of the model or create a Measurement Group which can be modified at any time.
f. Measurement groups can also be created from **Area** results. Modifying the group then adjusts the model to produce a desired area. For example, create a Measurement group for the area of a side of a box. While pulling the front of the box, you can enter a new area value for the side to complete the pull.



4. (Optional) Select options from the **Options** panel, or right-click and select them from the mini-toolbar.
5. Click and drag in the direction of the Pull arrow to create or edit a solid.
6. If the correct pull arrow is not highlighted, press **Tab** or click the Pull arrow you want to use.

- Press **Shift** to snap while pulling.
- Press the spacebar or click a dimension to enter a value.
- To dimension the pull, type the distance you want to pull and press **Enter**. You can also type a dimension in the dimension box in the mini-toolbar.
- You do not have to click and drag (or even click near) the Pull arrow. In fact, you should move the cursor away from the arrow for more precise control of the pull. The only important input to the Pull tool is the direction in which you are moving the cursor. When you pull, contiguous solids are automatically combined.
- If you want to pull in a head-on view, use the ruler at the bottom right corner of the Design window to pull instead of the Pull arrow. Slide the gray bar to the left to subtract material, and to the right to add material.
- If you pull through another object in the same component, the smaller object is merged into the larger one, and receives the larger object's properties. If you pull multiple, touching surfaces, the smaller surfaces are merged into the largest one. Select the **No Merge** option if you don't want objects merged.
- Hold the **Ctrl** key while pulling to create a copy of the object that is offset by the distance you pull. You can also hold Ctrl while revolving to create a copy of a surface. The influence of neighboring faces is taken into account to trim or extend the edges of the copied face, when appropriate. You can use the **Up To** tool guide with **Ctrl** to make a copy, and you can use the **Both Sides** option to make two copies that are offset in opposite directions from the original. You can also use the **Up To** tool guide to pull surfaces up to a reference edge.
- Press **Esc** to cancel the Pull.

See the topics in the Table of Contents under **Designing > Editing > Pulling** for the list of actions you can perform with the Pull tool.

Note: To do this faster:

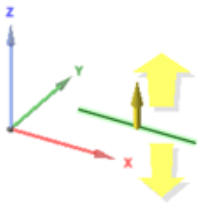
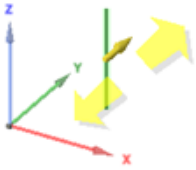
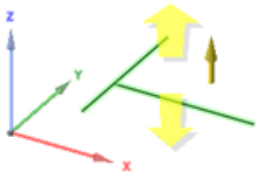
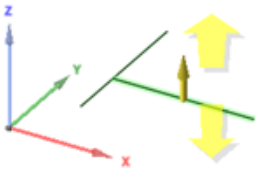
1. Select the faces, edges, and points you want to work with.

   In general, pulling a face increases the size of the solid, pulling an edge creates a surface, and pulling a point creates a line or curve.

2. (Optional) Alt+click the face or edge that will drive the pull.
3. Drag in the direction of the Pull arrow.

5.4.2.1.1. Pulling a Curve

When you select a curve with the **Pull** tool, the direction of the extrusion is determined by the plane of the curve.

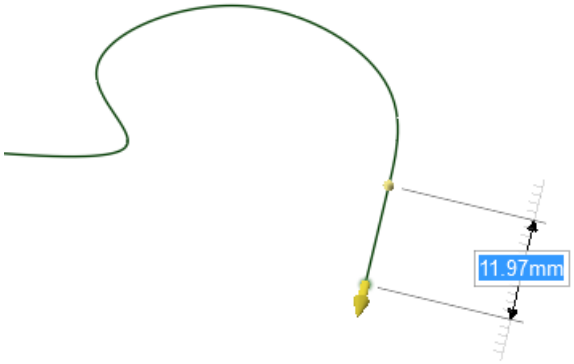
The default direction to pull a curve is along the Z axis.	
If a curve lies on the Z axis, the pull arrow points in the Y direction.	
If you select two curves on the same plane, the pull arrow points in the direction that is perpendicular to the plane of the curves.	
If you select a curve that touches another curve, the pull arrow points in the direction perpendicular to the plane of the two curves.	

5.4.2.1.2. Pulling Curve Ends

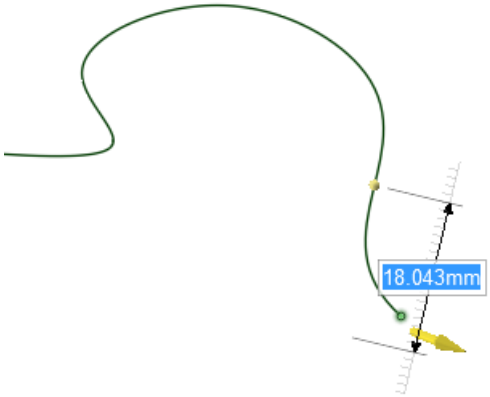
You can toggle between **Tangent** and **Natural** (curvature continuous) extension when dragging the end of a curve. To drag the extension Tangent to the curve, press and hold Ctrl prior to dragging. You can release the Ctrl key once you start dragging. If you do not press Ctrl, the extension is created Curvature Continuous.

- Works for splines (extended or trimmed-back)
- Works for arcs

Press Ctrl and drag to pull the curve end tangent to the curve.



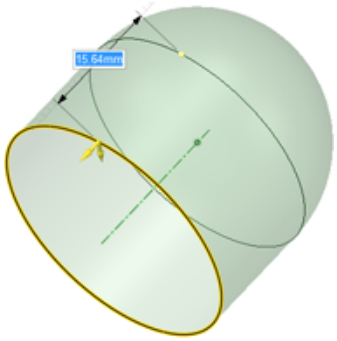
Without pressing Ctrl, pulling the curve end extends it with its natural curvature.



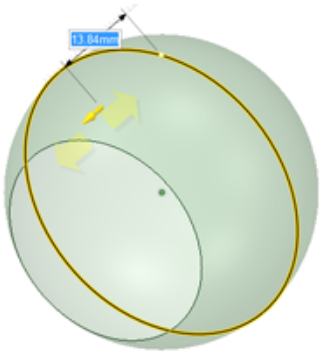
5.4.2.1.3. Pulling Surface Edges

Surface edges can be pulled normal to their neighboring face. Press the Ctrl key when you begin the pull.

Press Ctrl and drag to pull the edge tangent to the surface.



Without pressing Ctrl, pulling the edge extends it with its natural curvature.



5.4.2.2. Pull Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select	The **Select** tool guide is active by default. When this tool guide is active, you can perform standard selection tasks, and create natural offsets and rounds. Select a face, parallel faces, or surface edges to offset them. Select a solid edge to round it. Alt+click to select the driving face or edge for revolves, directed extrusion, sweeps, and drafts. Alt+double-click an edge to select an edge loop. Alt+double-click again to cycle through alternate edge loops. You can select objects across multiple components to pull.
Direction	Use the **Direction** tool guide to select a straight line, edge, axis, origin axis, plane, or planar face to set the pull direction.
Revolve	Select a face to pivot or select a face and edge to revolve. Then use the **Revolve** tool guide to select the straight line, edge, or axis around which you want to pivot or revolve.
Draft	Select any number of contiguous faces on the same body, then use the **Draft** tool guide to select the plane, planar face, or edge around which you want to pivot. None of the contiguous faces can be parallel to the neutral plane, face or edge around which you want to pivot.
Sweep	Use the **Sweep** tool guide to select the straight or curved lines or edges along which you want to sweep. Faces and edges can be swept, and the sweep trajectory cannot be in the same plane as the face.
Scale Body	Use the **Scale Body** tool guide to scale objects in 3D. See Scaling solids and surfaces.
Up To	Use the **Up To** tool guide to select the object that you want to pull to. The pulled object's face or edge will mate with the surface of the selected body or be pulled up to a plane through the selection. You can also use this tool guide to pull surfaces up to a reference edge or facet. When pulling Up To a Facet reference, a dimension is displayed for further editing. The object will be copied if you hold **Ctrl**. The selection used for **Up To** can come from the **Design** tree, if it can be determined. A single face surface is an example of an Up To selection in the model tree.

Full Pull	Extends an edge or face to the nearest face. This option works similar to the **Up To** tool guide, except you don't select the face to extend to. You can use **Full Pull** to automatically pull edges up to the closest faces that intersect with the object. The edges you select are extended in the direction of the Pull handle up to the next set of faces or edges that fully bound the extension. The original surfaces that the edges belong to are extended and new edges may be created. However, new faces are not created.

5.4.2.3. Pull Tool Options

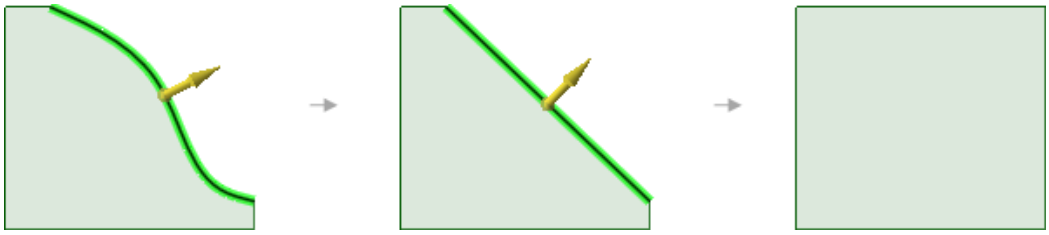
The following options are available in the **Pull** tool. Once you select the edge or face to pull, select these options from the Options panel or the mini-toolbar:

Add	Only add material when you pull. If you pull in a subtractive direction, no change will occur. You can combine this option with other Pull options.
Cut	Only remove material when you pull. If you pull in an additive direction, no change will occur. You can combine this option with other Pull options.
No Merge	Pulls without merging into other objects even when the object pulled intersects with an existing object.
Thicken surfaces	When this check box is selected and you pull a surface, the surface is extruded into a solid. This is the default behavior. When the option is deselected and you pull a surface, the surface is offset to a new location, changing the original surface. When you hold Ctrl and drag a surface with the option selected or deselected, the surface is copied and then offset.
Maintain Offset	Select this check box to maintain the offset relationship when pulling.
Pull Both Sides	Select a single, detached edge, imprinted edge, surface, or helix then click this option to pull both sides of the edge or surface at once.
Measure	Opens the **Measure** tool. Selecting a measurement result returns you to the **Pull** tool and displays the measurement value in a dimension box with an arrow pointing to the measured object. Modify the value for a one-time adjustment of the model or create a Measurement Group which can be modified at any time.
Mass Properties	Opens the **Mass Properties** tool. Selecting a mass property result returns you to the **Pull** tool and displays the measurement value in a dimension box with an arrow pointing to the measured object. Modify the value for a one-time adjustment of the model or create a Measurement Group which can be modified at any time.
Ruler	Select this option, then click to connect a ruler, oriented along the pull axis, to an anchor edge or face. You can use the ruler to dimension the pull. The direction must be specified to successfully create a ruler dimension. Press **Esc** to cancel the ruler dimension.
Round	When you are pulling an edge, select this option to create a rounded corner, which is also known as a fillet. **Note:** The default minimum radius is 0.1 mm. Smaller radii can be specified if the units are in micrometers, nanometers, or mils.

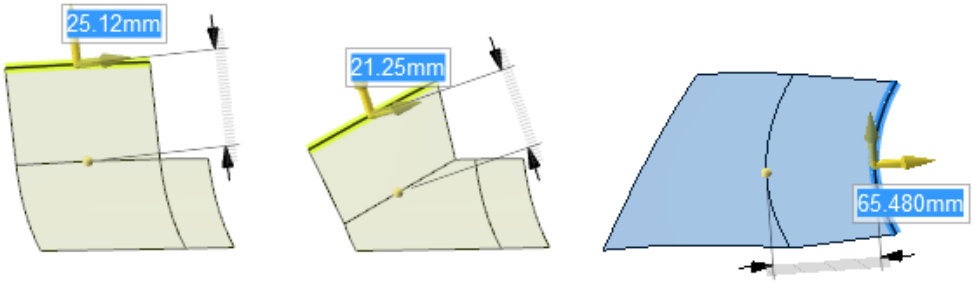
Chamfer	When you are pulling an edge, select this option to create a chamfer.
Extrude Edge	When you are pulling an edge, select this option to extrude the edge into a surface.
Copy Edge	When you are pulling an edge, select this option to create a copy of the edge. This option is selected by default. When this option is selected, all the points on the offset edge are the same distance from the initial edge.
Pivot Edge	When you are pulling an edge, select this option to pivot the edge along the selected Pull arrow.

5.4.2.4. Pull Tool Examples

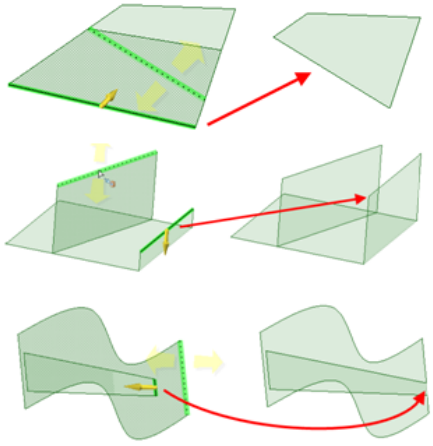
Pulling the edge of a surface first simplifies the edge, then its neighboring edges are extended (or trimmed).



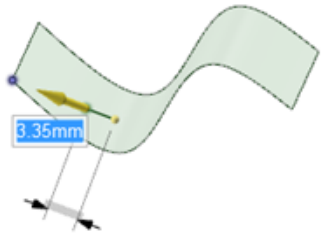
*Pulling the edge of a surface while holding **Ctrl** makes a new surface that is tangent to the edge..*



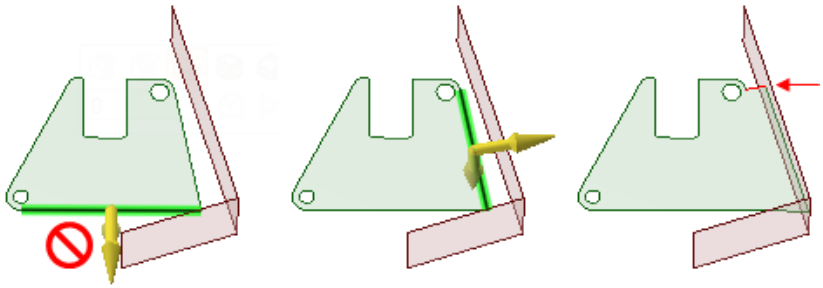
*Pulling edges up to other edges with the **Up To** tool guide.*



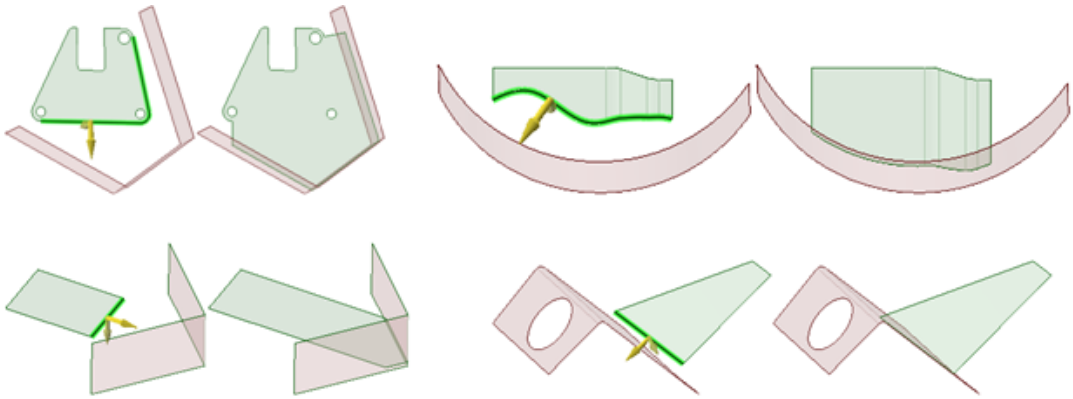
Pulling a point on a surface towards an Alt+clicked vertex.



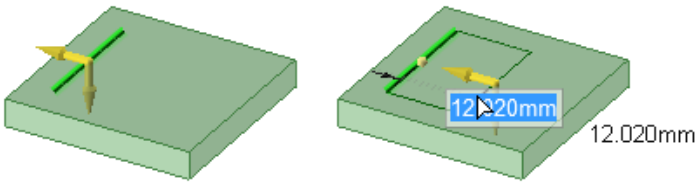
*Pulling edges with the **Full Pull** option. If you select the lower edge of the green surface shown below, you will receive an error because the neighboring surface does not extend past the end of the selected edge. The edge on the right side of the face can be pulled with the option, because the neighboring face extends beyond its length. A new edge is created, which is marked in red in the illustration on the right.*



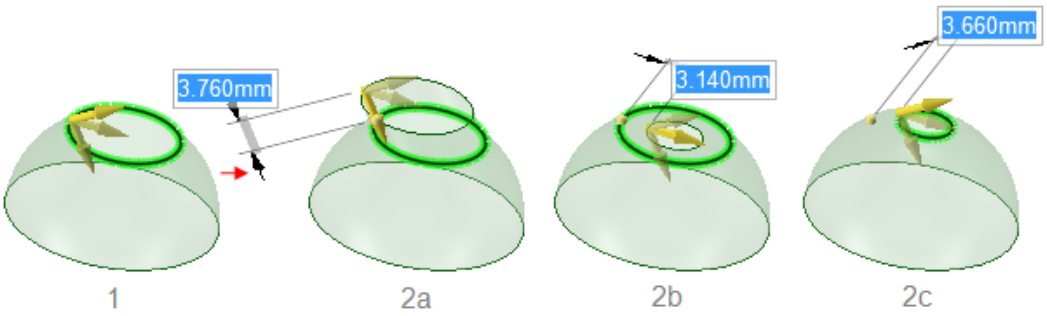
*Pulling edges to their nearest neighbor with the **Full Pull** option.*



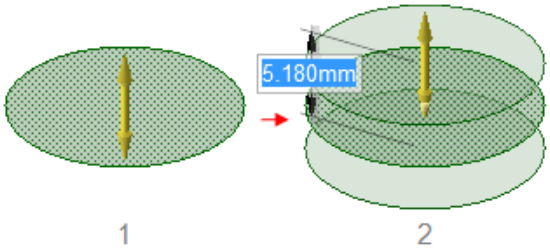
Pulling a sketched line on a planar face creates a surface in the same plane as the face



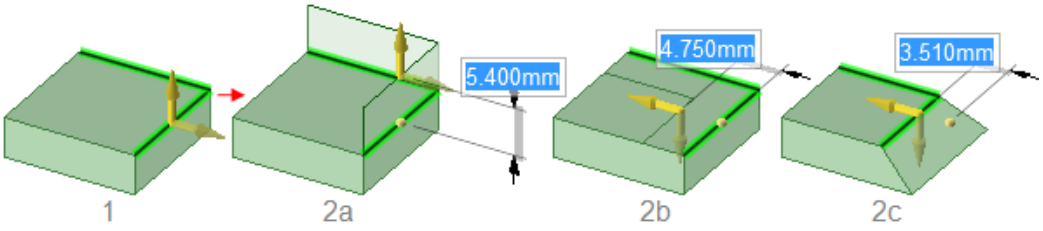
Pulling the edge of a toroidal surface. Three directions are available for pulling.



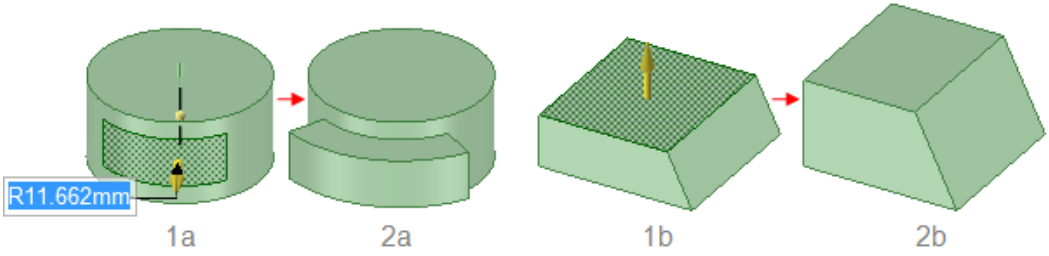
*Holding **Ctrl** while pulling a surface with the **Both Sides** option creates copies of a surface.*



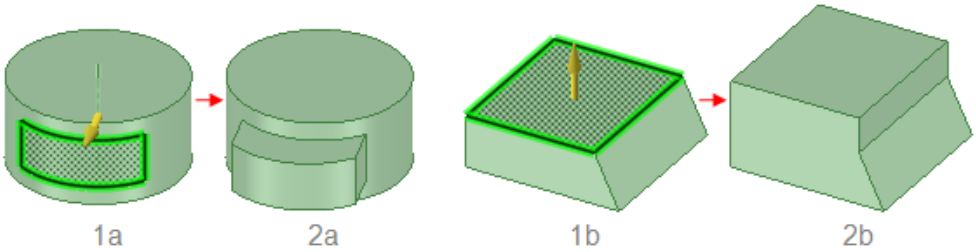
*Pulling two edges with the **Extrude** (2a), **Copy Edge** (2b), and **Pivot** (2c) options.*



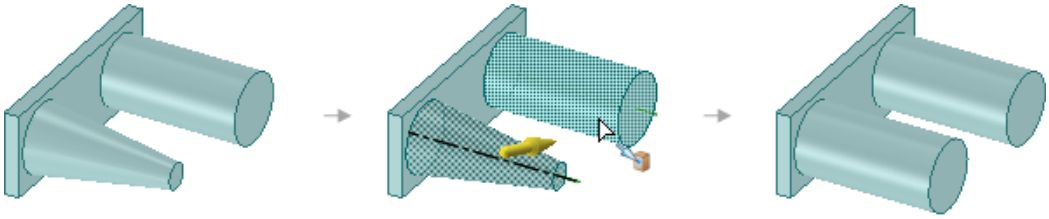
Pulling a face offsets it, and its edges are influenced by neighboring faces.



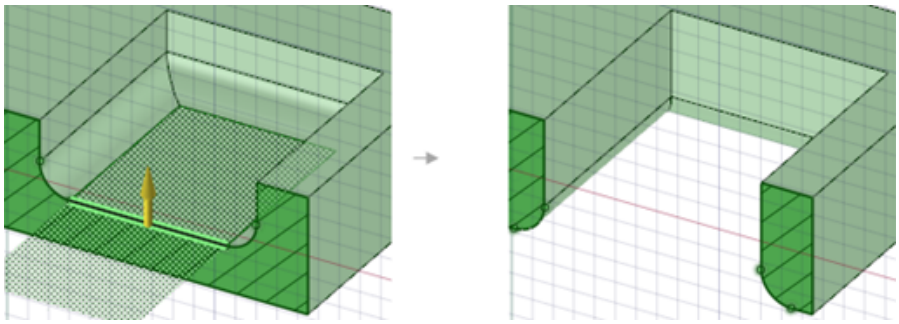
Pulling a face with its edges selected extrudes the face without influence from neighboring faces.



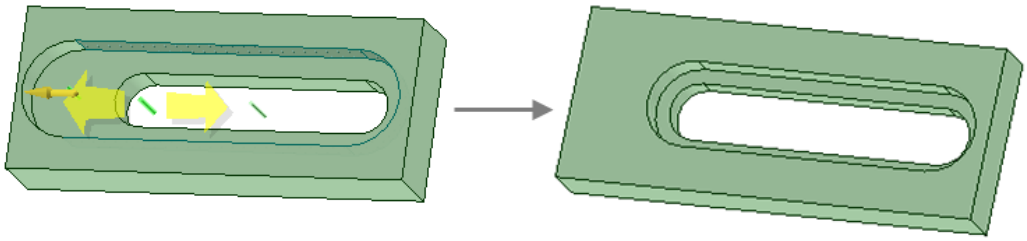
Pulling a conical face Up To a parallel cylindrical face replaces the cone with the cylinder if the axes are close together. Otherwise, the conical face is replaced with a cylindrical face that is coaxial to the cone and has the same radius as the cylinder.



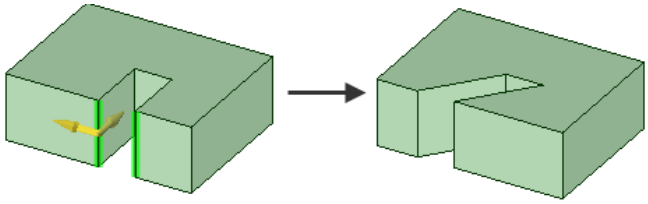
Pulling a pocket with rounded edges down through the bottom of a solid transfers the rounds to the resulting hole.



Pulling a surface up to a reference edge.



Pivot two separate edges together when pulling in one direction.



5.4.2.5. Offsetting or Extruding Faces

1. Select the **Pull** tool in the **Edit** group on the **Design** tab. The **Select** tool guide on the right side of the Design window should be active.
2. Select the face or surface you want to offset or extrude.

   You can select multiple surfaces and faces of solids to offset or extrude them simultaneously.
3. (Optional) Add edges to your selection.

   Select the edges of the face that you want to extrude when you pull. (Any unselected edges are defined by the neighboring geometry during the pull, creating an offset instead of an extrusion.)
4. (Optional) Select the **Direction** tool guide , then click the straight line, axis, or edge if you want to offset or extrude in a different direction.

You can also Alt+click a straight line, axis, plane, origin axis, or edge. Sometimes Discovery guesses your intent incorrectly when you Alt+click the direction line. If this happens, just use the Direction tool guide to correct it.

The pull direction is shown in blue and the pull arrow is reoriented.

5. Click and drag the face in the direction of the Pull arrow.

Dragging a:	**Does this:**
Solid face	Offsets the face in its natural offset direction
Solid face and all its edges	Creates an extrusion
Surface face	Thickens or thins the face
Corner edge of a solid	Creates a round, chamfer, or extrusion, depending on the selected option
Round or chamfer	Offsets the round or chamfer
Variable round	Uniformly offsets the variable round
Cylinder, torus, or cone	Offsets the cylinder, torus, or cone

To dimension the offset, type the distance you want to pull and press **Enter**.

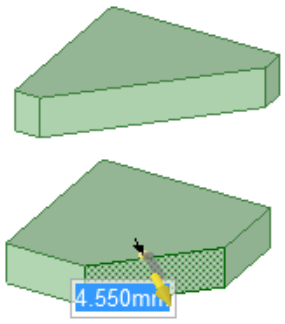
Press and hold **Ctrl** to offset a copy of the face as a surface.

Neighboring faces automatically extend to bound the offset face.

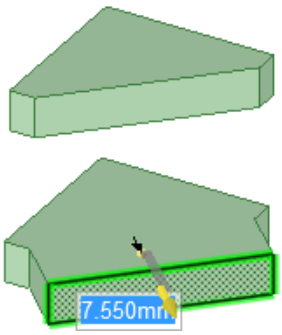
You can also use the Up To tool guide to pull up to any edge, plane, surface, or face in your design. If you pull a round, chamfer, or cylinder with the Up To tool guide, and pull up to similar geometry, the radius or offset is changed to the dimension of the Up To face you select.

The common pull options are described in Pull Options. Once you select the edge or face to pull, select these options from the Options panel, or right-click and select them from the mini-toolbar:

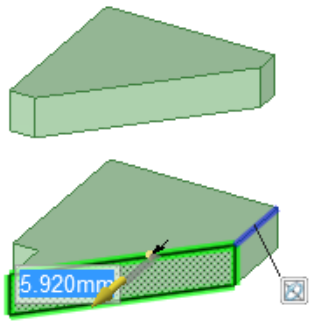
Examples



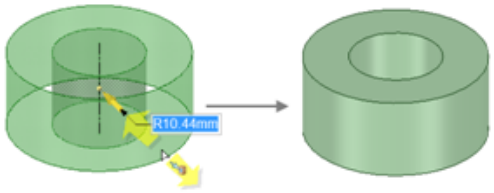
Selecting a face to offset



Selecting a face and edges to extrude



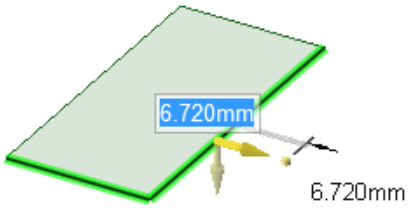
Selecting a face and edges, and setting a direction to extrude in a direction



Pulling an inner cylindrical surface up to an outer surface to create a solid ring

5.4.2.6. Extending or Extruding Surface Edges

You can extend or extrude the edges of any surface with the Pull tool. When you extend an edge, the pull extends the neighboring faces without creating a new edge. Extruding an edge creates edges.



1. Click **Pull** in the Edit group on the Design tab. The Select tool guide on the right side of the Design window should be active.
2. Select the outside edge of a surface.

   Ctrl+click to select multiple edges.
3. Click the Pull arrow oriented along the surface to extend the edge. Press **Tab** or click the other Pull arrow to extrude the edge in another direction.
4. (Optional) Ctrl+click the vertex of one or both neighboring edges to ignore their influence.
5. Drag in the direction of the Pull arrow to extend the edge or create a new surface perpendicular to the old one.

   If the correct pull arrow is not highlighted, press **Tab** or click the Pull arrow you want to use.

   The natural direction of a surface edge is in the plane of the surface.

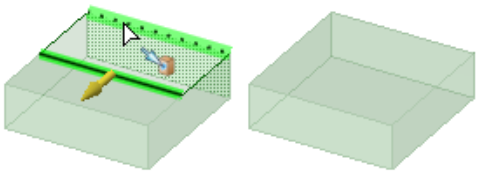
   You can use the Up To tool guide to pull a linear edge up to a face, surface, edge, or point. If the face or surface does not intersect the edge you are pulling, the edge will be pulled parallel to the selected object.

   To dimension the extension, type the distance you want to extend the surface while you are pulling and press **Enter**.

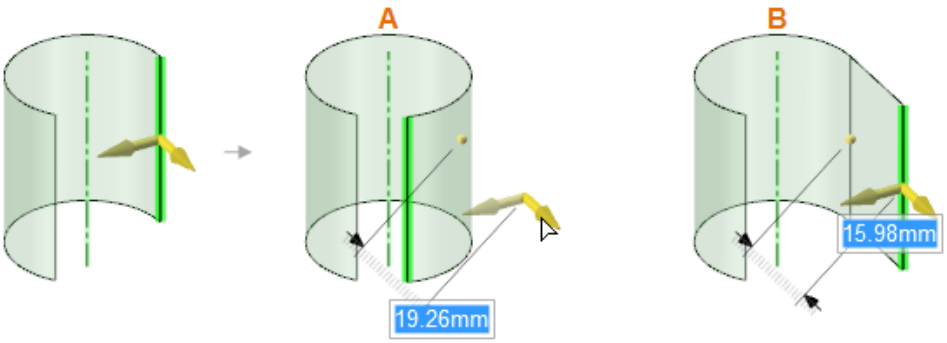
The common pull options are described in Pull Options. Once you select the edge or face to pull, select these options from the Options panel, or right-click and select them from the mini-toolbar.

5.4.2.6.1. Extending or Extruding Surface Edges Examples

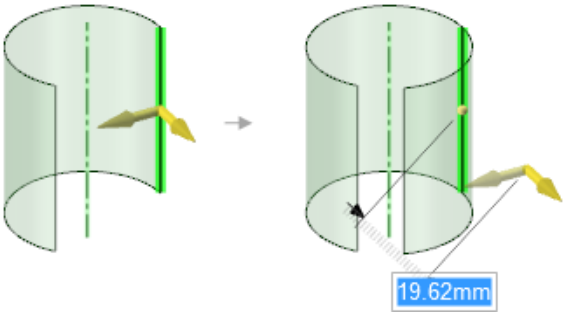
Pulling the edge of a surface up to another edge forms a solid if it closes the body.



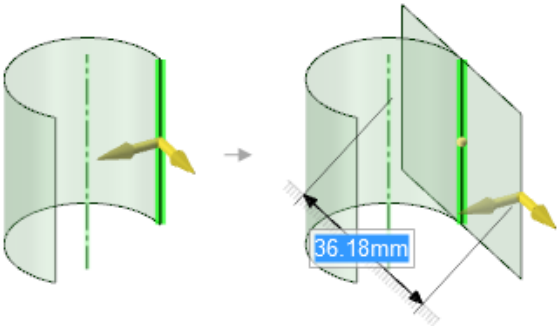
Pulling the edge of a cylindrical surface extends the surface along a cylindrical path (A) unless you hold Ctrl (B).



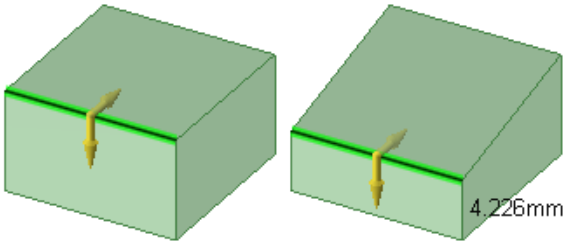
*Pulling the edge of a cylindrical surface with the **No Merge** option selected creates a new edge.*



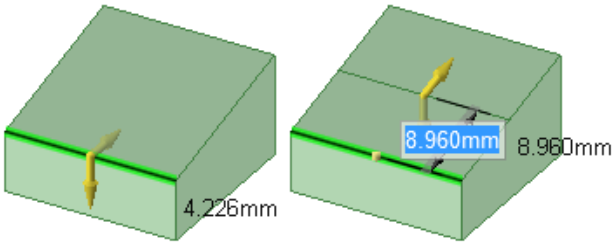
*Pulling the edge of a cylindrical surface with the **Both Sides** option and holding **Ctrl** creates a new surface tangent to the cylinder and extends in both directions from the edge.*



*Pulling an edge with the **Pivot Edge** option.*



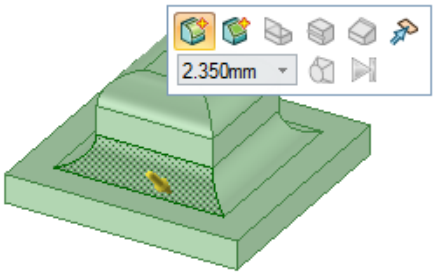
*Pulling an edge with the **Copy Edge** option*



5.4.2.7. Rounding Edges

Use the **Pull** tool's **Round** option to round the edges of any solid. You can also round an inside or outside corner of a surface. To create a rounded face between two faces, see Rounding between faces and surfaces.

You can change a constant- or variable-radius round with the **Pull** tool in section mode.



This section contains the following topics:

5.4.2.7.1. Rounding an Edge

1. Click the **Pull** tool in the Edit group on the Design tab.
2. Select the edge or edges you want to round. Double-click to select a tangent chain.

   You can also select the vertex of an inside or an outside corner of a surface.
3. Select the **Round** option in the Options panel or from the mini-toolbar. This option is automatically selected for edges that can be rounded.
4. Click and drag the edge in the direction of the Pull arrow.

   To dimension the round, press the Spacebar, type the radius, and press **Enter**.

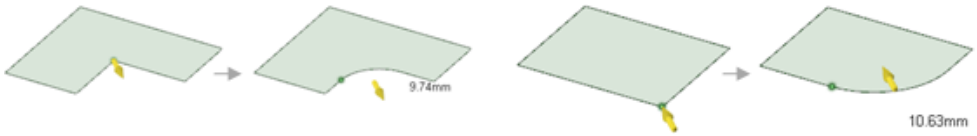
   When you pull an edge to create a round, the Pull tool determines whether it should be a constant radius or a variable radius round based on the geometry.

   Once you create a round, pulling an adjacent face also pulls the round.

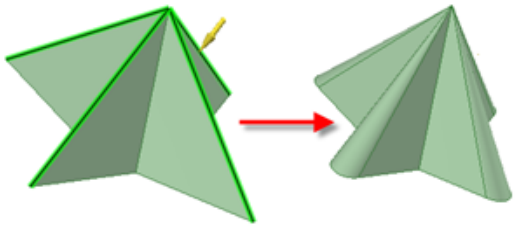
   When you round more than one edge at the same time, and one or more of the rounds cannot be created, only the successful rounds are created.

> **Note:** The face hidden by the round is remembered, so that if you fill the round it can be displayed. If you move a round, the face hidden by it is also moved.

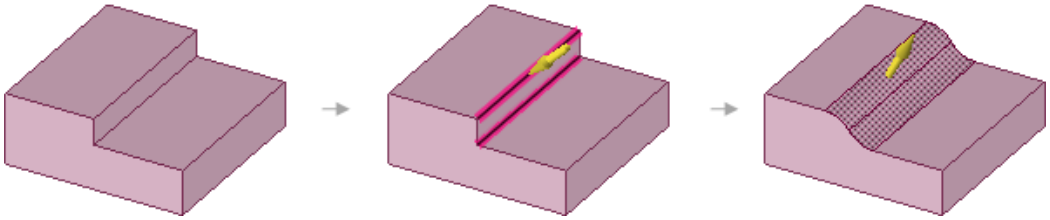
Rounding the inside and outside corner of two surfaces.



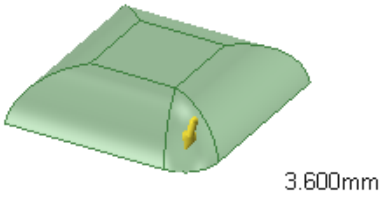
Pulling edges on geometry that automatically determines rounds will be variable radius.



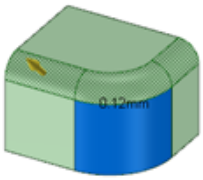
When creating surface-to-surface rounds, the expansion direction depends on the order in which you select the edges. In the example above, the rounds expand toward the left because the upper edge was selected first.



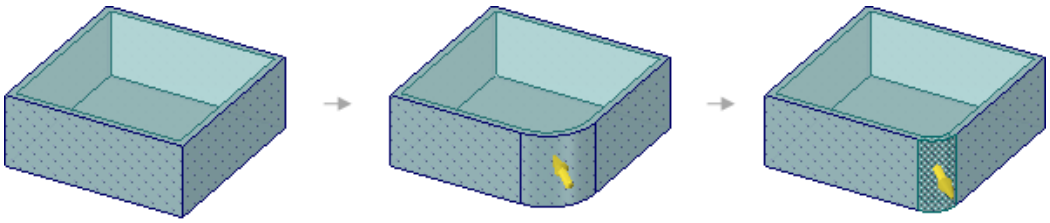
Rounding the edge created by two rounds with one edge not tangent.



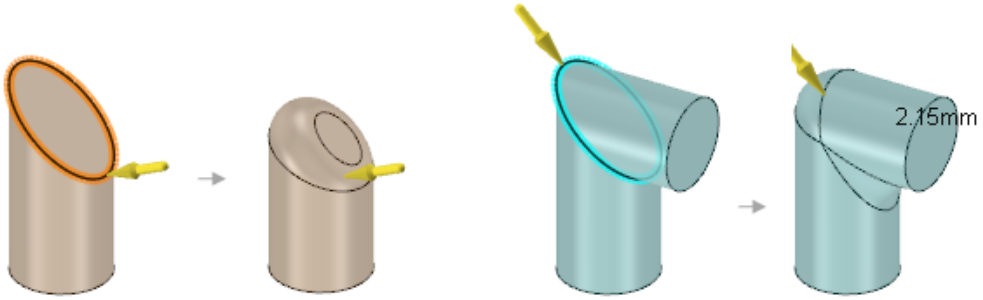
Pulling rounds in a tangent chain. Pulling the selected tangent chain does not change the round shown in blue.



Adding and changing a rounded edge on a shelled or offset part changes the inside faces.



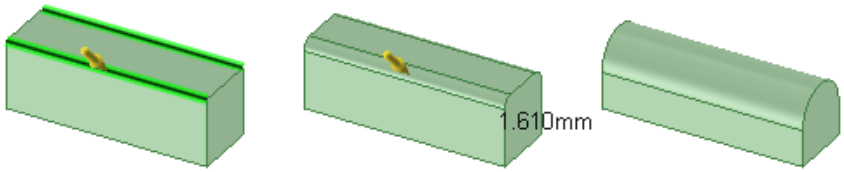
Rounding elliptical edges.



5.4.2.7.2. Creating a Full Round by Selecting Edges

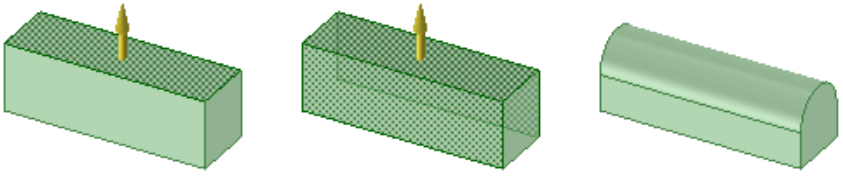
1. Click the **Pull** tool.
2. Select the two edges on opposite sides of the face you want to become a full round, or select the circular edge on the end of a cylinder.

   You can select more than one pair of edges, and create full rounds between the pairs.
3. Pull the edges down so they are just rounded.
4. Slowly pull the mouse in the opposite direction, until the faces are no longer round, and continue pulling until the full round appears.

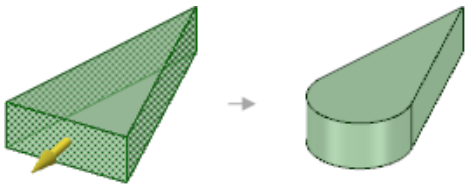


5.4.2.7.3. Creating a Full Round by Selecting Faces

1. Click the **Pull** tool.
2. Select the face where you want the full round.
3. Hold **Ctrl** and select the two side faces that will be tangent to the full round.
4. Right-click and select **Full Round**.



Creating a full round by selecting three faces



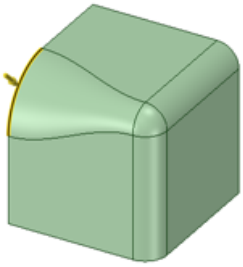
5.4.2.7.4. Converting to and Editing a Variable Radius Round

To convert a constant-radius round to a variable-radius round, right-click and select the **Edit as Variable Radius Round** option.

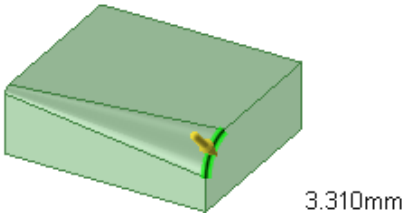
- Click the Pull arrow on an end of the round face and drag toward or away from the face to adjust the radius at the edge of the round.
- Click the Pull arrow pointing toward the center of the face and drag it along the face (or enter a length or percent) to set another point where you can adjust the round's radius.
- Hold **Ctrl** and drag to copy a control point.
- Press **Tab** to switch among and edit the dimensions by typing a number.
- Two rounds that intersect at a shared edge can be made variable together by selecting the shared sharp edge for modification. Both rounds become variable at that point. They can be made variable independently by selecting the sharp end edge that is not shared by the rounds.

  In the **Pull** tool, when you select an edge of a constant radius round, the **Smooth** option becomes enabled if the geometry allows it. When checked, the transition between radius at each end of the round will vary smoothly instead of linearly. Once set, the Smooth option remains set for subsequent rounds until you change it.

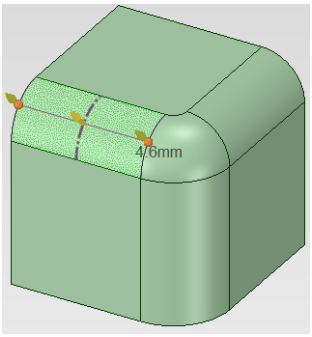
The image below shows a smooth variable radius round.



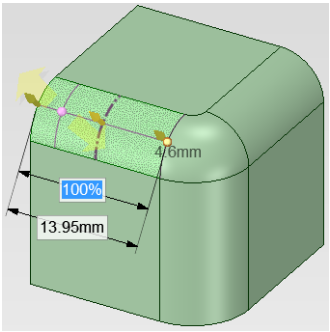
Editing a variable radius round. The variable radius round handles have a drag (ball) handle. The center Radius Handle cannot be dragged, all radii values are adjusted at once.



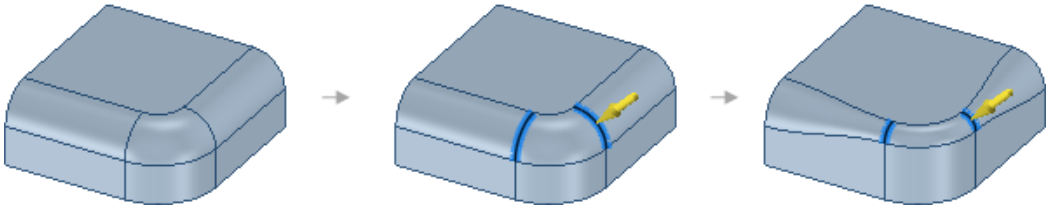
Click the ball and drag to add and place a new radius handle.



Setting another point for adjusting a variable round's radius



Selecting multiple edges to create a variable round across those edges



5.4.2.7.5. Removing a Rounded Edge with the Pull Tool

This method completely removes the rounded edge.

1. Click the **Pull** tool.
2. Select the rounded edge.
3. Select the **Round** option in the Options panel or from the mini-toolbar.

   The round is removed.

5.4.2.7.6. Removing a Rounded Edge with the Fill Tool

This method removes the rounded edge and creates a round group in the Groups panel. You can right-click the group and select **Reattach Round** to restore the rounded edge.

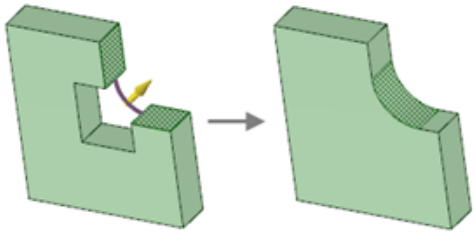
1. Click the **Fill** tool in the Edit group on the Design tab.
2. Select the rounded edge.
3. Click **Complete**.

   The round is removed and a round group is created.

5.4.2.8. Rounding Between Faces and Surfaces

You can create a rounded edge across the gap between two faces or surfaces that would meet at a corner. To create a rounded face on an edge, see Rounding edges.

When you round between faces of two solids, the solids must extend into the area where the rounded face is created.



This section contains the following topics:

5.4.2.8.1. Rounding Between a Face or Surface

1. Click **Pull** in the Edit group on the Design tab.
2. Select the faces you want to create a round between.
3. Select the **Round** option in the Options panel.

   The pull arrow changes to indicate that a round will be created between the faces and a curve is displayed to show you the direction of the round. Yellow cursor arrows show you the directions you can pull to create the round.
4. Press **Tab** to cycle through all of the possible rounds that can be created between two faces or surfaces.

   If the two faces you are rounding between are rounds, then Tab lets you choose either rounding between or changing the radius of both existing rounds.
5. Click and drag in the direction of the Pull arrow.

   To dimension the round, press the Spacebar, type the radius, and press **Enter**.

   When you pull an edge to create a round, the **Pull** tool determines whether it should be a constant radius or a variable radius round based on the geometry.

   Once you create a round, pulling an adjacent face also pulls the round.

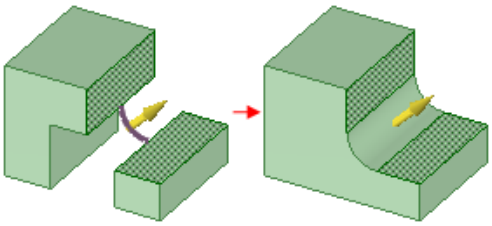
   When you round more than one edge at the same time, and one or more of the rounds cannot be created, only the successful rounds are created.

   When pulling Face-to-Face round,s you can match their radius to an existing round using the **Up To** tool guide.

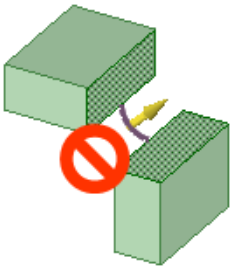
> **Note:** Unlike rounds made on edges, the faces hidden by the round are NOT remembered. If you fill the round it will be replaced by an edge.

5.4.2.8.2. Rounding Between Faces and Surfaces Examples

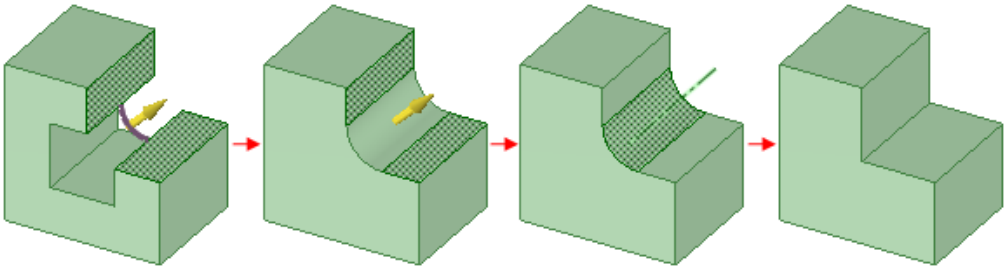
Rounding between faces on two parts. Material exists in the area where the rounded face is created, so this operation will succeed.



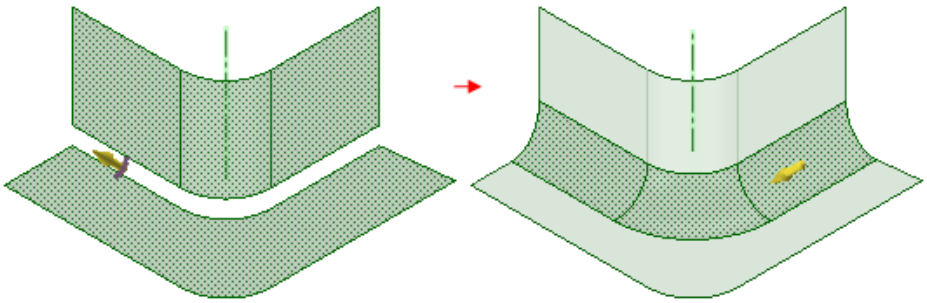
A rounded face cannot be created between these faces because material doesn't exist in the area between the parts where the round will be created.



This sequence shows creating a round between faces, then using the Fill tool to remove the round. The Fill replaces the round with an edge and not with the original geometry.



Rounding between two surfaces creates a rounded surface.

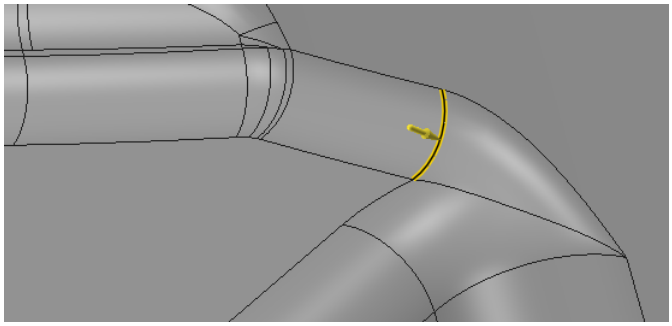


5.4.2.9. Modifying Rounds

Some modifications fail because the entire round chain (along with all chains required to be removed by the modification) are all included in the modification. In some cases, parts of the chain (or others, even far away ones) are restricted by surrounding geometry and the modification fails. To make modifications more robust, they are localized to the selected piece of the round.

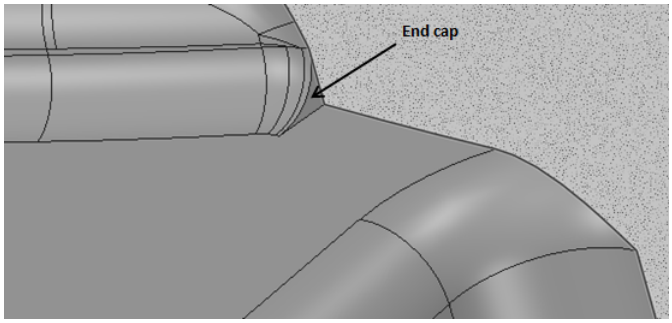
For this to work, the ends of the localized section need to be "auto-capped". In other words, they need to be able to be closed off by a plane triangular face, essentially normal to the last-visited construction edge. The following example illustrates this.

The radius is going to be increased at the selected edge.

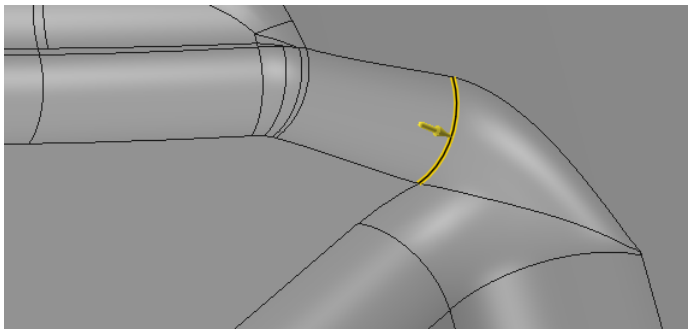


Localizing means that the round pieces on either side of the edge are removed first. For this to work, the ends of the removed pieces need to be "capped" with a triangular face or be the start or end point of the round chain. These end conditions are necessary to rebuild the new variable round between the ends.

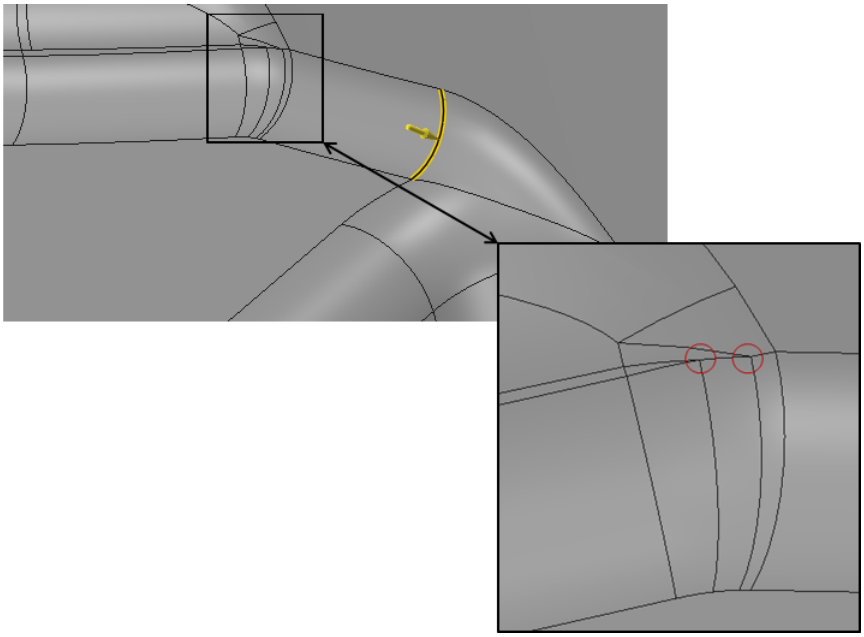
An end cap is shown in the example below.



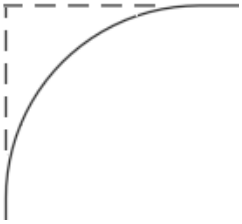
The Localized round can then be reconstructed with the modified radius as shown below.



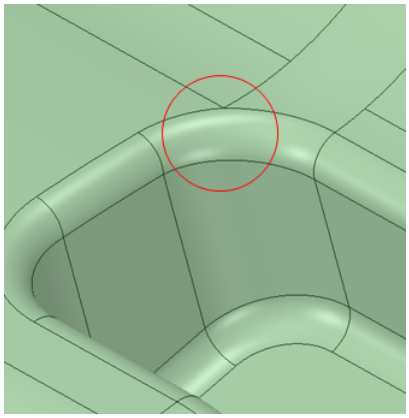
The image below shows an example where capping the end is difficult. The red circles highlight points where there is ambiguity for capping. It's difficult to construct a capping face normal to the construction edge. With Localization, the modification will fail if either of these edges is an end of a modified piece of the round. Without Localization, the modification would include these areas and the modification would fail.



In order to remove a round for modification, the side faces need to be able to Extend up and form a sharp edge as shown below.



In some cases, extra surface patches can cause ambiguous situations where the sharp edge cannot be determined and the round cannot be removed for modification. The image below is an example of a round that cannot be removed because of face extension ambiguity.



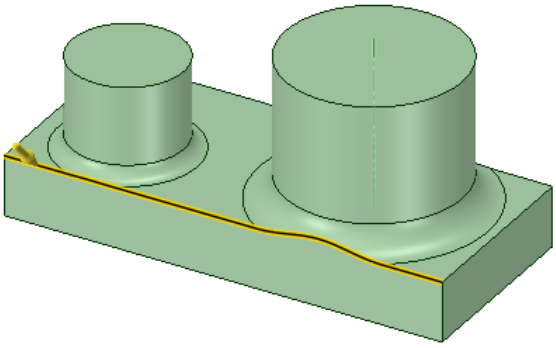
This section contains the following topics:

5.4.2.9.1. Round Order

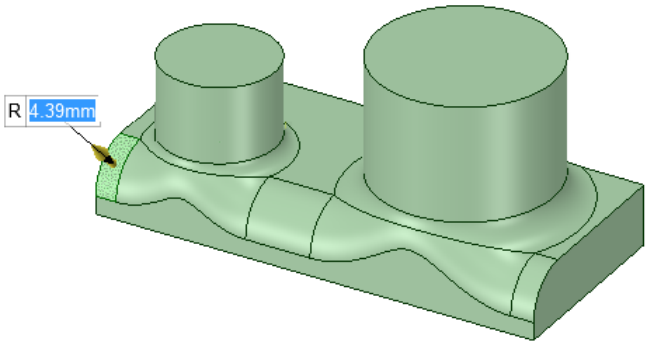
The order in which rounds are created influences the appearance and quality of the resultant geometry. The **Reverse Round Order** button in the Round mini-toolbar enables you to reverse the order of selected rounds that interact with other rounds.

You can change the order of round creation to clean up split rounds that produce extra edges as shown in the example below.

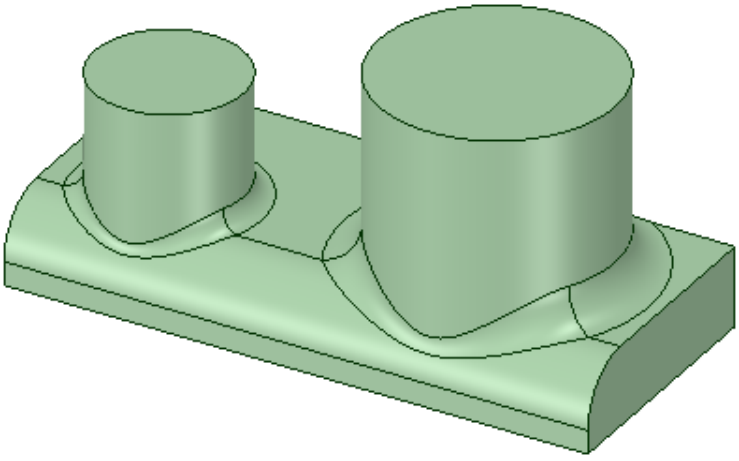
A round is applied to an edge which is broken by existing rounds.



Because the edge is broken, the new round is segmented.



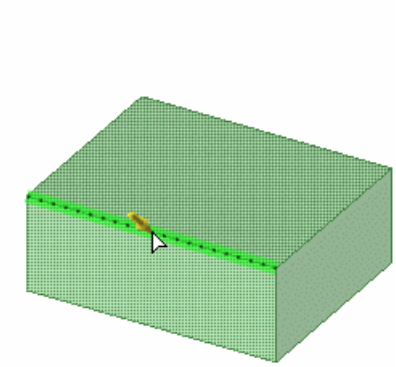
Reversing the round order produces smoother rounds.



5.4.2.10. Chamfering Edges

You can chamfer the edges of any solid by selecting the Pull tool's Chamfer option. Sketched chamfers that you make using the Create Rounded Corner tool with the Chamfer option are recognized as chamfers when they are extruded into a solid.

After you create the chamfer, you can adjust the setback distance of both sides and you can change its offset. Drag the arrows above or below the chamfer to change the distance, or drag the arrow that is perpendicular to the chamfer to change its offset. You can also pivot the edges and the chamfer's distance properties will be updated with the new values.



Note: When you make a hole in a chamfered face, the face is no longer a chamfer. You can still pull the face, or the hole, but you cannot change the chamfer to a round or dimension the chamfer.

This section contains the following topics:

5.4.2.10.1. Chamfering an Edge

1. Click **Pull** in the Edit group on the Design tab.
2. Select the edge or edges you want to chamfer. Double-click to select a tangent chain.
3. Select the **Chamfer** option in the Options panel or from the mini-toolbar.
4. Click and drag the edge in the direction of the Pull arrows.

   Pulling the arrow that is normal to the chamfer face changes the offset distance for the whole chamfer. Pulling either of the side arrows changes the setback distance of that side. You can press Tab to move among the arrows.

   To dimension the chamfer, right-click and enter a setback distance in the mini-toolbar, or press the space bar, type the setback distance, and press **Enter**.

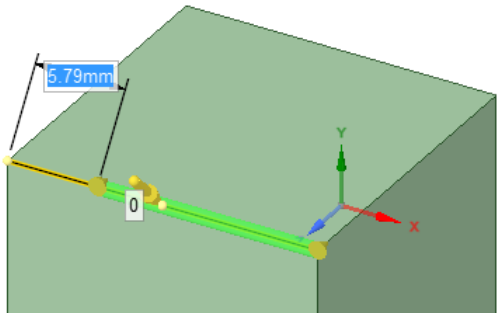
   To create a chamfer with the same setback distance and size as an existing chamfer, use the **Up to** tool guide and select an existing chamfer face. You can also select an edge with **Up To** to set the setback distance.

5.4.2.10.2. Chamfering Stop Faces

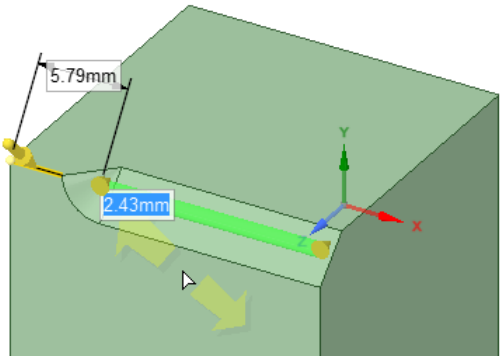
Stop Chamfers have conical stop faces that mimic the effect of an angled cutting tool rotating and then stopping somewhere in the middle of an edge.

Select a chamfered edge.

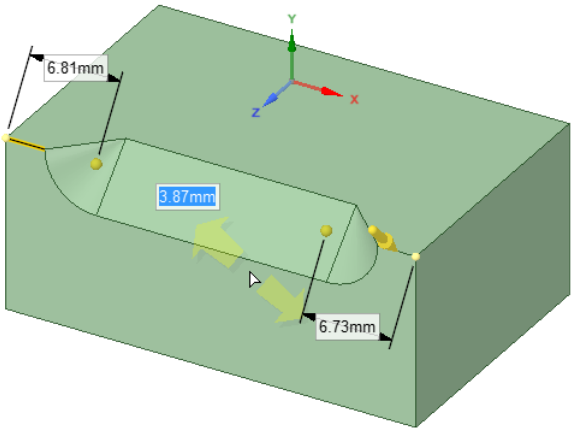
- *When you click the **Chamfer** option in the **Pull** tool, the edge ends switch to yellow cones that can be dragged. This is similar to **Pull Edge** in Sheet Metal and sets the point where the simulated cutter will stop.*



- *Note the conical stop face created at the chamfer's end.*

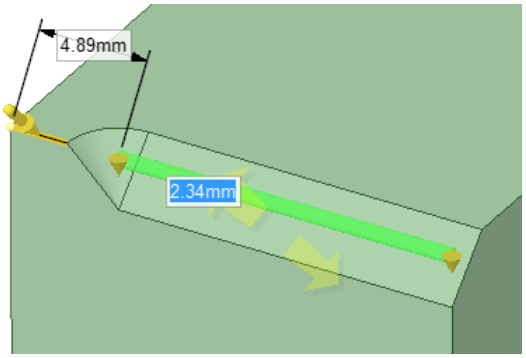


- *You can move both ends to create stop faces at both ends of the chamfer.*

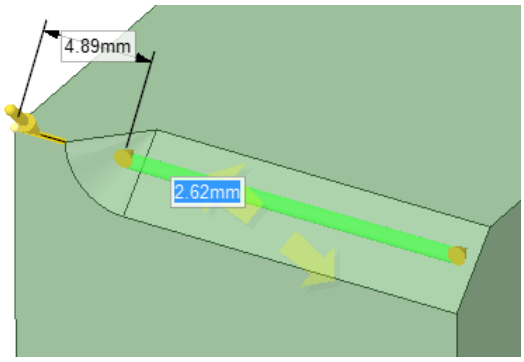


Reverse Chamfer Stop has been added to chamfer options. This flips the ‘direction' of the conical Stop Face.

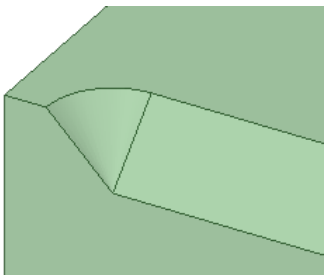
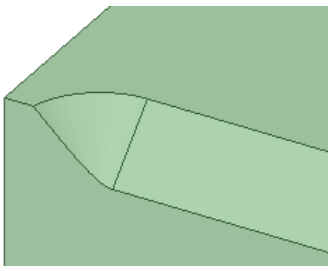
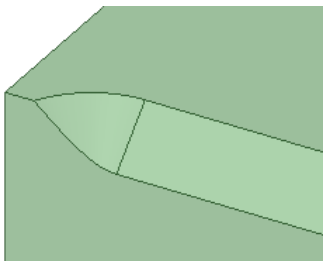
- *The handle used to set the location of the **Stop Face** is a cone that indicates the orientation of the cutting tool that would create the chamfer.*



- *Here it is 'reversed' or flipped 90 degrees.*

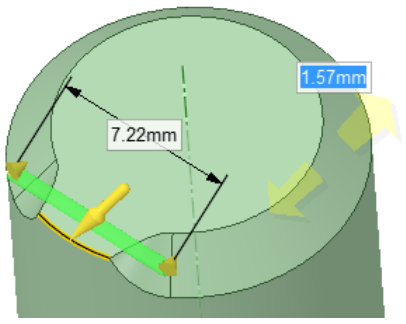


- An Apex offset input box has been added to chamfer options to offset the cone apex (the lowest point of the cone) past the furthest extent of the intersection of the virtual cutter with the material.

Apex offset = 0	*Apex offset = 1*	*Apex offset = 2*
		

- The linear dimension in the image refers to the arc length of the un-chamfered part of the original edge.

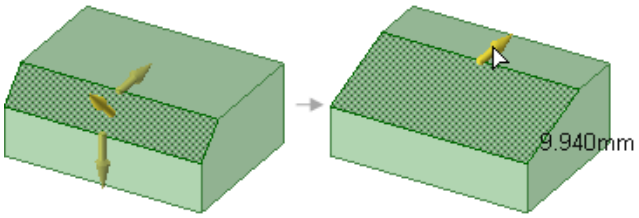
  Note in the example below that 7.22mm is the length of the arc between points.



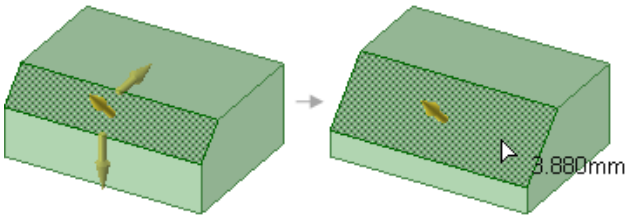
5.4.2.10.3. Modifying a Chamfered Edge

Select a chamfered edge or a chain of chamfered edges.

- To change the setback distance of one side of the chamfer, pull the arrow above or below the chamfer, as shown below. Press the space bar while pulling to type a distance value. You can also change both setback distances in the Properties panel and in the mini-toolbar. The **Up To** tool guide can be used to pull the setback up to an object.



- To change the offset distance of the chamfer, pull the arrow that is perpendicular to the chamfer, as shown below. Press the spacebar while pulling to type a distance value.



- To convert the chamfer to a normal face, select **False** in the Chamfer field of the Properties panel. You can select **True** to convert a face to a chamfer, however, faces with holes cannot be modified as chamfers. Changing this property does not change the geometry. See the instructions below to remove a chamfer from an edge.

> **Note:** Hold the Shift key while pulling a chamfer setback or offset to snap to existing edges.

5.4.2.10.4. Converting Between Rounded Edges and Chamfered Edges

1. Click **Pull** in the Edit group on the Design tab.
2. Select the edge or edges you want to convert to chamfers or rounded edges.
   Double-click to select a tangent chain.
   You can't convert a chamfered edge if you have changed the setback distance(s).
3. Select the **Chamfer** option or the **Round** option in the Options panel or from the mini-toolbar.
   The rounded edge is converted to a chamfered edge.

> **Note:** You cannot convert a chamfer to a round if you have changed its setback distance(s).

5.4.2.10.5. Making a Chamfer with the Same Setback and Offset as Another Chamfer

1. Click **Pull** in the Edit group on the Design tab.

2. Select an edge or an existing chamfer that you want to change.
3. Click the **Up To** tool guide.
4. Select the chamfer whose values you want to copy.
   The first chamfer will be changed so its offset and setback distances are the same as the second chamfer.

5.4.2.10.6. Removing a Chamfer with the Pull Tool

This method completely removes the chamfered edge.

1. Click **Pull** in the Edit group on the Design tab.
2. Select the chamfered edge.
3. Select the **Chamfer** option in the Options panel or from the mini-toolbar.
   The chamfer is removed.

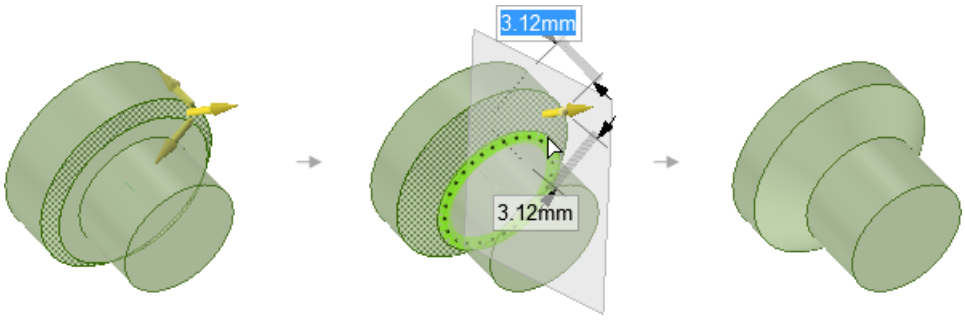
5.4.2.10.7. Removing a Chamfer with the Fill Tool

This method removes the chamfered edge and creates a chamfer group in the Groups panel. You can right-click the group and select **Reattach Chamfer** to restore the chamfered edge.

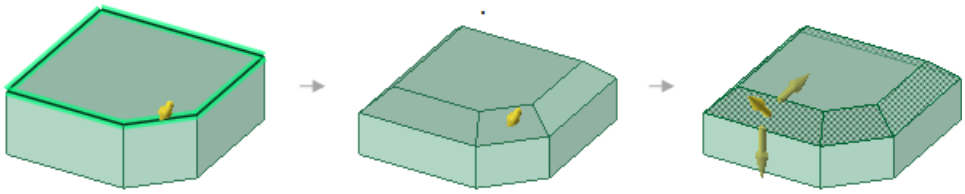
1. Click **Fill** in the Edit group on the Design tab.
2. Select the chamfered edge.
3. Click the **Complete** tool guide.
   The chamfer is removed and a chamfer group is created.

5.4.2.10.8. Chamfering Edges Examples

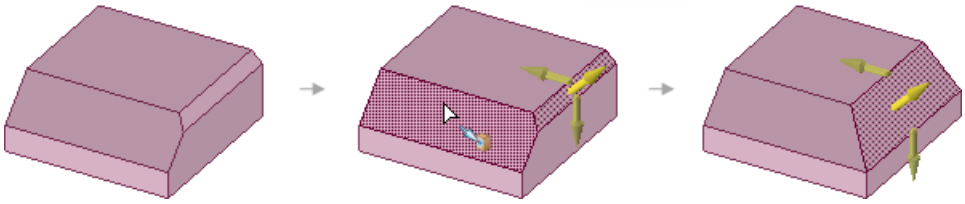
Pulling to enlarge an existing chamfer while holding Shift to snap to an edge. You can only snap to an edge in this case when you pull an existing chamfer.



*Using the **Up to** tool guide to change the offset and setback distances of a chamfer to match an existing chamfer.*

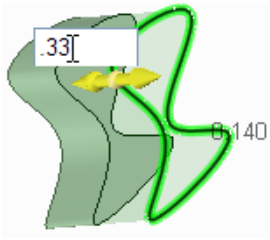


Creating a chamfer on multiple edges at the same time.



5.4.2.11. Extruding Edges

You can extrude the edge of any solid by selecting the **Pull** tool's **Extrude Edge** option. You can also extend and extrude surface edges.



This section contains the following topics:

5.4.2.11.1. Extruding an Edge

1. Click **Pull** in the Edit group on the Design tab. The Select tool guide on the right side of the Design window should be active.
2. Select the edge or edges you want to extrude.

   Ctrl+click to select multiple edges. Double-click to select a tangent chain.
3. Select the Extrude Edge option in the Options window, or from the mini-toolbar.

   The Pull arrow changes to show the two directions in which you can extrude the edge. One arrow is highlighted to show the primary direction.
4. If the arrow pointing the direction you want to pull is not highlighted, click the arrow or press **Tab** to change the direction.

Click and drag the edge in the direction of the Pull arrow.

To dimension the extrusion, right-click and enter a distance in the mini-toolbar, or type the distance *while you are pulling* and press **Enter**.

Press Ctrl to copy the edges.

Press Ctrl+C, then Ctrl+V to quickly copy and paste a selected set of edges *in place*.

You can click the Up To tool guide and click a face, edge, or point to pull up to. If the face does not intersect the edge you are pulling, the edge will be pulled parallel to the face. If you pull the edge of a surface up to another object, the result is automatically solidified if it creates a closed volume.

5.4.2.11.2. Extruding a Curve

1. Click **Pull** in the **Edit** group on the **Design** tab. The **Select** tool guide on the right side of the Design window should be active.
2. Select the curve or curves you want to extrude.

Ctrl+click to select multiple curves. Double-click to select a tangent chain.

3. Select the **Extrude Edge** option in the Options window, or from the mini-toolbar.

   The Pull arrow changes to show the two directions in which you can extrude the curve. One arrow is highlighted to show the primary direction.

4. If the arrow pointing the direction you want to pull is not highlighted, click the arrow or press **Tab** to change the direction.

Click and drag the curve in the direction of the Pull arrow.

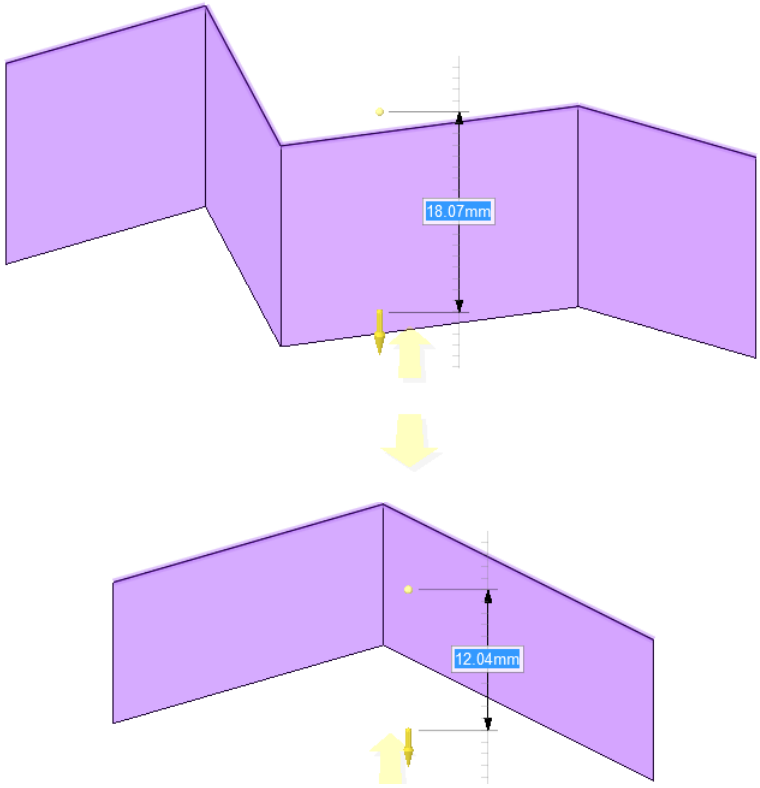
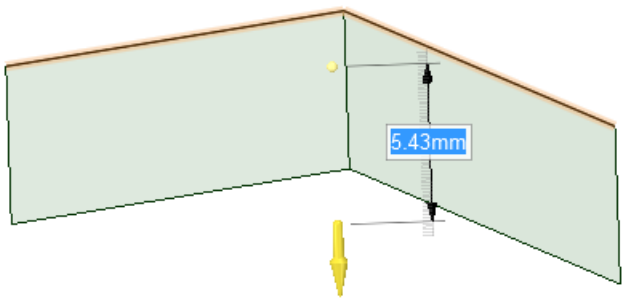
To dimension the extrusion, right-click and enter a distance in the mini-toolbar, or type the distance *while you are pulling* and press **Enter**.

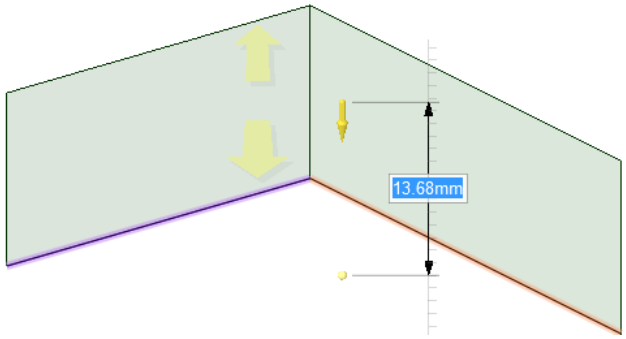
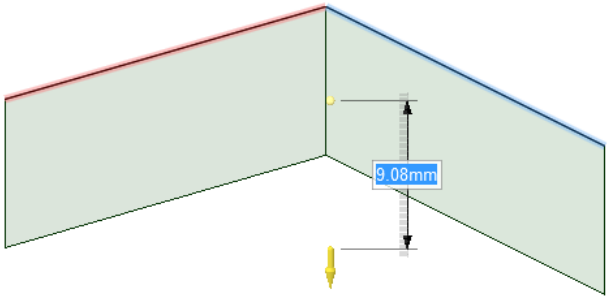
Press Ctrl to copy the curves.

Press Ctrl+C, then Ctrl+V to quickly copy and paste a selected set of curves *in place*.

You can click the **Up To** tool guide and click a face, edge, or point to pull up to. If the face does not intersect the curve you are pulling, the curve will be pulled parallel to the face. If you pull the curve of a surface up to another object, the result is automatically solidified if it creates a closed volume.

When extruding sketch curves into surfaces, the following rules apply to the resulting surface color.

If all curves being pulled are the same color AND the curve colors override their layer color, the surface is the same color as the curves.	
If all curves being pulled are the same color AND the curves take their color from a layer that is not the Active layer, the surface is the color of the Active layer and not the color of the curves.	

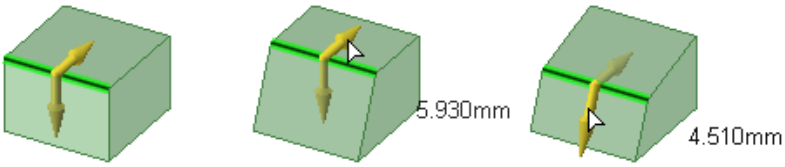
If all curves are not the same color, the surface uses the Active layer color.	
If all curves are not the same color AND take their colors from different layers, the surface uses the Active layer color.	

The common pull options are described in Pull Options. Once you select the edge or face to pull, select these options from the Options panel, or right-click and select them from the mini-toolbar:

5.4.2.12. Pivoting Edges

You can pivot the edge of any solid with the **Pull** tool's Pivot Edge option. You can also pivot two separate edges together when pulling in one direction.

To turn a cylinder into a cone, select the **Pull** tool and the Pivot Edge option. Then click the edge of the cylinder and pull it until it forms a point. To turn a truncated cone into a cylinder, click the small edge and pull it out.



This section contains the following topics:

5.4.2.12.1. Pivoting an Edge

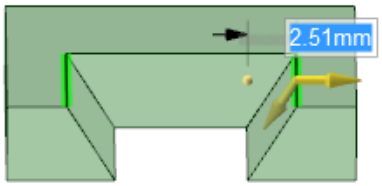
1. Select the **Pull** tool in the Edit group on the Design tab.
   The Select tool guide is enabled by default.
2. Select the edge or edges you want to pivot. Double-click to select a tangent chain or edge loop.
3. Select the **Pivot Edge** option in the Options panel, or from the mini-toolbar.
   The Pull arrow changes to show the two directions in which you can move the edge to pivot the connected faces. One arrow is highlighted to show the primary direction.
4. If the arrow pointing in the direction you want to pull is not highlighted, click the arrow or press **Tab**.
5. Drag the edge in the direction of the **Pull** arrow.

During the pull, the distance you have moved the edge is displayed. You can dimension the distance in Section and 3D mode, and dimension the angle in Section mode.

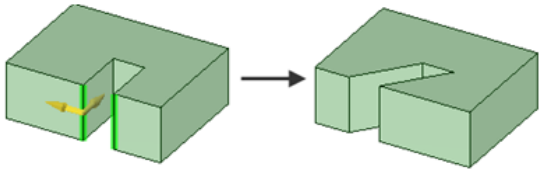
You can hold **Shift** to snap to existing geometry.

5.4.2.12.2. Pivoting Edges Examples

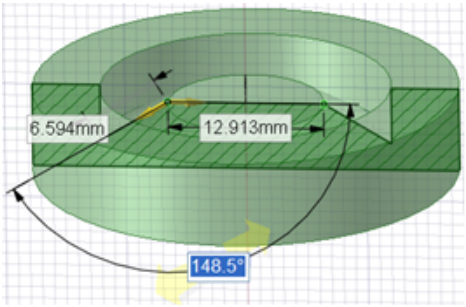
Simultaneously pivoting multiple edges.



Pivoting two separate edges together when pulling in one direction.



Dimensioning the pivot angle.



5.4.2.13. Revolving Faces

You can revolve any face or surface with the **Pull** tool. The face or surface can lie on both sides of the revolve axis.

1. Select the **Pull** tool in the Edit group on the Design tab. The Select tool guide on the right side of the Design window should be active.
2. Select the surfaces, faces, or solids you want to revolve.
3. Alt+click the straight line, axis, or edge to set the revolve axis. You can also select the **Revolve** tool guide, then click to set the revolve axis. The revolve axis is shown in blue.
4. Click and drag in the direction of the Pull arrow to revolve the selected object, select the Up To tool guide and click an edge, face, or plane, or select **Full Pull** from the Options panel or mini-toolbar.

   Neighboring faces automatically extend to define the boundaries of the revolved faces of solids. Revolving from a flat surface face has no neighboring faces, so it makes new face boundaries.

   To dimension the rotation, type the rotation angle and press **Enter**. The positive direction is shown by the Revolve handle.

Note: Alt+click an axis, then mouse over a face and pull to revolve the face around the axis.

The common pull options are described in Pull Options. Once you select the edge or face to pull, select these options from the Options panel, or right-click and select them from the mini-toolbar.

This section contains the following topics:

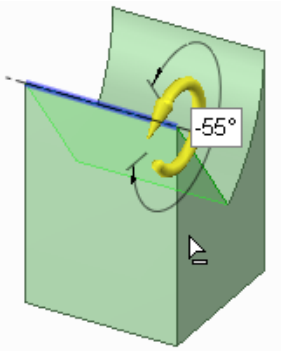
5.4.2.13.1. Revolve Tool Options

The following options are available once you select the edge or face to pull, and the **Revolve** tool guide:

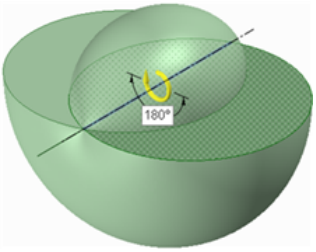
Revolve helix	Use this option to revolve the selected face or edge around the selected axis to create a helix. Specify the length and the pitch, and set the handedness of the helix by selecting or deselecting the **Right-Handed Helix** option. See Revolving Helices on page 228 for details.
Rotational rib	Use this option to revolve the selected face or edge around the selected axis to create a rotational rib.

5.4.2.13.2. Revolving Faces Examples

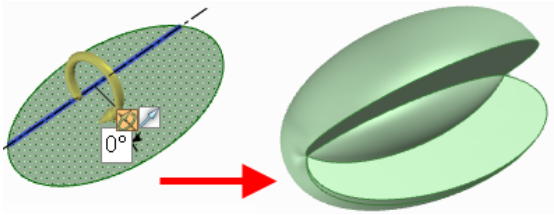
Revolve using cut.



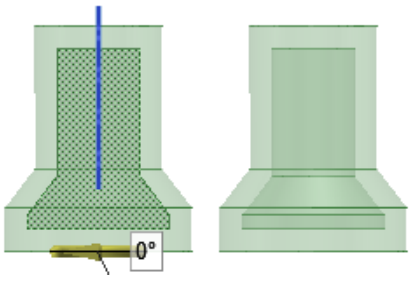
180 degree revolve.



Full revolve of an ellipse around an asymmetrical axis.

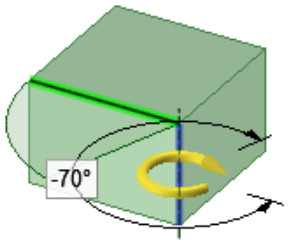


Revolving a surfacing using Cut, when the surface lies on both side of the axis and is enclosed within a solid.



5.4.2.14. Revolving Edges

You can revolve an edge to form a surface with the **Pull** tool. You can revolve the edge of a solid or surface.



This section contains the following topics:

5.4.2.14.1. Revolving an Edge of a Solid or Surface

1. Select the **Pull** tool in the Edit group on the Design tab. The Select tool guide on the right side of the Design window should be active.
2. Select the edges you want to revolve.
3. Alt+click a straight line, axis, or edge to set the rotation axis.

   You can also select the **Revolve** tool guide, then click the revolve axis. The revolve axis is shown in blue.
4. Click and drag in the direction of the Pull arrow to revolve the selected edge or select the Up To tool guide and click an edge or face to revolve up to.

   Ctrl+click to select edges of faces along with the face to force the pull tool to create new neighboring faces to contain these edges (essentially changing from a draft to a revolve). Unlike other CAD products, Discovery allows revolving planar and non-planar edges and faces about lines that do not lie in those planes. This allows skewed rotational solids and surfaces.

   You can use the **Up To** tool guide when you revolve edges.

   To dimension the rotation, type the rotation angle while the Revolve handle is displayed and press **Enter**.

5.4.2.14.2. Revolving an Edge of a Surface

1. Select the **Pull** tool in the Edit group on the Design tab. The Select tool guide on the right side of the Design window should be active.
2. Select the outside edge of a surface.
3. (Optional) Hold Ctrl and select an end point to maintain constant radius as the edge is revolved.

   If you don't select an end point, the edge will follow influencing sides of surfaces or solids, if they exist. A side influence can be another edge of the surface if it lies in the same plane, or a solid or surface that touches the end of the edge. See the examples below.
4. Alt+click a straight line, axis, or edge to set the rotation axis.

   You can also select the **Revolve** tool guide, then click the revolve axis. The revolve axis is shown in blue.
5. Click and drag in the direction of the Pull arrow to revolve the selected edge.

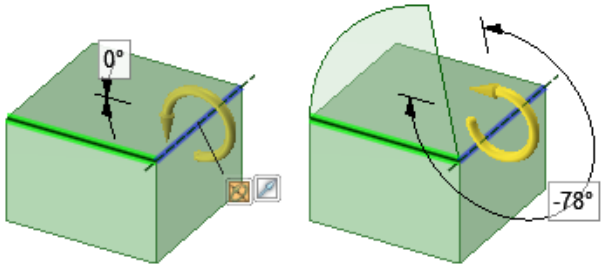
   To dimension the rotation, type the rotation angle while the Revolve handle is displayed and press **Enter**.

The common pull options are described in Pull Options. Once you select the edge or face to pull, select these options from the Options panel, or right-click and select them from the mini-toolbar.

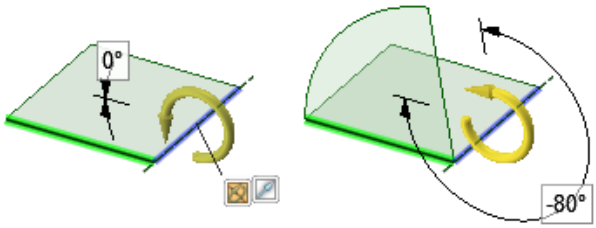
The **Revolve Options** are available once you select the edge or face to pull, and the **Revolve** tool guide.

5.4.2.14.3. Revolving Edges Examples

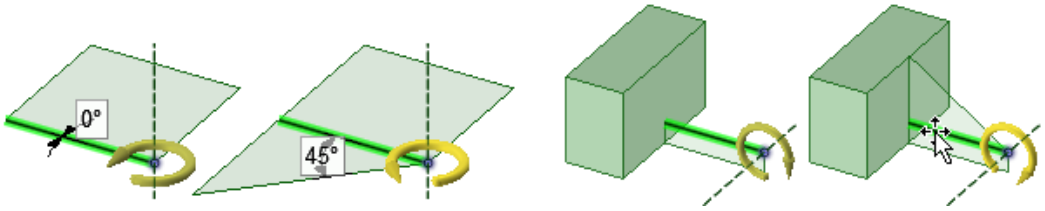
Revolving the edge of a solid to create a surface.



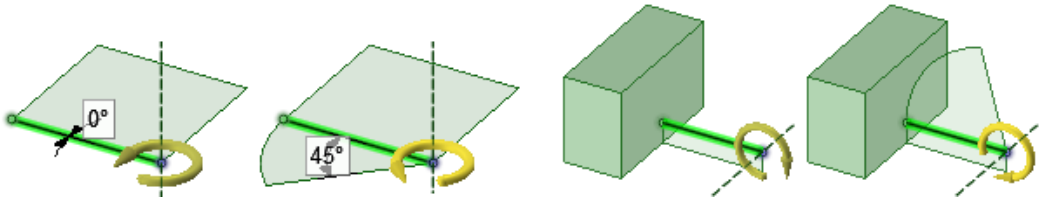
Revolving an edge of a surface that is not in the plane of the surface makes an edge with a constant radius because the edge has no side influence in this direction.



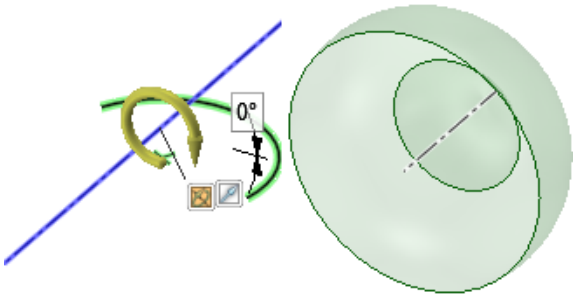
Revolving an edge of a surface with side influence makes an edge that is influenced by adjacent edges.



Revolving an edge of a surface with side influence with its end-point selected makes an edge with a constant radius.

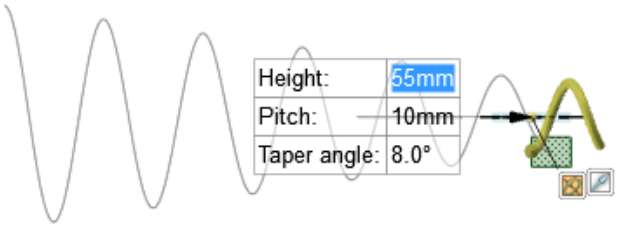


Revolving an edge that crosses the axis will create more than one surface or solid if the edge is not symmetrical across the axis.



5.4.2.15. Revolving Helices

You can revolve a helix with the **Pull** tool. Path and geometry are previewed as you pull along the axis or enter dimensions, as shown in the image below.



This section contains the following topics:

5.4.2.15.1. Revolving a Helix

1. Select the **Pull** tool in the Edit group on the Design tab. The Select tool guide on the right side of the Design window should be active.
2. Select the face or edge you want to revolve.
3. Select the **Revolve** tool guide.
4. Click the axis you want to revolve around.
5. Select the **Revolve Helix** option in the **Revolve Options** group.
6. For points, select the **Pull Both Sides** option in the general **Options** panel to create a helix in both directions.
7. Set the handedness of the helix by selecting or deselecting the **Right-Handed Helix** option in the Options panel. To create a **Left-Handed Helix**, deselect the **Right-Handed Helix** option.
8. Pull along the axis to create the helix dynamically.

   As you pull, you can enter dimensions in the helix dimension fields:

   - Pitch is the amount that the helix face shifts per 360 degrees of rotation.
   - Taper is the helix angle.

   Press **Tab** to switch between dimension fields and **Enter** to accept the dimensions and create the helix.

5.4.2.15.2. Creating a Helix Dimensionally

You can also create a helix by entering the height (the total length of the helix). Press **Tab** to switch between dimension fields and preview the helix. Click the **Full Pull** option to create the entire helix based on the dimensions.

The common pull options are described in Pull Options. Once you select the edge or face to pull, select these options from the Options panel, or right-click and select them from the mini-toolbar.

5.4.2.16. Sweeping

You can sweep a face, edge, surface, 3D curve, or other object along a trajectory with the **Pull** tool. Sweeping a face around a closed path creates a torus.

This section contains the following topics:

5.4.2.16.1. Sweeping a Face

1. Click **Pull** in the Edit group on the Design tab.
2. Select the profile you want to sweep.
3. Select the **Sweep** tool guide, then click the sweep trajectory. The sweep trajectory is shown in blue. Alt+double-click to select a tangent chain. Ctrl+click to add contiguous paths.

   You can also select the profile to sweep and then Alt+click the sweep trajectory or Alt+Ctrl+click multiple trajectories. The sweep tool is selected automatically.

   If the trajectory along which you want to sweep is a non-linear curve, you can also Alt+click the curve, which selects the Sweep tool guide automatically.

   If you select the **Normal to Trajectory** option, the sweep is aligned along the origin trajectory

   To quickly sketch a face perpendicular to a desired trajectory, select the end of the trajectory and select a sketching tool to place the sketch grid at that point. Then draw the face.
4. (Optional) Select options from the Options panel, or right-click and select them from the mini-toolbar.

   Select:

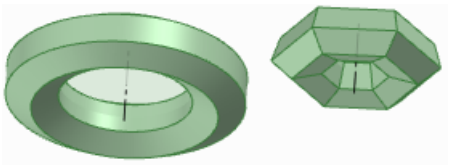
   - Select **Normal to Trajectory** to keep the swept surface normal to the sweep trajectory. If your sweep trajectory is perpendicular to the face you want to sweep, this option is enabled for you.
   - Select **Scale Sections** to control the effect of the vector from the 0-to-X trajectory so that it controls both orientation and scale of the sweep sections. Uncheck the box if you want the vector to control only the orientation of the sweep sections.
   - You can also select from the standard Pull options listed below.
5. Click and drag in the direction of the Pull arrow to sweep the selected objects, or select **Full Pull** from the Options panel or mini-toolbar to sweep the entire length of the trajectory. If you select Full Pull and the profile being swept is in the middle of the trajectory, it will be swept in both directions.

   You can also use the Up To tool guide to select a face or surface on which you want to end the sweep.

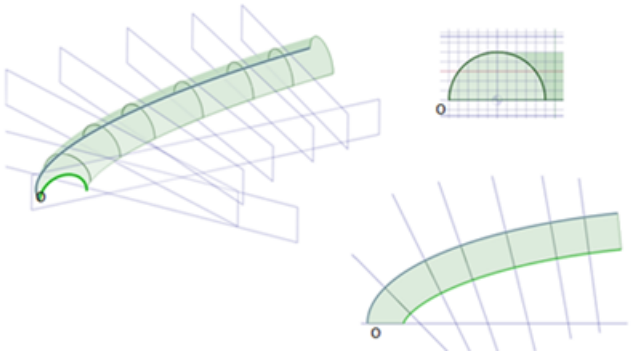
The common pull options are described in Pull Options. Once you select the edge or face to pull, select these options from the Options panel, or right-click and select them from the mini-toolbar.

5.4.2.16.2. Sweeping Examples

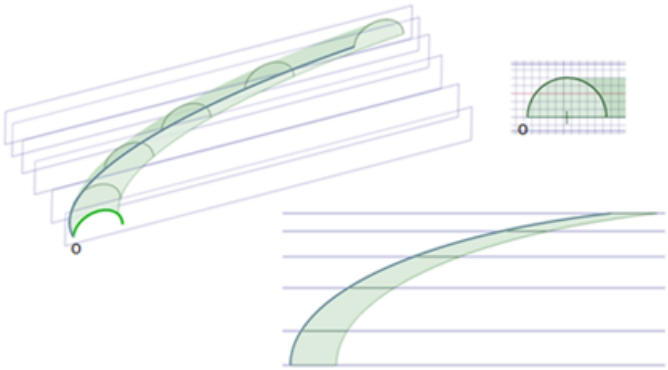
Hexagons swept around circular and hexagonal sweep paths.



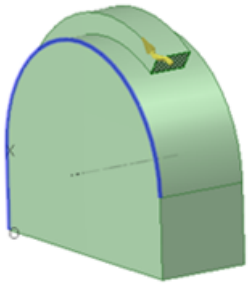
*Sweeping along a primary trajectory with **Normal to Trajectory** selected. Every section plane that is normal to the origin trajectory is the exact same profile as the initial section, limited by the extents of the trajectory.*



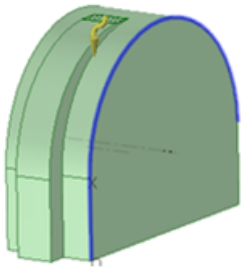
*Sweeping along a primary trajectory with **Normal to Trajectory** not selected. Every section plane that is parallel to the initial section profile is the exact same profile as that initial section, limited by the extents of the trajectory.*



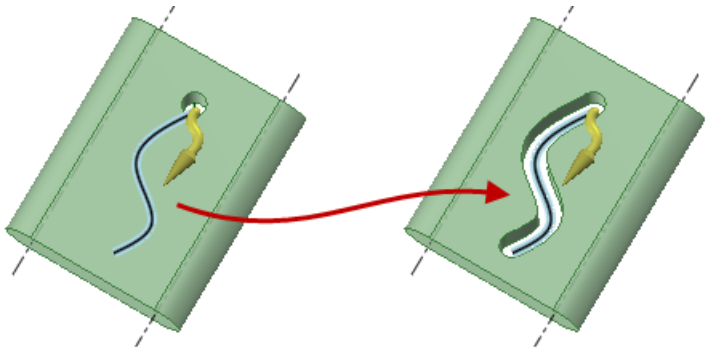
*Sweeping a rectangle along a trajectory with **Normal to Trajectory** selected.*



*Sweeping a rectangle along a trajectory with **Normal to Trajectory** not selected.*



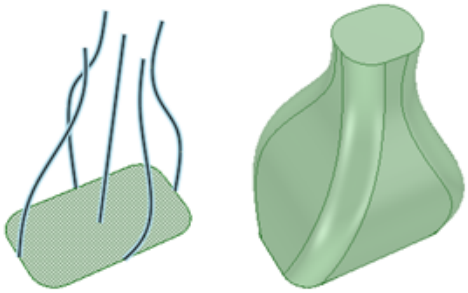
Sweeping a hole along a trajectory by selecting the axis of the hole, then the trajectory.



If you sweep a non-continuous profile, separate swept surfaces are created.

5.4.2.16.3. Sweeping with Multiple Trajectories

You can Alt+Ctrl+click to select multiple trajectories to sweep along. The trajectory you click first is the origin trajectory (labeled O) and the second trajectory is labeled X. If you select the Normal to Trajectory option, the sweep is aligned along the origin trajectory. If the object being swept is in the middle of the sweep, you can pull in either direction to sweep only in that direction.



Sweeps can vary the sketch entities (or edges) along the sweep trajectory. The trajectories can be thought of as guides that make continual changes to the profiles as if they had been dragged while in Sketch mode. Tangency and other constraints are maintained.

When you sweep using more than two trajectories and Scale Sections is turned off, the profile lines and curves will follow the trajectories individually and constraints are preserved. You can see an example in the image below.

You can use a chain of tangent edges as a guide for Sweep.

The extent (or length) of a sweep is limited by the shortest trajectory. If the profile being swept intersects at any point along the trajectory, you can pull in either direction to sweep only in that direction. You can also select Full Pull to create a bidirectional sweep that extends throughout the trajectory.

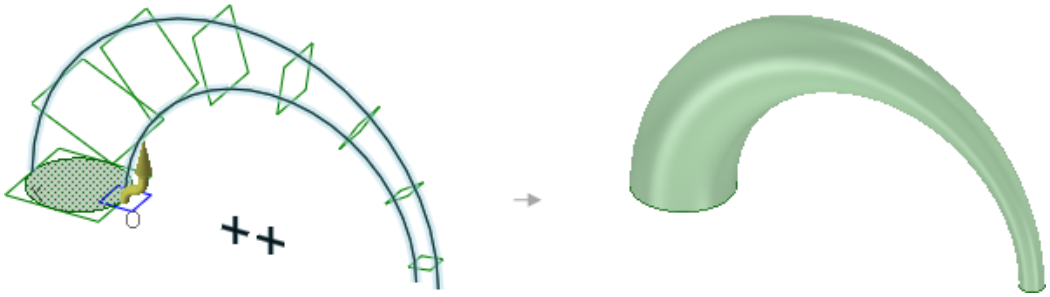
The sweep preview includes the following visual elements:

- The blue square indicates the direction of the origin trajectory at its start, and is localized around the origin trajectory's intersection with the profile to be swept.
- A right-angle symbol is displayed on the origin trajectory when you select the **Normal to origin trajectory** option.
- The preview rectangles are green and scale with the sweep if you select the **Scale sections** option.

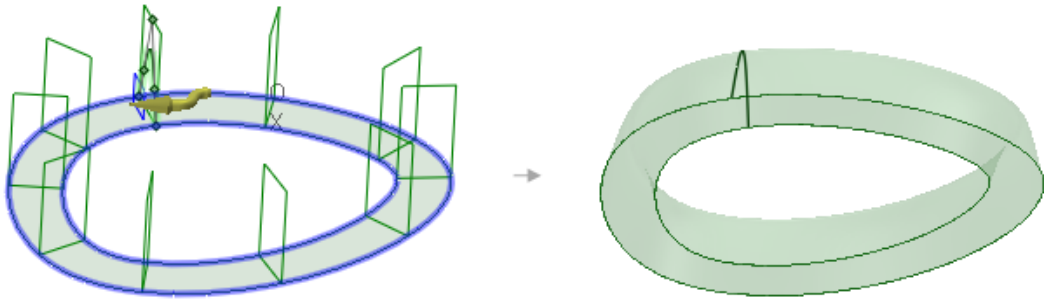
This section contains the following topics:

5.4.2.16.3.1. Sweeping with Two Guides

In a sweep with two guides, every section plane that is normal to the origin trajectory has a profile section that is aligned to the vector defined by the O-X vector in that plane, scaled by the length of the O-X vector, and limited by the extents of the shortest trajectory. If there are two or more trajectories and the spline that connects them (as a profile) is tangent to the surfaces that contain the trajectory edges, then the spline not only scales and aligns to the O-X vector, but also deforms to keep that profile tangent, at all points along the trajectories, to the neighboring surfaces. The profile does not have to touch the trajectories, but the profile plane has to intersect with the trajectories.

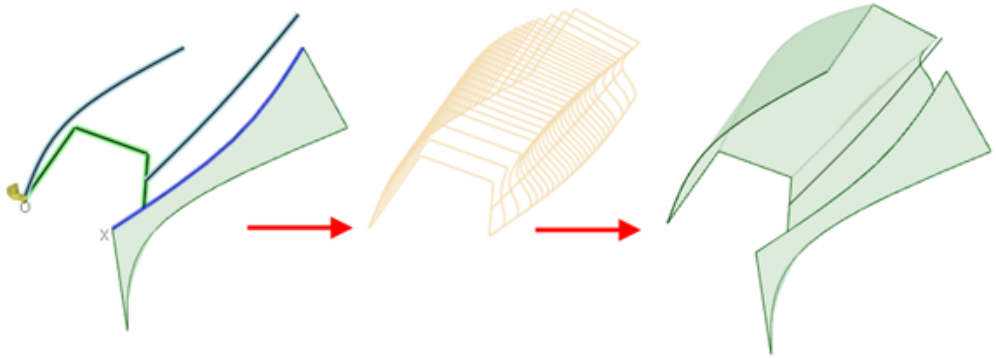


You can also sweep with two closed trajectories.



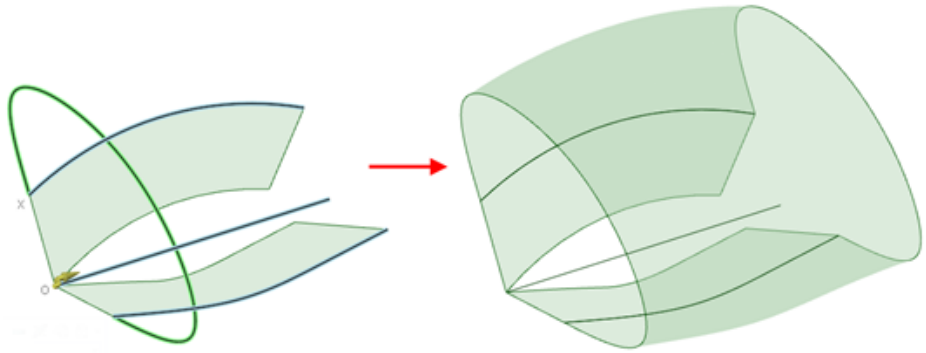
5.4.2.16.3.2. Sweeping Along More than Two Trajectories

If the profile is composed of lines and circles, then guide curves will control the shape of the profile much the same way a sketch is controlled by dragging the mouse. More specifically, there are three constraints: a line or circle that touches a guide curve will maintain the connection throughout the sweep, a circle with a guide curve at its center will stay centered on that guide curve through the sweep, and any members of the profile that are tangent will remain tangent. If the profile contains splines, then any lines or splines attached to a guide curve will deform so they remain attached throughout the sweep. In this case, neighboring faces that are tangent to the profile can be Alt+selected to specify that the profile should remain tangent to the neighboring face throughout the sweep.



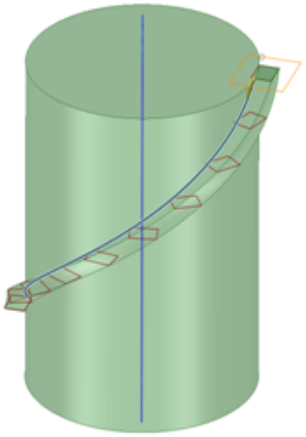
5.4.2.16.3.3. Sweeping Tangent to a Surface

If you select a sweep trajectory that is the edge of a surface, and the object being swept is tangent to that surface, then the initial tangency is maintained along the entire sweep, as shown in the figure below.



5.4.2.16.3.4. Sweeping with a Trajectory and an Axis

If you sweep a profile along a trajectory with an axis as the secondary trajectory, then the profile normal is maintained while the profile orientation changes as it is rotated around the axis.



5.4.2.16.4. Sweeping with an Axis to Set Orientation

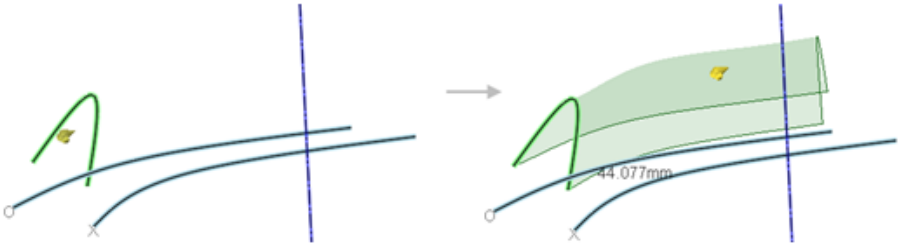
1. Click **Pull** in the Edit group on the Design tab.
2. Select the object you want to sweep.
3. Alt+click the primary trajectory.
4. Ctrl+Alt+click the secondary trajectory.
5. Click the Rotate tool guide and click the axis or line that will set the orientation during the sweep.
6. Drag to create the sweep.

   The object being swept must be in the plane that contains the axis and an intersection with the baseline trajectory.

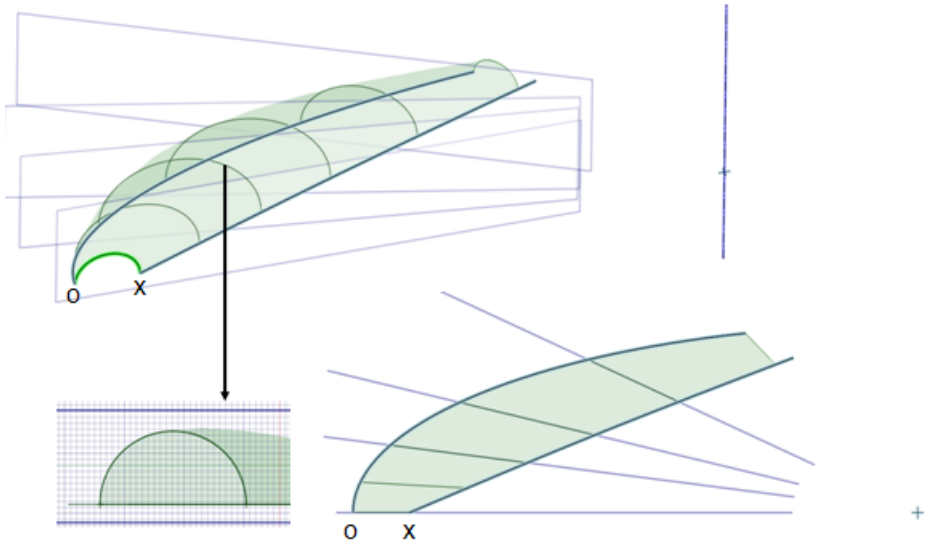
This section contains the following topics:

5.4.2.16.4.1. Set Orientation by Sweeping an Axis Examples

Setting the orientation of the selected object as it is swept along the trajectory.

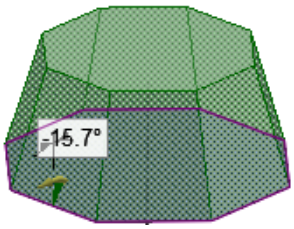


Variable sweep with axis. Every section plane that is cut through a point on the origin trajectory and through the selected axis has a profile section that is aligned to the vector defined by the O-X vector in that plane, scaled by the length of the O-X vector, and limited by the extents of the shortest trajectory. This makes every section appear to radiate around that selected axis.



5.4.2.17. Drafting Faces

You can draft faces around a plane or another face, edge, or surface with the **Pull** tool. Drafting changes the angle of the selected contiguous faces. For example, a mold designer drafts faces to facilitate removal of parts from the mold.



This section contains the following topics:

5.4.2.17.1. Drafting a Face

1. (Optional) Insert a plane around which you want to draft faces.

   You need to do this when you want to draft around a plane that doesn't already exist as geometry in the design.
2. Click **Pull** in the **Edit** group on the **Design** tab. The **Select** tool guide on the right side of the Design window should be active.
3. Select the face or contiguous faces you want to draft.
4. Alt+click the plane, face (including rounds), or surface about which you want to draft. When using a surface as a neutral plane, you can select the surface in the model tree.

   You can also select the Draft tool guide , then click the face or surface. The draft plane, face, or surface is shown in blue.

   > **Tip:** When you need to select more than one face or surface for the Alt-reference (or after selecting the Draft tool guide) then you have to hold the Ctrl and the Alt keys while selecting the additional references.

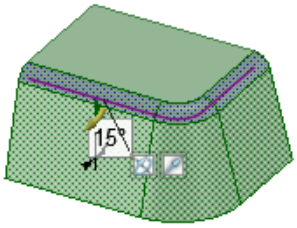
   If you select one edge loop, you can create an edge-driven draft. If you select two edge loops, you can create a split draft.
5. Click and drag in the direction of the Pull arrow to draft the selected faces.

   To dimension the draft, type the rotation angle while you are pulling and press **Enter**.

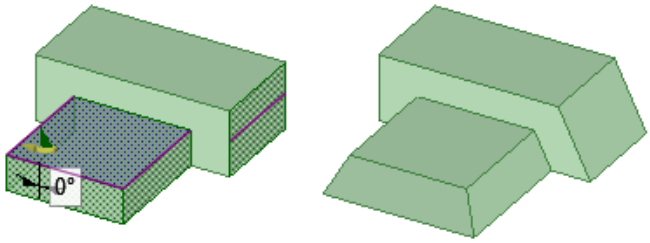
The common pull options are described in **Pull Options**. Once you select the edge or face to pull, select these options from the **Options** panel, or right-click and select them from the mini-toolbar:

5.4.2.17.2. Drafting Faces Examples

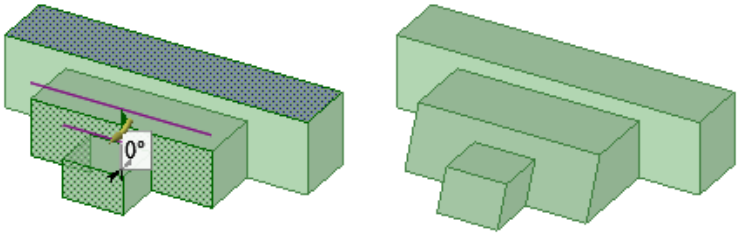
Drafting a face about a round.



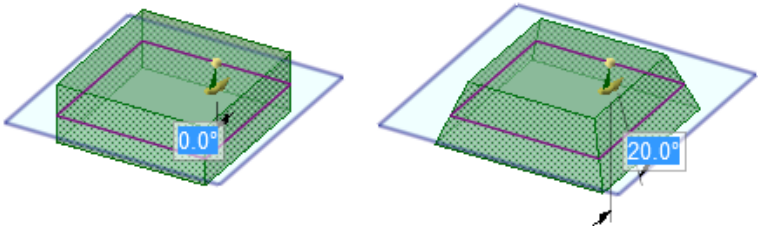
Drafting non-contiguous faces.



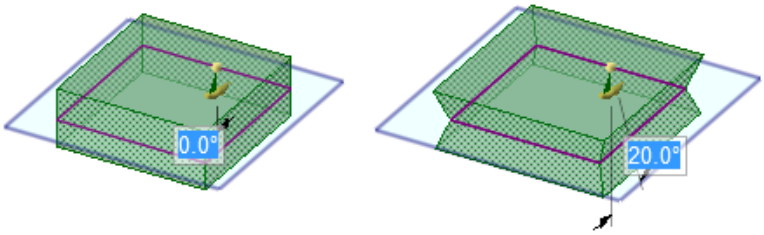
Drafting around a plane that does not touch the faces being drafted.



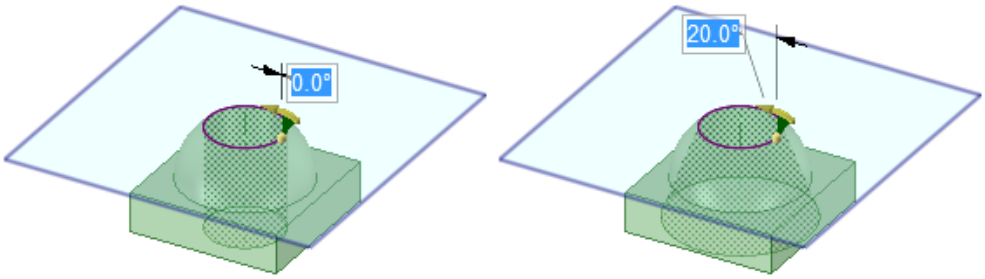
Drafting faces around a plane. The drafted faces pivot around the face or plane you select as the draft plane.



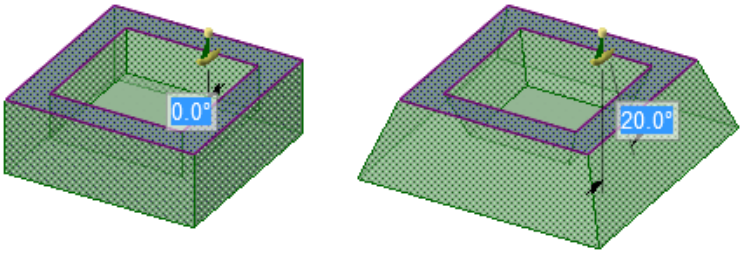
Drafting both sides around the plane.



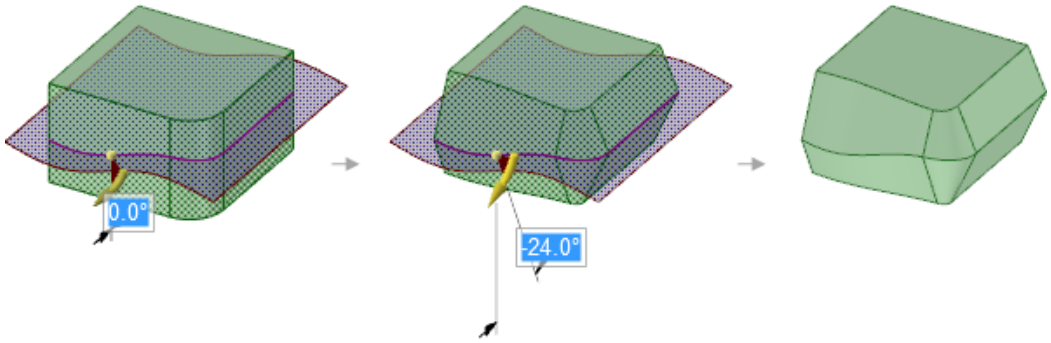
Drafting a face where a plane must be selected as the draft plane. See Inserting a plane for instructions to create a plane.



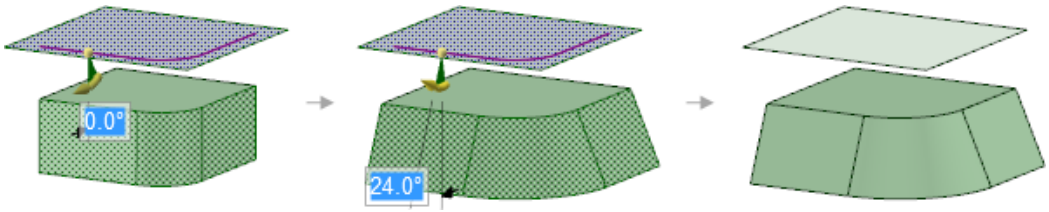
Drafting the offset faces (inside and the outside) of a shelled solid.



*Drafting around a spline surface with the **Draft both sides** option selected.*



Drafting a set of surfaces about a planar surface not attached to the solid.



5.4.2.18. Creating Slots

You can create slots from holes using the **Pull** tool. You can also edit slots. Slots maintain the relationship between their faces.

If you pull a hole with a rounded or chamfered edge into a slot, the slot will have a rounded or chamfered edge.

> **Note:** You must select the axis of the hole, rather than the axis of a round or chamfer on the hole. You will not be able to pull the slot if you select the axis of the round or chamfer.

This section contains the following topics:

5.4.2.18.1. Creating a Slot

1. Click **Pull** in the Edit group on the Design tab.
2. Select the axis of a hole.
   If you have trouble selecting the axis, hover first over the inside face of the hole and the axis will appear.
3. (Optional) Hold Alt and click a face, edge, or vertex to set the direction or trajectory of the Pull.
4. (Optional) Select the **Pull Both Sides** option to create the slot in both directions.
5. (Optional) Select **Full Pull** to pull the slot along the full length of the trajectory, if you selected one.
6. Pull the axis or face of a hole to create a slot.
   If you want to pull in a new direction, press the **Tab** key to change the direction of the Pull handles.

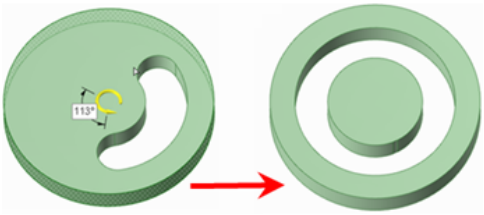
To remove the slot relationship, right-click a face of the slot and select **Remove Association**.

Any changes you make to the slot's face will affect only that face.

This section contains the following topics:

5.4.2.18.1.1. Creating a Curved Slot

1. Select the axis or face of a hole.
2. Select the **Rotate** tool guide and click the axis of the driving cylinder.
   You can also **Alt**+click the axis of the driving cylinder.
3. Pull the hole to create a curved slot.
   You can pull a curved slot 360 degrees to make a round cut.



5.4.2.18.1.2. Creating a Radial Slot

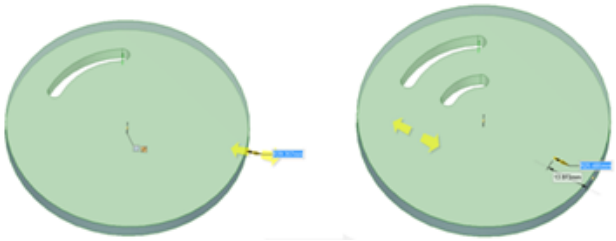
1. Select the axis or face of a hole.
2. Do one of the following:
   - **Alt**+click the face of the driving cylinder to set the direction of the Pull towards the axis of the driving cylinder. Click the **Pull Direction** tool guide and pull the hole to create a radial slot.
   - **Ctrl**+click the face of the driving cylinder to set the direction of the Pull towards the axis of the driving cylinder. Click the **Pull Direction** tool guide and pull the hole to offset the cylinder and create a radial slot simultaneously.
   - Select the Pull arrow that points toward the axis and pull.

   If you want to lengthen a radial slot while keeping it the same distance from a radial face, **Ctrl**+click the radial slot's axis and the face, then pull.

   If you want to move a slot along with a radial face **Ctrl**+select the hole's axis and the face, then **Ctrl**+pull.

5.4.2.18.1.3. Creating an Offset of a Radial Slot

1. Select an axis of a slot.
2. Hold the **Alt** key and select the cylinder face or face to set the offset direction.
3. Hold the **Ctrl** key and pull or enter a radius value.



5.4.2.18.1.4. Creating a Swept Slot

1. Select the axis of a hole.
2. Select the **Sweep** tool guide.

3. Click the sweep trajectory.

   A sweep mode arrow indicates the beginning of the sweep.

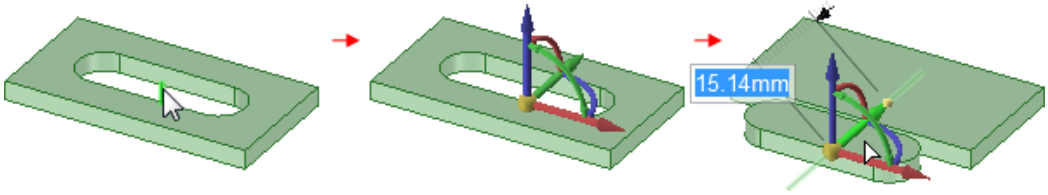
4. Click and drag in the direction of the Pull arrow to sweep the selected objects.

   You can also select **Full Pull** from the Options panel or the mini-toolbar to sweep the entire length of the trajectory. If you select Full Pull and the face or surface being swept is in the middle of the trajectory, it will be swept in both directions.

5.4.2.18.1.5. Editing a Slot

You can modify a hole that was dragged to form a slot by pulling on any of the slot axes.

- If the slot has a rounded or chamfered edge, double-click the round or chamfer to change its dimensions.
- If you select **Detach first** in the Options panel, then the slot will become a solid if you use the Move tool to move it off the body, as shown.



5.4.2.18.1.6. Moving a Slot Radially

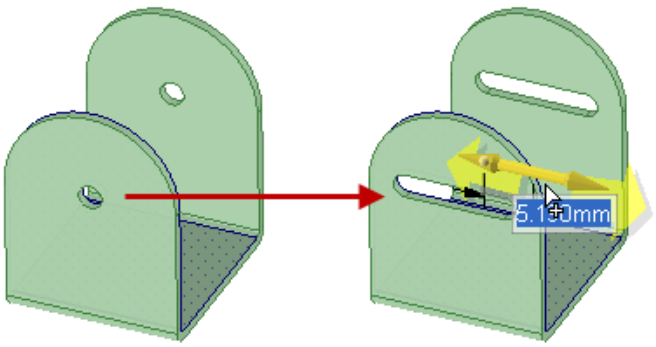
1. Select the **Pull** tool.
2. Select both axes of the slot.
3. Do one of the following:

   - **Alt**+click the face of the driving cylinder to set the direction of the Pull towards the axis of the driving cylinder. Click the **Pull Direction** tool guide and Ctrl-drag to move the slot radially.
   - **Ctrl**+click the face of the driving cylinder to set the direction of the Pull towards the axis of the driving cylinder. Click the **Pull Direction** tool guide and Ctrl+drag to offset the cylinder and move the slot.

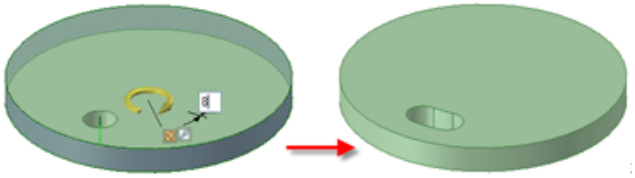
   You can also move a hole radially by pressing **Shift** while dragging it with the Pull tool.

5.4.2.18.2. Creating Slots Examples

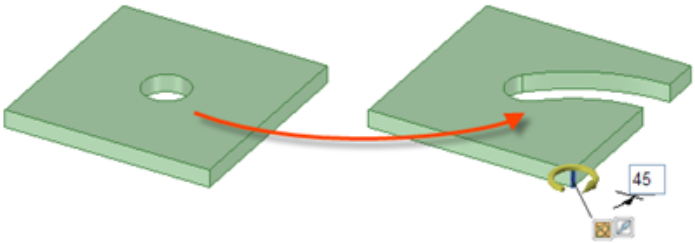
Pulling mirrored holes to create slots.



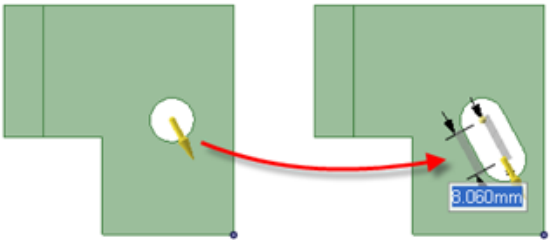
Creating a circular slot by entering an angle dimension with the Pull Both Sides option.



Creating a radial slot by selecting an axis or edge parallel to the hole's axis as the Pull direction.



Creating a slot using a vertex as the Pull direction.



5.4.2.19. Scaling Solids and Surfaces

You can scale solids and surfaces with the **Pull** tool. You can scale multiple objects in different components.

This section contains the following topics:

5.4.2.19.1. Scaling a Solid, Surface or Mesh

1. Click **Pull** in the Edit group on the Design tab.
2. Select a solid, surface, or mesh (meshes also have a **Scale** tool in the Facets tab).
3. Click the **Scale Body** tool guide.
4. Click a point, vertex, or origin to set the origin for scaling.
5. Pull to scale dynamically (a field appears to show the scale as you pull) or press the spacebar and enter the scale. You can enter mathematical expressions to calculate a scale factor.

   To scale using an annotation dimension, click the spark icon next to the dimension and type a new value.

> **Note:** Select the solid or surface, Alt+click a vertex to anchor the scale, and pull.

5.4.2.19.2. Scaling a Solid Asymmetrically

1. Click **Pull** in the Edit group on the Design tab.
2. Select a solid body (meshes can be scaled asymmetrically using the **Scale** tool in the Facets tab).
3. Click the **Scale Body** tool guide.
4. Click a point, vertex, or origin to set the origin for scaling.
5. Hold **Alt** and select an origin axis, or hold **Alt** and **Ctrl** and select a point and a line.
6. Drag in either direction of the line to scale the part in that direction.
   You can press the spacebar and enter a scale value while you are dragging.

5.4.2.19.3. Converting an Existing Solid or Surface from Millimeters to Inches

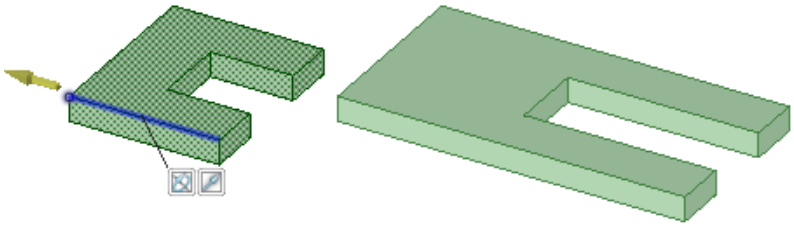
1. Change the units to inches as described above.
2. Select the Pull tool in the Edit ribbon group on the Design tab.
3. Select the object you want to convert.
4. Scale the object by 25.4.

5.4.2.19.4. Scaling a Datum Plane or Axis

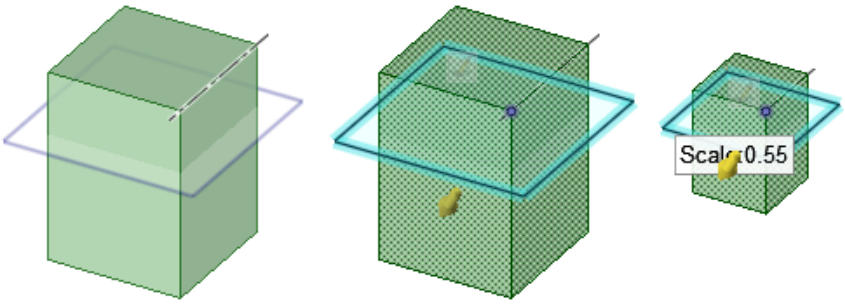
1. Click **Pull** in the Edit group on the Design tab.
2. Select the datum plane and/or axis **and** select a curve, surface, or solid.
   You must hold **Ctrl** and select the plane. Box-select will not select it.
3. Click the **Scale Body** tool guide.
4. Select an anchor point.
5. Drag in the direction of the Pull arrow.

5.4.2.19.5. Scaling Solids and Surfaces Examples

Scaling a solid asymmetrically.



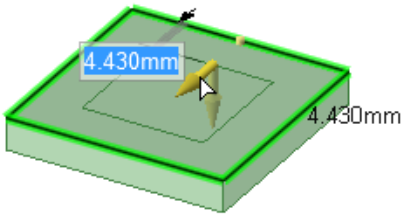
Scaling a plane and an axis with a solid.



5.4.2.20. Copying Edges and Faces

You can copy edges and faces by selecting the **Pull** tool's **Extrude Edge** option. You can also copy edges and faces with the Move tool.

When copying and pasting the edge of a surface as a curve or line, the lines are placed on the active layer and remain selected until you clear the selection. This feature is useful if you need references to rebuild a surface.



5.4.2.20.1. Copying an Edge or Edges

1. Click **Pull** in the Edit group on the Design tab.
2. Select the edge or edges you want to copy. Double-click to select a chain of edges.
3. Select the **Copy Edge** option in the Options panel or from the mini-toolbar.

   You can also Ctrl+drag with the Pivot Edge or Extrude edge options selected to copy the edge.

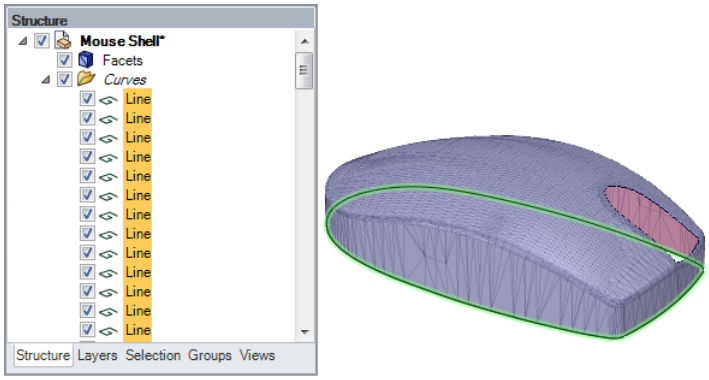
   The Pull arrow changes to show the two directions in which you can create copies of the edge. One arrow is highlighted to show the primary direction.
4. If the arrow pointing the direction in which you want to copy the edge is not highlighted, click the arrow or press **Tab** to change the direction.
5. Drag the edge in the direction of the highlighted Pull arrow.

   During the pull, the distance between the copied edge and the new edge is displayed. When you create an external edge, a surface is created between the copied edge and the new one.

> **Note:** Press **Ctrl** and drag to copy the selected face.

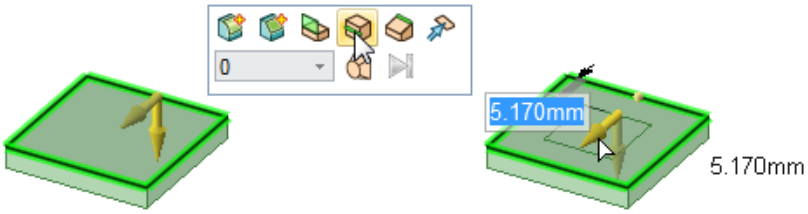
5.4.2.20.2. Copying Mesh Edges

When you Copy and Paste mesh edges, the result is a polyline.

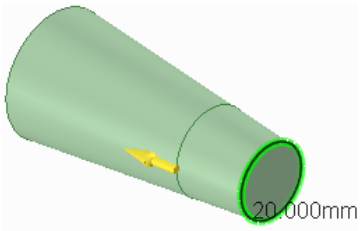


5.4.2.20.3. Copying Edges and Faces Examples

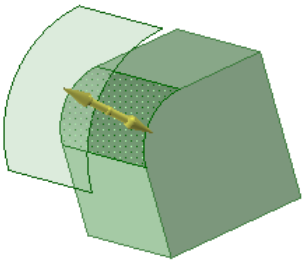
Copying edges to create new edges on a face.



When copying an edge, the edge adjusts based on the solid's geometry.



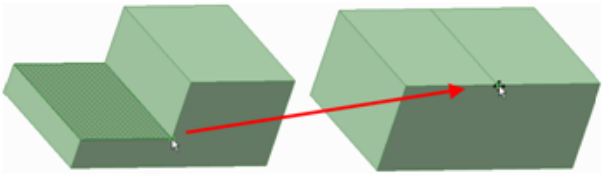
Copying a round face.



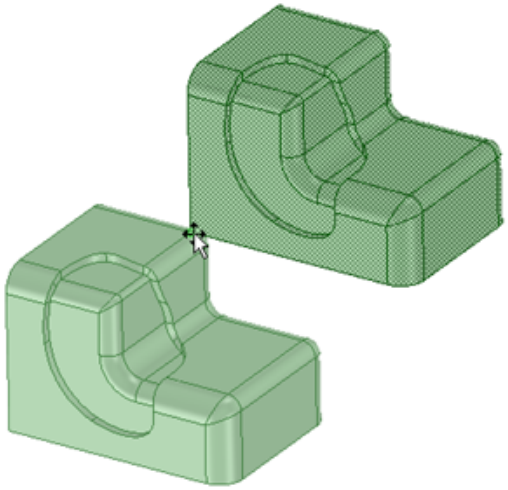
5.4.2.21. Pulling with the Select tool

You can pull with the **Select** tool or the Pull tool.

Dragging the vertex of a selected face to another vertex to pull the selected face.



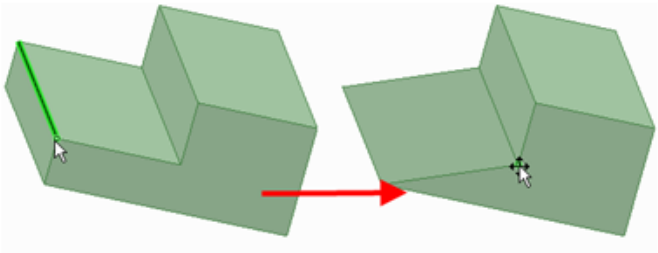
Dragging a copy of a solid back into the solid.



5.4.2.22. Pivoting an Edge with the Select Tool

You can pivot an edge with the **Select** tool or the Pull tool.

Drag the vertex of a selected edge to another vertex to pivot the selected face, as shown in the figure below.



5.4.2.23. Parameterizing Measurements Through Pull

1. On the **Design** tab, in the **Edit** group, click **Pull**.
2. Select an object to pull.
3. On the right side of the **Pull** tool HUD, select **Add a geometric parameter**.

   You are now in the **Pull – Parameterization** tool.
4. Select a second object to measure, or multiple objects to measure between.
5. On the right side of the **Pull – Parameterization** tool HUD, select the property to measure and parameterize. For example, some properties include **Area**, **Length**, **Mass**, **Volume**, but many more exist and depend on your geometry and selection.

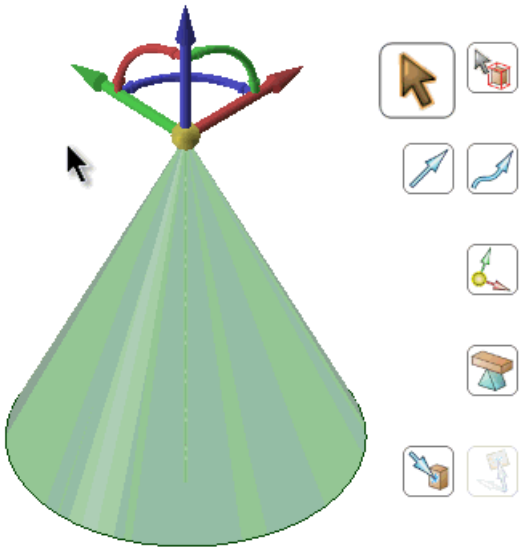
   A measurement is displayed in the graphics window closest to the object you select.

6. Click **Create parameter** to add the measurement to the defined variations table for parameterization.
   This exits the **Pull – Parameterization** tool and returns you to the **Pull** tool.
7. In the SID, click the **Defined parameters** hex to view the parameter in the defined variations table.

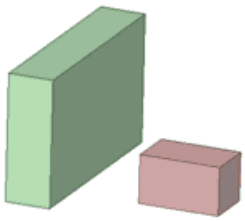
5.4.3. Move Tool

Use the **Move** tool to move any object in 2D or 3D, including drawing sheet views. The behavior of the **Move** tool changes based on what you have selected.

- If you select an entire object, such as a solid, surface, or sketch, you can translate or rotate the object.
- You can move one side of a solid, surface, or sketch to enlarge or reduce the size of the object.
- If you move an object into another object in the same component, the smaller object is merged into the larger one and receives the larger object's properties.
- Moving a component moves everything contained within the component.
- You can move a circular edge of a flat surface the same way you move a circular sketch curve.
- Mesh bodies, selected mesh facets, and mesh boundary loops can also be moved.
- Moving the apex of a cone changes the height. Anchor the **Move** tool to the outer face to scale the cone.



When you move a component that has been assembled using assembly constraints, the Move handle is positioned at the constraint and the axes that are constrained are disabled. If the assembly constraints only allow movement in one direction, then that direction will be automatically selected. For example, if you move a component with a Center Axes assembly constraint, the Move handle is positioned on the axis and you can only move the component in directions that will keep the axes aligned.



Note: If the Move handle appears disabled, check the model tree to determine if an assembly condition exists for the component you are trying to move.

Offset, mirror, and coaxial inferred relationships also affect Move.

Note: If you entered the **Design** tab with sheet metal features selected, the **Move** tool will work as it does in Sheet Metal. To work as usual, right-click the sheet metal part in the model tree and choose **Suspend Sheet Metal** in the context menu.

This section contains the following topics:

5.4.3.1. Moving Objects

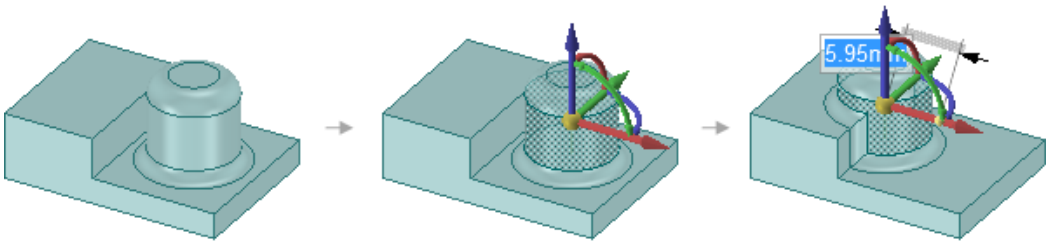
1. Click **Move** in the Edit group on the Design tab.
2. Select the object(s) that you want to move.
3. Select the following options:
   - **Move grid**: Select this option to move the sketch grid.
   - **Ruler**: Once you select an axis on the Move handle, select this option and click an edge or face to anchor the ruler. The ruler is oriented along the selected Move handle axis. Specify a value to use the ruler to dimension the move.
   - **Create patterns**: Select this option if you want to create a pattern by dragging selected objects with the Move tool. Dragging creates a copy of the selected object, moves it to a new location, and creates a pattern relationship. Select the **Maintain orientation** option to keep the initial orientation of the original object when you rotate or translate the pattern. See Creating a pattern.
   - **Detach first**: Detach the selected protrusions and depressions, move them, and reattach them at the new location.
   - **Maintain sketch connectivity**: Keeps the connections between a sketch curve and other curves that share its end points. If you deselect this option and move a sketch curve, the curve will move independent of other curves.
   - **Remember orientation**: Sets the orientation of the Move tool for the object. The orientation is only remembered for the current session. You can change the Move handle orientation by using the Direction tool guide, holding Alt and selecting a reference object, or by dragging a ball on the Move handle's axes. You can select the following options in the dropdown list to the right of the option:
     - **Default**: The Move handle orientation is determined by the object(s) you select.
     - **Global**: Saves the current Move handle orientation, and this orientation is used for all objects.
     - **Per Object**: Saves the current Move handle orientation for the selected object. When you select the object again with the Move tool active, the Move handle will be oriented at its saved orientation.
4. Click an axis and drag in that direction to move the selected object.

   A line extends from the Move handle axis to indicate the direction you selected for movement.

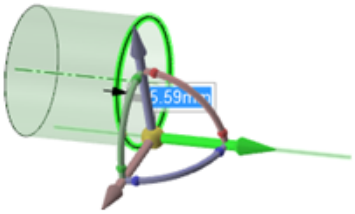
   If the move fails, the Move handle is repositioned to the last valid location and orientation. If you are trying to move a protrusion surrounded by round faces, you may need to fill the rounds.
5. You can also Alt + select a plane between Move handles to invoke free drag movement within that plane. Place the Move tool on any movable object and then hold the Alt key. Quarter circle planes appear between the Move handles. Selecting one changes the cursor to a free drag cursor and allows free movement within the plane. Select any Move handle to disable the free drag.

Note: The cursor does not need to be on the axis to move the selected object. In fact, you may find it easier to control the move if you drag some distance from the entity and the Move handle.

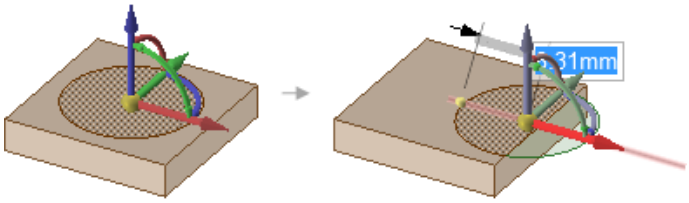
*Moving a protrusion with rounds that intersects with a stepped solid with the **Detach first** option*



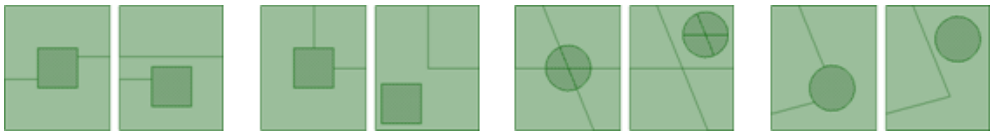
Moving a circular edge along a surface



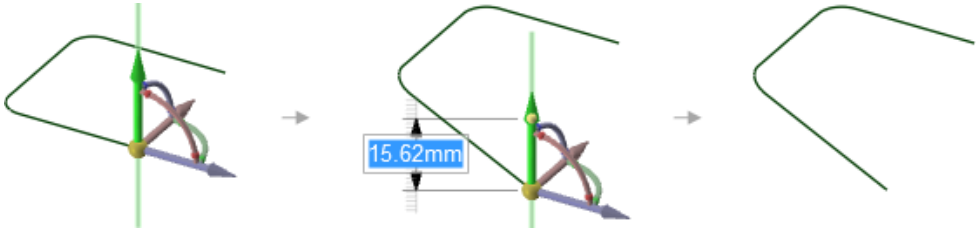
Moving an imprinted face off of a solid face creates a surface.



Moving imprinted edges on a face, and other intersecting imprinted edges are adjusted as needed. The examples above show how the edges are adjusted when the imprinted areas are moved.



Moving the end point of a line segment that has a tangent arc on its other end changes the arc so that it remains tangent to the line segment as you move its end point.



This section contains the following topics:

5.4.3.1.1. Moving Relative to Other Objects

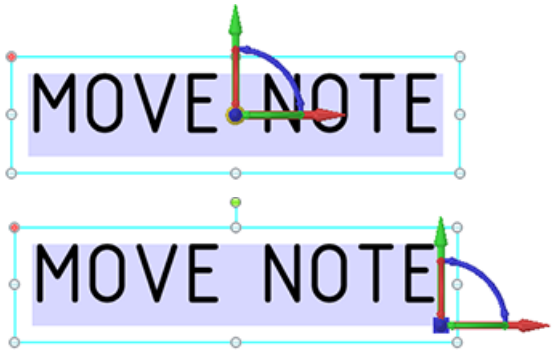
You can:

- **Snap to a co-planar face**: Hold **Shift** while dragging to snap to co-planar faces when you have the Move handle anchored to a planar face.
- **Move an object up to another object**: Click the **Up To** tool guide to move objects so the center of the Move handle is adjacent to the object.
- **Orient an object to another object**: Select the object to move and a Move handle axis, then click the **Orient to Object** tool guide and select a second object. The selected object will be rotated so the selected Move handle axis is aligned with the second object.

5.4.3.1.2. Changing the Anchor Location of the Move Handle

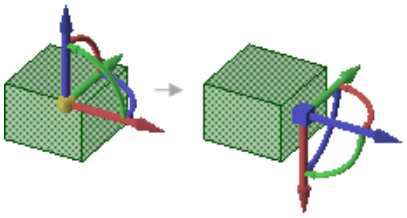
1. Drag the yellow center sphere on the Move handle to snap it to other geometry.

   Annotations have multiple Drag Points. The Move handle can be dragged near, and snapped to, any of these.



2. Click the **Anchor** tool guide and select the face, edge, or vertex on which to place the Move handle.
3. You can also click the **Origin** tool in the Design tab to insert an origin anywhere in your design that you want to anchor the Move tool.

The yellow center sphere turns into a blue cube when the Move handle is anchored.



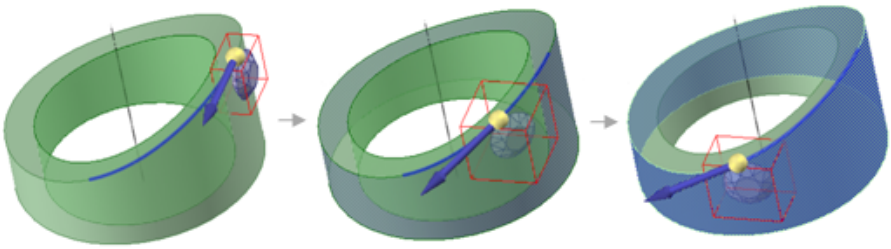
5.4.3.1.3. Changing the Direction or Trajectory for the Move

- Drag one of the small balls on the rotational axis to reorient the Move handle, or dimension the orientation by typing the rotation angle *while you are dragging*, then pressing **Enter**.
- You can also Alt+click a point or line, or click the **Move Direction** tool guide, then click a point or line, to orient one of the Move handle's axes toward that point or along that line.

  If you Alt+click a trajectory, you can move along the trajectory. Ctrl+Alt+click to add contiguous lines or edges to the trajectory.

  If you Alt+click a plane, the direction of movement is set perpendicular to the plane.

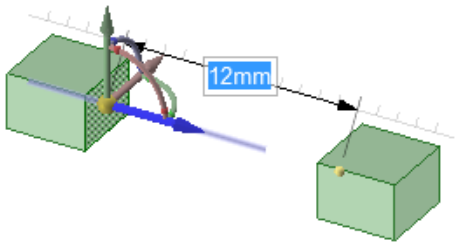
Normal to Surface - While moving the diamond along the selected trajectory, Ctrl+Alt+click the face of the cylinder to set the orientation of the diamond normal to the cylinder.



5.4.3.1.4. Dimensioning a Move

1. Follow the steps to move an object.
2. After you click an axis on the Move handle, click **Ruler** in the Options panel.
3. Click an edge or face to anchor the ruler.

   The ruler is oriented along the selected Move handle axis.
4. Type a distance and press **Enter**.

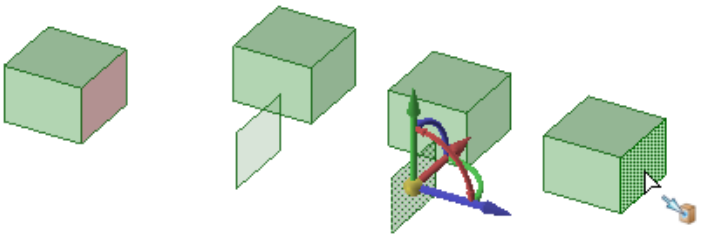


5.4.3.1.5. Copying an Object Using the Move Tool

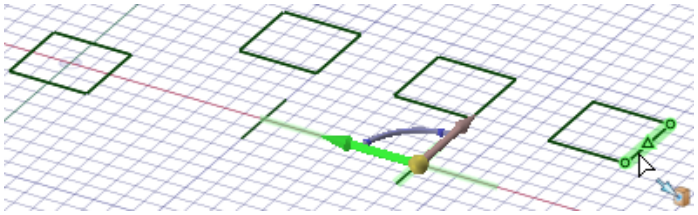
- Press **Ctrl** to copy the object selected for movement and place it at the location at which you drag or dimension the move.

  You can press the spacebar to dimension the move.
- Double-click the **Up To** tool guide to make multiple copies of the selected object. To exit this mode, select another tool or click in empty space in the Design window.

*Copying the red face multiple times by double-clicking the **Up To** tool guide to keep it active for more than one move*



*Making copies by double-clicking the **Up To** tool guide also works with sketches*



5.4.3.2. Move Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

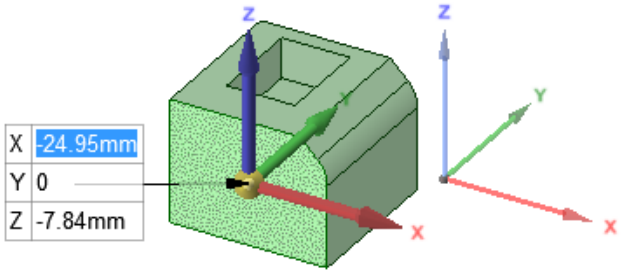
Select	The **Select** tool guide is active by default. When this tool guide is active, you can select faces, surfaces, solids, or components within the Move tool.
Select Component	Click any object with the **Select Component** tool guide to select the solid to which the object belongs. If the solid is the only object in its component, the component will be selected.
Move Direction	Select a point, vertex, line, axis, plane, or planar face with the **Move Direction** tool guide to orient the Move handle and set the initial direction of the move. (The object will not move until you drag.)
Move Along Trajectory	Select a set of lines or edges with the **Move Along Trajectory** tool guide to move the selected objects along that trajectory. For best results, perform Moves along trajectories in small increments. If the object to be moved is a protrusion, it will be detached, then reattached in the new location. When you move a protrusion along a trajectory, rounds are automatically removed. Ctrl+Alt+click a face to control the orientation of the object being moved or patterned along.
Anchor	Select an object, then use the **Anchor** tool guide to select the face, edge, or vertex that will anchor the move. You can anchor the Move handle to a temporary object, such as the intersection between two axes by Alt+Shift+clicking the two objects.
Fulcrum	Select an object, then use the **Fulcrum** tool guide to move other objects around it. Select a pattern member to anchor it, or select a component to explode an assembly. See Moving with the Fulcrum tool guide.
Move radially about axis	The **Move radially about axis** tool guide allows you to select an axis to move the selected objects radially about. Once you select an axis, the Move handle will reorient to have one axis parallel to the move axis and one axis in the radial direction.

Up To	Once you select the object to move and a Move handle axis, use the **Up To** tool guide to select the object you want to move up to. If a Move handle axis is selected, the Move is limited to that direction. If no handles are selected, the object is translated until the center of the Move handle lies on the selected reference. (A move handle must be selected to move up to the axis of an origin.) In a linear move to an intersecting object, the center of the Move handle is moved to the selected object. If the two objects do not intersect, the first object is moved along the desired direction up to the closest point to the second object. You can use this tool guide to: • Select a point along a trajectory or the axis of an origin to move up to. • Move the sketch grid in Sketch and Section modes. • Move an axis so it is coincident with another axis in a body. If you move the axis of a pattern, all pattern members will move together to the new location. You can double-click the **Up To** tool guide to keep it active. While the tool guide is active, it will copy faces and surfaces instead of moving them. To deactivate the tool guide, click it again, select another tool guide, or exit the Move tool.
Orient to Object	Once you select the object to move and a Move handle axis, use the **Orient to Object** tool guide to click an object. The selected object will be rotated until the selected Move handle axis is aligned with the clicked object. You can also use this tool guide to rotate the sketch grid in Sketch and Section modes.

5.4.3.3. Move Tool Options

The following options are available in the Move tool:

Move grid	Select this option to move the sketch grid.
Symmetric Move	Select this option to move symmetrically
Measure	Opens the **Measure** tool. Selecting a measurement result returns you to the **Move** tool. When you select move direction, the measurement value is displayed in a dimension box with an arrow pointing to the measured object. Modify the value for a one-time adjustment of the model or create a Measurement Group which can be modified at any time.
Ruler	Once you select an axis on the Move handle, select this option and click an edge or face to anchor the ruler. The ruler is oriented along the selected Move handle axis. Specify a value to use the ruler to dimension the move.
Maintain orientation	Select this option to maintain the orientation of the object when rotating or moving along a trajectory.
Create patterns	Select this option if you want to create a pattern by dragging selected objects with the Move tool. Dragging creates a copy of the selected object, moves it to a new location, and creates a pattern relationship.
Maintain orientation	Select the **Maintain orientation** option to keep the initial orientation of the original object when you rotate or translate the pattern. See Creating a pattern.

Detach first	Select this option to detach selected protrusions and depressions, move them, and reattach them at the new location.
Maintain sketch connectivity	Keep the connection between a sketch curve and other curves that share its end points. If you deselect this option and move a sketch curve, the curve will move independent of other curves.
Fixed beam	This option is for beams and causes the beam to remain fixed while the profile becomes offset from the beam.
Remember orientation	Sets the orientation of the Move tool for the object. The orientation is only remembered for the current session. You can change the Move handle orientation by using the Direction tool guide, holding Alt and selecting a reference object, or by dragging a ball on the Move handle's axes. You can select the following options in the dropdown list to the right of the option: • **Default**: The Move handle orientation is determined by the object(s) you select. • **Global**: Saves the current Move handle orientation, and this orientation is used for all objects. • **Per Object**: Saves the current Move handle orientation for the selected object. When you select the object again with the Move tool active, the Move handle will be oriented at its saved orientation.
Enter XYZ coordinates	This option allows you to enter X, Y, and Z distances to move relative to the World Origin. When you select the option, the Move handle re-orients to be parallel to the World Origin and displays X, Y, and Z input panels. 

5.4.3.4. The Move Handle

You will use the **Move** handle to move objects in 2D and 3D. The Move handle allows you to translate and rotate objects. You use the Move handle by clicking the axes of the Move handle and dragging to move the selected object.

When you select the object you want to move, and click one of the Move tools, Discovery guesses at the anchor point and orientation of the Move handle. If either of these is incorrect, you can change them.



This section contains the following topics:

5.4.3.4.1. Translating Objects Using the Move Handle

1. Click the axis that is aligned with the direction you want to move the selected object.

2. Drag in the direction of the axis to move the object.

   You can **Ctrl+drag** to create a copy of the object while moving.

5.4.3.4.2. Rotating Objects Using the Move Handle

Use the curved rotational axis that is aligned with the direction you want to rotate:

- Drag the axis to rotate the selected object.

  You can hold the Ctrl key to create a copy of the object while moving.

- Double-click the axis to rotate the object 90°. You can hold the Ctrl key while double-clicking to create a copy of the object at 90°.

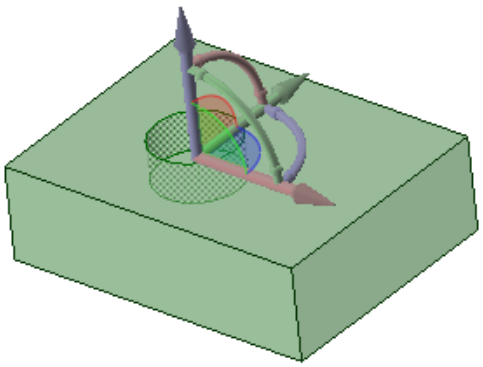
5.4.3.4.3. Moving Freely Within a Plane Using the Move Handle

Use the planes between Move handles to move within a plane.

- Hold the Alt key
- Quarter circle planes display between the Move handles
- Select the plane to move within
- Release the Alt key and start dragging.

  While dragging, the cursor changes to the free drag cursor.

- Clicking on any Move handle cancels free drag movement.



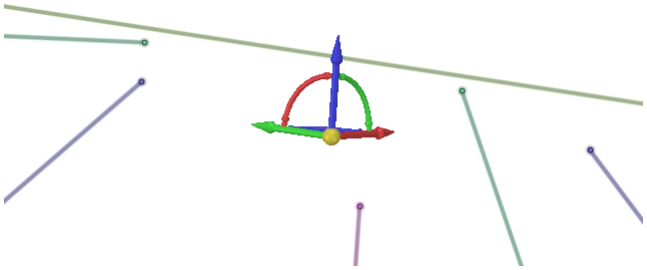
5.4.3.4.4. Realigning the Axes of the Move Handle

You can realign the Move handle in the following ways:

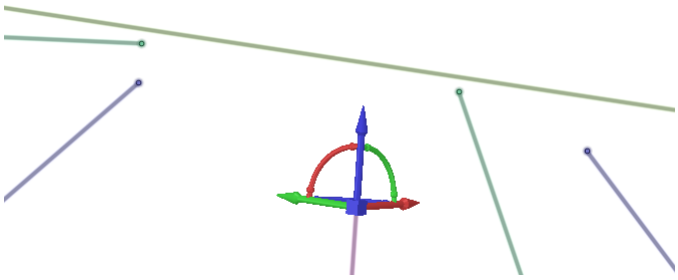
- Drag the small balls on each rotation axis of the Move handle to reorient it. While dragging, you can also press the spacebar to enter the angle directly.
- Drag the center ball of the Move handle (or use the Anchor tool guide) to place it on another solid, face, edge, or vertex. If you use this method, note that the selected object is still the one that is going to move.
- Alt+click a point or line (or use the Move Direction tool guide) to orient the closest linear axis toward that point or along the line.

5.4.3.4.5. Moving Multiple Beam Endpoints to a Single Point

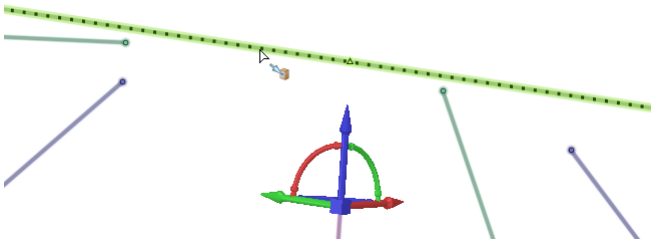
1. Click the Move tool and select the endpoints (you can also use box select).



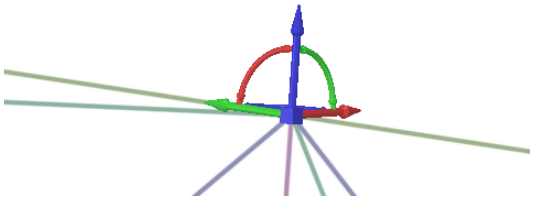
2. Anchor the Move Handle by using the Anchor tool guide or by dragging the yellow ball.



3. Click the Up To tool guide and select the Up To reference.



4. The endpoints will all snap to the single point.



5.4.3.5. Moving Symmetrically

Use the **Symmetric Move** option in the **Move** tool to move objects relative to each other about a plane as if they are mirrored objects but without the need to create a mirror association between the objects. Unlike the Mirror tool, the Symmetric move option can be used to move dissimilar geometry as well as similar.

You can use this option with an automatically determined virtual mirror plane based on the Move handle location or you can use the fulcrum tool guide to establish a fixed mirror plane.

With a fulcrum-selected mirror plane, geometry which is the same on both sides of the plane is found and moved automatically when moving geometry on one side of the plane. The center of each selected object determines which side of the plane the geometry lies on.

Faces, edges, vertices, section curves, datum planes, and sketch curves can be moved symmetrically. Curves that partially overlap the symmetry plane can be moved. The symmetric move option is not available for moving objects which have an established mirror association

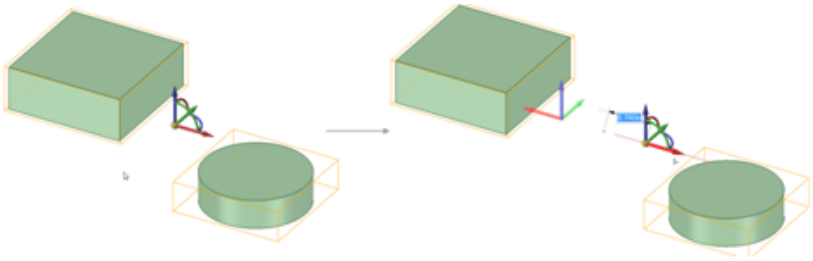
This section contains the following topics:

5.4.3.5.1. Moving Objects Symmetrically About a Virtual Mirror Plane

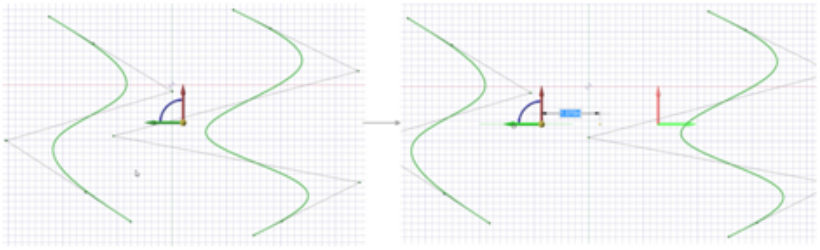
1. Select two or more objects.
2. Click the **Move** tool.
3. Click **Symmetric Move**in the Move options panel.
4. Click one of the translation arrows and drag. As you drag, a ghost mirror plane appears at the center of the Move handle, perpendicular to the direction you are dragging. Movements are mirrored about this plane. A ghost move handle is displayed on the other side of the virtual mirror plane to indicate the mirrored translation or rotation. Selected objects are moved symmetrically about the indicated plane.

   To reset the virtual mirror plane, change selection or toggle the **Symmetric Move** option.

Moving two solids symmetrically about a virtual mirror.



Moving two curves.

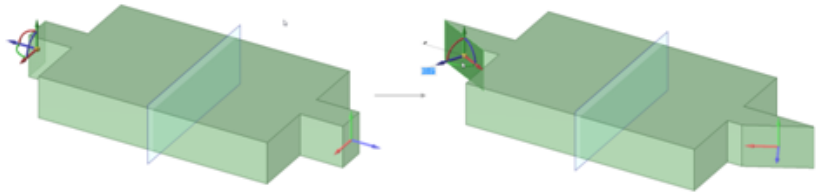


5.4.3.5.2. Moving Objects Symmetrically with a Fulcrum-selected Mirror Plane

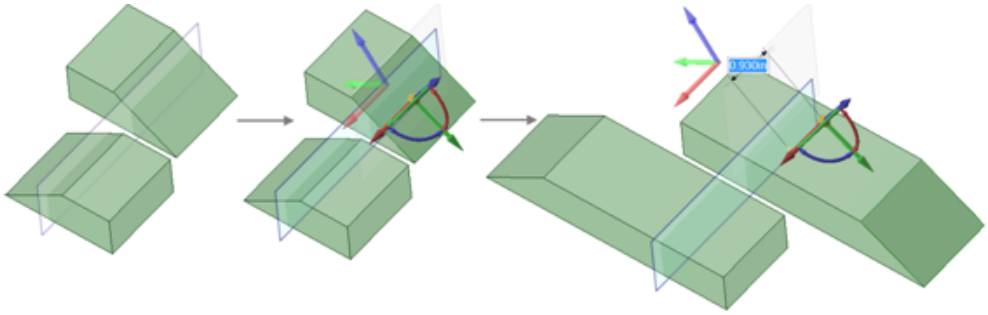
1. Create a plane or planar face.
2. Select one or more objects.
3. Click the Move tool.
4. Click **Symmetric Move** in the Move Options panel.
5. Click the **Fulcrum** tool guide.
6. Click the plane or planar face to specify the symmetry plane.

   Objects on the other side of the fulcrum plane with matching geometry are automatically detected and moved symmetrically about the plane.
7. Drag a rotation or translation arrow on the move handle to modify the geometry symmetrically about the fulcrum-selected mirror plane.

Pivoting an edge modifies the matching geometry on the other side of the fulcrum-selected mirror plane.



Moving faces that are symmetric to each other but lie partially on either side of the mirror plane. The center of each of the objects' bounding boxes is on either side of the plane, so the symmetric move tool can detect the symmetric geometry and the angled faces can be moved.



5.4.3.6. Creating Patterns

You can create a pattern of protrusions or depressions (including slots), points, components, planes, origins, or axes. You can also create patterns of sketch curves. You cannot create a pattern of threads. You can use the **Undo** button (**Ctrl+Z**) or **Redo** button (**Ctrl+Y**) when creating a pattern.

You can also create a pattern from a mix of object types, such as a pattern of holes (faces) and bolts (imported components). In Discovery, any pattern member can be used to modify the pattern after you create it. If the change cannot be made to all pattern members, the member that cannot change is still part of the pattern. Once you select a Pattern Type, the fill displays to show a pattern preview.

When you select a member of a pattern, you will see fields for the pattern count and distances relative to the member you have selected. Press **Tab** to move among the dimension and count fields.

- Icons next to the pattern's dimensions and counts show which will be changed if you change the value of the highlighted field. The closed lock indicates that a dimension won't change and the open lock shows you it will change. You can override a lock on a dimension by clicking its icon to toggle it between locked and unlocked.
- Dimension or count line placement, arrows, and value fields show you how the pattern will change if you change the value of a field.
- Your selection determines how changes to the distance and count affect the pattern's position. If you select one member of the pattern, the change is centered on that member. If you select all pattern members, the change is centered on the entire pattern, with the distance between members locked by default. You can control which direction the pattern will grow based on which member you select and which count or distance you change.
- Changing a pattern count in one direction does not change the distance between pattern members. Instead, the overall distance of the pattern will change. This is indicated by the closed lock icon.
- Circular patterns are anchored at the opposite end by default when you move one end.

With a pattern node selected in the model tree, you can:

- Change fill pattern parameters in the **Layout** section of the **Properties** panel. Changes that you make automatically update the pattern in the design window.
- To update lightweight fill patterns to fit the existing geometry of your design, select the lightweight pattern node, then right-click and select **Update Pattern**.

Note: Patterns of sketch entities and 3D curves are no longer patterns when they change into another form, such as a surface or solid. For example, if you pattern a C-shaped curve, then it will remain a pattern when you switch from sketch mode to 3D mode. But if you close off the curve to make a box before you switch to 3D mode, then the boxes will become surfaces and will no longer be a pattern.

This section contains the following topics:

5.4.3.6.1. Creating a Pattern

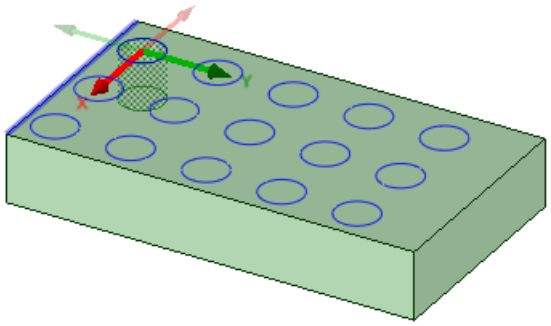
This section contains the following topics:

5.4.3.6.1.1. Creating a One- or Two-Dimension Linear Pattern

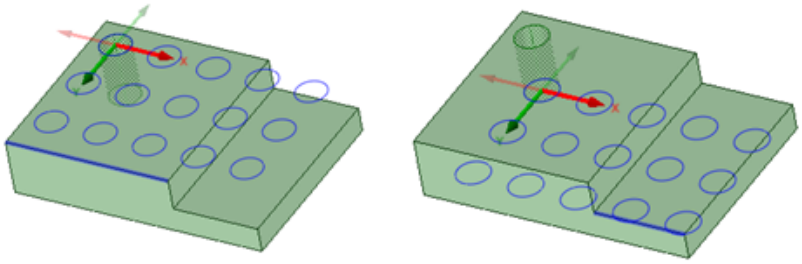
1. Click the **Linear** tool in the **Insert** group on the **Design** tab.
2. Select a protrusion, depression, body, sketch, points, axes, planes, origins, or 3D curves to be the first member (leader) of the pattern.
   The **Direction** tool guide is activated.
3. Select a line, edge, axis, or a set of points to set the direction of the pattern.
4. (Optional) Modify the Pattern options in the Options panel.
   - **Pattern Type**: One-dimensional or Two-dimensional.
   - Change **X Count**, or **X Pitch** values for One-dimensional patterns.
   - Change **X, Y** Count, or **X, Y** Pitch values for Two-dimensional patterns.

   As you switch between pattern types, or change pattern values, the pattern preview displays your changes in blue.
5. Select the **Create Pattern** tool guide, or press **Enter**, to complete the pattern.

The image shows the preview for a two-dimensional pattern. For one-dimensional, only the X-direction is displayed. Clicking the arrows flips their direction 180-degrees.



For through-all cuts, the preview is on the plane closest to the direction reference. For blind cuts, the preview is on the face of the selected pattern leader.



You can also create one-dimensional linear patterns on cylindrical faces as long as the pattern direction is either the cylinders axis or parallel to the axis.

To create a pattern of a pattern, follow the steps for the linear pattern, but select a member of a linear pattern as the first member of the pattern.

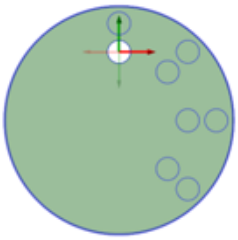
5.4.3.6.1.2. Creating a Circular Pattern

1. Click the **Circular** tool in the **Insert** group on the **Design** tab.
2. Select a protrusion, depression, body, sketch, points, axes, planes, origins, or 3D curves to be the first member (leader) of the pattern.
   The **Direction** tool guide is activated.
3. Select a line, edge, axis, or a set of points to set the direction of the pattern.
4. (Optional) Modify the Pattern options in the Options panel.
   - **Pattern Type**: One-dimensional or Two-dimensional.
   - Change **Circular Count**, or **Angle** values for One-dimensional.
   - Change **Circular Count**, **Angle**, **Linear Count**, or **Linear Pitch** values for Two-dimensional.

   > **Note:** Circular Count of one will create a single radial pattern.

   As you switch between pattern types, or change pattern values, the pattern preview displays your changes in blue.
5. Select the **Create Pattern** tool guide, or press **Enter**, to complete the pattern.

The image below shows the preview for a two-dimensional pattern. For one-dimensional, only the Angular direction is displayed. Clicking the arrows flips their direction 180-degrees.



You can create circular patterns on cylindrical faces, both around the face and along the axis.

5.4.3.6.1.3. Creating a Fill Pattern

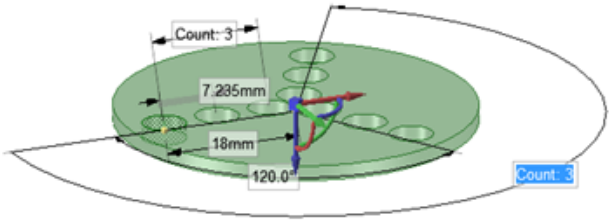
1. Click the **Fill** tool in the **Insert** group on the **Design** tab.

2. Select a protrusion, depression, sketch, points, axes, planes, origins, or 3D curves to be the first member (leader) of the pattern.
   The **Direction** tool guide is activated.
3. Select a line, edge, axis, or a set of points to set the direction of the pattern.
4. Modify the options in the Options panel:
   - Pattern **Type**: Grid or Offset
   - Modify the **X Spacing** or **Y Spacing**
   - Modify the **Margin** values. A dashed orange boundary line displays the pattern's marginal boundary.

   As you switch between pattern types, or change pattern values, the pattern preview displays your changes in blue.
5. Select the **Create Pattern** tool guide, or press **Enter**, to complete the pattern.

5.4.3.6.1.4. Creating a Radial Circular Pattern

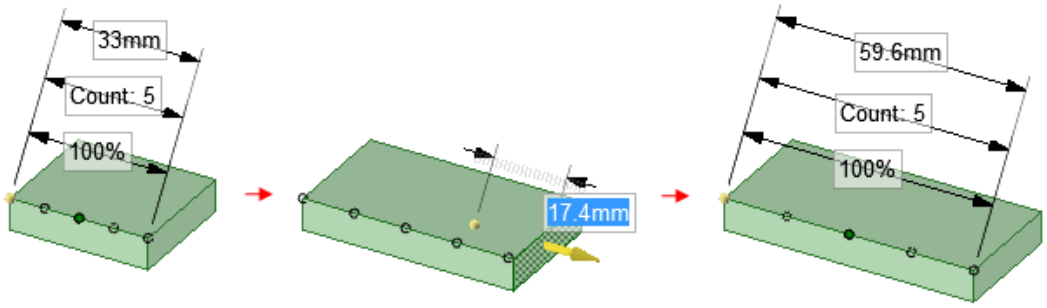
1. Select all radial pattern members.
2. Click the **Move** tool.
3. Re-anchor the Move tool on the circular axis.
4. Select the **Create patterns** check box in the **Options** panel.
5. Drag the radial pattern to form a circular pattern.



5.4.3.6.1.5. Create a Pattern of Points Along an Edge

1. Select a point to be the first member of the pattern.
2. Click the **Move** tool.
3. Select the **Create patterns** check box in the **Options** panel.
4. Click the **Move Along Trajectory** tool guide.
5. Click one of the edges that connects to the vertex.
6. Click the Move handle axis.
7. Drag to create the end pattern member and create the pattern.
8. Press **Tab** to change the count, length, and percent fields to edit the pattern of points.

All points are associated with the edge, so that when the edge changes, the points also change as shown.



5.4.3.6.1.6. Editing a Pattern's Properties

1. Select one pattern member to display the pattern count and dimensions.
2. Edit the pattern's properties.

   Press **Tab** to switch between the fields.

   Click the lock icon to lock or unlock a value.

   > **Note:** By default, **Pattern Length** is locked.

3. Press **Enter**.

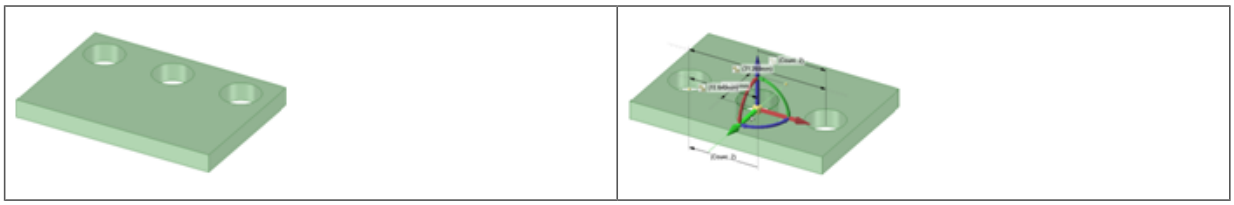
   The result of editing the count and spacing is relative to the member of the pattern you select.

   > **Note:** To quickly increase or decrease the number of pattern members, press **Tab** until the count field is highlighted, then hold the **Alt** key while you press the up or down arrow.

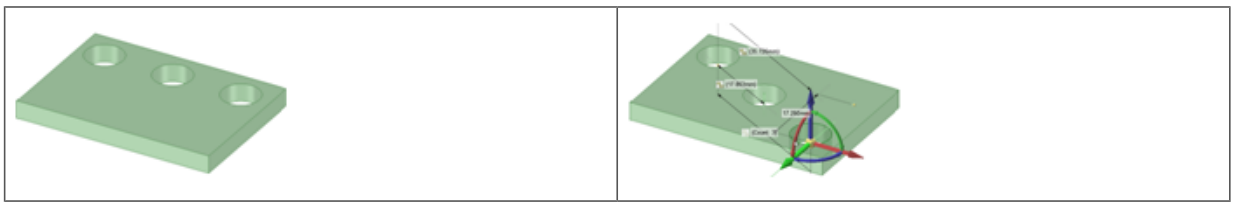
5.4.3.6.1.7. Moving a Pattern

1. Click the **Move** tool.
2. Select a pattern member to move a linear pattern or the pattern axis to move a circular or arc pattern.
3. Move the pattern member with the Move handle.

   If you move an interior member of a pattern and it is not anchored, all the pattern members move:



   If you move a member at one end of a pattern, the member at the opposite end is anchored and the pattern is skewed:

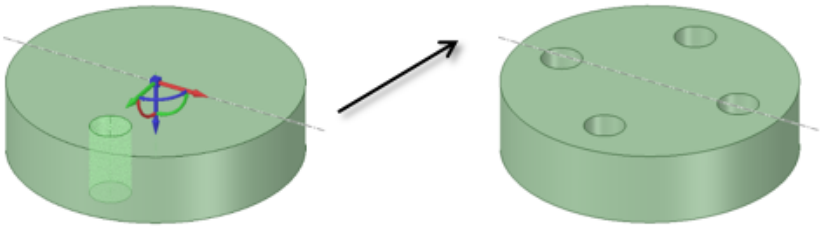


Note:

- If you anchor a different member than the member opposite the direction you are moving, **Move** skews the pattern.
- If you have a linear pattern in a radial direction and you move an interior member without setting an anchor, then the entire pattern shifts in the selected direction
- Use the **Up To** tool guide to move a pattern member up to another face or edge. Pattern dimensions (such as Count and Length) display as expected.
- You can also use the **Up To** tool guide in the Move tool to create a circular pattern by rotating up to a linear entity passing through the Move Handle origin.

5.4.3.6.1.8. Creating a Circular Pattern Using the Up To Tool

1. Relocate the Move handle to the axis of the cylinder.
2. Choose the rotation handle about the cylinder's axis.
3. Click **Up To** and select the horizontal axis to create the pattern.



5.4.3.6.1.9. Moving a Radial Pattern in a Linear Direction

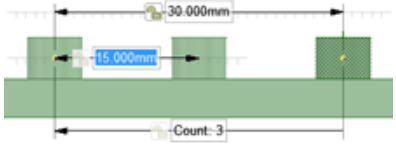
1. Click the **Move** tool.
2. Right-click a pattern member and click **Select** > **All Pattern Members**.
3. Select the **Direction** tool guide.
4. Click an object to set the direction of the move.
5. Drag the pattern.

5.4.3.6.1.10. Adjusting Pattern Distance or Spacing

1. Click **Select** or **Move** in the **Edit** group on the **Design** tab.
2. Select a pattern member.

This member will serve as the anchor, and other pattern members will move relative to this member.

3. Press **Tab** until the field you want to change is highlighted:
   - **Distance**: This field is the overall distance of the pattern. The pattern member you select is anchored, and the length will change relative to this member. The arrows indicate the direction of change. You can see this field at the top of the image below.
   - **Spacing**: This field is the spacing between pattern members. This field has arrows in both directions and is located between two pattern members. The field is highlighted in blue in the image below.
   - **Radial patterns**: If you select a member of a radial pattern, you can change the angle between pattern members and the distance from the pattern members to the center of the pattern.
4. Type a new distance or angle and press **Enter**.
   The spacing between all pattern members will change.



5.4.3.6.1.11. Removing a Pattern Member from the Pattern

- Right-click the face of the pattern member and select **Unpattern Member**. Doing this makes the member independent, so changes to the feature won't propagate to the pattern.

5.4.3.6.1.12. Creating a Pattern Group

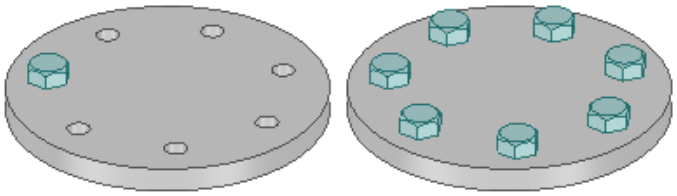
1. Select a pattern or pattern member in the Design window.
2. Click the **Groups** tab in the **Structure** panel.
3. Click **Create NS**.
   A group is created that displays the number of members in the pattern. You can click this group to select the pattern and change its parameters in the Design window.

5.4.3.6.1.13. Assembling Components on a Pattern

You can assemble a component with a pattern member, then propagate the component to all of the pattern members. The component will be copied and assembled to each pattern member.

1. Assemble a component to some geometry that is a pattern member using the **Align**, **Tangent**, or **Orient** tools.
2. Right-click the Assembly Condition in the **Design** tree and select **Update Components on Pattern**.

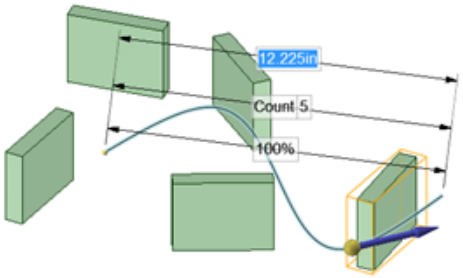
   The components will be propagated to the pattern, as shown.



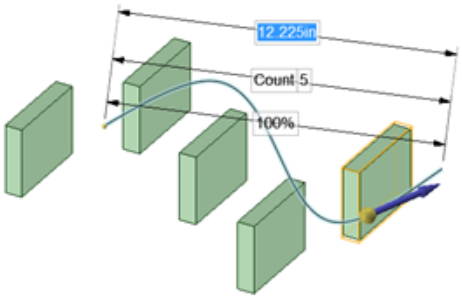
5.4.3.6.1.14. Creating a Pattern Along a Trajectory

1. Click the **Move** tool.
2. Select the object you want to pattern.
3. Click the **Move Along Trajectory** tool guide and select the curve to use as a trajectory.
   You can also hold **Alt** and double-click the curve.
4. Select **Create Patterns** in the Move options.
5. Pull the trajectory arrow.
6. Release the mouse button to display pattern dimensioning options, then enter values by tabbing through the options.
7. (Optional) Check **Maintain Orientation** in the Move options to keep the patterned objects in the same orientation as the original object.

Pattern along a trajectory without maintaining orientation.

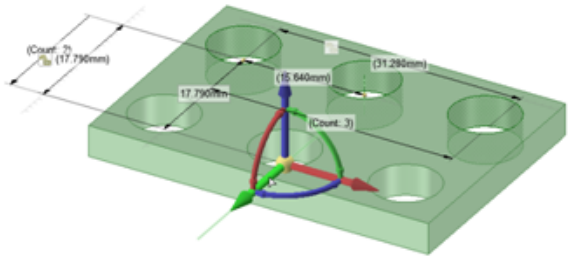


Pattern along a trajectory with orientation maintained.

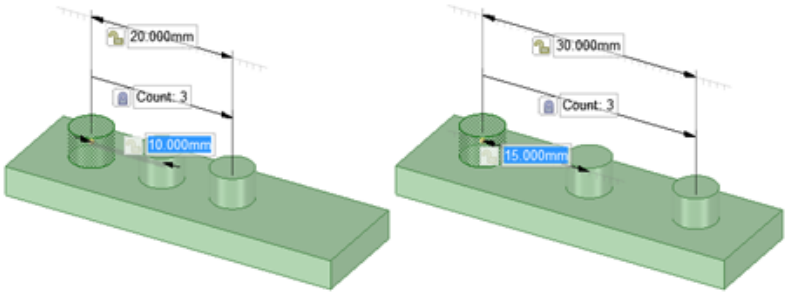


5.4.3.6.2. Creating Patterns Examples

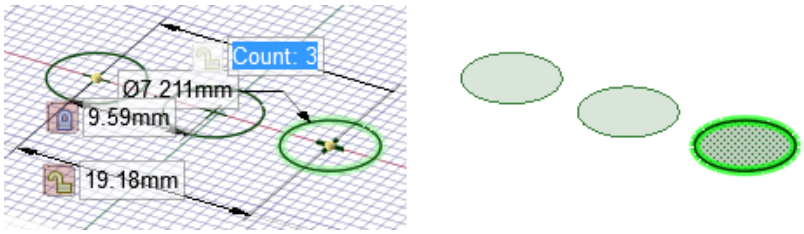
Rectangular pattern.



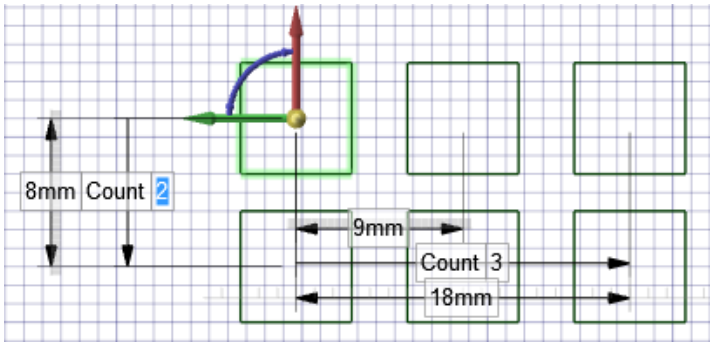
Changing spacing between pattern members changes the pattern's overall length when the length field isn't locked.



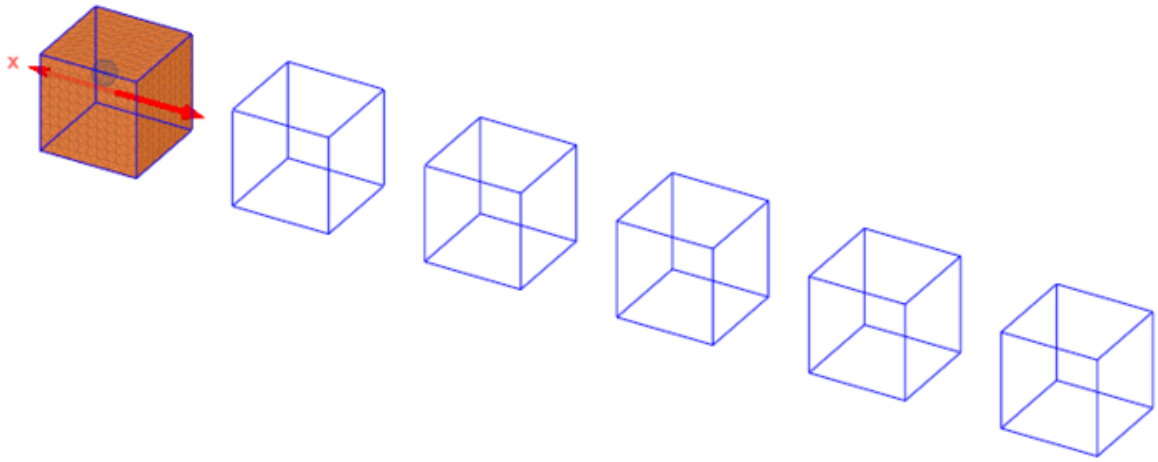
A pattern of sketch objects is no longer a pattern when you switch to 3D mode and the sketched objects are converted to surfaces.



Sketch curve patterns can propagate in two dimensions. The pattern above contains three squares in the X direction and two squares in the Y direction.



Faceted bodies can be patterned after selecting the body to pattern and direction.



5.4.3.7. Pivoting and Pulling Solids

You can use the **Move** tool to pivot or pull a solid.

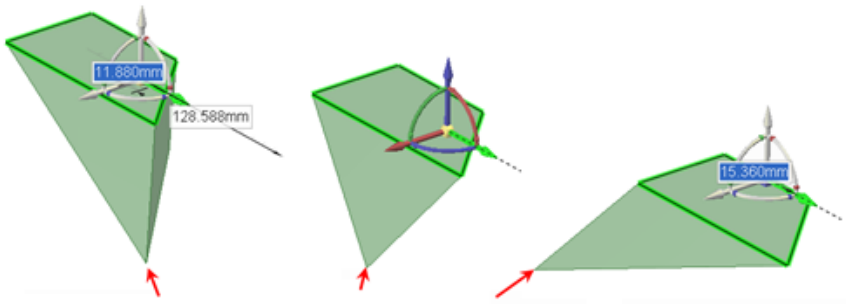
This section contains the following topics:

5.4.3.7.1. Pulling a Solid

1. Click the Move tool.
2. Select the face you want to move.
3. Use the Move handle to move the face.
   The solid is extended in the direction of the move

5.4.3.7.2. Pivoting a Solid

1. Select an edge loop on the solid that you want to pivot.
2. Click a linear axis of the Move handle.
3. (Optional) Click the Fulcrum tool guide and click an alternate plane to pivot around.
4. Drag to pivot the solid around a plane drawn through a point opposite the selected edge loop or to pivot the solid around the plane you selected with the Fulcrum tool guide.
   If the Move tool cannot maintain a planar or cylindrical face while pivoting, it will create a blended face.



5.4.3.8. Moving a Solid or Surface with the Select Tool

You can move solids and surfaces with the **Select** tool.

You can do any of the following:

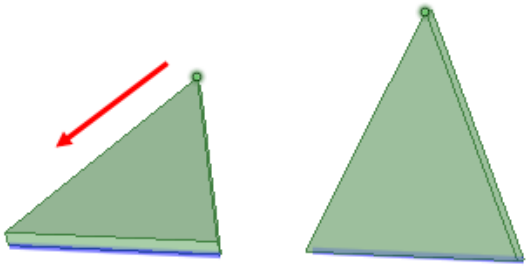
- Select a vertex of a solid or surface, then drag it to move the solid or surface. (Surfaces on the same plane are treated as a single surface and move together.)
- Alt+click a point, then drag the vertex to rotate in the plane of the screen.
- Alt+click an edge or two points, then drag the vertex to rotate the object around the axis defined by the line or points.
- You can place the dragged vertex on a plane or edge by dragging until the plane or edge is highlighted.

You cannot drag unfolded sheet metal parts in the Unfolded window by dragging their vertices with the Select tool.

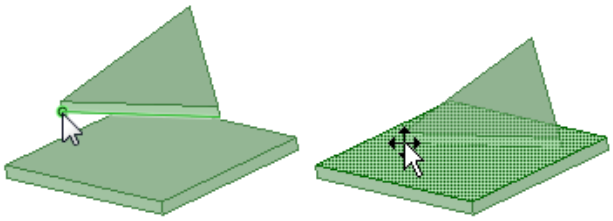
This section contains the following topics:

5.4.3.8.1. Moving Solids/Surfaces with the Select Tool Examples

The blue edge was selected using the Alt key, then the green vertex was dragged in the direction of the arrow. The part pivots around the blue edge.

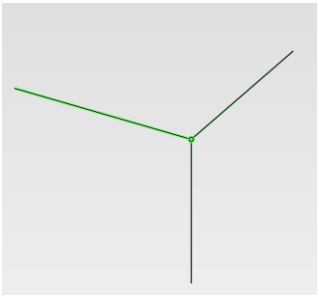


The vertex highlighted in green was dragged down until the face shown on the right was highlighted. The triangular part moves down so the vertex is in the same plane as the face.



When you hover and pre-highlight a vertex, the curve to which it belongs is highlighted. Using the mouse wheel, you can scroll through other curves attached to the vertex.

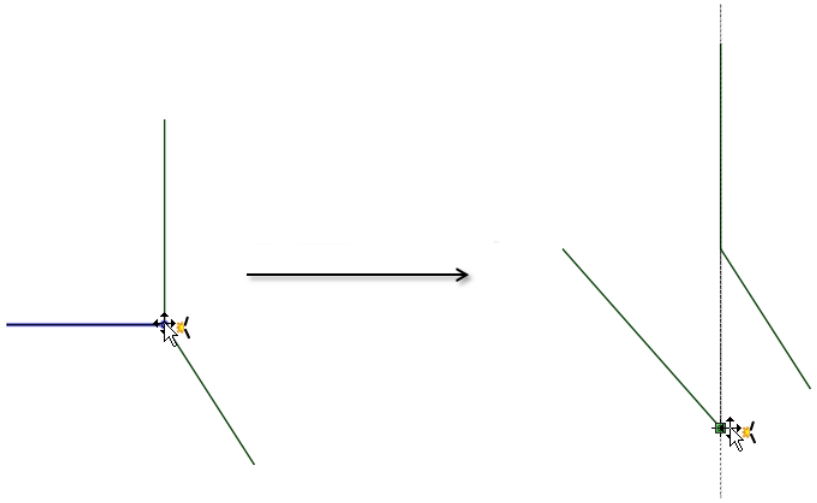
The cursor changes to the Move cursor when the mouse is over a vertex.



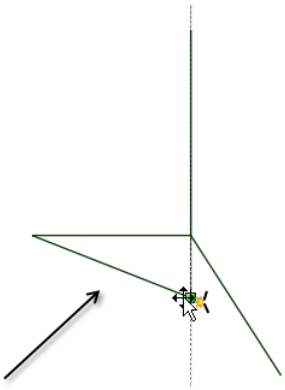
When the curve you want is highlighted, you can Alt-drag to move only the vertex of that curve.

Pressing the Alt key changes the Move cursor to the Detach-and-Move cursor.

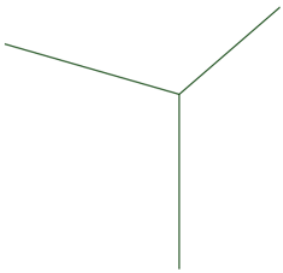
You do not need to hold the Alt key throughout dragging. You can release it once you start dragging.



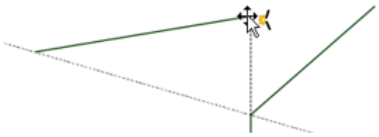
Drag a copy of the curve using Ctrl+Alt+drag.



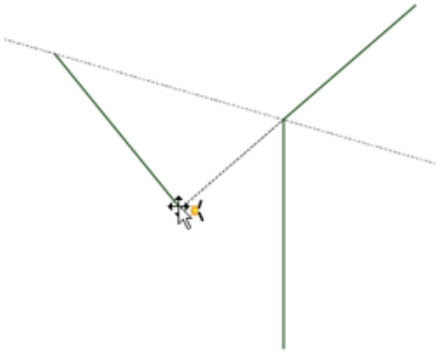
In the images above, notice the "source-generated dotted lines" extending from the other curve. This is similar to the behavior in Sketch Mode for aligning and snapping the vertex. By default, the system generates them extending from all curves that intersect at the selected-and-moving vertex.



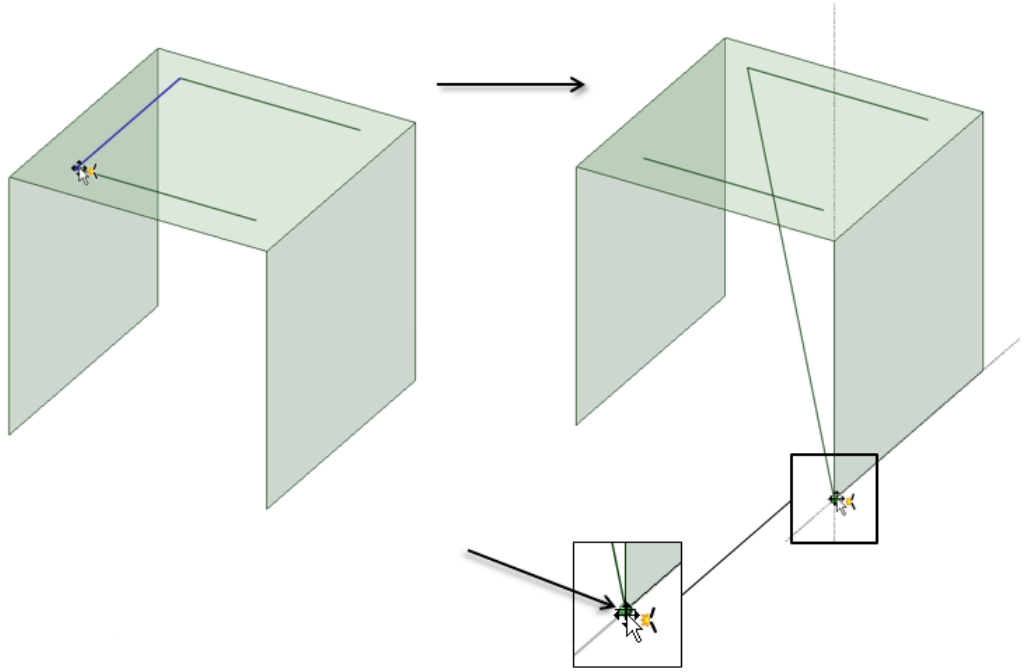
Drag the vertex vertically. The extension line for the affected curve is shown as well as the extension for the vertical curve.



Drag the vertex away from the vertical curve. The vertical extension line disappears, the affected curve extension stays and the extension for the third curve appears.

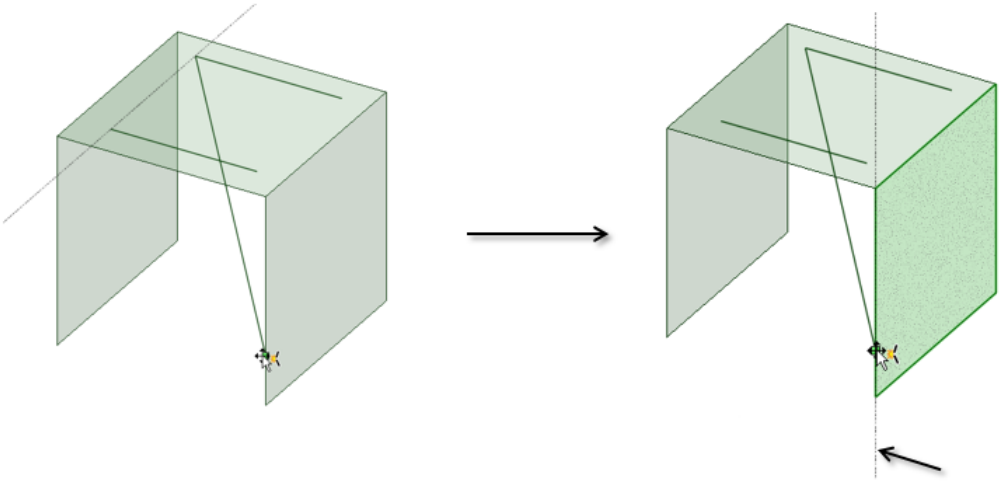


Sketch only allows snapping in the plane, but with Curve Dragging, you can snap in 3D.

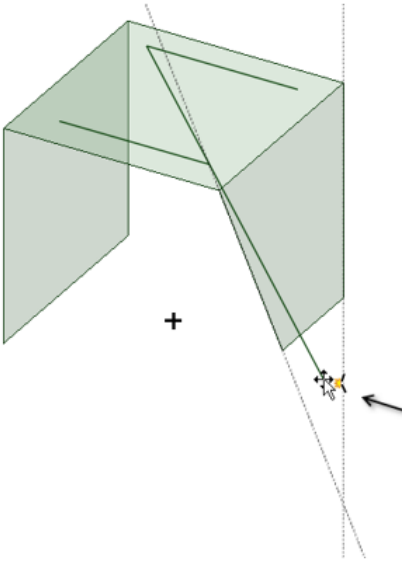


When you hover over any straight line or edge while dragging a vertex, a dotted vector will be generated. This allows you to snap to the extension of any existing line or edge.

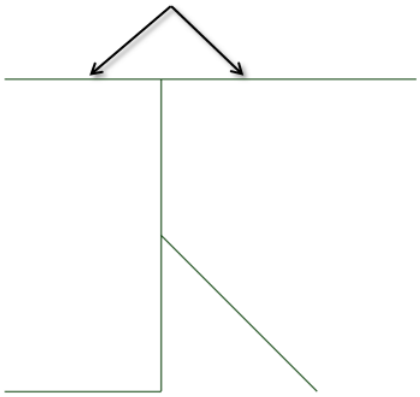
Since you can drag a vertex without holding down the Alt or Ctrl keys, you can Shift-hover over other objects. This displays extension lines for other lines or edges. The default, source-generated dotted lines are erased when you Shift-hover over an object.



Sequentially Shift-hovering on converging lines or edges lets you snap to their virtual intersection.

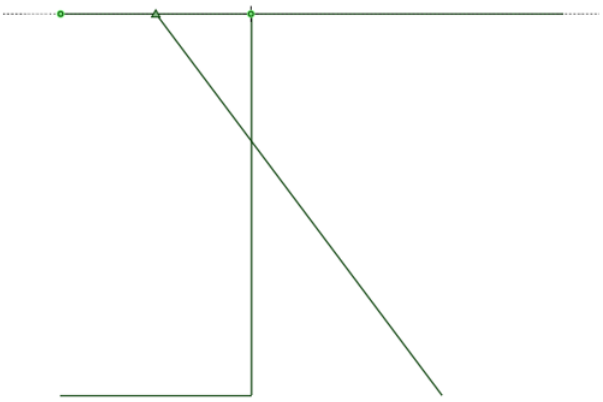


In the example below, the top horizontal line is a single curve, but with two "sub-curves" because the vertical curve's endpoint lies somewhere along its length. Although there is only one horizontal curve, it can be treated as split, for the purposes of dragging other vertices to these sub-curve endpoints or midpoints.

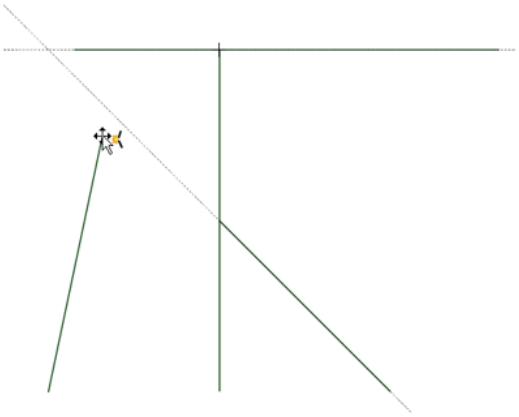


Sub-curve midpoints are available for snapping if you Shift-hover over the parent curve.

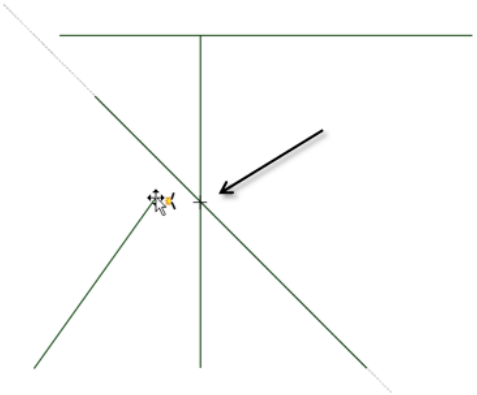
Green points display at the endpoints of the sub-curve and a triangle displays at the midpoint. This helps to determine what portion of the curve is being snapped to, in cases where it may be ambiguous.



The virtual intersection of the horizontal and slanted lines is available for snapping if you Shift-hover over both.

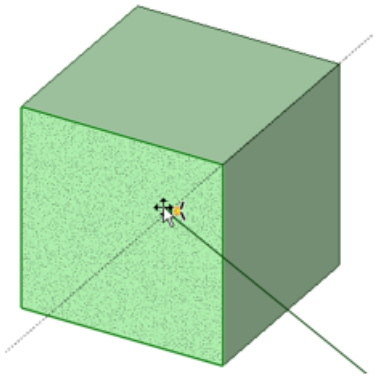


Curve intersections (for both straight and arc-shaped curves) are available for snapping and displayed with a "+" sign.

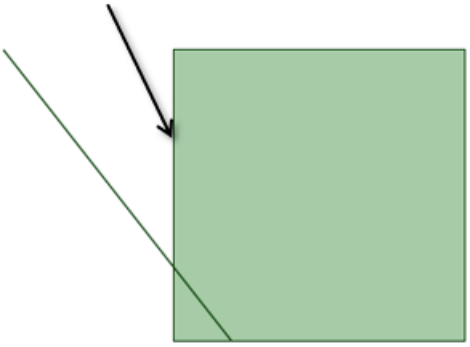


Face selection does not override edge extension snapping. In this example, the extension line is above the face. If the face had a higher priority, you could not snap to the extension line without reorienting the view.

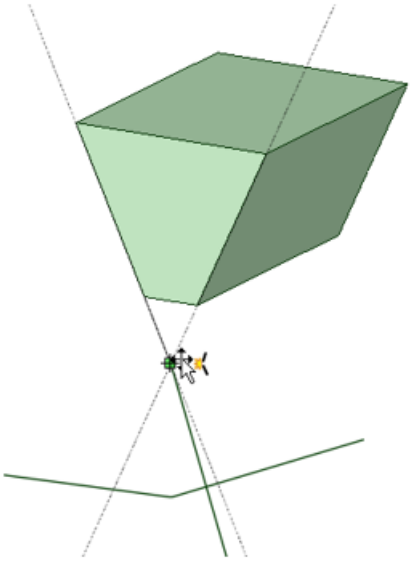
Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.



You can snap a vertex to a 3D point.



You can also snap a vertex to the virtual intersections of extended solid edges.



5.4.3.9. Moving with the Fulcrum Tool Guide

The **Fulcrum** tool guide fixes a point, edge, or face in 3D space. Then selected objects are moved relative to the fulcrum object. You can use this tool guide to pivot patterns or faces, or to create a simple exploded assembly.

1. Click **Move** in the **Edit** group.

2. Select the objects you want to move. You can select surfaces, solids, or components in the **Design** tree, or select point(s), edge(s), or face(s) in the design window.

   This object will be highlighted. Selected components will be highlighted with a bounding box.

3. Click the **Fulcrum** tool guide on the side of the Design window.

4. Select the object you want to use as the fulcrum. You can select surfaces, solids, or components in the model tree, or a point, edge, or face in the design window.

   This object will be highlighted in blue.

5. Drag an arrow or radius on the Move handle to move the object(s).

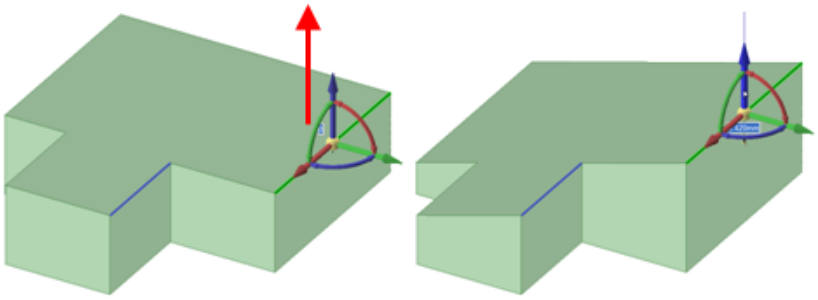
   Selected objects will move proportionally, relative to the fulcrum.

This section contains the following topics:

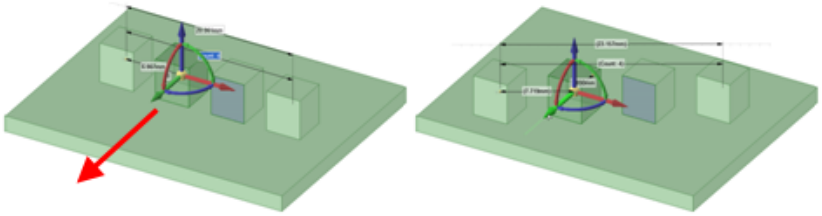
5.4.3.9.1. Moving with the Fulcrum Tool Examples

In the examples below, the object was dragged in the direction indicated by the red arrow.

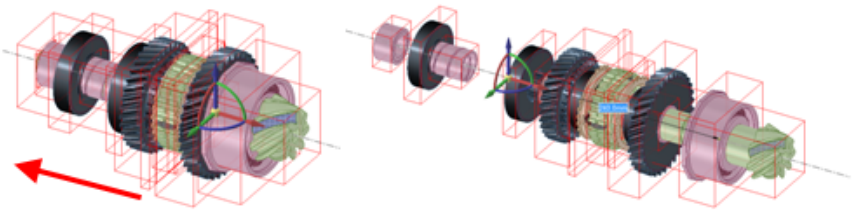
Moving an edge using another edge as the fulcrum point.



Moving a pattern with one pattern member as the fulcrum point. You must select all the faces on the pattern member that you move.



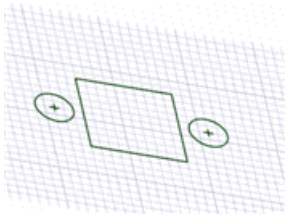
Creating a simple exploded view by using a face of one component as the fulcrum point. Select the components in the model tree that are part of the assembly you want to explode.



5.4.3.10. Move Grid Tool

Use the **Move Grid** tool to move the sketch grid. Make successive sketches by moving the grid after sketching closed line regions. These closed lines turn into regions when you move the grid.

The icon at the center of the sketch grid indicates the origin of the grid and moves with the grid as you move the grid.



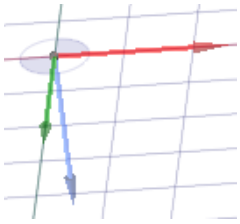
5.4.3.10.1. Moving the Sketch Grid

1. Click the **Move Grid** tool on the mini-toolbar or click the Move tool and check the **Move Grid** option.
2. (Optional) Select any sketch entities that you want to move along with the sketch grid.
3. Select a Move handle axis.
4. Drag along the axis of the Move handle to move or rotate the sketch grid.

   You can also right-click and select **Use Ruler Dimension**, and enter a value or press **Enter** to drag the grid. You can also use the standard Move tool guides when moving the grid.

   The Move handle can be moved around by dragging the center ball onto appropriate entities on the sketch grid.

5.4.3.10.2. Moving the Center of the Sketch Grid



1. Insert an origin.
2. Select an axis of the origin.
3. Switch to Sketch mode.

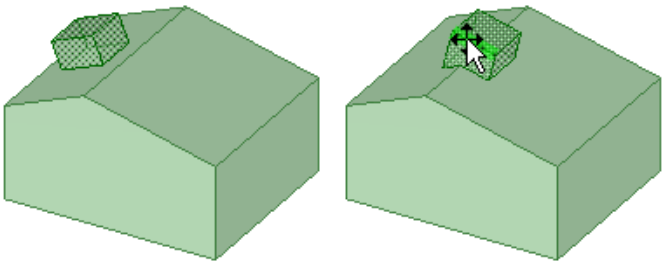
   You can also move the center of the sketch grid while sketching by using the Move Grid or Select New Sketch Plane tools in the Sketch mini-toolbar.

5.4.3.11. Moving Protrusions and Depressions

You can move protrusions and depressions with the **Select** tool or the Move tool.

1. Select a protrusion or depression.
2. Drag the protrusion or depression.

If two faces connect at an angle greater than 90 degrees, you can drag the protrusion across the two faces, as shown in the image below.



5.4.3.12. Parameterizing Measurements Through Move

1. On the **Design** tab, in the **Edit** group, click **Move**.
2. Select an object and a direction using the **Move** handle.
3. On the right side of the **Move** tool HUD, select **Add a geometric parameter**.

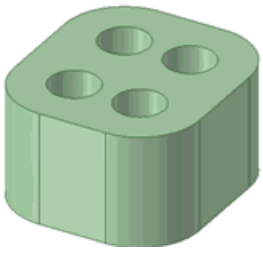
   You are now in the **Move – Parameterization** tool.
4. Select a second object to measure, or multiple objects to measure between. If insufficient selections are made, the Discovery application might generate a datum line or point to act as a move anchor or for measurement purposes.This datum line/point will not be visible in the design tree and will only appear when the parameter is selected. The datum lines/points should become visible, selectable, and moveable until the parameter is deselected. If the datum lines/points are moved, the parameter value may change accordingly.
5. On the right side of the **Move – Parameterization** tool HUD, select the property to measure and parameterize. For example, some properties include **Area**, **Length**, **Mass**, **Volume**, but many more exist and depend on your geometry and selection.
6. Click **Create parameter** to add the measurement to the defined variations table for parameterization.

   This exits the **Move – Parameterization** tool and returns you to the **Move** tool.
7. In the SID, click the **Defined parameters** hex to view the parameter in the defined variations table.

5.4.4. Fill Tool

Use the **Fill** tool to fill in the selected region with the surrounding surface or solid. **Fill** can "heal" many cuts made into geometry, such as chamfers and rounds, subtractive revolves, protrusions, depressions, and regions removed by removing regions in the Combine tool. When using Fill to fill a gusset, the bend geometry on which the gusset is defined remains intact.

The **Fill** tool can also be used to simplify surface edges and cap surfaces to form solids. You can select a combination of faces and edges to replace them with a single new face.



You can use the **Fill** tool in Sketch mode to fill a loop of sketch curves that is almost closed, but that has multiple small gaps. If the gaps are too large, multiple error messages appear to show you where the gaps are. You can also use it to concatenate multiple sketched curves.

You can also use the **Fill** tool when editing a layout. Fill functionality is useful when you sketch faces across section lines, but do not want the section lines to split the surfaces when you switch to 3D.

When you change the geometry of a model, for example, the model's length, height, or width, the model's fill pattern will automatically update to correctly re-fill to the model's new geometry.

This section contains the following topics:

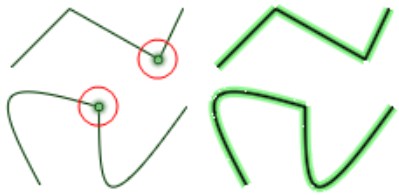
5.4.4.1. Using the Fill Tool

This section contains the following topics:

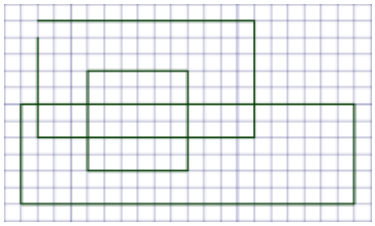
You can also use the **Delete** key to fill faces on a solid or surface.

5.4.4.1.1. Sketch and Layout Curves

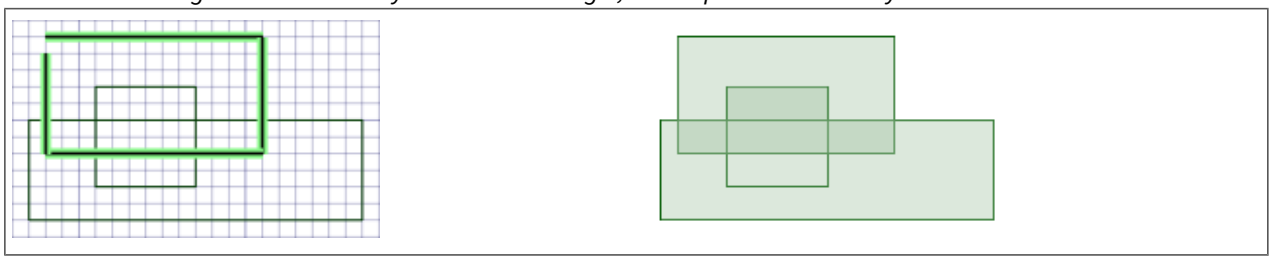
If you select one or more end points of sketch curves, the curves are concatenated into a single spline. Neighboring sketch curves are merged into one selectable item with an underlying spline. The spline is not displayed until the concatenated curve is modified:



Let's look at the following sketch curves:

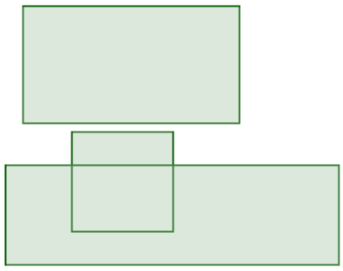


One of the rectangles isn't closed. If you fill this rectangle, the loop is automatically closed and it becomes a surface:



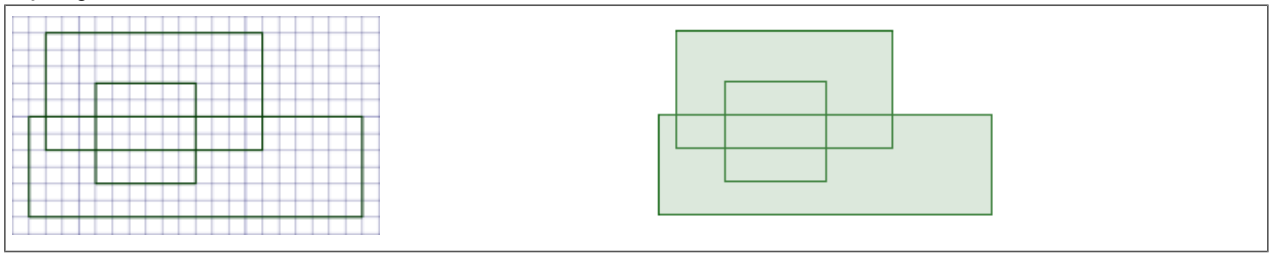
The gap in this loop is small enough for the Fill tool to automatically close. Larger gaps will not be closed. If a gap is less than 1.5 times the length of the minor grid spacing on the sketch grid, the edges are extended to close the gap. If the gap is larger, a message appears in the status bar and the gap's endpoints flash.

The Fill tool also created surfaces from the other closed loops of sketch curves. The darker shaded areas in the image above show where the surface parts overlap. If you move the rectangle, you can see the surface that was created automatically:



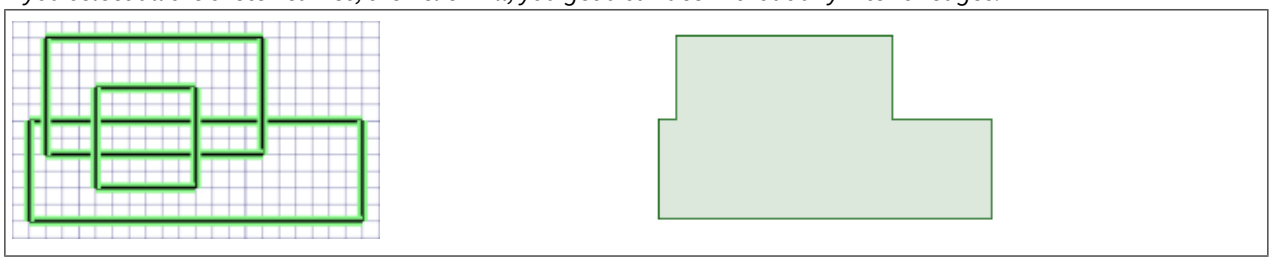
The result only has edges for the sketch curves that were *not* selected, because the selected curves were used to create a separate surface.

Now let's go back and close the open rectangle, and let the Fill tool automatically create surfaces without selecting any edges:

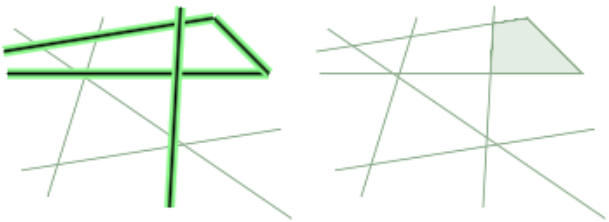


This surface has edges for every closed sketch curve. The same thing automatically happens if you go from sketch mode to 3D mode.

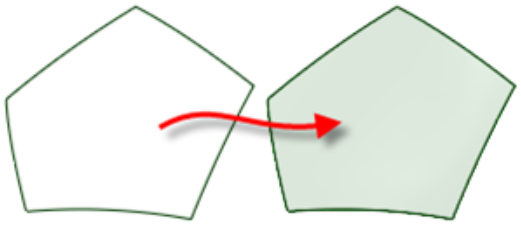
If you select all the sketch curves, then click Fill, you get a surface without any interior edges:



You can also fill layout and sketch loops:

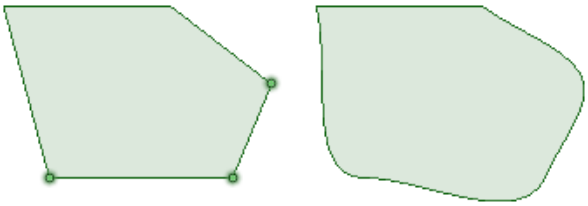


And a loop of non-tangent 3D curves:

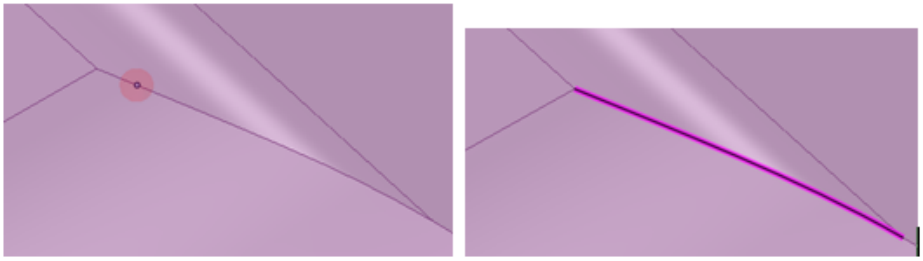


5.4.4.1.2. Vertices

You can select any number of edge points in any order. Each edge point is removed and a spline is created to make a smooth curvature change between the neighboring edges to the point.

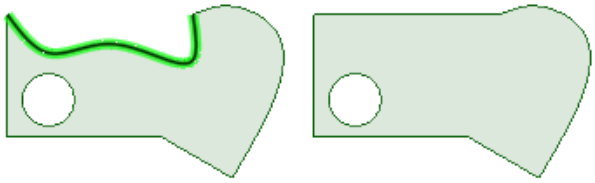


If you Fill a vertex on a solid, the system attempts to merge coincident edges into a single edge.

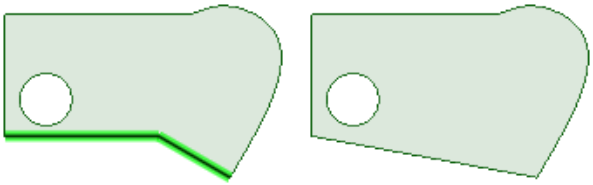


5.4.4.1.3. Edges on Surfaces

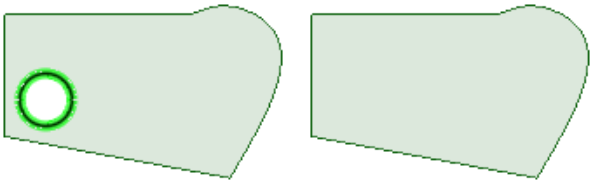
If you select one surface edge in the shape of a spline or arc, the edge is simplified into a straight line:



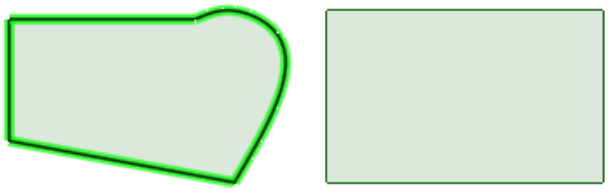
If you select two or more edges of a surface, the edges are simplified with a straight edge between the end points:



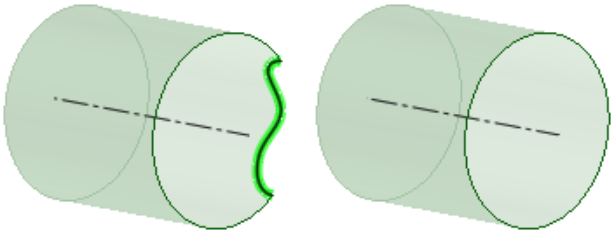
If you select an edge that is completely within a surface, the edge is removed:



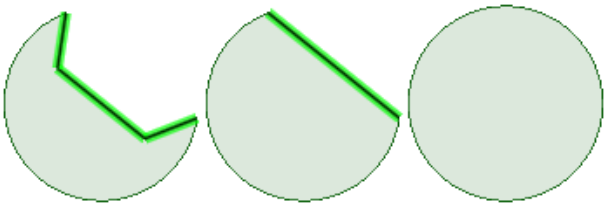
If you select all the edges that enclose a surface, the surface is simplified into a rectangle based on its extents:



If you select a single edge that lies on an analytic surface, Fill will simplify it with the neighboring edges:

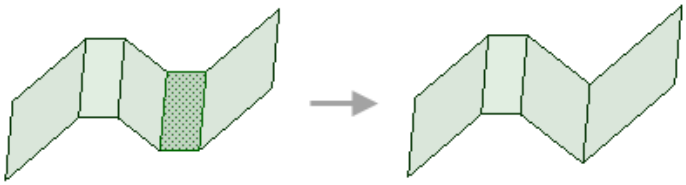


You can fill irregular gaps on a circular surface, and the gap is simplified into a straight edge. Use Fill again on the straight edge, and the edge becomes round:



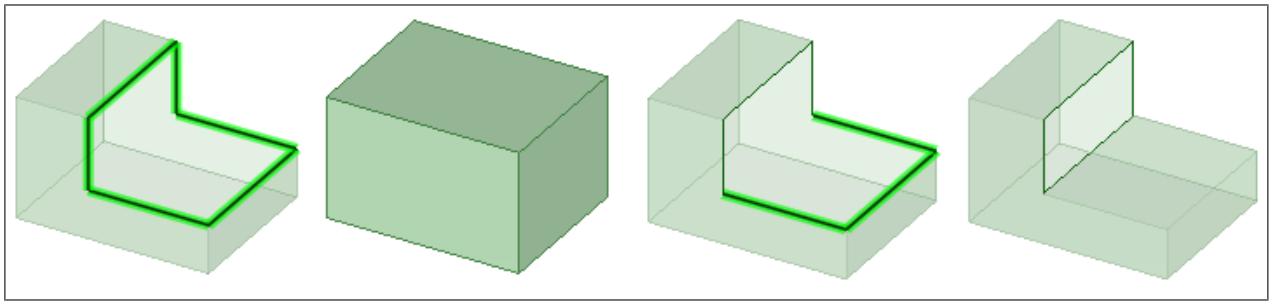
5.4.4.1.4. Chamfers on Surfaces

Select a chamfer on a surface and then use the **Fill** tool to fill the chamfer.

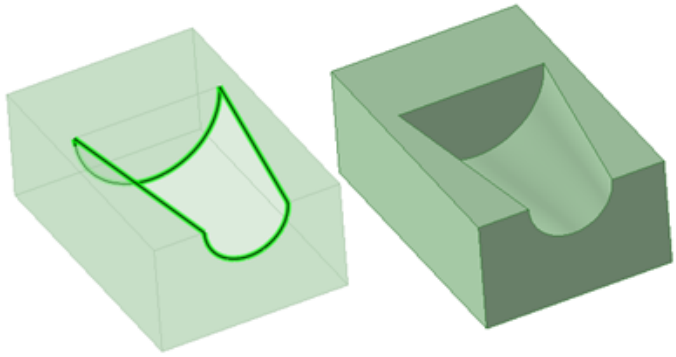


5.4.4.1.5. Edges on More Than One Face

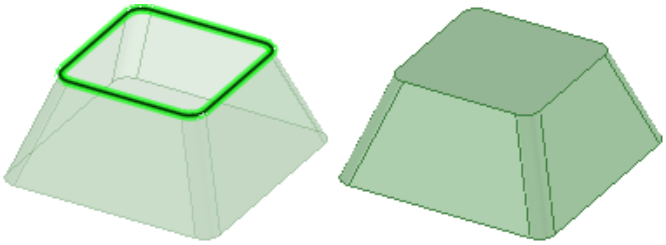
If you select a chain of open, planar surface edges, Fill creates faces based on the edges you select:



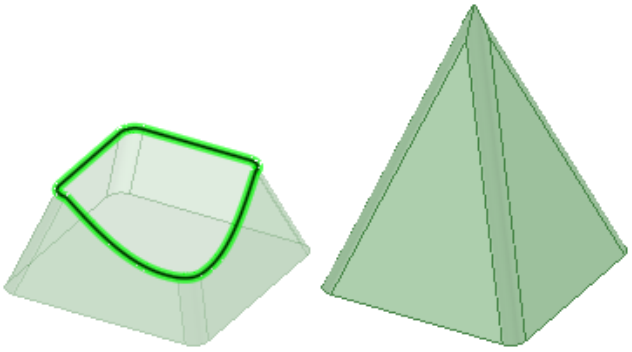
If you select an open edge loop that belongs to multiple faces, Fill attempts to cap it with an analytic surface (cylinder, cone, etc.):



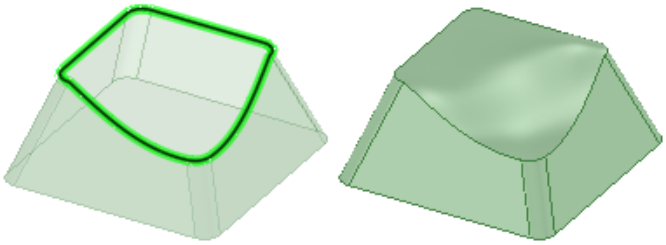
If you select a series of planar edges, you will get a planar face:



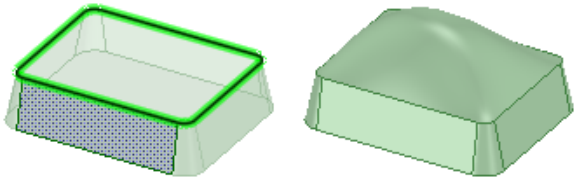
If you select a series of surface edges that are not planar, Fill extends neighboring faces if the Patch Blend option is off:



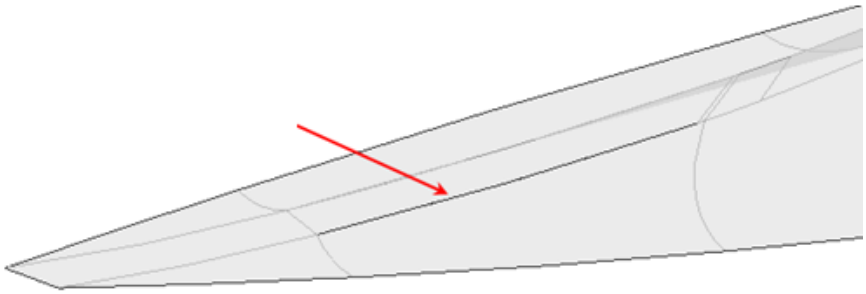
If the Patch Blend option is turned on,



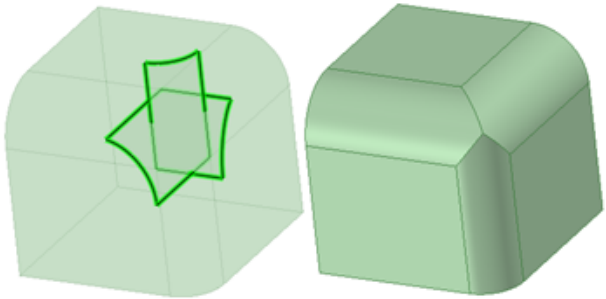
If you select an edge loop and Alt+click to select neighboring faces, the new face will be tangent to any faces you used Alt+click to select:



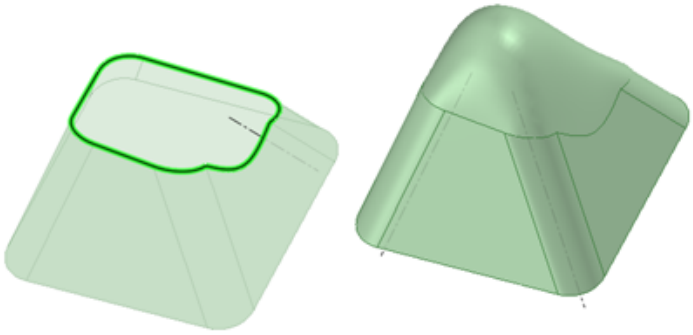
You can also extend neighboring faces to fill sliver gaps (double-click to get loop):



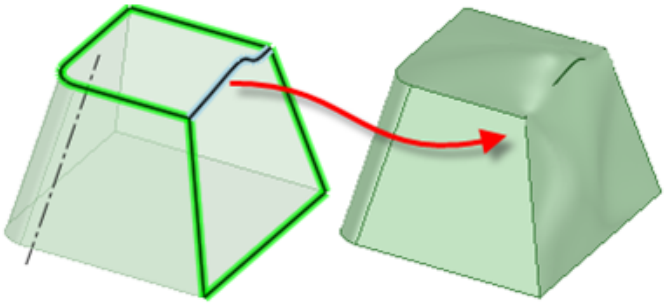
If you select open edges of a self-intersecting surface, Fill tries to form a solid and remove excess (also works in combine):



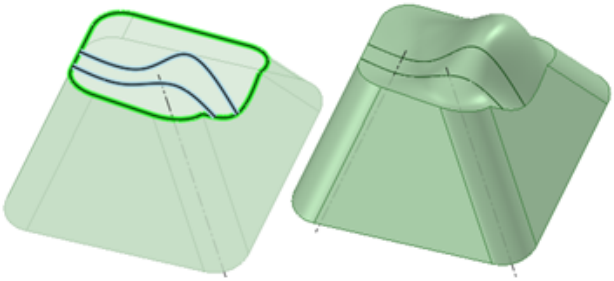
If you select an edge loop and use the Patch blend and Tangent extension options:



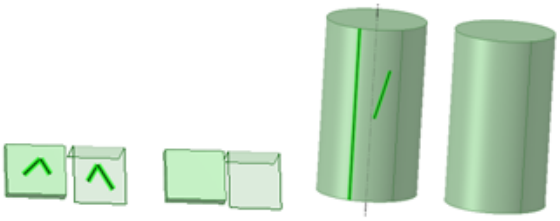
When you fill a loop of edges, you can use Alt and select curves and the new face(s) will pass through the curves:



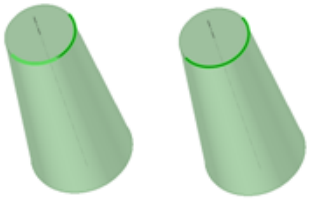
If you select an edge loop and guide curves, with or without the Tangent extension option (which applies to areas not influenced by guide curves):



If you select an imprinted edge on the face of a surface or solid, the imprinted edge is removed. This works the same as delete:

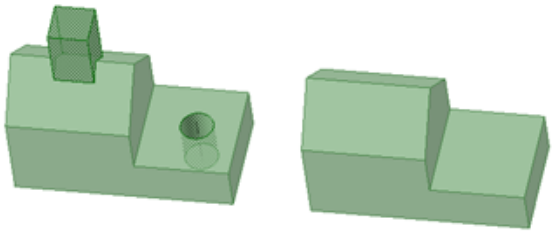


If you select laminar edge(s) of a solid or surface, Fill simplifies the edges by replacing them with a single edge with the same geometry:

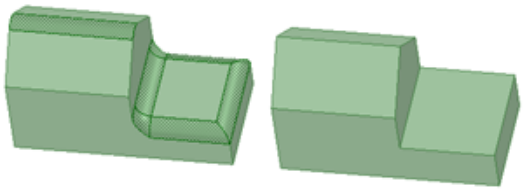


5.4.4.1.6. Solids

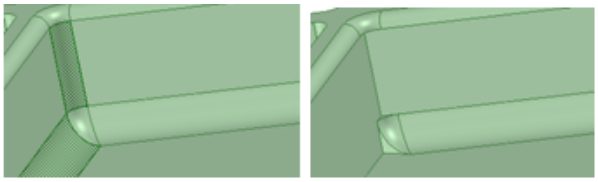
If you select faces, Fill deletes them and extends neighboring faces:



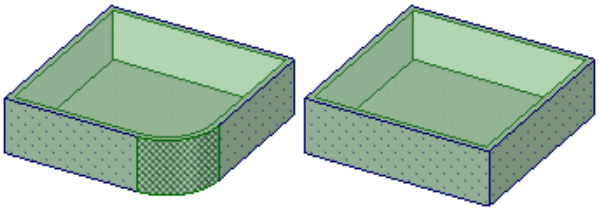
If you select a chamfer or round, Fill removes them and adds them to a named group:



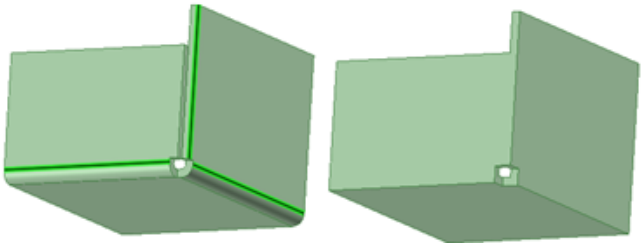
If you select rounds with neighboring rounds, Fill creates planar caps (because rounds should not be extended by definition):



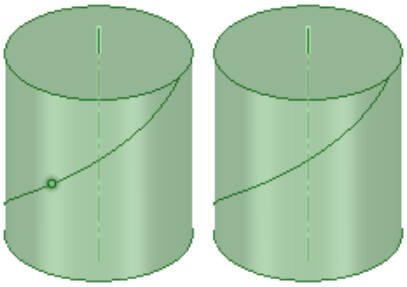
If you select rounds on shelled parts, both faces of the shell are filled:



If you select joint edges created in Sheet Metal, the joints are removed:



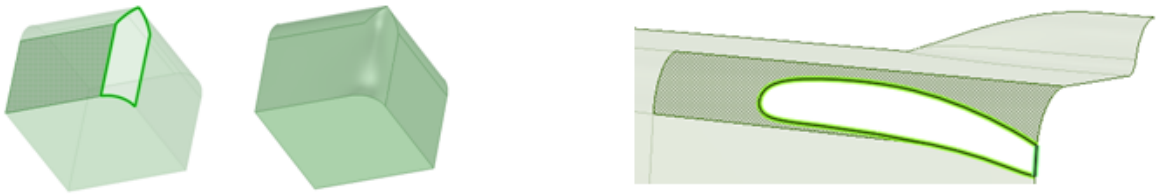
If you select two imprinted edges, as shown below, they are combined into one edge:



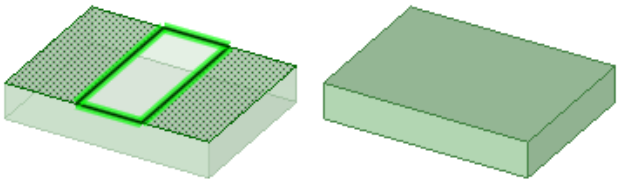
Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

5.4.4.1.7. Faces and Edges or Sketch Curves

With at least one face selected and at least one edge or sketch curve selected, Fill will remove the selected faces and create a single new face using the removed faces and the selected edges as inputs

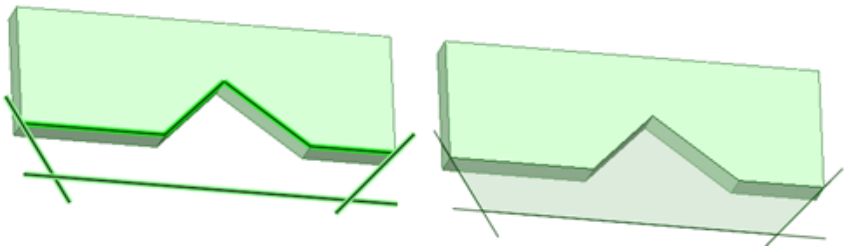


If you select two faces or surfaces that don't touch, you can replace them with a single face. You must double-click to select the gap between the faces.



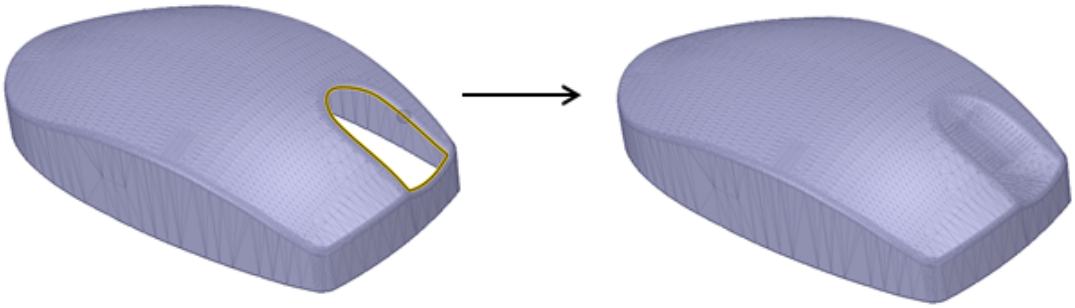
5.4.4.1.8. Any Combination of Sketch Curves, Layout Curves, Surface Edges, and Solid Edges

When you select any combination of sketch curves, layout curves, surface edges, and solid edges that lie in the same plane and form a closed loop, Fill creates a planar surface.



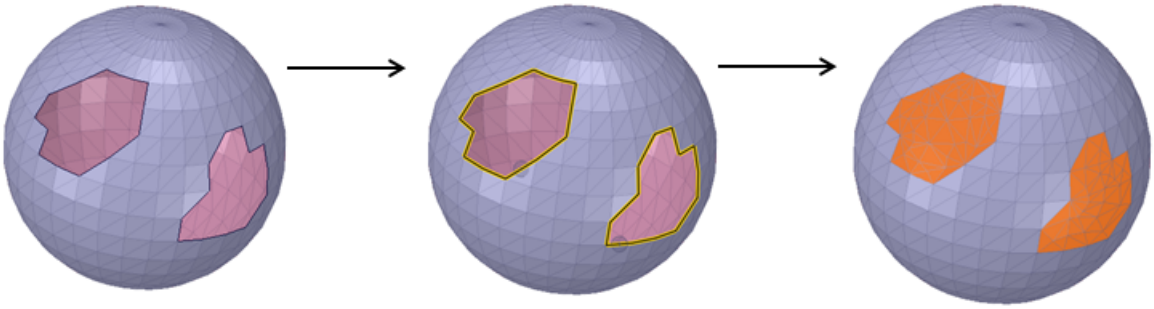
5.4.4.1.9. Meshes

Mesh internal loops and boundary loops can be filled with the **Fill** tool. The loops are filled with facets that consider the curvature of the neighboring facets. This produces a very smooth and uniform fill. The newly created facets are added to the selection set after the fill.

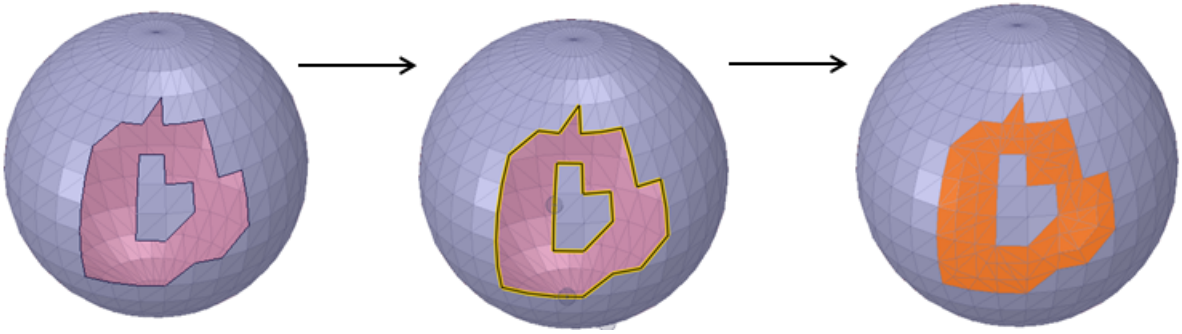


Multiple mesh loops can be filled at once in three cases.

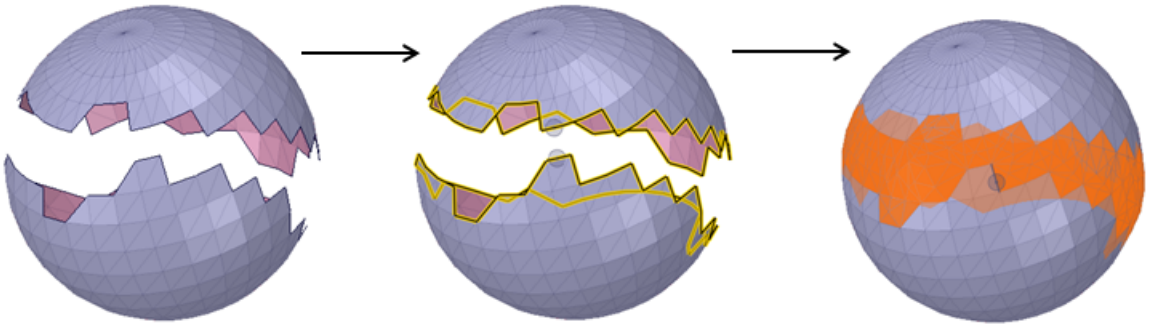
Separate loops on the Same mesh object that are NOT intended to be joined.



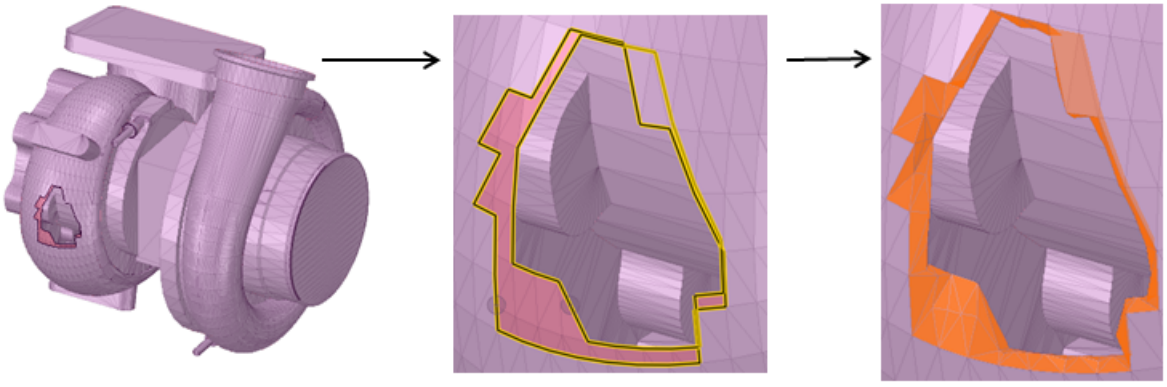
Separate loops on the Same mesh object that ARE intended to be joined



Separate loops on Different mesh objects that ARE intended to be joined



Loops on exterior and interior shells, when filled simultaneously, will create a wall thickness. Fill the interior and exterior loops separately to continue the shell.



You can also use the **Delete** key to fill faces on a solid or surface.

5.4.4.2. Filling a Region

1. Select the edges that define a surface region, or the faces that define a region within or on a solid.
   You can select an object in the model tree to simplify it.
   You can select faces and the **Fill** tool will automatically create a patch if you also select at least one edge.
2. Click the **Fill** tool or press **F**.

5.4.4.3. Filling Sketch or Layout Lines

1. Select a closed or almost closed loop of sketch lines.
2. Click the **Fill** tool or press **F**.
   If a gap is 1.5 times the length of the minor grid spacing on the sketch grid or less, the edges are extended to close the gap. If the gap is larger, a message appears in the status bar and the gap's endpoints flash.
   The mode is switched to 3D mode, and the filled loop becomes a surface.
   You can select the face of a solid when only the edge is displayed (such as in a drawing sheet view) using the scroll wheel. The edge becomes a slightly thicker line when the face is highlighted. If you fill lines in a layout, you can then pull the surface into 3D from the layout, but remain in edit layout mode after this action.
   You can fill lines and edges whether or not the sketched lines you want to fill were sketched in the same plane as the edges. (If the lines are imprinted on a face and become edges, filling those edges deletes them.)

> **Note:** Click the **Fill** tool in Sketch mode to fill any closed or almost closed loops and switch to 3D mode.

5.4.4.4. Fill Tool Guides

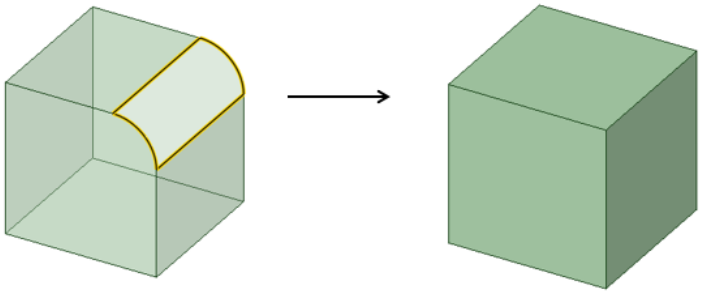
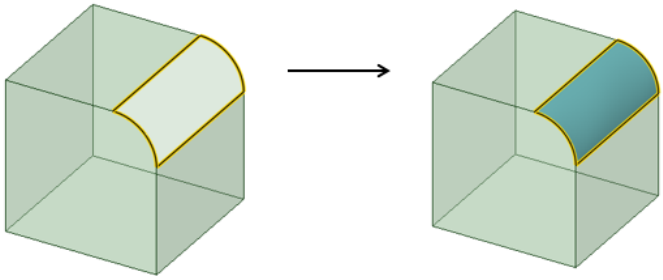
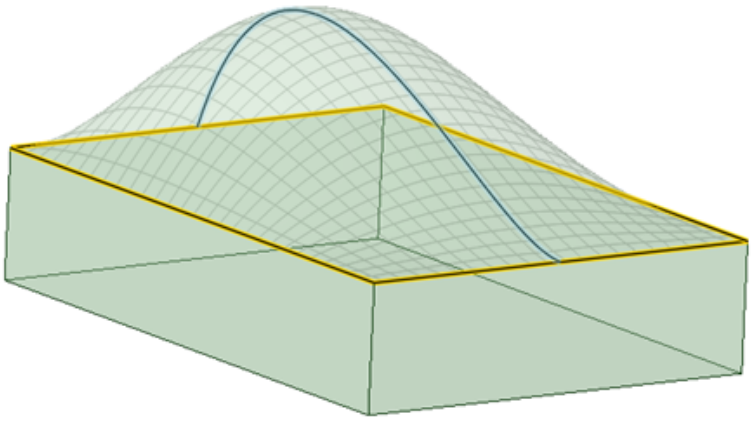
The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select	The **Select** tool guide is active by default. When this tool guide is active, you can select edges and faces to be filled. You can click an edge loop or use box-select to select multiple objects.
Select Guide Curves	The **Select Guide Curves** tool guide allows you to select a guide curve.

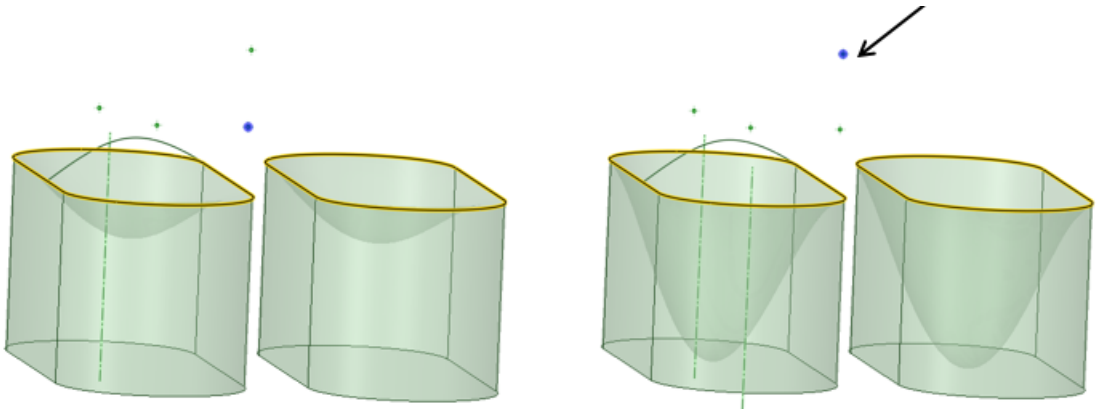
Complete	The **Complete** tool guide generates the filled face.

5.4.4.5. Fill Tool Options

The following options are available in the Options panel:

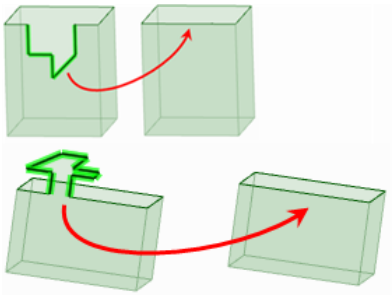
Extend Fill	Fills selected edges by extending the neighboring faces. 
Patch Fill	Fills the selected edges using the initial tangency of neighboring faces to create a smooth patch. 
Tangent to all sides	Available for Patch Fill. Makes the patch tangent when possible to the neighboring faces.
Show UV grid	Enabled for Patch Fill. Displays a grid on the Fill preview to help visualize the contours. Use the dropdown slider to adjust the density of the grid. 
Show deviation	Enabled for Patch Fill. Shows a Deviation analysis of the patch. You can set the color and scale used to display the analysis.

In the **Fill** tool, using the **Patch** option, fill multiple areas at once using Alt-selected tangency influences and preview the changes. In the example, the two loops are being filled simultaneously and previewed. Notice the preview change when a different Point is Alt-selected.

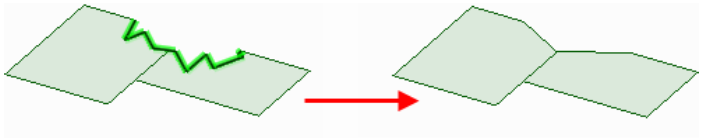


5.4.4.6. Fill Tool Examples

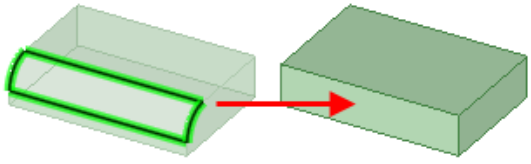
Simplifying edges.



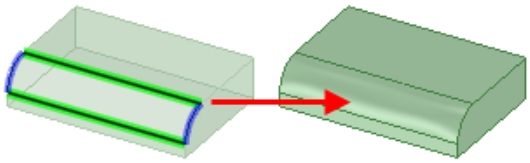
Simplifying edges across multiple faces.



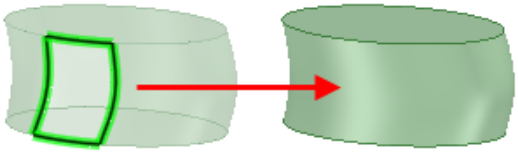
Filling with straight and curved edges.



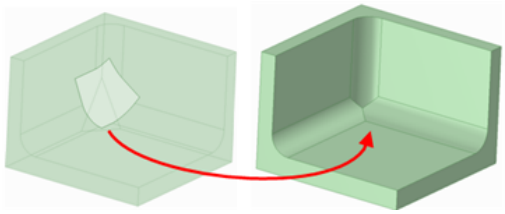
Filling with curved edges as guides.



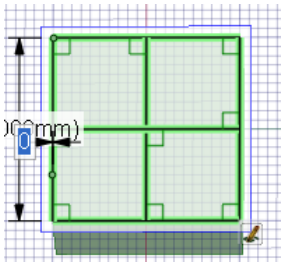
Capping a surface.



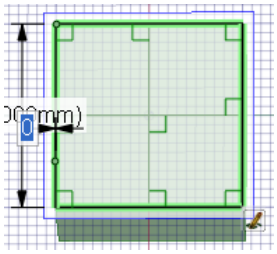
Capping a surface that crosses multiple edges.



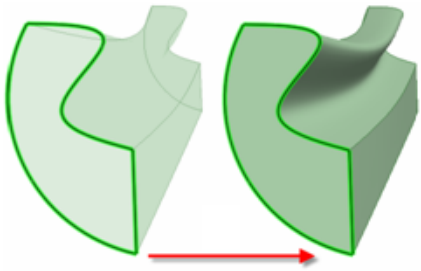
Selecting internal edges to keep them after filling.



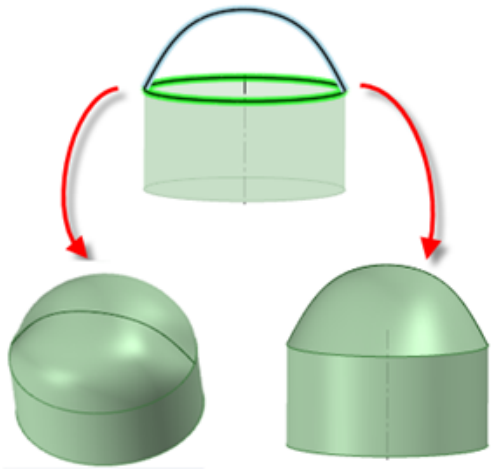
Selecting lines to simplify a surface by filling. Internal edges are removed.



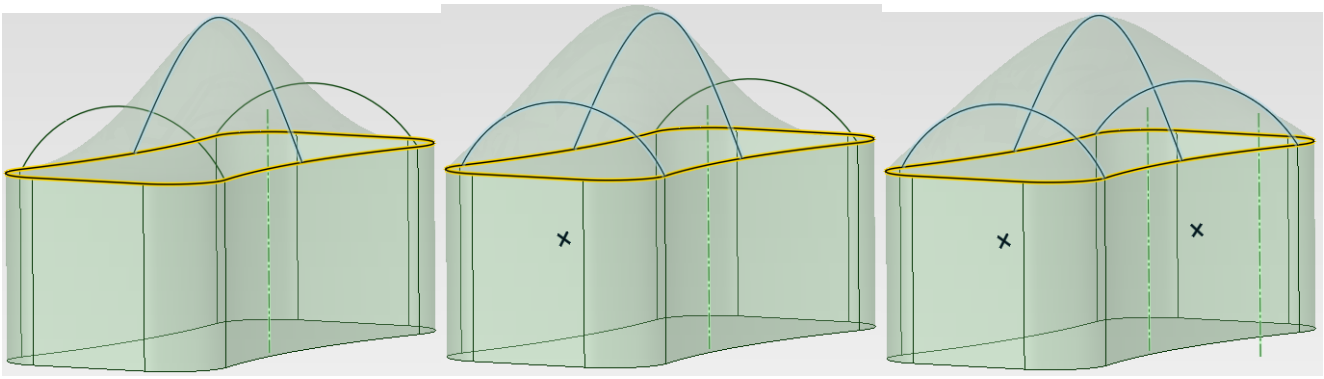
Filling edges to form a solid.



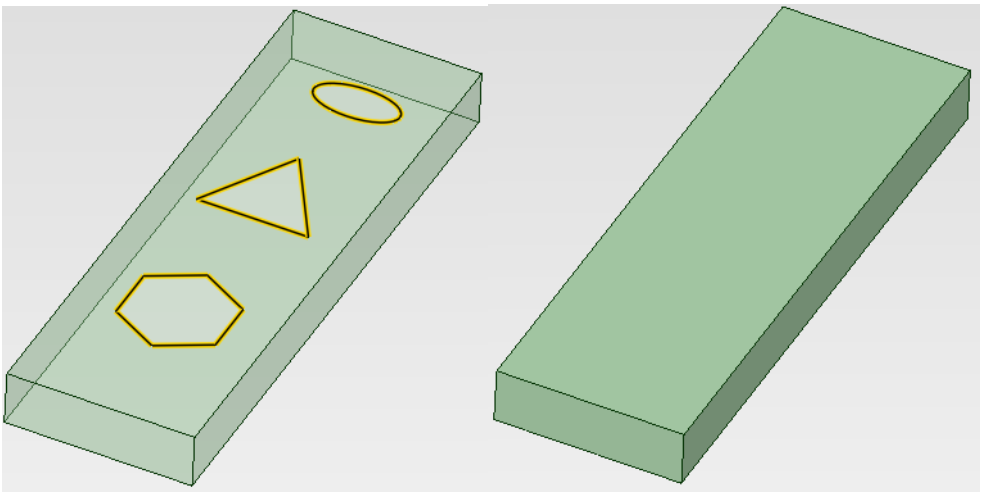
*Patch blend with and without tangent extension. The **Tangent extension** option is selected on the left, and not selected on the right.*



*When selecting **Guide Curves** in the **Fill** tool, preview allows continued selection of multiple Guide Curves. This enables you to understand how the Guide Curves influence the geometry.*

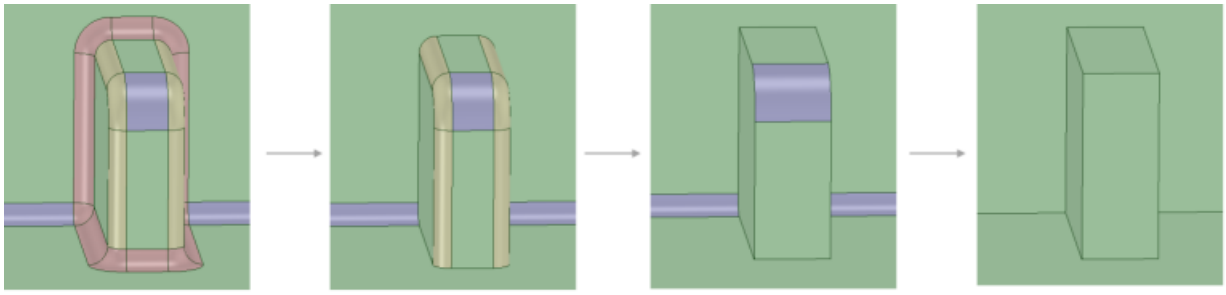


***Fill** works on multiple-selected, disconnected edge loops in surfaces. Use this as a shortcut instead of filling edge loops individually.*



5.4.4.7. Removing Rounds

- You will have a greater chance of success if you remove only a few rounds at a time. If you spend a lot of time removing rounds, you may want to review Discovery's tutorials or technical support information for advanced techniques you can use when removing rounds.
- Closely inspect your model from various angles and zoom levels, so you can best determine the areas of rounds you want to remove. Each area of a model is unique, and different areas may require different approaches to round removal.
- To ensure accurate and complete round removal, remove the rounds in the reverse order in which they were created, as shown in the example below of a model with colored rounds:



This section contains the following topics:

5.4.4.7.1. Filling a Round or Chamfer

1. Click the **Select** tool guide.
2. Select the round or chamfer.
3. Click the **Fill** tool or press **F** to fill the round and create caps if necessary.

   A Round Group is created in the Groups panel each time you fill a round.

   You can fill any round created in Discovery, even a round that removes the underlying face, until the round is changed by some other action.

5.4.4.7.2. Removing a Round Face

1. Select the round face.
2. Click the **Fill** tool or press **F**.

   If removing a face makes an invalid solid, the solid is converted to a surface.

   If a round face is difficult to remove, try Alt+clicking it with the Fill tool, or using the Replace tool.

5.4.4.7.3. Capping Multiple Round Faces

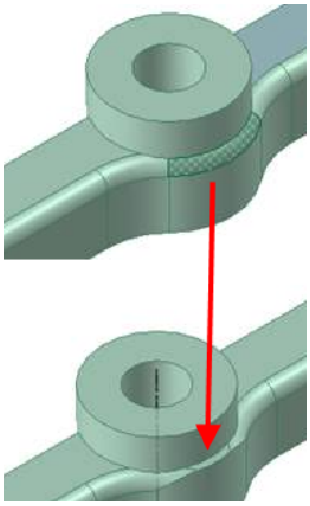
When multiple selected round faces are removed by the **Fill** tool, they are removed in a specific order so that they can be restored by right-clicking and selecting **Reattach Rounds** for each group, in reverse order.

Usually you can select all the round faces and click the **Fill** tool to remove and cap them all. However, sometimes this is not possible. In that case, select one round and fill it. If that works, undo and select that round and the next. Undo. Continue adding rounds to your selection and trying to fill them until the fill fails. Now you have identified one of the rounds that is causing the problem. Next, fill all the rounds that filled successfully. Finally, repeat this process in the other direction of the round tangent chain. Once you have filled all the rounds except for the one or two causing the problem, select the one causing the problem and its two neighbors. Then click Fill. This process allows more options for the extension of neighboring edges to intersect and cap the round.

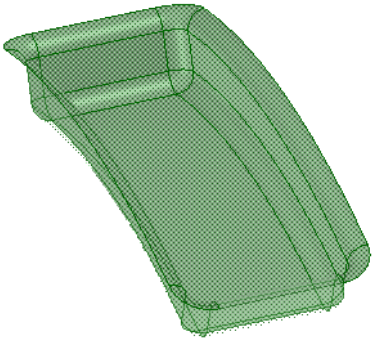
If you are experiencing difficulty filling a chain of round faces, first split the round faces, then fill the newly created faces, then fill the remaining round faces.

5.4.4.7.4. Removing Rounds Examples

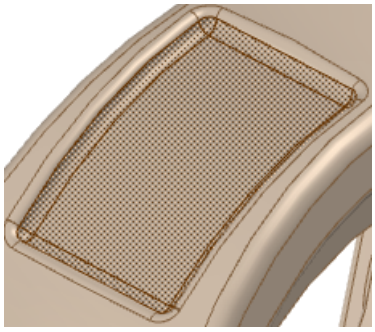
A difficult-to-remove round replaced before removing.



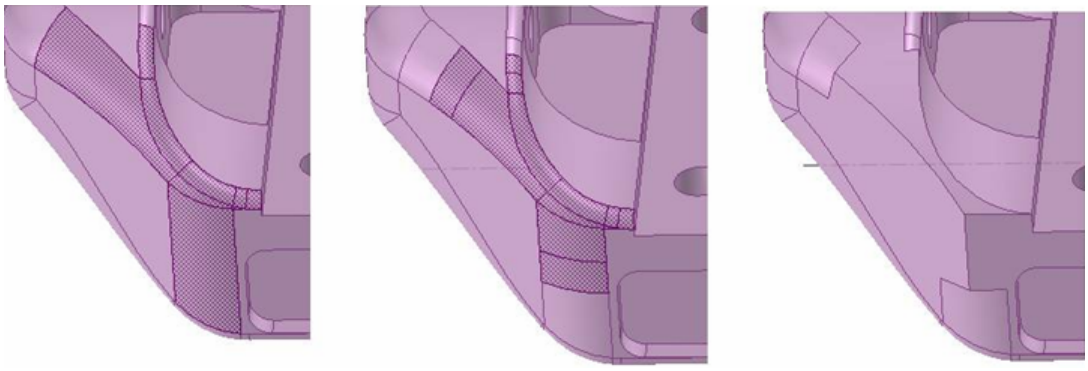
Filling a corner round that was created as a surface - surface round in an imported design.



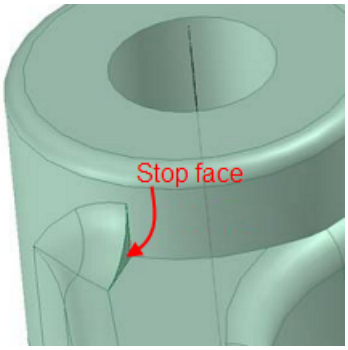
About to fill imported geometry bounded by surface rounds.



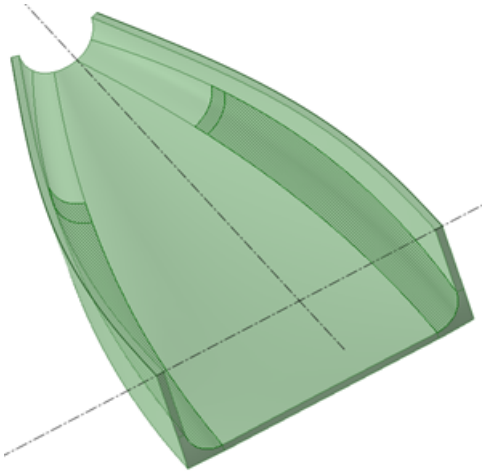
Filling a rounded edge chain - the original chain to be filled, chain after splitting round faces, and filled chain.



If a round cannot be removed, a "stop face" is added and round removal is discontinued (A stop face is a cap or vent face).



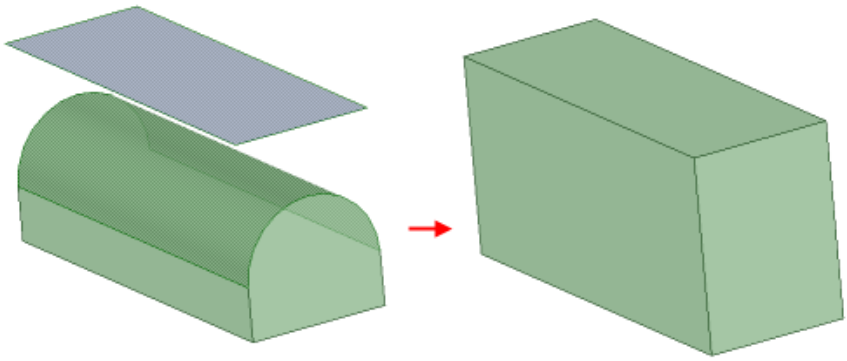
You can fill a chain of round faces when only one end of the round chain has a stop face.



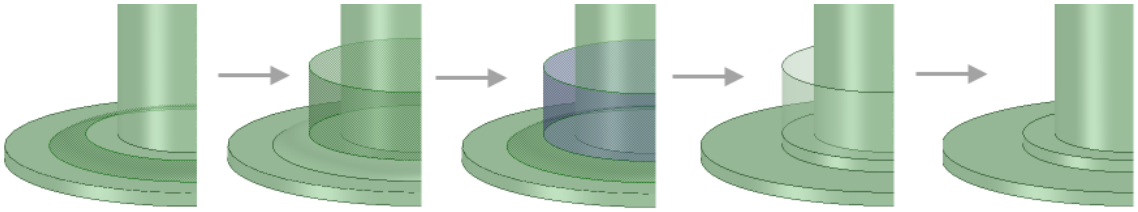
5.4.4.7.5. Removing Rounds with Replace Tool

As an option, when removing rounds, you can also use the Replace tool from the Edit group on the Design tab, to replace one face with another. You can replace multiple faces with a single face, replace a single face with multiple faces, or replace multiple faces with multiple other faces.

Full round replaced with a face.



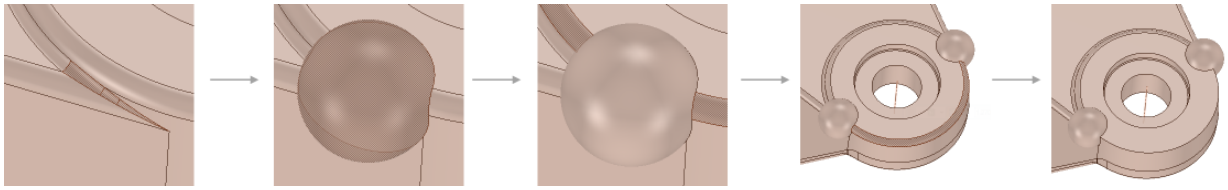
Progression using a face to remove a round.



5.4.4.7.6. Removing Rounds with Sphere Tool

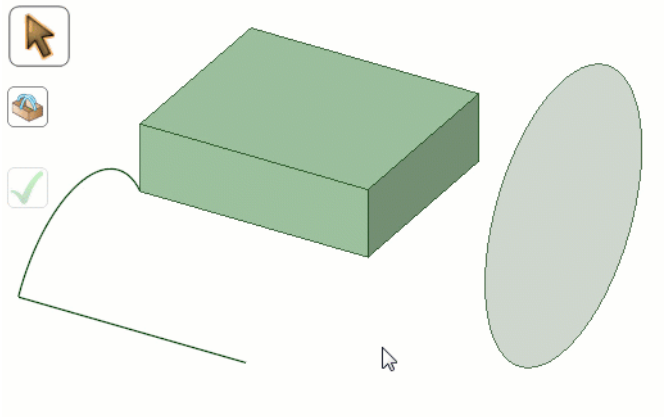
- Try using the Sphere tool from the Insert group on the Design tab. Using this tool, you can create a round sphere and place it at difficult geometry junctions. Once in place you can use the sphere to split the junction in order to remove the round. In some cases, you may want to create a chain of two or more spheres to help with removing larger or more complex rounds areas.
- To avoid trouble with removing spheres, it's good practice to fill the sphere immediately after you remove the rounds on either side of a sphere or sphere chain.

Using the Sphere tool to split and partially remove a round.



5.4.5. Blend Tool

You can create blends between points, edges, or faces.



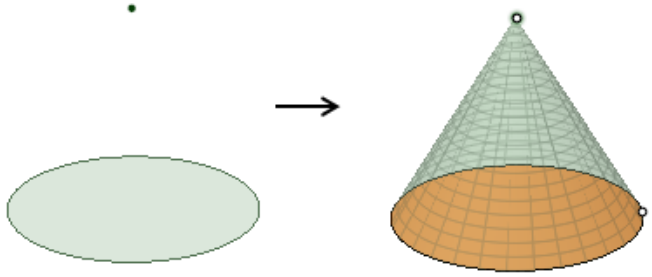
This section contains the following topics:

5.4.5.1. Creating a Blend

- Enter the Blend tool
- Select a point, edge, or face
- Ctrl+Select a corresponding point, edge, or face
  - You can select more than two objects

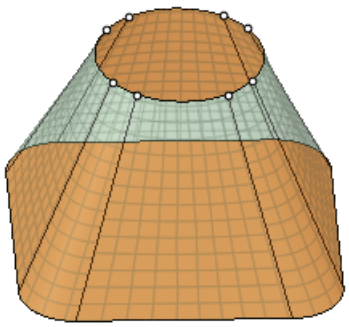
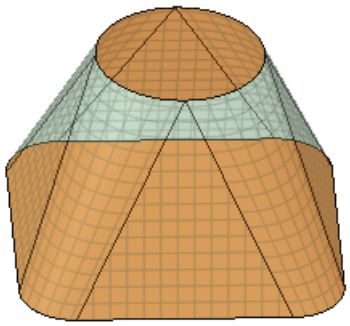
- The blend is previewed as you select
- Complete the blend

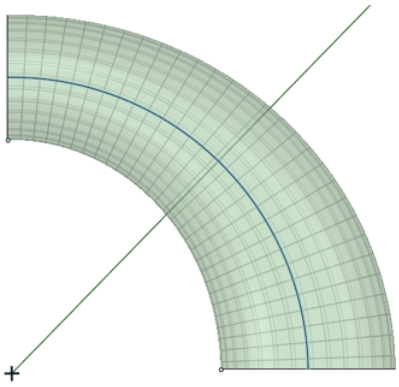
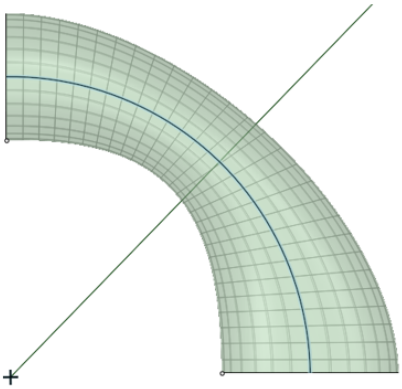
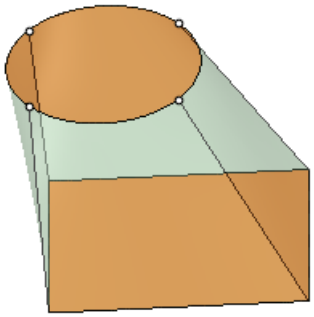
You can blend both closed and open sections to a single point as shown below.



5.4.5.2. Blend Tool Options

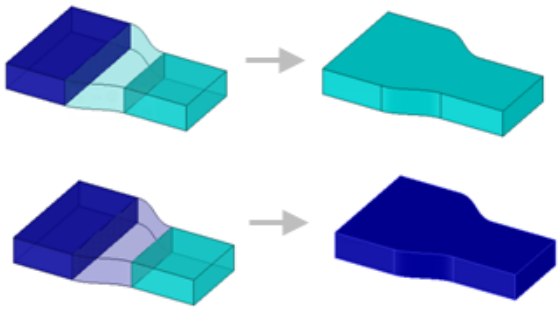
The following options are available in the Blend tool Options panel when you select the appropriate geometry for a blend.

Rotational blend	Create cylinders and cones whenever possible during the creation of a blend. You must have selected faces, points, or edges that can be rotated around a common axis.
Periodic blend	Go all the way around when blending. The blend will begin and end at the first selected object. You must have selected three or more faces, points, or edges that can be rotated around a common axis, and that also span an arc greater than 180 degrees. (Blending between 3 equal-radius circle faces creates a torus.)
Ruled sections	Create straight edges when you blend. When you blend between faces, this option has the same affect as selecting the face and its edges.
Local guides	Selected guide curves only influence areas near to them.
Minimal topology	Create a smooth face blend without unnecessary faces or edges for closed profiles with smooth vertices.
Sheet metal blend	Forces the tool to create developable surfaces. A developable surface is defined in mathematics as a surface with zero Gaussian Curvature (that is, a surface that can be flattened into a plane). The tool attempts to create planes, cylinders, and cones, in that order to maximize the planar areas. It is restricted to blending between two parallel planes.
	Sheet metal blend unchecked. 
	Sheet metal blend checked. 

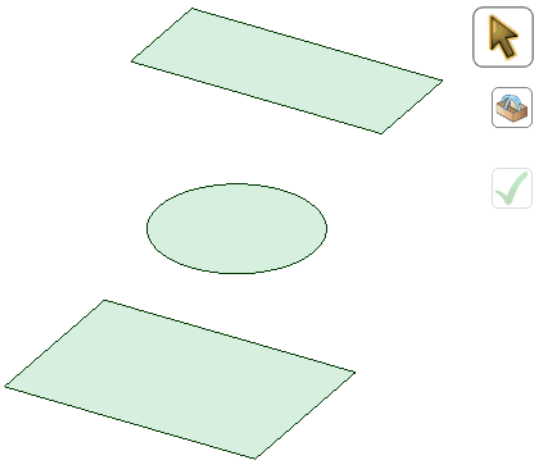
<table>
<tr><td>Normal to Centerline</td><td colspan="2">When on by default, Normal to Centerline forces the Blend algorithm to keep sections normal to the centerline. When off, the algorithm has more freedom to adjust sections so that a surface can be created.</td></tr>
<tr><td></td><td>Normal to centerline On.
  </td><td>Normal to centerline Off
  </td></tr>
<tr><td>Show UV grid</td><td colspan="2">This option is ON by default to help visualize contours by displaying a grid on the preview. Use the dropdown slider to increase or decrease the density of the grid.
  </td></tr>
</table>

Example

When blending between colored objects, the blended geometry takes on the color of the object that was selected first.



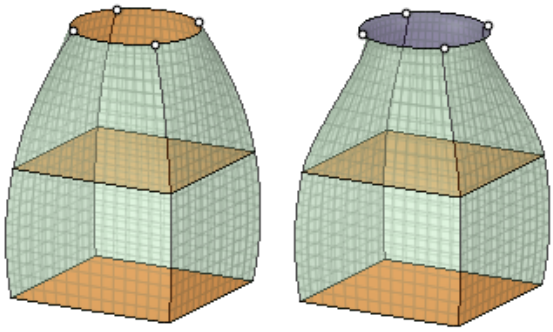
5.4.5.3. Blending Between Faces



You can blend between two or more faces with the Blend tool. Blended faces automatically simplify to analytic geometry when possible.

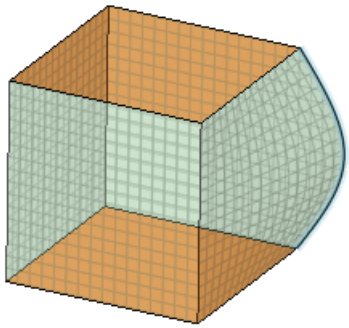
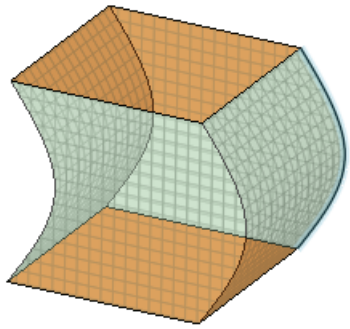
5.4.5.3.1. Blending Between Two Faces

1. Click **Blend** in the Design tab Edit group.
   The **Select** tool guide is enabled by default.
2. Select the first surface or face.
3. Ctrl+Select the second surface or face.
   A blend preview displays between the first two faces. Blend surfaces begin tangent to the edges of the initial face and end tangent to the edges of the last face. You can Alt+Click these faces to reverse the tangent direction.

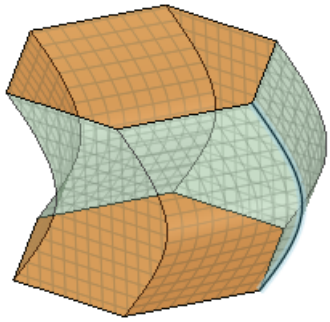
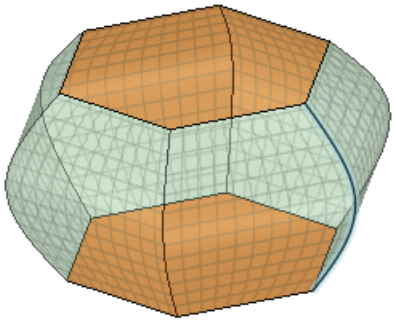


4. (Optional) Alt+click the edges or lines you want to use as guides for the blend. There is also a tool guide for selecting guide curves. Guide curves must touch all blend profiles.
5. Continue selecting surfaces or faces.
6. (Optional) Modify the blend by selecting from the following options:
   - **Rotational blend**: Create cylinders and cones whenever possible during the creation of a blend. You must have selected faces rotated around a common axis.
   - **Periodic blend**: Go all the way around when blending. You must have selected three or more faces rotated around a common axis, that also span an arc greater than 180 degrees. (Blending between 3 equal-radius circle faces creates a torus.)

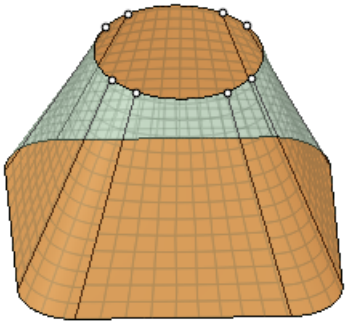
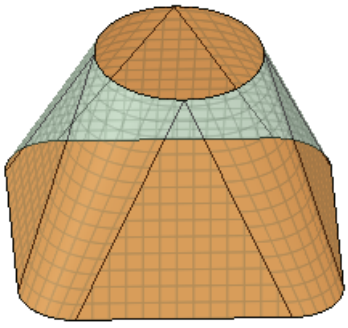
- **Ruled segments**: Create straight edges when you pull between three or more surfaces or faces.
- **Local Guides**: Selected guide curves only influence areas near to them.

Checked	*Unchecked*
	

- **Clocked guides**: Guide curves are oriented relative to the face edges instead being simply translated from vertex to vertex.

Clocked guides Unchecked	*Clocked guides Checked*
	

- **Sheet metal blend**: Forces the tool to create developable surfaces. A developable surface is defined in mathematics as a surface with zero Gaussian Curvature (for example, a surface that can be flattened into a plane). The tool attempts to create planes, cylinders, and cones, in that order to maximize the planar areas. It is restricted to blending between two parallel planes.

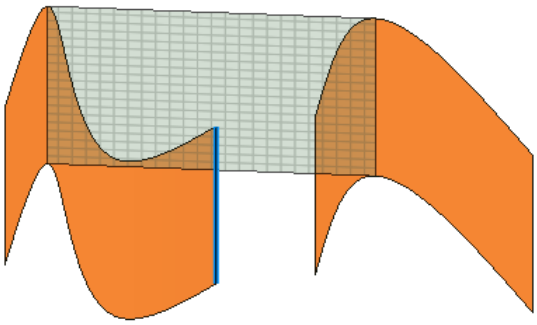
Sheet metal blend unchecked	*Sheet metal blend Checked*
	

7. Click **Complete** to create the blend.

This section contains the following topics:

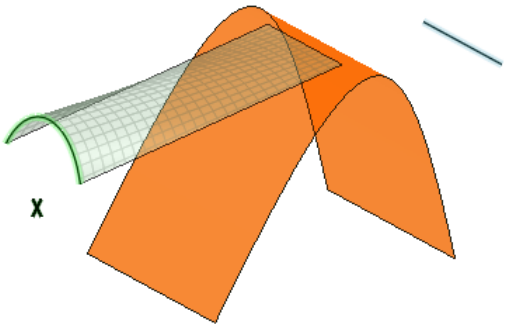
5.4.5.3.1.1. Blending a Surface Tangent to a Face Along a Given Direction

1. Select two curved faces and Alt+click a linear object, such as an axis, datum axis, sketch curve, or edge to indicate the direction for finding two tangent locations.
2. Where you click when selecting the surface determines the approximate location of the edge of the surface. The **Blend** tool previews the edges and the resulting surface.

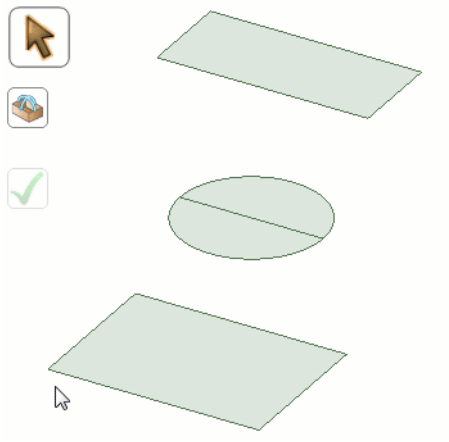


5.4.5.3.1.2. Blending a Surface Tangent to One Face and Through One Curve/Edge Along a Direction

Select a face and a curve and Alt+click a linear object to set the direction.



5.4.5.4. Blending Between Two or More Edges



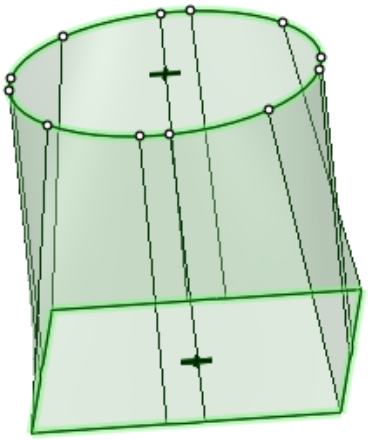
Use the **Pull** tool in the Design tab of the Edit group to blend between two or more edges. You can select guide curves for the edges to follow when creating the blend.

This section contains the following topics:

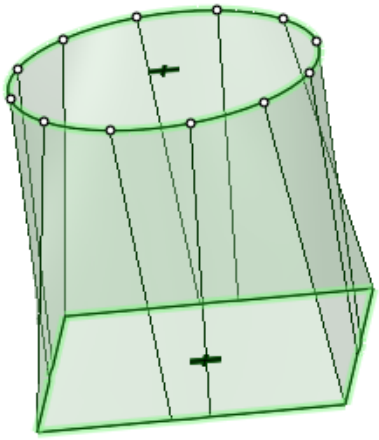
5.4.5.4.1. Blending Between Edges

1. Click **Blend** in the Design tab Edit group.
   The Select tool guide is enabled by default.
2. Select the first curve or edge.
3. Ctrl+Select the second curve or edge.
   A blend preview displays between the first two edges.
4. (Optional) Alt+click the edges or curves you want to use as guides for the blend. Guide curves must touch all blend profiles.
5. Continue selecting curves or edges.
6. (Optional) Modify the blend by selecting from the following options:
   - **Rotational blend**: Create cylinders and cones whenever possible during the creation of a blend. You must have selected edges rotated around a common axis.
   - **Periodic blend**: Go all the way around when blending. You must have selected three or more edges rotated around a common axis, that also span an arc greater than 180 degrees.
   - **Ruled sections**: Create straight edges when you blend between three or more curves or edges.
   - **Local Guides**: Selected guide curves only influence areas near to them.
   - **Clocked guides**: Guide curves are oriented relative to the edges instead being simply translated from vertex to vertex.

Clocked guides unchecked

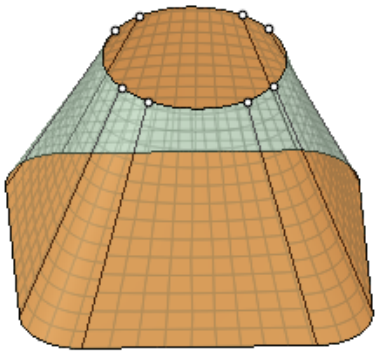


Clocked guides checked

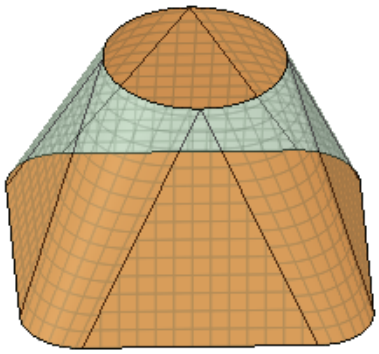


- **Sheet metal blend**: Forces the tool to create developable surfaces. A developable surface is defined in mathematics as a surface with zero Gaussian Curvature (that is, a surface that can be flattened into a plane). The tool attempts to create planes, cylinders, and cones, in that order to maximize the planar areas. It is restricted to blending between two parallel planes.

Sheet metal blend unchecked.



Sheet metal blend checked.

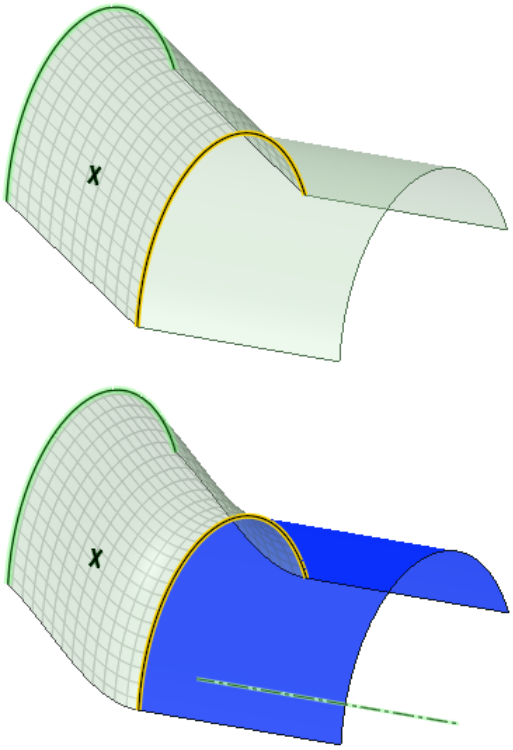


7. Click **Complete** to create the blend.

When a blend between splines will self-intersect, the splines are modified slightly to prevent this from happening.

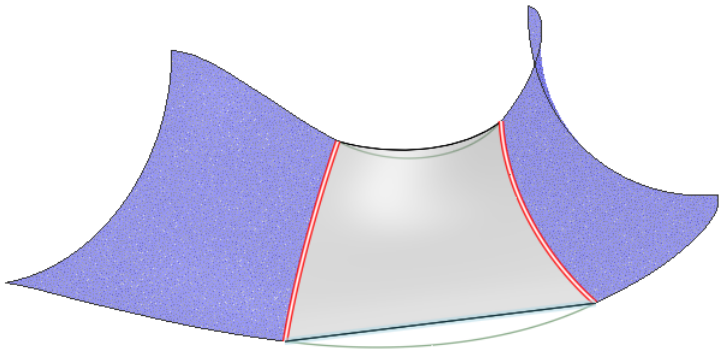
5.4.5.4.2. Adding Tangency

Make the blend tangent to adjacent faces by Alt+Selecting them. In the image below right, the blue face was Alt+selected to make the blend surface tangent to it.



The example below is one example of this.

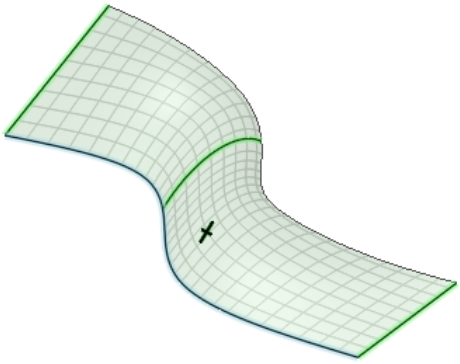
In the Blend tool, the system of Tangency and guide constraints sets up a system of equations that Discovery solves to produce the Blend result. Certain cases create a system of equations which is Inconsistent, meaning the equations produce a contradiction and have no solution. The figure below is an example of this.



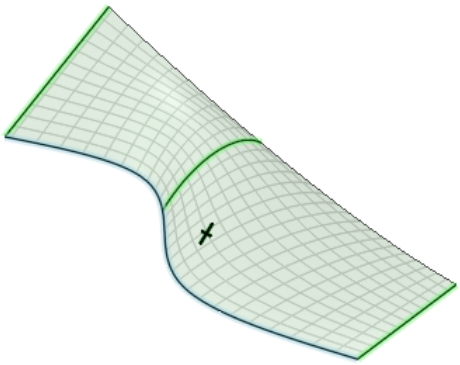
In some cases, when Blending with a combination of Tangency and Guide constraints, conflicts between the constraints can occur. In these cases, the conflicted geometry will highlight and an error message will be displayed.

5.4.5.4.3. Swept Blends

A Swept Blend is a blend between two or more edges without local guide influence. The entire blend is swept along the guide curve, as shown below.



If you check the Local guide option, the guide curve has a local effect on the blend.



Swept blends are different from sweeps. Swept blends need at least two profiles, while sweeps use a single profile. Sweeps give you limited control over the orientation and angle of the section along the trajectory, or at the end cap, because it is always determined by sweeping the profile along the trajectory.

With swept blends, you specify the section shape and orientation explicitly at the ends and at specific points along the trajectory (guide curve).

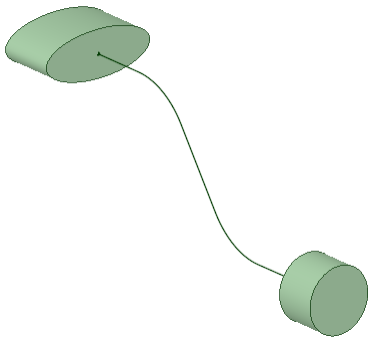
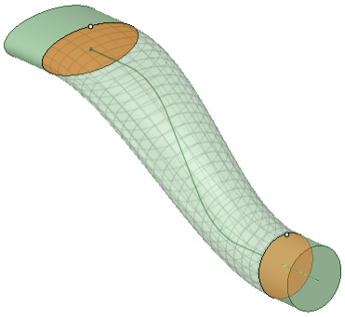
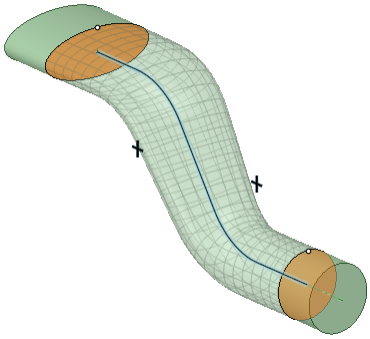
When you select more than one guide curve, the additional guide curves give you more control over the final shape of the blend. Each guide curve affects entities in the profile that are closest to the curve. Vertices between guide curves are controlled by curves blended between the neighboring guide curves.

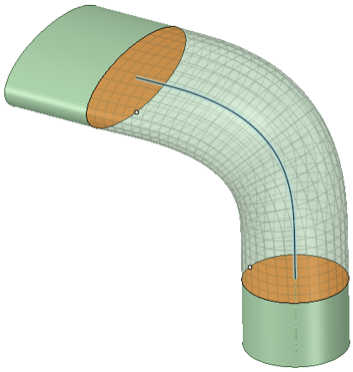
5.4.5.4.4. Centerline Blends

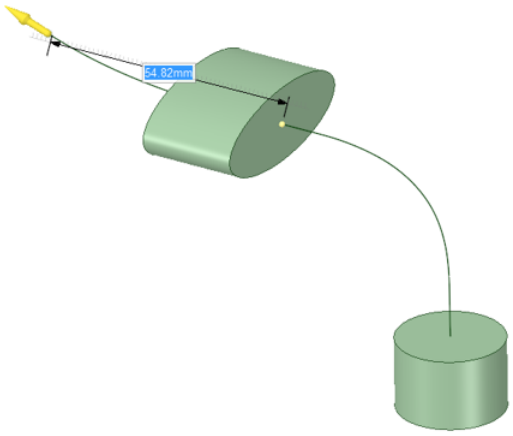
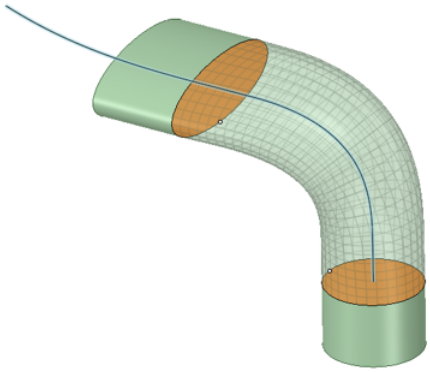
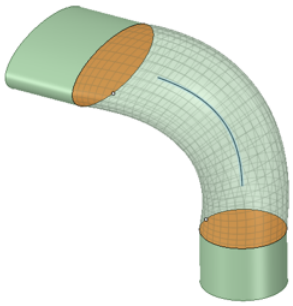
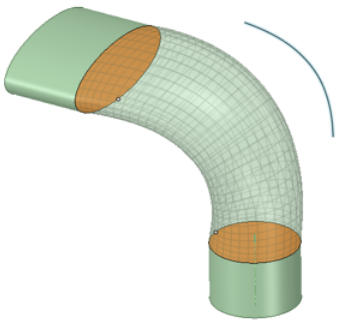
The **Blend** tool allows you to select a **Centerline Path** for the **Blend** to follow between two planar profiles. The **Centerline** does not have to touch either of the profiles, but the extension of the curves must pass through the profile. Also, although it is called a "centerline," the **Path** does not need to pass through the center of each profile. **Centerline** distinguishes itself from **Guide** curves, which must touch the profile edges. Multi-segmented curves or edges can be used as centerline blends provided the meet the following criteria:

- Meet end-to-end without gaps
- Are piece-wise continuous or tangent

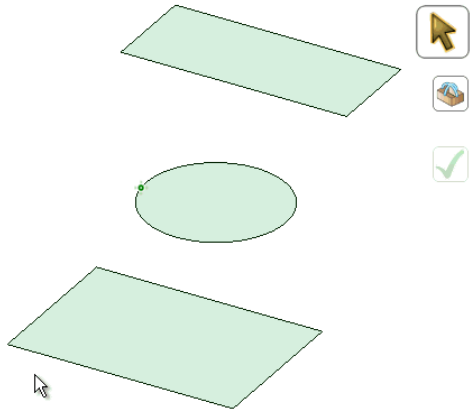
Centerline Blends Example

The centerline consists of three lines and two tangent arcs.	
Select the oval and circle faces to preview the default **Blend**.	
Select the **Centerline** tool and double-click the centerline to select the curve chain. The preview updates to show the **Centerline Blend**.	

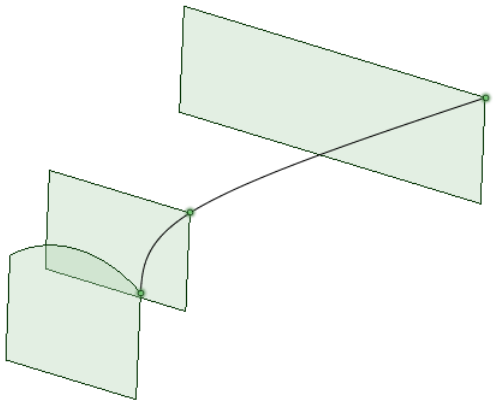
The centerline can pass through the profiles. **Note:** In these examples, the centerline has been changed to a spline.	

You can extend the spline using the **Pull** tool to pull it beyond the profile plane.	
The **Blend** can be created using the extended spline.	
The spline can be shortened so it does not touch the profile planes.	
The centerline can be moved normal to the original plane. In this example, the spline was moved to the right, which is normal to its original sketch plane.	

5.4.5.5. Blending Between Multiple Points



You can create a curve by blending between any combination of sketched points, vertices, or endpoints. Blend between any two points to create a line, or select multiple points to create a spline.



5.4.5.5.1. Blending Between Points

1. Enter the **Blend** tool.
2. Select the first point or vertex.
3. Ctrl+click to select multiple points or vertices.
   A blend preview displays between the first two points.
4. Ctrl+Select more points or vertices.
5. To make the curve tangent:
   - Ctrl+click neighboring edges of the vertices if you want the blend curve to begin tangent to one or more edges.
   - Alt+click one or more neighboring faces if you want the blend curve to be tangent to the face(s).
6. (Optional) Modify the blend by selecting from the following options:
   - **Rotational blend**: Create arcs whenever possible during the creation of a blend. You must have selected points rotated around a common axis.
   - **Periodic blend**: Go all the way around when blending. You must have selected three or more points rotated around a common axis, that also span an arc greater than 180 degrees.

- **Ruled sections**: Create straight line segments when you blend between three or more points.

7. Click **Complete** to create the blend.

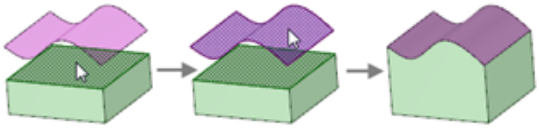
5.4.6. Replace Tool

Use the **Replace** tool in the Edit ribbon group on the Design tab to replace one face with another. You can replace multiple faces with a single face, replace a single face with multiple faces, or replace multiple faces with multiple other faces.

The **Replace** tool also lets you manually simplify or align complex faces and curves into planes, cones, and cylinders. (If you want to automate this process, use the Simplify tool in the Adjust group on the Repair tab to automatically find problem areas.)

This section contains the following topics:

5.4.6.1. Replacing a Face



1. Click the **Replace** tool in the **Edit** ribbon group on the **Design** tab.
2. Select the face you want to replace (that is, the *target*).
3. Select the face, surface, or plane you want to use to replace the target face (that is, the *source*).
   You can select sources in the Design window or in the model tree.

> **Tip:** You can also:
>
> 1. Click the target face and Alt+click the source face, surface, or plane with the **Select** tool.
> 2. Click the **Replace** tool to replace the target face with the source face.

5.4.6.2. Replacing Multiple Targets

1. Click the **Replace** tool.
2. Click the **Target** tool guide to make it sticky, then click each target face.
   You can also use any multiple-face selection method.
3. Select the source face, surface, or plane.
   You can select a source in the Design window or in the model tree.

5.4.6.3. Replacing a Target with Multiple Sources

1. Click the **Replace** tool.
2. Select the target face.

3. Click the **Source** tool guide to make it sticky, then click each source face, surface, or plane.

   You can select sources in the Design window or in the model tree and can also use any multiple-face selection method.

4. Click the **Complete** tool guide or press **Enter** to replace the target.

5.4.6.4. Replacing Multiple Targets with Multiple Sources

1. Click the **Replace** tool.
2. Click the **Target** tool guide, then click each target face.

   Youcan also use any multiple-face selection method.

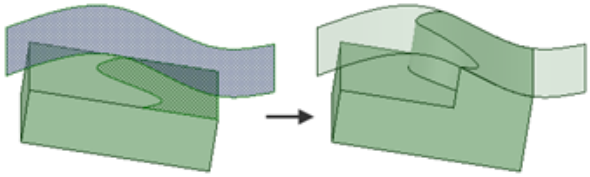
3. Click the **Source** tool guide twice to make it sticky, then click each source face, surface, or plane.

   You can select sources in the Design window or in the model tree and can also use any multiple-face selection method.

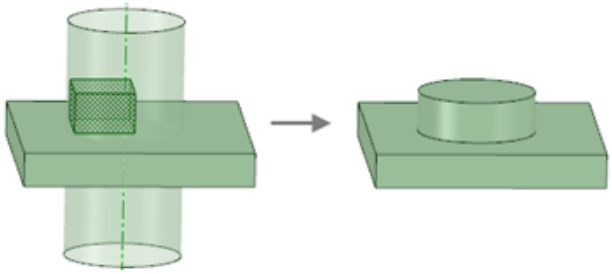
4. Click the **Complete** tool guide or press **Enter** to replace the targets.

5.4.6.5. Replacing Faces Examples

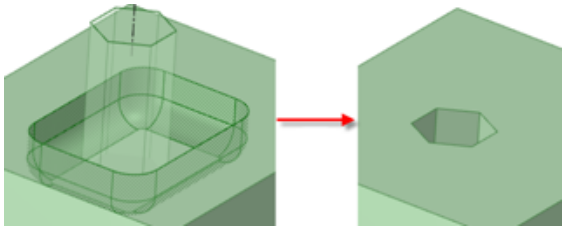
Replacing a target split face with a source surface.



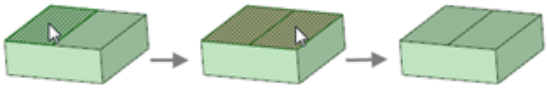
Replacing multiple target faces (the protrusion) with a single source face (the surface).



Replacing multiple target faces (in the depression) with multiple source faces (on the surface).



5.4.6.6. Aligning or Simplifying Faces



1. Click the **Replace** tool.
2. Click the **Target** tool guide, then click each face that you want to align or simplify.

   You can also use any multiple-face selection method.

   If the selected faces are candidates for alignment or simplification, they are highlighted in red. Faces can be aligned or simplified by the Replace tool if they are within Discovery 's alignment or simplification tolerance of ±0.0001m.
3. Click the **Complete** tool guide or press the **Enter** key to align or simplify the highlighted faces.

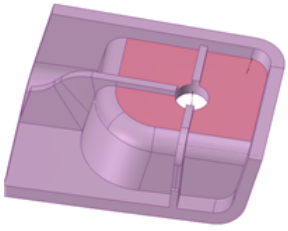
5.4.6.7. Simplifying Curves

1. Click the **Replace** tool.
2. Ctrl+click each curve that you want to simplify.

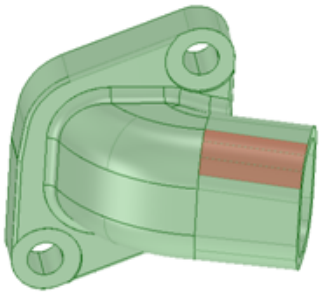
   If the selected curves are candidates for alignment or simplification, they are highlighted in red.
3. Click the **Complete** tool guide or press the **Enter** key to align or simplify the highlighted curves.

5.4.6.8. Replace Tool Example

Almost-planar faces highlighted for alignment



Faces highlighted for simplification



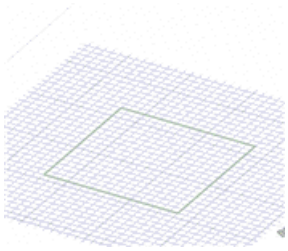
5.4.6.9. Replace Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Target	Select target faces using the sticky **Target** tool guide.
Source	Select source faces, surfaces, or planes using the sticky **Source** tool guide.
Complete	Click the **Complete** tool guide to replace the target face with the source face, or to simplify or align the target face.

5.4.7. Bend Tool

Use the **Bend** tool to bend straight lines and edges to form an arc. You can also use the **Bend** tool to adjust the radius of arcs and arced edges. Bend works on straight lines when you are sketching and when you are editing in cross-section.



This section contains the following topics:

5.4.7.1. Bending a Line or Edge

1. Click **Bend** in the Sketch group.
2. If you are working in Section mode, select the face that contains the edge.
3. Click and drag a line or arc to bend it or change its radius.
   Enter a value to dimension the radius and bend angle.

The common sketch options are described in Sketch Tool Guides and Options on page 133.

5.4.8. Mathematical Expressions

The following expression elements are available:

- Infix (dyadic) operators: **+ - * / ^**
- Prefix (monadic) operators: **+ -**
- Functions: **sin, cos, tan, asin, acos, atan, sqrt, log, log10, exp**
- Constants: **pi, e, root2, root3**
- Units: **m, cm, mm, yd, ft, ', in, ", deg, rad**

Normal precedence rules apply:

1 + 2 * 3 ^ 4 = 1 + (2 * (3 ^ 4)) = 163

Parentheses are required for expression arguments and optional for simple arguments:

- sqrt 2 = sqrt(2) = 1.4142...

- sqrt 2*2 = (sqrt 2) * 2 = 2.8284...
- sqrt(2*2) = 2

Missing operators are inferred:

- 1 1/2 = 1 + 1/2
- 1'6" = 1' + 6"
- 1ft 6in - 17in = 1ft + 6in - 17in
- 1 2 3 4 5 = 1 + 2 + 3 + 4 + 5 = 15
- (1)(2)(3)(4)(5) = (1) * (2) * (3) * (4) * (5) = 120
- 2(1 + 2) = 2 * (1 + 2) = 6
- sqrt 2 sqrt 2 = sqrt 2 * sqrt 2 = 2
- 4(4atan(1/5) - atan(1/239)) = 4 * (4 * atan(1/5) - atan(1/239)) = pi

Units are applied to previous terms if units were not specified and are applied to subsequent terms unless you override them:

- 1 + 1cm = 1cm + 1cm
- 1cm + 1 = 1cm + 1cm
- 1cm + 1 + 1mm = 1cm + 1mm + 1mm
- 1cm + 1 1/2 mm = 1cm + 1mm + 1mm / 2

Trigonometry functions work in radians by default, but you can enter degrees:

sin(45 deg)

Numbers support standard form, but e is a built-in constant:

- 2e2 = 200 2e 2 = 2 * e * 2 = 10.873...
- 2e-2 = 0.02 2e - 2 = 2 * e - 2 = 3.436...
- 2e1 = 20 2e = 2 * e

5.5. Intersecting

You can use the intersect tools to merge and split a solid or surface in your design with another solid or surface. You can merge and split solids or surfaces with other solids or surfaces, split a solid with a face, and split a face with another face. You can also project the edges of a face onto other solids and surfaces in your design.

Discovery's intersection capabilities include the full suite of geometry combination, all done with one major tool (**Combine**) and two minor ones (**Split Solid** and **Split**). Combine always takes two or more objects. The split tools always act on one object and that object is automatically selected from the cutter or projection face.

To understand what the **Combine** tool can do, the first step is to know which objects can be handled. Solid objects have faces that meet at corner edges. Edges that lie on faces can be deleted, but corners cannot. Surface objects have faces that come together at internal edges, and are surrounded by outside edges. Outside edges can be extended. Internal edges can be deleted if they are bounded by planar faces. For the purposes of combine, planes can be thought of as surface faces that extend across the design.

When surfaces enclose a volume, they automatically change into solid objects. When edges of the same surface become coincident, they will automatically merge. Planes cannot be split by any combine operation, but they can be used to split with. In general, layer, color, and visibility in the model tree propagates from the first selected item (the target) to the result. What happens as a result of the combination can be overridden with the options (both in the panel and the mini-toolbar). When you use the **Combine** or **Split** tools, the newly created objects have the layer properties of the previous objects.

You have complete control over the pieces that solids and surfaces get cut into. When appropriate, Discovery prompts you to remove regions, but you can choose to keep or remove those regions.

When using the intersect tools, the original, individual layers and colors of the objects are maintained.

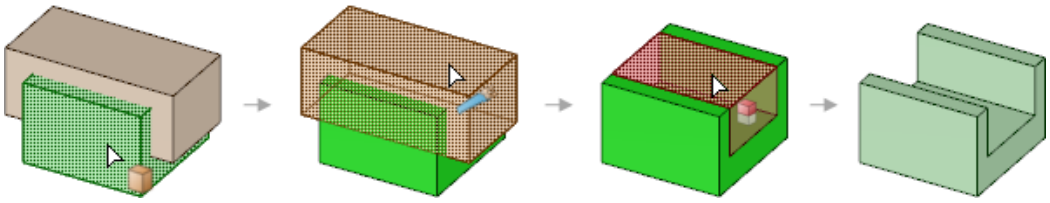
This section contains the following topics:

5.5.1. Intersect Ribbon Group

The **Intersect** ribbon group contains the following tools:

Combine	Use the Combine tool to merge and split solids and surfaces.
Split Solid	Use the Split Solid tool to split a solid by one or more of its faces or edges. Then select one or more regions for deletion.
Split	Use the Split tool to create an edge on a face by splitting it with another face or surface.
Project to Solid	Use the Project to Solid tool to create edges on a solid's face by extending the edges of another solid or surface.

5.5.2. Combine and Split Tools



The **Combine** tool is used to make combinations of objects. You can add (or merge) objects together and you can subtract (or split) objects from each other. These actions are also known as Boolean operations.

You can also specify a **Combine tolerance** to allow for merging or splitting of bodies which otherwise cannot be merged or produce slivers when split.

Tool guides for the **Combine** tool are sticky, and appear with a double outline when you click them. The tool guide remains selected so you can perform the same action repeatedly without holding the Ctrl key. To unstick a tool guide, click it again, click another tool guide, or click an empty place in the graphics window.

When you use a pattern with the **Combine** tool, the entire pattern is merged or used to cut the target.

This section contains the following topics:

5.5.2.1. Splitting Solids and Surfaces

1. Click **Design** > **Intersect** > **Combine** .
2. Specify the **Combine tolerance**, if applicable.

3. Select the target solid or surface that you want to split.

   You can select objects for **Combine** in the model tree.

   You can use temporary geometry to split an object.

   You can use a locked body as a cutter with the Combine tool, but you cannot use it as target.

4. (Optional) Select the following options:

   - **Merge when done**: Merge all newly-created, touching solids or surfaces when you exit the Combine tool.
   - **Imprint curves**: Create edges at intersections instead of selecting regions.
   - **Keep cutter**: Keep the cutting surface in your design. If this option is not selected, then the cutting surface is automatically deleted as soon as you select it.
   - **Make all regions**: Cut the target object with the cutter object and the cutter object with the target object. The target and cutter must be the same type of object.

5. Select one or more object to use as the cutter.

   Depending on the option you chose, the cutting surface will be kept or deleted. You can see the solids or surfaces created by the cut in the **Structure** panel.

   Move your mouse over the target solid to see the regions created by the cut.

6. (Optional) Select the split region(s) that you want to delete.

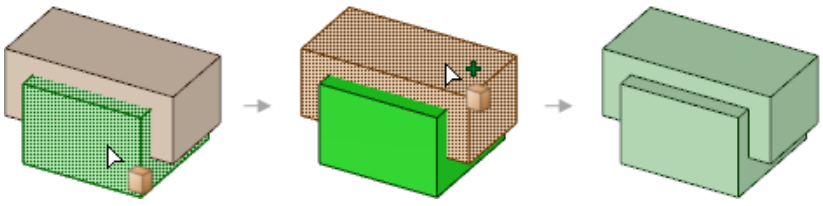
5.5.2.2. Merging Solids and Surfaces

1. Click **Combine** .
2. Specify the **Combine tolerance**, if applicable.
3. Select the target solid or surface.

   You can select objects for **Combine** in the model tree. You can also box select multiple solids or surfaces to merge them in one operation.

   Solids can be merged with solids, and surfaces with surfaces. Solids and surfaces can only be merged if the surfaces make a region that can be added to or cut out of the solid.

4. Click the **Select Bodies to Merge** tool guide or hold the **Ctrl** key.
5. Select the solid(s) or surface(s) that you want merged with the target.



Tip: Hold **Ctrl** and select the solids or surfaces you want to combine, and then click the **Combine** tool to merge them.

5.5.2.3. Making 3D Curves at the Intersection Between Faces

1. Click **Combine** .
2. Select the target solid or surface.

5.5.2.4. Combine Tool Guides

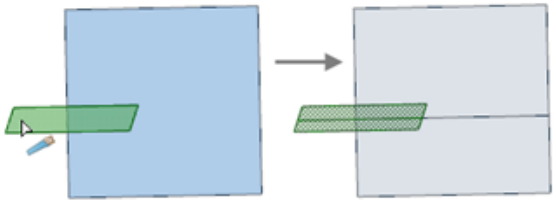
Tool guides for the **Combine** tool are sticky, and appear with a double outline when you click them. The tool guides remain selected so you can perform the same action repeatedly without holding the Ctrl key. To unstick the tool guide, you can click it again, click another tool guide, or click an empty place in the Design window.

Select Target	The **Select Target** tool guide is active by default. If you did not pre-select the target solid or surface, you can select it from within the Combine tool using the Select tool guide.
Select Bodies to Merge	Click the **Select Bodies to Merge** tool guide to select multiple solids or surfaces to merge together.
Select Cutter	The **Select Cutter** tool guide activates once you select a target. When this tool guide is active, click to select the solid or surface you want to use to cut the target. You can Ctrl+click when this tool guide is active if you need to add other bodies to your cutter selection.
Select Regions	The **Select Regions** tool guide activates once the target is cut. When this tool guide is active, mouse over the target to preview the regions created by the cut. Click a red region to delete it.

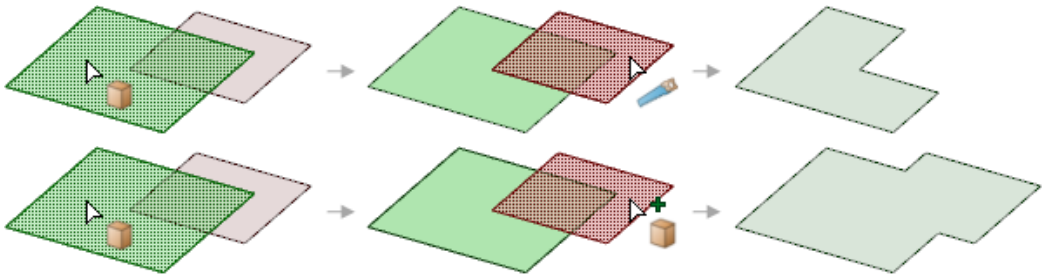
5.5.2.5. Combine Options

The following options are available for the **Combine** tool. Select one or more of these options from the Options docking panel, or right-click and select them from the mini-toolbar:

Combine tolerance	The default value is zero. Specify the Combine tolerance to merge or subtract bodies which cannot be merged otherwise or produce small slivers with the default setting.

Make solids	Solids are made when possible where the target and cutter objects intersect. When you select this option, you can choose any of the following: • **Merge when done**: Select this option to merge all newly-created, touching solids or surfaces when you exit the Combine tool. You can also click white space anywhere in the Design window to complete the merge. Hidden objects are not merged. This saves you the extra step of selecting all the cut-up regions after you are done with a complicated slice-and-dice session, and manually merging them all back together. • **Keep cutter**: Discovery assumes that you created a cutter object to be used only for cutting. If you want to keep the cutting surface in your design, select this option. If this option is not selected, then the cutting surface is automatically deleted as soon as you select it (that is cutter objects are normally used up unless you select this option). A kept cutter can be a surface or solid, but either way only the regions of the target can be removed. If you are splitting surfaces, check this option to prevent the cutter object from being split by the target object. This option is automatically selected if the cutter object is locked. • **Subtract from target**: Select this option to subtract the body created by the overlap of the target with the cutter body. • **Make all regions**: Select this option to cut the target object with the cutter object and the cutter object with the target object. Target and cutter must be the same type of object, either both solids or both surfaces. Because this option can create a large number of regions, it can be helpful to use this option along with the **Merge When Done** option to quickly merge all remaining regions when you click another tool or press **Enter** to finish using Combine. • **Create imprints**: Select this option to create edge imprints when subtracting bodies with coincident faces.
Make curves	Select this option to make 3D curves where the target and cutter object intersect. The curves are created in the active part, rather than the part that the first body belongs to. • **Imprint curves**: Select this option to create edges at intersections instead of creating 3D curves. You will not be able to preview regions for deletion. As soon as a region is selected for deletion, this option is disabled. The edges are created on the first body you select.
Extend intersections	**Extend intersections**: Select this option to extend the intersection of partially intersecting surfaces so the underlying surface is completely split. 
Transfer named selections	Select this option to transfer named selections while merging or subtracting bodies using the **Combine** tool.

5.5.2.6. Combine Examples



You can use Combine to cut or merge coincident surfaces. (The delete region step is not shown in the first illustration.)

5.5.2.7. Using Combine to Merge Solids and Surfaces

Use the **Combine** tool to merge solids and surfaces.

1. Select **Design** > **Intersect** > **Combine** or press **I**.
2. Specify the Combine tolerance, if applicable.
3. Click the first solid or surface.
4. Ctrl+click additional solids or surfaces to merge them.

   The model tree shows the merge.

> **Note:** Ctrl+click the solids or surfaces you want to combine in the model tree, then click the **Combine** tool to merge them.

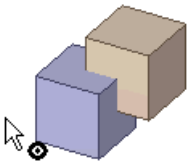
Solids can be merged with solids, and surfaces with surfaces. Solids and surfaces can only be merged if the surfaces make a region that can be added to or cut out of the solid. For example, if you select the face of a cylinder, copy and paste it, then you can merge it with the solid used to create it.

5.5.2.8. Merging Solids

You can merge two or more solids with the **Combine** tool.

This section contains the following topics:

5.5.2.8.1. Merging Two Solids

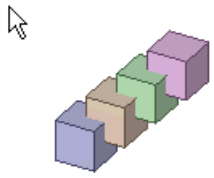


1. Select **Intersect** > **Combine** .
2. Specify the **Combine tolerance**, if applicable.

3. Click the first solid.
4. Ctrl+click the second solid to merge it with the first solid.

   The second solid takes on the color and visibility properties of the first solid. The model tree shows the merge.

5.5.2.8.2. Merging Multiple Solids



1. Specify the **Combine tolerance**, if applicable.
2. Select the solids you want to combine using box selection.
3. Select **Intersect** > **Combine** .

   The solids take on the color and visibility properties of the merged solid that appeared top-most in the model tree. The model tree shows the merge. You can also select the Combine tool, select the first solid, then Ctrl+click each individual solid you want to merge.

Notes on merging multiple solids:

- When there are two bodies selected, and the second body selected (the tool) has multiple instances, the tool is made independent before being merged with the target.
- When more than two bodies are selected, all instances that are consumed by the Merge are made independent. Those that are not merged are left as instances. The merge result is in the first non-instanced body of the selection.
- When all the bodies in an instance of a component are merged, the merged body is made in that component. The component is *Not* made independent.
- If all bodies of a component are consumed by the Merge, the empty components are now removed.

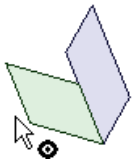
5.5.2.9. Merging Surfaces

You can merge surfaces with the **Combine** tool. The surfaces must share edges to be merged. If surfaces enclose a volume, you can use the **Combine** tool to quickly convert the enclosed volume into a solid and trim the extra.

Sometimes, if you manipulate a surface so that it becomes self-intersecting, you may need to create a solid from the surface that remains.

This section contains the following topics:

5.5.2.9.1. Merging Surfaces That Share Edges



1. Select **Intersect** > **Combine** .
2. Specify the **Combine tolerance**, if applicable.

3. Click the first surface.
4. Ctrl+click the second surface to merge it with the first surface.

   The second surface takes on the color and visibility properties of the first surface. The model tree shows the merge.

5.5.2.9.2. Merging Surfaces That Enclose a Volume

1. Specify the **Combine tolerance**, if applicable.
2. Select the surfaces that intersect to form an enclosed region using box selection.
3. Select **Intersect** > **Combine** .

   The enclosed region becomes a solid and takes on the color and visibility properties of the merged surface that appeared top-most in the model tree. The model tree shows the merge.

5.5.2.9.3. Creating a Solid from a Self-Intersecting Single Surface

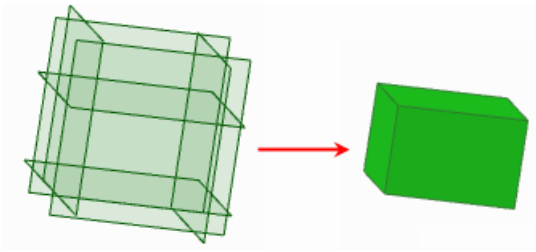
1. Click **Edit** > **Select**.
2. Select the "open" edge of the set of surfaces, shown in black.
3. Click the **Combine** tool.

5.5.2.9.4. Repairing a Self-Intersecting Surface

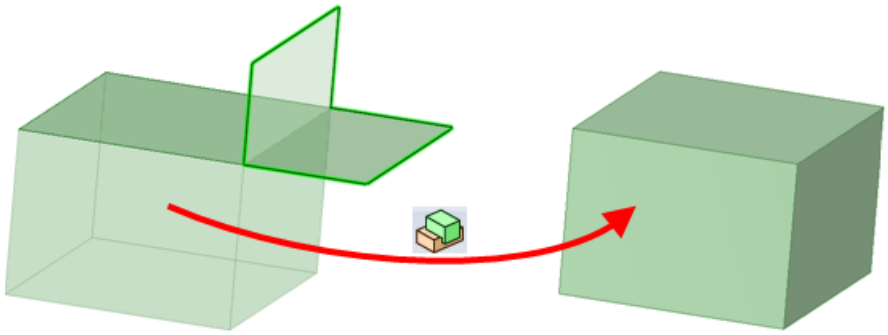
1. Click **Edit** > **Select**.
2. Double-click the edge loops.
3. Click the **Combine** tool.

5.5.2.9.5. Merging Surfaces Examples

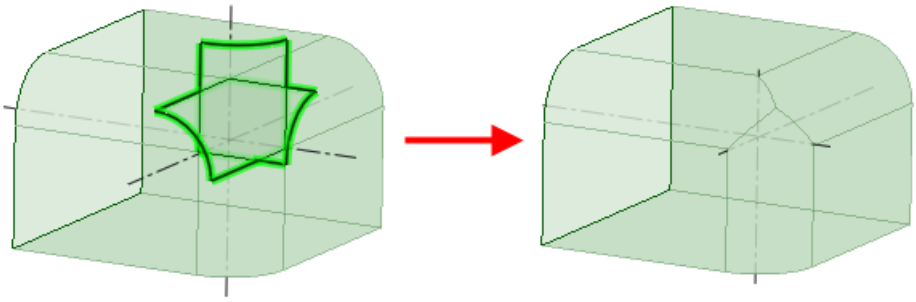
Six surfaces enclosing a volume, box-selected and combined with the Combine tool.



Repairing a self-intersecting surface with the Combine tool.

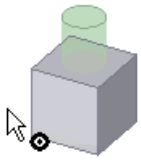


Ctrl+click to select an open edge loop of a surface, then close the surfaces with the Combine tool.



5.5.2.10. Merging Surface Protrusions

You can merge a surface that forms a protrusion to a solid.



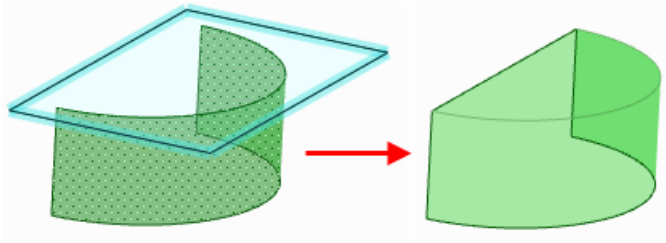
To merge a surface protrusion:

1. Select **Intersect** > **Combine** .
2. Specify the **Combine tolerance**, if applicable.
3. Click the first solid.
4. Ctrl+click a surface that could create a protrusion on the solid to merge it with the solid.

   The surface becomes a protrusion on the solid and takes on the color and visibility properties of the first solid. The model tree shows the merge.

5.5.2.11. Capping a Surface with a Plane

Ctrl+click a surface and a plane to cap the surface.

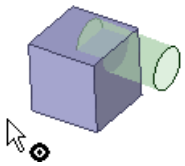


5.5.2.12. Removing Material From a Solid

Use the **Combine** tool to remove material from a solid based on the intersection of a solid or surface. You can remove the material defined by a surface that forms a depression, or remove a solid completely enclosed within another solid.

This section contains the following topics:

5.5.2.12.1. Removing Material From a Solid with a Surface That Forms a Depression



1. Select **Intersect** > **Combine** .
2. Specify the **Combine tolerance**, if applicable.
3. Click the solid you want to cut.
4. Click the surface you want to use to create a depression.
5. Mouse over the solid to see the regions created by the cut.
6. Click the region you want to delete.
   The regions you can delete are highlighted in red.

5.5.2.12.2. Removing an Enclosed Volume from a Solid

1. Create the exterior solid and interior solid in two different components.
2. Select **Intersect** > **Combine** .
3. Specify the **Combine tolerance**, if applicable.
4. Click the exterior solid.
5. Click the interior solid to use it as the cutter.
6. Click the interior solid to delete it.
   The regions you can delete are highlighted in red.

5.5.2.13. Splitting Bodies and Surfaces

Use the **Combine** tool to split bodies and surfaces.

1. Select **Intersect** > **Combine** or press **I**.
2. Specify the **Combine tolerance**, if applicable.
3. Select the body or surface you want to cut.

   The **Select Cutter** tool guide is activated.
4. (Optional) Control the behavior of the cut by selecting options.
5. Click the surface you want to use to cut the body.

   Depending on the option you chose, the cutting surface will be kept or deleted. Review the information in the Structure panel to see the bodies or surfaces created by the cut.

   Mouse over the target body to see the regions created by the cut.
6. Click each region you want to delete.

   When you are finished selecting the areas to be deleted, click the Select Target tool guide to combine some more, or select another tool.

> **Note:** If you want to split a body by a face of the body, select the **Split Body** tool.
>
> If you want to create an edge on a face, select the **Split** tool.

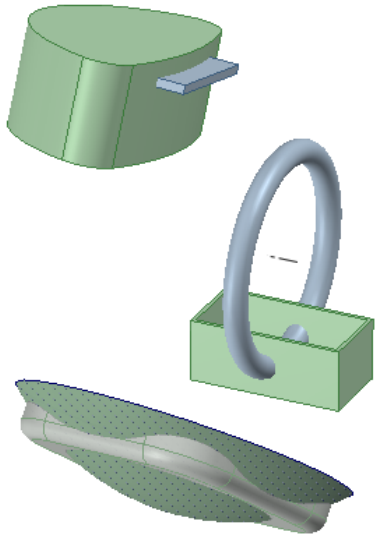
5.5.2.13.1. Splitting the Target Object with the Cutter Object and the Cutter Object with the Target Object

1. Select **Intersect** > **Combine** .
2. Specify the **Combine tolerance**, if applicable.
3. Select the **Make all regions** option.
4. Click the objects you want to cut.

   Target and cutter must be the same type of object, either both bodies or both surfaces.
5. Mouse over the surface to see the regions created by the cut.

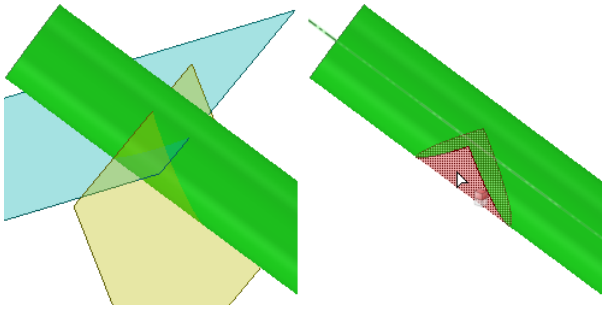
   Because this option can create a large number of regions, a best practice is to use this option along with the **Merge When Done** option to quickly merge all remaining regions when you click another tool or press **Esc** to finish using **Combine**.
6. Click the region you want to delete.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.



5.5.2.13.2. Splitting Bodies and Surfaces Example

Using two surfaces as the cutter, when the surfaces can't be merged outside of the region where they intersect the target. The two surfaces are merged to create a cut in the target.

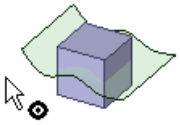


5.5.2.14. Splitting a Surface

Use the **Combine** tool to split solids. Surfaces can be split by solids, planes, patterns of planes, and other surfaces.

This section contains the following topics:

5.5.2.14.1. Splitting a Surface by a Solid or Plane



1. Select **Intersect** > **Combine** .
2. Specify the **Combine tolerance**, if applicable.
3. Click the surface you want to cut.
4. Click the solid, plane, or patterns of planes you want to cut the surface with.

5. Mouse over the surface to see the regions created by the cut.
6. Click the region you want to delete.

5.5.2.14.2. Splitting a Surface by Another Surface

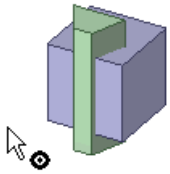
1. Select **Intersect** > **Combine** .
2. Specify the **Combine tolerance**, if applicable.
3. Click the surface you want to cut.
4. Click the surface you want to cut the surface with.
   You can Ctrl+click multiple surfaces that together completely intersect the target surface to fully cut the surface.
   You can box-select surfaces that only partially intersect the target surface to partially cut the surface.
5. Mouse over the surface to see the regions created by the cut.
6. Click the region you want to delete.

5.5.2.15. Splitting a Solid with a Surface, Plane, or Another Solid

Use the **Combine** tool to split solids. Solids can be split by surfaces, planes, patterns of planes, and other solids.

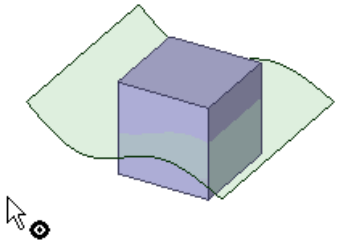
This section contains the following topics:

5.5.2.15.1. Splitting a Solid with a Solid



1. Select **Intersect** > **Combine** .
2. Specify the **Combine tolerance**, if applicable.
3. Click the solid you want to cut.
4. Click the solid you want to cut with.
5. Mouse over the solid to see the regions created by the cut.
6. Click the region you want to delete.
   The regions you can delete are highlighted in red.

5.5.2.15.2. Splitting a Solid with a Surface or a Plane



1. Select **Intersect** > **Combine** .
2. Specify the **Combine tolerance**, if applicable.
3. Click the solid you want to cut.
4. Click the surface you want to cut the solid with.
   You can Ctrl+click multiple surfaces that together completely intersect a solid.
   If available, you can select a pattern of planes from the model tree to cut the solid.
5. Mouse over the solid to see the regions created by the cut.
6. Click the region you want to delete.
   The regions you can delete are highlighted in red.

5.5.3. Split Body Tool

The **Split Body** tool works in two modes:

- *Slow mode* is activated by clicking a **Split Body** tool guide. In this mode, once a tool guide is clicked, it remains selected until another tool guide is clicked. This mode works just like any other tool.
- *Fast mode* automatically activates tool guides and moves you through the workflow. This predictive mode allows you to box-select multiple cutters to add even more efficiency.

You can use the **Split Body** tool to:

- Split a body by its edges
- Split a body by its faces
- Split a body by a plane
- Split a body by patterns of planes

You can select a body or surface from the model tree as the target and a plane or patterns of planes from the model tree as the cutter. When selecting patterns of planes as a cutter, you can also use Shift-Select or box select to select specific planes within the pattern.

This section contains the following topics:

5.5.3.1. Splitting a Body by One of its Faces

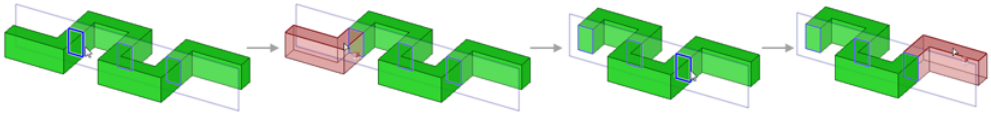
1. Click **Design** > **Intersect** > **Split Body** .
2. Select the body to be split.
3. Select the faces or edges you want to use to cut the body.
   Hold **Ctrl** and click or draw a box to select multiple faces or edges.
   You can create temporary geometry and select it to cut the body.
4. Mouse over the body to see the regions created by the cut.
5. (Optional) Click a region to delete it.
   When you are finished deleting regions, select another tool.
   If the selected edges do not completely encircle a portion of a body or surface, no region selection can occur.

> **Note:** Select a face with the Select tool, then select the Split Body tool to cut the body with the face.

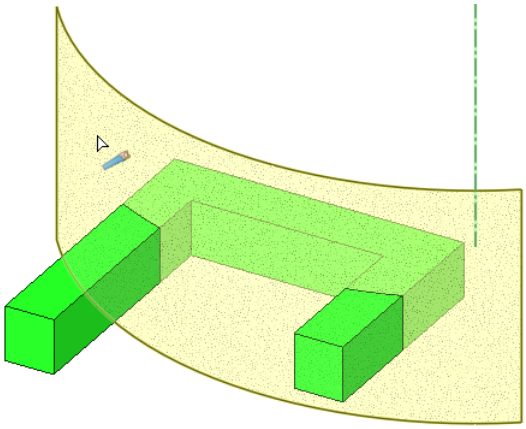
5.5.3.2. Splitting a Body Selectively

1. Click **Design** > **Intersect** > **Split Body** .
2. Select the body.
3. Check **ON** the **Local Slicing** option.
4. Click the cutter plane to see a blue outline which previews local slicing locations for splitting the body.
5. Click the blue outline preview to cut the body. The **Select Regions** tool guide is then activated however, holding the Ctrl key or clicking the **Select Cuts** tool guide allows you to create additional cuts.
6. Click a region to delete it.

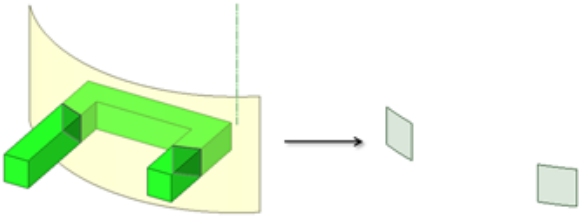
If you need to make multiple cuts on the same target body, manually click the **Select Cuts** tool guide to make it sticky before step 5.



A cylindrical surface can also be used for **Local Slicing**.



Use the **Create Split Surfaces** option to create surfaces at the intersection of the target and cutter.



5.5.3.3. Split Body Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Target	The **Select Target** tool guide is active by default. When this tool guide is active, click to select the body to split.
Select Cutter	The **Select Cutter** tool guide is active by default. When this tool guide is active, click to select the face you want to use to cut the body.

Select Cuts	The **Select Cuts** tool guide activates when the **Local Slicing** option is selected. Possible cuts are highlighted where the plane intersects the body. Mouse over the cuts to highlight them for selection.
Select Regions	The **Select Regions** tool guide activates once you once the body is cut by the face. When this tool guide is active, mouse over the target to see the regions created by the cut. The regions you can delete are highlighted in red.

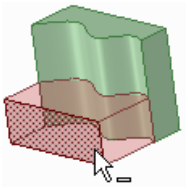
5.5.3.4. Split Body Tool Options

The following options are available in the Split Body tool. Once you select the edge or face to pull, select these options from the Options docking panel, or right-click and select them from the mini-toolbar:

Merge when done	Select this option to merge all touching bodies or surfaces when you exit the Split Body tool. Hidden objects are not merged.
Extend faces	Extends the selected cutter face to cut through the target body.
Create imprints	Create edge imprints when splitting a body by its edges, faces, a plane, or another body.
Local Slicing	Select this option with the **Select Cuts** tool guide to choose specific cuts made by a plane passing through the entire body, including those with a cylindrical surface. To **Create split surfaces**

5.5.3.5. Split Body Example

Body split by a spline face.



5.5.4. Splitting Faces or Edges

Use **Split** to split faces and edges. To create an edge on a face:

1. Click **Design** > **Intersect** > **Split** .

   The **Select Target** tool guide is enabled.

2. Select the face you want to split.
3. (Optional) Select a tool guide. Mouse over faces or edges in your design to preview the edge that will be created on the target. Dimensions are displayed along the U and V edge of the face.

> **Tip:** You can use the Tab key to cycle through the tool guide options, or with the pointer over the selected face, cycle through the dimensions.

4. Click to select the face, plane, or edge you want to use to split the selected face. You can select multiple faces and use box select.

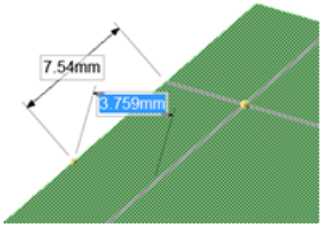
   You can hold **Shift** and hover over an edge while selecting the cutter points. The midpoint on the edge will be marked with a yellow ball in parentheses. The x and y dimensions will emanate from that point.

5. The **Select Results** tool guide remains active until there is only one split line remaining, so you can remove all but one split line without reactivating the tool guide.

This section contains the following topics:

5.5.4.1. Creating an Edge on a Face Example

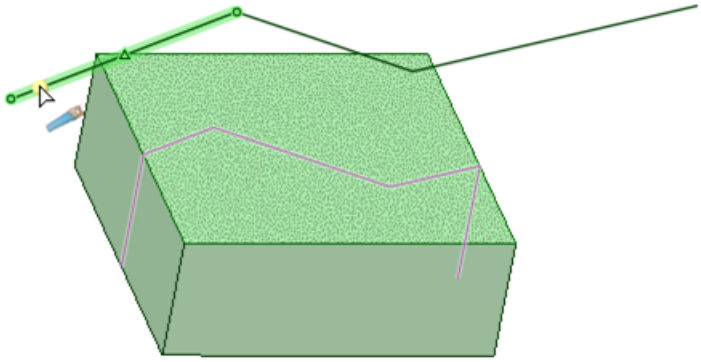
Dimensioning from the midpoint of an edge to split a face.



This section contains the following topics:

5.5.4.1.1. Splitting With Curves Outside of the Face

You can Split faces using Curves, whether they are touching the face or not. Connected curves can be used as the cutter by holding the **Shift** key. See the example.



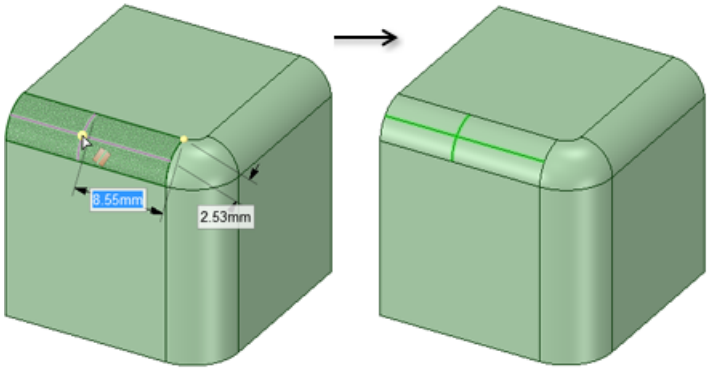
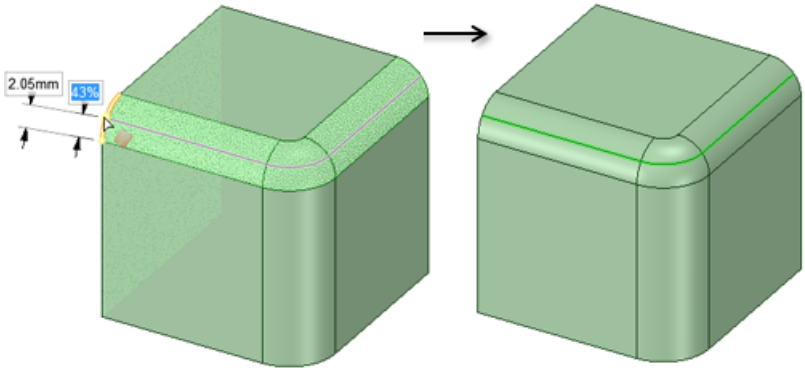
5.5.4.1.2. Splitting a Face With Another Face, Plane, or Patterns of Planes

1. Click **Design** > **Intersect** > **Split** .
2. Click the face to be split.
3. Click the **Select Cutter Face** tool guide.
4. Hover over the faces and planes in your design to preview the edge that will be created on the target. You can also select a patterns of planes in the model tree to split the face. When selecting patterns of planes as a cutter, you can also use Shift-Select to select specific planes within the pattern.
5. Click the face, plane, or patterns of planes to split the selected face with an edge.

5.5.4.1.3. Splitting a Face Using a Point on an Edge or Surface

Surfaces are parameterized using two variables labeled U and V. The location of each point on a surface can be described with coordinates (u, v). With the **Select UV Cutter Point** tool guide, you select a UV point on the surface or one of it's edges.

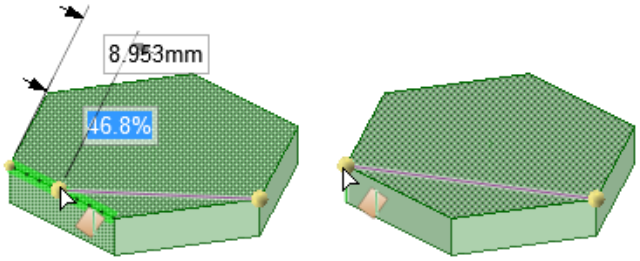
1. Click **Design** > **Intersect** > **Split** .
2. Click the **Select UV Cutter Point** tool guide.
3. Hover over a surface or it's edges to preview the split edges that will be created.
4. Click a point on the surface or edge to split the selected face.

Clicking a UV point on a surface splits it into four pieces.	
Clicking a UV point on an edge splits it into two pieces, along a constant U or V coordinate. For a chain of tangent faces, the split propagates across all tangent faces at the constant U or V coordinate.	

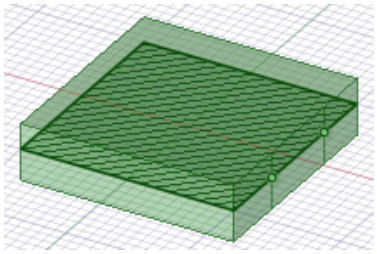
5.5.4.1.4. Splitting a Face Using Two Points

1. Click **Design** > **Intersect** > **Split** .
2. Click the **Select Two Cutter Points** tool guide and click a point on an edge.
3. Hover over the face or edges to preview the edges that will be created.
4. Click a point on another edge or on the face to split the selected face.

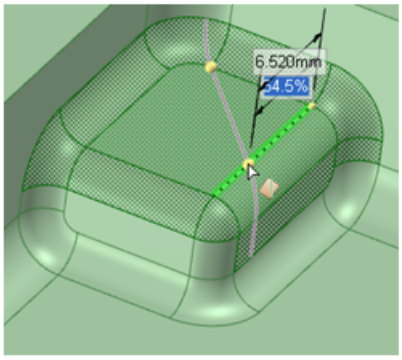
*Previewing edges that can be created using the **Select Cutter Point** and **Select Two Cutter Points** tool guides*



Splitting a face in section mode by selecting points on a section line



*Splitting multiple faces with the **Select Two Cutter Points** tool guide*

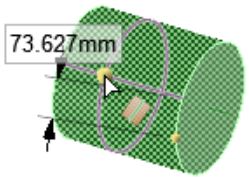


5.5.4.1.5. Splitting a Face Using a Perpendicular Line From a Point on an Edge

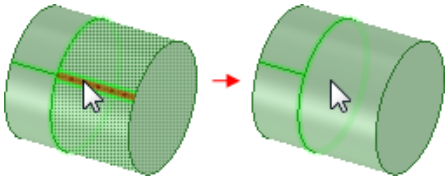
1. Click **Design** > **Intersect** > **Split** .
2. Click the **Select Perpendicular Cutter Point** tool guide and click a point on an edge.
3. Hover over edges of the face to preview the edges that will be created.
   The distance and percentage along the edge is displayed. You can press the **Space** bar and **Tab** key to change these values.

5.5.4.1.6. Splitting a Periodic Face

1. Click **Design** > **Intersect** > **Split** .
2. Click the **Select Target** tool guide and select the periodic face you want to split.
3. Click the point of the face where you want it split, as shown in the example.



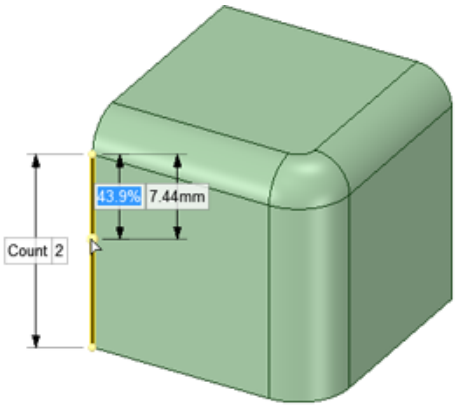
4. The **Select Results** tool guide is now active, so you can click edges that you don't want to keep, as shown on the left of the image below. The result after the two straight edges have been removed is shown on the right.

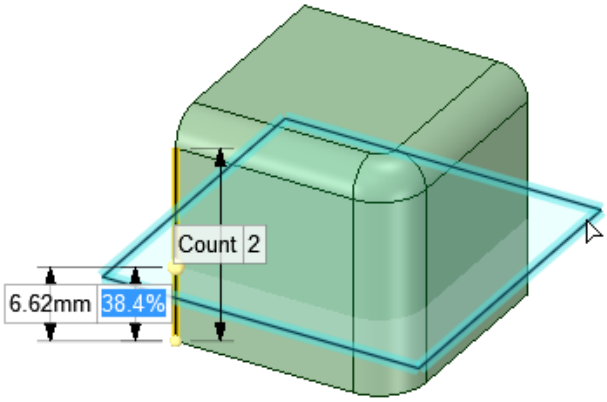


5.5.4.1.7. Splitting an Edge, Curve or Beam

1. Click **Design** > **Intersect** > **Split** .
2. Click the **Split Edge** tool guide and select the edge, curve, or beam you want to split.
   The edge highlights and the split is previewed with dimensions (length and percent) and a count. The count is the number of pieces the edge will be split into. The default count is two.
3. Click the point where you want to split.
   The **Select Results** tool guide is now active, so you can click edges that you don't want to keep.

Select edges that you don't want to keep as shown on the left of the example. The result after the two straight edges have been removed is shown on the right.





5.5.4.2. Split Face Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Target	The **Select Target** tool guide is active by default. If you did not pre-select the target face or surface, you can select it from within the Split tool using the Select tool guide. Ctrl+click multiple surfaces or solid faces in the same plane to split them all.
Split Edge	Use the **Split Edge** tool guide to split edges, curves, or beams.
Select Cutter Face	The **Select Cutter Face** tool guide activates once you select a target. When this tool guide is active, click to select the face or surface you want to use to create an edge on the target.

Select UV Cutter Point	The **Select UV Cutter Point** tool guide activates once you select a target. Mouse over an edge to preview the new edges that will be created. Click to create the edge on the selected face. You can mouse over an edge with this tool guide to display and edit the length along the edge, and the percentage of the edge that is between the first point and the end point.
Select Perpendicular Cutter Point	The **Select Perpendicular Cutter Point** tool guide activates once you select a target. Select an edge to preview where the perpendicular split will be made. Click to create the edge on the selected face. The face is split perpendicular to the edge you select.
Select Two Cutter Points	The **Select Two Cutter Points** tool guide activates once you select a target. Click to select the first point on an edge, then mouse over another edge to preview the new edge that will be created. Click to create the edge on the selected face. The shortest line is drawn between the two points. You can mouse over an edge with this tool guide to display and edit the length along the edge, and the percentage of the edge that is between the first point and the end point.
Select Results	Select the **Select Results** tool guide, then click newly created edges to remove any edges you don't want to keep.

5.5.4.3. Split Tool Options

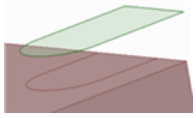
The following option is available in the **Split** tool:

Make Curves	Select this option to create sketch lines instead of splitting the face.

5.5.5. Project Tool

Use the **Project** tool to create edges on a solid's face by extending the edges of another solid, surface, sketch, or note text.

When you project onto a face, the tool makes a new surface to hold the projected edges, if possible.



This section contains the following topics:

5.5.5.1. Projecting the Edges of a Face, Surface, Sketch, or Note Text to a Solid

1. Click **Design** > **Intersect** > **Project** .
2. Select the edges, face, surface, sketch, or note text whose edges you want to project.

   Select an object in the model tree to project all the edges of the object.

   The projection occurs perpendicular to the selected object onto the nearest solid faces. The nearest solid faces are determined automatically.
3. (Optional) Click the **Select Direction** tool guide and select a face or edge to change the direction of the projection.
4. (Optional) Click the **Select Target Faces** tool guide and select the face(s) on which you want the edge projected.
5. Select from the following options in the Options panel:

- **Project through solids**: Project the edges on all faces through the entire solid instead of just the faces closest to the object you project.
- **Project silhouette edge** : Projects the outline, or silhouette, of a part. You must set the direction using the **Select Direction** tool guide.
- **Extend projected edges**: Extend the projected curves to the edge of the face(s) they are projected on.
- **Extend target faces**: Extend the target face when the projected face is larger than the target. See the example below.

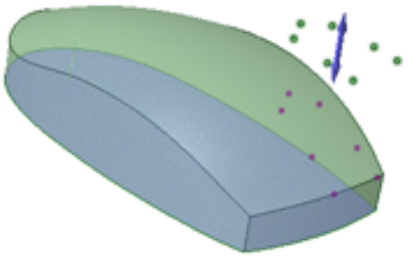
The projection preview is shown in purple, and will be updated based on the options and tool guides you use.

6. Click the **Complete** tool guide or press **Enter** to project the edges.

5.5.5.2. Projecting Points

Sketched 3D Points can also be projected onto surfaces using the **Project** tool. The projected points are created as Datum Points so that they can be pinned.

In the example below, points were sketched to trace out the shape of a mouse button and then projected down onto the top surface of the mouse. The workflow is the same as projecting Faces, Surfaces, Curves, Edges, or Note Text.



5.5.5.3. Project Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Curves	The **Select Curves** tool guide allows you to select the curves you want to project.
Select Direction	The **Select Direction** tool guide allows you to select the direction in which the curves will be projected. If you don't use this tool guide, the curves will be projected in both directions.
Select Target Faces	The **Select Target Faces** tool guide allows you to select the face(s) that the curves will be projected upon.
Complete	The **Complete** tool guide projects the edges.

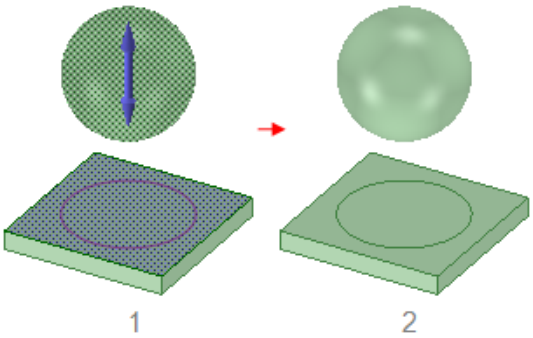
5.5.5.4. Project Tool Options

The following options are available in the Options panel:

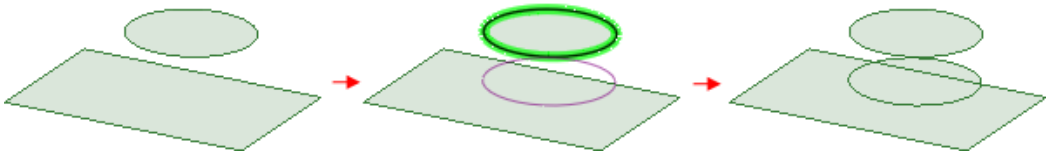
Project through solids	Project the edges on all faces through the entire solid instead of just the faces closest to the object you project.

Project silhouette edge	Projects the outline, or silhouette, of a part. You must set the direction using the **Select Direction** tool guide.
Extend projected edges	Extend the projected curves to the edge of the face(s) they are projected on.
Extend target faces	Extend the target face when the projected face is larger than the target. See the example below.

5.5.5.5. Projecting Examples



Projecting a sphere onto a planar face. Because a sphere has no edge to project, you must choose **Project silhouette edge** in the Options panel.



Projecting an edge with the **Extend target faces** option enabled. The rectangular face was chosen with the **Select Target Faces** tool guide.

5.6. Datum Objects and Relationships

This section contains the following topics:

5.6.1. Inserting Planes

You can define a plane by selecting various faces, edges, axes, or lines in your design. You can select any combination of coplanar axes, curves, edges, and coordinate axes.

The plane will extend a small amount beyond the edges of the face to make it easier to work with. The plane will maintain this margin around all the objects in your design, extending and trimming itself as you add, remove, or move solids within your design.

Datum planes adjust their extent based on zoom level and objects within the plane, including curves. The **Include Curves** property, set to true by default, is especially useful for beam-centric models with many curves.

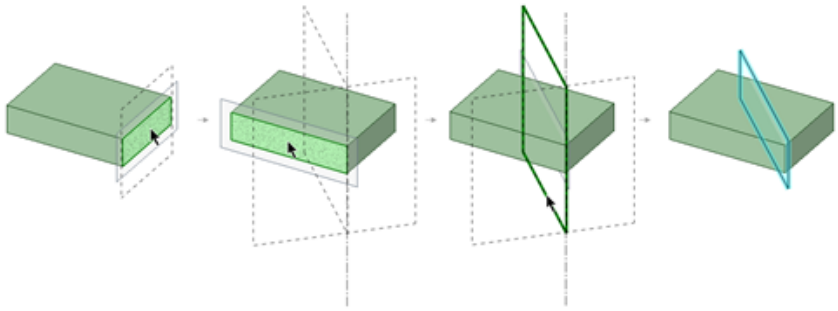
This section contains the following topics:

5.6.1.1. Inserting a Plane

1. Click **Plane** in the Create group on the Design tab.
2. Select one of the following:

Select a:	**To insert a plane:**
Planar face	That contains the face
Planar face and point	Through the point and parallel to the face
Planar face and edge	Through the edge and perpendicular to the face (useful when creating a sweep path)
Two parallel planar faces	At the midpoint between the two faces
Axial or cylindrical face	Tangent to the face at the selection point
Two axial faces with parallel axes	Tangent to both faces as close as possible to the selection points
Planar edges	That contains the edges
Axis	That contains the axis
Two axes	That contains the axes
Two origin axes	That contains the axes
An origin	Three orthogonal planes
Axis (or line) and one point	That contains the axis (or line) and the point
End point of any line	Through the end point and normal to the line at the end point
Control point of a spline	Through the selected point and perpendicular to the plane that contains the spline
Three points	That contains those three points
Line sketched on a plane	Through the line and normal to the plane
Line and point on the line	Through the line and the point
Two planar lines	That contains those two lines
Two points	Through the two points
Two points and a Plane or planar face	Through the two points and perpendicular to selected plane or planar face

3. (Optional) Click the **Build Plane** tool guide to select reference objects, then select a temporary (dotted) plane to create the one that you want from the possible planes. See Inserting temporary points, axes, and planes for detailed information about creating temporary planes.



An error message is displayed in the status bar if your selection does not define a plane.

- To insert a plane tangent to a cylinder and parallel to a plane, select a plane, **Ctrl**+click a cylinder, then click the **Plane** tool.
- To insert a plane tangent to a cylinder and perpendicular to a plane, select a cylinder, **Ctrl**+click a plane, then click the **Plane** tool.

5.6.1.2. Inserting a Plane Parallel to the Screen

1. Select the **Plane** tool.
2. Activate the **Align to Screen** tool guide .
3. Hover over geometry to preview the plane through the selected geometry and parallel to the screen.
4. Select the geometry to create the plane.

5.6.1.3. Inserting a Plane Through the Mid-point Between Two Points

1. Select the **Plane** tool.
2. In **3D** mode or **Section** mode, Alt+Shift+click two points to create a temporary plane that includes the midpoint.
3. Click the temporary plane.

5.6.1.4. Clipping Your Design with a Plane

Right-click a plane and select **Clip with Plane**. To restore the view of your design, right-click the plane and select **Clip with Plane** again.

Planes, axes, and annotations are not clipped.

5.6.1.5. Changing the Appearance of a Plane

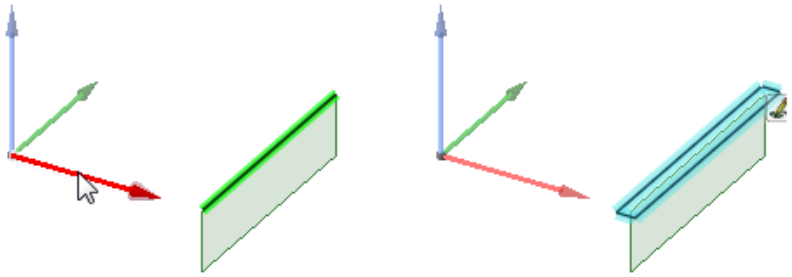
You can use the **Fill, Outline**, and **Clipface** options to change the appearance of a plane.

- **Fill:** Set to **Automatic** by default. **Automatic** causes geometry behind the plane to look lighter than geometry outside the plane boundaries. If you change to **No Fill**, all geometry will have the same opacity.
- **Outline:** Set to **Automatic** by default. For **Automatic** you see the outline of the plane. If you change to **No Outline**, the plane is invisible unless selected.
- Setting planes to **No Fill** and **No Outline** is useful when dimensioning parts. It is easier to see annotations when planes are less visible.
- **Clipface:** Sets the clipface color override for each clipping plane in the design window. It does not change the color of the plane itself, just the color of the face the plane is touching.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

5.6.1.6. Inserting a Plane Example

An edge and the axis of a coordinate object selected to create a plane.



5.6.2. Inserting Axes

You can extend any axis in your design to make it available for revolves or rotations. The axis will extend a small amount beyond the edge of the space containing your design. It will maintain this margin extending and trimming it as you add, remove, or move objects in your design. Axes are not linked to the geometry used to create them (unless they are an extension of existing axes); they must be explicitly selected to move along with other items.

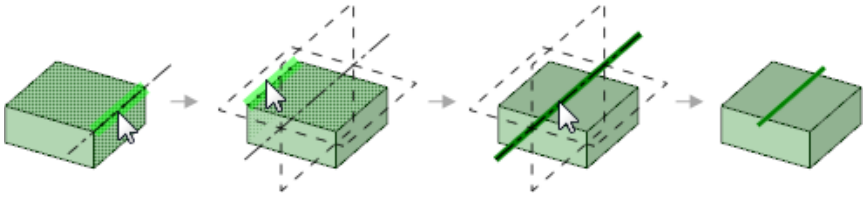
This section contains the following topics:

5.6.2.1. Inserting an Axis

1. Click **Axis** in the Create group on the Design tab.
2. Select one of the following:

Select:	To insert an axis:
Axial face	That contains the face's axis
Two, non-parallel planar faces	At the intersection of the theoretical planes containing the two faces
Linear edge	That contains the edge
Cylinder and tangent plane	At the intersection of the cylinder and plane
Line	That contains the line
Origin axis	Along the origin's axis
Circle or arc	That contains the center point and is normal to the line
Point and a plane	Through the point and normal (perpendicular) to the plane
Two points	Through both points
Three axes	At the point where a cylinder's axis would be positioned between the three axes

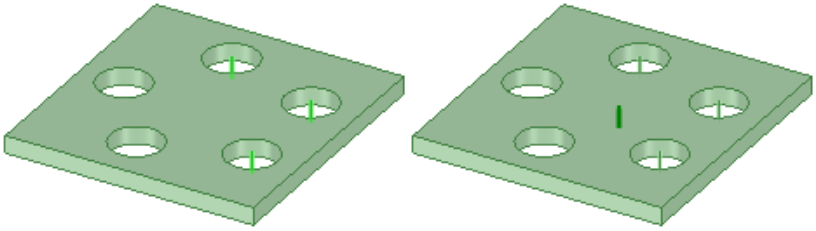
3. (Optional) Click the **Build Line** tool guide to select reference objects, then select a temporary (dotted) line to create the one that you want from the possible axes. See Inserting temporary points, axes, and planes for detailed information about creating temporary lines.



5.6.2.2. Inserting an Axis Through the Mid-point Between Two Points

1. Click in the Edit group on the Design tab.
2. Hold Alt+Shift and select two points to create two temporary axes that cross at the midpoint.
3. Click the temporary axis.
4. Click **Axis** in the Insert group on the Design tab.

5.6.2.3. Inserting an Axis Example



Creating an axis by selecting the axes of three cylinders

5.6.3. Inserting Points

You can create 3D points anywhere on the 3D geometry. Alternatively, you can provide a precise location to insert a point.

1. Click **Point** in the Create group on the Design tab.
2. Select a point, vertex, or location on a face, surface, curve, or edge.
   - World Origin coordinates are shown for selected Datum Points in the Status Bar.
   - Points can be patterned in the **Move** tool and **Linear** and **Circular Pattern** tools.
   - Points can be dragged and dropped in the model tree.
   - **Move to New Component** and **Move Each to New Component** work for selected Points.

5.6.4. Inserting Coordinate Systems

You can insert a cartesian or cylindrical coordinate system at any location in your design where you can anchor the **Move** tool. You can anchor a ruler to a coordinate system, and snap to coordinate systems while in Sketch mode. This allows you to dimension from a coordinate system and quickly move the sketch grid to a coordinate system. You can also insert a coordinate system at a solid's center of mass or volume.



This section contains the following topics:

5.6.4.1. Inserting a Coordinate System

1. Click **Coordinate System** in the Create group on the Design tab.
   A coordinate system is previewed attached to the cursor.
2. From the Tool Guides, select whether you want to insert a cartesian or cylindrical coordinate system.
3. Move the cursor to position the coordinate system. The coordinate system will adjust its orientation based on the type geometry it is on.
4. Click to place the coordinate system.

5.6.4.2. Inserting a Coordinate System Based on Selected Faces

1. Select one, two, or three faces.
2. Click **Coordinate System** in the Create group on the Design tab.

   The coordinate system is created based on the following:

   - If one planar face is selected, the coordinate system is created at the center of the face.
   - If two intersecting faces are selected, the origincoordinate system is created with XY in the first plane, Y aligned in the intersection of the two planes.
   - If three intersecting faces are selected, the coordinate system is created with XY in the first plane, Y aligned in the second, and Z placed at the intersection of the third plane with Y.
3. From the Tool Guides, select whether you want to insert a cartesian or cylindrical coordinate system.

5.6.4.3. Inserting a Coordinate System Using the Move Tool

1. Click **Move** in the **Edit** group on the **Design** tab.
2. Position the move handle where you want the coordinate system to appear.
3. Select the **Coordinate System** tool in the Create group to insert a coordinate system.
   The coordinate system appears in the graphics window and in the model tree.
4. From the Tool Guides, select whether you want to insert a cartesian or cylindrical coordinate system.

This section contains the following topics:

5.6.4.3.1. Inserting a Coordinate System at the Mid-point Between Two Points

1. Select the **Move** tool.
2. Alt+Shift+click two points to create a temporary point at the mid-point.
3. Click the temporary point to place the Move handle there.
4. Select the **Coordinate System** tool in the Create ribbon group.
   The coordinate system appears at the midpoint.

5. From the Tool Guides, select whether you want to insert a cartesian or cylindrical coordinate system.

5.6.4.3.2. Inserting a Coordinate System at the Center of Mass or Volume

1. Click the **Mass Properties** tool from the **Inspect** group on the **Measure** tab.
2. Select the solid at whose center you want to place the coordinate system.
   Axes appear at the center of mass or volume.
3. Select the **Coordinate System** tool in the Create ribbon group.
   The coordinate system appears at the center of mass or volume.
4. From the Tool Guides, select whether you want to insert a cartesian or cylindrical coordinate system.

5.6.4.3.3. Placing the Sketch Grid on a Coordinate System

1. Select the vertex of a coordinate system.
2. Enter sketch mode.
   The sketch grid is now located on the x and y axis of the selected coordinate system.

5.6.4.3.4. Defining the Coordinate System for a Newly Created Component

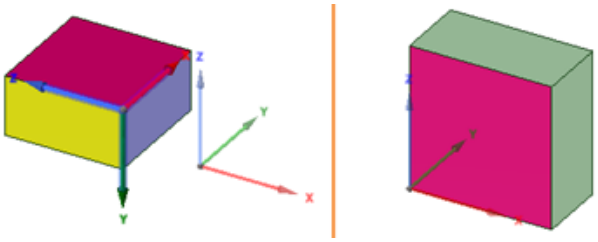
1. Insert a coordinate system.
2. Alt+click the coordinate system.
3. Right-click a solid and select **Move to New Component** from the context menu.

   You can also create a new component and then drag and drop the solid into that component while the coordinate system is Alt-selected.

4. Right-click the component in the model tree and select **Open Component**.
   The component opens in a new design window.
5. Display the world origin.
   The world origin in the new design window is positioned at the same location as the alt-selected coordinate system.

5.6.4.3.5. Inserting a Coordinate System Example

Alt+selecting a coordinate system when creating a new component. The image on the left shows the World Origin and a coordinate system created on the solid. The image on the right is the component opened in a new design window with the world origin in the location it was placed on the object.



5.6.5. Pinned Datums

Datums can be **Pinned** to the location at which they were created using **RMB** > **Pinned**.

Pinned Datums will move if their creation references are moved.

If a Datum is created and subsequently moved from its original (created) position (with respect to its creation references) **Pinned** will be grayed out.

The following apply to all pinned datums:

- Pinned datum icons in the tree are highlighted to distinguish them from normal.
- Ctrl+drag to copy in the **Move** tool works for all pinned datums. The original datum remains pinned but the new copy is unpinnable.
- Datum Planes and Axes are now automatically unpinned when they can no longer be constructed from the reference objects. For example, when a reference face is changed from a plane to a cylinder.
- The Datum Plane, Line, and Point tools are sticky (they stay on until you enter another tool) like normal tools.
- If any of a Pinned Datum's references are deleted, the system forces the datum to be unpinned. The datum will not be placed with respect to the remaining references. This applies to Planes, Lines, and Points.
- Datum Planes, Lines, and Points that are constructed exclusively from other pinned datums become pinned by default. For example, a datum axis will be pinned by default if it is constructed from a point pinned to a face.
- The Move handle is disabled when all selected objects are pinned datums (axes, planes). Since they are pinned, these objects cannot be Moved. If unpinned datums, or other objects, are included in the selection, the Move handle is enabled, but the pinned datums will not Move. To allow copying, the Move handle becomes enabled when the **Ctrl** key is pressed.

The following apply to pinned datums in components:

- You can Cut/Copy/Paste by parent component.
- You can set the Visibility by parent component.
- Note that when pinned datums are moved to a new component, they become unpinned and subsequently unpinnable
- Datums can only be pinned if *All* of the references are in the same component. Allowing references from multiple components can result in unintentional Assembly-like constraints.

This section contains the following topics:

5.6.5.1. Pinned Planes

- In the graphics area, pinned datums are also given a green color and they have their corners rounded. The image here shows the plane selected to emphasize the color.
- The pinned datum reference is also highlighted when the plane is selected.
- If any of the pin references are deleted, the plane is automatically unpinned.
- You can NOT pin datum planes created from temporary references (using ALT+SHIFT.)
- Only datum planes that are NOT moved from their original position can be pinned. Planes moved away, and then back to their creation references, CANNOT be pinned.
- Planes created using the **Build Plane** tool guide cannot be pinned. The references used in creating these planes are temporary and are not remembered outside of the tool.

5.6.5.2. Pinned Axes

- Un-pin a datum line if any of the references are deleted.
- Pinned axes are given a green color to distinguish them from normal axes.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

- Pinned axes attached to pinned datum points on faces will adjust when the faces are edited with **Pull** > **Offset**, **Move**, **Bend**, etc.
- Axes created using the **Build Line** tool guide cannot be pinned. The references used in creating these planes are temporary and are not remembered outside of the tool.

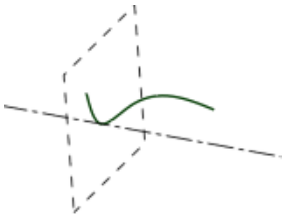
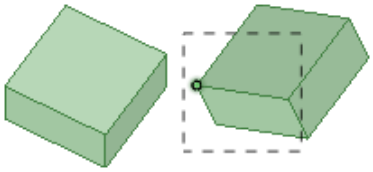
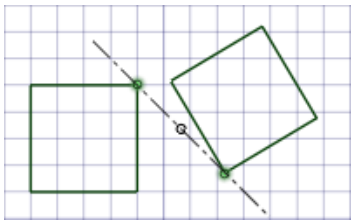
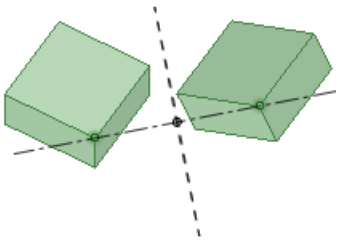
5.6.5.3. Pinned Points

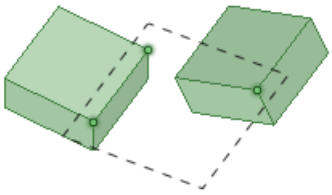
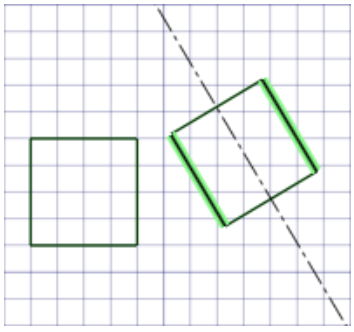
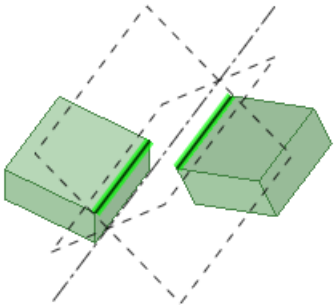
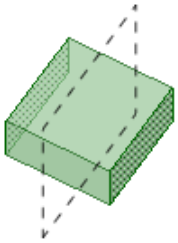
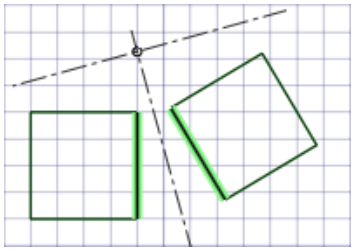
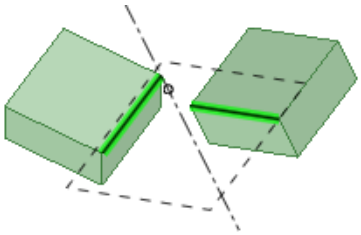
- Currently pin to an existing point/vertex, or to the midpoint between two other points.
- Face and Edge point connections have been added.
- You can now box-select Points
- Pinned Datum Points now store their latest location to ensure proper placement if the pin reference becomes invalid by deletion or geometry change.
- 3D Datum Points can now be moved with the Move tool if they are not pinned.
- When you Bend a face that has pinned datum points, the points follow the face at a well-chosen location.

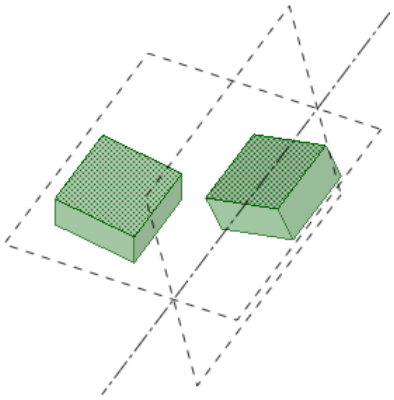
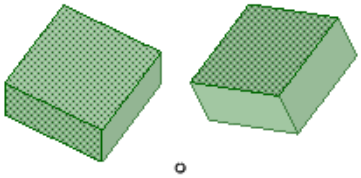
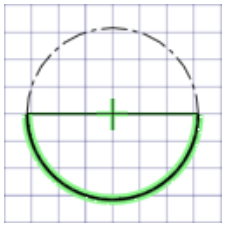
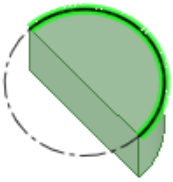
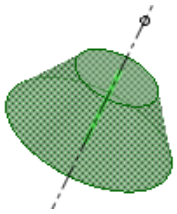
5.6.6. Inserting Temporary Points, Axes, and Planes

You can create temporary points, axes, and planes while working with other tools. Temporary objects can be used to dimension to and create other objects. For example, you can use them to create a ruler dimension, or anchor the Move handle on them.

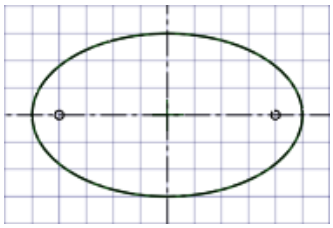
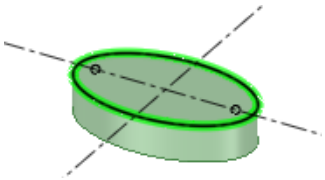
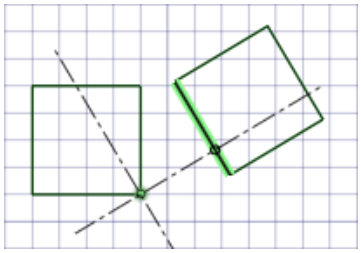
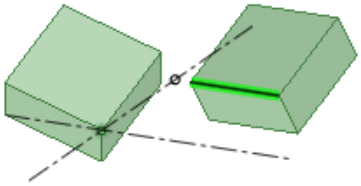
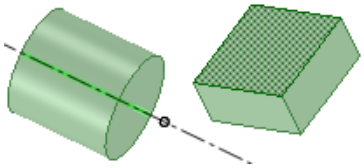
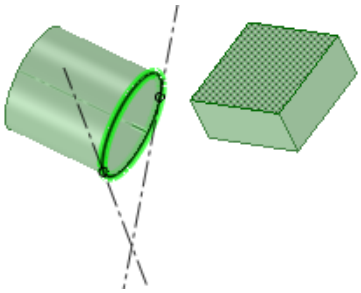
To clear a temporary object, select the object and press Delete. A temporary object also disappears when you click in an empty space with the Selection tool or create another temporary object.

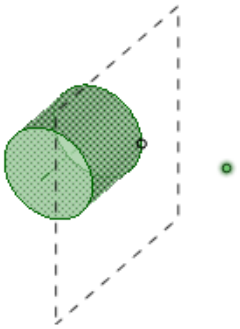
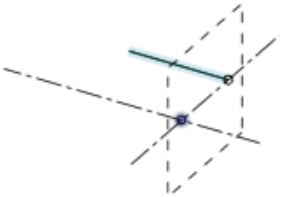
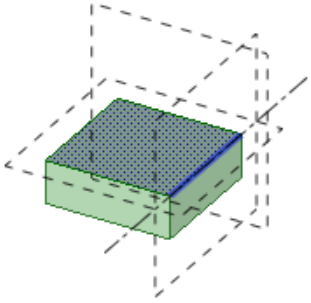
Alt+Shift+click	**2D Example**	**3D Example**
A point	*For curves: A tangent line and a perpendicular plane.* 	*For solids: A plane that is normal to the view direction (or parallel to the plane of the screen).* 
Two points	*2D: A midpoint between two points and a bisecting line.* 	*3D: A midpoint between two points and a bisecting line.* 

Alt+Shift+click	2D Example	3D Example
Three points	*Not applicable in 2D*	*A plane.* 
Two parallel lines or edges	*A line midway between two lines (2D).* 	*A line midway between two edges and planes that intersect the edges and the line (3D).* 
Two parallel planes	*Not applicable in 2D*	*A plane midway between two planes.* 
Two non-parallel lines, edges, or axes	*Two bisector lines and the intersection point (2D).* 	*An axis between the two points at the closest approach between each other, and a point and a midplane at the middle distance along that axis (3D).* 

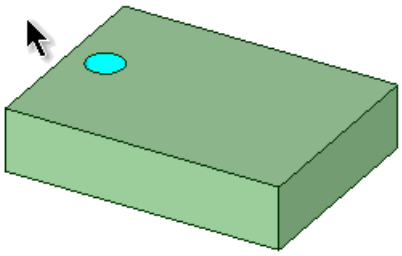
Alt+Shift+click	2D Example	3D Example
Two non-parallel planes	*Not applicable in 2D*	*Two bisecting planes and the edge where the planes intersect.* 
Three planes	*Not applicable in 2D*	*The point where the three planes intersect* 
An arc	*2D: A circle that completes the arc.* 	*3D: A circle that completes the arc.* 
A conical face and its axis	*Not applicable in 2D*	*The point where the conical face would come to a point.* 

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Alt+Shift+click	2D Example	3D Example
Ellipse	*2D: The major and minor foci.* 	*3D: The major and minor foci.* 
A line or edge and a point that is not in line with the line or edge	*Axis from the point normal to the edge and a second axis parallel to the edge through the point.* 	*Axis from the point normal to the edge and a second axis parallel to the edge through the point.* 
An axis and an intersecting face or surface	*Not applicable in 2D*	*The point at the intersection of the axis and the plane of the face.* 
A curve and an intersecting face or surface	*Not applicable in 2D*	*The points where the face would intersect the curve, and the axes that pass through the points, are tangent to the curve, and are on the same plane as the curve. One point and axis are created if the face would only intersect the curve at one point.* 

Alt+Shift+click	2D Example	3D Example
A point and a face or surface	*Not applicable in 2D*	*Plane that is tangent to the face and a point that is normal to the axis from the face to the point.* 
A point and a line or edge	*2D and 3D produce the same result, shown on the right.*	*A plane through the line or edge, or a plane that is normal to the plane that goes through the point. If a line is chosen at a particular point, then that location on the line can also be used as a point reference.* 
A line or edge and a planar face	*Not applicable in 2D*	*Planes that are perpendicular to the face and go through the line or edge.* 

5.6.7. Linear Patterns



You can create patterns of:

- Protrusions or depressions (including slots)
- Points, planes, origins, or axes
- Sketch curves
- Notes
- Lightweight text
- Components

Patterns can be made from a mix of object types, such as a pattern of holes (faces) and bolts (imported components). You cannot create a pattern of threads. You can use the **Undo** button, **Ctrl+Z** or the **Redo** button, **Ctrl+Y** when creating a pattern.

This section contains the following topics:

5.6.7.1. Creating a One- or Two-dimensional Linear Pattern

1. Click the **Linear** tool in the Create group on the Design tab.
2. Select a protrusion, depression, body, sketch, points, axes, planes, origins, or 3D curves to be the first member (leader) of the pattern. The **Direction** tool guide activates.
3. Select a line, edge, axis, or a set of points to set the direction of the pattern.
4. (Optional) Modify the Pattern options in the Options panel.

   Pattern Type: One-dimensional or Two-dimensional.

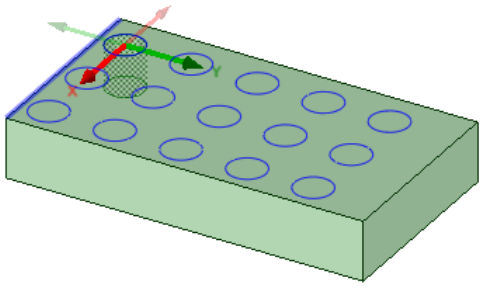
   Change **X Count**, or **X Pitch** values for One-dimensional

   Change **X, Y** Count, or **X, Y** Pitch values for Two-dimensional

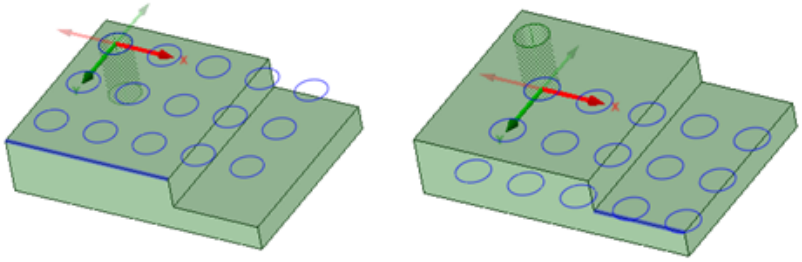
   As you switch between pattern types, or change pattern values, the pattern preview displays your changes in blue.
5. Select the **Define Pattern** tool guide.
6. Select the **Create Pattern** tool guide, or press **Enter**, to complete the pattern.

When the pattern is previewed, any members which cannot be created are shown in red. Members that can be created are shown in blue.

The image below shows the preview for a two-dimensional pattern. For one-dimensional, only the X-direction is displayed. Clicking the arrows flips their direction 180-degrees.



For through-all cuts, the preview is on the plane closest to the direction reference. For blind cuts, the preview is on the face of the selected pattern leader.



You can also create one-dimensional linear patterns on cylindrical faces as long as the pattern direction is either the cylinders axis or parallel to the axis.

In Discovery, any pattern member can be used to modify the pattern after you create it. If the change cannot be made to all pattern members, the member that cannot change is still part of the pattern.

When you select a member of a pattern, you will see fields for the pattern count and distances relative to the member you have selected. Press **Tab** to move among the dimension and count fields.

- Icons next to the pattern's dimensions and counts show which will be changed if you change the value of the highlighted field. The closed lock indicates that a dimension won't change and the open lock shows you it will change. You can override a lock on a dimension by clicking its icon to toggle it between locked and unlocked.
- Dimension or count line placement, arrows, and value fields show you how the pattern will change if you change the value of a field.
- Your selection determines how changes to the distance and count affect the pattern's position. If you select one member of the pattern, the change is centered on that member. If you select all pattern members, the change is centered on the entire pattern, with the distance between members locked by default. You can control which direction the pattern will grow based on which member you select and which count or distance you change.
- Changing a pattern count in one direction does not change the distance between pattern members. Instead, the overall distance of the pattern will change. This is indicated by the closed lock icon.

> **Note:** Patterns of sketch entities and 3D curves are no longer patterns when they change into another form, such as a surface or solid. For example, if you pattern a C-shaped curve, then it will remain a pattern when you switch from sketch mode to 3D mode. But if you close off the curve to make a box before you switch to 3D mode, then the boxes will become surfaces and will no longer be a pattern.

5.6.7.2. Creating a Pattern of a Pattern

Follow the steps for the linear pattern, but select a member of a linear pattern as the first member of the pattern.

5.6.7.3. Editing Linear Pattern Properties

1. Select one pattern member to display the pattern count and dimensions.
2. Edit the pattern's properties.

   Press **Tab** to switch between the fields.

   Click the lock icon to lock or unlock a value.
3. Press **Enter**.

   The result of editing the count and spacing is relative to the member of the pattern you select.

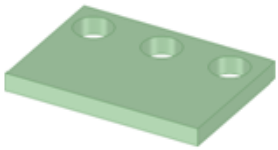
> **Note:** To quickly increase or decrease the number of pattern members, press **Tab** until the count field is highlighted, then hold the **Alt** key while you press the up or down arrow.

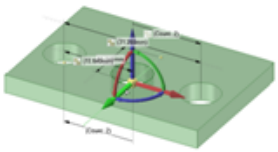
This section contains the following topics:

5.6.7.3.1. Moving a Pattern

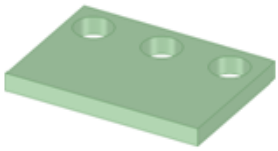
1. Click the **Move** tool.
2. Select a pattern member to move a linear pattern or the pattern axis to move a circular or arc pattern.
3. Move the pattern member with the Move handle.

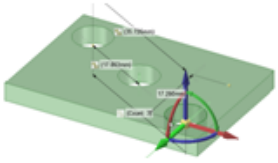
   If you move an interior member of a pattern and it is not anchored, all the pattern members move:





   If you move a member at one end of a pattern, the member at the opposite end is anchored and the pattern is skewed:





   If you anchor a different member than the member opposite the direction you are moving, Move skews the pattern.

   If you have a linear pattern in a radial direction and you move an interior member without setting an anchor, then the entire pattern shifts in the selected direction

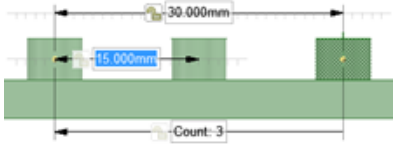
   Use the **Up To** tool to move a pattern member up to another face or edge. Pattern dimensions (such as Count and Length) display as expected.

5.6.7.3.2. Adjusting Pattern Distance or Spacing

1. Click **Select** or **Move** in the **Edit** group on the **Design** tab.
2. Select a pattern member.

   This member will serve as the anchor, and other pattern members will move relative to this member.
3. Press **Tab** until the field you want to change is highlighted:
   - **Distance**: This field is the overall distance of the pattern. The pattern member you select is anchored, and the length will change relative to this member. The arrows indicate the direction of change. You can see this field at the top of the image below.
   - **Spacing**: This field is the spacing between pattern members. This field has arrows in both directions and is located between two pattern members. The field is highlighted in blue in the image below.
   - **Radial patterns**: If you select a member of a radial pattern, you can change the angle between pattern members and the distance from the pattern members to the center of the pattern.
4. Type a new distance or angle and press **Enter**.

   The spacing between all pattern members will change.



5.6.7.3.3. Removing a Pattern Member from the Pattern

- Right-click the face of the pattern member and select **Unpattern Member**.
- Select the member components in the tree, then right-click and select **Source** > **Make Independent** from the context-menu.

Doing this makes the member independent, so changes to the feature won't propagate to the pattern.

5.6.7.3.4. Creating a Pattern Group

1. Select a pattern or pattern member in the Design window.
2. Click the **Groups** tab in the **Structure** panel.
3. Click **Create NS**.

   A group is created that displays the number of members in the pattern. You can click this group to select the pattern and change its parameters in the Design window.

5.6.7.3.5. Assembling Components on a Pattern

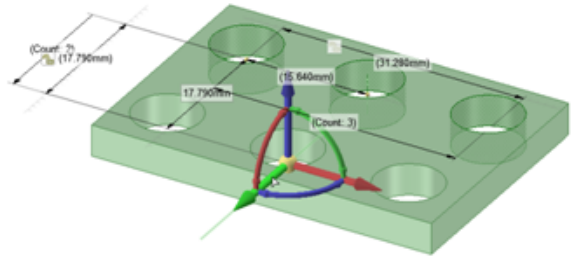
You can assemble a component with a pattern member, then propagate the component to all of the pattern members. The component will be copied and assembled to each pattern member.

1. Assemble a component to some geometry that is a pattern member using the **Align**, **Tangent**, or **Orient** tools.
2. Right-click the Assembly Condition in the **Design** tree and select **Update Components on Pattern**.

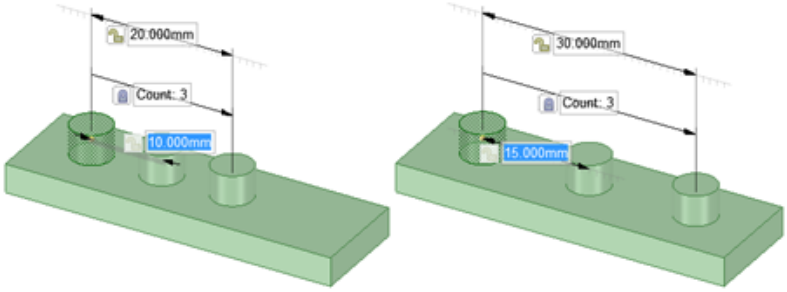
   The components will be propagated to the pattern.

5.6.7.3.6. Linear Pattern Examples

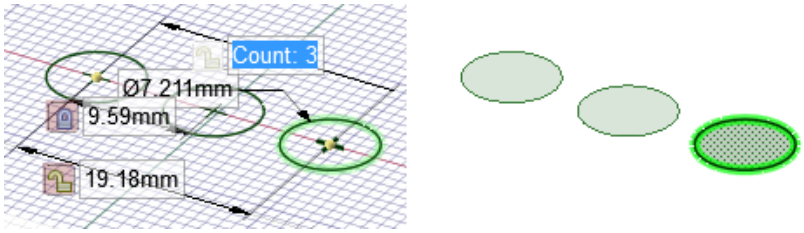
Rectangular pattern.



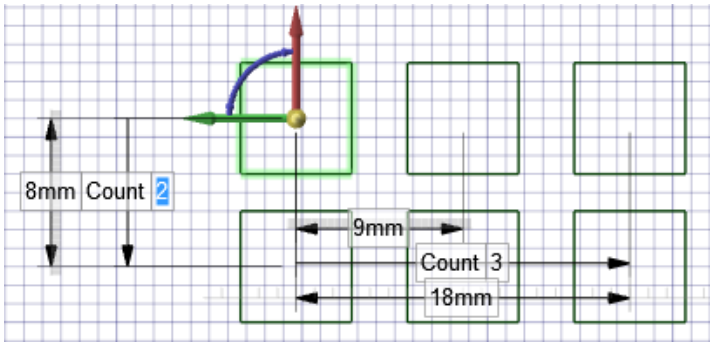
Changing spacing between pattern members changes the pattern's overall length when the length field isn't locked.



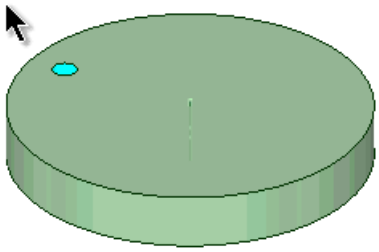
A pattern of sketch objects is no longer a pattern when you switch to 3D mode and the sketched objects are converted to surfaces.



Sketch curve patterns can propagate in two dimensions. The pattern above contains three squares in the X direction and two squares in the Y direction.



5.6.8. Circular Patterns



You can create patterns of:

- Protrusions or depressions (including slots)
- Points, planes, origins, or axes
- Sketch curves
- Notes
- Lightweight text
- Components

Patterns can be made from a mix of object types, such as a pattern of holes (faces) and bolts (imported components). You cannot create a pattern of threads. You can use the **Undo** button, **Ctrl+Z** or the **Redo** button, **Ctrl+Y** when creating a pattern.

This section contains the following topics:

5.6.8.1. Creating a One- or Two-dimensional Circular Pattern

1. Click the **Circular** tool in the Create group on the Design tab.
2. Select a protrusion, depression, body, sketch, points, axes, planes, origins, or 3D curves to be the first member (leader) of the pattern. The **Direction** tool guide activates.
3. Select a line, edge, axis, sheet metal marker, or a set of points to set the direction of the pattern.
4. (Optional) Modify the Pattern options in the Options panel.

   Pattern Type: One-dimensional or Two-dimensional.

   Change **Circular Count**, or **Angle** values for One-dimensional.

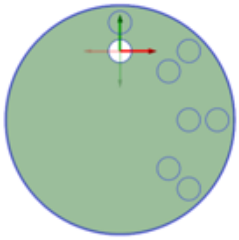
   Change **Circular Count**, **Angle**, **Linear Count**, or **Linear Pitch** values for Two-dimensional.

   Note that a Circular Count of one, in a two dimensional pattern, will create a single radial pattern.

   As you switch between pattern types, or change pattern values, the pattern preview displays your changes in blue.
5. Select the **Define Pattern** tool guide.
6. Select the **Create Pattern** tool guide, or press **Enter**, to complete the pattern.

   When the pattern is previewed, any members which cannot be created are shown in red. Members that can be created are shown in blue.

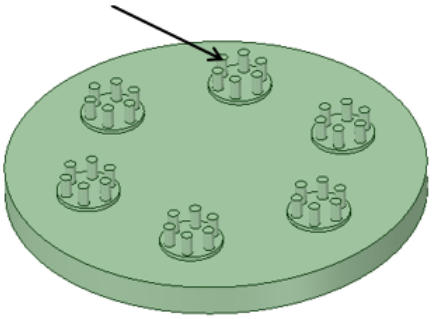
   The image below shows the preview for a two-dimensional pattern. For one-dimensional, only the Angular direction is displayed. Clicking the arrows flips their direction 180-degrees. You can create circular patterns on cylindrical faces, both around the face and along the axis.



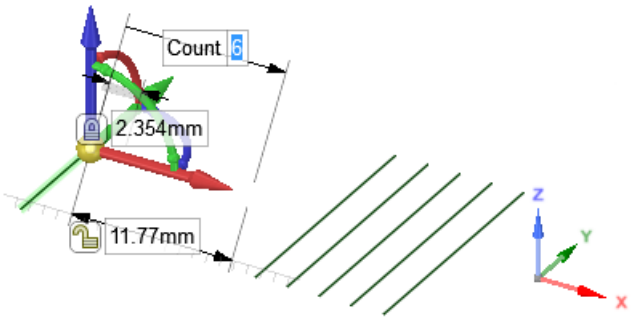
In Discovery, any pattern member can be used to modify the pattern after you create it. If the change cannot be made to all pattern members, the member that cannot change is still part of the pattern.

When you select a member of a pattern, you will see fields for the pattern count and distances relative to the member you have selected. Press **Tab** to move among the dimension and count fields.

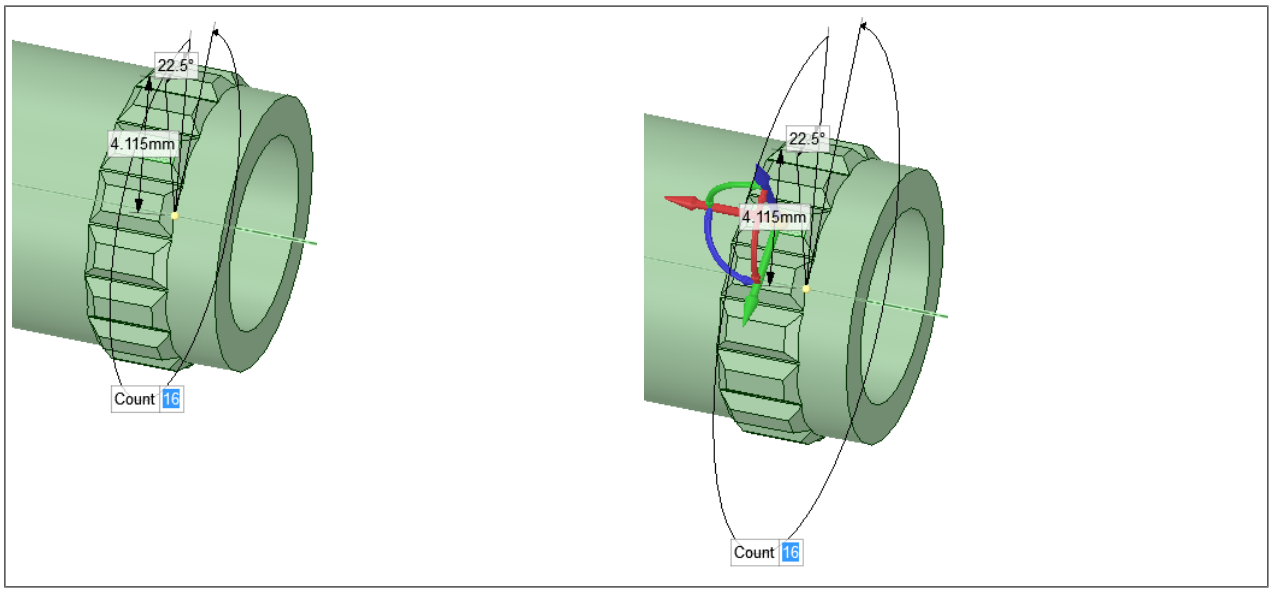
- Icons next to the pattern's dimensions and counts show which will be changed if you change the value of the highlighted field. The closed lock indicates that a dimension won't change and the open lock shows you it will change. You can override a lock on a dimension by clicking its icon to toggle it between locked and unlocked.
- Dimension or count line placement, arrows, and value fields show you how the pattern will change if you change the value of a field.
- Your selection determines how changes to the distance and count affect the pattern's position. If you select one member of the pattern, the change is centered on that member. If you select all pattern members, the change is centered on the entire pattern, with the distance between members locked by default. You can control which direction the pattern will grow based on which member you select and which count or distance you change.
- Changing a pattern count in one direction does not change the distance between pattern members. Instead, the overall distance of the pattern will change. This is indicated by the closed lock icon.
- Circular patterns are anchored at the opposite end by default when you move one end.
- In patterns of patterns, associations of children of the first pattern are maintained. An example is shown below. The height of all the bosses is changed by changing one. You can do this by using the **Move** tool to pull up, adjusting the height.



- **Patterned curves** can now be moved. Previously, you could move one of the members and the rest would not follow. The example on the right shows the previous behavior. The line on the left was used to create a six member pattern. When the line is later moved, the other members do not follow.



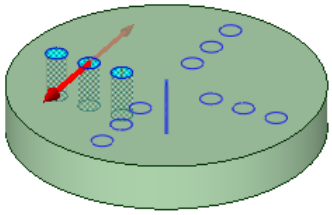
- **Pattern dimension display** now remains when you enter the **Move** tool. Previously, the pattern dimensions disappeared and only the faces were displayed. This led to confusion about what was selected for the Move. The images below show the new, correct behavior.



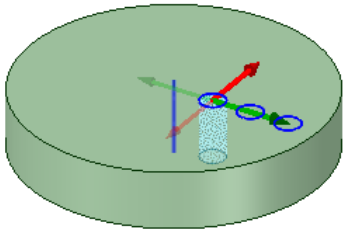
> **Note:** Patterns of sketch entities and 3D curves are no longer patterns when they change into another form, such as a surface or solid. For example, if you pattern a C-shaped curve, then it will remain a pattern when you switch from sketch mode to 3D mode. But if you close off the curve to make a box before you switch to 3D mode, then the boxes will become surfaces and will no longer be a pattern.

5.6.8.2. Creating a Radial Circular Pattern

1. Select all radial pattern members.
2. Click the **Circular pattern** tool.
3. Set the Pattern type to **One-dimensional**.
4. Enter the **Circular count** and **Angle**.
5. Click the circular axis.



To create a single radial pattern, set the Pattern type to **Two-dimensional** and enter a **Circular Count** of 1.



5.6.8.3. Editing Circular Pattern Properties

1. Select one pattern member to display the pattern count and dimensions.
2. Edit the pattern's properties.

   Press **Tab** to switch between the fields.

   Click the lock icon to lock or unlock a value.
3. Press **Enter**.

   The result of editing the count and spacing is relative to the member of the pattern you select.

> **Note:** To quickly increase or decrease the number of pattern members, press **Tab** until the count field is highlighted, then hold the **Alt** key while you press the up or down arrow.

This section contains the following topics:

5.6.8.3.1. Moving a Pattern

1. Click the **Move** tool.
2. Select a pattern member to move a linear pattern or the pattern axis to move a circular or arc pattern.
3. Move the pattern member with the Move handle.

   If you have a linear pattern in a radial direction and you move an interior member without setting an anchor, then the entire pattern shifts in the selected direction.

Use the **Up To** tool to move a pattern member up to another face or edge. Pattern dimensions (such as count and length) display as expected.

5.6.8.3.2. Moving a Radial Pattern in a Linear Direction

1. Click the **Move** tool.
2. Right-click a pattern member and click **Select** > **All Pattern Members**.

3. Select the **Direction** tool guide.
4. Click an object to set the direction of the move.
5. Drag the pattern.

5.6.8.3.3. Adjusting Pattern Distance or Spacing

1. Click **Select** or **Move** in the **Edit** group on the **Design** tab.
2. Select a pattern member.

   This member will serve as the anchor, and other pattern members will move relative to this member.
3. Press **Tab** until the field you want to change is highlighted:
   - **Distance**: This field is the overall distance of the pattern. The pattern member you select is anchored, and the length will change relative to this member. The arrows indicate the direction of change. You can see this field at the top of the image below.
   - **Spacing**: This field is the spacing between pattern members. This field has arrows in both directions and is located between two pattern members. The field is highlighted in blue in the image below.
   - **Radial patterns**: If you select a member of a radial pattern, you can change the angle between pattern members and the distance from the pattern members to the center of the pattern.
4. Type a new distance or angle and press **Enter**.

   The spacing between all pattern members will change.

5.6.8.3.4. Removing a Pattern Member from the Pattern

- Right-click the face of the pattern member and select **Unpattern Member**.
- Select the member components in the tree, then right-click and select **Source** > **Make Independent** from the context menu.

5.6.8.3.5. Creating a Pattern Group

1. Select a pattern or pattern member in the **Design** window.
2. Click the **Groups** tab in the **Structure** panel.
3. Click **Create NS**.

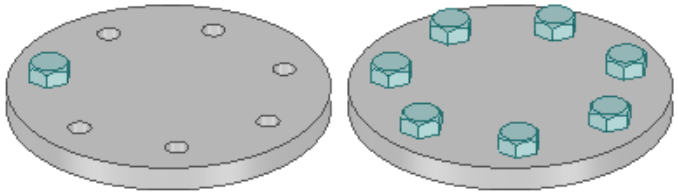
   A group is created that displays the number of members in the pattern. You can click this group to select the pattern and change its parameters in the Design window.

5.6.8.3.6. Assembling Components on a Pattern

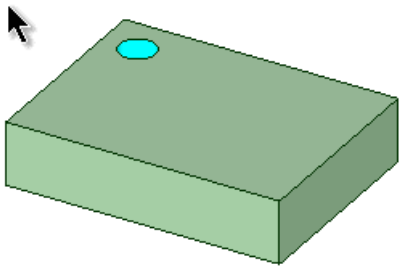
You can assemble a component with a pattern member, then propagate the component to all of the pattern members. The component will be copied and assembled to each pattern member.

1. Assemble a component to some geometry that is a pattern member using the **Align**, **Tangent**, or **Orient** tools.
2. Right-click the **Assembly Condition** in the **Design** tree and select **Update Components on Pattern**.

   The components will be propagated to the pattern, as shown below.



5.6.9. Fill Patterns



You can create **Fill** patterns of:

- Protrusions (including sheet metal forms)
- Depressions (including slots and standard holes)
- Faces (Closed curves on a face)

Components can be assembled to a Fill pattern. For example, a bolt component assembled into a pattern of holes.

Threads cannot be patterned.

You can use the **Undo** button, Ctrl+Z or **Redo** button, Ctrl+Y when creating a pattern.

This section contains the following topics:

5.6.9.1. Creating a Fill Pattern

1. Click the **Fill** tool in the Create group on the Design tab.
2. Select a protrusion or depression to be the first member (leader) of the pattern. The **Direction** tool guide activates.
3. Select a line, edge, axis, or a set of points to set the direction of the pattern.
4. Modify the options in the Options panel

   Pattern **Type**: Grid, Offset, or Skewed

   Modify the **X Spacing** or **Y Spacing** for Grid or Offset patterns.

   Modify **Row X Offset**, **Row Y Offset**, **Column X Offset**, and **Column Y Offset** for Skewed patterns.

   Modify the **Margin** values. A dashed orange boundary line displays the pattern's marginal boundary.

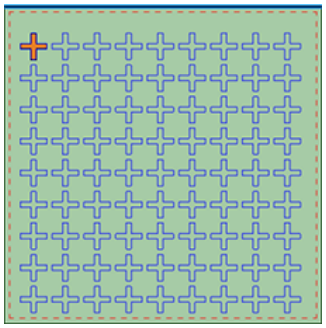
   As you switch between pattern types, or change pattern values, the pattern preview displays your changes in blue.
5. Select the **Create Pattern** tool guide, or press **Enter**, to complete the pattern.

When the pattern is previewed, any members which cannot be created are shown in red. Members that can be created are shown in blue.

In Discovery, any pattern member can be used to modify the pattern after you create it. If the change cannot be made to all pattern members, the member that cannot change is still part of the pattern.

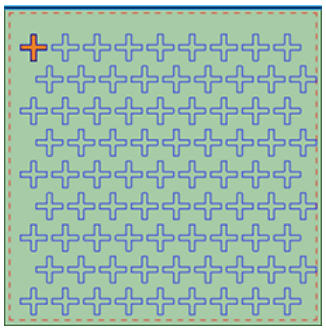
5.6.9.2. Grid Patterns

Grid uses the X and Y Offset values to produce a simple grid pattern.



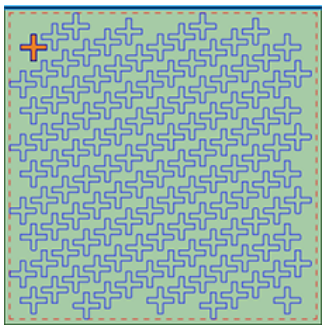
5.6.9.3. Offset Patterns

Offset uses the X and Y Offset values to produce a grid but also offsets alternate rows.

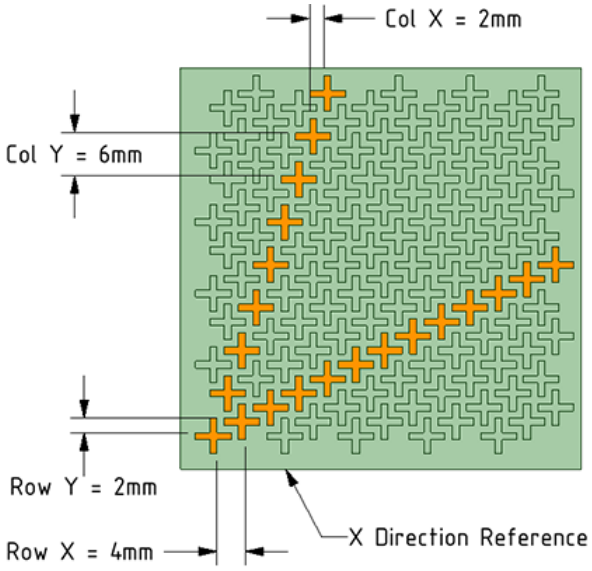


5.6.9.4. Skewed Patterns

Skewed allows you to get a tighter nesting of pattern members using individual X and Y offsets for rows and columns.



The meaning of the offsets is shown graphically below.



5.6.9.5. Editing Fill Pattern Properties

With a fill pattern node selected in the model tree, you can:

- Change fill pattern parameters in the **Layout** section of the **Properties** panel. Changes that you make automatically update the pattern in the design window.
- To update lightweight fill patterns to fit the existing geometry of your design, select the lightweight pattern node, then right-click and select **Update Pattern**.

This section contains the following topics:

5.6.9.5.1. Moving a Pattern

1. Click the **Move** tool.
2. Select a pattern member to move a linear pattern or the pattern axis to move a circular or arc pattern.0
3. Move the pattern member with the Move handle.

   If you move an interior member of a pattern and it is not anchored, all the pattern members move:



   If you move a member at one end of a pattern, the member at the opposite end is anchored and the pattern is skewed:



If you anchor a different member than the member opposite the direction you are moving, Move skews the pattern.

If you have a linear pattern in a radial direction and you move an interior member without setting an anchor, then the entire pattern shifts in the selected direction

Use the **Up To** tool to move a pattern member up to another face or edge. Pattern dimensions (such as count and length) display as expected.

5.6.9.5.2. Removing a Pattern Member from the Pattern

To remove a pattern member from the pattern:

1. Right-click the face of the pattern member and select **Unpattern Member**.
2. Select the member components in the tree, then right-click and select **Source** > **Make Independent** from the context-menu.

5.6.9.5.3. Creating a Pattern Group

1. Select a pattern or pattern member in the Design window.
2. Click the **Groups** tab in the **Structure** panel.
3. Click **Create NS**.

   A group is created that displays the number of members in the pattern. You can click this group to select the pattern and change its parameters in the Design window.

5.6.9.5.4. Assembling Components on a Pattern

You can assemble a component with a pattern member, then propagate the component to all of the pattern members. The component will be copied and assembled to each pattern member.

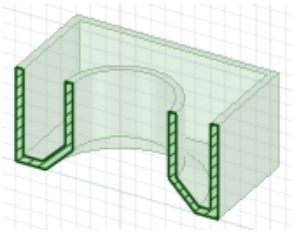
1. Assemble a component to some geometry that is a pattern member using the **Align**, **Tangent**, or **Orient** tools.
2. Right-click the **Assembly Condition** in the **Design** tree and select **Update Components on Pattern**.

5.6.10. Shelling a Solid

Use the Shell tool in the Insert ribbon group on the Design tab to remove one of the faces of a solid and create a shell of a specified thickness. You can then use the Shell tool to remove other sides of the shell. Discovery automatically creates an offset relationship between the sides of the solid for you.

You can also create a *closed shell* without removing a face.

Shelled solids can have chamfers and rounds, and the faces can be drafted. The chamfer or rounded edge will be created and maintained on both offset faces. A shelled solid with a chamfer is shown in cross-section below.



This section contains the following topics:

5.6.10.1. Creating a Shell

1. Click the **Shell** tool in the Create ribbon group on the Design tab.
   Mouse over the solids in your design to highlight the faces that could be removed.
2. (Optional) Enter a value into the dimension field to change the thickness of the shell.
   Enter a negative number to create the shell thickness from the outside of the solid.
3. Select the face you want to remove.
   The face is removed and a shell is created. The baseline of the offset is shown in blue.
4. (Optional) Continue clicking to remove additional faces.

5.6.10.2. Creating a Closed Shell

1. Select the solid.
2. Click the Shell tool in the Create ribbon group on the Design tab.
   An internal shell is created.

5.6.10.3. Editing a Shell

Right-click the shell to change its thickness.

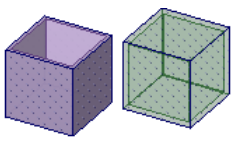
5.6.10.4. Shell Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

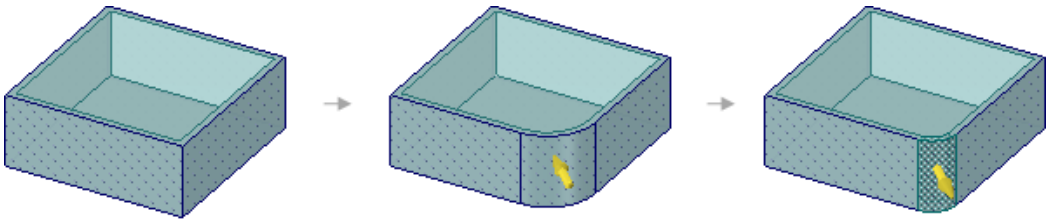
Remove Faces	The **Remove Faces** tool guide is active by default. Select a face of the solid to remove it and create a shell. Ctrl+click to remove multiple faces.
More Shell	If you create a shell, then add a protrusion onto it, you can extend the shell through the protrusion by clicking the **More Shell** tool guide, then clicking the newly added protrusion.
Complete	The **Complete** tool guide creates the shell and reactivates the **Remove Faces** tool guide so you can create another shell.

5.6.10.5. Creating a Shell Example

An open shell and a closed shell.



Adding and changing a rounded edge on a shelled or offset part changes the inside faces.



5.6.11. Creating Offset Relationships

Use the **Offset** tool to create an offset relationship between two faces. This relationship will be maintained in the other 2D and 3D editing tools. For example, when you select an offset face with the Pull tool, the offset dimension is displayed. Baseline faces do not display a dimension.

This section contains the following topics:

5.6.11.1. Creating an Offset Relationship

1. Click the **Offset** tool in the Create ribbon group.

   Mouse over your design to see the faces eligible for offset relationships.
2. (Optional) Select options.

   Select the **Find All Same Offset** option if you want to select all contiguous face pairs that have the same offset distance as the pair you selected. If this option is not selected, the offset relationship is only created for the selected pair of faces.
3. Click the first face.
4. Click the second face.

   You have now established an offset relationship between the face pair and other contiguous face pairs with the same offset distance (if the **All the Same** option is selected). The **Toggle Baseline** tool guide activates.
5. (Optional) Click to select the baseline face(s).

   When you move one of the faces with a design tool, the other face in the offset pair moves to maintain the offset relationship. When you fill one face of an offset pair, the other face is also filled.

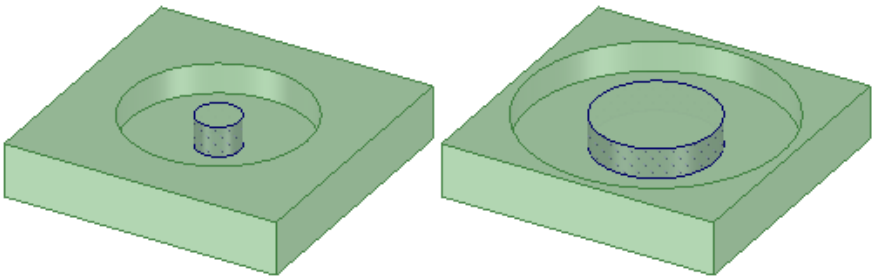
5.6.11.2. Offset Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Face Pair	The **Face Pair** tool guide is active by default.
Toggle Baseline	When the **Toggle Baseline** tool guide is active, click the faces that you want to be the baseline of the offset.

5.6.11.3. Creating Offset Relationship Example

An offset relationship between two cylindrical faces causes both to change when you change the diameter of either cylinder.



5.6.12. Creating Mirror Relationships

Use the **Mirror** tool in the **Insert** ribbon group on the **Design** tab to mirror any geometry that can be moved with the **Move** tool. You can insert a mirror plane between two faces, associating the faces so you can edit them together. You can copy a solid, surface, face, or curve from one side of a mirror plane to the other.

You can mirror curves, points, and axes without leaving sketch mode or cross section mode. When you mirror a curve, changing its geometry does not propagate to the other side.

You can mirror solids that contain patterns of features on a face, and the pattern will be updated on both sides. The pattern must be created before the solid is mirrored.

The plane becomes a persistent mirror once you create geometry using the mirror. The mirror relationship is maintained when you use other 2D and 3D tools to edit your design.

Mirrored components have a mirrored icon in the model tree and maintain their mirror relationships when they are exported.

Mirrored objects are created on the same layer as the original object. Mirrored points do not maintain their mirror relationship.

This section contains the following topics:

5.6.12.1. Mirroring an Object

1. (Optional) Create the plane you want to use as a mirror using the Insert Plane tool and position it with the Move tool.
2. Click **Mirror** in the Create group.

   The **Select Mirror Plane** tool guide is enabled.
3. Click the plane to use as a mirror.
4. Click the **Mirror Body** or **Mirror Face** tool guide.
5. Mouse over the solids or faces in your design to preview the solid, surface, or face that will be created on the other side of the selected mirror plane.
6. Click the object you want to mirror.
7. The mirror plane is created and will persist in other tools.

   You can click objects in your design or click a component in the model tree.

If you enclose a volume by mirroring faces, a solid is created.

5.6.12.2. Removing a Mirror Face or Plane

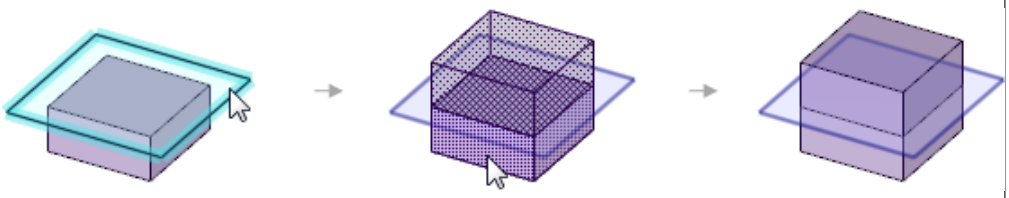
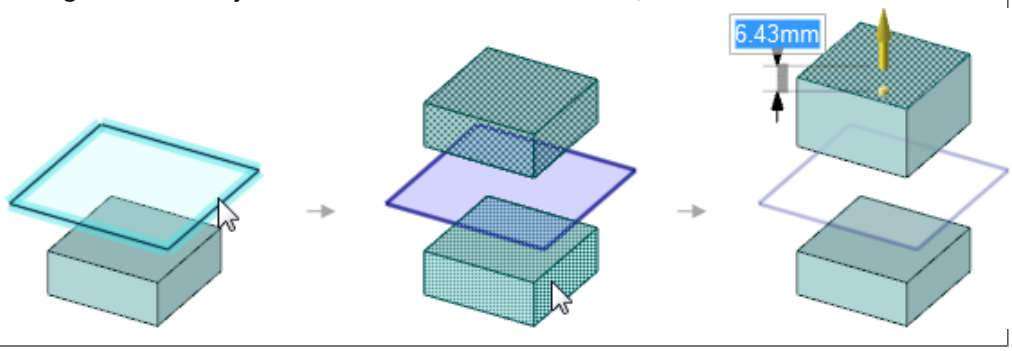
1. Click **Mirror** in the Create group.

   The Select Mirror Plane tool guide is enabled.

2. Click the **Remove Mirror** tool guide.
3. Click the face from which you want to remove the mirror relationship. You can also Box-select any number of faces.

   You can temporarily disable a plane in any tool. Select a mirrored face to display the mirror plane, then click the mirror icon to disable the mirror. Click the mirror icon again to enable the mirror.

5.6.12.3. Mirror Tool Options

Merge mirrored objects	When you use a face as the mirror plane, or when the mirror plane lies on a face of the source object, the two mirrored objects will be merged. Deselect this option to make a new object, as shown below. 
Create mirror relationships	If you deselect this option, only the geometry is created and not the mirror relationship. Changes to either object will not be reflected in the other, as shown below. 

You can create a mirror plane between two symmetrical faces in your design.

5.6.12.4. Setting Up a Mirror Relationship

1. Click **Mirror** in the Create group.

   The **Select Mirror Plane** tool guide is enabled.
2. Click the **Setup Mirror** tool guide.

   As you mouse over the faces in your design, the eligible faces are highlighted. A face is eligible if it is symmetrical to another face in your design.
3. Click a face.
4. Click a parallel face.

   The second face is highlighted in blue and the mirror plane appears halfway between the two faces. Edits to one face will now be mirrored on the other face. Geometry in between the two faces will be updated accordingly.

You can create mirror relationships by finding similar faces on the other side of an existing mirror.

5.6.12.5. Finding Similar Faces on the Other Side of a Mirror

1. Click the **Setup Mirror** tool guide.
2. Click a face.
   Ctrl+click to select multiple faces.
3. Alt+click a mirror plane.
   Identical faces equidistant from the mirror plane are highlighted in blue, and a mirror relationship is created.

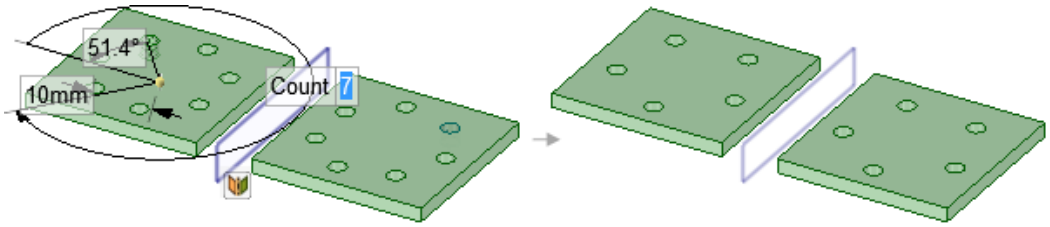
5.6.12.6. Mirror Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

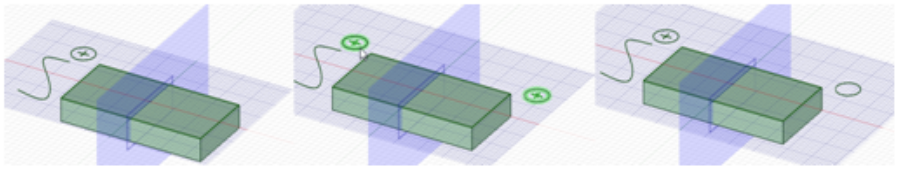
Mirror Plane	The **Mirror Plane** tool guide is active by default. Select a face or plane to use it as a mirror. (You can also use this tool guide to select another mirror plane to use if one is already selected.
Mirror Body	The **Mirror Body** tool guide activates automatically once you select a mirror face or plane. Mouse over the solids in your design to preview the solid that will be created by the mirror. Click a solid to mirror it.
Mirror Face	Once you select a face or plane to use as a mirror, use the **Mirror Face** tool guide to select the faces you want to mirror. Mousing over the faces before you click them previews the face that will be created by the mirror. Click a face to mirror it.
Mirror Component	The **Mirror Component** creates a left-hand transform and does not create a mirror relationship between the two components or subassemblies.
Setup Mirror	The **Setup Mirror** tool guide can be used any time to create a mirror plane between two faces. Only those two faces will be affected by the mirror. To create a relationship among many faces, Ctrl+select faces and then select a mirror plane to auto-detect identical faces equidistant from the mirror plane.
Remove Mirror	Use the **Remove Mirror** tool to remove the mirror relationship between faces. You can select to faces with a mirror relationship, or use box-selection to select any number of mirrored pairs.

5.6.12.7. Mirroring Examples

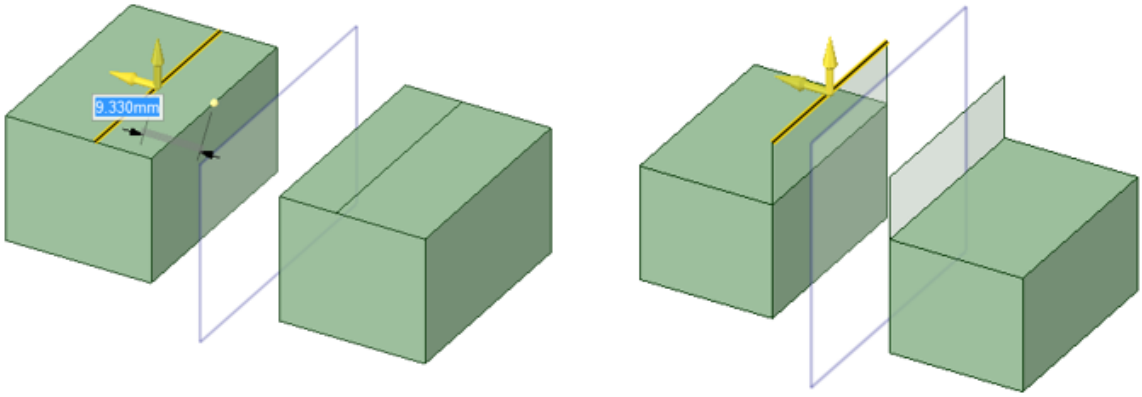
A mirrored solid with a pattern of features is updated when the count is changed from 7 to 5 on one side.



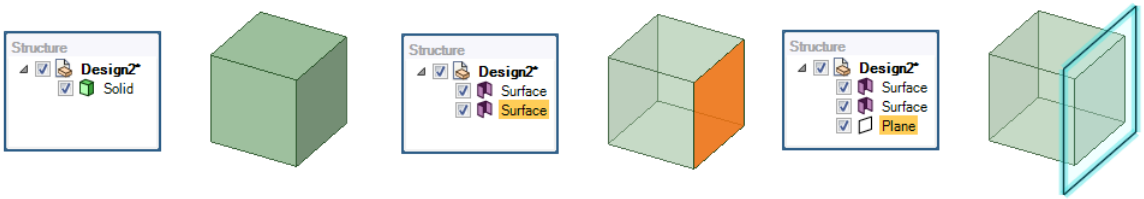
Mirroring a sketched circle. Modifying the circle does not change the geometry of the mirrored circle.



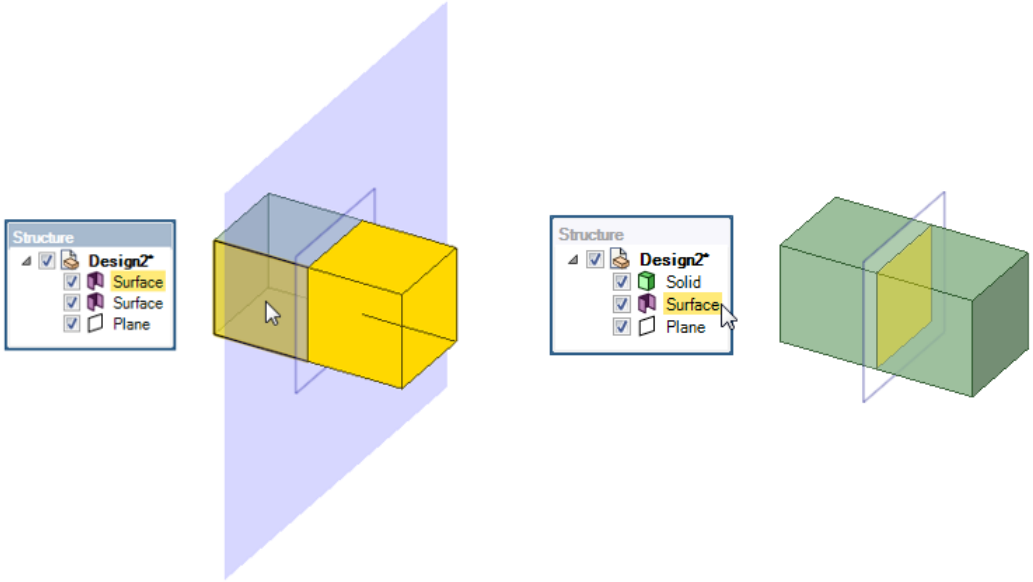
*Mirror associations are maintained in the Pull tool using the **Copy Edge** and **Extrude Edge** options. The behavior for **Copy Edge** is on the left and **Extrude Edge** is on the right.*



A mirrored surface body that becomes a closed body after the mirror now becomes a closed body. Start with a solid body, detach a face to make two surface bodies, and create a plane at the opening.



Mirror the surface body to see a solid body with the detached surface in the mirror plane.



5.7. Bodies

This section contains the following topics:

5.7.1. Geometry from Equations

You can use the **Equation** tool to create geometry surfaces or curves from equations. Select from several preset equations or write custom equations to create the geometry.

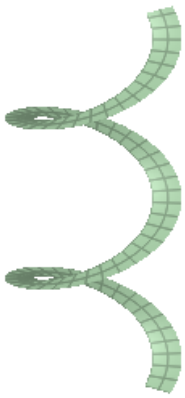
Design > **Equation**

This section contains the following topics:

5.7.1.1. Surfaces from Equations

You can select from a number of preset 3D equations or write custom equations in the tool.

The image below shows a Helicoid surface.



The equation is constructed in the Options panel.

- You can enter an equation in the input fields for X, Y, and Z is for 3D equations.
- Clicking the icon to the right of the X, Y, or Z input fields opens an editor for more complex equations.
- Use the **Show UV grid** option to display a grid on the surface and adjust the grid density using the dropdown slider. The grid helps visualize the surface contours.

This section contains the following topics:

5.7.1.1.1. Creating a Preset Equation

1. Select the **Equation** tool.
2. (Optional) Click the **Select Origin** tool guide and select an equation origin.
3. Select **Surface** and choose an equation from the **Surface type** dropdown list.
4. Edit the equation as needed in the Options panel.
5. Click **Complete** to create the equation surface.

5.7.1.1.2. Creating a Custom Equation

1. Select the **Equation** tool.
2. (Optional) Click the **Select Origin** tool guide and select an equation origin.
3. Select **Surface** and choose **Custom** from the **Surface type** dropdown list.
4. Enter the equation in the input panels.
5. To create a parameter to use in the equation enter it in square brackets "`[ ]`"
   - For example: `[L]`
   - The parameter is added to the Parameters list in the Options panel
6. Click **Complete** to create the equation surface.

5.7.1.1.3. Saving a Custom Equation

The **Save Custom Equation** button next to the **Surface type** dropdown list allows you to save your custom equations.

Saved equations are added to the **Surface type** dropdown list and can be used in other documents and across multiple sessions. If you do not see a newly saved equation in the list, open/create a new document or close and re-open the current document.

5.7.1.1.4. Editing an Equation

1. Select the **Equation** tool.
2. The **Select Body** tool guide is active.
3. Select the body to edit its equation.
4. (Optional) Click the **Select Origin** tool guide and select a different origin.
5. Edit the equation as needed in the Options panel.
6. Click **Complete** to create the equation surface.

5.7.1.1.5. Scaling an Equation Surface

Use the **Scale** parameter and slider in the Options panel to scale an equation surface.

3D scaling with the **Pull** tool also works on equation surfaces.

5.7.1.1.6. Syntax and Predefined Functions

- Parameters include a name and are enclosed in square brackets `[L]`
- Functions include a function name followed by parenthesis containing any values as arguments

  For example: `Sine([t]-[r])`

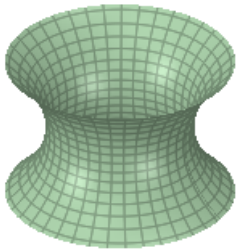
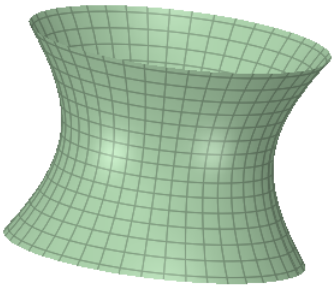
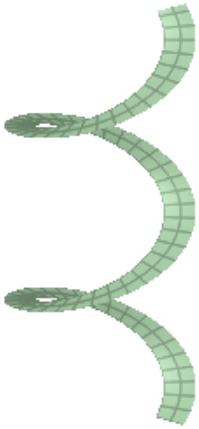
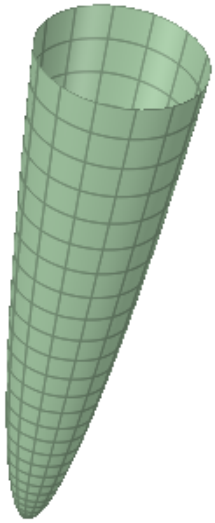
The table below shows common predefined functions included.

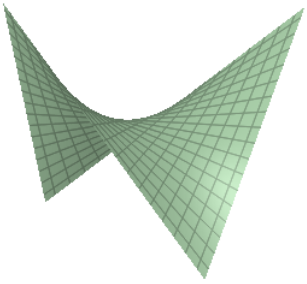
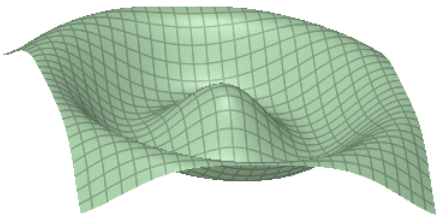
Abs	Returns the absolute value of a specified number
Acos	Returns the angle whose cosine is the specified number
Asin	Returns the angle whose sine is the specified number

Atan	Returns the angle whose tangent is the specified number
Cos	Returns the cosine of the specified angle
Sin	Returns the sine of the specified angle
Sqrt	Returns the square root of the specified number
Tan	Returns the tangent of the specified angle

5.7.1.1.7. Preset Surface Examples

The preset equations include those shown in the following table.

Catenoid 	*One Sheet Hyperboloid. Two sheets consists of separate top and bottom Hyperboloid surfaces, and can be made with two equations with inverted ranges.* 
Helicoid 	*Paraboloid* 

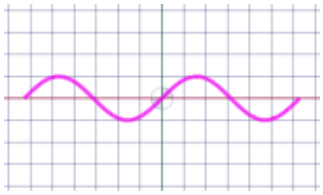
Hyperbolic Paraboloid	*Radial Wave*
	

5.7.1.2. Curves from Equations

Use the **Equation** tool to create curves based on mathematical equations.

You can select from a number of preset equations or write custom equations.

The image below shows a Sine Wave equation.



The equation is constructed in the Options panel.

- You can enter an equation in the input fields for X and Y. Z is for 3D equations. You can enter a value to offset the curve in the Z-direction.
- Clicking the icon to the right of the X, Y, or Z input fields opens an editor for more complex equations.

This section contains the following topics:

5.7.1.2.1. Creating a Preset Equation

1. Select the **Equation** tool in the Design tab.
2. (Optional) Click the **Set Grid Location** tool guide and select a different sketch plane
3. Select **Curve** and choose an equation from the **Curve type** dropdown list.
4. Edit the equation as needed in the Options panel.
5. Click **Complete** to create the equation curve.

5.7.1.2.2. Creating a Custom Equation

1. Select the **Equation** tool in the Design tab.
2. (Optional) Click the **Set Grid Location** tool guide and select a different sketch plane.
3. Select **Curve** and choose **Custom** from the **Curve type** dropdown list.
4. Enter the equation in the input panels.
5. To create a Parameter to use in the equation enter it in square brackets "`[ ]`"

- For example: `[L]`
- The parameter is added to the Parameters list in the Options panel

6. Click **Complete** to create the equation curve.

5.7.1.2.3. Saving a Custom Equation

The **Save Custom Equation** button next to the **Curve type** dropdown list allows you to save your custom equations.

Saved equations are added to the **Curve type** dropdown list and can be used in other documents and across multiple sessions. If you do not see a newly saved equation in the list. open/create a new document or close and re-open the current document.

5.7.1.2.4. Editing an Equation Curve

1. Select the **Equation** tool in the Design tab.
2. The **Select Curve** tool guide is active.
3. Select the curve to edit.
4. (Optional) Click the **Set Grid Location** tool guide and select a different sketch plane.
5. Edit the equation as needed in the Options panel.
6. Click **Complete** to create the equation curve.

> **Note:** Editing an Equation Curve in 3D is limited to the range of the equation. For example, if a Sine curve has a range of 4π, you can Pull an end to anywhere within the that range. You cannot make it longer (for example, 6π)
>
> If the curve has been Split, the ends of any of the pieces can be Pulled within the range of the original equation.

5.7.1.2.5. Scaling an Equation Curve

Use the **Scale** parameter and slider in the Options panel to scale an equation curve.

3D scaling with the **Pull** tool also work on equation curves.

5.7.1.2.6. Syntax and Predefined Functions

- Parameters include a name and are enclosed in square brackets `[L]`
- Functions include a function name followed by parenthesis containing any values as arguments

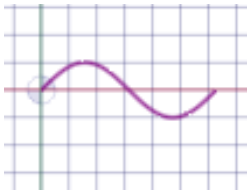
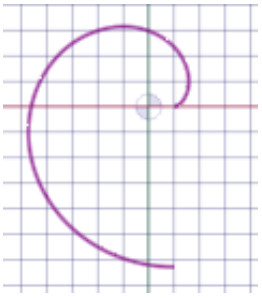
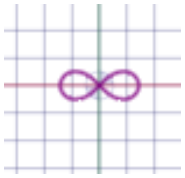
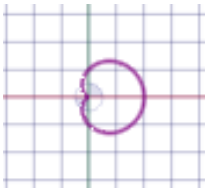
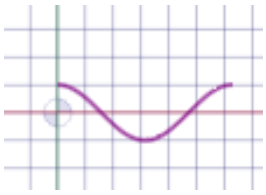
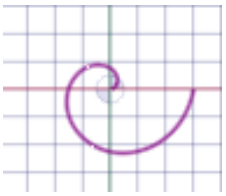
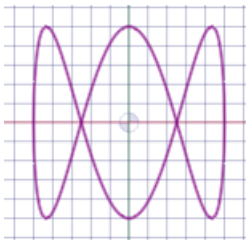
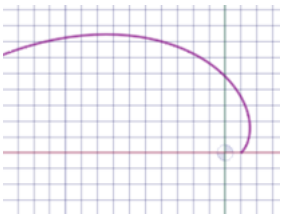
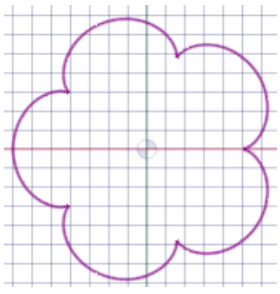
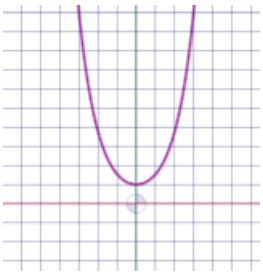
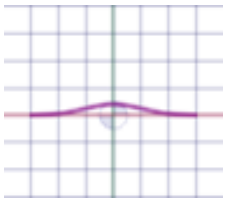
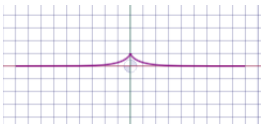
  For example: `Sine([t]-[r])`
- The table below shows common predefined functions included

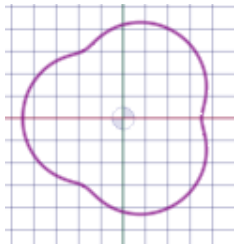
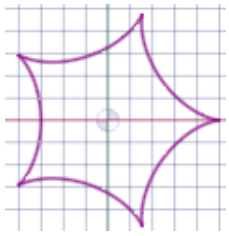
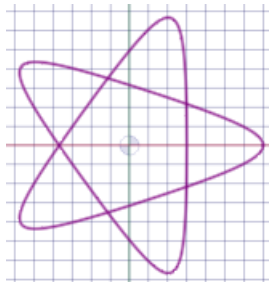
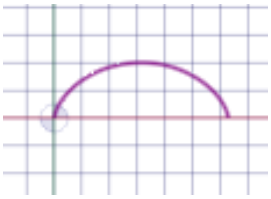
Abs	Returns the absolute value of a specified number
Acos	Returns the angle whose cosine is the specified number
Asin	Returns the angle whose sine is the specified number

Atan	Returns the angle whose tangent is the specified number
Cos	Returns the cosine of the specified angle
Sin	Returns the sine of the specified angle
Sqrt	Returns the square root of the specified number
Tan	Returns the tangent of the specified angle

5.7.1.2.7. Preset Curve Examples

The preset curves include those shown in the following table. All curves have default values.

Sine Wave 	**Involute of Circle** 	**Lemniscate (Bernoulli)** 	**Limaçon** 
Cosine Wave 	**Archimedes Spiral** 	**Lissajous** 	**Logarithmic Spiral** 
Epicycloid 	**Catenary** 	**Normal Distribution** 	**Tractrix** 

Epitrochoid	Hypocycloid	Hypotrochoid	Trochoid
			

There are also 3D curves included in the presets. These are shown in the table below. All curves are at default values.

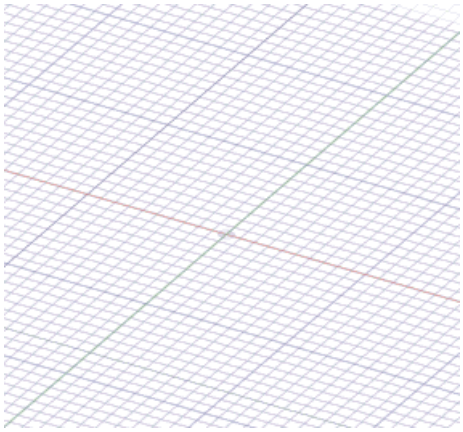
Helix	Slinky
	
Line	**Viviani's**
	

5.7.2. Creating Cylinders

Use the **Cylinder** tool to sketch the cylinder's axis in 2D and create its diameter in 3D.

The sketch grid must be visible in the workspace before you can draw.

If you select a 3D curve or edge before you click the **Cylinder** tool, the cylinder is created as a swept pipe.



This section contains the following topics:

5.7.2.1. Drawing a Cylinder

1. Click **Design** > **Body** > **Cylinder** .
2. (Optional) If you want to dimension the first end point of the axis, press **Shift** and hover the mouse over a line or point to create a dimension relative to that line or point.
3. Click or press **Enter** to set the first end point of the axis.
4. (Optional) Dimension the axis.
5. Click or press **Enter** to set the other end point of the axis.

   By default, the axis is dimensioned to its first end point. You can also press **Shift** and hover the mouse over another line or point to create a dimension relative to that line or point.
6. (Optional) Dimension the cylinder's diameter.
7. (Optional) Select options from the Options panel, or right-click and select them from the mini-toolbar.
8. Click or press **Enter** to set the diameter of the cylinder.

5.7.2.2. Creating a Swept Pipe

1. Select a 3D curve or edge.

   If the curves or edges include angles, a spherical surface is added at the angle. If you use the Fill tool on the spherical joints, the corners are converted to sharp corners. See the examples below.
2. Click **Design** > **Body** > **Cylinder** .
3. Click in the Design window to set the diameter of the cylinder.

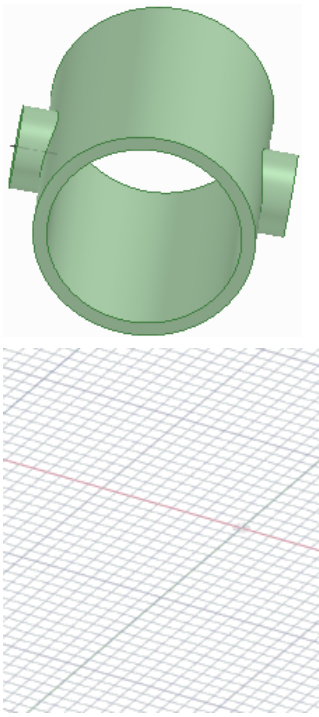
5.7.2.3. Cylinder Tool Options

The following options are available in the **Cylinder** tool.

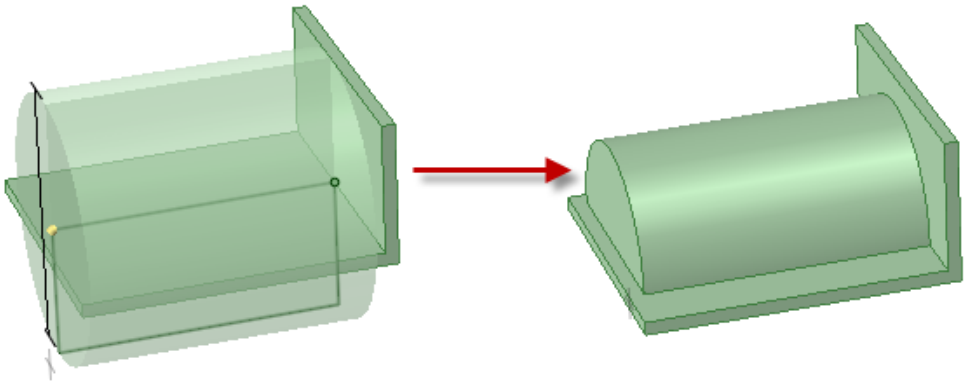
Add	Add material to create a solid cylinder.
Cut	Remove material when the cylinder intersects another object.
No Merge	Creates a cylinder without merging into other objects even when the cylinder intersects with an existing object.
Near-side body only	Add material only on the near side when the cylinder intersects another object.

5.7.2.4. Cylinder Examples

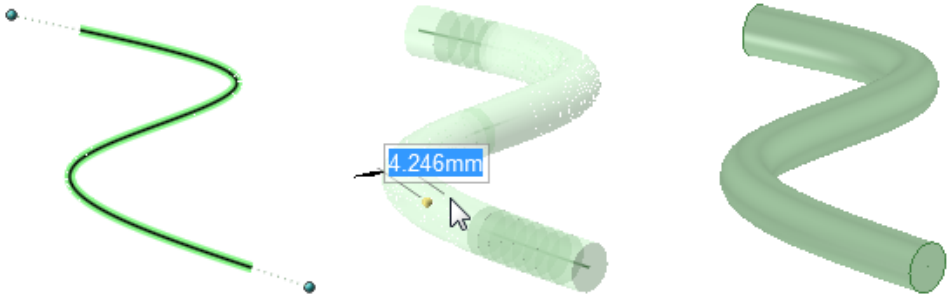
Tube with Nubs



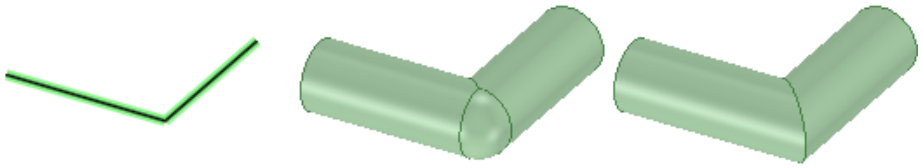
Using the Near-side body only option to add cylinder material only on the near side of a thin placement wall.



Creating a swept pipe with the Cylinder tool.



Creating a swept pipe with the Cylinder tool, then using Fill on the spherical joint to create a sharp corner.



5.7.3. Creating Spheres

You can quickly create a sphere using the **Sphere** tool.

1. Select the **Design** > **Body** > **Sphere**.
2. (Optional) Click **No Merge** in the Properties panel if you don't want the sphere merged with existing bodies.
3. (Optional) Click **Cut** in the Properties panel to remove material from existing bodies where they overlap the sphere.
4. Click to set the center of the sphere and the plane in which the sphere's radius is dimensioned.

   As you move the mouse, you can see a preview of the sphere. Sphere creation works best when you move the cursor in the x-y direction of the plane indicated by the first click.
5. Click to set the radius of the sphere.

This section contains the following topics:

5.7.3.1. Creating a Sphere Over a Face or Edge

1. Select one or more faces or edges.
2. Select the **Design** > **Body** > **Sphere**.

   A sphere is created that minimally covers the selected faces and/or edges.

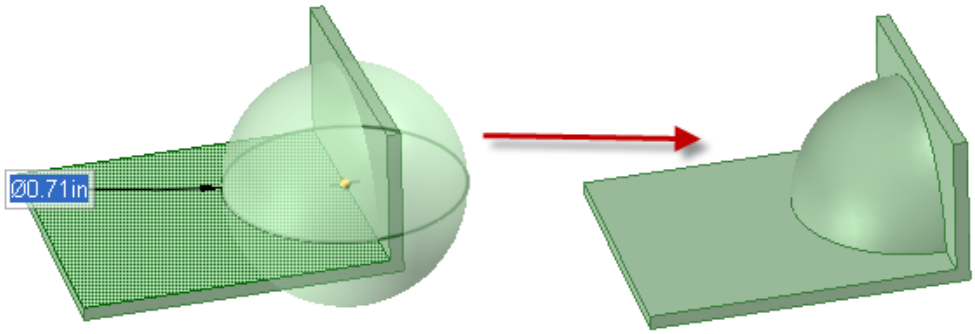
5.7.3.2. Sphere Tool Options

The following options are available in the **Sphere** tool.

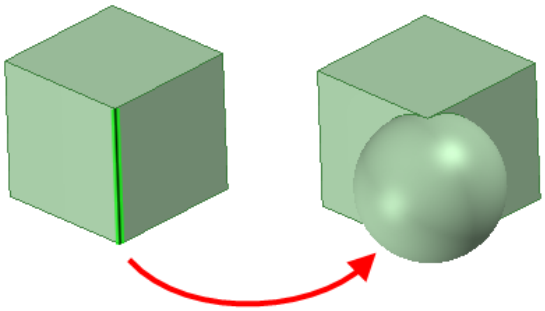
Add	Add material to create a solid sphere.
Cut	Remove material when the sphere intersects another object.
No Merge	Creates a sphere without merging into other objects even when the sphere intersects with an existing object.
Near-side body only	Add material only on the near side when the sphere intersects another object.

5.7.3.3. Sphere Examples

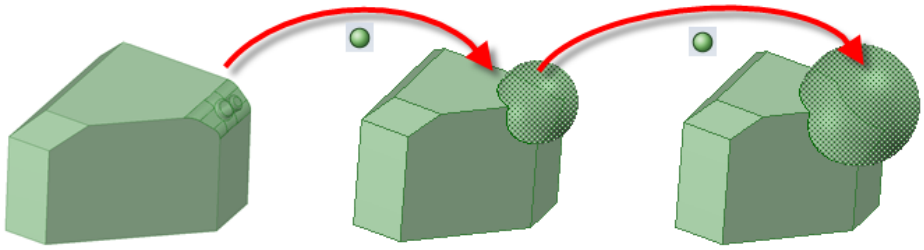
Using the Near-side body only option to add sphere material only on the near side of a thin placement wall.



Creating a sphere over a face or edge.



Selecting faces and clicking the Sphere tool to create a sphere, then clicking the Sphere tool multiple times to enlarge the selected sphere.

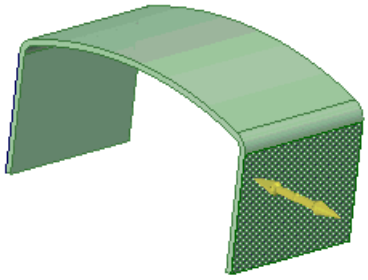


5.8. Dimensions

You can dimension every element in your design, from lines in sketches to faces of solids. In Discovery, dimensions are not constraints. Rather, they are tools for precise control during the creation or modification of a design. In Discovery, if you do want to save a dimension with your design, use the **Ruler Dimension** option when pulling or moving. You can save the ruler dimensions as **Groups** for later edits.

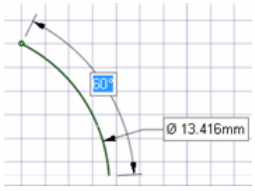
Whenever a dimension field appears, you can press the spacebar or click it to enter a value, and press Tab to switch between fields.

You can use mathematical expressions in a dimension.



This section contains the following topics:

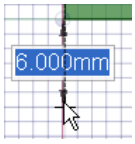
5.8.1. Dimensioning a Sketch Line During Creation



1. Press the spacebar (or just type) to enter a value in the highlighted field.
2. Press **Tab** to switch between dimension fields.
3. Repeat step 2 until you have entered all the dimensions.
4. Press **Enter** to accept the values and return to sketching.

   The dimensions persist until you select another tool or begin drawing another sketch object.

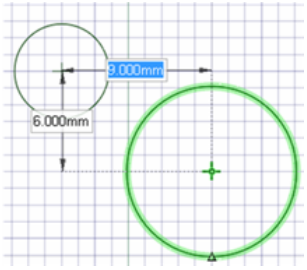
5.8.2. Dimensioning the Start, End or Middle Point of a Sketch Line from Another Point in Your Sketch



1. Hover the mouse over the point from which you want to dimension.
2. Press **Shift**.

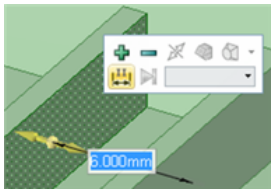
   As you move your mouse around the sketch grid, a dimension will appear from the point you indicated to the mouse location.
3. Press the spacebar (or just type) to enter a value in the highlighted field.
4. Press **Tab** to switch between dimension fields.
5. Repeat step 4 until you have entered all the dimensions.
6. Press **Enter** to accept the values and place the point that begins or ends your line.

5.8.3. Dimensioning Existing Sketch Lines



1. Click the **Select** tool.
2. Select the sketch object you want to change.
3. Dimension the item's size or location by doing one of the following:
   - Press the spacebar (or just type) to enter a value in the highlighted field.
   - Drag the selected item to change its size or location.
   - Hover over a point in your design and press **Shift** to dimension between the selected object and that point.
   - Press **Shift** while dragging to dimension from the current mouse location.

5.8.4. Dimensioning While Moving or Pulling



1. Select a direction for the move or pull.
2. Press the spacebar (or just type) to enter a value in the highlighted field.
3. Press **Tab** to switch between dimension fields.
4. Repeat step 3 until you have entered all the dimensions.
5. Press **Enter** to accept the values and move or pull the selected object the distance you entered.

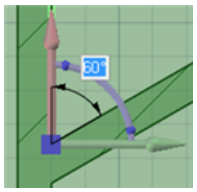
5.8.5. Creating a Ruler Dimension

1. Click **Design** > **Edit** > **Pull** or press **P**.
2. Select the faces or edges whose location you wish to specify.
3. Select a direction for the dimension.
4. Select **Options** > **Create Ruler Dimension** or right-click and select it from the mini-toolbar.
   The start point of the dimension is set as the location of the **Pull** arrow or **Move** handle.
5. Click an object to set the end point of the dimension.
   Use the scroll wheel if multiple objects appear at the same point in the Design window.
6. Enter a value.
7. Press **Enter** to accept the value and complete the move or pull.

Press **Esc** to hide the ruler dimension.

You can make more than one change per ruler dimension.

5.8.6. Creating an Angular Ruler Dimension



1. Select the **Move** tool and switch to **Section** mode.
2. Select the section line (that represents a face) that you want to rotate.
3. (Optional) Anchor the **Move** handle to the object around which you want to rotate by dragging the center sphere or using the **Anchor** tool guide.
4. Select the rotational axis of the **Move** handle.
5. Select **Create Ruler Dimension** from the **Options** panel or right-click and select it from the mini-toolbar.

   An angular dimension indicator appears from the red linear axis of the **Move** handle.
6. Select the end reference for the angular dimension.
7. Enter a value for the dimension.

5.9. Detach

You can detach individual pieces of a sketch, or detach objects or faces in 3D. You can detach protrusions to move them with the **Move** tool's **Detach First** option.

This section contains the following topics:

5.9.1. Detaching in 2D

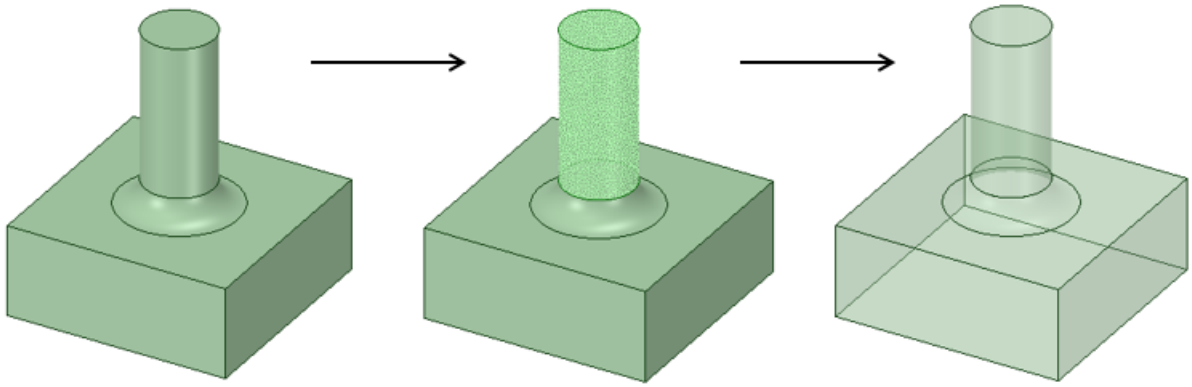
Alt+drag with the **Select** tool to detach the selected item when sketching. Use the **2D Move** tool to detach items and move them.

5.9.2. Detaching in 3D

1. Click the **Edit** > **Select** tool.
2. If you want to detach an object, **Ctrl+**click all its faces to select them.
3. Right-click the object and select **Detach** from the context menu.

5.9.3. Detaching a Cylinder

1. Click the **Edit** > **Select** tool.
2. If you want to detach a cylinder, **Ctrl+**click the cylinder surfaces to select them.
3. Right-click the object and select **Detach** from the context menu.

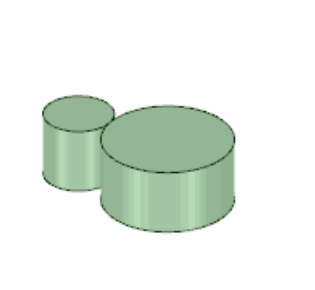


5.10. Assemble

This section contains the following topics:

5.10.1. Tangent Tool

The **Tangent** tool enables if a model is loaded, and it aligns two faces so they are tangent or aligns a face tangent with a line, point, or plane. Possible face types include planes, cylinders, spheres, and cones.



Note: You can use the **Tangent** tool to move objects without assigning assembly conditions. If the objects belong to different components, uncheck **Create conditions** in the Assembly Options panel before moving.

This section contains the following topics:

5.10.1.1. Defining a Tangent Condition

1. Click **Tangent** in the **Assembly** group on the **Design** tab.
   The **Align** tool guide is enabled by default.
2. Select an edge or face of the component that you want to move.
   You can Ctrl+click to select multiple objects.
   The **Reference** tool guide is enabled.
3. Select an edge or face of the component that you want to remain stationary.
   The components align.
   The constraint is added to the components in the model tree.

If you want to align to the other side of the plane, right-click the assembly condition in the model tree and select **Reverse Sense**.

5.10.1.2. Doing It Faster

Hold **Ctrl** and select the component that you want to move and the component that you want to remain stationary, then click **Tangent** in the **Assembly** group on the **Design** tab.

Reversing the Sense of a Tangent Condition

Right-click a **Tangent** condition in the model tree and select **Reverse Sense** to align the components to the opposite side of the alignment plane.

You can also modify the **Reversed** value to **True** or **False** in the **Assembly Condition** section of the **Properties** panel.

5.10.1.3. Defining an Offset Distance for a Tangent Condition

1. Select the **Tangent** condition in the model tree.
2. Change the angle value in the **Offset** property in the **Properties** panel.

5.10.1.4. Making Objects Tangent without an Assembly Condition

1. Click **Tangent** in the Assembly group on the Design tab.
   The Align tool guide is enabled by default.
2. Uncheck **Create conditions** in the Options panel.
3. Select an edge or face of the object that you want to move.
   You can Ctrl+click to select multiple objects.
   The Reference tool guide is enabled.
4. Select an edge or face of the object that you want to remain stationary.
   The components align.

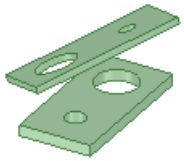
5.10.1.5. Making Objects Tangent Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Align	The **Align** tool guide allows you to select the component to move.
Reference	The **Reference** tool guide allows you to select the component to remain stationary.

5.10.2. Align Tool

The **Align** tool aligns two points, lines, planes, origins, or a combination of these elements. If you select a cylindrical or conical face, then the axis is aligned. If you select a spherical face, then the center point is aligned. You can define a ball joint assembly condition using Align by selecting the face of the ball and then the face of the socket. The ball rotates within the socket no matter where you place the move handle on the ball part.



Note: You can use the Align tool to move objects without assigning assembly conditions. If the objects belong to different components, uncheck **Create conditions** in the Assembly Options panel before moving.

This section contains the following topics:

5.10.2.1. Aligning Objects

1. Click **Align** in the Assembly group on the Design tab.
   The Align tool guide is enabled by default.
2. Select an edge or face of the component that you want to move.
   You can Ctrl-click to select multiple objects.
   The Reference tool guide is enabled.
3. Select an edge or face of the component that you want to remain stationary.
   The components align.
   The constraint is added to the components in the Design tree.

Note: The **Align** tool aligns objects within a tolerance of 0.1 mm. This tolerance cannot be modified.

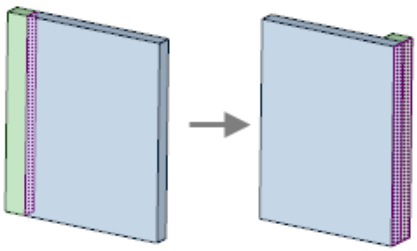
5.10.2.2. Doing It Faster

Use the Select tool to click the component you want to move, then Ctrl+click the component you want to remain stationary. Then click **Align** in the Assembly group on the Design tab.

5.10.2.3. Reversing the Sense of an Align Condition

Right-click an **Align** condition in the model tree tree and select **Reverse Sense** to align the components to the opposite side of the alignment plane.

You can also modify the **Reversed** value to **True** or **False** in the **Assembly Condition** section of the **Properties** panel.



5.10.2.4. Defining an Offset Distance for an Align Condition

1. Select the **Align** condition in the model tree.
2. Change the angle value in the **Offset** property in the properties panel.

5.10.2.5. Aligning Objects without an Assembly Condition

1. Click **Align** in the Assembly group on the Design tab.
   The Align tool guide is enabled by default.
2. Uncheck **Create conditions** in the Options panel.
3. Select an edge or face of the object that you want to move.
   You can Ctrl+click to select multiple objects.
   The Reference tool guide is enabled.
4. Select an edge or face of the object that you want to remain stationary.
   The components align.

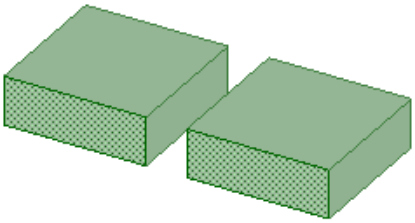
5.10.2.6. Align Objects Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

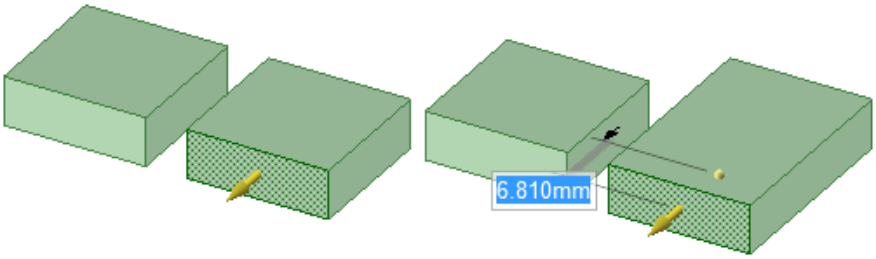
Align	The **Align** tool guide allows you to select the component to move.
Reference	The **Reference** tool guide allows you to select the component to remain stationary.

5.10.2.7. Aligning Objects Examples

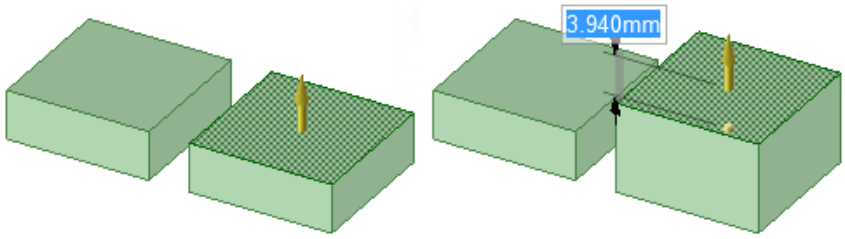
*Clicking the **Align** assembly condition in the model tree highlights the faces.*



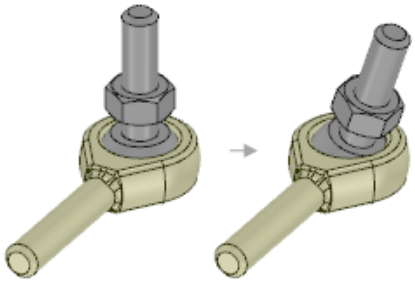
Pulling one face of the aligned pair changes one object and moves the other.



Pulling an adjacent face has no affect on the aligned pair.

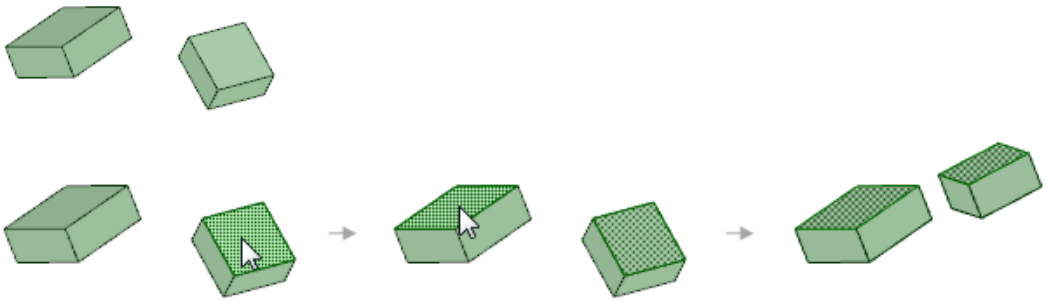


Defining a ball socket using align.



5.10.3. Orient Tool

The **Orient** tool enables if a model is loaded, and it rotates components so the selected elements are oriented in the same direction.



> **Note:** You can use the **Orient** tool to move objects without assigning assembly conditions. If the objects belong to different components, uncheck **Create conditions** in the Assembly Options panel before moving.

This section contains the following topics:

5.10.3.1. Defining an Orient Condition

1. Click **Orient** in the **Assembly** group on the **Design** tab.
   The **Align** tool guide is enabled by default.
2. Select an edge or face of the component that you want to move.
   You can Ctrl+click to select multiple objects.
   The **Reference** tool guide is enabled.

3. Select an edge or face of the component that you want to remain stationary.
   The components align.
   The constraint is added to the components in the model tree.

5.10.3.2. Doing It Faster

Hold **Ctrl** and select the component that you want to move and the component that you want to remain stationary, then click **Orient** in the **Assembly** group on the **Design** tab.

Reversing the Sense of an Orient Condition

Right-click an Orient condition in the model tree and select **Reverse Sense** to align the components to the opposite side of the alignment plane.

You can also modify the **Reversed** value to **True** or **False** in the **Assembly Condition** section of the **Properties** panel.

5.10.3.3. Defining an Offset Angle for an Orient Condition

1. Select the **Orient Direction** condition in the model tree.
2. Change the angle value in the **Offset** property in the **Properties** panel.

5.10.3.4. Orienting Objects without an Assembly Condition

1. Click **Orient** in the Assembly group on the Design tab.
   The Align tool guide is enabled by default.
2. Uncheck **Create conditions** in the Assembly Options panel.
3. Select an edge or face of the object that you want to move.
   You can Ctrl+click to select multiple objects.
   The Reference tool guide is enabled.
4. Select an edge or face of the object that you want to remain stationary.
   The components align.

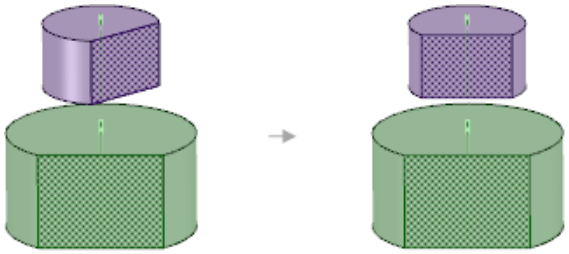
5.10.3.5. Orienting Objects Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Align	The **Align** tool guide allows you to select the component to move.
Reference	The **Reference** tool guide allows you to select the component to remain stationary.

5.10.3.6. Orienting Objects Example

Orienting two components that have an Align constraint on their axes turns the second part you select.



5.10.4. Rigid Tool

The **Rigid** tool locks the orientation and position of two components to each other.

> **Note:** The **Rigid** tool is enabled when you select appropriate objects that belong to different components.

This section contains the following topics:

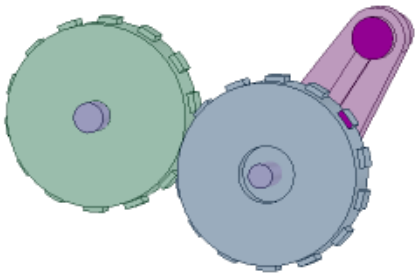
5.10.4.1. Defining a Rigid Condition

1. Hold **Ctrl** and select an edge or face of the two components that you want to remain stationary with each other.
   You can also select two components in the model tree.
2. Click **Rigid** in the **Assembly** group on the **Design** tab.
   The constraint is added to both components in the model tree.

5.10.5. Gear Tool

The **Gear** tool constrains two objects so one of the objects rotates in response to the rotation of the other object. Gear conditions can be created between two cylinders, two cones, a cylinder and a plane, or a cone and a plane.

> **Note:** The **Gear** tool is enabled when you select appropriate objects that belong to different components.



This section contains the following topics:

5.10.5.1. Defining a Gear Condition

1. Hold **Ctrl** and select an edge or face of the two component that you want to define as gears.

2. Click **Gear** in the Assembly group on the Design tab.
   The constraint is added to the components.

5.10.5.2. Reversing the Sense of a Gear Condition

Right-click a Gear condition in the model tree and select **Reverse Sense** to align the components to the opposite side of the alignment plane.

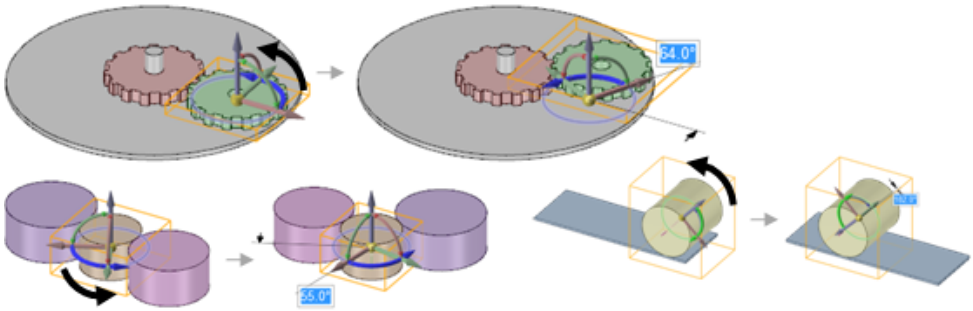
You can also modify the **Reversed** value to **True** or **False** in the Assembly Condition section of the Properties panel.

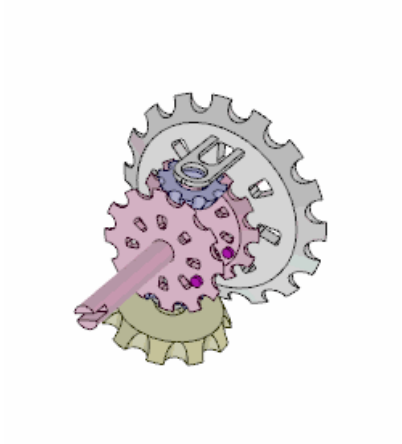
5.10.5.3. Reversing the Rotation Direction of a Gear Condition

1. Select the Gear condition in the model tree.
2. Right-click and select **Reverse Rotation Direction**.
   By default, gears rotate in the opposite direction to the gear turning them. You can reverse this to simulate a belt or chain drive, in which both gears spin in the same direction.

5.10.5.4. Gear Tool Examples

Several examples of gear constraints





The animated example shows how anchoring different parts in the assembly affects the behavior of the gears. First the gray component is turned without anchoring it or the rose components. Then the rose component that is highlighted is anchored and the gray component is turned. Watch the purple buttons on the rose components to see the difference.

5.10.6. Anchor Tool

The **Anchor** tool fixes the position of a single component in 3D space.

> **Note:** The **Anchor** tool is enabled when you select an appropriate object that belongs to a component.

This section contains the following topics:

5.10.6.1. Defining an Anchor Condition

1. Select an edge or face of the component that you want to remain stationary.
   You can also select the component in the model tree.
2. Click **Anchor** in the Assembly group on the Design tab.
   The constraint is added to the component. If the component is selected with the Move tool, the move handle is disabled.

To remove the anchor condition, right-click the **Anchor** object under the component in the model tree.

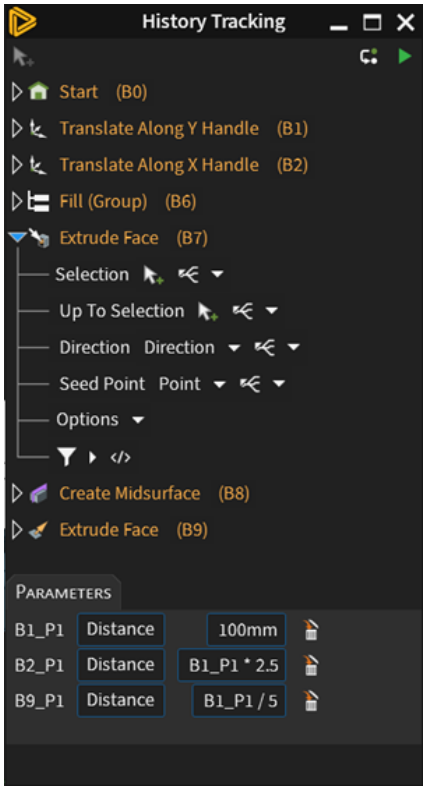
5.11. Automate

The **Automate** group includes the following:

- **History Tracking** for recording modeling and/or physics changes in script blocks and replaying operations or workflows.
- **Script Editor** for creating and running scripts to perform repetitive tasks.
- **Extension Builder** for creating custom extensions to easily add buttons and tools to groups and tabs on the ribbon.

5.11.1. History Tracking

History Tracking can be used for replaying operations or workflows after a CAD update or in parametric scenarios. Modeling and/or physics changes are recorded in script blocks. As you perform operations, they are recorded in steps, with each step representing a script block that can be used to recreate that part of your workflow. You can make changes to the settings and parameters, and play back the operations to recreate the workflow.



> **Note:** When you stop tracking history, the window closes. Additional modeling or physics changes will delete the history.

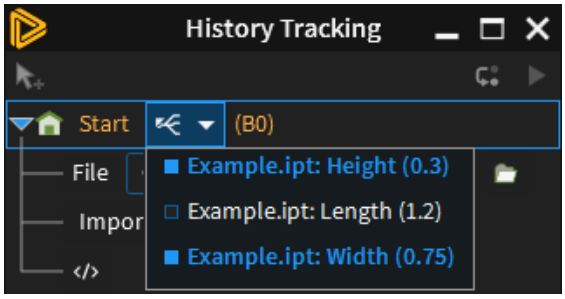
Known Limitations and Caveats

- History tracking is available when you are in the Model, Explore, and Refine stages. It is not available:
  - in the Refine stage, when using multiple simulations,
  - when multiple graphics display tabs are open,
  - when using the Script Editor or Extension Builder.
- Physics changes are represented by a single Simulation step. This step records a snapshot of the physics, which is then reapplied after the workflow or out of date steps are replayed.
- Physics changes within custom blocks are not supported. Any physics changes in a custom block will be overwritten when the Simulation step is reapplied at the end of the workflow.

5.11.1.1. Using History Tracking

To track your modeling and/or physics workflow:

1. Select **History Tracking** in the **Automate** group in the **Design** tab.
2. If you import the CAD model with existing parameters via the associative interfaces, the Start step captures the import of the model, along with the Import Options used. Parameters set up in your source CAD system are also available in the Start step.



If you make changes to the model in the source CAD system, you can right-click the Start step and select **Update From Source CAD** to update the model.

3. Set up the modeling and/or physics workflow.

   Note that the physics changes are not recorded as individual history steps, but they are represented by a single Simulation step.

4. Make changes to the set up:

   - For modeling steps, edit individual steps to change options or parameters. You can also make parametric changes using parameters from the source CAD system or the parameters defined.
   - For physics changes, you can update the geometry using parameters from the source CAD system. Make changes to the simulation set up as required.

5. Replay the workflow from the Start or for the out of date steps to recreate the workflow.

   Note the block states to verify that the workflow replayed completely. You may need to address blocks that are out of date, unplayed, or show errors or warnings for the workflow to be recreated correctly.

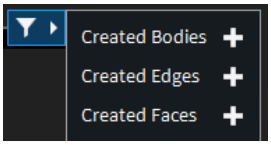
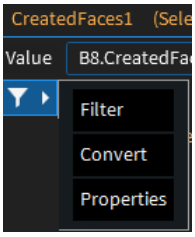
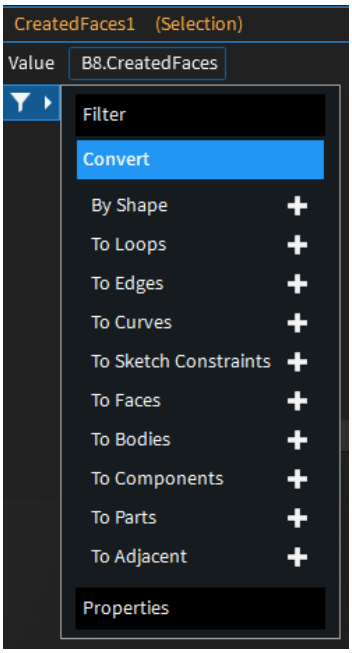
5.11.1.2. History Tracking Options

To work with an individual step in the **History Tracking** window, expand and edit individual step properties for changing options, parameters etc., and apply the changes to the overall design of the model. You can also change the parameters in the source CAD system.

View and edit the options for the operation by clicking the **Options** dropdown in the step.

	Edit and track the changes to the sketch. Click **Complete** to update the step with the sketch changes or click **Cancel** to retain the previous sketch.
	• Click in the main **History Tracking** window to create a new selection step to record geometry selections. • Click within a step to open the **Select** HUD for selecting a new reference or adding to the selection. Click **Complete** to update the step with the new selection or click **Cancel** to retain the previous selection.

	Create a variable or add a parameter with a unique label. You can also click the dropdown menu and select **Create Variable** or **Add Parameter** to create a variable or add an existing parameter. • Each parameter has a unique label and is added to the **Parameters** list. • You can use the parameters to define other variables using basic mathematical expressions. **Note:** When working with expressions, you must be careful with units.

	Click **Output Options** to create output parameters. • Create a parameter for the bodies, edges, or faces created by clicking the + icon next to the appropriate selection.  • Convert or filter created selections based on size, geometry, or other requirements to improve selection. Choose from the **Convert**, **Filter**, and **Properties** methods available.  You can convert selections to types (such as faces or edges), or convert by shape, size, etc. You can choose to filter faces, edges, parts, bodies, or components or select based on properties such as plane, normal, smart variables, area, perimeter, etc. Click the + icon next to the appropriate selection. 

</>	Click **Edit script** to view and/or edit the block script in the **Script Editor**. **Note:** You can only view a single block script at a time. • Click  to insert the current selection as a script snippet. • Click  to have the debugger execute the function call and then pause after the function returns. Choose to cancel or accept the script changes when you exit the **Script Editor**. Refer to Script Editor on page 748 for detailed descriptions of the **Script Editor** functions.
	Move to the next **Step** in the workflow.
	Click **Continue** to play through all the steps to bring you to the last step in the workflow.
Options	Right-click a step and select from these options: • **Rename** • **Move To** - Move to the state defined by a specific step. • **Force Regenerate** - Force Regenerate the selected step. • **Delete** - Delete the selected step. • **Cut** / **Paste Below** - Cut the selected steps and paste them below the selected step. • **Insert Custom Block** - Insert a custom block. Edit the custom block script to create a custom block. • **Suppress** / **Unsuppress** - Suppress or Unsuppress the selected state. • **Group** - Group selected blocks together. • **Hide Non-Model Changing Blocks** - Hide non-model changing blocks, for example, changing the visibility of objects. This option is enabled by default. • **Collapse All** - Collapse all steps.

6: Changing the Display

The tools you use to customize the appearance of your design within the design window are found in Discovery's Display tab.

- Customize your design by modifying which objects are displayed, the style in which solids and edges are displayed, and the color in which solids appear in your design.
- Paint display properties from one object to another.
- Create layers to save different customizations and display characteristics.
- Show or hide workspace tools.

This section contains the following topics:

6.1. Clipboard

Select a tool from the Clipboard ribbon group to cut, copy, or paste any 2D or 3D object. You can use these tools any time, even when you are designing with other 2D or 3D tools.

Objects are pasted into the active component. Objects that can be named (components, bodies, axes, or planes) will retain their names when pasted.

When you cut and paste a solid or surface, the object retains its color and face style.

This section contains the following topics:

6.1.1. Pasting an Object into the Graphics Window

1. Cut or copy the object.
2. Select the **Paste** tool from the Clipboard ribbon group.

   You can also press **Ctrl+V** or right-click in the graphics window and select **Paste** from the context menu.

   If you do not specify a location to paste, the copy is placed in its original location relative to the world origin.

   When you paste an object and it is located outside of the visible area in the graphics window, the zoom is adjusted to make the object visible.

> **Note:** Face edges that are Copied and Pasted create Curves. In some cases, there may be a very small deviation between the curve and the underlying edge that was copied. Most cases involve spline edges because of the way spline curves are constructed. A small number of cases may occur where lines and arcs connect due how the connecting vertex is calculated.

6.1.2. Pasting an Object onto a Face

1. Select all the faces of the object.
2. (Optional) Alt+click the face of the object you want to attach to the face.

   The face is highlighted in blue.
3. Cut or copy the selection.

4. Click the face onto which you want to paste the object.
5. Paste the object.

   The object is pasted on the face at the location you clicked. The blue face is attached to the clicked face.

6.1.3. Pasting a Round onto Another Edge

1. Select the face of the round.
2. Cut or copy the face.
3. Click the edge onto which you want to paste the round.
4. Paste the round.

6.1.4. Cutting an Object

1. Select the object.
2. Select the **Cut** tool from the Clipboard ribbon group.

   You can also press **Ctrl+X** or right-click in the graphics window and select **Cut** from the context menu. If you cut the face of a solid, it is converted to a surface.

6.1.5. Copying an Object

1. Select the object.
2. Select the **Copy** tool from the Clipboard ribbon group.

   You can also press **Ctrl+C** or right-click in the graphics window and select **Copy** from the context menu.

6.1.6. Deleting an Object

1. Right-click the selected object (or set of objects).
2. Press **Delete**.

   If you want to delete something and fill the gap with neighboring geometry, or create faces in the gap, you should use the Fill tool.

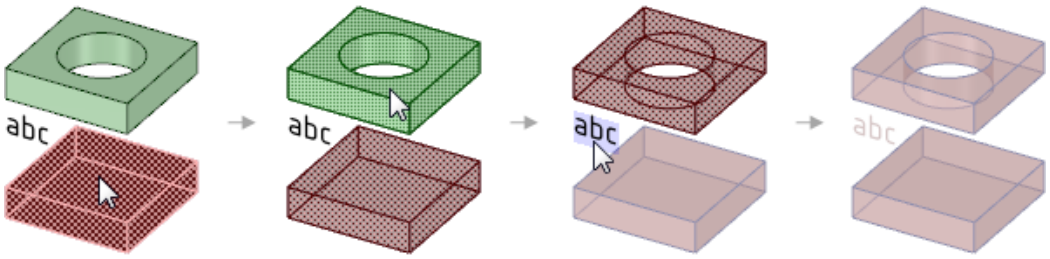
   If you delete the face of a solid, it is converted to a surface.

6.1.7. Style Painter

Use the **Style Painter** tool to apply the display properties of one object to another object. The tool applies color and transparency intelligently from one object type to a different object type.

You can select a component in the model tree as the target for the **Style Painter** tool. The color and transparency are applied to all solids in a component. You cannot select a component as the source object, and only the solids in the target component are changed.

In the example below, the red part is semi-transparent and has blue edges. The style from the red block was painted to the green block and to the note. The color, transparency, and edge color of the red block was copied to the green block. Only the color of the red block was copied to the note, because a note doesn't have transparency or edge color.



6.1.7.1. Painting Display Properties from One Object to Another

1. Click **Style Painter** in the Clipboard group.
2. Select the source object that you want to copy the display properties from.
3. Select the target object that you want to copy the display properties to.

   The **Select Target** tool guide automatically becomes active so you can paint the properties of the source to as many targets as you like by simply clicking on them.

   If you can't select the object you want, such as an edge, try scrolling the mouse wheel over the object.
4. If you want to paint properties from a different source object, click the **Select Source** tool guide or hold **Ctrl** and select a different source object.

   The **Select Target** tool guide automatically becomes active again so you can click as many targets as you want.

6.2. Orient

Select a tool from the Orient ribbon group to orient your design in the workspace. You can use these tools at any time, even when you are designing with other 2D or 3D tools.

> **Note:** Drag the middle mouse button to spin, Shift+drag it to zoom, and Shift+drag it to zoom. You can switch between spinning, panning, and zooming by pressing and holding Ctrl (to pan) or Shift (to zoom). When you release the key, you will return to spinning.

This section contains the following topics:

6.2.1. Spin Tool

When you click the **Spin** tool, it stays enabled until you click it again, press **Esc**, or click another tool.



This section contains the following topics:

6.2.1.1. Spinning Your Design

1. Click **Spin** in the Orient group on the Design tab.

   You can also select one of the following from the **Spin** dropdown:

   - **On Center** to spin around the center of your design.
   - **On Cursor** to spin around the cursor location.

2. Click and drag to spin your design.

   You can also rotate around a highlighted object. Select a line, edge or axis, then press **Alt** and drag. You can set this as the default spin action by setting the advanced Discovery option **Rotate about preselected object in spin**. This option also applies when using the middle mouse button to spin when working in any tool.

   You may find it easier to reach a desired orientation if you use short mouse drags to spin the design a little bit at a time.

   If you double-click a face while using the Spin tool, the face is zoomed so that it fills the design window. You can always zoom using the scroll wheel, or Shift-click the middle mouse button and drag up or down.

6.2.1.2. Setting the Spin Center

Right-click anywhere in the graphics window, select **Spin Center,** and select one of the following:

- **Set**: Sets the spin center to the center of the object you have selected. If you don't have an object selected, the spin center is set to the spot where you right-click. If you previously set the spin center, setting it again will change it to your current selection. You don't need to clear the spin center to set it again.
- **Clear**: Clears the spin center. When you rotate your view in 3D, the spin will be centered based on the option you select in the Spin tool.
- **Locate**: Centers the view on the spin center, if one is set.

6.2.1.3. Displaying the Spin Center Indicator

Select **Spin Center** in the Show group on the Display tab to see an indicator of your spin center in the graphics window when you spin your view. The indicator only appears while you spin, and it changes when you have a spin center set. The default indicator is shown below on the left. The indicator below is shown when you have the spin center set.

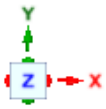


6.2.1.4. Orientation Shortcuts

An orientation indicator is located in the lower left corner of the graphics window. You can use it to snap to views and rotate the view.



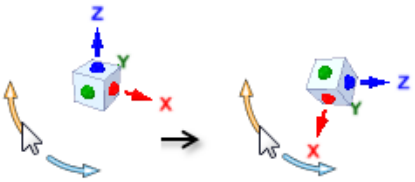
Click an axis or one of the colored balls to snap the view to snap that axis normal to the screen. The balls are the intersection of an axis on the opposite side of the block from it's label. The image below shows the result of clicking on the Z-axis.



As the mouse gets near the indicator, arrow arcs appear.



Click an arrow to rotate it 90° in that direction.



You can also drag on the arrow to rotate the view. As you drag, the view rotates in 5° increments. To rotate smoothly, press and hold the Shift key while dragging.

6.2.2. Pan Tool

Use the **Pan** tool to move your design within the graphics window.

When you click the **Pan** tool, it stays enabled until you click it again, press **Esc**, or click another tool.



6.2.2.1. Panning Your Design

1. Select the Pan tool from the Orient ribbon group or status bar.
2. Drag to move your design around the graphics window.

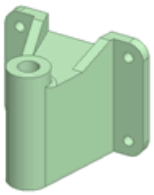
   If you double-click a face while using the Pan tool, the face is zoomed so that it fills the design window. You can always zoom using the scroll wheel, or Shift-click the middle mouse button and drag up or down.

> **Note:** When working in any tool, Ctrl+click the middle mouse button and drag to move your design.

6.2.3. Zoom Tool

Use the **Zoom** tool to display your design closer or farther away in the graphics window. You can zoom the design to fill the graphics window, zoom into an area, or zoom in or out a preset amount.

When you click the **Zoom** tool, it stays enabled until you click it again, press **Esc**, or click another tool.



This section contains the following topics:

6.2.3.1. Zooming In and Out

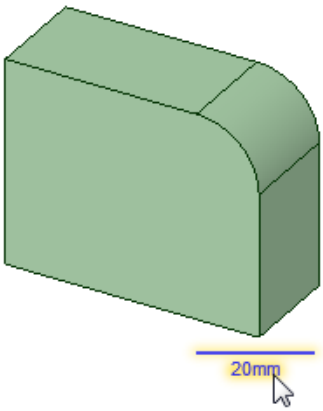
1. Select the **Zoom** tool from the Orient ribbon group or status bar.
2. Click where you want to center the zoom.
3. Drag down to zoom into your design; drag up to zoom out.
   You can also use the scroll wheel to zoom in and out.

> **Note:** When working in any tool, you can use the scroll wheel to zoom, or Shift-click the middle mouse button and drag up or down. You can also press **Ctrl+** or **Ctrl-** to zoom in or out a preset amount.

6.2.3.2. Using the Zoom Legend

You can toggle the Zoom Legend on and off using the Show menu in the Display ribbon group.

The Zoom Legend highlights when you hover over it, indicating you can click and drag it to a new location.



6.2.3.3. Zooming the Design to Fit the Design Window

Select **Zoom > Extents** or press **Z**.

The design is zoomed so that it fills the graphics window. When working with a drawing sheet, it will fit the drawing sheet to the Design window. If you resize the graphics window, the design will also be resized until it again fills the graphics window.

6.2.3.4. Zooming into a Selected Area

1. Select **Zoom > Zoom Box In**.

   You can also right-click in the graphics window and select **View > Zoom Box In** from the menu.

2. Click and drag to select the area.

   Two rectangles appear as you draw. The dotted rectangle shows your selection; the solid rectangle shows what will be displayed in the graphics window. When you mouse-up, the design pans and zooms until it fits within the area.

6.2.3.5. Zooming In and Out a Preset Amount

Select **Zoom > Zoom In** to bring your design closer. Select **Zoom > Zoom Out** to move your design further away.

When working in any tool, press **Ctrl** and **+** or **Ctrl** and **-** to zoom in and out a preset amount.

6.2.4. Switching Between the Spin, Pan, and Zoom Commands

While one of the Spin, Pan, and Zoom commands in the status bar is active, you can switch to any of the others using the **Ctrl** and **Shift** keys or the middle-mouse button.

When the Spin or Pan tools are active, you can always zoom using the scroll wheel.

To activate Spin or Pan when in another Orient tool:

Active Command	Press and Hold
Spin	**Shift** to pan
Spin	**Ctrl+Shift** to switch to the Pan tool
Zoom	**Shift** to pan
Zoom	**Ctrl+Shift** to switch to the Pan tool

6.3. Style

This section contains the following topics:

6.3.1. Color Usage

You can modify the color of edges, faces and bodies when selecting an object.

See Working with Layers to change the layer color or move an object to a different layer.

Use the **Style Override** tool in the Style ribbon group of the Display tab to override the layer color for solids, surfaces, faces, and curves.

If you change the color when nothing is selected, the color will become the default for new objects. This allows you to set the color for things you haven't created yet, without using layers and default layer colors.

6.3.1.1. Changing Object Transparency

By default, the transparency of an object is determined by its layer color and whether or not the object is a surface. Surfaces are displayed as semi-transparent to visually distinguish them from solids. You can override the transparency of objects using the **Color** tool.

In a graphics editing application, color is assigned to an Alpha channel (ARGB, for example). Discovery follows this standard convention and also treats transparency as a property of the object's color.

The hierarchy of rendering overrides is as follows:

- The opacity setting for the layer is used if no other transparency is set for the object.
- The transparency setting from the **Color** tool or **Style Painter** overrides the transparency of the layer.
- **Style Override** makes the object opaque or transparent, regardless of face or layer settings. These overrides are automatically deselected when you change the transparency of an object using the **Color tool** or **Style Painter** .

This section contains the following topics:

6.3.1.1.1. Changing an Object's Transparency

1. Select one or more solids, surfaces, faces, curves, or components in the graphics window or model tree.
   Hold the **Ctrl** key to select more than one object.
   If you select a component, then the color of all of the component's children will be changed.
2. Click **Color** in the Style group on the Display tab.
3. Choose the **Opacity** slider and adjust as appropriate.

6.3.1.1.2. Overriding an Object's Transparency

1. Select one or more solids, surfaces, faces, curves, or components in the graphics window or model tree.
   Hold the **Ctrl** key to select more than one object.
   If you select a component, then the color of all of the component's children will be changed.
2. Click **Style Override** in the Style group on the Display tab.
3. Select one of the options:
   - **Transparent** to make the object 35% transparent.
   - **Opaque** to make the object completely opaque.

   Deselect so neither override is selected if you want to use the object's transparency setting.

> **Note:** If any subset of the body's faces is made transparent, **Style Override** is disabled, because there is a mix of opaque and transparent faces.

6.3.1.1.3. Reapplying the Layer Color and Transparency

1. Select one or more solids, surfaces, faces, or curves in the graphics window or model tree.
   Hold the **Ctrl** key to select more than one object.
2. Select **Remove Style Override** from the **Color** tool in the Style group on the Display tab. You can see this option in the image of the Color tool above.

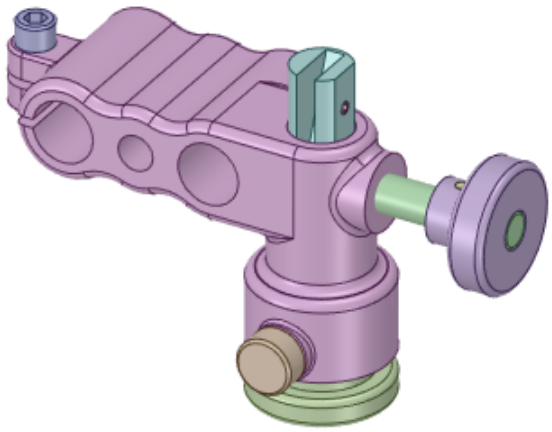
6.3.1.1.4. Applying Per-face Transparency

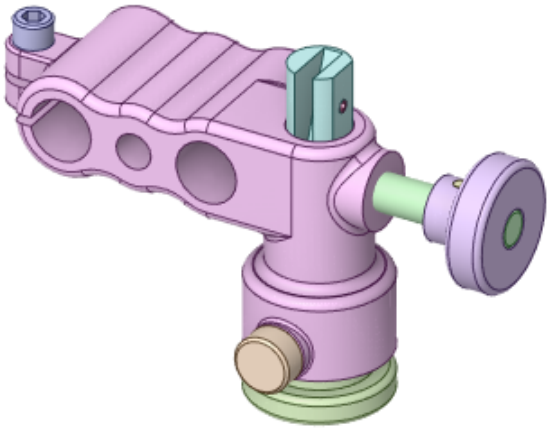
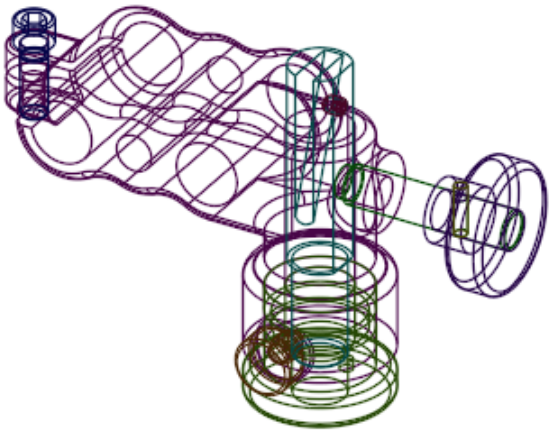
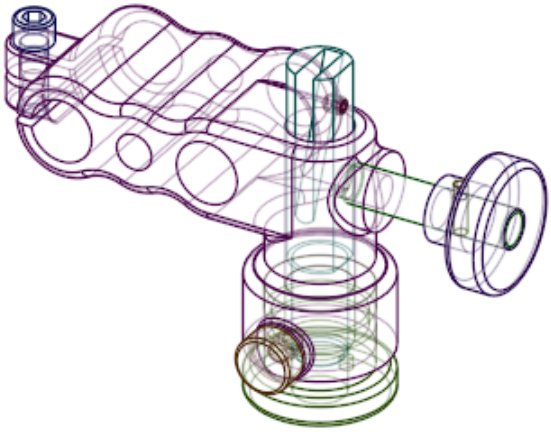
1. Select a face.
2. Open **Color** in the **Display** tab and set the **Transparency** for the selected face.
   By default, the **Style** property will be **By Color**, **By Style** so the **Body Style** overrides the **Face** style and the **Transparency** is overridden.
3. Open **Style Override** in the **Display** tab and note that **Opaque** is the active **Body Style**.
4. Click the active style (**Opaque**) to disable it.
   The face transparency displays.

6.3.2. Graphics Styles

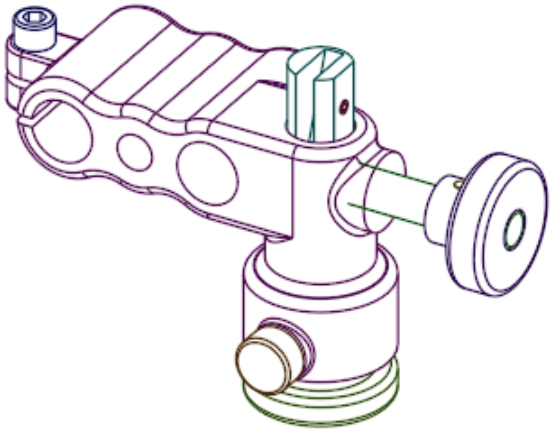
Discovery offers several different styles to view your design. You can apply styles to your entire design or drawing sheet, to individual layers, or to individual views in your drawing sheet.

Graphics styles are available from the **Graphics** tool in the Display tab of the Style group. All objects in the graphics window will be displayed in the style you select. The following styles are available:

Shaded: Displays solids and surfaces as three-dimensional, shaded objects.	

Enhanced Shaded: Solids and surfaces are displayed as three-dimensional, shaded objects (the default). The shading represents the exposure of the scene to ambient lighting, with shadows and highlights to improve rendering.	
Wireframe: Only the edges of objects are displayed. The wireframes are the same color as the edges.	
Hidden Line: Objects are displayed as wireframes with hidden lines displayed in a light gray.	

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

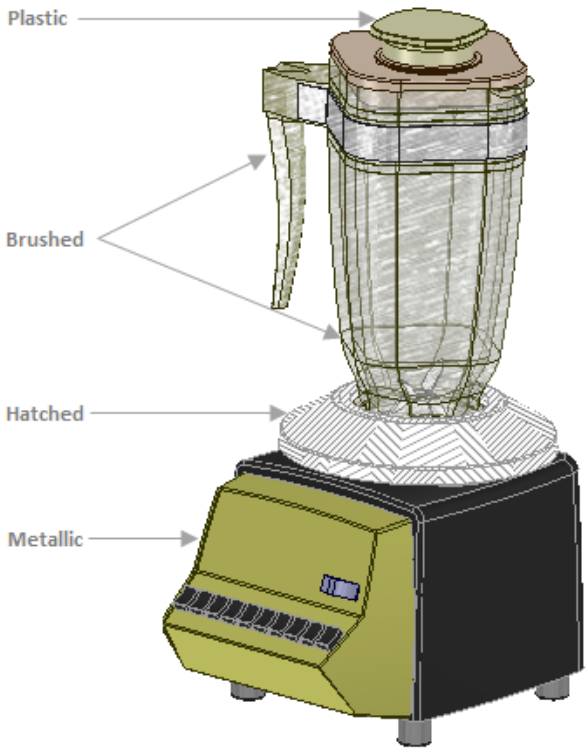
Hidden Line Removed: Objects are displayed as wireframes and hidden lines are not shown.	

6.3.3. Rendering Styles

The **Rendering Style** tool, in the Style group of the Display tab, allows you to change how solid faces are rendered. Assigning different rendering styles to a model helps you to better visualize parts, assemblies, or components as you develop your design. You can choose between:

- **Metallic**: a shiny, light reflective finish
- **Plastic**: a matte finish
- **Brushed**: a sketched finish
- **Hatched**: a hatch line finish
- **Finish**: a matte or gloss finish

All four rendering styles are illustrated below: the top of the pitcher is set to plastic, the handle and pitcher are brushed, the blade base is hatched, and the base of the pitcher is metallic.



6.3.3.1. Changing the Rendering Style of a Solid or Surface

1. Select the solid or surface.
2. Click **Rendering Style** in the Style group on the Display tab.
   This icon changes to reflect the current setting, but always appears as a shaded sphere.
3. Select **Hatched, Brushed, Metallic** or **Plastic** to apply that surface style to the selected solid or surface.

6.3.4. Style Overrides

Use the **Style Override** tool in the Style ribbon group of the Display tab to override the layer color for solids, surfaces, faces, and curves. The color of solids, surfaces, and components are normally determined by the layer color; however, you can override the layer colors for objects. By default, all objects are placed on Layer0. You can modify the color of edges, faces and bodies when selecting an object.

See Layers on page 406 to change the layer color or move an object to a different layer.

If you change the color when nothing is selected, the color will become the default for new objects. This allows you to set the color for things you haven't created yet, without using layers and default layer colors.

This section contains the following topics:

6.3.4.1. Overriding the Layer Color of a Solid, Surface, Face, or Curve

1. Select one or more solids, surfaces, faces, curves, or components in the graphics window or model tree. Select a body by selecting one of its faces or edges, then selecting **Body** as the **Target**.
   If you select a component, then the color of all of the component's children will be changed.

2. Click **Color** in the Style group on the Display tab.
3. Select a color. If the color you want is not in the list, click **More Colors** and select a color or create a new custom color.
4. Choose **Randomize Colors** to let the system assign colors to the selected objects
5. (Optional) Set the transparency of the object. See Changing object transparency.

6.3.4.2. Overriding Edge Color

1. Select any edge on a solid or surface.
2. Click **Color** in the Style group on the Display tab.
3. Select a color.

   This color will be used for all edges of the solid.

6.3.4.3. Removing Color Overrides and Returning the Object to the Layer Color and Transparency

1. Select one or more solids, surfaces, faces, curves, or components in the graphics window or model tree. You can select a body by selecting one of its faces or edges, then selecting **Body** as the **Target**.

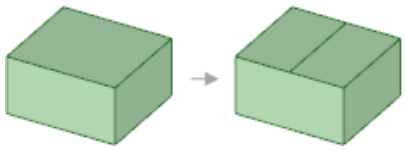
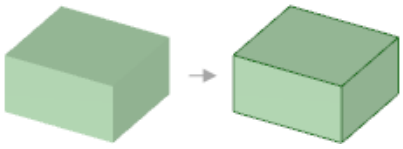
   If you select a component, the color of all of the component's children will be changed.
2. Click **Color** in the Style group and select **Remove Color Override**.

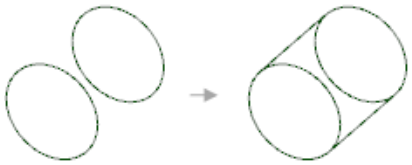
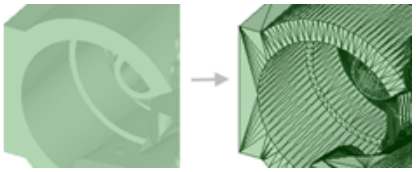
6.3.5. Edge Styles

Use the **Edges** tool in the Style group on the Display tab to customize which edges are displayed. These settings are saved with the document, but are only used for 3D display. The edge display options apply to the design in the current graphics window tab, and not other designs you may have open in other tabs in the graphics window.

See Applying colors to change the color of edges.

Edge options are available from the **Edges** tool in the Display tab of the Style group. The following styles are available:

Tangent: Display lines indicating tangent edges and edges that do not span a face.	
Surface: Display all edges on surfaces.	
Solid: Display all non-tangent edges on a solid.	

Silhouette: Display the lines that indicate the silhouetted edges of all curved surfaces. This option affects only Wireframe, Hidden Line, and Hidden Line Removed graphics styles	
Facets: Display facet edges on an STL model. You can switch the display of **Internal** mesh edges and **Open** meshes edges on boundaries. You can sketch on the mesh object and your sketch tools will snap to the facets.	
Soft: Display the soft edges of imported SketchUp models. This option is ON by default.	

6.3.6. Layers

Layers can be thought of as a grouping mechanism based on visual characteristics. Visual characteristics include visibility, color and line style. Layers are managed in the Layers panel and accessed and modified with the **Layer** tool in the Display tab of the Style group.

Layers are especially useful when you want to hide annotation planes.

Sheet metal parts have a set of default layers when the part is in an unfolded state. They are:

- Dimension
- Bend Lines Up
- Bend Lines Down
- Bend Dimensions
- Forms

6.3.6.1. Working With Layers

Click **Layers** in the Style ribbon group of the Display tab to open **Layers**. Select a layer and right-click to access the context menu to do any of the following:

Sort A - Z **Sort Z - A**	Sort layers alphabetically.
Add Layer	Create a new layer. This layer becomes the active layer. Any newly created objects are automatically placed on the active layer.
Duplicate Layer	Duplicate a layer.
Activate Layer	Select a layer to be the active layer. Any newly created objects are automatically placed on the active layer.
Assign to Layer	Select one or more objects to assign to the layer.
Show	Show the layer.

Hide	Hide the layer.
Hide Others	Hide the other layers.
Select Objects	Select the object
Rename	Rename the selected layer. **Note:** Layer0 cannot be renamed.
Delete	Delete the selected layer. **Note:** Layer0 cannot be deleted.
Remove Empty Layers	Deletes any layers that are empty.

6.4. Grid

You can modify whether the sketch grid is displayed, and how the geometry above or below the grid is displayed in each graphics window. You may want to use one style when you are examining a component, and another when you are creating new geometry within a component.

This section contains the following topics:

6.4.1. Showing or Hiding the Sketch Grid

Check the **Show Sketch Grid** box in the Grid ribbon group on the Display tab to display the sketch grid. The sketch grid appears in all the sketching tools. Displaying the sketch grid allows you to snap to grid lines and provides a visual cue to the orientation of your sketch within your design

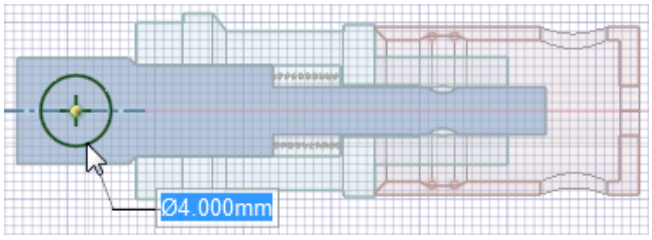
6.4.2. Determining How Geometry Above or Below the Grid is Displayed

- Check the **Fade Scene** box to make the geometry under the sketch grid more transparent.

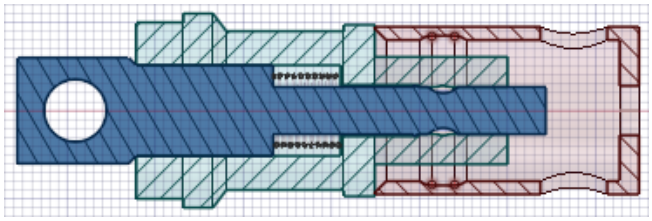
  Sketch curves outside the current sketch plane appear faded, while layout curves are unaffected.
- Check the **Clip Scene Above Grid** box to hide the geometry above the sketch grid.

6.4.3. Show Sketch Grid Examples

Fading the scene below the grid makes it easier to see when you sketch in section mode.



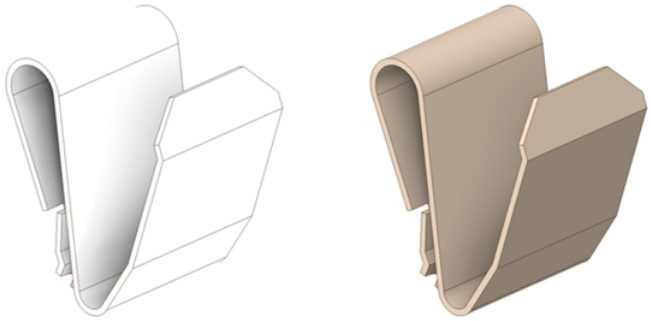
Examining a section with the scene clipped above the grid.



6.5. Display

You can adjust the display of workspace tools and the display of your design in the graphics window using the **Show** dropdown list of tools in the Display ribbon group on the Display tab.

World Origin	Check the **World Origin** box to display the axes that set the default orientation of the design in the Design window.
Spin Center	Check the **Spin Center** box to mark the center of the spin when using the Spin tool. (This is the same as the **Show Spin Center** Discovery option.)
Zoom Legend	Check the **Zoom Legend** box to display the zoom legend in the Design window.
Face Highlight	Check the **Face Highlight** box to enable pre-highlighting of faces.
Face Spotlight	Check the **Face Spotlight** box to show a spotlight attached to the cursor when passing over faces.
Body Glow	Check the **Body Glow** box to enable highlighting for pre-selection and selection of bodies. This setting applies to all windows.
Vertices	Check the **Vertices** box to show all vertices in the model. Vertices are not shown on closed periodic edges (for example, on a cylinder). This setting works on a per-window basis, enabling it in one window has no effect on other windows.
Tesselation	
Body Interference	Check the **Body Interference** box to show any bodies that are intersecting. This setting works on a per-window basis, enabling it in one window has no effect on other windows.

Ambient Occlusion Only	Check the **Ambient Occlusion Only** box to render the display with a diffuse, non-directional shading effect that approximates how light should be shining on any specific surface based on the light source and, if included, the environment. The shading represents the exposure of the scene to ambient lighting, for example, interior surfaces of a model are typically more occluded and will appear darker than the exposed outer surfaces. This option can be used when the Enhanced Shading option is enabled.  In the example, the image on the left is rendered with **Ambient Occlusion** and shows the shadows and highlights that are missing in the image on the right.
Adjacent Entities	Check the **Adjacent Entities** box to display faint highlighting on adjacent faces when you hover over an edge and on adjacent edges when you hover over a vertex. Scrolling the mouse wheel switches between adjacent entities. This feature is useful in selecting the correct edge or face to extrude.
Offset Baseline Faces	Check the **Offset Baseline Faces** box to display offset relationships with blue shading.
Standard Holes	Check the **Standard Holes** box display Standard Holes in blue. When unchecked, Standard Hole faces display in the normal face color.
Lightweight Components	Check the **Lightweight Components** box to display Lightweight components. This also has a flyout for setting Lightweight component transparency.
Environment	Check the **Environment** box to display according to the settings in the **Appearance** panel.

6.6. Visibility

The Discovery application offers several ways to change the visibility of your display, including Hiding and Showing Objects on page 409 and clipping, either with volumes or with planes.

This section contains the following topics:

6.6.1. Hiding and Showing Objects

You can use any of these methods for setting the visibility of objects in the graphics window:

Note: Shift+click and Ctrl+click multiple objects to work with them as a group.

View only a single object	Right-click an object in the graphics window and select **Hide Others** to view only that solid, surface, or curve.
Make all objects visible	Right-click anywhere in the graphics window and select **Show All** to make all objects visible.
Keep an object visible even if you hide its layer	Right-click an object in the model tree and select **Always Visible** from the context menu. The object will remain visible if you hide its layer, but this setting has no effect on other visibility commands such as **Isolate** or hiding its parent component by deselecting its check box in the model tree.
Hide an object	Deselect the check box in the model tree to hide the object in the graphics window. The object icon is displayed in gray. You can also right-click an object in the graphics window and select **Hide** to turn the visibility of the object off.
Set object to the layer visibility	Select the check box next to the object in the model tree to set the visibility of the object to the layer visibility. If the layer is hidden, then the object will be hidden. If the layer is visible, then the object will be visible. **Note:** If the layer visibility is on, the icon appears normally. If the layer visibility is off, the icon appears like the  icon. You cannot work with hidden objects in the graphics window.

You can use any of these methods for setting the visibility of faces:

Hide one or more faces	Right-click a face and select **Face** > **Hide Face** to hide the selected faces. Hidden faces allow objects behind them to be selected without using the scroll wheel.
Hide all faces except for those selected	Right-click a face and select **Face** > **Hide Other Faces** to hide all faces but those selected.
Invert the current visibility state for all faces	Right-click a face and select **Face** > **Inverse Face Visibility** to make visible hidden faces (even those hidden By Body) and hide any visible faces.
Show all faces	Right-click a face and select **Face** > **Show All Faces** to make all faces visible, whether they were hidden By Face or By Body

Note: You can also select **Show Faces** and **Hide Faces** from the **Visibility** section of the **Display** tab.

6.6.2. Clip with Volume Tool

The **Volume Clip** tool creates a spherical or cubic clip volume around an object so that you can selectively isolate a specific region, element, or section of a design. You have the option of a spherical or cubic volume, and if you want to clip your design interactively or based on a selection. You can also clip your design based on a specified location and size.

Using **Volume Clip** can be particularly helpful when you closely view and work on a specific feature of interest within a complex model. For example, after isolating a design area, you can use the **Missing Faces** tool to help detect and fix missing faces on a body.

As you develop a model, you can also work with **Plane Clip** to create a spherical clip volume around an object so that you can selectively isolate a specific region, element, or section of a design.

This section contains the following topics:

6.6.2.1. Clipping a Design Interactively

To clip the viewable and selectable geometry interactively by the radius of a virtual sphere or cube:

1. Select **Volume Clip** from the **Visibility** group of the **Display** tab.
2. Rotate and zoom in to your model to locate the model element or area you want to view.
3. Select a **Shape** for the volume clip, either a cube or sphere.
4. Specify the **Method** as **Set interactively**.
5. Select the model element or area and select **Clip with volume interactively** to clip the viewable and selectable geometry by the radius of a virtual volume.
6. Hover over the center of the element you want to view and then Click+drag to create your clip volume. The area you selected to preview magnifies and displays in a sphere as clipped, or isolated, from the rest of the model.
7. Hover over the area if you want to identify all of the various design elements.

Once you have created your clip, select **Clear the clip area** to re-display your design with no clipping. You can also change the size of your clip or move it using the **Resize the clip area** and **Move the clip area** tool guides on the left side of the HUD.

6.6.2.2. Clipping a Design Based on a Selection

To set the extent of the clip based on the current selection:

1. Select **Volume Clip** from the **Visibility** group of the **Display** tab.
2. Rotate and zoom in to your model to locate the model element or area you want to view.
3. Select a **Shape**, then set the **Method** to **Set from selection**.
4. Select the model element or area to clip based on the selected geometry and shape.
5. Select **Clip with volume at selection**. The model is clipped at the current selection based on the specified shape.
6. Hover over the area if you want to identify all of the various design elements.

Once you have created your clip, select **Clear the clip area** to re-display your design with no clipping. You can also change the size of your clip or move it using the **Resize the clip area** and **Move the clip area** tool guides on the left side of the HUD.

6.6.2.3. Clipping a Design Based on a Location

To set the extent of the clip based on the current selection:

1. Select **Volume Clip** from the **Visibility** group of the **Display** tab.
2. Rotate and zoom in to your model to locate the model element or area you want to view.
3. Select a **Shape**, then set the **Method** to **Use precise location**.
4. Select the model element or area to clip based on the selected geometry and shape.
5. Enter the **XYZ** coordinates (based on the World Origin) for the volume (sphere or cube) center, as well as the **Size** of the clip.
6. Select **Clip with volume at the precise location** to clip the viewable and selectable geometry based on the specified location and size.
7. Hover over the area if you want to identify all of the various design elements.

Once you have created your clip, select **Clear the clip area** to re-display your design with no clipping. You can also change the size of your clip or move it using the **Resize the clip area** and **Move the clip area** tool guides on the left side of the HUD.

6.6.2.4. Clip Volume Tool Guides and Options

Clip Volume Tool Guides

The following tool guides are available for Clip Volume:

Select	Set the clip shape, selection, or location.
Move clip	Move the clipped volume frame to a different area of the model.
Resize clip	Adjust the size of the clipped volume frame.

Clip Volume Tool Options

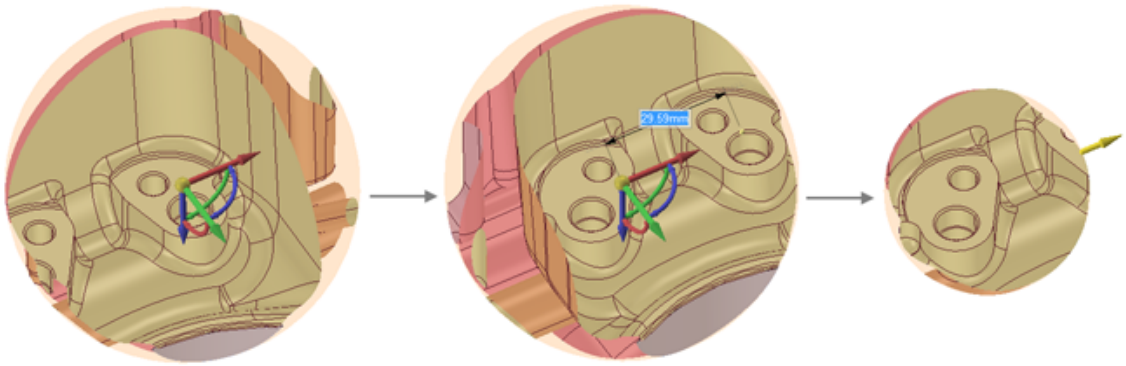
The following tool options are available for Clip Volume:

Clip	Create a clip based on the selected shape, selection, or location.
Clear the clip area	Clear the set selections.
Extract geometry	Cap off clipped boundaries according to clip type (by volume or by planes). Adds **Geometry Snapshot** to the model tree.
Shape	Set the shape of the clip volume to a sphere or cube.
Method	Specify how the clip is created. Choose from: • **Set interactively**: Create the clip volume interactively using Click+drag. • **Set from selection**: Set the virtual clip area based on the current selection. • **Use precise location**: Manually create the clip by entering the size of the clip and the coordinates (based on the World Origin) for the volume (sphere or cube) center.

6.6.2.5. Clip with Volume Examples

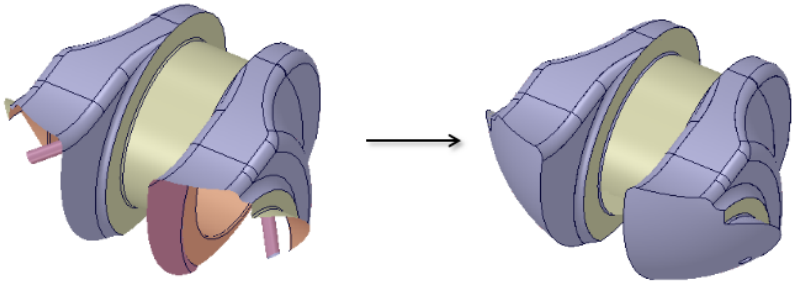
Examples

Use the **Move** option in the **Volume Clip** HUD to move the clipped volume frame to a different area of the model, or the **Resize** option to adjust the size of the clipped volume frame.

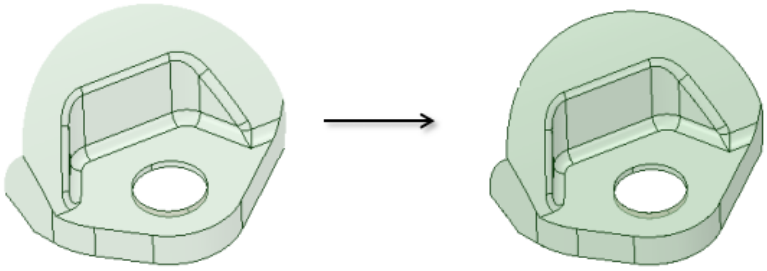


Use the **Extract Geometry** option to cap off clipped boundaries according to Clip type (spherical or cube volume or by planes). The result is shown in the model tree as a **Geometry Snapshot**.

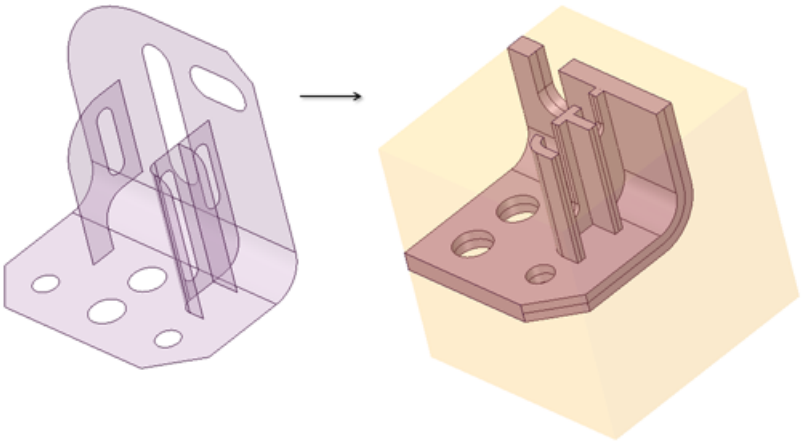
For intersections with solids, **Extract Geometry** creates a **Geometry Snapshot** that is a solid.



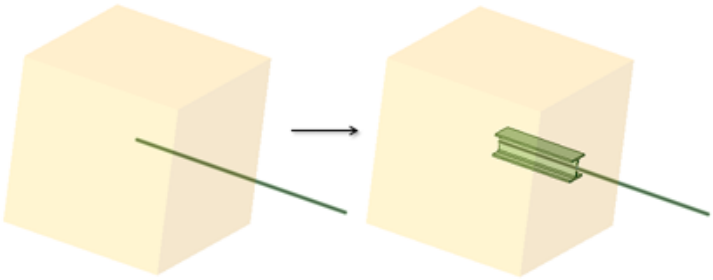
For intersections with surface models, **Extract geometry** also creates a **Geometry Snapshot**, but displays the surfaces with a more opaque, solid look.



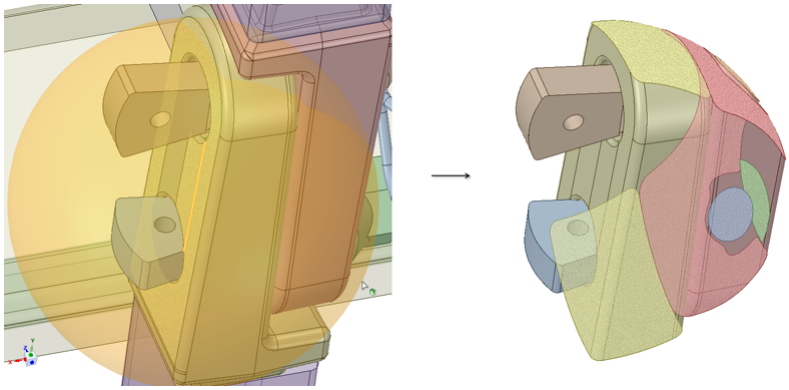
For intersections with mid-surface models, or any surface with an assigned thickness, **Extract geometry** creates a **Geometry Snapshot** with the assigned thickness.



For beams, **Extract geometry** creates a **Geometry Snapshot** showing their actual geometry.



Extract geometry also creates a named selections group that shows the faces created at the boundary.



6.6.3. Clip with Plane Tool

You can use multiple planes to quickly clip your design to show only the geometry on one side of a plane or the other.

The **Plane Clip** tool options enable you to clip using a plane, reverse the direction of the clipped view, and merge clips together. Planes, axes, and annotations are not clipped.

By default, when you create a plane, the model tree displays a plane icon. You can select a visible plane you want to clip from either the model tree or the graphics window, then select **Plane Clip** from the **Display** tab.

You can select up to ten planes to set a clipped volume. Planes selected for clipping display with orange boundaries in the graphics window.

As you develop a model, you can also work with **Volume Clip** to create a spherical clip volume around an object so that you can selectively isolate a specific region, element, or section of a design.

This section contains the following topics:

6.6.3.1. Clipping a Design with Planes

To clip a design with one or more planes:

1. Select a plane, or use Ctrl+Select to select multiple planes.
2. Select **Plane Clip** from the **Display** tab to display the tool HUD.
3. Select **Clip with plane** in the HUD.

The plane or planes you clipped display with a red outline. Icons update in the model tree. With the plane(s) selected, you can modify values in the tool options.

To restore the view of your design, select **Clear the clip area**. You can also reverse the direction of the clipped plane.

By default, when clipping is initially set, additional selected planes automatically combine using an "and" operation. This way you can isolate the inside of two parallel planes or the inside of three pairs of parallel planes (one of the most common usages for this feature).

With a group (two or more) of planes, you can also choose to merge them together. This option is essentially an "or" operation that defines a plane group that you intend to combine with the rest of the planes. Once you define and combine a group of planes, the merged set is put at the beginning of the logical set, and the rest of the planes follow with their default "and" states.

6.6.3.2. Reversing the Direction of a Clipped View

To reverse the direction of your clipped view:

1. Select a plane.
2. Select **View model on other side of plane**.

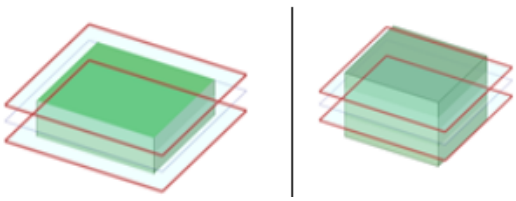
The geometry of your design flips to what is on the other side of the plane. To return to the original clipped view of your design, select the plane and select **View model on other side of plane** again.

6.6.3.3. Combining Clips

To merge two or more clips into a single clip:

1. Ctrl+Select more than one clipped plane.
2. Select **Combine clips**.

The image below shows the result of combining two planes. Sets of parallel planes default to keeping the material inside:



6.6.3.4. Plane Clip Tool Options

The following tool options are available for Clip Plane:

Plane Clip	Quickly clip your design to show only the geometry on one side of a plane.
View model on other side of plane	Flip the geometry view to what is on the other side of the plane. To return to the original clipped view of your design, select the plane and select **View model on other side of plane** again.
Combine clips	Merge two or more planes into a single clip.

7: Subdivision Surface Tools

The **Subdivision Surface** tools offer a modeling alternative to NURBS. In many cases, this approach lends itself to easier to manipulation and exploration of smooth, organic designs. The subdivision surface is a curved limit surface represented by a coarser polygonal control cage. You can begin modeling with a basic shape having a subdivision surface. After creating the basic shape, you can use the various tools in the **SubD** tab to further subdivide the surfaces and add detail to form more complicated shapes.

You can also convert an existing faceted or solid model to a subdivision surface or a hybrid model containing both subdivision surfaces and analytic faces such as planes or cylinders. You can then perform editing operations to add in details as needed.

You can:

- Use the tools in the **Create** group to create the basic shape or convert bodies to and from Subdivision surfaces. You can also create the warped body for a CAD part.
- Export results from a topology optimization as a hybrid model. The hybrid model contains both subdivision surfaces for regions of the model where you want to allow for flexible and freeform editing and traditional surfaces such as planes or cylinders for regions requiring precise topology.
- Use the tools in the **Edit** and **Modify** groups to make changes to the design.
- Use the options in the **View** group to view the **Limit Surface**, the **Control Cage**, or both. You can also use the **Show** dropdown to customize the model appearance while using the subdivision surface tools.

This section contains the following topics:

7.1. Creating Subdivision Surfaces

Use the **Primitive** tool in the **Create** group to create basic shapes.

1. Select the basic shape for the subdivision body to be created.

   You can choose to create a cube, pyramid, or torus.

2. Specify the origin and dimensions for the subdivision body. Select an existing face, edge, or vertex to position the subdivision body, if needed.

   The specified dimensions are used to create the control cage for the subdivision body, the actual subdivision body dimensions may vary.

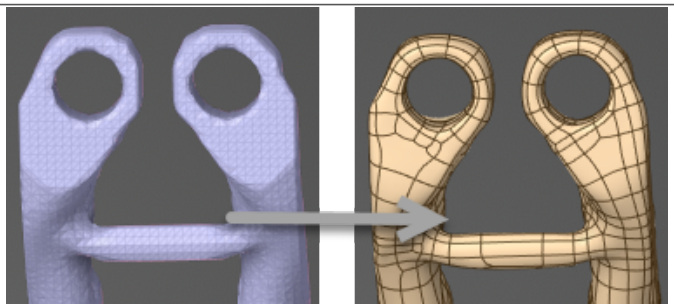
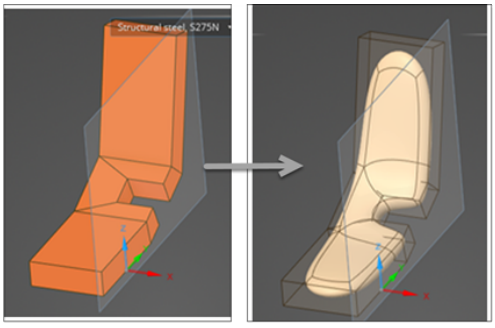
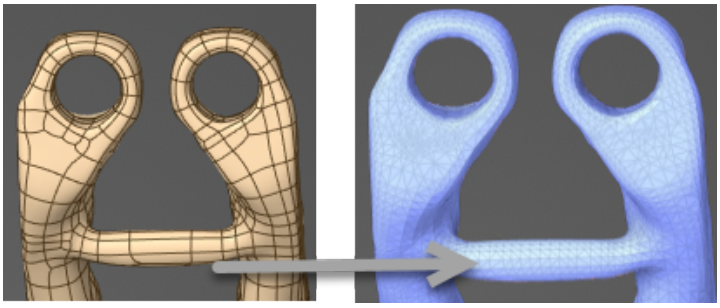
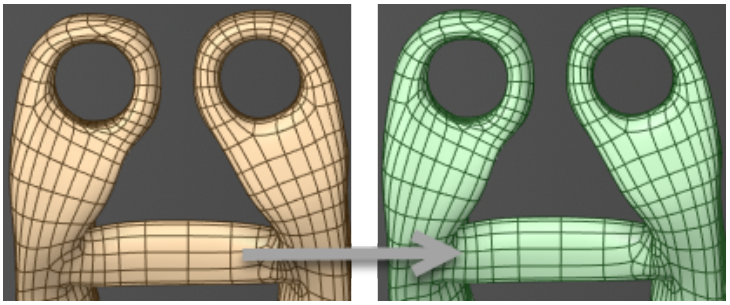
3. Click **Complete** to create the subdivision body.

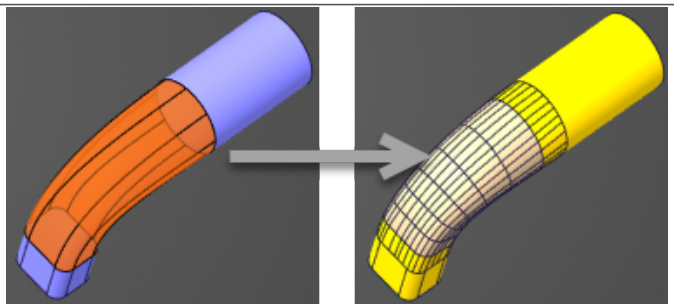
7.2. Converting To and From Subdivision Surfaces

Use the **Convert** tool to convert to and from subdivision surfaces.

1. Select the appropriate option for the **Conversion type**.

Option	Description
Facets to SubD	Use the **Facets to SubD** option to convert faceted bodies to subdivision surface bodies by specifying an appropriate **Fit tolerance**.

Option	Description
	
Solid to SubD Cage	Use the **Solid to SubD Cage** option to convert solid bodies to subdivision surfaces. This tool can be used for converting models which have faces with mostly four sides. 
SubD to Facets	Use the **SubD to Facets** option to convert subdivision bodies to faceted bodies. 
SubD to Solid	Use the **SubD to Solid** option to convert subdivision bodies to solid bodies. 
Region to SubD	Use the **Region to SubD** option to convert a selected region to subdivision surfaces. The resulting hybrid model contains subdivision surfaces for the selected region of the model and retains the remaining traditional surfaces.

Option	Description
	

2. Select the body to be converted. For the **Region to SubD** option, select the region of the model to be converted. The selected region should be tangent.

> **Note:** Currently, no two converted regions can touch.

3. **Keep original bodies** is selected by default to retain the original body after the conversion.
4. Choose the other options for the conversion, if applicable.
   a) For the **Facets to SubD** and **Region to SubD** conversion, specify an appropriate **Fit tolerance**.

      The **Fit Tolerance** depends on how dense the faceted mesh or model region is. Specifying a smaller tolerance value automatically adds points and adjusts the control cage to minimize the distance between the faceted mesh or original model and the subdivision surface.

   b) For the **SubD to Facets** conversion, choose from the additional options.

Option	Description
Max distance	Controls how far away the facet edges are from the subdivision edges. For the same value of **Max distance** and **Aspect ratio**, decreasing the **Max angle** will result in more facets along curved subdivision edges.
Max angle	Controls how far away the facet edges are from the subdivision edges. For the same value of **Max angle** and **Aspect ratio**, decreasing the **Max distance** will result in more facets along a curved edge, bringing the facet edges closer to the subdivision surface edge.
Aspect ratio	When selected, facets will be created with an aspect ratio close to the value specified. Set a value from 1 to 10 to prevent distorted triangles on the faceted body.
Max edge length	When selected, facets will be created with edge lengths less than or equal to the value specified.

5. Click **Complete** to convert the body.

7.3. Converting Topology Optimization Results to Hybrid Models

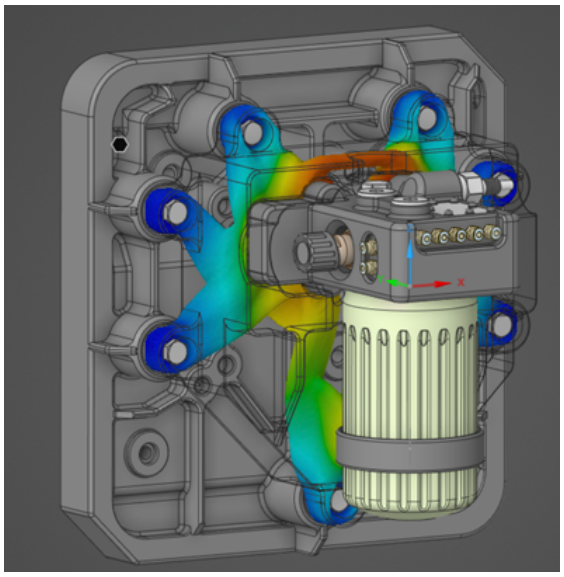
You can export results from a topology optimization as a hybrid model. The hybrid model contains both subdivision surfaces for regions of the model where you want to allow for flexible and freeform editing and traditional surfaces such as planes or cylinders for regions requiring precise topology.

1. Set up the **Topology Optimization** analysis to compute an optimal structural design for a selected region of your model with specific design goals and constraints.
2. Once you have a faceted geometry result you want to save, click **Add optimized body to model** in the **Results Arc** to save it to the model. The new faceted geometry is now listed in the Model tree.
3. Right-click the faceted body in the Model tree and select **Convert to SubD hybrid**.

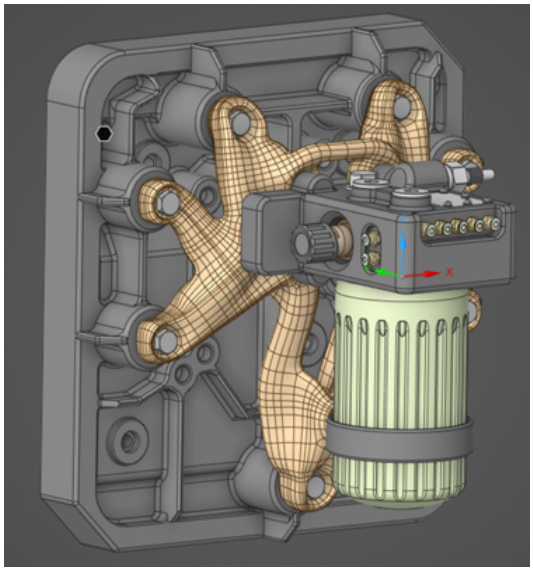
   You can find a course about SubD modeling on the Ansys Innovation Space.
4. (Optional) Once you have created the hybrid model of the resulting optimized shape, you can right-click the hybrid model in the Model tree select the **Create Validation Simulation** option to create a validation simulation that includes all of the physics conditions from the topology optimization simulation applied to the hybrid model.

The examples show the initial optimized body which is converted to a hybrid body. You can also see the cylindrical faces preserved where the supports were applied.

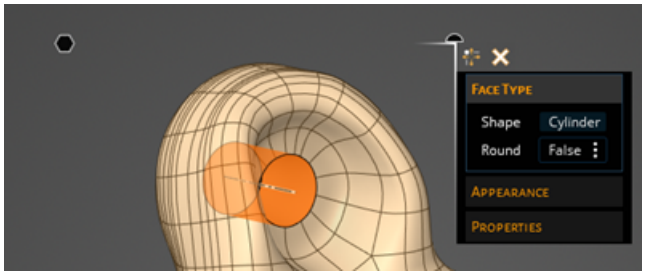
Initial optimized body



Converted hybrid body



Preserved cylindrical faces for supports



You can also use the **Auto Skin** tool to create the SubD hybrid model from faceted models that are relatively smooth and error free.

This approach can be used for models with topology optimization results from the Mechanicalapplication, which can be brought in and converted to a SubD hybrid model in the Discovery application. These models usually have groups/facet regions set on import.

The facet regions are converted to a combination of traditional and SubD faces. The SubD hybrid model retains the traditional analytic faces such as planes or cylinders for regions that require precise topology. The remaining facets are converted to SubD faces which allow for flexible editing.

See Creating the SubD Hybrid Model on page 486 for more information.

7.4. Using the Warp Tool

Use the **Warp** tool to create the warped body for a CAD part. The tool requires either two or three bodies and a tolerance value to create the warped body.

1. Select the CAD part for which the warp body is to be created.
2. Select the initial faceted body which represents the CAD part. If this faceted body has a displacement field on it then those are the only bodies which are needed for the warp operation.
3. (Optional) If the selected faceted body does not have a displacement field, select another faceted body representing the deformed triangle surface mesh before the **Complete** tool guide is activated.

   The deformed faceted body must have the same number and ordering of vertices as the first selected faceted body since it is used to calculate the displacement field.
4. (Optional) Choose to exclude specific faces from the selection, if required.
5. Specify the **Fit tolerance** to determine how closely the generated body conforms to the deformed faceted body.

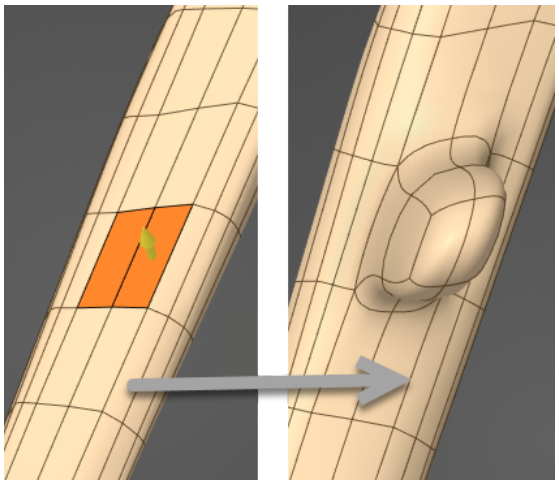
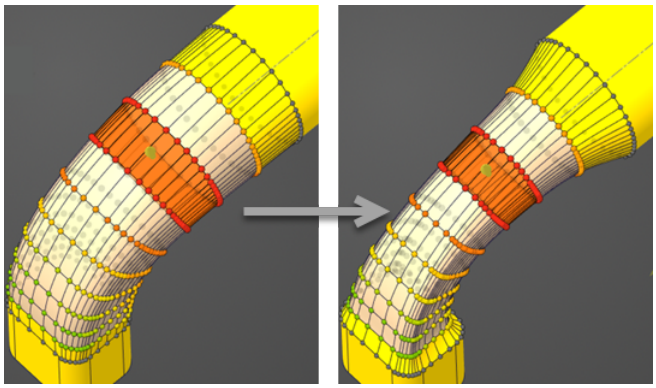
   The fit tolerance controls how closely the generated warp body conforms to the deformed faceted body. A tighter tolerance generates a heavier model (with more control points), while a bigger tolerance generates a looser but smoother fit.
6. Use **Show preview** to see the preview for the warp body.

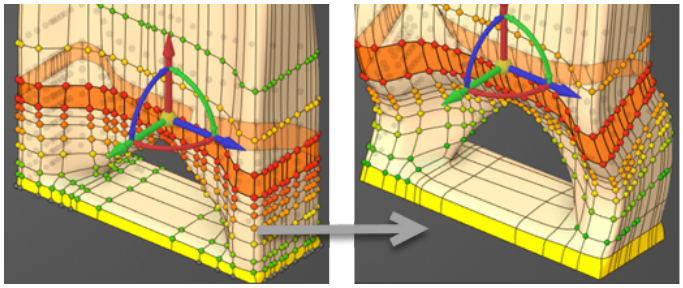
   Adjust the **Preview opacity** and use the **Displacement factor** slider to see an animated preview of the displacement.
7. Click **Complete** to create the warp body.

7.5. Editing Subdivision Surfaces

Use the tools in the **Edit** group to edit the Subdivision surfaces.

Select	With the **Select** tool active, click to select objects (vertices, edges, faces, bodies) from the display. Use the options in the **Select** ribbon dropdown to choose from the optional selection modes, if required.

Pull	Use the **Pull** tool to offset, extrude, revolve, sweep, and draft faces. You can use also it to extrude, copy, or pivot edges or move vertices of the selected faces along their normals. The action of the **Pull** tool depends on which faces and edges you select to work with, and which faces, planes, or edges you select to drive the change. Additional options are available for subdivision surfaces. • **Extrude** *Extrude the selected subdivision body faces or edges.*  • **Proportional** *Pull the selected faces along with the surrounding faces in proportion based on the **Radius** specified.* 

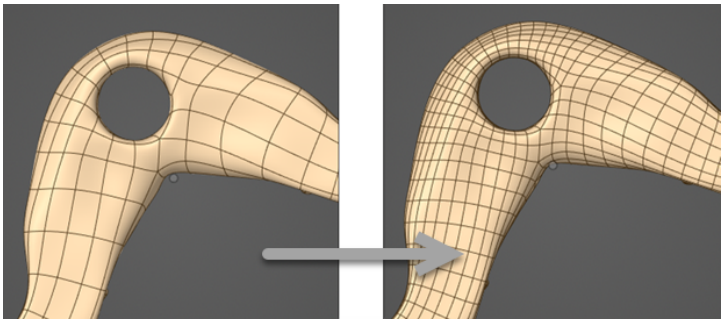
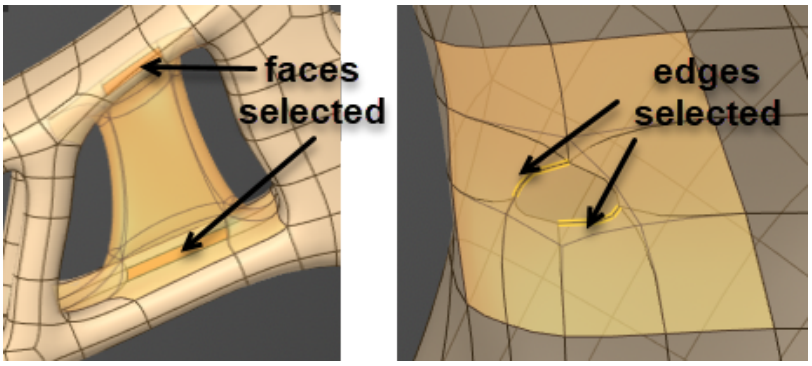
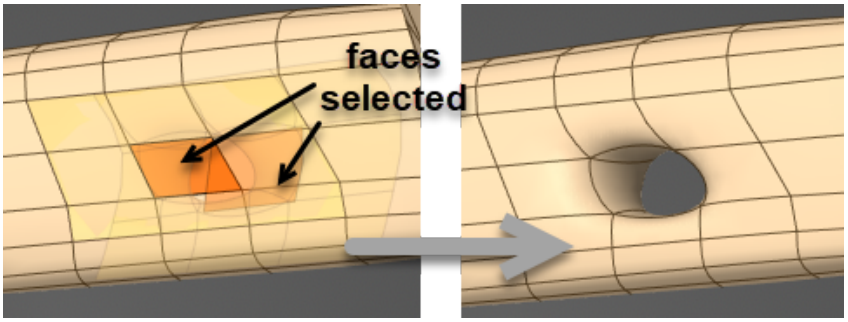
Move	Use the **Move** tool to move bodies or select faces, edges, or vertices to translate and/or rotate. The **Move** tool includes an additional option for subdivision surfaces. Select **Proportional** to move the selected faces along with the surrounding faces in proportion based on the **Radius** specified. 
Fill	Use the **Fill** tool to fill the selected region or edges with neighboring or simplified geometry.
Blend	Use the **Blend** tool to create a blend between selected faces, surfaces, edges, or curves.
Shell	Use the **Shell** tool to create a shell by removing a face.
Replace	Use the **Replace** tool to replace a face or curve with another face or curve, or simplify a face.
Mirror	Use the **Mirror** tool to create a duplicate of the selected object using a plane or face as the plane of symmetry. You can mirror curves, points, axes, faces, planes, and solids. Editing an object on one side of the mirror will apply that same edit to the corresponding object on the other side of the mirror. Select **Reverse mirror** to reverse the mirror direction, allowing you to control which portion of the model is mirrored.

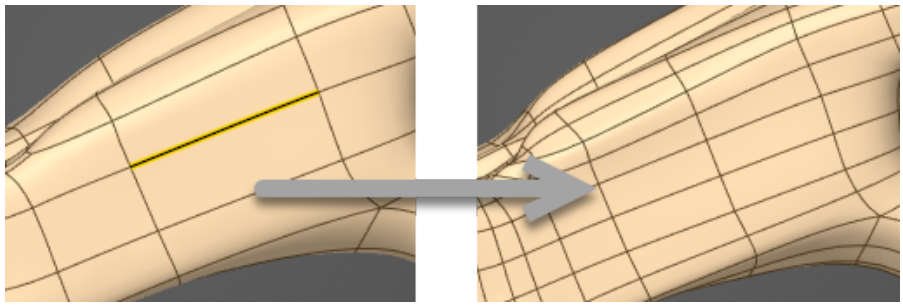
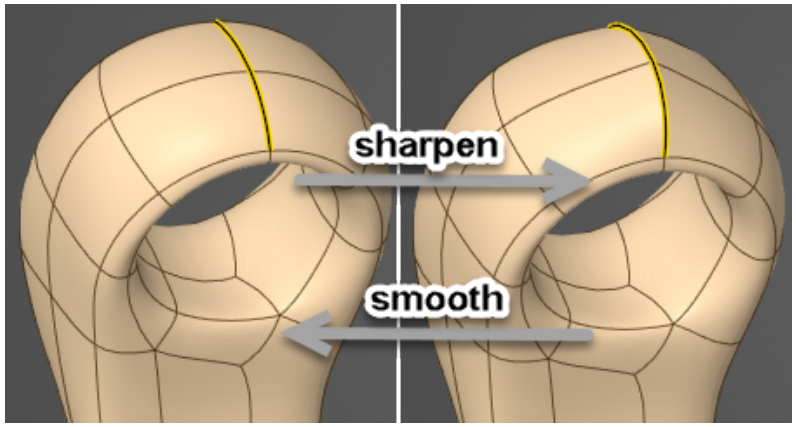
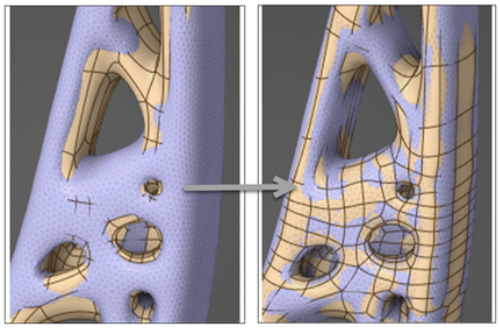
In addition to the tools in the **Edit** group, you can use the tools in the **Measure** tab (unless otherwise indicated) for measuring and analyzing the quality of the design.

> **Note:** The accuracy of the mass properties calculation for hybrid bodies is based on displayed facets, approximations can occur based on faceting.

7.6. Modifying Subdivision Surfaces

Use the tools in the **Modify** group to further modify the Subdivision surfaces.

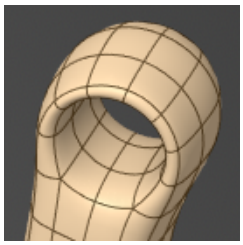
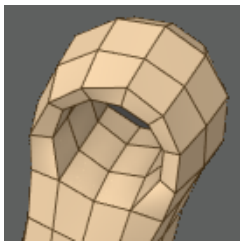
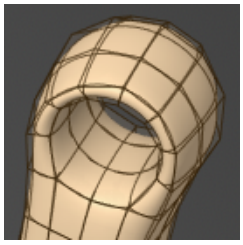
Subdivide	Use the **Subdivide** tool to add more faces to a selected body without changing its shape. Specify the **Cut count** (1-3) or use the slider to set the value indicating the number of subdivisions. For a cut count *n*, every edge is subdivided into $e = 2^n$ edges. Every four-sided face is subdivided into e^2 faces. For example, for a cut count of 1, every edge will be subdivided into $2^1 = 2$ edges. Four-sided faces will be subdivided into $2^2 = 4$ faces. 
Bridge	Use the **Bridge** tool to connect two selected regions. You can select faces or edges. When selecting edges, select a pair of edges to connect.  You can also create holes using the **Bridge** tool if the connection between the faces is already filled with material. Ensure there is enough topology around to allow the hole. You may need to use the **Subdivide** tool first. 

Split	Use the **Split** tool to split the selected edge and create four-sided patches (in the normal direction to the edge). 
Crease	Use the **Crease** tool to create a crease around selected edges or vertices. Specify an appropriate value for the **Crease factor**. Use the **Sharpen** option to make edges sharper. Use the **Smooth** option to smooth sharp edges. 
Snap	Use the **Snap** tool to snap the subdivided surface to the faceted body. This tool is useful if you convert from a faceted body to a subdivision surface and it deviates from the original body. Select the subdivision surface body and then select the faceted body, plane, or secondary selection to snap to. 
Smooth	Use the **Smooth** tool to smooth the subdivision surface body to the faceted body selected. Select the subdivision surface body and then select the faceted body to smooth the subdivision surface.

7.7. Viewing Subdivision Surfaces

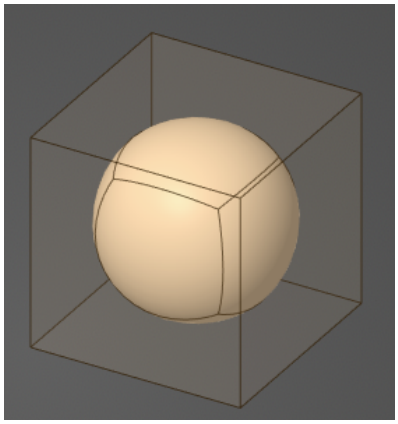
Each shape is defined over a polygonal control cage that controls its shape. The control cage is repeatedly subdivided to produce the limit surface. Control cages contain vertices, edges and faces which can be edited or modified to develop a unique shape.

- Use the View group options to view the **Limit Surface**, the **Control Cage**, or both.

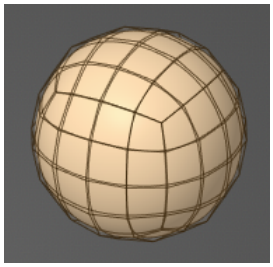
Limit Surface	
Control Cage	
Both	

The control cage completely defines the limit surface. Manipulating the control cage has a predictable effect on the limit surface.

In this example, even though the control cage looks like a cube, the limit surface is curved, almost spherical.



As you subdivide further, you can see that the control cage is closer to the shape of the limit surface. The limit surface is mostly smooth, except around extraordinary vertices, which have more or less than 4 faces.



> **Note:** The SubD control cage cannot be viewed for SubD hybrid bodies.

- Use the **Show** dropdown to customize the model appearance while using the subdivision surface tools.
  - **Proportional radius** shows the affected region when using the **Proportional** option in the **Pull** or **Move** tool.
  - **Hybrid body colors** shows the hybrid bodies by color.

8: Facets

The Discovery application has tools that allow you to prepare models for Additive Manufacturing, that is, 3D Printing.

3D Printers use STL files to build up a model in layers. The **Facets** tab contains tools for working with facets and preparing them for export to a 3D printer. The ribbon is organized in groups of tools that, from left to right, step through the process of creating a Faceted body for the printer.

Edge Display has toggles for displaying:

- **Internal** facet edges are shared by two connected facets
- **Open** facet edges belong to one facet
- **Over-connected** facet edges are connected to three or more facets and cannot be resolved by 3D printers

This section contains the following topics:

8.1. Selecting Facets

You can select facets and then modify the selection using the following tools in the **Select** group in the **Facets** tab. The dropdown menu on the **Select** option also allows you to select facets **Using Box**, **Using Lasso**, **Using Polygon**, or by **Using Paint**. You can also choose to **Extend Facets Selection** or **Select Facets by Geometry**, which enables the transfer of selections from one body to another.

Expand	Use the **Expand** tool to add an additional row of facets around the current selection. Clicking this tool multiple times allows you to expand the selection by one row each click.
Shrink	Use the **Shrink** tool to reduce the current selection by one row of facets. Clicking this tool multiple times allows you to shrink the selection by one row each click.
Fill	Use the **Fill** tool to select all remaining facets that are bounded by the current selection.

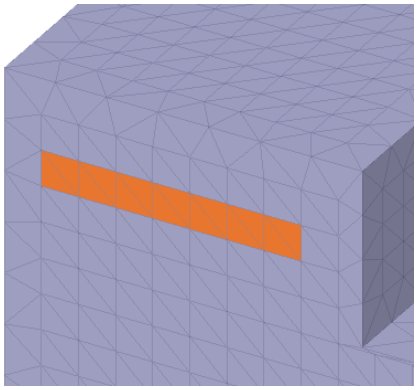
This section contains the following topics:

8.1.1. Using the Expand Tool

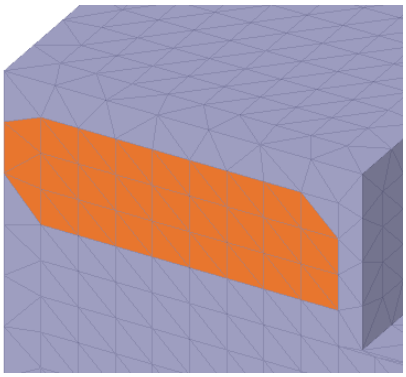
The **Expand** tool lets you select additional rows of facets around the current selection.

To expand facets:

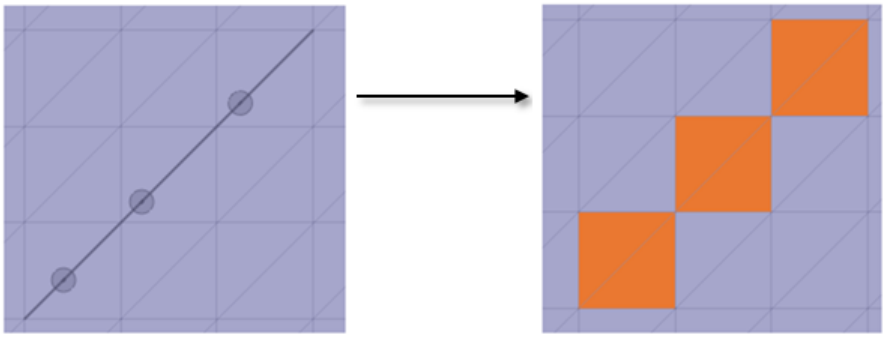
1. Select facets.



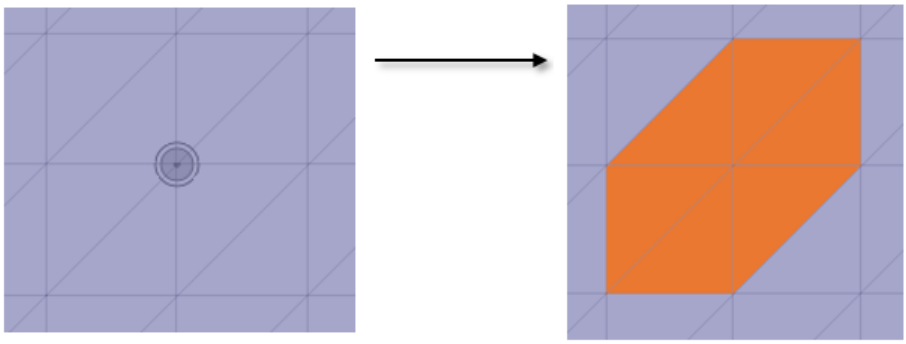
2. Click **Expand** to select a row of facets around the current selection.



Using the same steps above, facet edges can be expanded to adjacent triangles.



Facet vertices can also be expanded to adjacent triangles.

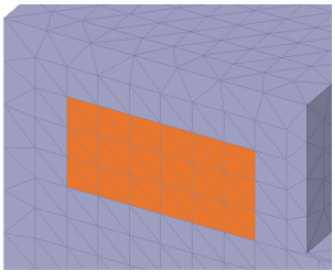


8.1.2. Using the Shrink Tool

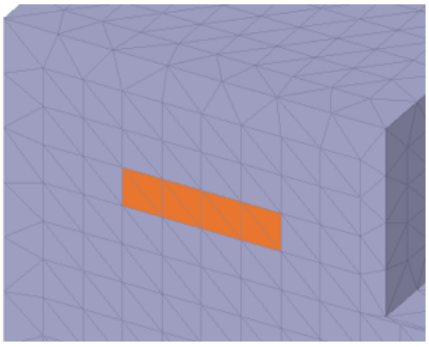
The **Shrink** tool lets you reduce the current selection by one row of facets.

To shrink facets:

1. Select facets.



2. Click **Shrink** to reduce the current selection by one row of facets.

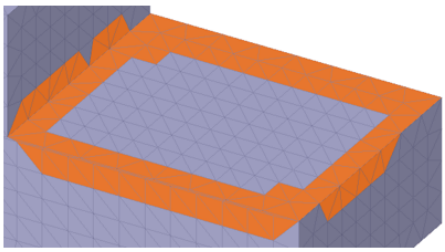


8.1.3. Using the Fill Tool

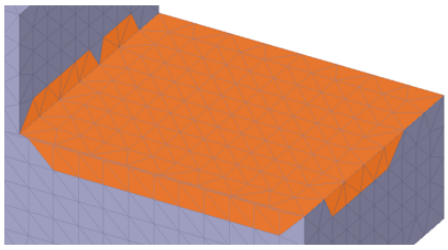
The **Fill** tool selects all remaining facets that are bounded by the current selection.

To fill facets:

1. Select facets.



2. Click **Fill** to select all remaining facets.

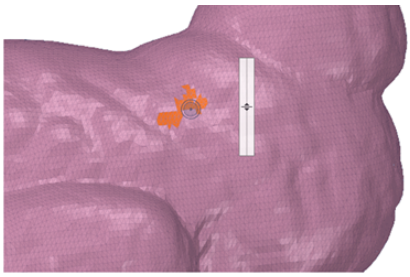


> **Note:** If the **Fill** adds less than five facets, a warning is issued.

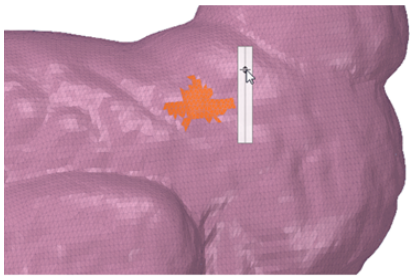
8.1.4. Using a Slider to Adjust Selection

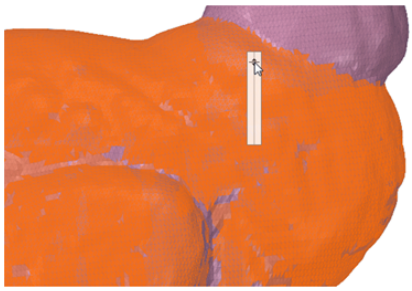
After double-clicking a facet, a slider can be used to adjust the initial selection.

1. Double-click facets.



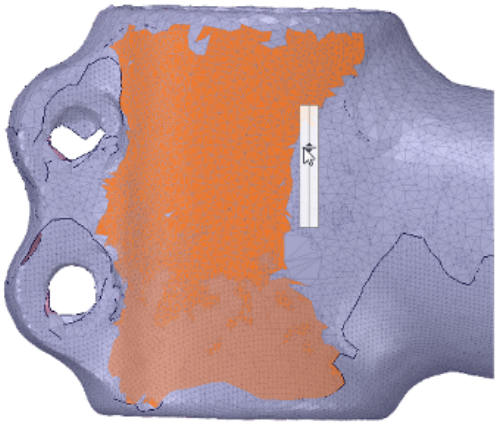
2. Move the slider up and down to adjust the initial selection.



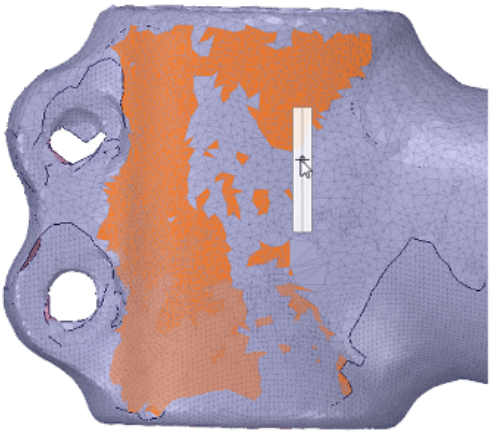


The selected facets can be added to a group. The slider will be dismissed after triple-clicking the facet selection.

To deselect facets, select facets with Box-Select or the Double-Click Slider.



While holding Ctrl, use the Double-Click Slider to unselect facets.

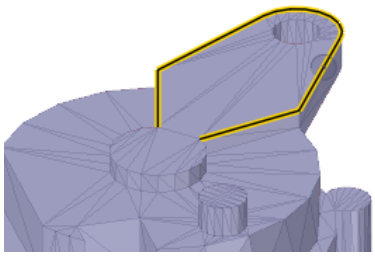


8.1.5. Selecting Facet Edges

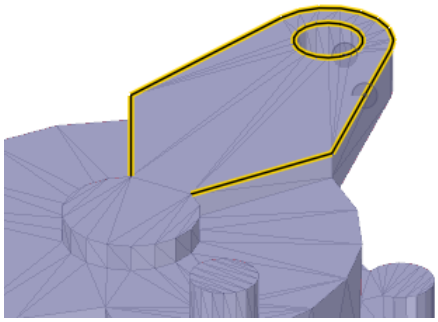
In addition to selecting facets, you can select facet edges and edge loops.

Use **Shift+Double-click** on sharp (>60°) internal edges that form an isoline (edges that are nearly continuously tangent to each other as the line propagates).

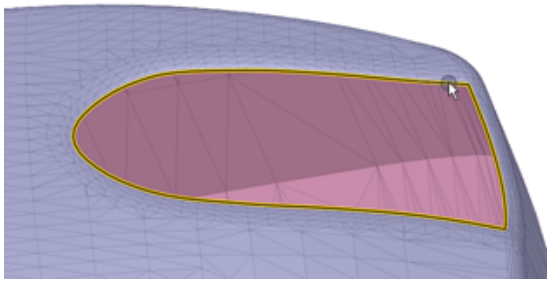
The selection will be the best polyline found and not necessarily a complete loop. As you can see in the image below, the selection stops at the cylinder because adjacent edges are not within the tolerance (<10°) to be considered tangent.



Use **Ctrl+Shift+Double-click** on sharp edges to select multiple mesh edge isolines.



Double-click the Boundary mesh edges to highlight and select the boundary edges



8.2. Faceted Body Cleanup

3D Printing requires that the faceted body be watertight and free of self-intersecting facets, or other defects. The **Cleanup** tools identify and fix facet defects.

The **Cleanup** group in the **Facets** tab contains the following tools for finding and fixing problems in faceted bodies.

Check Facets	Use the **Check Facets** tool to list problems with the faceted body that will adversely affect the 3D Printing process.
Auto Fix	Use the **Auto Fix** tool to automatically search for, and fix any facet defects.
Shrinkwrap	Use the **Shrinkwrap** tool to create a faceted wrapper that closes gaps, around the model.
Holes	Use the **Holes** tool to find and fix holes in faceted bodies.
Intersections	Use the **Intersections** tool to highlight areas of self-intersection.
Fix Sharps	Use the **Fix Sharps** tool to find and fix sharp spikes that need to be flattened.
Over-Connected	Use the **Over-Connected** tool to find and fix facets connected to too many other facets.

> **Note:** Additive Manufacturing technology is capable of producing interior voids (enclosed, hollow spaces) that would not be possible with traditional manufacturing methods. Depending on the additive process used, the void may contain unused resin, powder, or support materials. If voids need to be in the finished part, it is good practice to build it in halves or add access holes to remove the unused materials.

This section contains the following topics:

8.2.1. Automatically Cleaning up Faceted Bodies

The **Check Facets** and the **Auto Fix** tools are quick ways to find and fix problems in faceted bodies.

This section contains the following topics:

8.2.1.1. Using the Check Facets Tool

Use the **Check Facets** tool to list problems with the faceted body that will adversely affect the 3D Printing process.

Problems detected include the following:

- Multiple pieces (facet regions or shells)
- Non-watertight facets
- Self intersections
- Over-connected (non-manifold) edges

To check facets:

1. Select a Faceted body in the model tree.
2. Click the **Check Facets** tool.
   A dialog appears listing any issues found.
3. Click **Zoom to fit** in the dialog to zoom in on selected problems in the list.

The **Check Facets** tool can also be used in an Action-Object workflow by entering the tool and then selecting the faceted body.

8.2.1.2. Auto Fix Tool

Use the **Auto Fix** tool to automatically search for, and fix any facet defects. Geometry operations performed by the tool do not remove Named Selections.

The **Auto Fix** tool performs the following sequence:

- Fix self-intersections
- Close holes using the following methods:
  - If there is surrounding curvature, the hole is closed using the Patch method.
  - If the surrounding edges are approximately planar, the hole is closed using the Cap method.
- Recheck for self-intersections and fix them
- Remove void shells
- Fix inverted normals
- Fix non-manifold edges and vertices

This section contains the following topics:

8.2.1.2.1. Using the Auto Fix Tool

To use **Auto Fix** on a faceted body:

1. Select a faceted body in the display window.
2. Select the types of faceting problems to fix. Choose from:
   - Self-intersection

- Holes
- Non-manifold
- Fold overs

3. Click **Complete** to fix the faceting problems.

   The problem areas are fixed.

8.2.1.2.2. Auto Fix Tool Options

Select the types of faceting problems to be fixed.

Self-intersection	Fix self-intersecting facets in the faceted body.
Holes	Fix holes in the faceted body.
Non-manifold	Fix non-manifold facets in the faceted body.
Fold overs	Fix folded over facets in the faceted body.

8.2.2. Shrinkwrap Tool

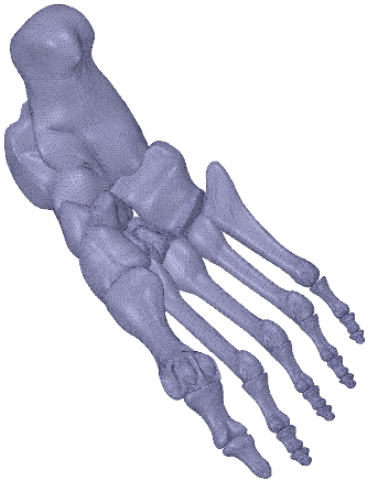
The **Shrinkwrap** tool creates a faceted wrapper around the selected solid or faceted bodies. You can wrap a single body or several bodies. For example, if you have an assembly with multiple parts fastened together but there are small gaps between the component parts, the Shrinkwrap tool allows you to create a single, faceted model that closes such gaps.

This section contains the following topics:

8.2.2.1. Creating a Shrinkwrap

1. Select the **Shrinkwrap** tool and select a solid or faceted body. The initial size is chosen based on the size of the selected body.

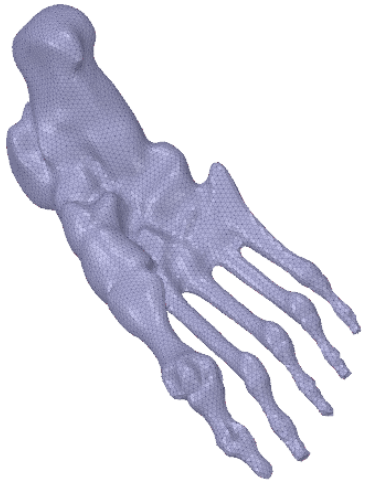
   The foot model used in this example, is a faceted model from scanned data. Notice that there are many tiny facet regions and small gaps between the bones of the foot.



2. Change the Shrinkwrap Options as desired.

3. (Optional) Select faces where the shrinkwrap will fit tightly using the **Select Tight-Fit Faces or Facets** tool guide.
4. Click **Complete** to create the Shrinkwrap.

   The completed shrinkwrap is shown below. Notice how the tiny facet regions have been wrapped into a single faceted body.



8.2.2.2. Shrinkwrap Tool Guides and Options

Shrinkwrap Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Bodies	The **Select Bodies** tool guide is active by default. It allows you to select solid or faceted bodies to shrinkwrap.
Select Tight-Fit Faces or Facets	The **Select Tight-Fit Faces or Facets** tool guide allows you to select faces where the shrinkwrap will fit tightly.
Complete	The **Complete** tool guide completes the shrinkwrap.

Shrinkwrap Tool Options

The following options are available for shrinkwrapping:

Size	Enter a value for the average facet size that will be used for shrinkwrapping. Distances under this size may be walked over, while distances above this size will be considered acceptable and will not be closed.
Keep original bodies	When enabled, the original faceted bodies will not be removed after the Shrinkwrap is created.
Preserve features	When enabled, this keeps hard edges and doesn't smooth them. Set the Angle threshold which decides which edges to keep.
Secondary size	Specify a facet size (smaller than the Size) to apply to selected facets. This allows you to refine the shrinkwrap locally.

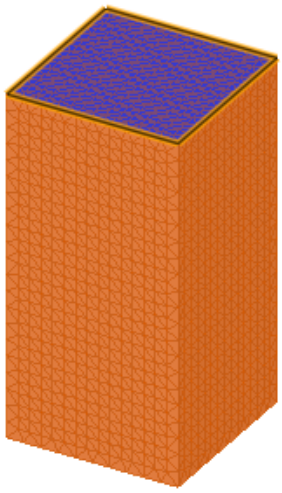
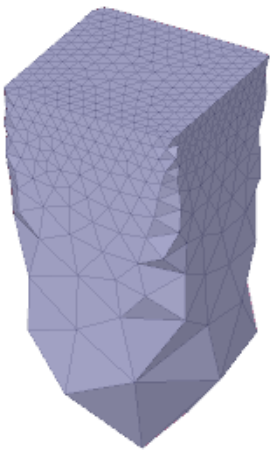
Max size	Enable this option to make the triangles conform more to the shape with different sized triangles based on curvature. Specify the maximum triangle size for curvature dependent faceting.
Remove interior	When enabled, wraps the selected object to cover all openings and remove interior voids. For models with large openings or holes, you can use this option to use a small enough facet size based on the geometry to capture significant details but also cover the larger openings when the shrinkwrap is created.
Fit tolerance	Specify the fit tolerance to be used for shrinkwrapping to cover openings and remove interior voids. Specifying a value of zero will automatically compute facet sizes based on geometry.

8.2.2.3. Shrinkwrap Examples

Using the Select Tight-Fit Faces or Facets Tool Guide

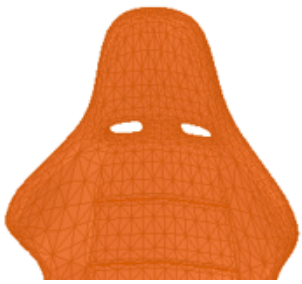
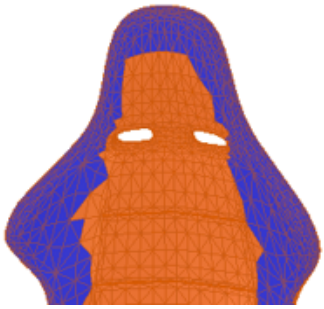
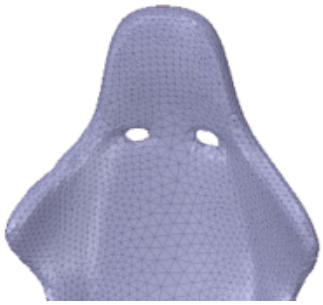
Use the **Select Tight-Fit Faces or Facets** tool guide to control shrinkwrap facets in selected areas.

The following simple example shows how the tool guide works.

*Click the **Select Tight-Fit Faces or Facets** tool guide and double-click the top face. This face will be tightly fit in the shrinkwrap.* 	*Notice that the facets fit tightly on the selected face. It is important to note that the controlling effect falls off as a function of distance from the selected area.* 

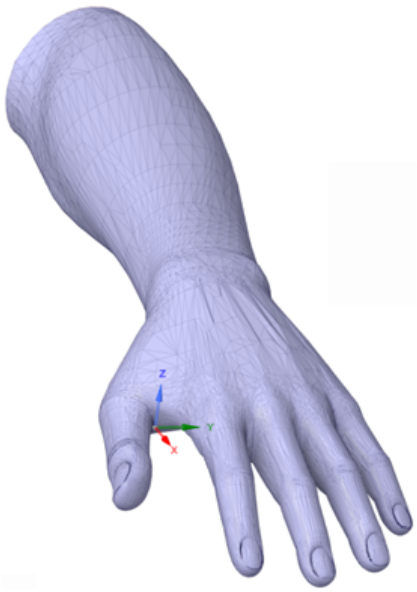
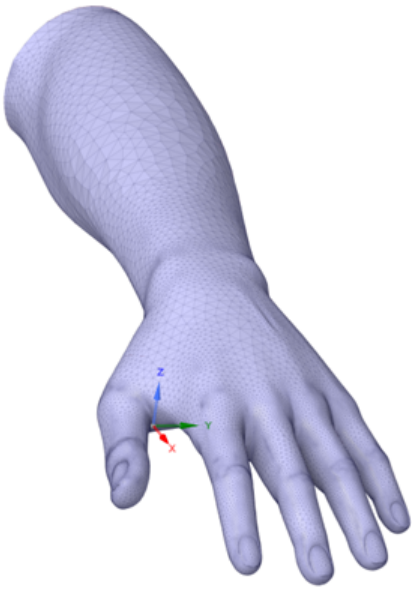
Using a Secondary Facet Size

Secondary size is more useful for refining the shrinkwrap to preserve the initial shape. An example is shown below.

This faceted model has a distinct profile shape.	
Using the default size and no Secondary size specified, Shrinkwrap produces a very rough approximation of the profile.	
1. Enable the **Secondary size** option and specify a smaller value for Secondary size. 2. Click the **Select Tight-Fit Faces or Facets** tool guide and select facets around the profile to apply the secondary size.	
Click **Complete** and see that the secondary size follows the profile.	

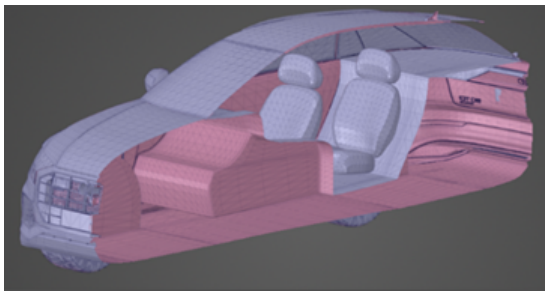
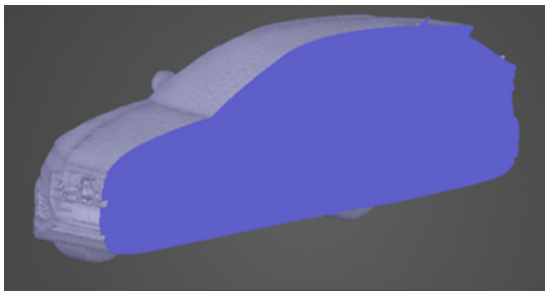
Using Shrinkwrap for Curvature Dependent Faceting

Use the Shrinkwrap options to make the triangles conform more to the shape with different sized triangles based on curvature.

Original faceting.	*Faceting after shrinkwrap using **Max size** and **Curvature angle**. Larger triangles can be seen on the forearm and smaller triangles by the nails where curvature is greater.*
	

Using the Remove Interior Option

Use the **Remove interior** option to wrap the selected object to cover all openings and remove interior voids.

This example has both small details that should be captured during shrinkwrapping as well as large openings.	*Faceting after shrinkwrap using **Remove interior** with a **Fit tolerance** of zero. Specifying a value of zero will automatically compute facet sizes based on geometry.*
	

8.2.3. Holes Tool

You can find and fix holes using the **Holes** tool. It works like other Find-and-Fix tools in the **Repair** tab.

This section contains the following topics:

8.2.3.1. Finding and Fixing Holes in Faceted Bodies

1. Select the **Holes** tool to find and highlight holes.

The tool highlights holes that can be filled.

2. Set any of the **Options** described below as needed.
3. Use the **Select Problem** tool guide to holes to fill.
4. Click **Complete** to fill all holes or the selected holes.

8.2.3.2. Holes Tool Guides and Options

Holes Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Problem	The **Select Problem** tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.
Select Geometry	The **Select Geometry** tool guide allows you to select geometry not highlighted by the tool
Exclude Problem	The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.
Select Guide Curves	The **Select Guide Curves** tool guide allows you to select curves to influence the shape of the patch.
Exclude Regions	The **Exclude Regions** allows you to exclude regions in multi-region patches.
Complete	The **Complete** tool guide fixes all remaining problem areas.

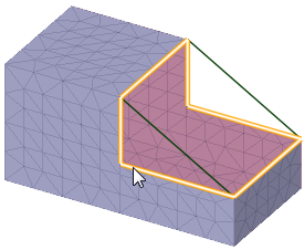
Holes Tool Options

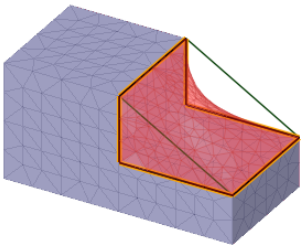
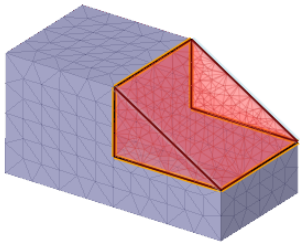
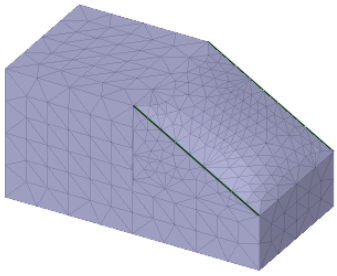
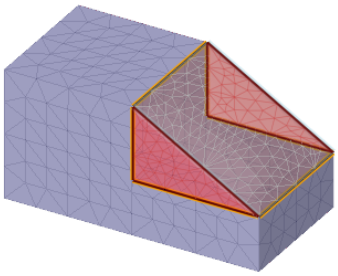
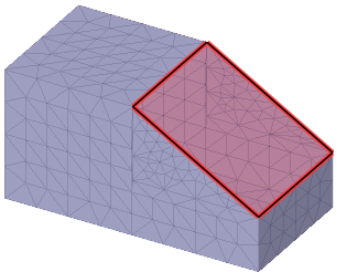
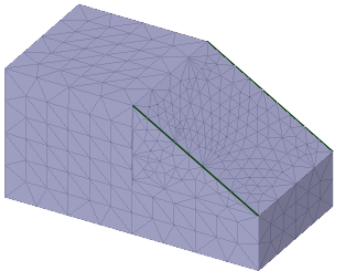
The following options are available to control how holes are filled:

Patch	Attempts to match the contour of the surrounding facets.
Cap	Creates a simple cap over the opening.
Together	Considers multiple openings together.
Separately	Considers multiple openings separately.

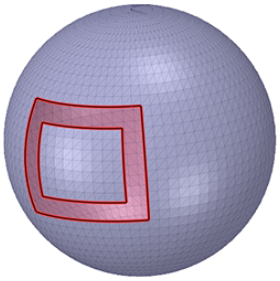
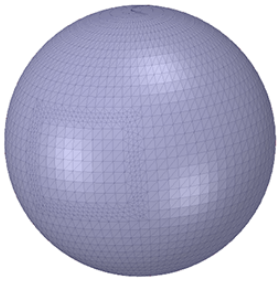
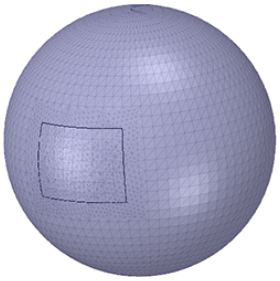
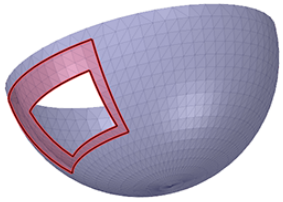
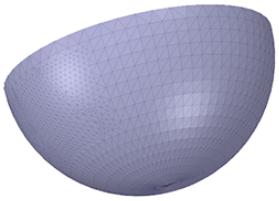
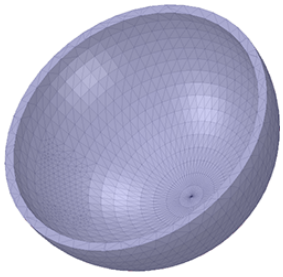
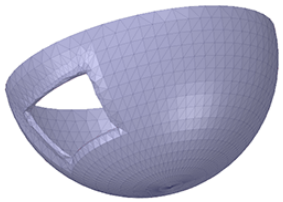
8.2.3.3. Fixing Holes Examples

The following example shows the use of a few of the tool guides and the Cap option.

Click the **Select Geometry** tool guide and select the contour.	

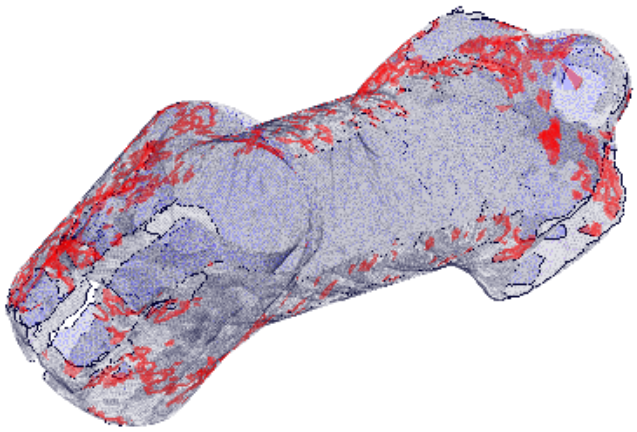
The preview shows that the default patch will not produce an angled face.	
Click the **Select Guide Curves** tool guide and select the curves, or Alt-select them. notice that the preview now goes through the curves. With the Select Guide Curves tool guide active, it is considered sticky (as if the Ctrl key were being held down) to allow multiple selection and selection toggling.	
Clicking **Complete** shows that the face is not flat.	
To make it flat, Undo the fill and use the **Exclude Regions** tool guide to exclude the angled face. Notice that the excluded region is shown in a dim gray to indicate it will not be included.	
In the Options panel, set the Cap option. Then click the **Select Problem** tool guide and the select the remaining loop.	
Click **Complete**.	

The following examples show the use of the **Together** and **Separately** options.

Two loops on the outside of a sphere is a case that should be filled together, as they are both intended to be occupying the same physical space (that is, the outside of the sphere). 	*The **Together** option ensures that the patch completes the shape.* 
*If the **Separately** option is used, the gap is closed (one at a time) so a loop and a separate surface remains.* 	*This shelled bowl is an example where the **Separately** option is more likely to be used.* 
*The **Separately** option ensures that each patch completes the surrounding shape both outside and inside.*  	*The **Together** option considers the surrounding shape of both holes at the same time and transitions between the two holes.* 

8.2.4. Intersections Tool

Self-intersecting facets can cause problems for 3D printing. Use the **Intersections** tool to highlight areas of self-intersection.



This section contains the following topics:

8.2.4.1. Checking for Intersections

1. Select the **Intersections** tool.
2. Select a Faceted body in the Graphics window.
   Areas of self-intersection are highlighted. The model is also made transparent if any intersections are found.
3. Click **Complete** to fix the intersections.

8.2.4.2. Intersections Tool Guides

Complete	The **Complete** tool guide fixes any intersections found.

8.2.5. Fix Sharps Tool

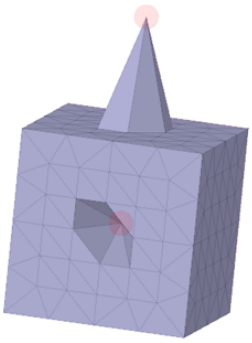
Faceted bodies may contain sharp spikes that need to be flattened. You can use the **Fix Sharps** tool to find and fix any sharps in the mesh. It works like other Find-and-Fix tools in the Repair tab.

This section contains the following topics:

8.2.5.1. Finding and Fixing Sharps

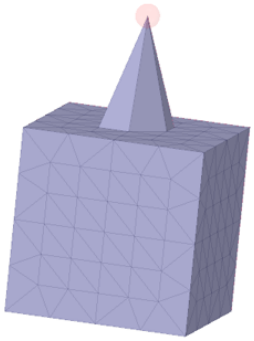
1. Select the **Sharps** tool .
2. Based on the options settings, if any sharps are found, they are highlighted.

   Convex and concave sharps are found.



3. (Optional) Change the option settings as needed.
4. Use the **Select Problem** tool guide to select and fix individual sharps.

   The concave sharp is flattened.



5. Use the **Exclude Problem** tool guide to exclude problem areas.
6. Use the **Select Geometry** tool guide to select geometry not found as a problem area.
7. Click **Complete** to fix all remaining highlighted sharps.

8.2.5.2. Fix Sharps Tool Guides and Options

Fix Sharps Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Problem	The **Select Problem** tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.
Select Geometry	The **Select Geometry** tool guide allows you to select geometry that was not found as a problem area.
Exclude Problem	The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.
Complete	The **Complete** tool guide fixes all remaining problem areas.

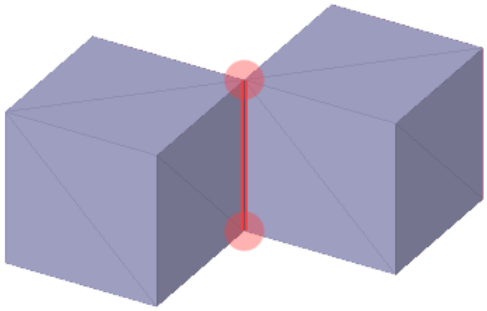
Fix Sharps Tool Options

The following options are available to control how sharps are fixed:

Sharp edges	This applies to the angle between normals of facets that share an edge. Any edges with an edge angle greater than the specified angle will be found.
Sharp vertices	This applies to the angle between normals of any two facets that share a vertex. Any vertices with an angle greater than the specified angle will be found.
Convex	Convex sharps are found. These are sharps that point outward.
Concave	Concave sharps are found. These are sharps that point inward.

8.2.6. Over-Connected Tool

Over-connected facets cannot be 3D printed. There can be over-connected edges and over-connected vertices. Over-connected edges are facet edges that are shared by more than two facets. In the image below, two bodies share a single edge which means the shared edge belongs to four facets. In this case, the vertices at the end of the edge are also over-connected because there is no volume beneath them.

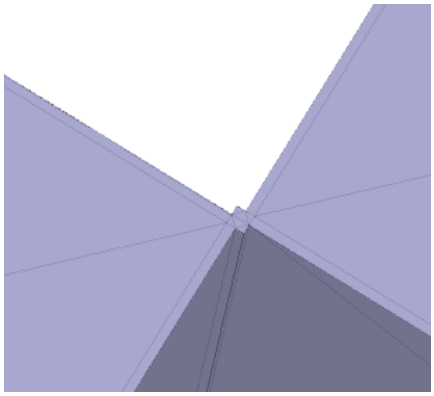


This section contains the following topics:

8.2.6.1. Finding and Fixing Over-Connected Facets

1. Select the **Over-connected** tool.
   The tool finds any over-connected facets and highlights them.
2. Select highlighted areas to fix using the **Select Problem** tool guide.
3. Exclude problem areas using the **Exclude Problem** tool guide.
4. Use the **Select Geometry** tool guide to select geometry that is not a problem.
5. Click **Complete** to fix all remaining areas.

When the **Over-connected** tool is used on the example shown above, it creates a small faceted volume around the over-connected edge. The image below shows the result when zoomed-in to the edge.



8.2.6.2. Over-Connected Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Problem	The **Select Problem** tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.
Select Geometry	The **Select Geometry** tool guide allows you to select geometry that was not found as a problem area.
Exclude Problem	The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.
Complete	The **Complete** tool guide fixes all remaining problem areas.

8.3. Organizing Faceted Bodies

The **Organize** group has the following tools for organizing faceted bodies and regions.

Convert	Use the **Convert** tool to convert solid bodies into facets bodies.
Separate	Use the **Separate** tool to separate each disconnected faceted region of a faceted body into individual faceted bodies. A typical use of this tool is to separate the inner and outer regions of a shelled faceted body.
Separate All	Use the **Separate All** tool to separate all of the faceted bodies in the design that have multiple disconnected regions into individual faceted bodies. This would be used for designs with multiple shelled faceted bodies.
Join	Use the **Join** tool to combine multiple faceted bodies into a single body and stitch them together when possible..

A single faceted body can have any number of regions. A common example is a shelled faceted body which has both an inner and an outer faceted region in one body. Another example can be created by saving a Discovery model, which contains several separate solid bodies, as an STL file. It gets imported into the Discovery application as a single faceted body but has several separate facet regions.

This section contains the following topics:

8.3.1. Convert Tool

Use the **Convert** tool to convert solid bodies into facets bodies.

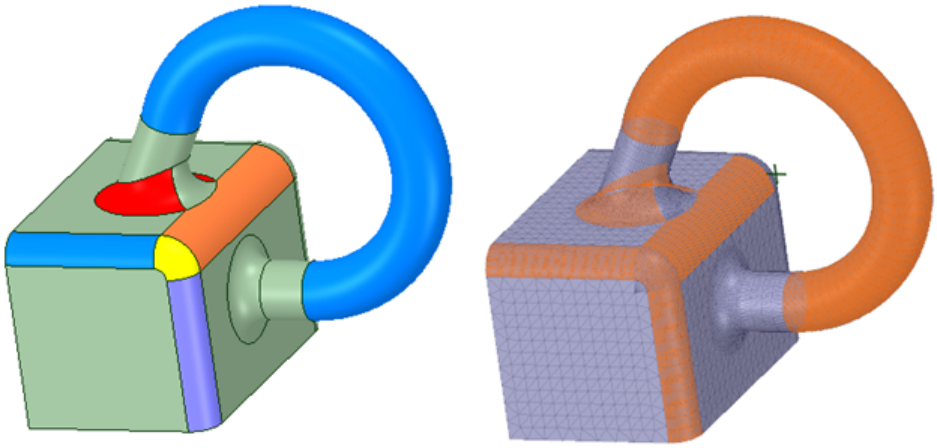
The **Convert** tool has a Progress Bar and a Stop button for meshes that take longer to generate.

This section contains the following topics:

8.3.1.1. Converting a Solid Body to a Faceted Body

1. Click the **Convert** tool.
2. Select a faceted body.
   The selected faceted body highlights in blue
3. Choose a **Cap option** (see images below).
4. Select a plane, or surface, to use as a cutter.
5. The faceted body is split into two separate faceted bodies.

When converting a B-Rep model to a faceted model, you can double-click a face to select the underlying topology. In this example, the left image shows the original model. The right image shows the faceted model with faces selected (shown in orange). The selected faces correspond to the colored faces in the original model.



8.3.1.2. Convert Tool Options

The following options are available for converting solid bodies to faceted bodies:

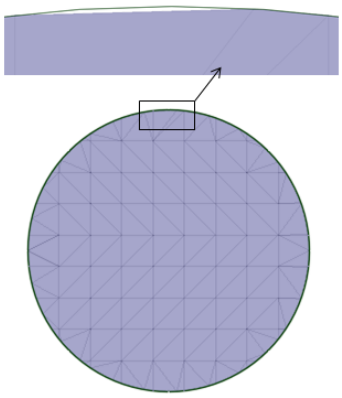
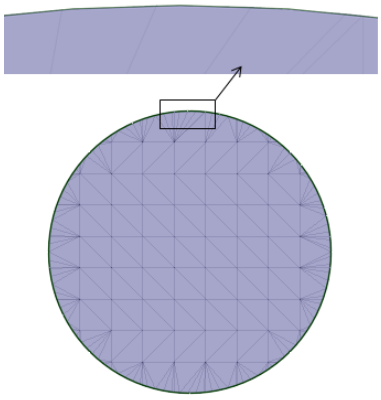
Keep original bodies	Retain the original solid bodies after conversion
Max distance	Controls how far facet edges are away from model edges.
Max angle	Controls how far facet edges are away from model edges.
Aspect ratio	Enter a number from 1 to 10. Facets will be created with aspect ratios near the specified value.
Max edge length	Generated facets will have edge lengths less than or equal to the value entered.

- Use the **Keep original bodies** option if you want to retain the original solid bodies. Converting a solid body to a faceted body consumes the original solid.

These are the same options (geometrically-speaking) as the ones that control STL *export*.

Regardless of the context in which these options are used, they control the facet creation required by facet modeling tools, whether it is for the explicit, user-directed **Convert**, or the behind-the-scenes conversion required by other facet modeling tools.

- **Max distance** controls how far facet edges are away from model edges. The images below show the effect of Max distance on the mesh of a cylinder. These examples keep the Aspect ratio fixed at 3 and the Max edge length set to 4 mm.

Max Distance = 0.75 mm *Max angle = 20-degrees*	*Max Distance = 0.01 mm* *Max angle = 20-degrees*
	

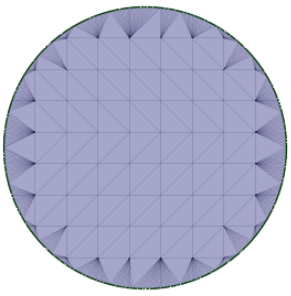
Notice how the settings change the facets along the edge of the circle. The interior is a flat plane, so 45-degree triangles are the most efficient facets.

The example on the left shows the default Max distance of 0.75 mm. This refers to the gap seen in the zoomed-in (top) image. Decreasing the Max distance to 0.01 mm results in more facets along the edge, which brings the facet edges closer to the cylinder edge, as seen in the example on the right.

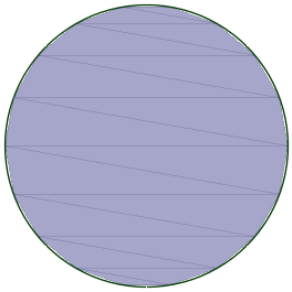
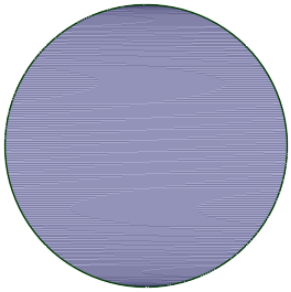
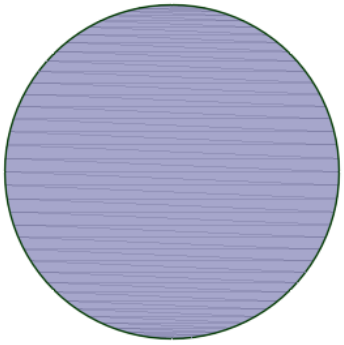
- **Max angle** also controls how far facet edges are away from model edges. Decreasing the Max angle will result in more facets along curved edges. This example shows the result of decreasing the Max angle from 20 degrees to 1 degree.

Max Distance = 0.75 mm

Max angle = 1-degree



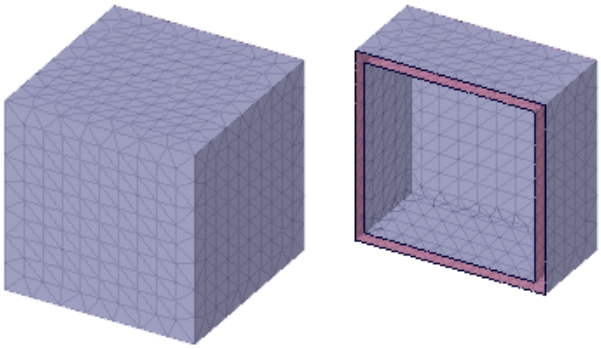
- **Aspect ratio** will prevent distorted triangles. Without a fixed Aspect ratio and without a Max edge length, you can get more distorted triangles as shown in the following images.

No set Aspect Ratio or Maximum Edge Length *Max Distance = 0.75 mm* *Max angle = 20-degrees* 	*No set Aspect Ratio or Maximum Edge Length* *Max Distance = 0.75 mm* *Max angle = 1-degree* 	

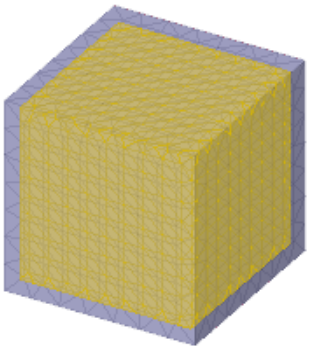
8.3.2. Using the Separate Tool

1. Select the **Separate** tool .
2. Use the **Select Shells** tool guide to select a shelled faceted body and separate it.
3. Use the **Select Meshes** tool guide to select a faceted body with disconnected regions and separate them.
4. Select a facet shell or facet region.
   The selection is then converted to a separate faceted body.

The image shows a shelled faceted body. The section view shows the interior and exterior regions.



After separation, there are two faceted bodies. The image shows the interior faceted body highlighted.



8.3.2.1. Separate Tool Guides and Options

Separate Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Shells	The **Select Shells** tool guide is active by default. Use it to select a shell and separate it from its faceted body.
Select Faceted Bodies	Use the **Select Faceted Bodies** tool guide to select a faceted body and separate it into pieces.

Separate Tool Options

The following options are available:

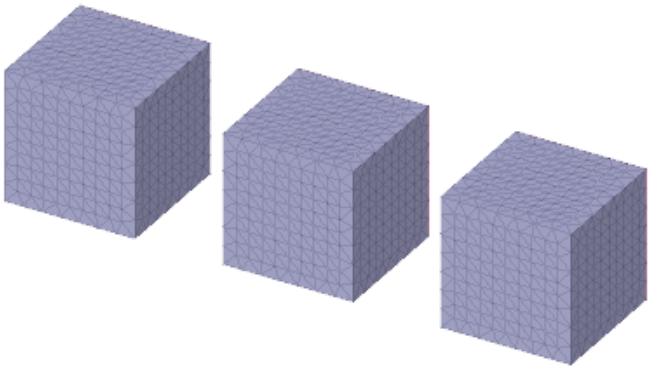
Over-connected	Include over-connected facet regions (regions joined at a single edge or point)
Connected	Include only normally connected facet regions

8.3.3. Using the Separate All Tool

Click the **Separate All** tool or right-click a faceted body in the model tree and click **Separate All**.

All facet regions, in all facets bodies, are converted to separate facets bodies.

The image below shows a simple example of a single faceted body with multiple disconnected regions.



8.3.4. Using the Join Tool

1. Click the **Join** tool.
2. Use the **Select Faceted Bodies** tool guide to join selected bodies one-by-one.
   a) Select a body to serve as the target. All subsequently selected bodies will be joined to the target.
   b) Select bodies to join to the target.
3. Use the **Select Target Faceted Body** tool guide to join all facets bodies to the selected target body.
   a) Select the target body.

b) All other faceted bodies are automatically joined to the target.

8.3.4.1. Join Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Faceted Bodies	The **Select Faceted Bodies** tool guide is active by default. Use it to select faceted bodies to join one-by-one. The first body selected is the target to which all other selected bodies are joined.
Select Target Faceted Bodies	Use the **Select Target Faceted Bodies** tool guide to select a target faceted body. All other faceted bodies are then automatically joined to the target body.

8.4. Editing Faceted Surfaces

Use the **Pull** or **Move** tool to edit the faceted bodies or surfaces.

- Use the **Pull** tool to offset, extrude, revolve, sweep, and draft faces. You can use also it to extrude, copy, or pivot edges or move vertices of the selected faces along their normals.

  The action of the **Pull** tool depends on which faces and edges you select to work with, and which faces, planes, or edges you select to drive the change.

- Use the **Move** tool to move bodies or select faces, edges, or vertices to translate and/or rotate.

These tools work slightly differently on faceted bodies or faces than they do on regular solids.

This section contains the following topics:

8.4.1. Using the Pull or Move Tool

The **Pull** and **Move** tools enable you to edit the faceted bodies or surfaces.

To pull or move facets:

1. Select the **Pull** or **Move** tool from the **Design** tab.
2. Select the facets to be moved. You can select a faceted body, a set of vertices (of the facet body) defining the region of interest, or a set of control vertices (a subset of the region of interest).
3. Specify the transformation for the selection.

   The transformation in this case is approximately like extrusion (without modifying existing connectivity or adding new vertices).

4. (Optional) Select **Proportional** and specify an appropriate value for **Radius**.

   This option enables deformation to be propagated proportionally to a region of the faceted mesh, rather than exclusively to the selection. In contrast to the default transformation (extrusion without modifying connectivity or adding new vertices) this takes the selected regions and the specified radius to generate the regions of interest on the mesh. The boundaries of the regions of interest are fixed vertices for this transformation (that is, the deformation will not propagate to or beyond these fixed vertices).

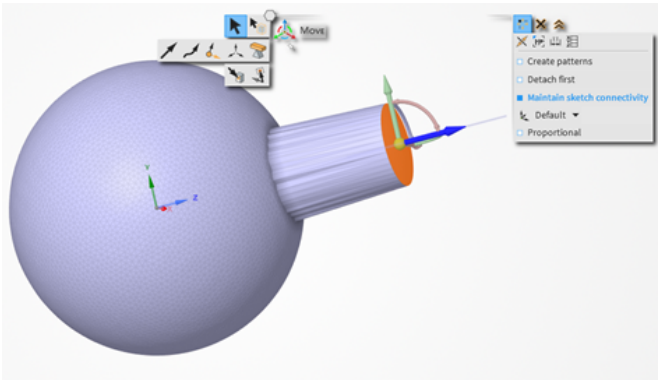
   The transformation specified determines the control vectors or the per-control-vertex transformation which defines an offset for each control vertex selected.

- For the **Pull** tool, control vectors are calculated for each control vertex from the vertex normals, multiplied by the offset specified.
- For the **Move** tool, control vectors are calculated from the transformation matrix computed by the tool based on the rotation/transformation and offset specified.

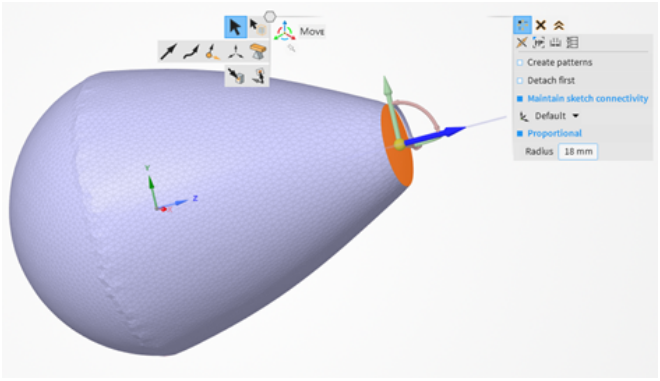
> **Important:** If you select multiple patches/regions/selections of the faceted mesh to deform at the same time, the same transformation will be applied to all selections, regardless of location on facet body.

This example shows a comparison of the default transformation with the proportional deformation for the same offset magnitude.

Default transformation



Proportional deformation



8.5. Modifying Faceted Bodies

The **Modify** group contains tools for combining, splitting, shelling, scaling, and thickening faceted bodies. The table below summarizes the tools.

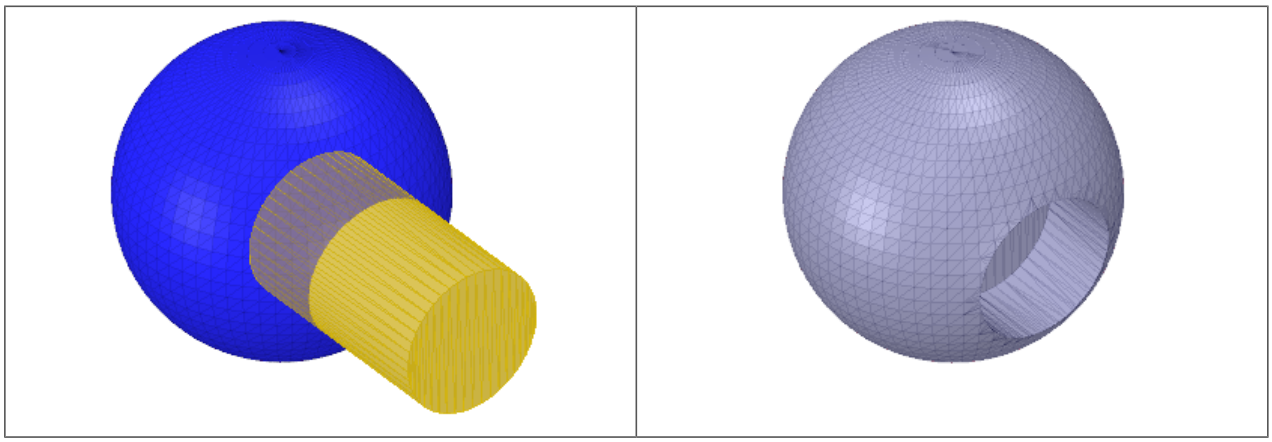
Merge	Use the **Merge** tool to combine two faceted bodies into a single body.
Subtract	Use the **Subtract** tool to subtract an intersecting region from one faceted body using the other intersecting faceted body.
Intersect	Use the **Intersect** tool to create a new faceted body from the intersecting region of two faceted bodies.

Split	Use the **Split** tool to split a faceted body using a cutter.
Shell	Use the **Shell** tool to create a thin wall on a closed faceted body.
Scale	Use the **Scale** tool to scale the selected faceted body relative to a selected point.
Thicken	Use the **Thicken** tool to add thickness to a faceted surface body.

This section contains the following topics:

8.5.1. Merge Tool

The **Merge** tool combines two faceted bodies, or a faceted body and a solid body.



Merging faceted bodies is a boolean operation similar to the **Combine** tool in the **Design** tab.

For Boolean tools, ensure that:

- The first faceted body you select is altered by the second
- The second faceted body is "consumed" by the first
- After the operation, the first faceted body is still in the model tree but the second is not

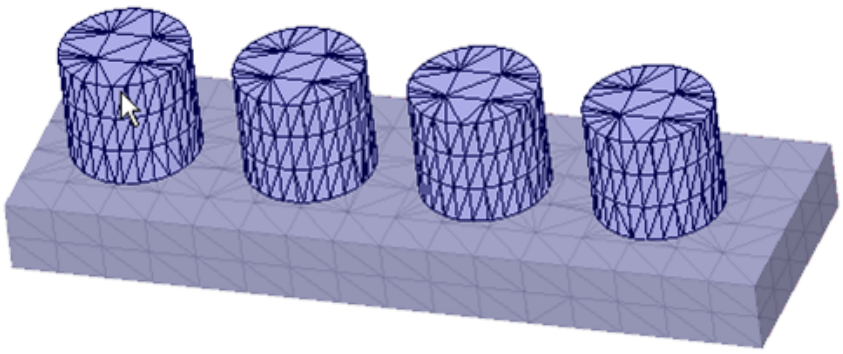
This section contains the following topics:

8.5.1.1. Merging Faceted Bodies

1. Select the **Merge** tool.
2. Select the first faceted body.
   The first faceted body is highlighted in blue
3. Select the second faceted body or a solid body.
   The two bodies are merged into one.
   If a solid body is merged with a faceted body, the solid body is converted to a faceted body

The **Merge** tool also works on two Open faceted bodies to automatically create the union that produces a single, closed faceted body. An example would be two halves of a shelled body merged into a closed shelled body.

Patterned components are up-selected in the Boolean tools. In the example below, the cylinders were made as a pattern. Highlighting one pattern member highlights the entire pattern.



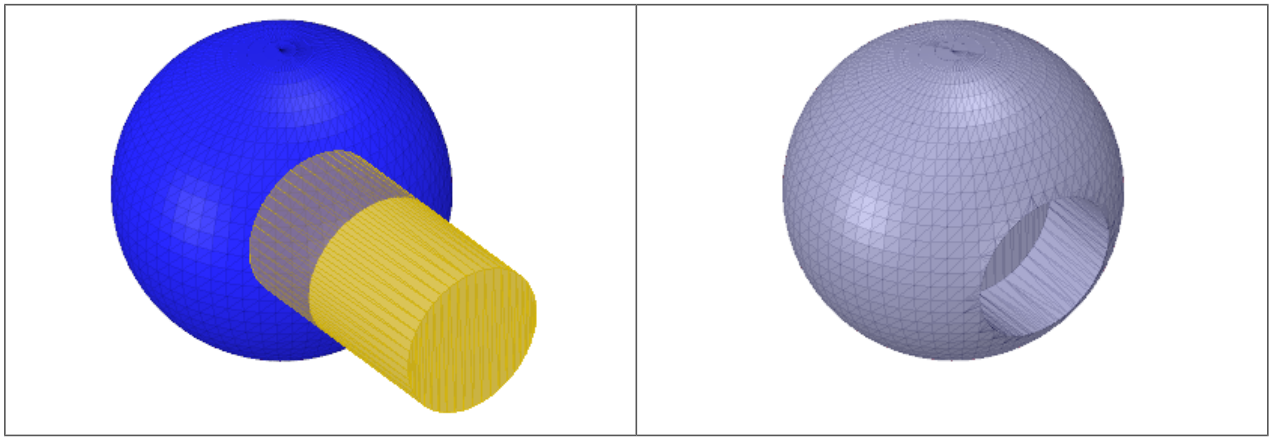
8.5.1.2. Merge Tool Options

The following options are available to control how bodies are merged:

Keep original bodies	Retain the original bodies after the intersection.

8.5.2. Subtract Tool

The **Subtract** tool removes the intersecting region between two faceted bodies, or a faceted body and a solid body.



Subtracting faceted bodies is a boolean operation similar to the **Combine** tool in the **Design** tab.

For Boolean tools, ensure that:

- The first faceted body you select is altered by the second.
- The second faceted body is "consumed" by the first.
- After the operation, the first faceted body is still in the model tree but the second is not.

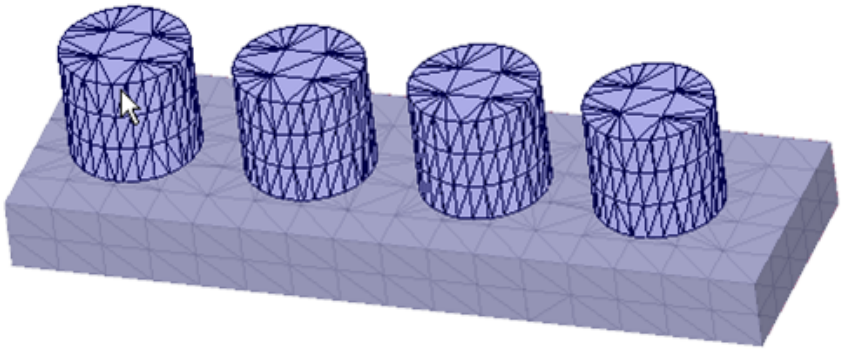
This section contains the following topics:

8.5.2.1. Subtracting Faceted Bodies

1. Select the **Subtract** tool .
2. Select the first faceted body.
   The first faceted body is highlighted in blue.

3. Select a second faceted body or a solid body to use as a cutter, or box-select multiple cutters.
   The second body is subtracted from the first.

Patterned components are up-selected in the Boolean tools. In the example below, the cylinders were made as a pattern. Highlighting one pattern member highlights the entire pattern.



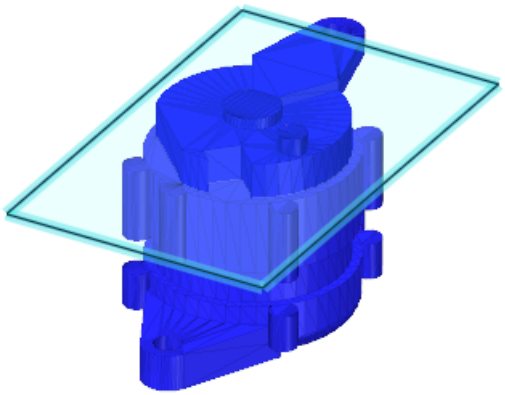
8.5.2.2. Subtract Tool Options

The following options are available to control how bodies are subtracted:

Keep original bodies	Retain the original bodies after the intersection.
Keep cutter	Retain the cutter body after the operation is complete.

8.5.3. Split Tool

Faceted bodies can be split using a plane or surface as a cutter.



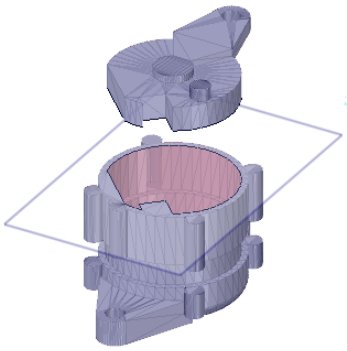
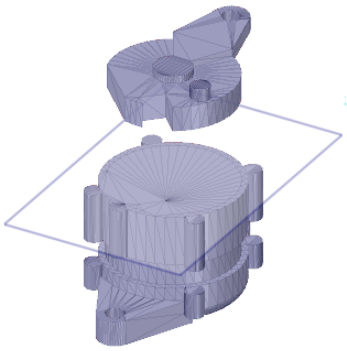
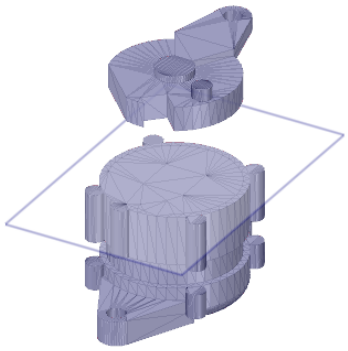
This section contains the following topics:

8.5.3.1. Splitting a Faceted Body

1. Click the **Split** tool.

2. Select a faceted body.
   The selected faceted body highlights in blue.
3. Choose a Cap option (see images below).
4. Select a plane, or surface, to use as a cutter.
5. The faceted body is split into two separate faceted bodies.

The images below show examples of the different options.

No cap	*Cap*	*Cap with internal points*
		

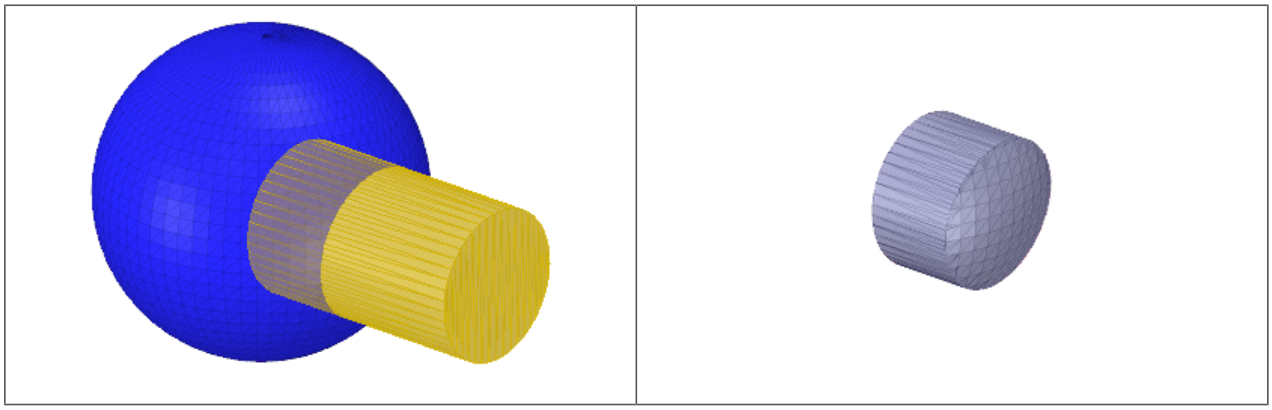
8.5.3.2. Split Tool Options

The following options are available to control how bodies are split:

No cap	Leave an opening after the split.
Cap	Cap the opening that results from the split
Cap with internal points	Cap the opening that results from the split but use extra points in the interior of the cap to produce better triangles.

8.5.4. Intersect Tool

The **Intersect** tool creates a new faceted body from the intersecting portion of two faceted bodies, or a faceted body and a solid body.



Intersecting faceted bodies is a boolean operation similar to the **Combine** tool in the **Design** tab.

For Boolean tools, ensure that:

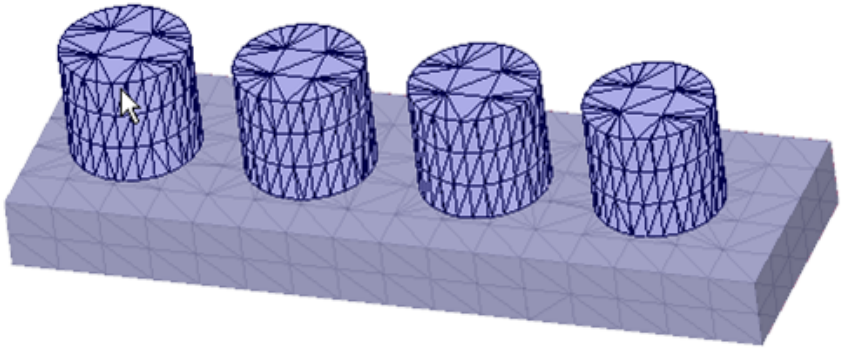
- The first faceted body you select is altered by the second.
- The second faceted body is "consumed" by the first.
- After the operation, the first faceted body is still in the model tree but the second is not.

This section contains the following topics:

8.5.4.1. Intersecting Faceted Bodies

1. Select the **Intersect** tool .
2. Select the first faceted body.
   The first faceted body is highlighted in blue.
3. Select the second faceted body, or a solid body.
   The two bodies are intersected.

Patterned components are up-selected in the Boolean tools. In the example below, the cylinders were made as a pattern. Highlighting one pattern member highlights the entire pattern.



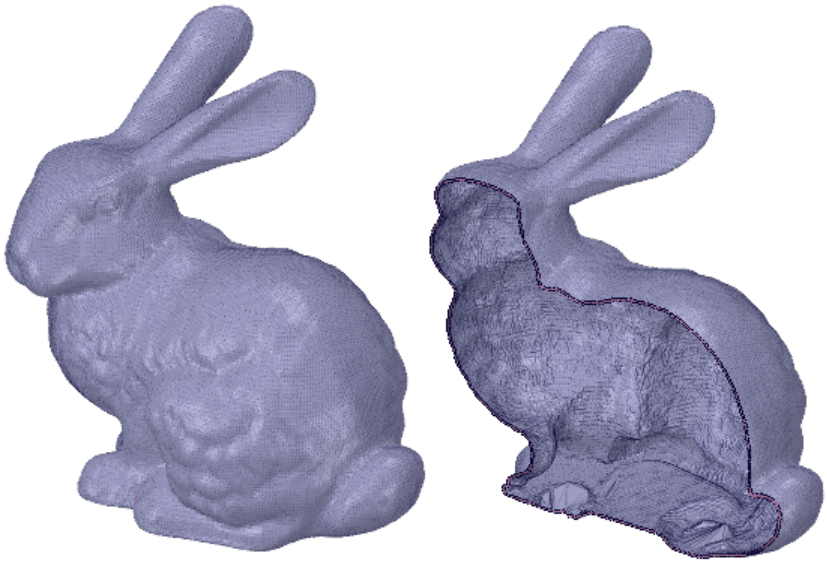
8.5.4.2. Intersect Tool Options

The following tool option is available:

Keep original bodies	Retain the original bodies after the intersection.

8.5.5. Shell Tool

Use the **Shell** tool to add thickness on the inside or outside of a faceted body.



This section contains the following topics:

8.5.5.1. Shelling a Faceted Body

To shell a faceted body:

1. Click the **Shell** tool.
2. Select a faceted body.
3. Select **Inside** or **Outside** to create the shell wall on the inside or outside the body and enter a **Thickness** of the shell wall.
4. (Optional) Select **Keep original bodies** to create a new shelled faceted body while retaining the original body.
5. (Optional) Choose to add a **Basic infill** structure for adding strength to the shelled body.

   Select an infill shape from the **Extrusions**, **Lattices**, or **Minimal Surfaces** available. Set the infill sizing and preview options. For **Lattices**, you can select an additional trim option.
6. Click **Complete**.

   The shell is added and a new faceted body is shown in the model tree

8.5.5.2. Shelling and Adding Infill Using Basic Shapes

Printing the model could take a long time and a lot of material, if the entire volume is filled.

Infill lets you shell the interior of a model and add an internal support structure. This provides strength while reducing material costs and printing time.

The tool has several options for controlling the size, shape and percentage of the volume filled.

To shell by adding an infill using basic shapes:

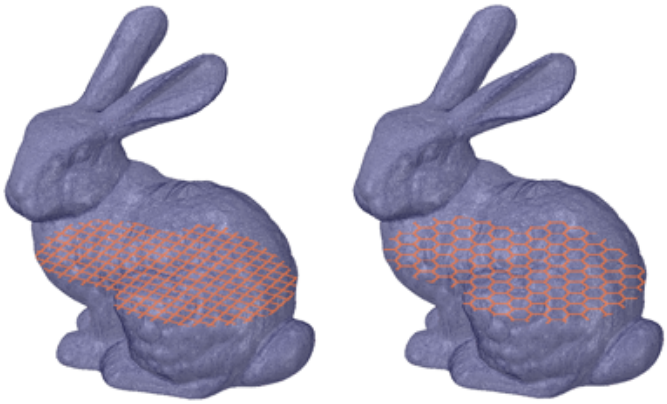
1. Select the **Shell** tool.
2. Set the **Thicken direction** to **Inside**.
3. Set the **Thickness** of the shell wall.
4. Choose whether or not to keep the original faceted body after it is shelled.
5. Choose the **Basic** option.

6. Select the faceted body to shell.
7. Choose an **Infill Shape**.
   - Use the dropdown list to select a group: Extrusions, Lattices, or Minimal Surfaces.
   - Select a desired infill shape.
8. Set the **Infill Size** options.
9. Adjust the Preview.
   - Choose a Preview **Direction** (X, Y, or Z).
   - Use the **Offset** slider to move the preview section along the Preview Direction.
   - Click the **Reset** button to go back to the original Offset location.
10. Click **Complete**.

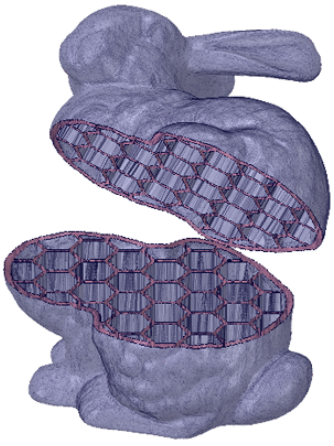
    The Shell is added and a new Faceted Body is shown in the model tree.

The images below show preview for **Square** and **Hex** Infill shapes.

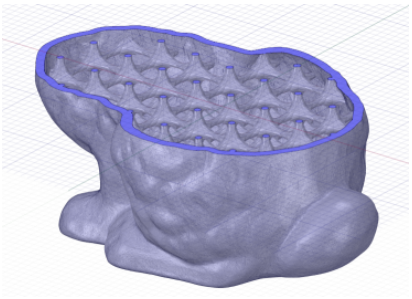
The Y-axis is chosen for the preview Direction and the Offset was adjusted to place the preview near the middle of the model along the Y-axis.



*The image below shows the model shelled with a **Hex** infill. The body was split and moved to expose the Infill.*



*The image below shows the model shelled with a **3D lattice** infill.*



Note: Lattice infills are sensitive to the coarseness of the faceting. Coarse facets produce lattice infills faster. Coarseness is controlled when you convert a solid body to a faceted body using the **Convert** tool. When converting, use a Maximum Angle in the neighborhood of 30° to get fewer facets and speed up the lattice calculations.

8.5.5.3. Shell Tool Guides and Options

Shell Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Solid or Faceted Body	The **Select Solid or Faceted Body** tool guide is active by default. It allows you to select solid or faceted bodies to shell.
Select Solid or Faceted Body	The **Select Solid or Faceted Body** tool guide allows you to select geometry or facets to be removed when shelling.
Complete	The **Complete** tool guide completes the shell.

Shell Tool Options

The following options are available:

Inside	Remove material inside and leave a thin wall to the inside of the original body.
Outside	Remove material inside and leave a thin wall to the outside of the original body.
Thickness	Enter a value for the shell thickness in model units.
Keep original bodies	Create a new shelled faceted body and retain the original body.
Basic Infill	Choose the type of infill structure to be created for adding strength to the shelled body when it is 3D printed. See **Basic Infill** tool options in the table below.

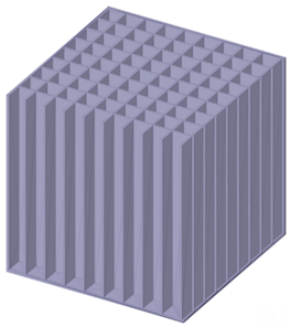
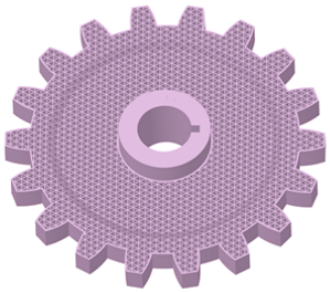
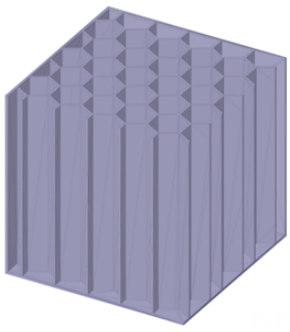
Basic Infill Tool Options

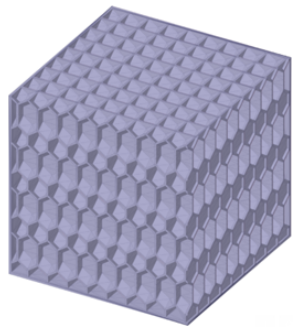
The following options are available:

Shape	The infill shape options are separated into three groups with multiple options in each. See Basic Infill Shapes on page 461 for descriptions of the infill shapes available.
Sizing Options	The sizing options are linked. Changing one will adjust the others.
Fill %	Percentage of the shell volume filled by material
Length	Determines the size of the basic shapes used in the infill pattern
Thickness	Sets the infill wall thickness
Preview Options	Allow you to control how the preview is displayed and positioned
Direction	Choose the X, Y, Z axis, or the Screen to set the preview direction.
Offset	Use the dropdown slider to drag the preview along its normal direction. Use the Reset button to go back to the previous location.

8.5.5.4. Basic Infill Shapes

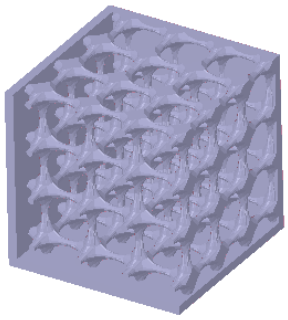
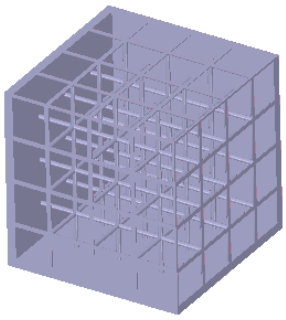
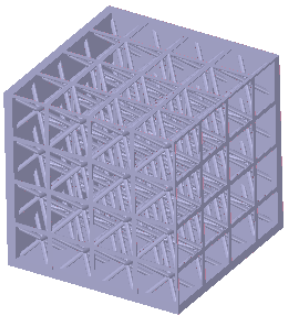
Extrusions

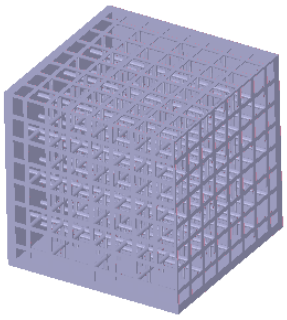
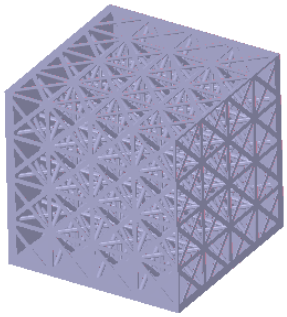
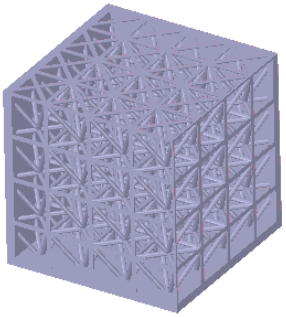
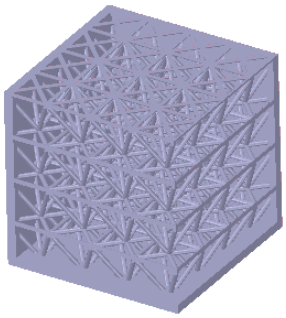
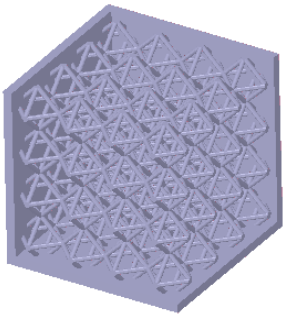
Square Extruded square grid.	
Triangle Extruded triangle grid.	
Hex Extruded hexagonal grid.	

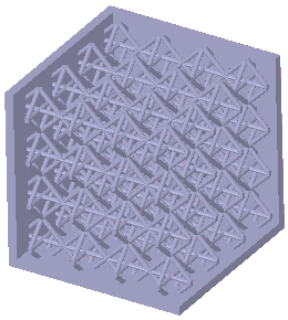
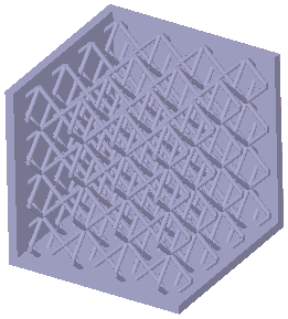
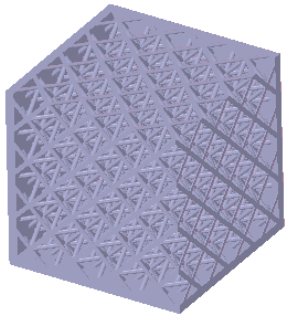
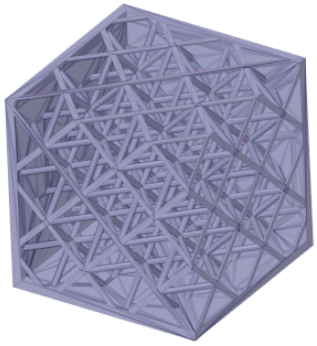
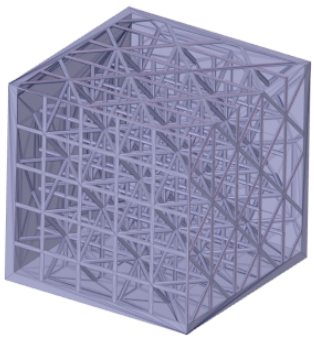
3D Hex 3D hexagonal grid similar to honeycomb structures. Cross-sections alternate between hexagons and square.	

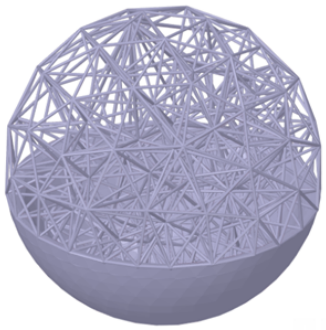
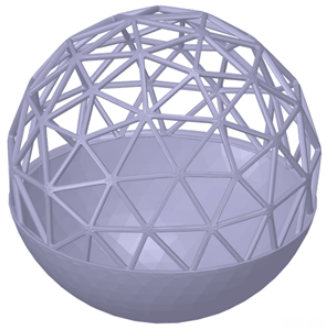
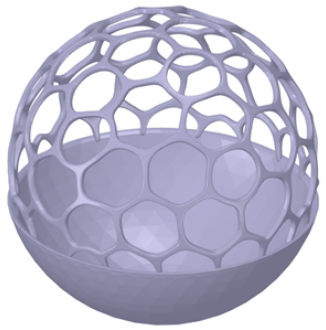
Lattices

Select a Lattice structure and Lattice trimming option.

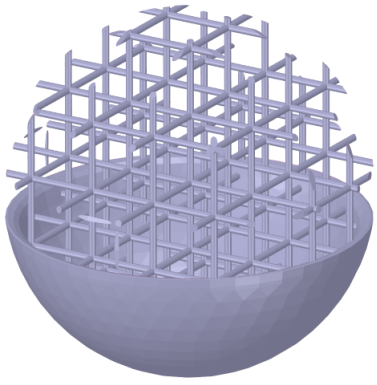
Lattice Three dimensional lattice infill pattern	
Regular Cube Lattice Simple cube lattice infill pattern	
Cube Lattice with Center Supports Cube lattice with supports from all corners of the cube intersecting at the center of the cube	

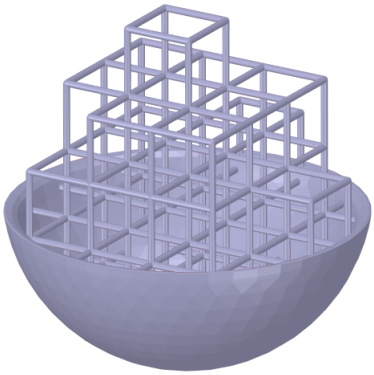
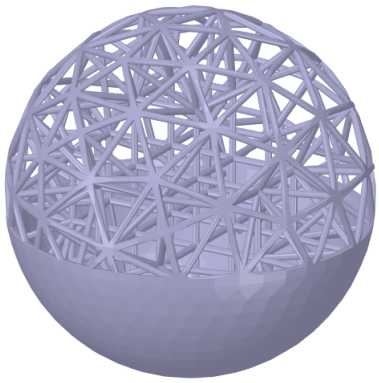
Cube Lattice with Side Cross Supports Cube lattice with cross supports between edges on each face of the enclosing cube	
Cube Lattice with Side Diagonal Supports Cube lattice with diagonal supports between vertices on each face of the enclosing cube	
Cube Lattice with Bottom Center Cube lattice with supports that meet at the bottom face center of the enclosing cube	
Cube Lattice with Bottom Center Without Vertical Supports Cube lattice with supports that meet at the bottom face center of the enclosing cube but without vertical supports	
Double Pyramid Lattice Double pyramid lattice with lateral supports	

Double Pyramid Lattice with Cross Double pyramid lattice with internal cross supports	
Diamond Lattice Double pyramid without the lateral supports	
Double Pyramid Lattice and Face Diagonals Double pyramid in the center of the lattice with diagonals on faces of the enclosing cube	
Octahedral-1	
Octahedral-2	

Tetrahedral Faceted Lattice Boundary conforming lattice structure, where the lattice elements follow the edges of tetrahedra.	
Triangle Surface Faceted Lattice Uniform surface lattice, smoothly connected at the vertices with triangular holes.	
Hexagon Surface Faceted Lattice Smoothed Voronoi dual to the triangular surface mesh. Holes are on average hexagonal.	

Select a lattice trim option:

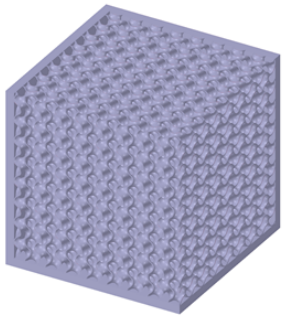
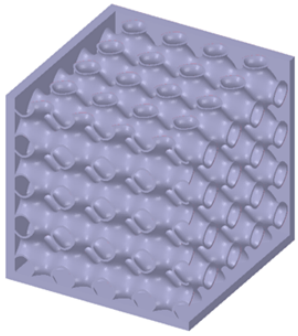
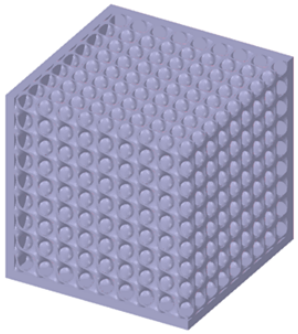
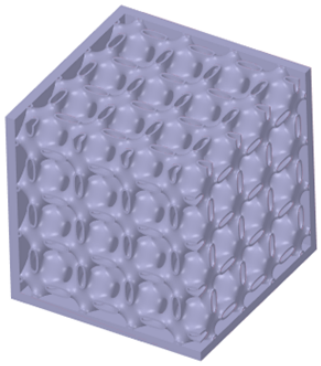
None: No explicit trimming is done. The internal lattice structure is trimmed by the original part, which may leave dangly lattice pieces that are not connected to other lattice pieces	

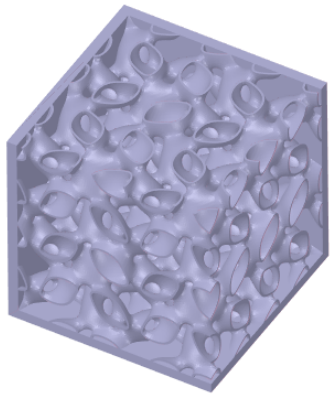
Remove partial segments: trims the dangling pieces back so that every lattice has at least two lattice edges ending in it.	
Boundary conforming: builds a Triangular Lattice between the surface of the geometry shell and the interal lattice structure.	

Note:

- The **Boundary conforming** and **Remove partial segments** options are not available for the following lattice types, since they are already boundary conforming by design.
  - Tetrahedral Faceted Lattice
  - Triangle Surface Faceted Lattice
  - Hexagon Surface Faceted Lattice
- Lattice trimming is not available for the three dimensional infill pattern "**Lattice**".

Minimal Surfaces

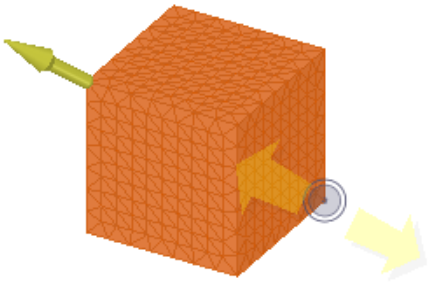
Gyroid Belongs to Triply Periodic Minimal Surfaces.	
Schwarz P Two intertwined congruent labyrinths, each with the shape of an inflated tubular version of the simple cubic lattice.	
Schwarz D Two intertwined congruent labyrinths, each having the shape of an inflated tubular version of the diamond bond structure.	
Neovius A triply periodic minimal surface, dividing space into two infinite non-equivalent labyrinths.	

Lidinoid A triply periodic minimal surface similar to a gyroid.	

You can choose to add a smooth transition from the minimal surface infills to the boundary of the original part. Select **Boundary transition** and specify an appropriate value for **Radius** to add a smooth blend to the boundary.

8.5.6. Scale Tool

Faceted bodies can be scaled smaller or larger relative to a selected point.



This section contains the following topics:

8.5.6.1. Scaling a Faceted Body

1. Click the **Scale** tool.
2. Select the faceted body (or bodies) in the Graphics window.
3. Select a point on the faceted body to scale about.
4. (Optional) Switch on/off the scaling directions in the Options panel to scale non-uniformly. By default, all directions are selected for uniform scaling.

   For non-uniform scaling, use the **Orient handles** tool guide to select an origin to orient the direction handles.
5. Drag to change the scale, or press the Spacebar and specify a scale value.

You can also scale the faceted body using the **Pull** tool.

8.5.6.2. Scale Tool Guides and Options

Scale Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Scale faceted body	The **Scale faceted body** tool guide is active by default. This tool guide allows you to select faceted bodies to be scaled.
Select point	The **Select point** tool guide allows you to select a point to scale about.
Orient handles	The **Orient handles** tool guide allows you to select an origin to orient the direction handles for non-uniform scaling.

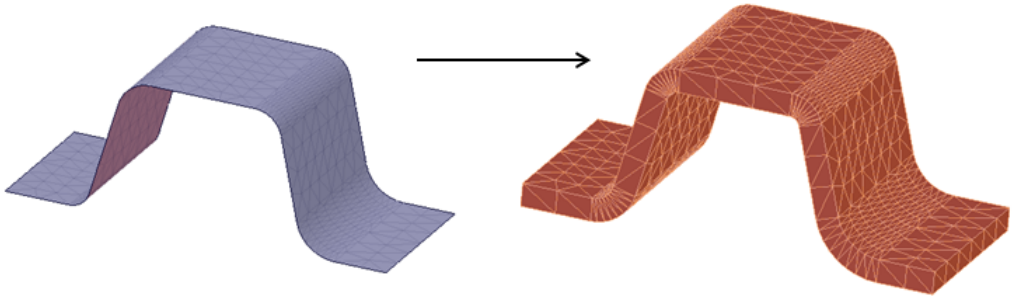
Scale Tool Options

The following tool option is available:

Scaling directions	You can toggle On/Off the X, Y, and Z directions to scale non-uniformly. All directions are On by default for uniform scaling.

8.5.7. Thicken Tool

Use the **Thicken** tool to add thickness to a surface faceted body.



This section contains the following topics:

8.5.7.1. Thickening a Surface Faceted Body

1. Create or import a surface faceted body.
2. Select the **Thicken** tool.
3. Choose a thickening **Direction** in the options panel and enter a thickness value in the options panel.
4. Select the surface faceted body.
5. Exit the tool or Undo, change the options, and reselect the surface faceted body.

8.5.7.2. Thicken Tool Options

The following options are available to control thickening:

Inside	Add thickness to the inside.
Outside	Add thickness to the outside.
Both	Add thickness in both inward and outward directions.
Thickness	Enter the total thickness value in model units.
Fix intersections	Fix intersections that occur because of the thickening.

The **Zoom Legend** appears at the lower-right of the screen. If the zoom legend is on before entering this tool or is turned on while using this tool, it will remain on when exiting the tool.

8.6. Adjusting Facets

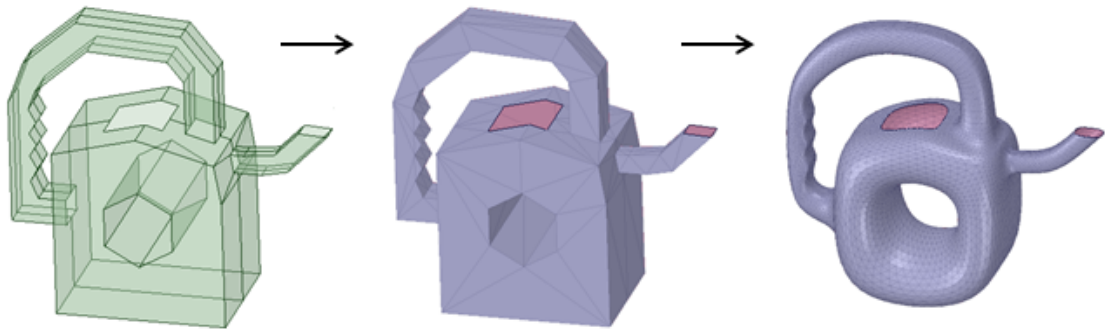
The **Adjust** group contains tools for smoothing facets, reducing the number of facets, and making facets more uniform (Regularize).

Smooth	Use the **Smooth** tool to smooth out rough facet regions.
Reduce	Use the **Reduce** tool to reduce the number of facets in a faceted body.
Regularize	Use the **Regularize** tool to make the aspect ratios of selected facets more consistent.

This section contains the following topics:

8.6.1. Smooth Tool

The **Smooth** tool is typically used on imported faceted bodies.



This section contains the following topics:

8.6.1.1. Smoothing Facets or a Faceted Body

1. Select the **Smooth** tool.

2. Change the **Smooth Options** if desired.
3. Select facets or a faceted body.
4. Click **Complete** to smooth the selection.
5. You can continue to click **Complete** to further smooth the facets.

You can also select facets or a faceted body and then select the **Smooth** tool.

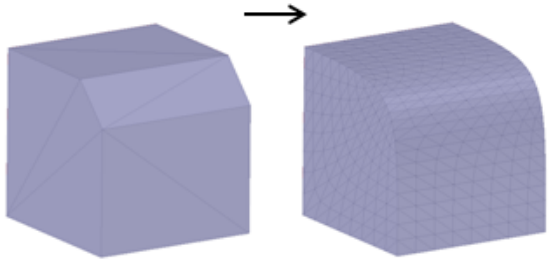
8.6.1.2. Smooth Tool Options

The following options are available for smoothing facets:

Angle threshold	Determines if an angle stays sharp or gets smoothed.
Flatten Peaks	**Flatten Peaks** works locally on slightly rough areas. Topology is maintained and local spikes are reduced.
Add Facets	The **Add Facets** option allows vertex positions to move, achieving generally smoother results but possibly with significant changes in volume and triangle count.
Volume Aware	**Volume Aware** smoothing keeps the triangle count the same and tries to maintain the original volume.

Angle threshold

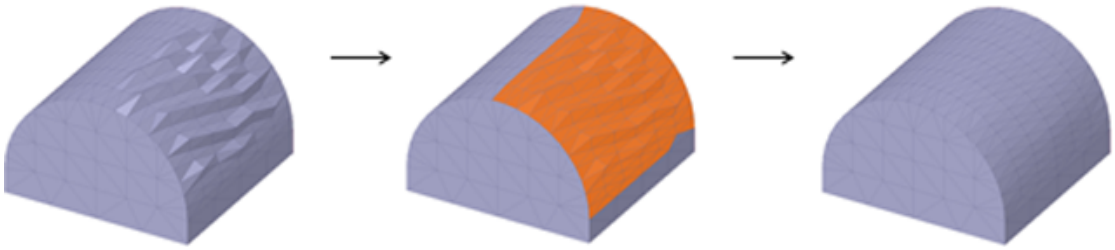
This option determines if an angle stays sharp or gets smoothed. Vertices on edges with faces coming together at angles above the **Angle threshold** will not be moved or changed. The example below shows a block that has all 90° edges except for two which are 45°. With an angle tolerance of 60°, only the two 45° edges get smoothed and the 90° edges are left alone.



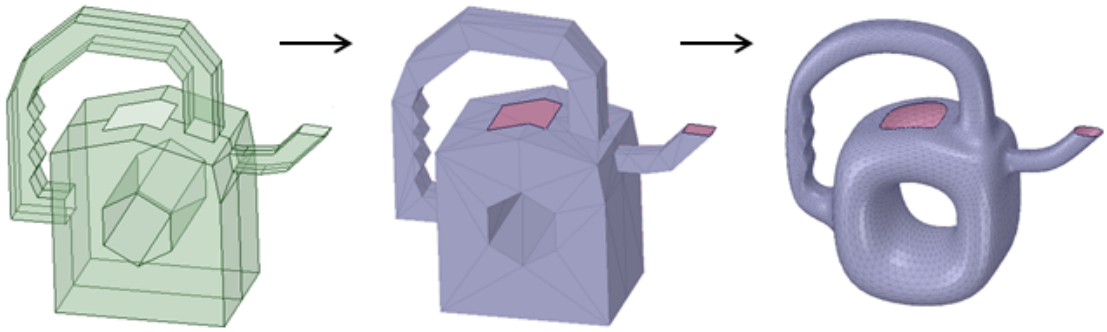
Smooth type

Determines which of the following algorithms is used to do the smoothing.

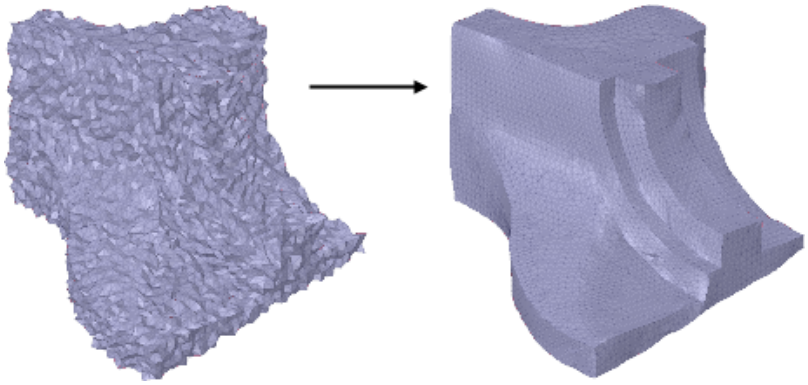
- **Flatten Peaks**:
  - Flatten Peaks keeps the triangle count the same.
  - It mostly knocks down spikes.
  - The example below shows how repeated application of Flatten Peaks smoothing works on slightly rough areas. The number of facets remains the same and the spiked areas are brought down closer to the surroundings.



- **Add Facets**:
  - The original vertex positions and the new vertex positions are permitted to move.
  - The amount of movement depends on how smooth the surroundings are.
  - In sharp areas, the amount of movement will be large.
  - In smoother areas, the amount of movement may be barely noticeable.
  - Because of the movement, the Add Facets option will, in general, produce smoother results.
  - The Add Facets option can produce results that are noticeable as shown in the example below. The final result is achieved with several smoothing passes.

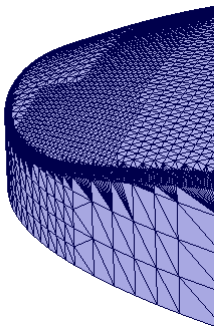
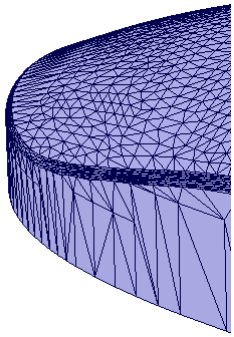


- **Volume Aware**:
  - Volume Aware does not change topology.
  - It tries to maintain the sharps and minimize volume loss.
  - The example below shows how Volume Aware smoothing works on the rough faceted body.



8.6.2. Reduce Tool

The **Reduce** tool reduces the number of facets, which influences 3D printing time. You can control the amount of reduction through the Reduce Options.

Original mesh	*Reduced mesh*
	

This section contains the following topics:

8.6.2.1. Reducing the Number of Facets

1. Click the **Reduce** tool.
2. Change the **Reduce Options** if desired.
3. Select a faceted body or box-select facets.
4. Click **Complete**.

   The model or selection is re-faceted with fewer triangles based on the **Triangle reduction** and **Maximum deviation** specified.
5. You can continue to click **Complete** to reduce further.

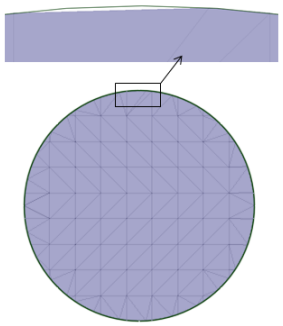
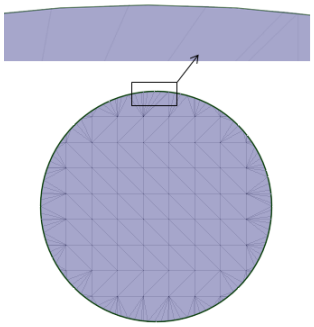
You can also make your selection and then select the tool.

8.6.2.2. Reduce Tool Options

The following options are available for reducing the number of facets with the **Reduce** tool:

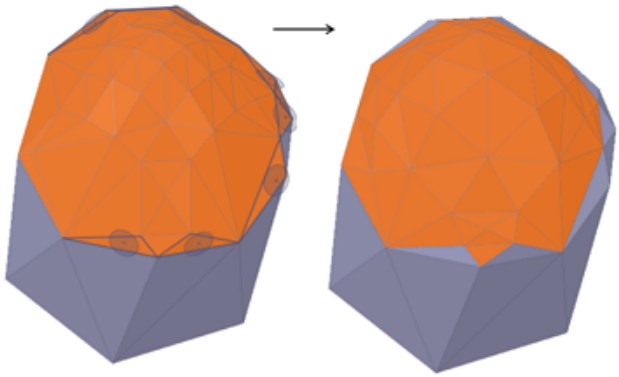
Triangle reduction	The percentage reduction you want to achieve. The default is 20%.
Maximum deviation	Controls the distance between triangle edges and the actual edges of the part. It influences the number of triangles used to approximate curved edges. See the images below.

The following images show how the maximum deviation specified influences the number of triangles used to approximate curved edges.

Larger maximum distance	*Smaller maximum distance*
	

8.6.3. Regularize Tool

The **Regularize** tool improves the quality of selected facets by making their aspect ratios more consistent. You can control the process with the Regularize Options. Geometry operations performed by the tool do not remove Named Selections.



This section contains the following topics:

8.6.3.1. Regularizing Selected Facets

To regularize a selection of facets:

1. Select the **Regularize** tool.
2. Change the **Regularize Options** if desired.
3. Select a faceted body or box-select facets.

> **Note:** The default value for regularize is determined based on the selected facets on the body.

4. Click **Complete**.
   The model or selection is re-triangulated.
5. You can continue to click **Complete** to regularize further.

You can also make your selection and then select the tool.

8.6.3.2. Regularize Tool Options

The following options are available for regularizing facets:

Angle threshold	Determines if an angle stays sharp or gets smoothed.
Max edge length	Sets the maximum edge length.
Deviation	Specifies the maximum distance between the new and old faceting.
Curvature dependent	Varies the facet size based on curvature. Enabling this option makes the triangles conform more to the shape with different sized triangles.

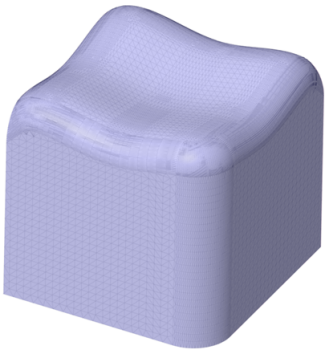
The **Angle threshold** option sets the facet-to-facet angle limit when regularizing the selected facets. Vertices on edges with faces coming together at angles above the Angle Threshold will not be moved or changed.

The **Zoom Legend** appears at the lower-right of the screen. If the zoom legend is on before entering this tool or is turned on while using this tool, it will remain on when exiting the tool.

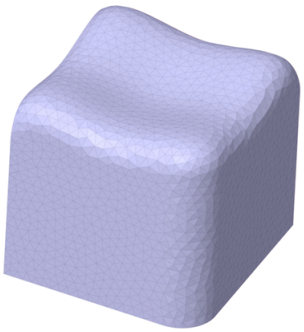
8.6.3.3. Regularize Tool Examples

The following examples show the use of the **Regularize** tool:

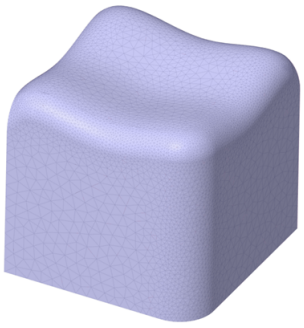
Original faceting.



Faceting after applying Angle threshold and Max edge length.



Faceting after applying Angle threshold and Deviation, with Curvature dependent enabled.



8.7. Inspecting Faceted Bodies

The **Inspect** group contains tools for checking faceted bodies for overhangs, thickness, and cavities. These items need to be considered for 3D printing.

Overhangs	Use the **Overhangs** tool to detect and highlight areas of the faceted body that protrude out at angles up to 90°
Thickness	Use the **Thickness** tool to check the minimum thickness of a faceted body.
Cavity	Use the **Cavity** tool to find cavities that are completely enclosed inside a faceted body.

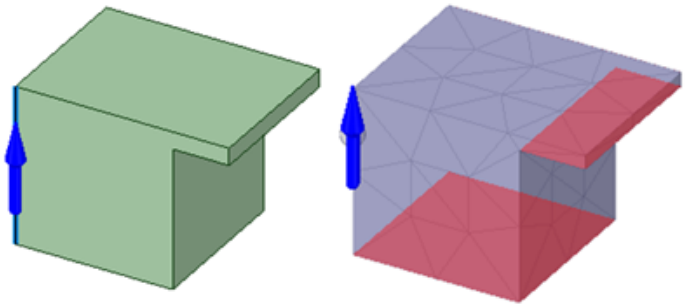
This section contains the following topics:

8.7.1. Overhangs Tool

Overhanging geometry is an important consideration in 3D printing because it may require special supports. This can add cost and setup time to the process.

The **Overhangs** tool detects and highlights areas of the mesh that protrude out at angles up to 90°. You can set the **Overhang Angle** option to look for any angle up to 90°.

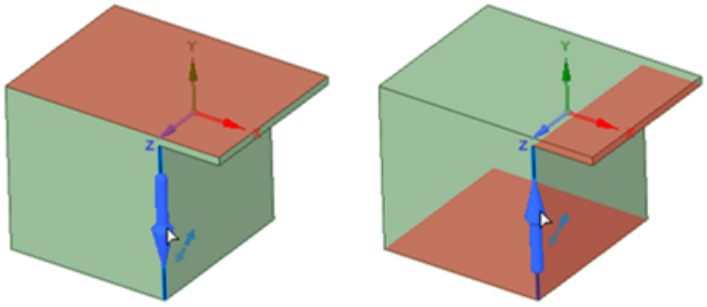
The Overhangs tool works on both faceted and solid bodies. You can choose to select highlighted overhangs for additional processing or to modify the body to minimize the overhang area.



This section contains the following topics:

8.7.1.1. Checking a Body for Overhangs

1. Click the **Overhangs** tool.
2. Select a body.
3. Click the **Select Direction** tool and select a solid or facet edge to set the direction to look for overhangs. ALT-select also works for selecting a direction.



8.7.1.2. Overhangs Tool Guides and Options

Overhangs Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Body	Select the body to locate overhangs.
Convert to selection	Add overhanging facets or geometry found by the Overhangs tool to the selection.
Select direction	Select the direction to look for overhangs.
Minimize overhang area	When the Overhangs tool finds overhanging facets or geometry, create a copy of the faceted body, and orient it to a position that minimizes the overhang area.

Overhangs Tool Options

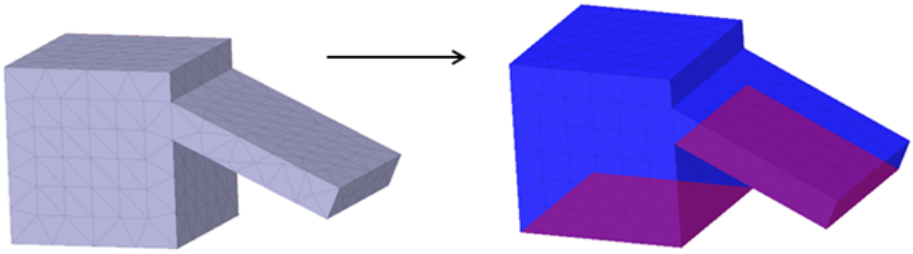
The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Overhang angle	Enter a value in degrees. Faces that overhang by *More Than* the specified angle will be detected and highlighted.
Ignore small areas	Enter a value. Overhangs with areas *Smaller Than* the specified value will be ignored.

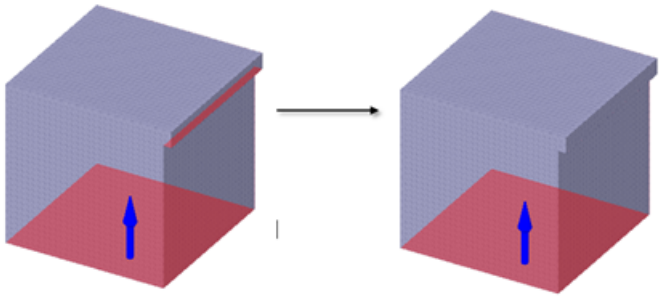
8.7.1.3. Overhangs Tool Examples

The following options are available for detecting overhangs:

*In the example below, the overhang is at 65°. The **Overhang Angle** is set to 64°.*



*In the example below, there is a small overhang with an area of 20 mm^2. It is found when **Ignore Small Areas** is checked **OFF**, but ignored when the option is checked **ON** and set to 30 mm^2.*

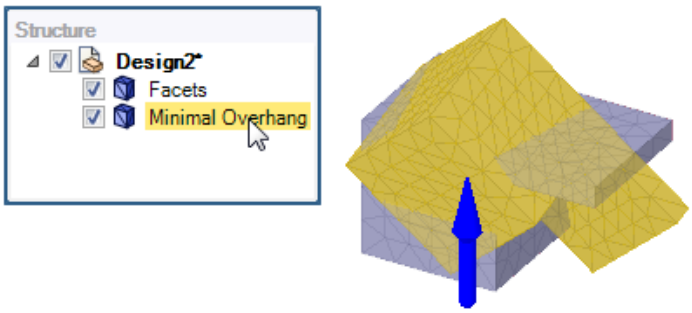


8.7.1.4. Converting the Highlighted Overhang(s) to a Selection

When the **Overhangs** tool finds overhanging facets or geometry, click the **Convert to Selection** tool guide to add them to a selection.

8.7.1.5. Minimizing the Overhang

When the **Overhangs** tool finds overhanging facets or geometry, click the **Minimize Overhang Area** tool guide. This creates a copy of the Facets body and then orients it to a position that minimizes the Overhang area. This orientation could be used in 3D printing to have minimal overhang.



8.7.2. Thickness Tool

Thickness is important for planning 3D printing. The **Thickness** tool lets you measure the thickness based on facet normals or relative to the printer vertical direction.

This section contains the following topics:

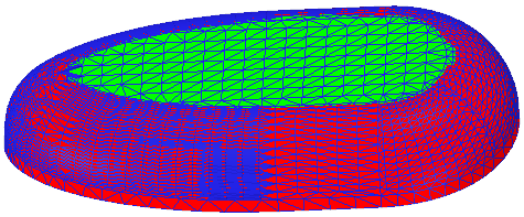
8.7.2.1. Checking Faceted Body Thickness

1. Select the **Thickness** tool.
2. Enter a **Minimum thickness** in the Options panel or use the default value.

   You can also click the arrow to open a slider for setting the Minimum thickness. The Thickness display changes dynamical as you drag the slider.
3. (Optional) Click the **3D Printing Direction** tool guide and select an face, edge, point or axis parallel to the printer vertical direction.
4. Select a faceted or solid body.

   Although solid bodies can be selected in the **Thickness** tool, they will produce no results.

The results are shown shaded with areas that are less than the Minimum thickness shown in red. An example is shown below.



8.7.2.2. Thickness Tool Guides and Options

Thickness Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Faceted Body	The **Select Faceted Body** tool guide is active by default. Use it to select a faceted body to analyze.
3D Printing Direction	Use the **3D Printing Direction** tool guide to select a line, edge, or axis for the 3D printing direction.

Thickness Tool Options

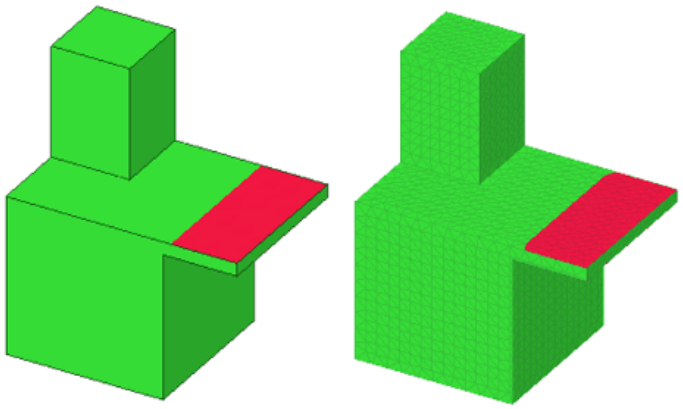
The following options are available:

Minimum thickness	Enter a value in model units. Any areas that have a thickness *Less Than* the minimum thickness will be detected and highlighted.
Sample spacing	Controls the maximum distance between sample points. Smaller values will create more samples and better visualization, but will also increase the computation time of the tool.

The **Zoom Legend** appears at the lower-right of the screen. If the zoom legend is on before entering this tool or is turned on while using this tool, it will remain on when exiting the tool.

8.7.2.3. Thickness Examples

The **Sample spacing** tool is intended for solid bodies. The images below show the tool applied to a solid body and faceted body.



8.7.3. Cavity Tool

The **Cavity** tool finds cavities that are completely enclosed inside faceted bodies. It's important to identify cavities because they can trap excess 3D printer material. The **Cavity** tool works on both faceted and solid bodies.

This section contains the following topics:

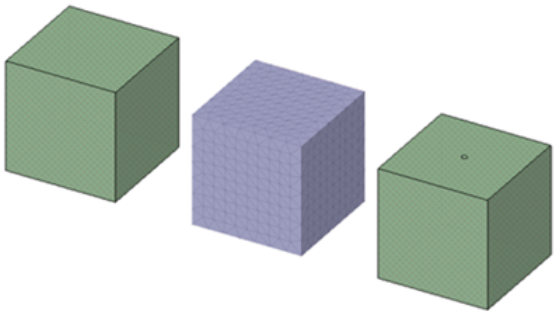
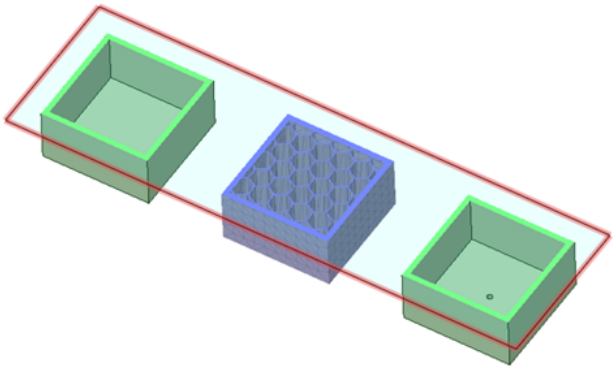
8.7.3.1. Checking a Body for Cavities

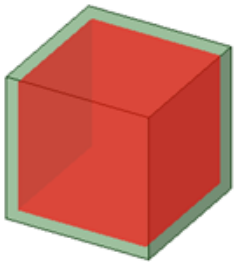
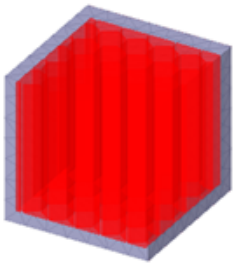
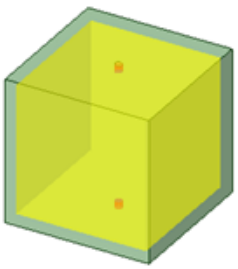
1. Select the **Cavity** tool.
2. Optional: Check **Max Opening** and enter a value for finding small openings.
   - Solid bodies can have small openings used for draining excess material
   - These small holes can prevent cavities from being found in these areas.
3. Select a body.

   Cavities are detected.

8.7.3.2. Checking for Cavity Examples

In this example, three bodies will be checked for cavities. The left solid body is a shelled block, the faceted body in the middle is shelled with a Basic Hex Infill, and the right solid body is shelled with a 1 mm diameter hole through the top and bottom walls.

A clipping plane is used to show the interior details.	
The **Cavity** tool gives the following results.	

The left solid body has one cavity.	
The middle faceted body has 42 cavities due to the Hex Infill.	
The right solid body has a through hole so no cavities are found. However, the through hole is small (1 mm) and small openings can cause problems for draining the excess material. Click the **Max Opening** option and enter "1". One cavity is now found and the small openings are highlighted in red.	

8.7.3.3. Cavity Tool Options

The following tool option is available:

Max Opening	Set the max opening value to detect cavities.

8.8. Reverse Engineering

This section contains the following topics:

8.8.1. Auto Skin Tool

The **Auto Skin** tool generates the CAD geometry from faceted models that are relatively smooth and error free. For example, the starting point of your design may be a clay model which is scanned and output to one of the supported faceted data types. After importing the faceted data as a Facet Mesh, use the **Auto Skin** tool to create the CAD geometry.

You can also use the **Auto Skin** tool to create the SubD hybrid model for models with topology optimization results from the Mechanical application.

Note:

- The faceted model should be smooth and free from defects (for example, the model should be manifold with no sharp edges, self intersections, etc.). If the model boundary is not smooth, the CAD geometry creation may fail or result in incorrect geometry.
- Only a single faceted body can be converted at a time.
- The conversion is dependent on the number of facets, inflection points, curves, etc. Performance will be impacted when skinning models with a large number of facets (above 200,000 triangles).
- The **Skin Surface** tool can be used to manually create any missing patches.

This section contains the following topics:

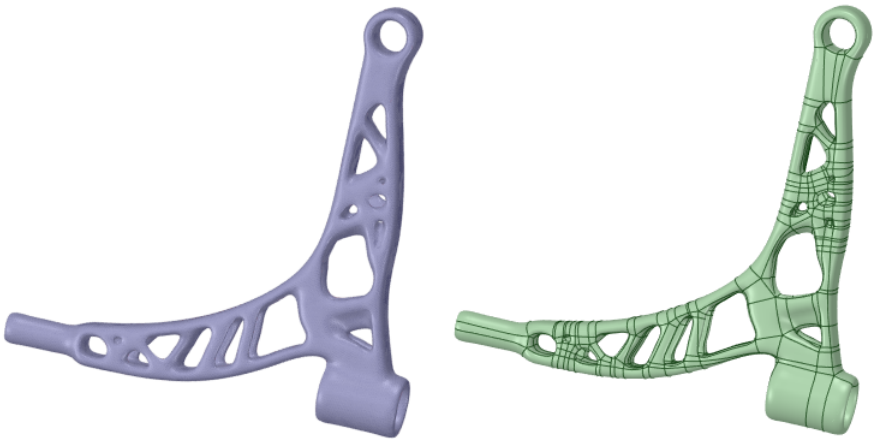
8.8.1.1. Creating the CAD Geometry

1. Import or create a facet mesh.
2. Use the tools in the **Cleanup** group in the **Facets** tab to ensure the model is smooth and watertight before skinning.
3. Select the **Auto Skin** tool in the **Reverse Engineering** group of the **Facets** tab.
4. The **Select Faceted Body** tool guide is active by default. Select the faceted body.
5. The **Select Solid Bodies** tool guide is activated when the mesh selected has groups or facet regions. Select the solid bodies which match the facet regions.

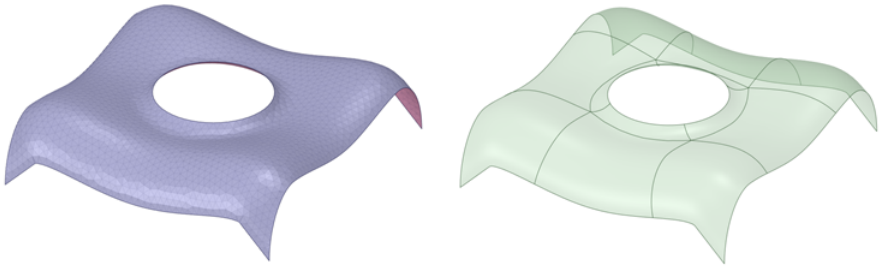
   For example, models with topology optimization results from the Mechanical application contain groups/facet regions. These models will generally have the facet regions set on import.

   To create facet regions in the Discovery application, select the facets and then right-click and select **Add Facet Region** from the context menu.
6. Click **Complete** to convert the model to CAD geometry.

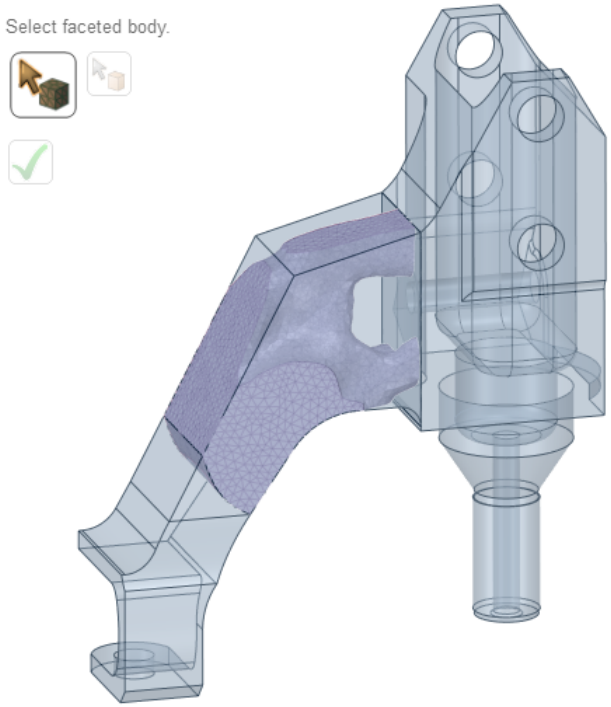
This example shows the conversion of a faceted model to CAD geometry.



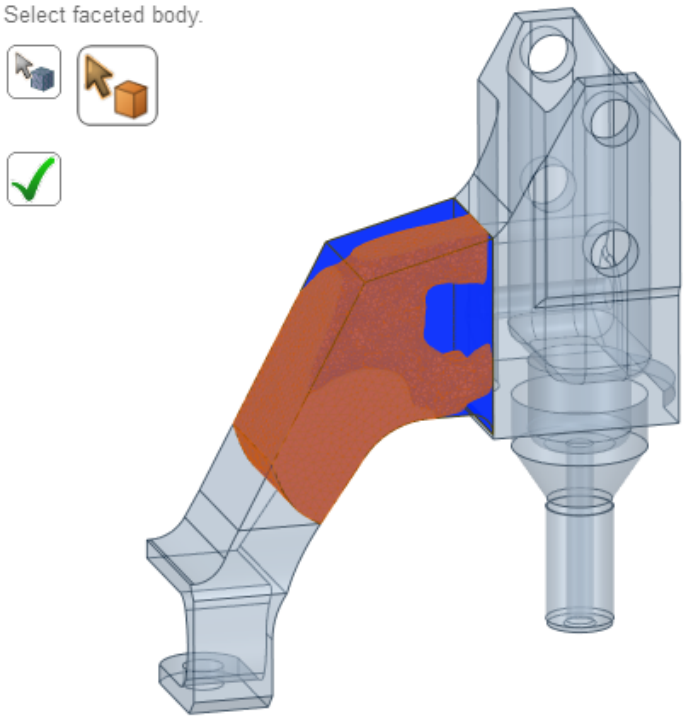
This example shows the conversion of a faceted model with an open body to CAD geometry.



This example shows a model with a facet region.

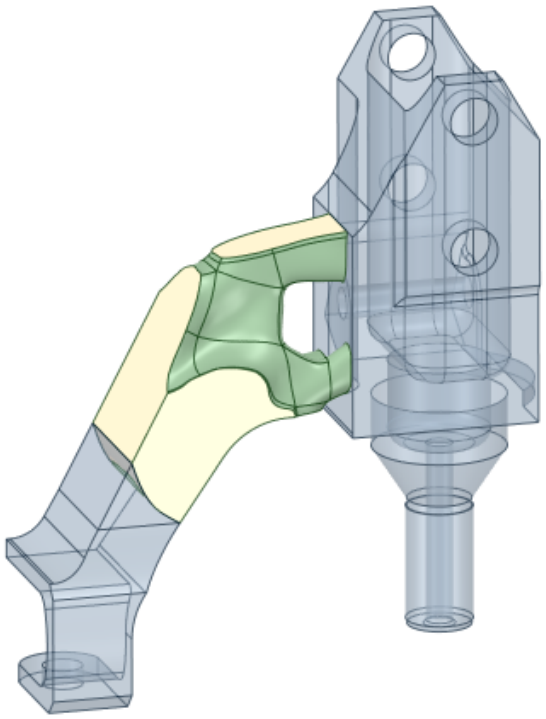


Select the faceted mesh and the solid bodies which have faces that match the facet regions as shown in the example.

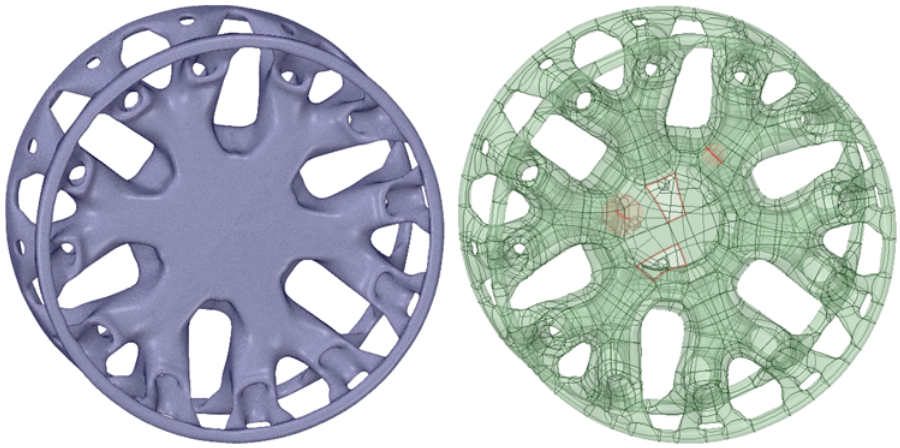


The CAD faces will be trimmed to the facet regions. Triangles on the mesh which do not match a selected CAD face will be autoskinned.

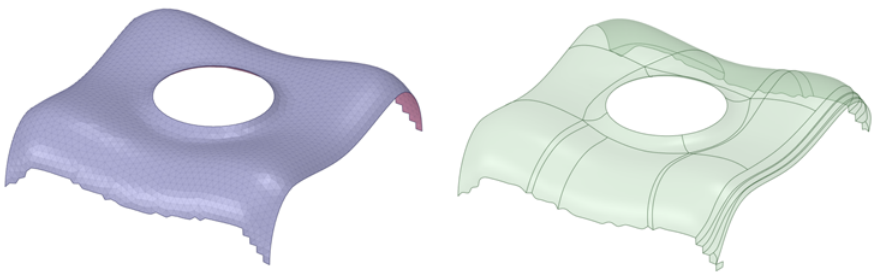
In the example below, the green faces are the faces generated by the autoskin algorithm. The beige/white ones are copies from the selected CAD faces. These faces are trimmed if the facet region corresponding to a CAD face does not cover the whole face (as seen in the example).



7. Verify the CAD geometry. In the example, the CAD geometry created has missing patches.



If the geometry has missing patches, use the **Skin Surface** tool to create the missing patches.
If the boundary is not smooth, the CAD geometry creation may fail or result in incorrect geometry.



8.8.1.2. Creating the SubD Hybrid Model

You can also use the **Auto Skin** tool to create the SubD hybrid model from faceted models that are relatively smooth and error free.

This approach can be used for models with topology optimization results from the Mechanical application, which can be brought in and converted to a SubD hybrid model in the Discovery application. These models usually have groups/facet regions set on import.

> **Note:** To create facet regions in the Discovery application, select the facets and then right-click and select **Add Facet Region** from the context menu.

1. Import or create a faceted model.
2. Use the tools in the **Cleanup** group in the **Facets** tab to ensure the model is smooth and watertight.
3. Select the **Auto Skin** tool in the **Reverse Engineering** group of the **Facets** tab.
4. The **Select Faceted Body** tool guide is active by default. Select the faceted body.
5. The **Select Solid Bodies** tool guide is activated. Select the solid bodies which match the facet regions.
6. Click **Complete** to convert the model to a SubD hybrid model.

   The facet regions are converted to a combination of traditional and SubD faces. The SubD hybrid model retains the traditional analytic faces such as planes or cylinders for regions that require precise topology. The remaining facets are converted to SubD faces which allow for flexible editing.

   See Subdivision Surface Tools on page 417 for more information on SubD tools.

8.8.1.3. Auto Skin Tool Guides and Options

Auto Skin Tool Guides

Select Faceted Body	The **Select Faceted Body** tool guide is active by default. Select the faceted body to be converted to CAD.
Select Solid Bodies	The **Select Solid Bodies** tool guide will be activated if the mesh selected has groups or facet regions. Select solid bodies which will be used to generate CAD faces which match the facet regions. The CAD faces will be trimmed to the facet regions.
Complete	Converts the faceted model to CAD geometry.

Auto Skin Tool Options

The following options are available:

Create CAD	Use the **Create CAD** option to create the CAD geometry from the faceted model.
Create SubD hybrid	Use the **Create SubD hybrid** option to create the SubD hybrid model from faceted models with topology optimization results.

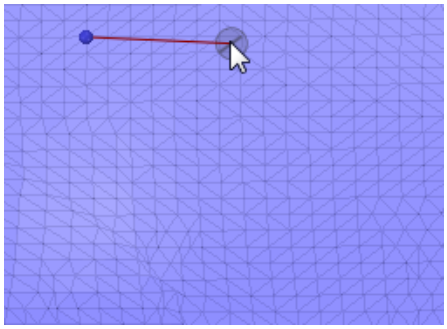
8.8.2. Skin Surface Tool

The **Skin Surface** tool helps you reverse-engineer a surface model from faceted data. For example, the starting point of your design may be a clay model which is scanned and output to one of the supported faceted data types. After you import the faceted data as a facet mesh, use the **Skin Surface** tool to sketch and create surface patches on the facets, and stitch them together into a surface body.

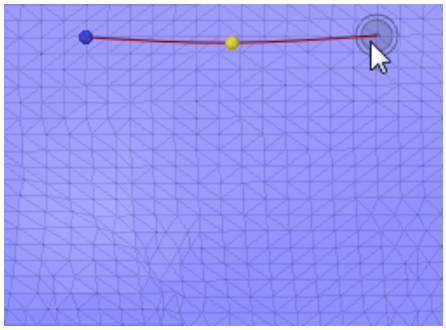
This section contains the following topics:

8.8.2.1. Sketching a Surface Patch

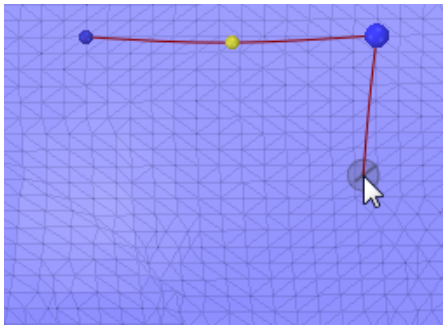
1. Import or create a facet mesh.
2. Select the **Skin Surface** tool in the **Reverse Engineering** group of the **Tools** tab.
3. Single-click the mesh where you want to start the sketch.



4. Sketch the desired shape as follows:
   - Single-click to create internal edge points, which can later be moved to adjust the shape of the edge

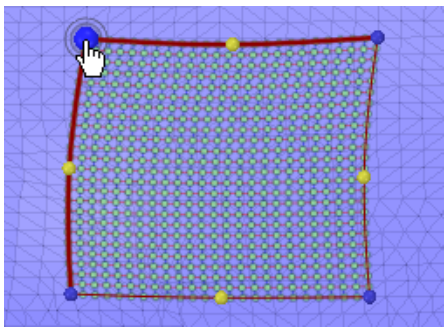


   - Double-click to end the edge and create a corner

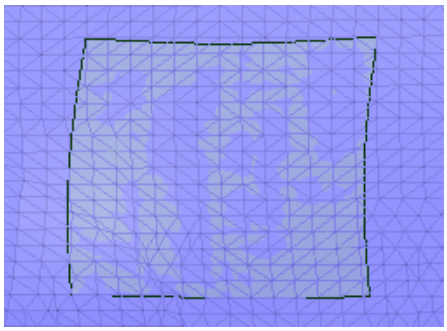


5. Continue creating edges and corners.

6. Close the shape by single-clicking on the start point.



7. The surface is shown in preview and you can drag any of the sketched points to adjust the shape of the sketch.
8. If you need to start over, simply hit the Escape key and the current sketch will be removed.
9. Click **Complete** to create the surface.



8.8.2.2. Skin Surface Tool Guides and Options

Skin Surface Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Boundary	The **Select Boundary** tool guide is active by default. When this tool guide is active, you sketch surface patch boundaries on the facets.
Select Geometry	The **Select Geometry** tool guide lets you choose planes or existing patch edges to create boundary loops for surface patches.

Skin Surface Tool Options

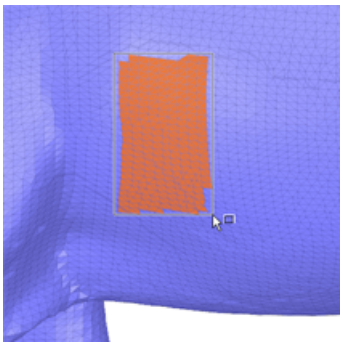
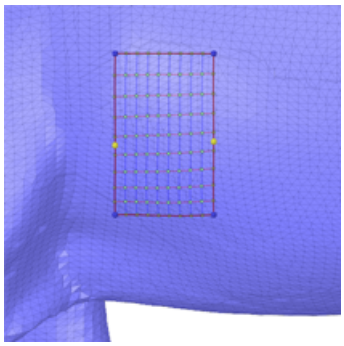
The following options are available:

Samples	The value entered determines the number of control points on the control curves for the surface. By default, the number of sample points is fixed and cannot be changed. If you switch on **Full preview**, you can enter a new number of sample points. In **Full preview** mode you can enter a new number of points directly, or use the dropdown slider.

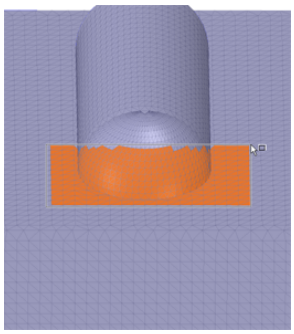
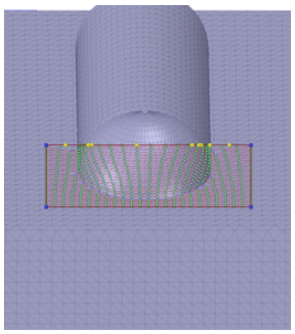
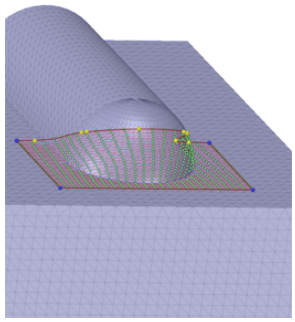
Full preview	When you switch on **Full preview**, you can control the number of sample points used in the surface. In **Full preview** mode, when you change the number of sample points, the preview dynamically updates. The speed of the dynamic updates is affected by the number of sample points. Fewer points leads to faster updates.
Tolerance	The tolerance controls the number of control points and the control curves created, and therefore how closely the patches match the faceted body. Lower tolerance values will result in more points and generate more precise curves on the patches.

8.8.2.3. Using Box-select

If you want to work with rectangular patches, use box-select instead of sketching them freehand. With the **Select Boundary** tool guide active, simply box-select the area where you want the sketch.

Box-select facets.	*Sketch is made from the box-selection*
	

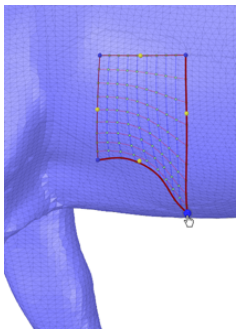
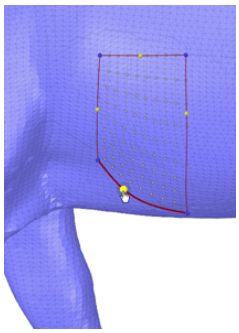
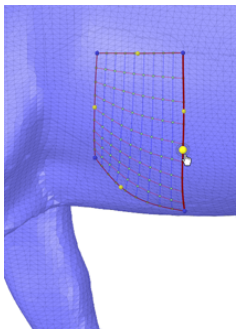
Notice that control points are automatically added in areas with curvature. This can be seen more dramatically in the images below.

Box-select across a curved area.	*Control points are auto-created.*	*Change view to see the curvature.*
		

8.8.2.4. Editing Curvature in a Sketch

While in Preview, you can modify the shape of a sketch by dragging the corners and control points. Control Points are the internal points on edges that allow you to adjust the edge's curvature. You can also add control points to give you more curvature control on edges.

The images below show a sketch being adjusted to better fit to the contours of the model and show a control point being added for further adjustments.

Dragging a corner.	*Dragging a control point.*	*Adding a control point*
		

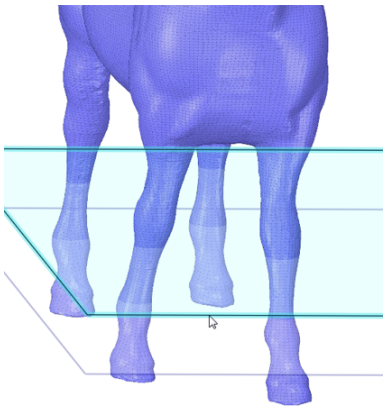
8.8.2.5. Periodic Surfaces

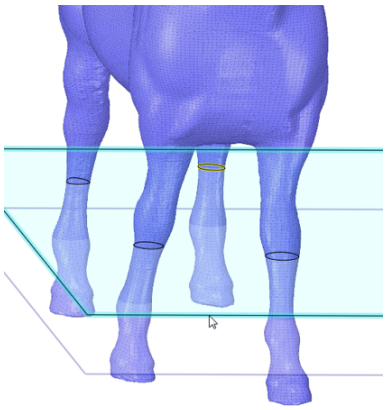
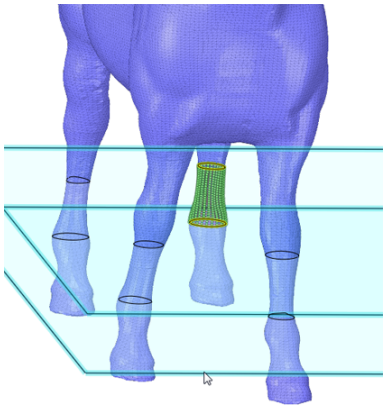
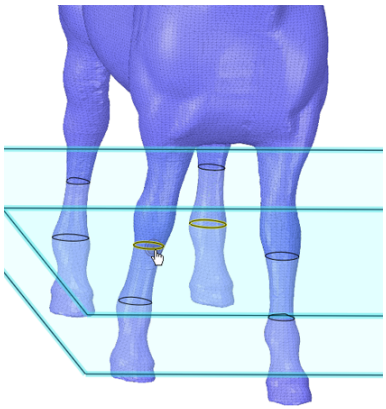
Surfaces that wrap around cylindrical, and roughly cylindrical, volumes are called **periodic surfaces**. They look like a rectangular patch wrapped around the volume.

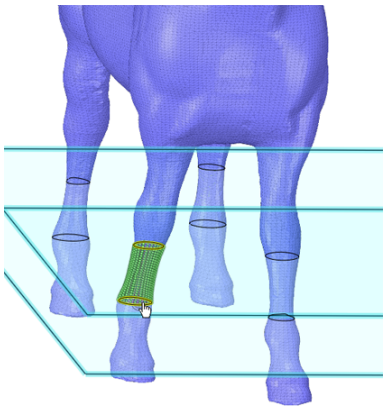
Sketching these by hand can be tedious because you need to rotate the model to expose more facets on which to sketch.

The **Select Geometry** tool guide streamlines the process by letting you select planes that create sections through the volume. The sections are then used to automatically generate sketch curves. Selecting two planes some distance apart automatically, creates the periodic surface.

In the following example, there are multiple periodic surfaces possible. Skin Surface previews one by default. If it is not the one you want, you simply select the edges of the one you do want.

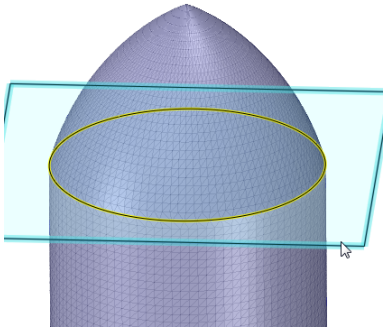
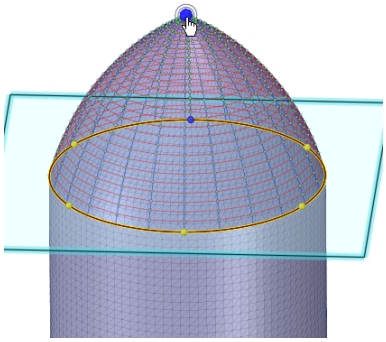
Two planes intersect the four legs.	

Selecting the first plane displays sections on all four legs.	
Selecting the second plane displays the other four sections and the default surface.	
To create a different surface, select the first section of the surface you want.	

Select the second section to display the surface preview. Click Complete to create the surface.	

Conical Patches:

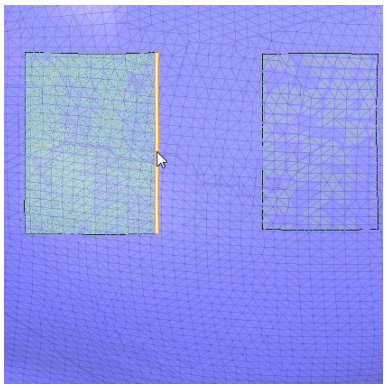
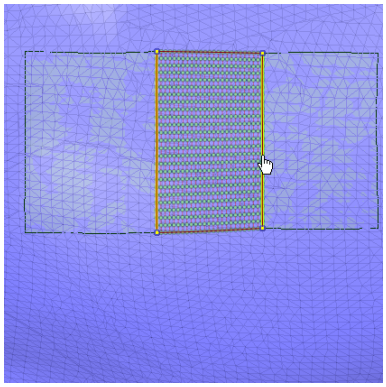
Periodic surfaces can also be conical. The following example shows how to create a conical surface patch.

You can either sketch a periodic edge manually, or use the Select Geometry tool guide and a plane to create a periodic edge on the section.	
Click the apex of the cone to preview the surface. Click Complete to create the surface.	

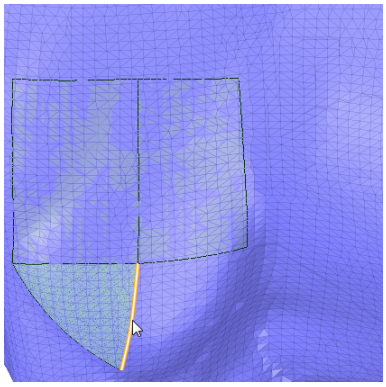
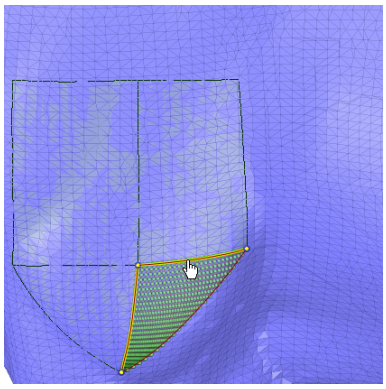
8.8.2.6. "Stitching" Surfaces Examples

Existing skin surfaces can be "stitched" into a single surface patch. Examples are shown below.

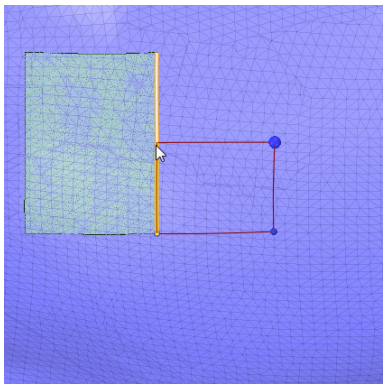
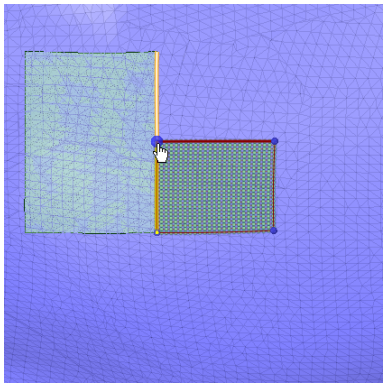
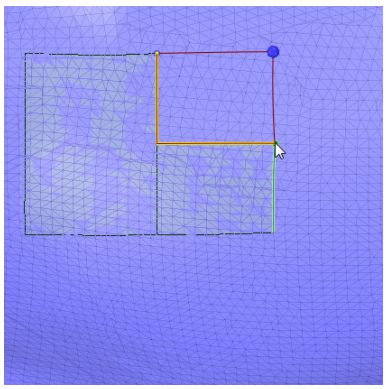
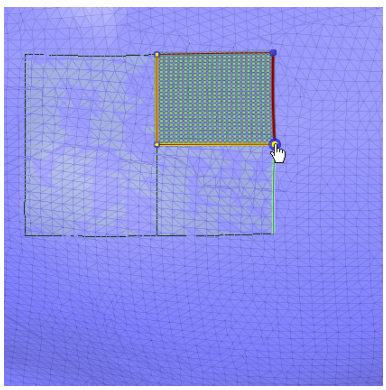
Use the Select Geometry Tool Guide to Select Rectangular Edges:

You can bridge the gap between rectangular patches using the Select Geometry tool guide. pre-highlight and select one edge.	
Pre-highlight and select the second edge to preview the surface. Click Complete to create the surface.	

Use the Select Geometry Tool Guide to Select Triangular Edges:

The same procedure works for triangular patches. Pre-highlight and select the first edge.	
Pre-highlight and select the second edge to preview the surface. Click Complete to create the surface.	

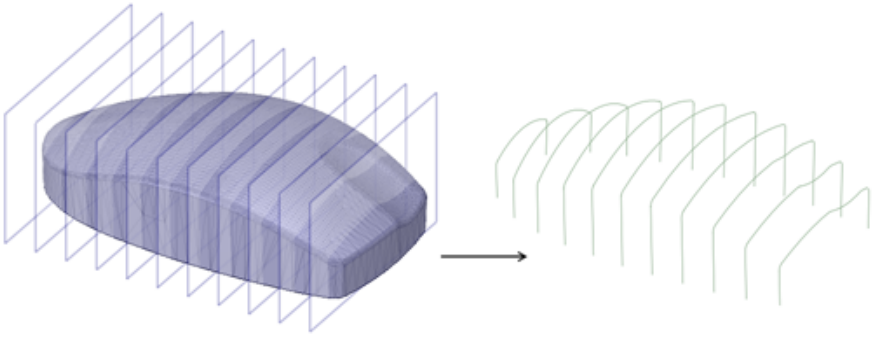
Snap to Existing Patch Edges and Points:

Withe the Select Boundary tool guide active, you can snap to existing surfaces. Here, the sketch started by snapping to the bottom corner. The color and size of the start point indicate that it is snapped to the lower right corner of the existing surface. The sketch then proceeds in a counterclockwise direction. The end of the third edge snaps to the existing edge, which is highlighted to indicate it will be used. The fourth edge is auto-completed by the tool and is shown on top of the snapped edge.	
A single click completes the third and fourth edges and the surface previews. Click Complete to create the surface.	
The next sketch begins at the upper right corner of the first surface. Like the previous sketch, the color and size of the start point indicate that it is snapped. The sketch proceeds in a clockwise direction. When the second edge reaches the upper right corner of the previous surface, the corner and edge of that surface highlight to indicate they will be used. Two edges can auto-complete to complete the loop. They are highlighted on top of the existing edges that will be used.	
A single click completes the second, third, and fourth edges and the surface previews. Click Complete to create the surface.	

8.8.3. Extract Curves Tool

The **Extract Curves** tool creates curves through design mesh cross sections.

Curves are created on multiple design mesh sections.



This section contains the following topics:

8.8.3.1. Creating Edges with Extract Curves

1. Open the **Facets** tab and click **Extract Curves** in the **Reverse Engineering** group.
2. The **Select Section Planes** tool guide is active by default.
3. Select one or more planes that intersect the mesh.
4. Click **Complete** to create the curves.

8.8.3.2. Extract Curves Tool Guides and Options

Extract Curves Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Section Planes	The **Select Section Planes** tool guide is active by default. Use it to select one or more planes that intersect the faceted body.
Select Curve	Use the **Select Curve** tool guide to select a new plane intersecting the faceted body.

Extract Curves Tool Options

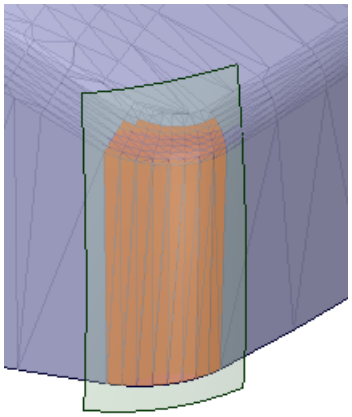
The following options are available:

Fit curves	Use **Fit curves** to create lines or arcs based on faceted models. From here, you can project the curves to a sketch to rebuild a solid model from a faceted model.
Tolerance	When using the **Fit curves** option, the **Tolerance** setting adjusts how closely the created curves match the faceted body. Lower numbers will generally use more lines or arcs, generating a more precise curve.
Auto-merge	Merge lines and arcs to form spline curves.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

8.8.4. Fit Spline Tool

The **Fit Spline** tool allows you to create a "best fit" spline surface through selected mesh facets.



This section contains the following topics:

8.8.4.1. Fitting a Spline Surface on a Mesh

1. Select mesh facets individually, or use box-selection.
2. Choose **Fit Spline** from the Reverse Engineering group of the Tools tab.
3. The surface is created (see example above).

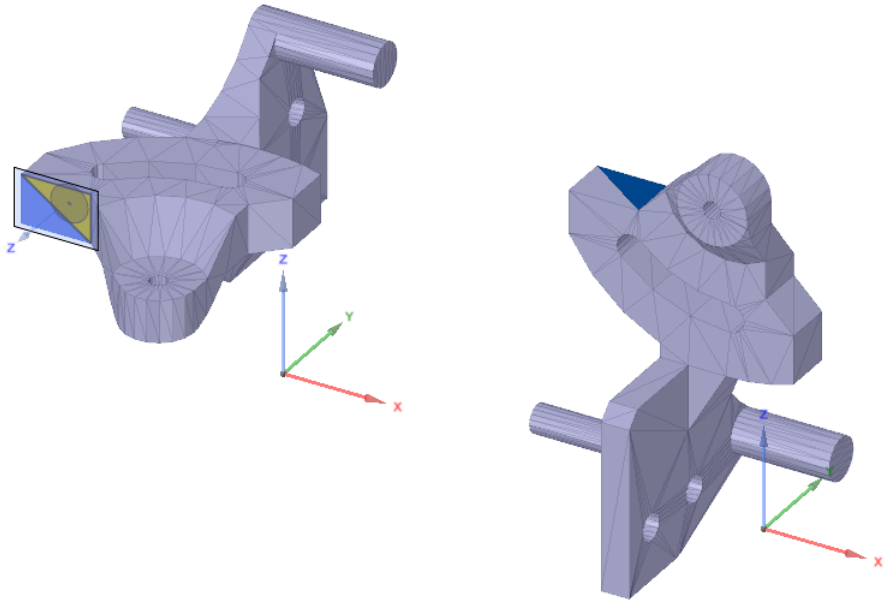
8.8.5. Orient Facets Tool

The **Orient Facets** tool in the **Reverse Engineering** group of the **Facets** tab allows you to select roughly planar areas of triangles or cylindrical areas of triangles to align to the World Origin axes.

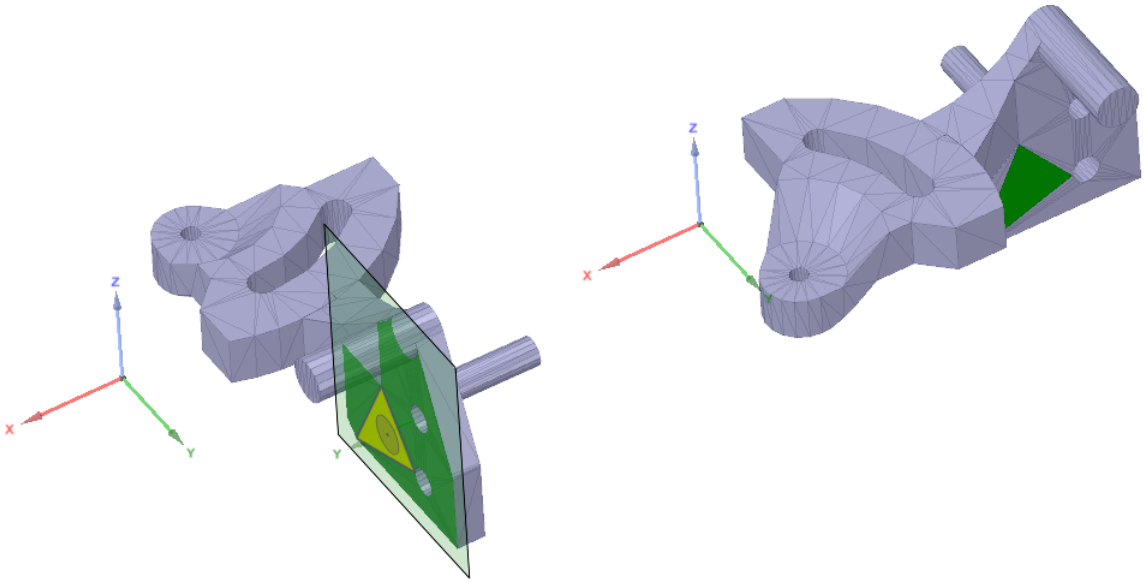
This section contains the following topics:

8.8.5.1. Orienting a Faceted Body

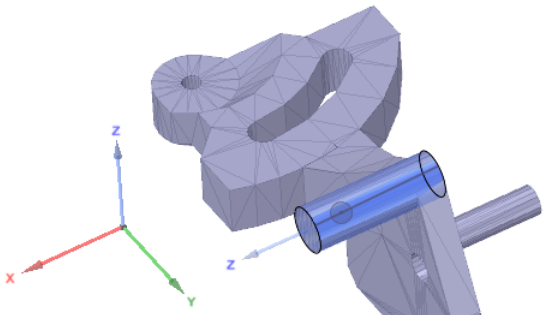
1. Open the **Facets** tab and click **Orient Facets** in the Reverse Engineering group of the **Facets** tab.
2. Hover over the triangles to see a preview of the axis it would align to.
3. Select the highlighted triangle.
4. The facet aligns as previewed. The selected triangle is shown in blue. Its normal is aligned to the Z-axis.

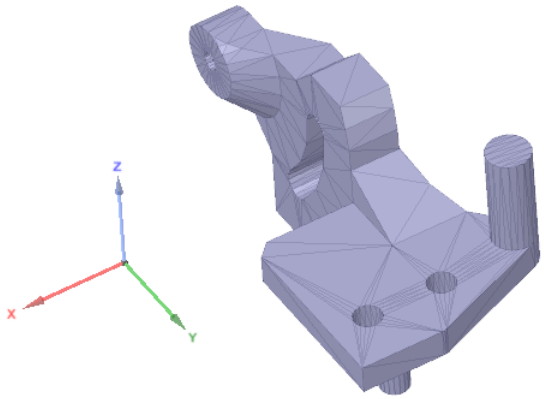


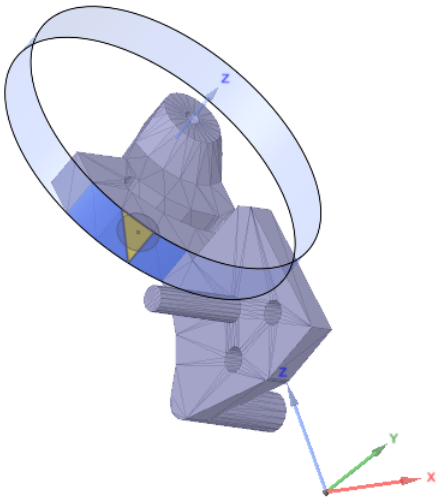
5. With each selection, the align-to axis alternates between Z and Y. In the next example, the normal of the selected triangle (shown in green) aligns to the Y-axis.

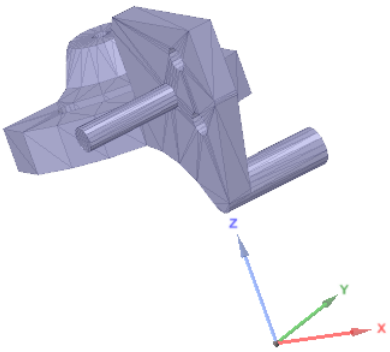


As you hover over triangles, cylindrical areas are detected. The following examples show cylindrical areas being aligned to the Z-axis.


Z
X
Y
Z


Z
X
Y


Z
Z
Y
X


Z
Y
X

9: Detailing

You can detail your designs to communicate with others or to submit your designs for review with the tools on the Detail tab.

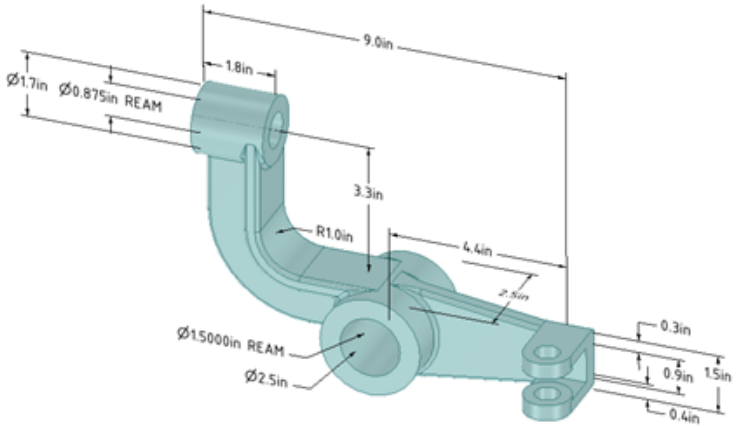
Press **Esc**, then **S** to end any detailing action and return to the **Select** tool.

This section contains the following topics:

9.1. Dimension Tool

Use the **Dimension** tool to add a measurement to your design, drawing sheet, or 3D markup.

An annotation plane cannot be moved to a sub-component after you add dimensions because the references would be lost.



This section contains the following topics:

9.1.1. Creating a Dimension Annotation

1. Click the arrow under the **Dimension** tool and select **Dimension**.
2. If you are creating a dimension in 3D, click a face to create an annotation plane on which to place the dimension.

   Mouse over the faces of your design to preview the eligible annotation planes. (In Sketch and Section mode, the sketch grid defines the annotation plane.) If multiple objects occur at your cursor location, use the scroll wheel or arrow keys to highlight each one.

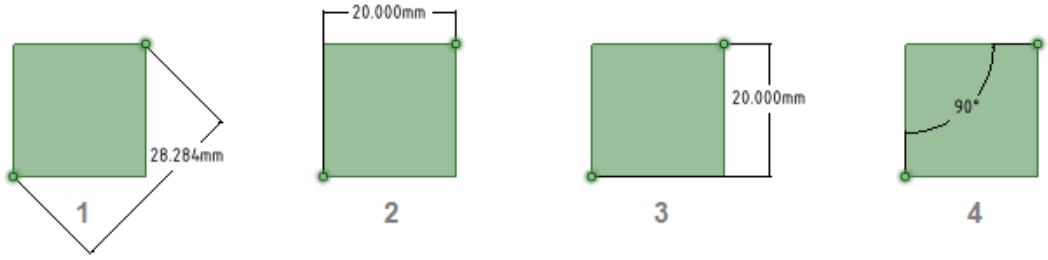
   To create an annotation plane for a cylindrical face, select the cylinder's axis.

   If you need to change the annotation plane, right-click and click **Select New Annotation Plane** from the context menu and select a new annotation plane.
3. Click an edge or face.

   Where you click a circle determines whether you will measure from the circle's center, near, or far edge. To select the center click the top, bottom, left, or right side of the circle.
4. Mouse over your design to preview the possible dimensions.
5. Click a second object if you want to dimension between two objects.

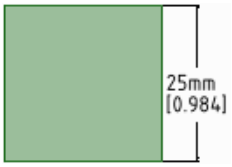
6. (Optional) Select a dimension orientation in the Options panel. You can also select the orientation for the first and second reference.
7. Click to create the dimension.

9.1.2. Creating Dimension Annotations Examples

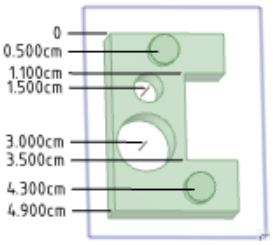


Annotations with orientation changed in the Options panel

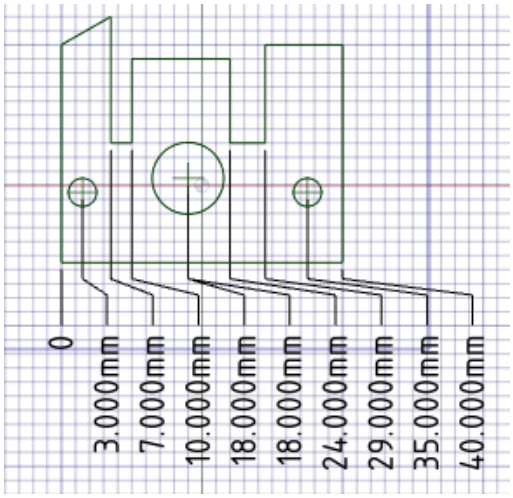
1. **Dimension Orientation** set to Aligned
2. **Dimension Orientation** set to Horizontal
3. **Dimension Orientation** set to Vertical
4. **1st Reference Orientation** set to Horizontal, **2nd Reference Orientation** set to Vertical



Ordinate dimension annotations



Ordinate dimensions for a planar face



Automatic jog points with closely spaced ordinate dimensions

9.1.3. Dimensioning Between Virtual Points

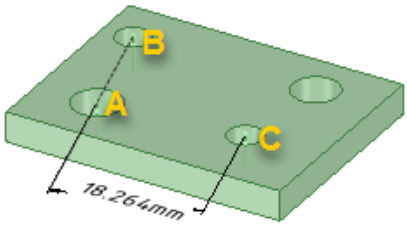
You can create dimensions from the line between two points or the mid-plane between two lines, even when that line doesn't exist as an edge.

1. Click **Dimension** in the Annotation group on the Detail tab .
2. Select a point (A) as your starting point.
3. Hold **Ctrl** and select a second point (B).

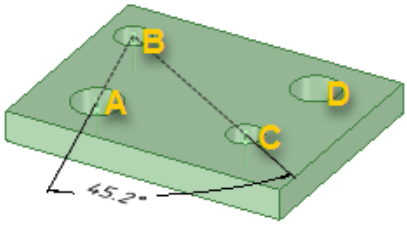
   This creates a virtual line between points A and B, which will be the baseline of your dimension.

4. Select a third point (C).

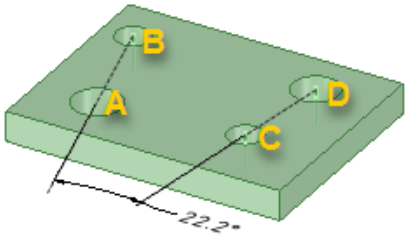
   Now you have a dimension between line AB and point C:



   If you hold **Ctrl** when you select point C, the dimension is created between line AB and line BC:



5. (Optional) If you select point C without holding **Ctrl**, and hold **Ctrl** to select a fourth point (D), the dimension is created between line AB and line CD:



6. Click anywhere in the annotation plane to place the dimension.

This section contains the following topics:

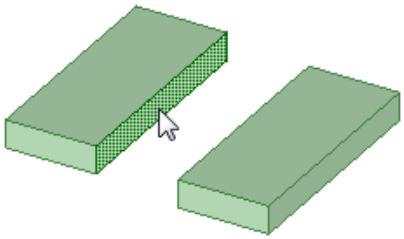
9.1.3.1. Dimensioning to a Virtual Sharp

1. Click **Dimension** in the Annotation group on the Detail tab .
2. Hold **Ctrl** and select a line.
3. Hold **Ctrl** and select a converging line.

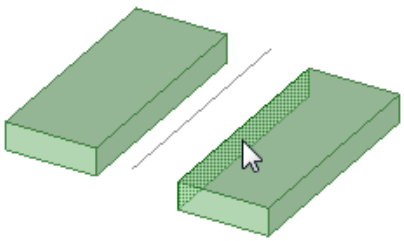
   The head of the note leader is attached to the virtual sharp. You can also drag the virtual sharp's end point, draw another leader to the virtual sharp, create virtual sharps in cross-section, for rounds, and between an angled and straight edge.
4. (Optional) To end the dimension at another virtual sharp,
   a) Hold **Ctrl** and select a line.
   b) Hold **Ctrl** and select a converging line.
5. Click anywhere in the annotation plane to place the dimension.

9.1.3.2. Dimensioning to a Mid-plane Line

1. Click **Dimension** in the Annotation group on the Detail tab .
2. Define the mid-plane line:
   a) Select a face or edge:



   b) Hold **Ctrl** and select a second face or edge:



   The mid-plane line is displayed as a gray line, and the beginning of the dimension is attached to this line.
3. Select the face, edge, or point you want to dimension to.

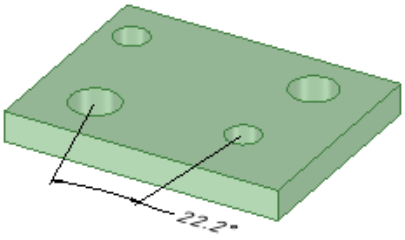
The end of the dimension is attached to the face, edge, or point.

4. Click to place the dimension annotation.

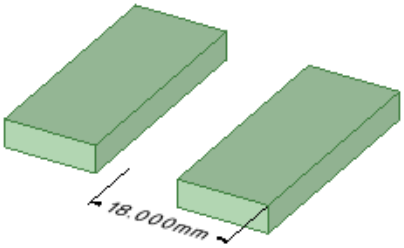
> **Note:** You can reverse these steps to begin the dimension with an existing line or point and end the dimension with a mid-plane line. You can also dimension between two mid-plane lines.

9.1.3.3. Examples

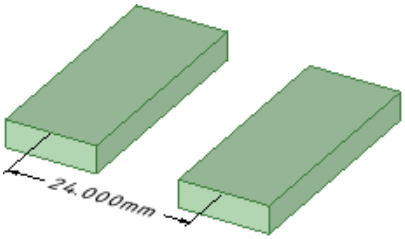
A dimension between virtual points.



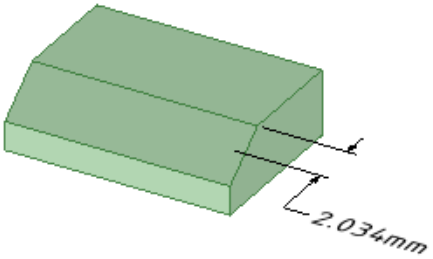
A dimension from a mid-plane line to an edge.



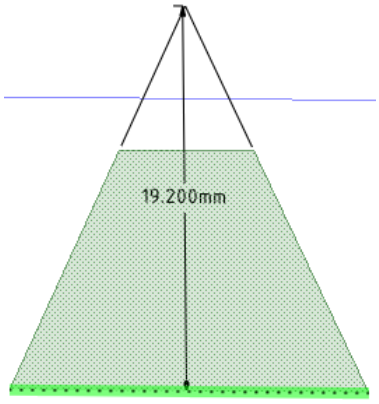
A dimension between two mid-plane lines.



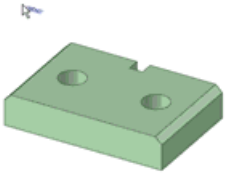
A dimension to the virtual sharp of a chamfered edge.



A virtual sharp.



9.2. Note Tool



Notes are shown in the model tree. Like curves, **Notes** can live in datum planes or drawing sheets. Unlike curves, they do not live directly in a part. So, in a part, you will not see a Notes folder.

This section contains the following topics:

9.2.1. Creating a Note

1. Select the **Note** tool from the Annotation ribbon group in the Detail tab.

   Mouse over the faces of your design to preview the eligible annotation planes. (In Sketch and Section mode, the sketch grid defines the annotation plane.)
2. Click a face to create the plane on which to place the note.

   To create an annotation plane for a cylindrical face, select the cylinder's axis.

   If you need to change the annotation plane, right-click and click **Select New Annotation Plane** from the context menu and select a new annotation plane.
3. Click to place the note on the plane.
4. Enter the text of the note.
   - Click Ω in the mini-toolbar to insert a symbol into your note at the cursor location.
   - Click in the mini-toolbar to insert a dynamic field. Dynamic fields include current values from a variety of properties.
   - You can format the note text, and enter the text by cutting, copying, and pasting the text from other notes or dimension annotations.
   - The application recognizes many fonts stored on your computer. If you paste text from outside of the Discovery application, and the font is not recognized, the default font will be used.
5. Adjust the orientation of the note by dragging the rotation handles.

9.2.2. Creating a Note Field

Field codes can be included within a note. Because the text in field codes is variable text (the text within a field is tied to code), when you change a field code in your note or drawing sheet, for example, a date format, the field automatically updates.

Some field codes are bi-directional. For example, adding the **Scale** field code to a drawing sheet enables you to change the scale by editing the note, or, change the scale by changing the value from the **Scale** dropdown in the **Sheet Setup** group in the **Detail** tab.

1. Click within the text of the note and place the cursor where you want the field to appear.
2. Right-click to display the mini-toolbar.
3. Click in the mini-toolbar to display the **Insert Field** window.

   The **Fields** tab displays the properties available for insertion. (Document properties are those that appear in the Properties panel when you click the top-level design in the model tree.)
4. Select a value from the category dropdown to filter the properties displayed in the **Fields** list.

   If you click **Selected Object**, you can click any object in the Design window or model tree to make its properties available.

   If you select **Formula**, you can enter an expression, and include any numeric fields within the expression.
5. Click a property in the **Fields** list.
6. Click the **Format** tab to format the text within the field.

   The formatting options are based on the type of the property value. For example, strings can be formatted with upper case, lower case, initial capitals, or title case.
7. Click **OK** to insert the formatted, dynamic field into the note at the cursor location.

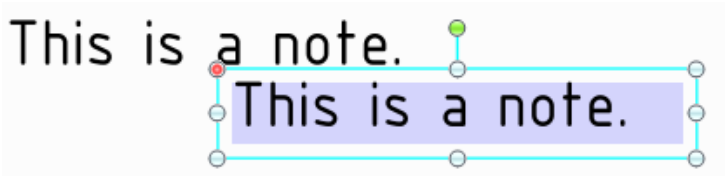
   If the field is empty, check to make sure that the property you selected has a value by selecting the appropriate object and viewing the Properties panel.

9.2.3. Copying a Note

Ctrl+drag a note to copy it.

You can also do the following:

- Click a note.
- The note displays inside the text box border.
- Click the border.
- Press Ctrl+c
- Press Ctrl+v
- The note copy displays below and to the right of the original note.



- If you paste the note on a different sheet, it will paste in the same location as in the original sheet. It does not paste at the cursor location.
- If you are zoomed in, the note will paste in the center of the view. If there is another note at that location, the new note will paste below and to the right of that note.

9.2.4. Editing a Note

1. Select the note to move, size, or rotate it.

   If you select a single note, you can edit, resize, and rotate the note. If you select multiple notes, you can only move or change formatting.

   To move the note box, mouse over the edge of the box until the cursor changes to , then drag the note.

   To size the box containing the note, drag the handles of the note box (the white circles).

   To rotate the note, drag the rotation handle (the green circle). Press **Shift** to snap to angular increments.

2. Select the text of the note to reformat it.

   Click a field to edit it. Changing the value of a string also changes it in the Properties panel.

3. Modify the note's properties in the Properties panel.

   - Circular Text
     - **Type -**

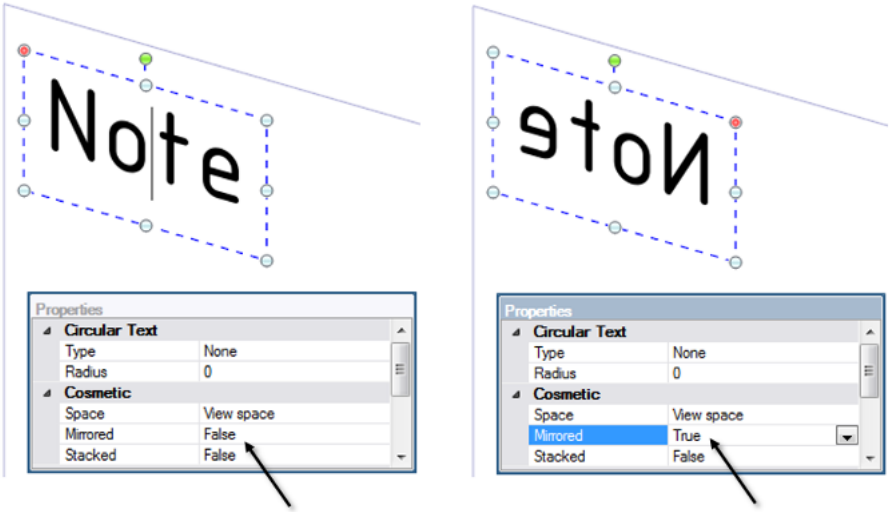
       Choose from:

       - **None**
       - **Clockwise**
       - **Counterclockwise**

     - **Radius -** Enter a radius for the note circle.

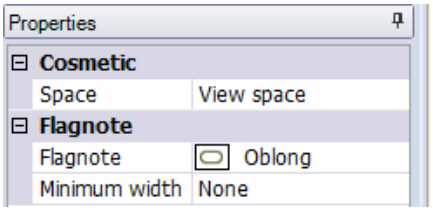
   - Cosmetic
     - **Space** - Choose from:
       - **Model Space** sizes the text based on the actual measurements of the objects in your design.
       - **View Space** sizes the text based on the view of the design in the Design window.

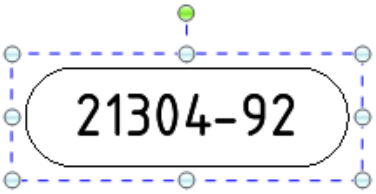
     - **Mirrored** - Set this property to True to display the text as if it is seen in a mirror. Notes seen from behind can appear 'backwards,' which can be confused as 'mirrored.' Setting this property confirms the Note is mirrored intentionally.



     - **Stacked** - Set this property to True to stack the text vertically instead of horizontally.

- Flagnote
  - **Flagnote** is a dropdown menu of border shapes to put around the text.
  - **Minimum width** prevents the border from resizing automatically to fit the content of the note.





- Position

  Anchor influences the position and size of a note within a plane. Where you anchor a note determines the direction that the text will fill the note. For example, if you select the Left Top position, as you type, the expanded text box flows from left to right and top to bottom. The choices are:

  - Left Top
  - Left Bottom
  - Right Bottom
  - Right Top
  - Left Center
  - Right Center
  - Top Center
  - Bottom Center
  - Center

    When moving a note or circular note, the note's anchor (or circular note's text center) now snaps to an edge, a sketch point, or an axis. Select the Move tool, and then select the note's red anchor and drag it to an edge, sketch point, or axis. The red anchor will snap to the note's anchor location that is closest to the drag point. The Anchor position updates in the Position group of the Properties panel as you move the anchor to a new location.

9.2.5. Fitting Text Within a Note Box

- RMB Click the note and open the **Autofit** dropdown menu.
- Choose one of the following options:
  - **Do not autofit:** The text box adjusts to the size of the text and grows as you type. There is no blank space around the note and making the text larger or smaller adjusts the box accordingly.
  - **Resize text height on overflow:** The text always fits the width of the box. If you make the box wider, text from the second line will move up to the first line.
  - **Shrink text on overflow:** The text adjusts uniformly (width and height) and scales to fit in the text box.
  - **Shrink text horizontally on overflow:** The width of the text changes but the height remains the same.

9.2.6. Adjusting the Text Width

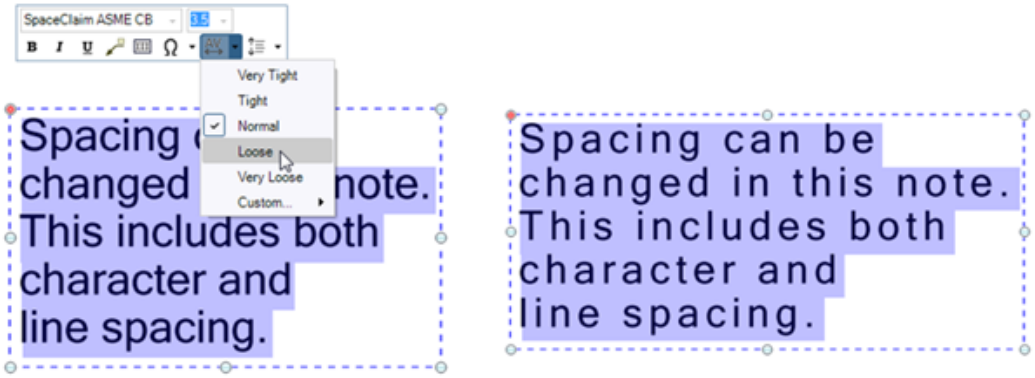
The Font group in the Detail tab has tools for increasing and decreasing text width.

- Click the **Width Factor** dropdown menu and select from the width factor options.
- Click the **Increase Width Factor** button to increase the width in 0.1 increments.
- Click the **Decrease Width Factor** button to decrease the width in 0.1 increments.

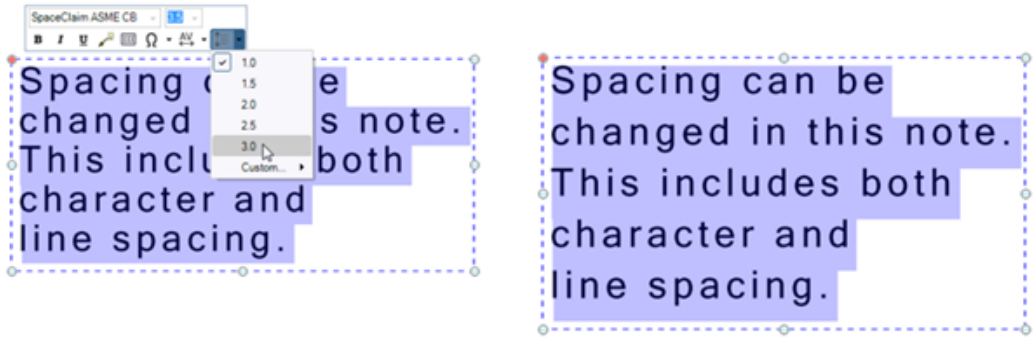
9.2.7. Adjusting Line and Character Spacing

Character and Line spacing can be changed in Notes.

- Click the character spacing dropdown menu and select from the character spacing options.



- Click the line spacing dropdown menu and select from the line spacing options.



9.2.8. Aligning Multiple Notes

1. Select multiple notes.
2. In the RMB menu, choose from the following:
   - **Align Lefts**: Align the left side of each note box.
   - **Align Centers**: Vertically align the center of each note box.
   - **Align Rights**: Align the right side of each note box.
   - **Align Top Lines**: Align the bottom of the first line of text in each note box.
   - **Align Middles**: Horizontally align the middle of each note box.
   - **Align Bottom Lines**: Align the bottom of the last line of text in each note box.

- **Make Same width**: Make each note box the same width based on the first note selected. Some text may be scaled accordingly.
- **Make Same Height**: Make each note box the same height based on the first note selected. Some text may be scaled accordingly.
- **Make Same Size**: Make each note box the same size (height and width) based on the first note selected. Some text may be scaled accordingly.
- **Distribute Horizontally**: The space between each note box is distributed evenly horizontally.
- **Distribute Vertically**: The space between each note box is distributed evenly vertically.
- **Remove Horizontal Spacing**: The even horizontal spacing is removed and any significant overlap is equalized.
- **Remove Vertical Spacing**: The even vertical spacing is removed and any significant overlap is equalized.

Any notes rotated differently are not affected by the above commands.

9.2.9. Attaching and Detaching Notes to Other Notes

You can attach notes to other notes using the **Attach Notes** option in the RMB menu. The first note selected is the parent note (highlighted in blue) and any other notes you select become child notes (highlighted in dark blue).

1. Select two or more notes
2. RMB click
3. Select **Attach Notes**
4. To detach a note RMB select it and choose the **Detach** option.

When notes are attached, moving the parent note moves all the notes as one. However, moving a child note will move that note individually.

You can attach a note to the note fields of objects created with tools in the Detail tab. For example, Surface Finish symbols, Welding symbols, Datum Targets, Datum symbols, notes within a Symbol, and Dimensions are some tools that have note fields. Any notes can be attached to any other notes. Dimensions, however, can only be parent notes since they are attached to the model.

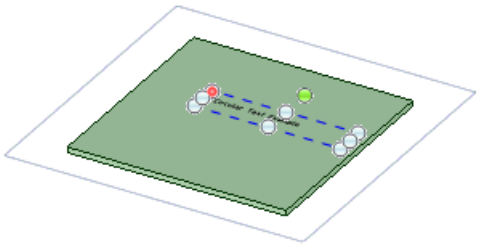
9.2.10. Displaying the Text of an Annotation and Hiding the Plane

1. Create two layers, one for notes, and one for the annotation planes.
2. Place the note on one layer and the annotation plane on another layer.
3. Turn off the visibility of the layer that contains the annotation plane.

9.2.11. Creating Circular Notes

Use the **Note** tool to create and edit circular Note text. This text type can orient either clockwise (CW) or counterclockwise (CCW). The circle center is indicated by a cross-hair marker. The circle radius displays as a visual guideline. Use the size handle to size the circle. You can adjust the orientation of the note by dragging the rotation handle. Enter or change new radius settings in the Options panel.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.



With circular text you can create:

- Single line text
- Multi-line text. Type the first line of text, and then press **Enter** to create an additional line of text.
- Formatted single or multiple line text. Use the mini-toolbar to format circular note text. You can change the font, font size, and style of all or some of the text; however, you cannot underline circular text.
- A note with a True Type or a Microsoft Windows 3.1 font (Modern, Roman, and Script). True Type fonts may be used when etching or laser-cutting in sheet metal, where the width of the character is determined by the width and/or focus of a laser beam.

9.2.11.1. Creating a Circular Note

1. Select the **Note** tool from the Annotation ribbon group in the Detail tab.

   Mouse over the faces of your design to preview the eligible annotation planes. (In Sketch and Section mode, the sketch grid defines the annotation plane.)

2. Click a face to create the plane on which to place the note.

   To create an annotation plane for a cylindrical face, select the cylinder's axis.

   If you need to change the annotation plane, right-click and click **Select New Annotation Plane** from the context menu and select a new annotation plane.

3. Click to place the note on the plane.
4. Enter the text of the note.
5. In the Properties panel, select **Clockwise** or **Counterclockwise** from the **Type** dropdown.
6. Click the fact to update the note.

Once you create the circular note text, you can:

- Click Ω in the mini-toolbar to insert a symbol into your note at the cursor location.
- Click in the mini-toolbar to insert a dynamic field. Dynamic fields include current values from a variety of properties.
- Format the note text, and enter the text by cutting, copying, and pasting the text from other notes or dimension annotations.
- Position the note text relative to the note's anchor by using the **Anchor** dropdown in the Properties panel. Adjust the note's diameter by dragging the note's anchor.
- Adjust the orientation of the note by dragging the rotation handles. Drag the center cross-hair marker to move the text.
- Use leading spaces (inserted before text or within text) and trailing spaces (inserted after text) to position text around the circle radius as needed.
- Convert the text to lightweight text in sheet metal. Lightweight text is transparent, and is used to represent text objects in a virtual way (so that further operations on the part do not have to recalculate all the new graphics data

dealing with text). In lightweight mode, the text cutouts seen by the user are only recreated when the text is edited, as opposed to every time any other action is performed on the model.

- Move the note: Simply drag a red anchor point to snap to an edge, a sketch point, or an axis. The red anchor will snap to the note's anchor location that is closest to the drag point.

9.3. Adding Hyperlinks to Annotations

Hyperlinks allow access to more content in your notes and dimensions. Some examples are:

- Images
- Web pages
- Word documents

Once a hyperlink is established, you can hover over the annotation text to display the link in the status bar and Ctrl+Click to go to the link.

This section contains the following topics:

9.3.1. Adding a Hyperlink to a Note

1. Highlight the annotation text.
2. RMB click
3. Select the **Hyperlink** option.
4. The Insert Hyperlink dialog box opens.
5. Enter an existing file path or web page link. You can also click the **Browse for File** button or **Browse the Web** button.
6. Click the **OK** button.

9.3.2. Editing a Hyperlink

1. Highlight the annotation text.
2. RMB click
3. Select the **Edit Hyperlink** option.
4. Edit the link in the **Edit Hyperlink**dialog box.

9.3.3. Opening a Hyperlink

1. Highlight the annotation text.
2. RMB click
3. Select the **Open Hyperlink** option.
4. The link opens in the appropriate media.

You can also hover over the annotation text and Ctrl+Click to open the link in the appropriate media.

9.3.4. Removing a Hyperlink

1. Highlight the annotation text.

2. RMB click
3. Select the **Remove Hyperlink** option.

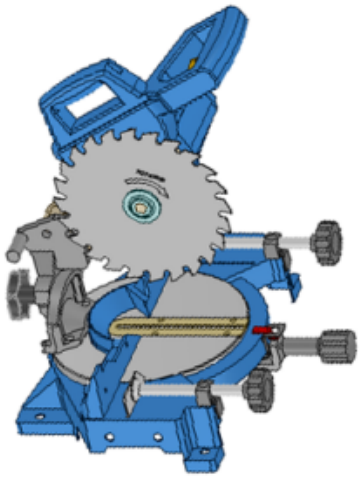
You can also edit the hyperlink and click the **Remove Hyperlink** button in the dialog box.

9.4. Bill of Materials (BOMs) Tool

You can insert a Bill of Materials (BOM) onto your design, drawing sheet, or 3D markup slide. BOM values are automatically populated for both internal and external components that are part of an assembly. The Discovery application only itemizes components. If the design only has solids in the model tree, they will not be included in the BOM. The QTY column in a design's BOM table shows the number of sub-components, not items, that are included in an assembly.

When a drawing includes a BOM table that contains data added with a newer service pack of the Discovery application than you are using, the new columns are displayed in red. This indicates that the data cannot be changed, and can only be viewed.

You can include X, Y, and Z dimensions of components in a BOM table. The extents shown are the exact extents of all solid and surface geometry in the component, for both visible and hidden solids and surfaces. Sketch curves, bitmaps, and other entities are not included in the extents. Use these dimensions to create a cut list of parts in an assembly of panels in the BOM, along with their number and dimensions. See the instructions for adding a column to a BOM table below.

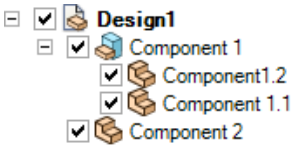


ITEM NO.	PART NUMBER	QTY
1	washer__blade.prt	2
2	blade.prt	1
3	housing__motor.prt	1
4	handle__motor.prt	1
5	trigger.prt	1
6	__casting	1
7	mount-1.prt	1
8	mount-2.prt	1
9	knob__miter-angle.prt	1
10	knob__clamp.prt	2
11	mount__clamp.prt	2
12	clamp.prt	2
13	clip__clamp.prt	2
14	platen.prt	1
15	guard__platen__upper.prt	1
16	spring-index.prt	1
17	bracket__index.prt	1
18	indicator.prt	1
19	knob__angle-set.prt	1
20	base.prt	1
21	backstop.prt	1

This section contains the following topics:

9.4.1. Adding a Bill of Materials to a Document

1. Click the arrow next to the **Bill of Materials** tool in the Annotation group on the Detail tab and select a type:
   - **Top level**: Creates a table that includes only parent components (or parts):



ITEM NO.	PART NUMBER	QTY
1	Component 1	1
2	Component 2	1

- **Parts only**: Creates a table that includes all child components:

ITEM NO.	PART NUMBER	QTY
1	Component 1.1	1
2	Component 2	1
3	Component1.2	1

- **Indented**: Creates a table that includes all components, with child components indented:

ITEM NO.	PART NUMBER	QTY
1	Component 1	1
1.1	Component 1.1	1
1.2	Component1.2	1
2	Component 2	1

The last type of BOM you created is remembered and will be automatically chosen if you click the BOM icon instead of the dropdown arrow.

2. (Optional) If you created a view of a component on the drawing sheet, select the component on the Options panel to create a BOM for just that component.
3. (Optional) Modify the Bill of Material's properties in the Properties panel.
   - The **Hidden components** option allows you to control whether or not hidden components will be included in the BOM.
   - There are many other options for controlling the cosmetics of the BOM.
4. Click to place the Bill of Materials on the design, drawing sheet, or 3D markup slide. The table appears in gray until you place it, and it changes to black after it is placed.

   To cancel, press the **Esc** key before you click to place the table.

   The BOM appears in the model tree.

   You can copy and paste a BOM, and you can copy the contents from a BOM table and paste them into Microsoft Excel.

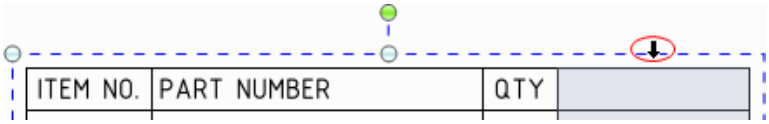
9.4.2. Editing a Bill of Materials

- When you change a value in a BOM table, the property is changed. You can also select the BOM and edit its properties in the Properties panel.
- Select a row or column, right-click, and select from the menu to Insert rows or columns.
- Select a row or column, right-click, and select from the menu to Delete rows or columns.
  - You can only delete rows that you insert
  - You cannot delete all columns. If there is one column left, it cannot be deleted.

- Select a row, right-click, and select **Split Table** from the menu to divide the BOM into two tables, for example, if necessary to fit on the drawing sheet.
- Re-size a table by dragging the circles at its corners or edges.
- Rotate a table by dragging the green circle at the top of the table.

9.4.3. Adding Columns to a BOM Table

1. Right-click an existing column and select **Insert** > **Insert Columns to the Left** or **Insert Columns to the Right**.
2. Select the new column by clicking above the first row of the column, like this:



3. Select a **Column Type** in the Properties panel.
4. Select a **Column Value** if you selected Document Property, Material Property, or Custom Property for the column type. The values in the column will be automatically filled-in.

   If you select Material Property for the column type and Mass or Volume for the column value, then the column data will only be filled-in when a material has been assigned to the object.

9.4.4. Exporting a Bill of Materials

1. Select a BOM table, then click its edge to select the entire table. The edge will change from a dashed line to a solid line when you have the entire table selected.
2. Right-click the table and select **Save Table As**.
3. Select HTML, XML, BOM Template (.scbomtemplate) for the **Save as type**.

If you choose 'BOM Template' as the type, all the formatting, header, table properties, and contents are saved to an .xml file which you can edit.

9.4.5. Using a BOM Template

1. Click the arrow on the BOM tool.
2. Select **Template** from the dropdown list.
3. Select the .scbomtemplate document and click the **Open** button.
4. Place the BOM table.

9.4.6. Sorting a Bill of Materials

You can sort the columns in a BOM table.

1. Select a column.
2. Choose **Sort by** column in the RMB menu.
3. The values are sorted numerically and then alphabetically in ascending order.
4. Sort the column a second time to sort in descending order.

You can manually edit any values in the BOM but any sorts have to be re-done to update the table.

Sorting works with indented and regular BOM's.

Sequential sorting is remembered across columns. Below is an example of two sequential sorts:

1. First sort items alphabetically by name.
2. Then sort numerically by Quantity.
3. Result:
   a. Rows are put in ascending order according to Quantity
   b. Items with duplicate quantities are then put in alphabetical order according to their names. In the image below, items are sorted by Quantity (1, 2, 3). Then all items with a Quantity of 1 are sorted alphabetically.

ITEM NO.	FIRST SORT	SECOND SORT
1	Blocks	1
1.1	Block 2	1
1.2	Block 3	1
1.3	Block 5	1
1.4	Block 6	1
1.5	Block 1	2
1.6	Block 4	3
2	Cylinders	1
2.1	Cylinder 2	1
2.2	Cylinder 3	1
2.3	Cylinder 5	1
2.4	Cylinder 6	1
2.5	Cylinder 1	2
2.6	Cylinder 4	3

9.4.7. Grouping Rows in a Bill of Materials

You can group rows in a BOM table based on a selected Custom Property.

1. Add a column to the BOM
2. Select the column and set the **Column Type** property to Custom Property
3. Set the **Column Value** property to be the Property that you want to group
4. Set the Group Using property to be the Property that you want to group
5. You can control the row numbering for the grouped rows using the **Start groups from** property
   - Number items sequentially
   - 102, 102, ...
   - 1001, 1002, ...
   - 10001, 10002, ...

Notes on row grouping:

- Top Level and Parts Only BOM's only (not Indented)
- Only one grouping is allowed per BOM
- Sorted by Custom Property Values in alphabetical order (inside groups, normal sorting order applies)

Example:

BOM with "Part Type" custom property column added

ITEM NO.	PART NUMBER	QTY	Part Type
1	Part 1	1	
2	Part 2	1	
3	Stock Part 4	1	STOCK
4	Stock Part 1	1	STOCK
5	Ordered Part 2	1	ORDER
6	Stock Part 3	1	STOCK
7	Ordered Part 3	1	ORDER
8	Stock Part 2	1	STOCK
9	Ordered Part 1	1	ORDER

Group Using property set to "Part Type" and default row numbering

ITEM NO.	PART NUMBER	QTY	Part Type
1	Part 1	1	
2	Part 2	1	
101	Ordered Part 2	1	ORDER
102	Ordered Part 3	1	ORDER
103	Ordered Part 1	1	ORDER
201	Stock Part 4	1	STOCK
202	Stock Part 1	1	STOCK
203	Stock Part 3	1	STOCK
204	Stock Part 2	1	STOCK

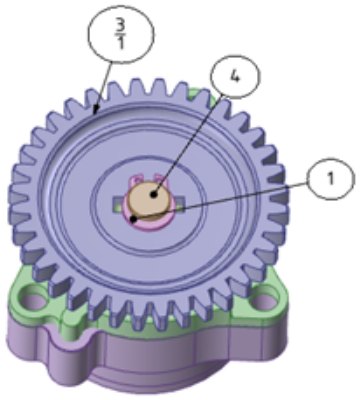
Start groups from property set to Number rows sequentially

ITEM NO.	PART NUMBER	QTY	Part Type
1	Part 1	1	
2	Part 2	1	
3	Ordered Part 2	1	ORDER
4	Ordered Part 3	1	ORDER
5	Ordered Part 1	1	ORDER
6	Stock Part 4	1	STOCK
7	Stock Part 1	1	STOCK
8	Stock Part 3	1	STOCK
9	Stock Part 2	1	STOCK

9.5. Balloon Tool

Use the **Balloon** tool to add BOM balloon notes. The content of the balloon updates when changes are made to the BOM table.

The document must contain a BOM table for the Balloon tool to be enabled.



This section contains the following topics:

9.5.1. Adding BOM Balloon Notes

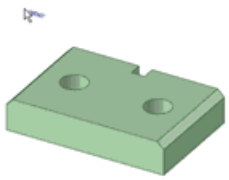
1. From the dropdown menu on the **Bill of Materials** tool in the Annotation ribbon group, select a type:
   - Item number
   - Part number
   - Item number and quantity
2. Click a face or edge in the drawing sheet or Design window to attach a balloon to the object.
3. Click again to place the balloon.

9.5.2. Automatically Adding BOM Balloon Notes to a View

1. Right-click a view.
2. Select **Autoballoon** and then **Item Number**, **Part Number**, or **Item Number and Quantity**.
   These menu options are only available when a view has a BOM.

9.6. Note Leader Tool

Use the **Note Leader** tool to create an arrow from your note.



This section contains the following topics:

9.6.1. Creating a Note Leader

1. Click **Note Leader** in the Annotation group on the Detail tab.
   Mouse over notes on the active annotation plane to see the possible ways to connect the note leader to the note.

2. Click a note's connection point to draw the first segment of the note leader.

   Mouse over your design to highlight the geometry to which you can attach the end of the note leader. You can also attach leaders to temporary objects.

3. For a segmented line, click to set each point of the note leader line.
4. End the note leader by clicking a vertex, edge, or face to attach the end of the note leader, or double-click to end the note leader at any point.

   The end of the note leader is an arrow unless you attach it to a face.

9.6.2. Snapping a Leader to 45° Increments

While creating a Leader, hold the Shift key to snap the Leader to the nearest 45° increment.

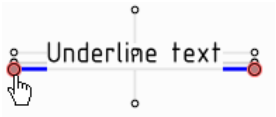
1. Click to set the start point of the Leader.
2. Start dragging to get near the desired orientation.
3. Press and hold the Shift key.
4. The Leader snaps to the nearest 45° increment.
5. Continue holding Shift down and dragging to set the length.
6. Click to set the endpoint of the Leader.

9.6.3. Underlining Note Text with a Leader

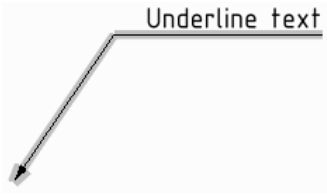
1. Create a Note
2. Enter the Leader tool.
3. Attach a Leader to the underline location.
4. Complete the Leader.
5. Select the Leader.
6. Select the **Underline** option from the context-menu.

   RMB Underline

   The text is underlined.



When you start the leader, hovering near the note text displays attachment points. The image above shows the underline location highlighted.



Select the leader and use the context menu to select Underlined. This toggles the leader to an underlined leader.

9.6.4. Inserting an All-Around Symbol

Right-click the note leader and select All Around.

9.6.5. Changing a Note Leader's Segments

Drag a segment of the note leader line to move it and its surrounding boundary points. Right-click the note leader and select **Add Jog Point** to create a new segment.

You can delete note leader line segments by deleting the jog points that border the segment.

9.6.6. Changing a Note Leader's Properties

1. Right-click the note leader and select **Properties**.
2. Modify the Arrow **Style**, **Length**, and **Width** values.
3. Set the **All Around** value to **True** to display an all-around symbol. Select **False** to hide it.

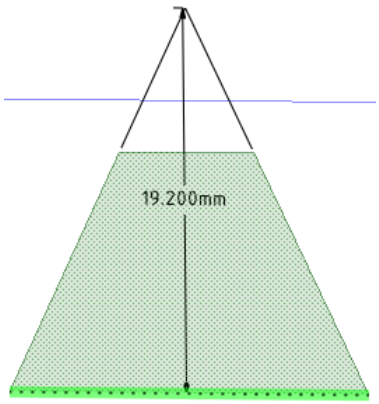
9.6.7. Attaching a Note Leader to a Virtual Sharp

1. Select the **Note Leader** tool from the Annotation ribbon group in the Detail tab.
2. Ctrl+click a line.
3. Ctrl+click a converging line.

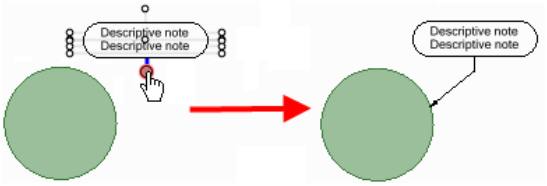
   The head of the note leader is attached to the virtual sharp. You can also drag the virtual sharp's end point, draw another leader to the virtual sharp, create virtual sharps in cross-section, for rounds, and between an angled and straight edge.

9.6.8. Note Leader Examples

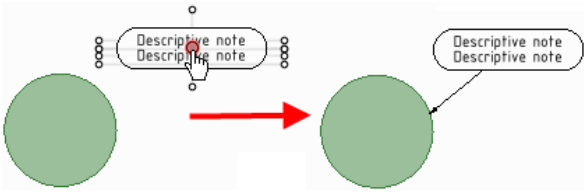
A virtual sharp.



Attaching to the bottom connection point of a note to create a leader with a jog or shoulder.



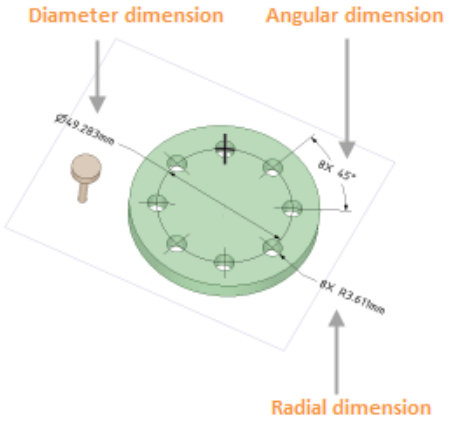
Attaching to the center connection point of a note to create a straight leader.



Use the center connection point to create circle flag notes or BOM balloon notes according to the JIS standard, without a jog or shoulder.

9.7. Bolt Circle Tool

- You can create a Bolt circle for either one- or two-dimension circular patterns.
- A Bolt circle displays in the **Design** tree as an Annotation Plane. You can deselect the Annotation Plane check box to hide the Bolt circle.
- Bolt circle information does not update if you modify the options for your pattern, such as Circular count or Angle.
- The count and angular dimension are separate and can be deleted individually.



This section contains the following topics:

9.7.1. Creating a Bolt Circle

1. Select the **Bolt Circle** tool from the Annotation ribbon group in the Detail tab.
2. In your circular pattern design, click the face of your design that holds the pattern member to create a datum plane on which to place your Bolt circle.

   The application creates a datum plane for the Bolt circle.
3. Select a pattern member.

The application automatically adds dimension call outs, arrows, and lines to the annotation plane. You can click an arrow or line to view or modify each dimension's values in the Properties panel.

9.8. Table Tool

You can place a table on an annotation plane using the **Table** tool in the Annotation ribbon group.

You can copy annotation dimensions from drawings and paste them into tables, and their values will be updated when the dimensions change.

This section contains the following topics:

9.8.1. Adding a Table

1. Add an Annotation plane or activate an existing plane.
2. Click the Table tool .
3. Drag to create the table with as many columns and rows as you need. The default cell size is determined by the font size in the Detail tab.

9.8.2. Table Editing Operations

- Select the table, then drag to select multiple cells. Once multiple cells are selected, you can right-click and select **Merge Cells** or **Unmerge Cells**.
- Right-click a cell and insert columns and rows.
- Select one or more cells and right-click to access the other context menu functions, which allow you to add and remove columns and rows.
- Click within a table cell to edit its contents.
- Drag a column or row boundary to resize the column or row.
- Select rows, columns and cells by positioning the cursor to the left of the row, to the top of the column, and in the internal left side of the cell.
- Select and then drag a row or column to move it within the table.
- Apply font and paragraph properties to selected text.
- Press **Tab** to navigate through table cells.
- Resize a table by dragging the circles at its corners or edges.
- Rotate a table by dragging the green circle at the top of the table.
- Copy an annotation dimension and paste it into a table cell. The dimension value will be updated automatically if the dimension changes.

9.8.3. Moving or Rotating a Table

1. Click the outer, dotted border of the table to select it.
2. Use the:
   - Arrow keys to move the table.
   - Ctrl+arrow keys to move the table by a very small amount.
   - Round handle at the top of the table to rotate the table.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

9.8.4. Deleting Text Within a Table

1. Select all the cells in the table.
2. Press **Delete**.

9.8.5. Deleting a Table

1. Select the table.
2. Select the rectangular border.
3. Press **Delete**.

9.8.6. Modifying Table Properties

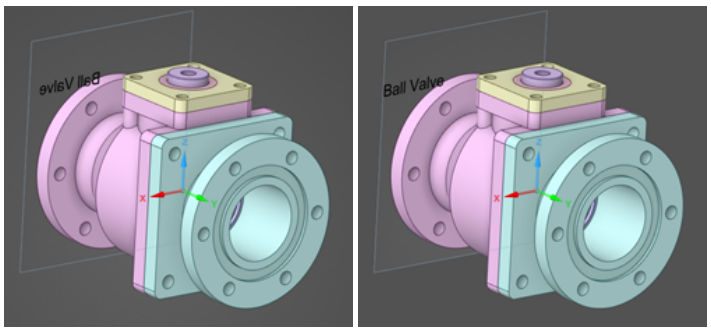
1. Click the outer, dotted border of the table to select it.
2. Enter values in the Properties panel.
   You can modify:
   - Cell alignment and margins.
   - Position of the table anchor.
   - Number of rows and columns, column width, and row height.

9.8.7. Copying Table Contents

Press Ctrl+C or right-click and select **Copy**. You can paste the data from a table into Microsoft Excel.

9.9. Update Annotations Reading Direction Tool

After moving or adjusting a model with annotations, you can use the **Update Annotations Reading Direction** tool to make them readable.



This section contains the following topics:

9.9.1. Reorienting Annotations

1. After adjusting and moving your annotated model, you may notice that the annotations can become unreadable.

2. Click **Update Annotations Reading Direction** in the ribbon to make the annotations in your model readable.

9.10. Center Line Tool

You can put a center mark on any circle, arc, cylinder end, or sphere, and you can put center lines on any cylindrical face on a drawing sheet.

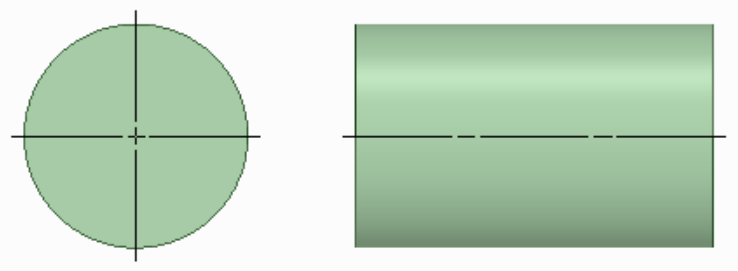
This section contains the following topics:

9.10.1. Adding a Center Mark or Center Line

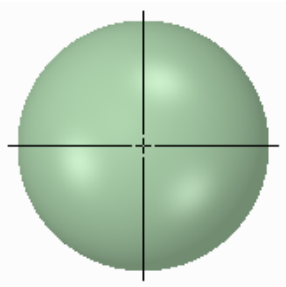
1. Click the **Center Line** tool .
2. Click an:
   - Edge of the end of the cylinder or hole to add a center mark.
   - Cylindrical face to add a center line.
   - Sphere to add a center mark.

9.10.2. Center Marks and Lines Examples

Center mark on the end of a cylinder and center line along the face of a cylinder.



Center mark on a sphere.



9.11. Detail Dimensions Tool

The **Detail Dimensions** tool adjusts the display size of 3D annotations relative to model size. It helps maintain readability for large or small geometry, scaling all annotations uniformly in real time without changing their values or units.

This section contains the following topics:

9.11.1. Scaling 3D Annotation Dimensions

1. Select an annotation in your model.
2. Open the **Scale** dropdown menu from the ribbon and choose your preferred annotation size.

> **Note:** The scale is shown as a ratio (1:1, 1:2, 20:1), where the first number represents the model size and the second represents the annotation size. Higher ratios result in smaller annotations, while lower ratios make annotations appear larger. This helps adjust visibility without changing the model.

9.12. Engineering Notebook

Engineering Notebook can help you keep your design organized and ensure that you don't miss any important details.

You can automatically generate views of your design, which you can then customize by adding, removing, or editing them to suit your needs.

To rearrange your views, simply drag and drop them to the desired location. And because they are saved within your design, you won't have to worry about losing any important information.

This section contains the following topics:

9.12.1. Creating an Engineering Notebook

Before using **Engineering Notebook** for the first time, you need to expose the **Detail** tab.

1. Go to **File Menu > Settings > Customize > Ribbon Tabs**.
2. Click  to expose the tab in the ribbon. The icon will turn to  , indicating that it is now available in the ribbon. Changes take effect immediately.

Launching Engineering Notebook

To launch **Engineering Notebook** click the **Detail** tab and then the **Engineering Notebook** ribbon button.

> **Note:** **Engineering Notebook** is disabled when you are running **History Tracking**.

An **Engineering Notebook** containing front, top, and right-side views appears in the **Design** window, and the sheet appears in the structure tree. The views are added as nodes in the structure tree with visibility toggles for showing and hiding and a context menu for other view operations.

Modifying one view changes the related views as appropriate. You can navigate between sheets using the window tabs found at the lower-right of the **Design** window.

Using the Structure Tree Context Menu

Multiple visibility options give you control over what views you're seeing in the sheet. You can choose to **Show** or **Hide** an individual view through the context menu or by toggling the view icon in the structure tree. Other visibility options allow you to **Hide Others** or **Inverse Visibility** of the views in your sheet.

You can right-click the root node in the structure tree and select **Add Sheet** to add a sheet to your **Engineering Notebook**. New sheets are blank except for a trimetric view of the model in the upper right corner.

9.12.2. Sheet Setup

In the **Details** tab, there are options to adjust your drawing sheet in the **Sheet Setup** group.

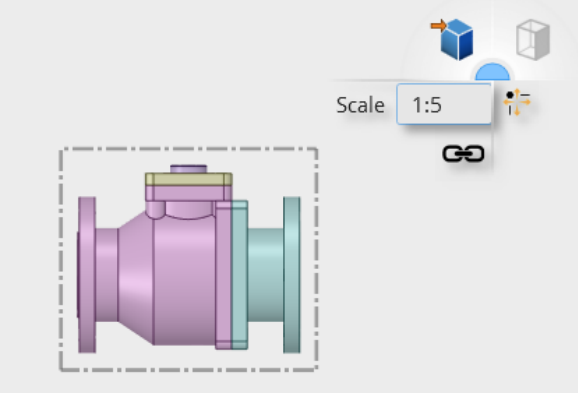
Click **Orientation** and choose **Portrait** or **Landscape** to adjust your sheet's orientation.

Choose **Paper Size** and choose one of the options to adjust the drawing sheet size.

9.12.3. Using the Drawing Scale

On the drawing sheet, you have the option to enter a ratio in the **Scale** field. By entering a ratio of 1:1, you can display views of your design at its actual size. Entering different ratios in the Scale field allows you to adjust the size and presentation of the views.

When you select a view, the semi-halo opens. You can click the **Link** button to link the view's scale to the sheet or unlink it to be independent. By default, the view scale is linked to the sheet scale.

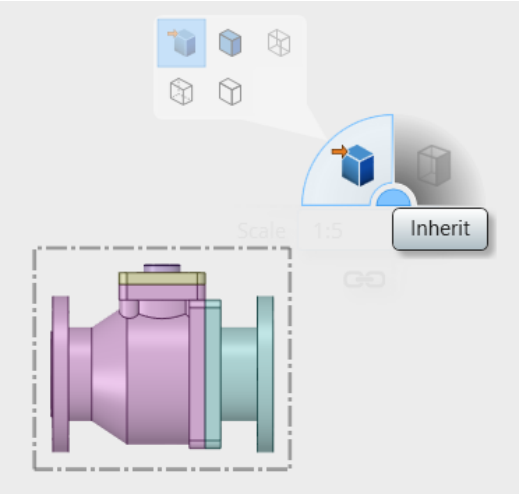


Note: If you make the scale independent, a note is added to the view to show the scale.

You can change the scale of the view independently from the sheet scale.

9.12.4. Rendering View Styles

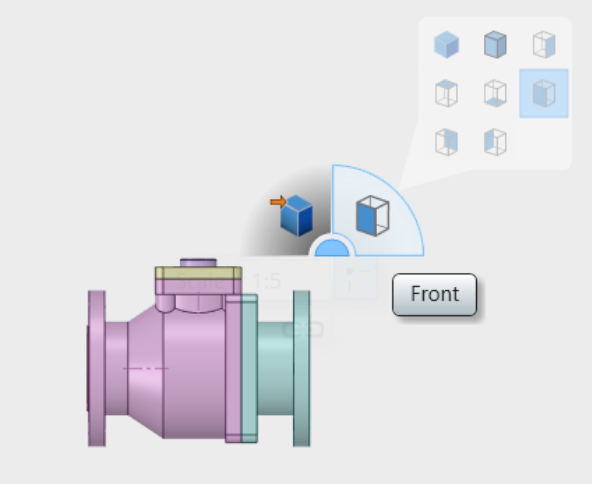
You can hover over the **Inherit** button in the semi-halo to view the current rendering style for a view. The default rendering style is inherited from the 3D model.



You can select different rendering options from the **Inherit** button flyout menu, including **Shaded**, **Wireframe**, **Hidden Line**, and **Hidden Line Removed**.

9.12.5. Orienting Views

You can hover over the **Orientation** button in the semi-halo to view the current orientation for a view.



You can select different orientation options from the **Orientation** button flyout menu, including **Isometric**, **Trimetric**, and the six standard model views.

9.13. Engineering Notebook Views

You can add and remove views from the Engineering Notebook, move them around, and modify their properties. You can create general views, detail views, projected views, broken views, cross-section views, and broken-out section views. These views are all related to the view used to create them, and inherit properties from that view.

This section contains the following topics:

9.13.1. Mastering Views

Selecting a View on the Drawing Sheet

1. Activate the **Select** tool.

> **Note:** If you are in another tool, press **Esc** to cancel the current action, then press **Esc** again to exit the tool and activate the **Select** tool.

2. Mouse just outside the view on the drawing sheet or scroll over the design shown in the view until a dashed box around the view is displayed.
3. Click the box to select the view.

Moving a View on the Drawing Sheet

1. Select the view.
2. Drag the view with the **Select** tool to move it.

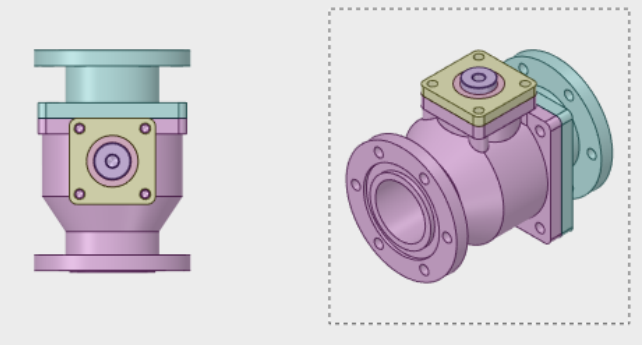
   Projected and cross-section views move together as a group.

9.13.2. General View Tool

Use the **General View** tool to add a new, independent view.

Creating a New, Independent View

1. Click the **General** view tool.
2. Click the drawing sheet to add a new isometric view of your model.



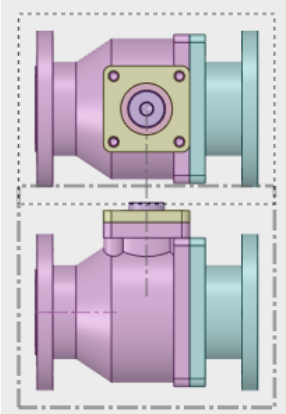
9.13.3. Projected View Tool

Projected views show another side of the model.

Use the **Projected** view tool to add projected and auxiliary views to your drawing sheet.

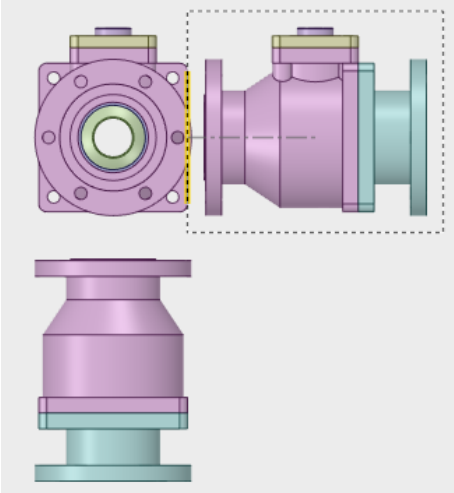
Creating a Projected View

1. After selecting the **Projected** view tool from the ribbon, click within a selected view and move your mouse toward the top, bottom, left or right to preview a projected view.
2. Click to create the projected view on the drawing sheet.



Creating an Auxiliary View

1. Click the **Projected** view tool from the ribbon.
2. Click a highlighted face or edge within the selected view to create an auxiliary view.



9.13.4. Cross Section View Tool

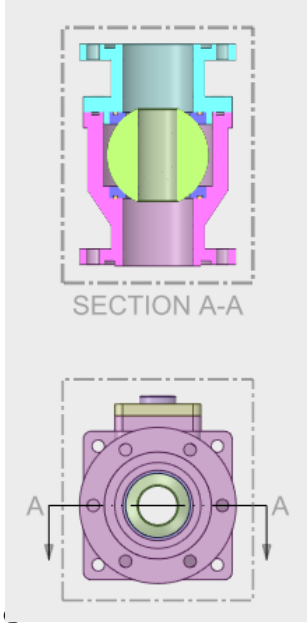
Use the **Cross Section** view tool to add cross section views to your drawing sheet. Cross-section views show interior details of your design.

To create a cross-section view, you must already have at least one view. You can add a cross-section to any view type.

Creating a Cross Section View

1. After selecting the **Cross Section** view tool from the ribbon, select the view to convert to a cross section.

2. Mouse over the projected view to set the depth of the cross



3. Click to place the indicator and convert the selected view to a cross section.

9.13.4.1. Cross Section View Tool Guides and Options

Tool Guides

Select Section View	Select this tool option and then click a view to that you would like cross sectioned. Move your cursor to a projected view to set the depth of the cross section.
Select Reference Geometry Inside Drawing View	Select to place the section line. You can use **Horizontal**, **Vertical**, and **Through Selection** options to select the appropriate arrow orientation and **General View** and **Auxiliary View** to create the desired view.
Set Pivot for Aligned Sections	Select this tool guide and click somewhere on the section line to set the pivot point.
Create Offset Section	Select this tool guide and click the section line at a point where you want to create an offset. Drag the side of the section line you want to have pass through a different detail and continue until you establish all necessary offsets.
Create Aligned Section	Select this tool guide and click somewhere on the section line to set a cutter point.

Tool Options

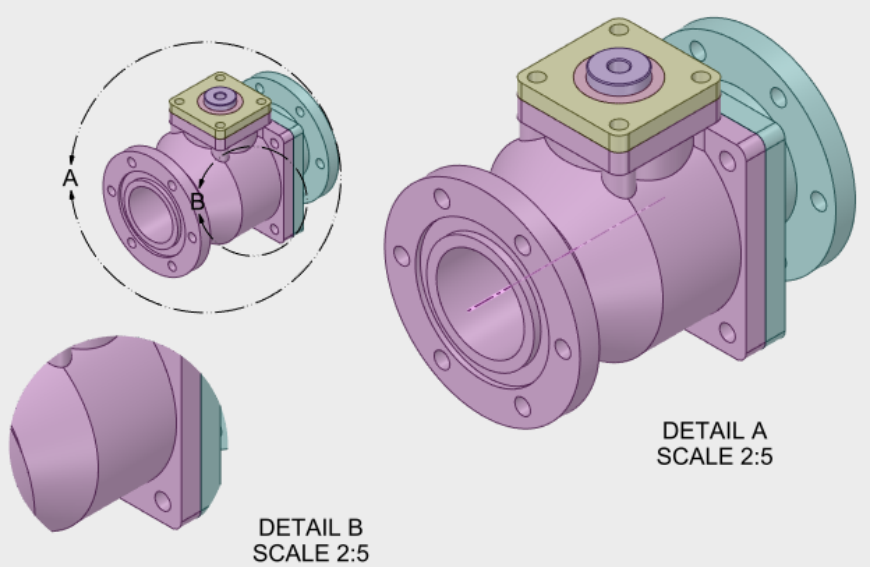
Create a total cross-section	Check to set the view's section type property to **Total**. Uncheck this option to set the property to **Area**.
Horizontal/Vertical/Through Selection	Choose the direction to orient the arrows. Select **Through Selection** to orient the arrows based on the highlighted object.
General View	Create as **General View** to place the view in any location on the drawing sheet.

Auxiliary View	Create as **Auxiliary View** to link the orientation of the cross-section view to the view containing the arrows.

9.13.5. Detail View Tool

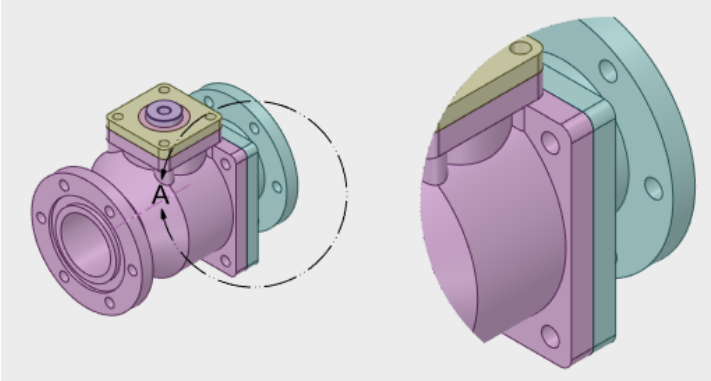
Use the **Detail** view tool to add a detail view on the drawing sheet.

Detail views allow you to magnify a portion of another view to show more detail. To create a detail view, you must already have at least one view on your drawing sheet.



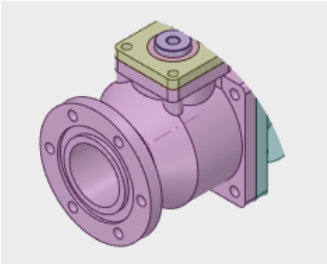
Creating a Detail of Selected View

1. Click the **Detail** view tool to add a detail view on the drawing sheet.
2. Choose **Create detail of selected view** to create a new detail view with the boundary drawn on an existing view.
3. Choose the **Sketch Boundary Type** to determine the shape of the new view.
4. Click an object to set the anchor point for scaling, click to set the boundary center, then click to draw the boundary.
5. Click again to place the detail view on the drawing sheet.



Converting a View to a Partial View

1. Click the **Detail** view tool to add a detail view on the drawing sheet.
2. Choose **Convert view to partial** to convert an existing view into the detail view.
3. Choose the **Sketch Boundary Type** to determine the shape of the new view.
4. Click an object to set the anchor point for scaling, click to set the boundary center, then click to draw the boundary.
5. Click again to place the detail view on the drawing sheet.



9.13.5.1. Detail View Tool Options

Create detail of selected view	Create a new detail view with the boundary drawn on an existing view.
Convert view to partial	Convert an existing view into the detail view.
Circle/Rectangle/Spline	Select the **Sketch Boundary Type** for the new view.

9.13.6. Broken View Tool

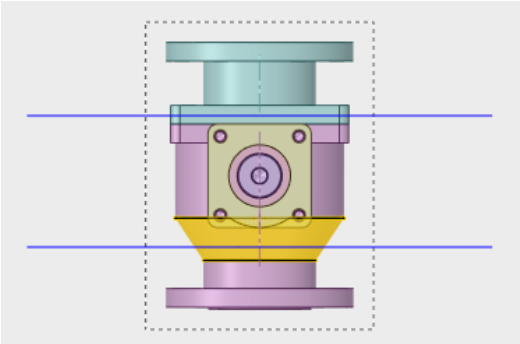
Use the **Broken** view tool to add breaks to shorten an existing view.

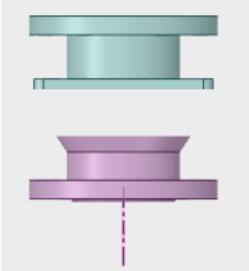
Broken views are a way of foreshortening long views without changing the scale.

When you break a view, a new view is created and both the original and the new view are converted to partial views. An alignment is set up to match the cut orientation.

Creating a Broken View

1. Click a view geometry to set the start of a break.
2. Click again to set the end of the break.
3. Use the options panel to control the **Cut Direction**, **Break Line** and **Default Gap**.





9.13.6.1. Broken View Tool Options

Cut Direction: Horizontal/Vertical/Aligned to Geometry	Sets the cut direction for the broken view.
Break Line: Straight Cut/Freehand Split Cut	Set the cut to a straight line or draw the desired cut.
Default Gap	Set the initial gap between broken view pieces.

9.13.7. Broken-out Section View Tool

Use the **Broken-out Section** view tool to create a partial cross section view.

Broken-out sections reveal internal details of a specific part of an existing view. To create one, simply choose an attachment point within the view, determine the desired section depth, and sketch the boundary of the section. This will remove the area within the boundary up to the section depth, providing a clear view of the exposed interior details.

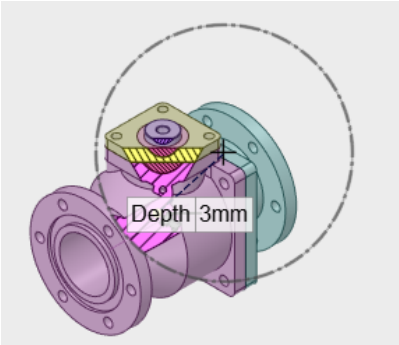
This section contains the following topics:

Creating a Broken-Out Section View

1. Click a view geometry to set the anchor point through which the section plane is placed.
2. Set the depth in options. The default is zero.

> **Note:** If the attachment point is on a hole axis, a zero depth creates a section through the center of the hole.

3. Choose the boundary type in options: **Circle** or **Spline**.
4. Draw a circular or closed spline region to cut away model geometry and show the partial cross section.



9.13.7.1. Broken-out Section View Tool Options

Boundary Type: Circle/Spline	Sets the boundary type for the view.
Depth	Sets the depth for the view. The default setting is zero. If the attachment point is on a hole axis, a zero depth creates a section through the center of the hole.

10: Measuring and Analyzing

The **Measure** tab contains the tools you use for measuring, displaying interference, and analyzing quality in your design.

Inspect	Tools for displaying measurements of the edges, faces, and solids in your design.
Interference	Tools for displaying edges where solids intersect each other or volumes created by the intersection of solids, surfaces, and components in your design.
Quality	Tools for detecting anomalies or discontinuities in surfaces that shading cannot show.

This section contains the following topics:

10.1. Quick Measurements

A quick measurement is displayed in the status bar when you select a single object or a pair of objects. This measurement is displayed using the units and precision set in the Units options for the current document. The **Measure** tool also has a **Units** option to switch units for the measured values without changing units for the document.

In the Discovery application, you can expand the Quick Measurement field and change the Model unit system for the current document.

If the text in the quick measure area is cut off, move your mouse over the text area to show all of the text.

To measure:	**Select:**
Distance between two objects	• Two parallel linear objects (lines, edges, axes) • Two parallel planar objects (faces, surfaces, planes) • Two points or vertices • One point or vertex and one curve or edge (curved or linear) - displays the shortest distance from the point to the curve or edge • One point and one analytic face, edge, or curve - displays the projected distance of point onto face, edge, or curve • Two parallel, but not concentric, cylindrical faces - displays the distance between the axes • Two non-concentric circular edges or arcs - displays the distance between the centers
Length of an object	• A linear object (line, edge)
Radius of an object	• A circular object (circular edge or curve, cylinder, sphere)
Angle between two objects	• Two non-parallel linear objects (lines, edges, axes) • Two non-parallel planar objects (faces, surfaces, planes) • Two analytic curves that share an end point - displays the angle between the curves at the point where they meet
Offset	• Two concentric cylindrical faces • Two circular edges that are in the same plane

To measure:	Select:
X, Y, Z coordinates from the world origin	• A point or vertex

10.2. Inspect

Select a tool from the **Inspect** ribbon group to display measurements for edges, faces, solids, and environmental indicators in your design.

Measure	Use the **Measure** tool to display measurements of the edges and faces in your design. Select from this tool's menu to display edges and volumes of intersection.
Mass Properties	Use the **Mass Properties** tool to display volume information for the objects in your design.
Environmental Properties	Use the **Environmental Properties** tool to display cumulative environmental information such as climate change cost, embodied energy, and the percentage of recyclable material mass for solid bodies in your design.

This section contains the following topics:

10.2.1. Measure Tool

Use the **Measure** tool to display measurements of the edges and faces in your design. Any values displayed on screen are automatically copied into the clipboard and can be pasted into another document.

The **Measure** tool is accessible in the ribbon and from within the **Pull** and **Move** tools. Measurement values are selectable only when the tool is invoked within **Pull** or **Move**.

You can select units for measurement in the **Units options**. You can also modify the **Precision** and **Angular Precision** values in the Measure Tool options.

The document origin is displayed by default. You can hold **Alt** and select the origin or its axes as reference objects, and the distance in that direction (or all three) is displayed.

If you hold **Alt** and select an origin, then you will see a preview of the X, Y, and Z distance from the origin. This preview changes as you move your mouse over objects in the Design window. If you hold **Alt** and select any plane, then the preview shows the distance from the point under the mouse to the plane. If you hold **Alt** and select a line or axis, then the preview shows the distance from the point under the mouse to the line. Clicking on an object sets the dimensions in the results box.

Negative values are displayed for the results when you select an origin and the measurement is negative in the direction of one or more axes.

This section contains the following topics:

10.2.1.1. Viewing a Measurement

1. Click **Measure** in the **Inspect** group on the **Measure** tab.
   Mouse over your design to preview the faces and edges eligible for measurement.
2. Select points, curves, edges, or faces to display measurement information.

Ctrl+ Selecting multiple objects displays measurement between them as appropriate. The value **Min. distance between** objects displays an exact measurement of the minimum distance between the objects you selected.

Selecting multiple faces displays their total area.

Triple-click a body to display its total surface area. Selecting a whole body in the **Measure** tool also reports the overall X, Y, and Z direction extents.

3. (Optional) Hold **Alt** and select a reference object.

   The reference can be a point, curve, edge, face, plane, axis, or origin.

   You can measure to an axis of an origin object.

4. Hover over a measurement value to display the semi-halo.

   - Click **Copy** to copy the measurement value to the clipboard for pasting into Notes, etc.
   - Click **Create output parameter** to create an output parameter from the measurement.
   - Click **Record script** to record parameters as a script in the script editor window, when scripting is enabled.

   When measuring edges and curves, the **Minimum** and **Maximum Curvature** locations are highlighted in the geometry when you hover over the result.

10.2.1.2. Measure Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select objects to measure	The **Select objects to measure** tool guide is active by default. This tool guide allows you to select the object you want to measure.
Select plane or origin	The **Select plane or origin** tool guide allows you to set the measurement directions by selecting an origin, origin axis, line, or plane. You can hold **Alt** and select the origin to display the X, Y, and Z coordinates from the origin to the object being measured or between the objects if you select two objects. You can select a plane or origin in the model tree or the Design window.

10.2.2. Mass Properties Tool

Use the **Mass Properties** tool to display volume information for the objects in your design.

> **Note:** The accuracy of the mass properties calculation is based on displayed facets, approximations can occur based on faceting.

If you use the tool on a surface, it will display the total surface area. If there are multiple surfaces on the same plane, the tool displays the total surface area for all the surfaces. To display the surfaces' individual surface areas, use the Measure tool.

This section contains the following topics:

10.2.2.1. Viewing Mass Properties of an Object

1. Click **Measure** > **Inspect** > **Mass Properties**.

With the **Select Body** tool guide active, select a solid in the **Design** tree or by triple-clicking it in the Design window to display its volume, center of mass, and principal moments and axes.

2. A small origin appears at the center of volume with its axes oriented in the direction of the principal axes. To calculate the principal moments and axes for another point, hold **Alt** and select an origin to add it to your selection.

   To calculate the actual moment of inertia, multiply the moment measurement by the density of the solid.

   The axis is indicated with x, y, z values. In the figure above, these values are (0, -1, 0). The first number is the red axis of the origin, the second is the green axis, and the third is the blue axis.

3. For beam objects ensure that the **Material Name** property is set as appropriate to display the mass and center of mass properties in addition to the volume.

10.2.2.2. Measuring the Projected Area of an Object or Objects

1. Click **Measure** > **Inspect** > **Mass Properties**.
2. Click the **Select datum plane** tool guide.
3. Select the objects you want to measure.

   You can also hold **Alt** and select the object without using the tool guide.

   The projected area and estimated precision is displayed in the Design window.

10.2.2.3. Measuring the Properties of a Planar Section

1. Click **Measure** > **Inspect** > **Mass Properties**.
2. Click the **Select Planar Face or Section Loop** tool guide and select a planar area.

> **Note:** Mass properties for planar sections are approximated using edge tessellation.

10.2.2.4. Selecting a Coordinate System

1. Click **Measure** > **Inspect** > **Mass Properties**.
2. Select the objects you want to measure.
3. Click the **Select Coordinate System** tool guide and select a coordinate system or coordinate system axis.

   The center of volume and moment and axis are measured from the origin/axis chosen rather than the default origin (0, 0, 0).

10.2.2.5. Measuring a Projected Area

You can measure the projected area of one or more objects. The objects are projected onto a plane, and the area of their silhouette is measured.

To measure the project area of an object or objects:

1. Click **Measure** > **Inspect** > **Mass Properties**.
2. Select the objects you want to measure.
3. Select **Options** > **Show projected area** to see a shadow of the projected area.

4. Click the **Select Datum Plane** tool guide or hold **Alt** and select the plane you want the objects' silhouette projected on.

   The projected area and estimated precision is displayed in the Design window.

10.2.2.6. Mass Properties Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Body	The **Select Body** tool guide is active by default. This tool guide allows you to select bodies to measure mass properties, total surface area, and principal moments and axes.
Select Planar Face or Section Loop	The **Select Planar Face or Section Loop** tool guide allows you to select a planar face or cross section curve loop in Section mode to measure area or cross-section properties.
Select Datum Plane	The **Select Datum Plane** tool guide allows you to select a datum plane to calculate projected area.
Select Coordinate System	The **Select Coordinate System** tool guide allows you to select a coordinate system or coordinate system axis.

10.2.3. Environmental Properties Tool

Use the **Environmental Properties** tool to display information about environmental stressors. The environmental impact is evaluated based on indicators such as embodied energy, climate change in terms of equivalent carbon emissions, and if the material has potential to be recycled.

This section contains the following topics:

10.2.3.1. Displaying Environmental Properties

1. On the **Measure** tab, click **Environmental Properties**.
2. With the **Select bodies** tool guide active, select one or more bodies to display the environmental property data.

   Measures for **Climate change (CO_2-eq)**, **Embodied energy**, and the amount of potentially **Recyclable material mass** as a percentage are listed in the options panel. These values are based on the assigned materials and volume of the selected bodies.
3. (Optional) To track the environmental properties of the selected body, click the value and select **Create output parameter** in the semi-halo. The values will update whenever the assigned material is modified or geometry changes.

   When you create output parameters, the Defined variations hex appears in the SID. There you can investigate parameterized geometry changes, make interactive modifications to the model, and perform interactive solves.
4. (Optional) To record a script, on the **File Menu**, click **Script Editor**. When the **Script Editor** window is active, a **Record measurement as a script** button is available in the semi-halo of the environmental property.

10.2.3.2. Environmental Properties Tool Guides and Options

Environmental Properties Tool Guides

The following tool guide is available:

Select bodies	Selects the bodies for which to display environmental property data. This information depends on the assigned material and volume of the object.

Environmental Properties Tool Options

The following tool options are available in the semi-halo for each property.

Copy text	Copies the value to the clipboard.
Create output parameter	Creates a parameter of the environmental property which you can track. This parameter appears in the Defined variations table.
Record measurement as a script	Records the measurement as a script. This option is available after opening the **Script Editor** window from the **File Menu**.

10.3. Interference

Use the **Curves** tool to display the edges where solids intersect each other.

Use the **Volumes** tool to display volumes created by the intersection of solids, surfaces, and components in your design.

This section contains the following topics:

10.3.1. Displaying the Edges of Intersection

1. Click **Measure** > **Interference** > **Curves**.
2. Select intersecting objects to see their edges of intersection.

10.3.2. Displaying Volumes of Intersection

1. Click **Measure** > **Interference** > **Curves**.
2. Select objects to display their intersection in green.

   You can also select multiple faces by box-selecting or selecting in the model tree.

   Interference with thread intersections are not reported.
3. (Optional) Click the **Create Volumes** tool guide and select an area of interference to create a new solid from the volume.

10.3.3. Interference Tool Guides and Options

Interference Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Targets	The **Select Targets** tool guide allows you to select the objects for which interference will be shown.
Create Volume	The **Create Volume** tool guide allows you to select an area of interference and create a new solid from the volume.

Interference Tool Options

The following tool option is available:

Collapse to parent component	All of the component's sub-components are treated as part of a single solid when performing the volume interference calculation. Select this option to optimize performance.

10.4. Quality

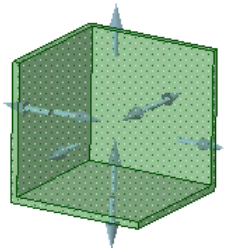
The tools in the **Quality** ribbon group allow you to detect any anomalies or discontinuities in a surface that shading cannot show.

Normal	Use the **Normal** tool to display the normal direction ("inside" or "outside") of faces or surfaces.
Grid	Use the **Grid** tool to display the curves that define any face or surface in your design.
Curvature	Use the **Curvature** tool to display a fringe graph of the curvature along curves or edges.
Dihedral	Use the **Dihedral** tool to display a fringe graph of the angle between two faces along the selected edges.
Draft	Use the **Draft** tool to display a fringe graph of the selected surfaces, where each value is an angle measurement.
Stripes	Use the **Stripes** tool to reflect a virtual 3D cube "room" on the selected faces.
Deviation	Use the **Deviation** tool to show the distance from a source, or reference, body to a selected body or Mesh body.

This section contains the following topics:

10.4.1. Normal Tool

The **Normal** tool will display the normal direction of faces or surfaces in your design. The normal direction is the "outside" of the face or surface. The normal can be flipped when you import models. Use this tool to check and fix face normal directions.



This section contains the following topics:

10.4.1.1. Displaying Normal Directions for Faces

1. Click **Measure** > **Quality** > **Normal**.
2. Select an object, face, or surface. **Ctrl+click** to select multiple faces and surfaces, or select an object in the model tree.

   The normal direction of each face or surface at the point where you click is displayed.
3. If you find an incorrect normal, right-click the face of a surface or surface body and select **Reverse Face Normal**.

10.4.1.2. Normal Tool Options

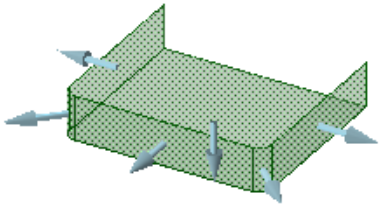
The following options are available for the **Normal** analysis tool:

View the normal direction using:

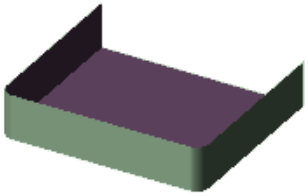
Arrow	Display the normal direction using an arrow.
Color	Display the normal direction using colors. Select colors for the front ("outside") and back ("inside") using the dropdown menus. Use contrasting colors to make it easier to see which faces were incorrectly reversed on import.

10.4.1.3. Normal Tool Examples

Displaying face normals using arrows.



Displaying face normals using colors.



10.4.2. Grid Tool

Use the **Grid** tool to display the curves that define any face or surface in your design. This tool displays a visual representation of the mathematical expression of the surface. The grid lines represent the mathematical expression used to generate the surface. You can use it to identify a low quality surface. For example, a face may look fine, but the underlying mathematical representation is unnecessarily complicated or poorly parameterized. You would have problems changing the face using other tools.

If you find a face with underlying problems, you can delete the face, then use the **Fill** tool with the **Patch Blend** option or the **Replace** tool to generate a new face.

This section contains the following topics:

10.4.2.1. Viewing a Face or Surface Grid

1. Click **Measure** > **Quality** > **Grid**.
2. Select a face or surface. **Ctrl+click** to display the grid on multiple faces and surfaces.
   Click in an empty area in the Design window to hide the grid.

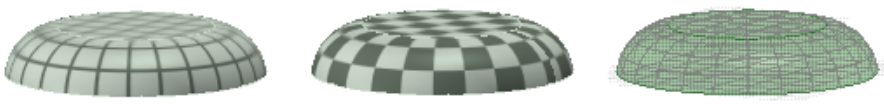
10.4.2.2. Grid Tool Options

The following options are available for the **Grid** analysis tool:

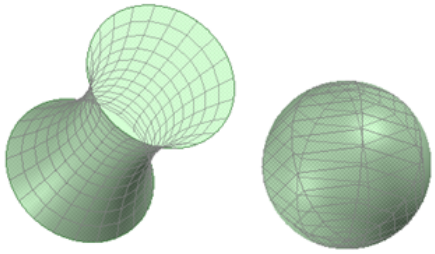
Shading	Select **Grid** or **Checker** from the **Texture** menu. Use this option to increase performance with large or complex designs.
Wire	Slide the scale or click **-** or **+** to set the scale of the grid.

10.4.2.3. Displaying a Face Grid Examples

Face grids with grid shading, checker shading, and wire



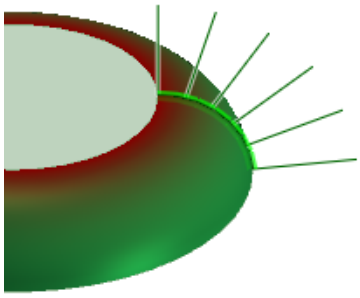
Face grids displayed on a blended face and on a sphere.



10.4.3. Curvature Tool

Use the **Curvature** tool to show curvature values along faces or edges. You can use this analysis to identify areas with very tight curves that can be difficult to offset or turn into thin solids.

You can also use the visualization to evaluate curvature continuity along a set of faces or edges. Curvature continuity is when the curvature values change in a smooth, continuous manner. You can see continuous curves because the data points will gradually change length or the transition between face coloring is even and smooth. Sudden changes in the length of the data points or color on a face indicates non-continuous curvature.



This section contains the following topics:

10.4.3.1. Displaying Edge Curvature

1. Click **Measure** > **Quality** > **Curvature**.
2. Select a line, curve, or edges. Ctrl+click to select multiple lines or edges, or select an object in the model tree.

   You can change the shading colors in the options panel.

   You can change the color, scale, and density of the data points in the options panel.

   > **Tip: Select Edges** in the selection filter, then box select to select all the edges in your design.

   The visualization for edges shows lines at data points along the edge. The greater the curve's radius, the longer the data point. Straight edges won't show any data points because they don't have curvature.

10.4.3.2. Displaying Face Curvature

1. Click **Measure** > **Quality** > **Curvature**.
2. Select a face or Ctrl+click to select multiple faces.

   You can change the shading colors in the options panel.

In addition to the face curvature display, the absolute values for minimum and maximum curvature are displayed in the **Properties** panel.

The curvature with the smallest radius is shown in the maximum color (red is the default). The curvature with the smallest radius, or no radius, is shown in the minimum color (gray is the default).

Each line length of the fringe graph represents a curvature value C = absolute value (1/r) when a surface or body is selected.

10.4.3.3. Curvature Tool Options

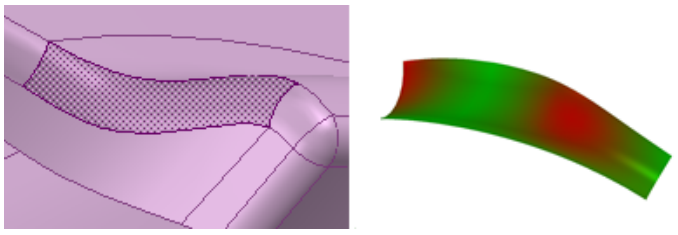
The following options are available for the **Curvature** analysis tool:

Show curvature using:

Color	Select a color for the fringe graph from the dropdown menu.
Scale	Slide the scale or click **-** or **+** to increase or decrease the relative size of the fringe graph.
Density	Slide the scale or click **-** or **+** to increase or decrease the density of sampling (the number of lines created along the edge).
Shading	Select **Shading** to display face curvature using color. Select colors for minimum and maximum curvature from the dropdown menus.

10.4.3.4. Curvature Tool Examples

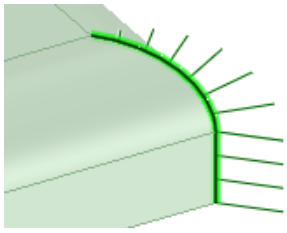
Displaying face curvature using color.



10.4.4. Dihedral Tool

The **Dihedral** tool can be used to visualize tangency (or lack of tangency) between two faces. You won't see any data points if you select an edge between two tangent faces. Non-tangent faces will show data points because the angle is more than 0. The larger the angle between the faces, the longer the fringe lines will be at each test point.

You can use the **Repair** dihedral tool guide to make the faces tangent. You should only use this tool guide if the faces are already close to tangent. Neighboring edges are not modified to be tangent as well, so the results can be unexpected if the faces are not close to tangent.



This section contains the following topics:

10.4.4.1. Displaying a Dihedral Graph

1. Click **Measure** > **Quality** > **Dihedral**.
2. Select a dihedral edge. You can **Ctrl+click** to select multiple edges.

   Data points show the angle between the faces along the edge. Longer data points are shown for larger angles, or, variable dihedral angles

   The **Analysis** section of the **Properties** panel displays both minimum and maximum edge dihedral angle values.

10.4.4.2. Dihedral Tool Options

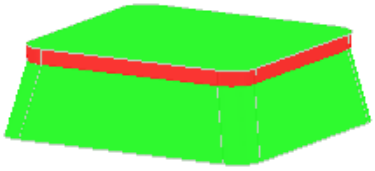
The following options are available for the **Dihedral** analysis tool:

Color	Select a color from the dropdown menu.
Scale	Slide the scale or click **-** or **+** to decrease or increase the relative size of the fringe graph.
Density	Slide the scale or click **-** or **+** to decrease or increase the number of points along the edge where the angle is measured and displayed.

10.4.5. Draft Tool

The **Draft** tool helps you identify the amount of draft and direction on each face in a design.

You can use this tool to analyze parts that will be molded. Faces shown in red (the default color for a negative draft angle) may not release properly from a mold. You should specify the draft angle you consider necessary for proper release. The draft angle may depend on a number of factors, including material choice and design constraints.



This section contains the following topics:

10.4.5.1. Displaying a Draft Angle

1. Click **Measure** > **Quality** > **Draft**.
2. Click the **Draft Direction** tool guide and select a face, surface, or the World Origin to set the draft direction.

   If you don't first set the draft direction, the angle is measured with respect to the Z axis.
3. Set the **Angle** in the options panel.

   The face highlight color is based on the angle you specify. A face will be highlighted in the positive color if the angle of the face is greater than the Angle value and the negative color if its angle is smaller in the direction you specify. You can change these colors in the **Options** panel.
4. Click the **Draft Select** tool guide and select a face to check its draft angle. You can also Ctrl+click to select solids and surfaces, box-select or select objects in the model tree.

A color graph of the selected surfaces is displayed. Each value is an angle measurement between the surface and the selected plane.

Set the probe location(s) to see the draft value at the selected locations.

You can also see the draft angle at any given location by hovering over the display.

10.4.5.2. Draft Tool Guides and Options

Draft Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Draft Select	The **Draft Select** tool guide is active by default. Use it to select the face(s) for which you want to display draft angles.
Draft Direction	Use the **Draft Direction** tool guide or Alt+click to select an alternate reference for the draft direction.
Draft Probe	Use the **Draft Probe** tool guide to set a probe location. Use the Ctrl key to select multiple probe locations.
Draft Curve	Use the **Draft Curve** tool guide to select a temporary curve (created by the faces selected, draft direction, and angle of one or both sides) to create the curve in the model tree. Curves appear in the current layer color.

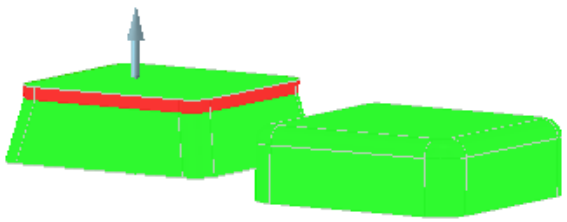
Draft Tool Options

The following options are available:

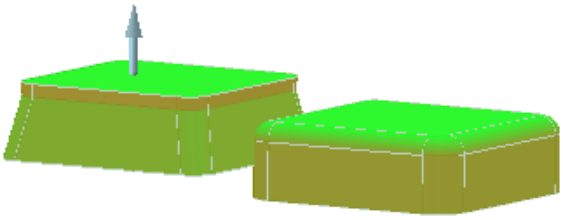
Angle	Slide the scale or click **-** or **+** to set the minimum angle to display.
Direction	Select **One** or **Both**to display the draft angle in one or both directions.
Color	Select colors from the drop-down menus to indicate positive and negative draft angles.
Color Transition	Slide the scale or click**-** or **+** to adjust the color transition between sharp and smooth. The higher this setting, the more gradual the transition between positive and negative faces.
Shadow Lines	Display shadow lines where the angle of the face is the same as the draft angle.

10.4.5.3. Draft Tool Examples

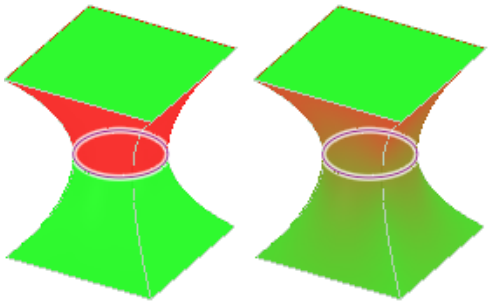
Draft analysis with color transition low (sharp).



Draft analysis with color transition set to high (smooth).



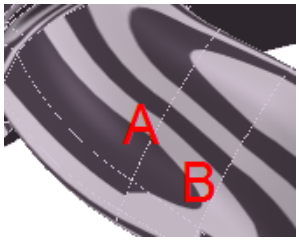
The shadow line on these examples show where the angle of the curved face is the same as the draft angle. The example on the left is shown with the Color Transition option set low, and the example on the right shows the Color Transition set high.



10.4.6. Stripes Tool

The **Stripes** tool will reflect an infinite striped plane on the selected faces (or on all the faces of the selected solid). This tool is useful to show how the smoothness of a surface.

Use this tool to visualize and check tangency and curvature continuity between faces. For example, two surfaces may appear to have a smooth transition, but the **Stripes** tool will reveal an irregularity.



In the example above, the area marked with **A** has good continuity. The stripes line up almost perfectly across the edge. The area marked with **B** does not have good continuity. The stripes do not quite line up at the edge.

If you find irregularities in your design, you can repair them using the following tools:

- **Tangency** (on the **Prepare** tab)
- **Merge Faces** (on the **Prepare** tab)
- **Fill** with the **Patch Blend** option (on the **Design** tab)

This section contains the following topics:

10.4.6.1. Displaying Stripes

1. Click **Measure** > **Quality** > **Stripes**.

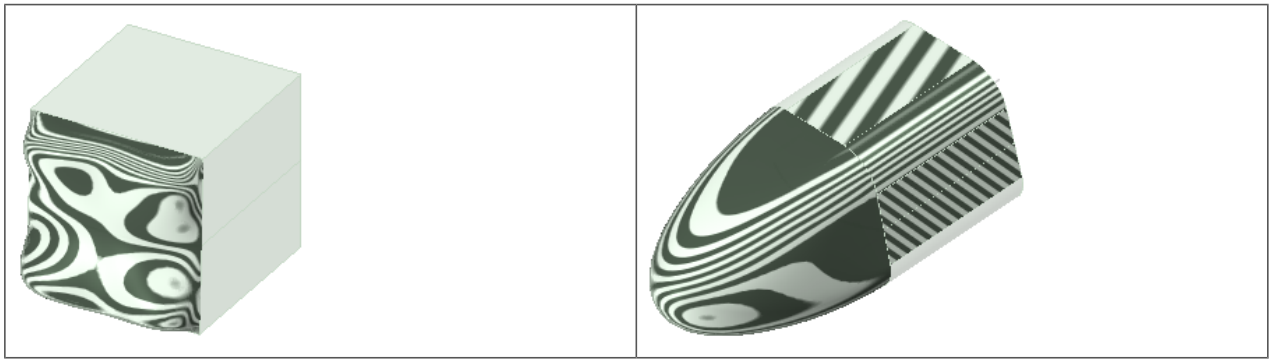
2. Select the face that you want to appear striped. Ctrl+click to select multiple faces or select an object in the model tree.

10.4.6.2. Stripes Tool Options

The following options are available with the **Stripes** analysis tool:

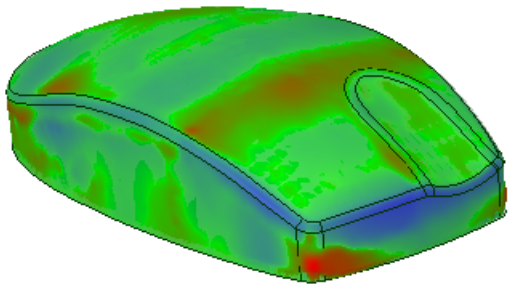
Front, **Back**	Select front and back colors for the pattern from the dropdown menu.
Density	Slide the scale or click **-** or **+** to increase or decrease the stripe density (the number of lines in the pattern).

10.4.6.3. Stripes Tool Examples



10.4.7. Deviation Tool

Use the **Deviation** tool to show the distance from a source, or reference, body to a selected body or mesh body. Use this analysis to see how closely the geometry of two bodies match up. For example, you would use the **Deviation** tool when you reverse engineer a design from mesh data to see how closely your design matches the mesh.



The **Deviation** tool also works to show deviation between meshes.

This section contains the following topics:

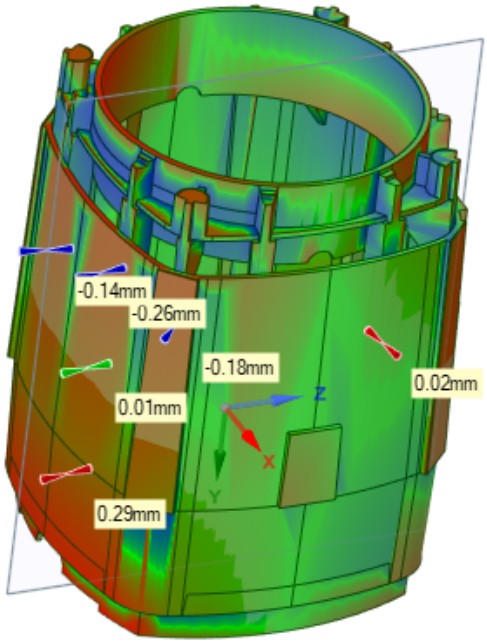
10.4.7.1. Displaying Deviation

1. Click **Measure** > **Quality** > **Deviation**.
2. Select a body or a mesh as the source, or reference, body.

3. Select a second body as the target body.
   When you have two bodies selected, the system automatically calculates the deviation.

> **Note:** You can select only one mesh or faceted body.

4. Set the probe location(s) to see the deviation value at the selected locations.
   You can also see the deviation value at any given location by hovering over the deviation display.



> **Note:** In a small number of cases, deviation results may be exaggerated. You should take sample measurements in the areas of maximum inside and outside deviation to rule out the chance of exaggeration.

10.4.7.2. Deviation Tool Guides and Options

Deviation Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Body	The **Select Body** tool guide is active by default. Use it to select the source and target bodies.
Deviation Probe	Use the **Deviation Probe** tool guide to set a probe location. Use the Ctrl key to select multiple probe locations.

Deviation Tool Options

The following options are available:

Swap Source/Target	Swap the faceted bodies selected as source and target bodies.
Deviation	• **Inside distance** - Choose a color for displaying distances to sample points inside of the target body. • **Outside distance** - Choose a color for displaying distances to sample points outside of the target body. • **Tolerance** - Display deviation results above a specified tolerance. Choose a color for displaying distances within the specified tolerance • **Transition** - Slide the scale or click **-** or **+** to increase or decrease the smoothness of the transition between colors in the display.
Sample Spacing (Source Body)	Select the method for defining the sample point spacing. • **Body** - Use body faceting vertices as sampling points • **Percentage** - Specify a percentage of the body's diameter to define sample spacing • **Length** - Specify a length to use as the sample spacing
Show	Display controls: • **Shading** - Use textured shading to indicate deviation • **Porcupine** - Use lines to indicate deviation • **Sample points** - Show sample points in the deviation display • **Target** - Show the target body in the display **Transparent** - If the target body is displayed, show it as transparent

10.4.8. Control Points Tool

Use the **Control Points** tool to display the control points on a spline face, edge, or curve. You can display control points on multiple spline entities simultaneously.

This section contains the following topics:

10.4.8.1. Displaying Control Points

1. Select **Measure** > **Quality** > **Control Points**.
2. Select a spline face, spline edge, or spline curve to display the control points.
   Use Ctrl to select multiple spline entities.

11: Repairing Problems

The Discovery application can import a variety of native and neutral CAD formats, but you may need to clean up and repair this data for use in the application and for CAE. The **Repair** tab includes tools you can use to repair imported models and prepare your designs for export and analysis.

In general, when you need to repair geometry imported from another source, you use the tools in the order they are presented in the ribbon (left to right).

This section contains the following topics:

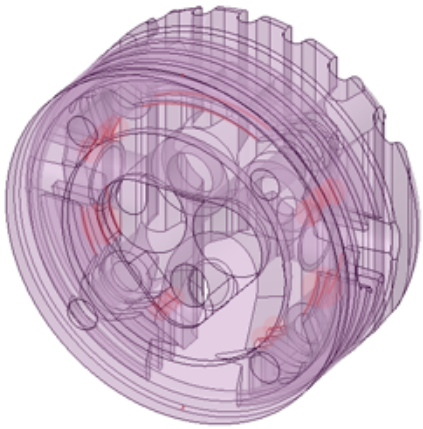
11.1. Navigate Through Issues

The Navigate ribbon group allows you to quickly jump through all the problems identified by a tool on this tab.

11.1.1. Navigating Identified Issues

1. Click the **Next** or **Back** buttons to view each problem.
2. Click **Select** to convert the problem area into a selection and exit the tool for manual fixing.
3. Select **Zoom to Fit** to zoom in on each problem when you click next or back.
4. Select **Clip Volume** to put a Clip Volume spheres around the problem areas. You can then use the Next and Back buttons to make them current.

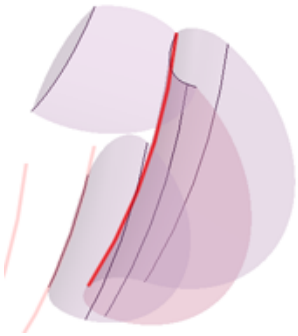
Problem areas are found.



Clip Volume places clipping spheres around the problem areas.



Zoom to fit the current problem.



11.2. Solidify

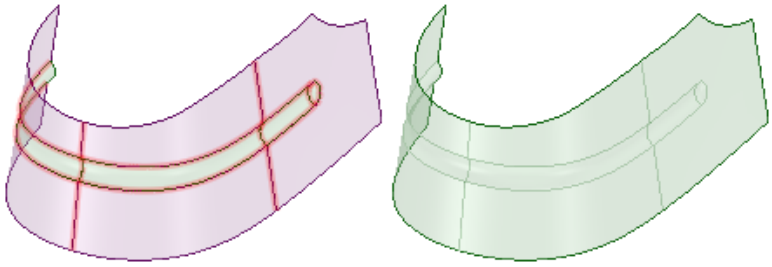
Tools in the **Solidify** group fix problems that may exist in imported data, which prevent the model from being solid.

Stitch	Use the **Stitch** tool to combine surface part faces that are touching at their edges.
Gaps	Use the **Gaps** tool to repair gaps between faces.
Missing Faces	Use the **Missing Faces** tool to fill missing faces.

This section contains the following topics:

11.2.1. Stitch Tool

The **Stitch** tool combines surface part faces that are touching at their edges. When the merged faces form a closed surface, a solid is automatically created. You can use this tool to repair multiple surface parts that are in separate components. Coincident faces are detected and removed before Stitch merges surfaces into a single body.



This section contains the following topics:

11.2.1.1. Stitching Faces Together

1. (Optional) Select one or more surface parts or a components.
   Selecting objects before starting the tool will limit the tool's results to those objects.
   If you select a component, all surface parts in the component that have touching faces will be merged.
2. Click **Stitch** in the Solidify group on the Repair tab.
   The edges of faces that can be merged are highlighted.

3. (Optional) Change the **Maximum distance** slider in the Options panel to adjust the distance between parts that are automatically detected.
4. Select the **Check for coincidence** check box to find any edges or planes that are coincidental to each other.
5. Click the highlighted faces that you want to merge.

   The faces are merged when you select them.
6. (Optional) Click the **Select Geometry** tool guide to select faces that were not automatically detected.
7. Click **Complete** to merge all highlighted faces.

11.2.1.2. Stitch Tool Guides and Options

Stitch Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Problem	The **Select Problem** tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.
Select Geometry	The **Select Geometry** tool guide allows you to select faces that were not automatically found. Hold Ctrl to select multiple objects or box select in the design area.
Exclude Problem	The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.
Complete	The **Complete** tool guide merges the highlighted faces.

Stitch Tool Options

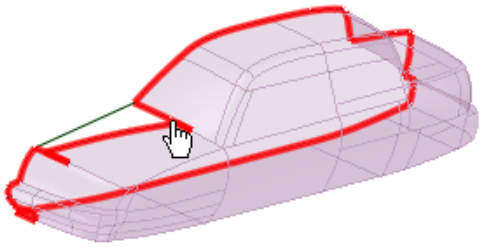
The following options are available:

Maximum distance	The maximum distance between faces that is automatically detected by the tool. The maximum distance value can be adjusted using a slider in the Options panel.
Check for coincidence	Set this option to check for coincidence within the model.
Maintain components	When you call a stitch, stitching will isolate each component and attempt to sew surface bodies within that component. It will not stitch surface bodies across components. This option is on by default.

11.2.2. Gaps Tool

The **Gaps** tool removes gaps between faces. These gaps are usually found on parts imported from other CAD systems when the native format allows faces to fit together loosely.

This tool only works for edges that are paired. Paired edges are edges that are within the maximum distance along their length or that share an end point and are within the maximum angle you set in the tool options. Use the **Missing Faces** tool if you need to repair a part with edges that are not paired. When a gap is adjacent to a larger hole, this tool only repairs the gap and not the hole.



This section contains the following topics:

11.2.2.1. Removing Gaps

1. Click **Gaps** in the Solidify group on the Repair tab.
   The tool will automatically detect and highlight gaps in an object, as shown in the image above.
2. Select the object(s) you want to change:
   - Use the **Select Problem** tool guide to select highlighted areas that you want to change. The cursor will change to a hand when you move the mouse over a problem area. This tool guide is active by default.
   - Use the **Select Geometry** tool guide to select edges that were not automatically detected.
3. Set the following options:
   - **Maximum Angle**: The maximum angle between any neighboring edge pairs in the potential missing face edge loop.
   - **Maximum Distance**: The maximum distance between any pair of edges in the potential missing face edge loop.
4. Use the controls in the Navigate ribbon group to view each problem one at a time before you fix it.
   - Click **Next** or **Previous** to step through and highlight each identified problem.
   - Select **Zoom to Fit** if you want to automatically zoom in on the problem in the design area when you click Next or Previous.
5. Click the **Complete** tool guide.
   The application adjusts neighboring faces to remove the highlighted gaps.

11.2.2.2. Gaps Tool Guides and Options

Gaps Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Problem	The **Select Problem** tool guide is active by default. This tool guide allows you to select problem areas that are automatically found by the tool.
Select Geometry	The **Select Geometry** tool guide allows you to select edges that were not automatically found. Hold Ctrl to select multiple edges or box select in the design area.

Exclude Problem	The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.
Complete	The **Complete** tool guide fills faces between all highlighted edges.

Gaps Tool Options

The following options are available:

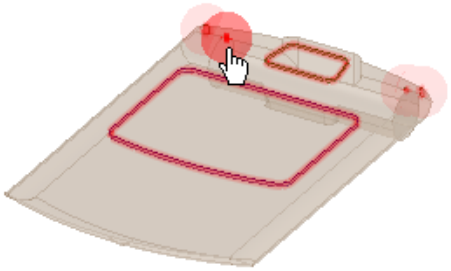
Maximum angle	The maximum angle between any neighboring edge pairs in the potential missing face edge loop.
Maximum distance	The maximum distance between any pair of edges in the potential missing face edge loop.

11.2.3. Missing Faces Tool

The **Missing Faces** tool automatically detects and fills missing faces on an object. This tool should be used to find missing faces on imported designs.

Use the **Fill** tool to fill faces when you know where the edges of the new face should be. Use the **Missing Faces** tool to identify missing faces and fill them automatically or choose which missing faces you want created.

As a precaution, if your design includes any open edge loops, first be certain that the loop(s) are not simply imported parts that may have been designed as a surface body with open regions.



This section contains the following topics:

11.2.3.1. Finding Missing Faces

1. Click **Missing Faces** in the Solidify group on the Repair tab.
   The tool will automatically detect and highlight missing faces, as shown in the image above.
2. Select the object(s) you want to change:
   - Use the **Select Problem** tool guide to select highlighted areas that you want to change. The cursor will change to a hand when you move the mouse over a problem area. This tool guide is active by default.
   - Use the **Select Geometry** tool guide to select edge loops that were not automatically detected.
3. Set the following options:
   - **Minimum Angle**: The minimum angle between all neighboring edge pairs in the potential missing face edge loop.

- **Minimum Distance**: The minimum distance between all of edges in the potential missing face edge loop.

  > **Note:** The tool looks for missing faces that meet *both* of the minimum measurements.

- **Fill**: Fix the missing face by extending neighboring faces until they intersect.
- **Patch**: Fix the missing face by creating a new face through the bounding edges of neighboring faces. To repair edge loops, select Patch and insert a new blended face. This new face will include each relevant tangent neighboring face, enabling you to close any selected edge loops with the smoothest possible inside surface, thereby creating a more realistic design.
- **Allow multiple faces**: Fix a hole in the surface with two or more missing faces.

4. Use the controls in the Navigate ribbon group to view each problem one at a time before you fix it.
   - Click **Next** or **Previous** to step through and highlight each identified problem.
   - Select **Zoom to Fit** if you want to automatically zoom in on the problem in the design area when you click Next or Previous.
5. Click the **Complete** tool guide.
   The application creates a new face using the method you selected in the Options panel.

11.2.3.2. Missing Faces Tool Guides and Options

Missing Faces Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Problem	The **Select Problem** tool guide is active by default. This tool guide allows you to select problem areas that are automatically found by the tool.
Select Geometry	The **Select Geometry** tool guide allows you to select edges between faces that were not automatically found. Hold Ctrl to select multiple edges or box select in the design area.
Exclude Problem	The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.
Complete	The **Complete** tool guide creates new faces in areas that are highlighted.

Missing Faces Tool Options

The following options are available:

Minimum angle	The minimum angle between all neighboring edge pairs in the potential missing face edge loop.
Minimum distance	The minimum distance between all of edges in the potential missing face edge loop.
Fill	Fix the missing face by extending neighboring faces until they intersect.

Patch	Fix the missing face by creating a new face through the bounding edges of neighboring faces. Faces with a high level of detail on a boundary may have one or more short edges. To make a smooth patch on a boundary, short edges are automatically removed when patching.
Allow multiple faces	Fix a hole in the surface with two or more missing faces.

11.3. Fix

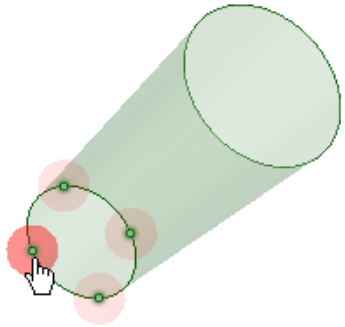
Tools in the **Fix** group fix problems that may exist in imported data, which can exist in a solid model but cause problems when you edit or modify the geometry.

Split Edges	Use the **Split Edges** tool to remove extraneous points on edges.
Extra Edges	Use the **Extra Edges** tool to detect and remove edges that are not needed to define the model.
Duplicates	Use the **Duplicates** tool to detect and fix duplicate faces.

This section contains the following topics:

11.3.1. Split Edges Tool

The **Split Edges** tool detects and merges coincident edges that do not mark the boundaries of new faces.



This section contains the following topics:

11.3.1.1. Consolidating Split Edges

1. Click **Split Edges** in the Fix group on the Repair tab.
   The tool will automatically detect and highlight split edges.
2. Select the object(s) you want to change:
   - Use the **Select Problem** tool guide to select highlighted areas that you want to change. The cursor will change to a hand when you move the mouse over a problem area. This tool guide is active by default.
   - Use the **Select Geometry** tool guide to select edges that were not automatically detected.
3. (Optional) Select **Suppress topologies** to suppress the entities from your selection.

4. Use the controls in the Navigate ribbon group to view each problem one at a time before you fix it.
   - Click **Next** or **Previous** to step through and highlight each identified problem.
   - Select **Zoom to Fit** if you want to automatically zoom in on the problem in the design area when you click Next or Previous.

5. Click the **Complete** tool guide.
   The highlighted areas are consolidated by merging coincident edges as appropriate.

11.3.1.2. Split Edges Tool Guides and Options

Split Edges Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

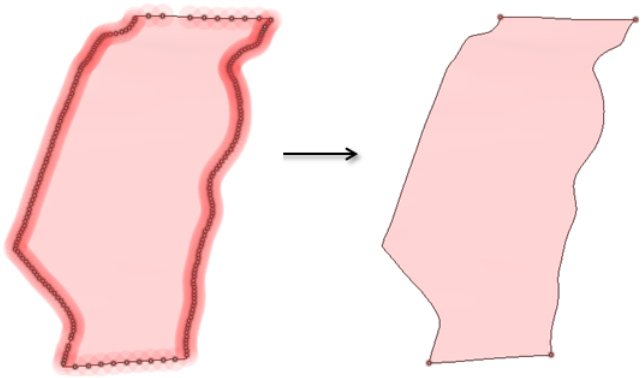
Select Problem	The **Select Problem** tool guide is active by default. This tool guide allows you to select problem areas that are automatically found by the tool.
Select Geometry	The **Select Geometry** tool guide allows you to select points that were not automatically found. Hold Ctrl to select multiple points or box select in the design area.
Exclude Problem	The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.
Complete	The **Complete** tool guide consolidates all highlighted areas.

Split Edges Tool Options

The following options are available:

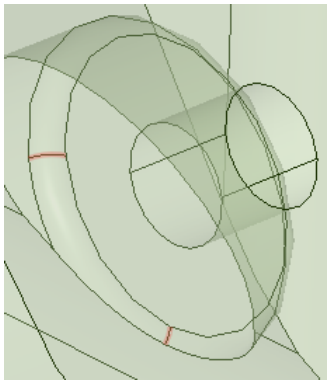
Maximum Length	Set the Maximum Length to search the model for split edges that are no longer than the specified length.
Minimum Angle	Use the Minimum Angle option to detect open edges on surfaces that are close to tangent. An open edge is defined as an edge on the outside boundary of a surface and not connected to another surface. This option must be enabled for the tool to find the edges.
Suppress topologies	Select Suppress topologies to suppress the entities from your selection.

The image below shows an example of using the Minimum Angle option to find open edges that are close to tangent. After fixing the edges found, they are made tangent, resulting in continuous tangent edges.



11.3.2. Extra Edges Tool

The **Extra Edges** tool works like the **Merge Faces** tool but operates on edges. Instead of merging two faces by selecting the faces, you select the edges between faces to remove the extra edge and merge the faces.



You should only merge faces that are tangent or close to tangent. Otherwise, the results may not be what you expect.

> **Note:** The Extra Edges tool is only intended to be used as preparation for analysis. Merging faces simplifies the model by removing edges and makes the model more difficult to modify.

This section contains the following topics:

11.3.2.1. Removing Edges

1. Open a model which needs edges removed.
2. Click the **Extra Edges** tool in the Fix group of the Repair tab.
   Edges which can be removed are highlighted.
3. (Optional) Select **Suppress topologies** to suppress the entities from your selection.
4. Click the **Complete** tool guide to remove all edges at once.
5. You can also select areas to be excluded from fixing.
   The extra edges are removed and the faces merged as appropriate.

11.3.2.2. Extra Edges Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Problem	The **Select Problem** tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.
Select Geometry	The **Select Geometry** tool guide allows you to select edges that were not automatically found. Hold Ctrl to select multiple objects or box select in the design area.
Exclude Problem	The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.
Complete	The **Complete** tool guide merges the highlighted faces.

Extra Edges Tool Options

Suppress topologies	Select **Suppress topologies** to suppress the entities from your selection.

11.3.3. Duplicates Tool

The **Duplicates** tool detects and fixes duplicate faces on surfaces. The application highlights the duplicates and will remove them all, or you can select duplicates to exclude from being fixed. You can optionally choose to detect and fix duplicate faces from solids as well.

This section contains the following topics:

11.3.3.1. Finding and Fixing Duplicate Faces

1. Open a model which needs faces removed.
2. Select the **Duplicates** tool in the Fix group of the Repair tab.

   The tool highlights the Duplicate faces to be removed.
3. (Optional) Select **Allow face removal from solids**, if required.
4. Click the **Complete** tool guide to remove all duplicates at once.
5. You can also select areas to be excluded from fixing using the **Exclude Problem** tool guide and then click **Complete**.

   The application removes the duplicate faces.

11.3.3.2. Duplicates Tool Guides and Options

Duplicates Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Problem	The **Select Problem** tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.
Select Geometry	The **Select Geometry** tool guide allows you to select faces that were not automatically found. Hold Ctrl to select multiple objects or box select in the design area.
Exclude Problem	The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.
Complete	The **Complete** tool guide removes the highlighted faces.

Duplicates Tool Options

The following tool option is available:

Allow face removal from solids	Enables you to detect and fix duplicate faces from solids.

11.4. Fix Curves

Tools in the **Fix Curves** group fix problems that may exist in imported data, which cause problems when working with curves.

Fit Curves	Use the **Fit Curves** tool to improve selected curves by replacing them with lines, arcs, or splines.
Curve Gaps	Use the **Curve Gaps** tool to detect and fix gaps between curves.
Duplicate Curves	Use the **Duplicate Curves** tool to detect and remove duplicate curves.
Small Curves	Use the **Small Curves** tool to detect and remove small curves and fix the resulting gaps.

This section contains the following topics:

11.4.1. Fit Curves Tool

The **Fit Curves** tool attempts to create fewer and better curves to replace selected curves which may not be continuous or tangent. You can also replace a curve with simple line segments as in the example above.

This section contains the following topics:

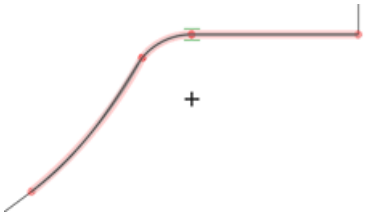
11.4.1.1. Fixing and Replacing Small Curves

1. Click the **Fit Curves** tool in the Fix Curves group of the Repair tab.
2. Select or box-select the curves you want to fix.
3. Set the Maximum distance option to determine the curves that will be found and how closely new curves will follow old curves.
4. Choose Fix Options to set the types of replacement curves to use.
   - **Lines**

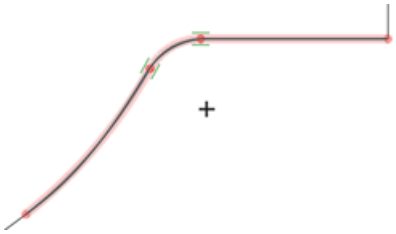
- **Arcs**
- **Splines**
- **Correct tangency**: Check this option to make arcs tangent that are otherwise slightly non-tangent.

5. Small curves are highlighted and new curve start and end points are indicated.
6. Click problem areas to fix them or click the **Complete** tool guide.

The images below show the effect of the **Check tangency** option.



Without the **Check tangency** option checked, only one tangency is found.



With the Check tangency option checked, a slightly non-tangent arc is made tangent.

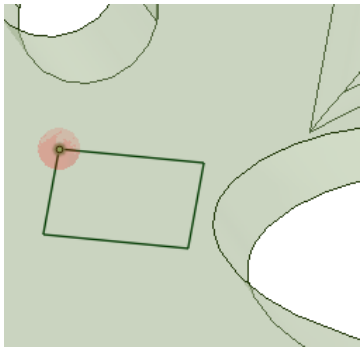
11.4.1.2. Fit Curves Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Geometry	The **Select Geometry** tool guide allows you to select geometry that were not automatically found. Hold Ctrl to select multiple objects or box select in the design area.
Complete	The **Complete** tool fixes the curves.

11.4.2. Curve Gaps Tool

The **Curve Gaps** tool finds any gaps between curves and closes them. It can extend and/or move lines to close the gaps.



This section contains the following topics:

11.4.2.1. Fixing Curve Gaps

1. Click the **Curve Gaps** tool in the Fix Curves group of the Repair tab.
2. Set the Maximum distance option or keep the default value
3. Gaps within the maximum distance tolerance are highlighted.
4. Click problem areas to fix them or click the **Complete** tool guide.

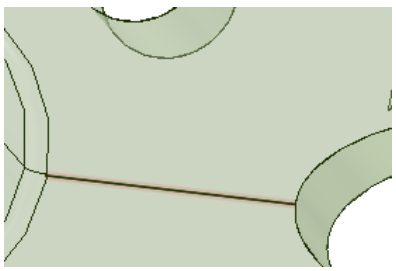
11.4.2.2. Curve Gaps Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Problem	The **Select Problem** tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.
Select Geometry	The **Select Geometry** tool guide allows you to select geometry that were not automatically found. Hold Ctrl to select multiple objects or box select in the design area.
Exclude Problem	The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.
Complete	The **Complete** tool repairs the gaps.

11.4.3. Duplicate Curves Tool

The **Duplicate Curves** tool finds any overlapping curves and deletes any extra curves.



This section contains the following topics:

11.4.3.1. Fixing Duplicate Curves

1. Click the **Duplicate Curves** tool in the Fix Curves group of the Repair tab.
2. Duplicate curves are highlighted.
3. Click problem areas to fix them or click the **Complete** tool guide.

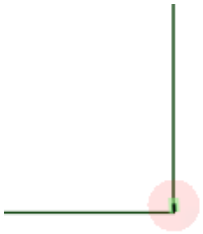
11.4.3.2. Duplicate Curves Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Problem	The **Select Problem** tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.
Select Geometry	The **Select Geometry** tool guide allows you to select geometry that were not automatically found. Hold Ctrl to select multiple objects or box select in the design area.
Exclude Problem	The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.
Complete	The **Complete** tool removes the duplicate curves.

11.4.4. Small Curves Tool

The **Small Curves** tool finds any curves smaller than a specified length.



This section contains the following topics:

11.4.4.1. Finding Small Curves

1. Click the **Small Curves** tool in the Fix Curves group of the Repair tab.
2. Set the Maximum length option or keep the default value.
3. Curves within the maximum length tolerance are highlighted.
4. Click the problem areas to fix them or click the **Complete** tool guide.

11.4.4.2. Small Curves Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Problem	The **Select Problem** tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.
Select Geometry	The **Select Geometry** tool guide allows you to select geometry that were not automatically found. Hold Ctrl to select multiple objects or box select in the design area.
Exclude Problem	The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.
Complete	The **Complete** tool removes the small curves and repairs the gaps they leave.

11.5. Adjust

Tools in the **Adjust** group help you modify characteristics of your design that will affect analysis.

Merge Faces	Use the **Merge Faces** tool to combine two or more faces into a single face.
Small Faces	Use the **Small Faces** tool to remove small faces from your design.
Simplify	Use the **Simplify** tool to simplify complex faces and curves.
Inexact Edges	Use the **Inexact Edges** tool to repair edges that do not fit together precisely.

This section contains the following topics:

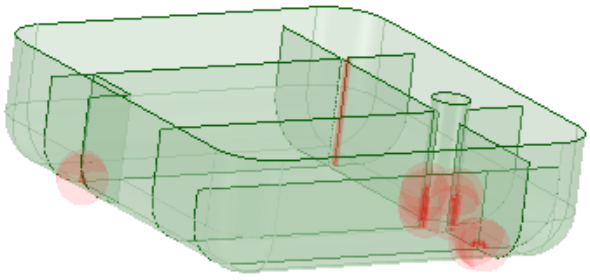
11.5.1. Merge Faces Tool

The **Merge Faces** tool replaces two or more neighboring faces with a single new face that closely fits the original faces. Use this tool to simplify a model before you export it for analysis. Merging faces can result in a smoother mesh on the solid.

You should only merge faces that are tangent or close to tangent, otherwise, the results may not be what you expect.

You cannot select edges when using the **Merge Faces** tool. You should use the Fill tool on the Design tab when you need to select an edge and a face to fill in a missing face. The Missing Faces and Gaps tools on the **Prepare** tab also perform this function.

> **Note:** The merge faces tool is only intended to be used as preparation for analysis. Merging faces simplifies the model by removing edges and makes the model more difficult to modify.



This section contains the following topics:

11.5.1.1. Merging Neighboring Faces

1. Click **Merge Faces** in the Adjust group of the Repair tab.
2. Select two or more faces you want to merge.
3. If you want to retain tangency between the new face and an existing face, use the **Maintain Tangency** tool guide and select the face with which you want to retain tangency.
4. Click the **Merge Faces** tool guide.

   The application replaces the selected faces with a new face.

11.5.1.2. Merge Faces Tool Guides

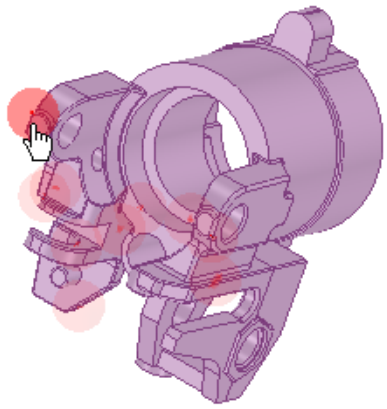
The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select	The **Select** tool guide is active by default. You can also select by drawing a box in the design window.
Maintain Tangency	The **Maintain Tangency** tool guide allows you to select faces with which you want to retain tangency. The new face will be tangent to the face(s) you select. You can also use Alt+click to select faces for tangency.
Complete	The **Complete** tool guide replaces the selected faces with a single face.

11.5.2. Small Faces Tool

The **Small Faces** tool detects and removes small and sliver faces in your design. You may want to remove these faces before you export the design for analysis if they will have a negligible impact on the analysis accuracy but a significant impact on its speed.

If the small face is tangent to a neighboring face, the tool will merge the small face with the neighboring face. If no neighboring face is tangent, the tool will extend neighboring faces to remove the small face.



This section contains the following topics:

11.5.2.1. Removing Small and Sliver Faces

1. Click **Small Faces** in the Adjust group of the Repair tab.

The tool automatically finds small faces in the active component and highlights them in the design area, as shown in the image above.

2. Select the object(s) you want to change:
   - Use the **Select Problem** tool guide to select highlighted areas that you want to adjust. The cursor will change to a hand when you move the mouse over a problem area. This tool guide is active by default.
   - Use the **Select Geometry** tool guide to select faces that were not automatically detected.
3. Set the **Maximum area** and **Maximum width** in the Options panel. Faces that are smaller than the maximum area and thinner than the maximum width will be found and highlighted for adjustment.

   You can alternatively use the **Select Sample Face** tool guide to select a sample face to set the maximum area.
4. (Optional) Select **Suppress topologies** to suppress the entities from your selection.
5. Use the controls in the Navigate ribbon group to view each problem one at a time before you fix it.
   - Click **Next** or **Previous** to step through and highlight each identified problem.
   - Select **Zoom to Fit** if you want to automatically zoom in on the problem in the design area when you click Next or Previous.
6. Click the **Complete** tool guide.

   The selected faces are removed.

11.5.2.2. Small Faces Tool Guides and Options

Small Faces Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Sample Face	The **Select Sample Face** tool guide allows you to select a sample face to set the maximum area.
Select Problem	The **Select Problem** tool guide is active by default. You must select objects that you want to adjust in the model tree while this tool is active.
Select Geometry	The **Select Geometry** tool guide allows you to select small and sliver faces that were not automatically detected. The face will only be removed if it is smaller than the maximum area and/or width specified.
Exclude Problem	The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.
Complete	The **Complete** tool guide removes all the faces that are highlighted.

Small Faces Tool Options

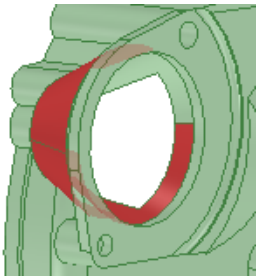
The following options are available:

Maximum area	Faces that are smaller than the maximum area will be found and highlighted for adjustment.
Maximum width	Faces that are thinner than the maximum width will be found and highlighted for adjustment.

Suppress topologies	Select **Suppress topologies** to suppress the entities from your selection.

11.5.3. Simplify Tool

The **Simplify** tool examines a design and simplifies complex faces and curves into planes, cones, cylinders, lines, arcs, etc. This automates the one-by-one simplify capability found in the **Replace** tool.



This section contains the following topics:

11.5.3.1. Simplifying a Design

1. (Optional) Select the faces you want to simplify.

   You may want to pre-select faces on large designs because it can be easier to simplify a region at a time.
2. Click **Simplify** in the Adjust group of the Repair tab.
3. (Optional) Click the **Select Geometry** tool guide to select faces to simplify that aren't automatically detected.
4. Click the **Select Problem** tool guide and select a highlighted face to simplify the face or click the **Complete** tool guide to simplify all highlighted faces.

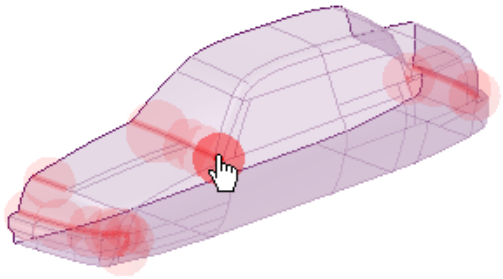
11.5.3.2. Simplify Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Problem	The **Select Problem** tool guide is active by default. This tool guide allows you to select problem areas that are automatically found by the tool.
Select Geometry	The **Select Geometry** tool guide allows you to select faces and curves that were not automatically found. Hold Ctrl to select multiple objects or box select in the design area.
Exclude Problem	The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.
Complete	The **Complete** tool guide replaces all the highlighted objects with simplified faces and curves.

11.5.4. Inexact Edges Tool

The **Inexact Edges** tool finds and repairs edges that have been inaccurately defined and do not meet precisely. These types of edges are usually found in designs imported from other CAD systems, particularly from conceptual design systems.



This section contains the following topics:

11.5.4.1. Fixing Inexact Edges

1. Click **Inexact Edges** in the Fix group of the Repair tab.
   The tool will automatically detect and highlight gaps in an object, as shown in the image above.
2. Select the object(s) you want to change:
   - Use the **Select Problem** tool guide to select highlighted areas that you want to change. The cursor will change to a hand when you move the mouse over a problem area. This tool guide is active by default.
   - Use the **Select Geometry** tool guide to select edges that were not automatically detected.
3. Use the controls in the Navigate ribbon group to view each problem one at a time before you fix it.
   - Click **Next** or **Previous** to step through and highlight each identified problem.
   - Select **Zoom to Fit** if you want to automatically zoom in on the problem in the design area when you click Next or Previous.
4. Click the **Complete** tool guide.
   The application redefines the edges so they meet precisely.

11.5.4.2. Inexact Edges Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Problem	The Select Problem tool guide is active by default. This tool guide allows you to select problem areas that are automatically found by the tool.
Select Geometry	The **Select Geometry** tool guide allows you to select edges that were not automatically found. Hold Ctrl to select multiple edges or box select in the design area.
Exclude Problem	The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.

Complete	The **Complete** tool guide recalculates all the highlighted edges to increase the precision between faces or surfaces.

11.6. Inspect

Select a tool from the **Inspect** ribbon group to check for problems in your design.

11.6.1. Check Geometry Tool

You can use the Check Geometry tool to look for any geometry errors on visible bodies in your model. Select errors and warnings in the **Check Geometry** panel to repair.

> **Note:** The **Check Geometry** tool will not work on lightweight bodies. See General Settings for File Import and Export on page 95 for more information about lightweight file formats.

To check the geometry:

1. Select parts in your model and click **Check Geometry** in the ribbon of the **Repair** tab.

   Alternatively, right-click the visible body in the model tree and select **Check Geometry** from the context menu.

   A **Check Geometry** panel will open listing any errors, warnings, and informational messages.

   > **Note:** If you have not selected any geometry prior to clicking **Check Geometry** in the ribbon, the application will check all visible bodies for problems.

2. In the **Check Geometry** panel, you can perform the following actions:
   a. Use **Zoom to fit** to help you find the location of the issues selected in the list by automatically zooming in on the area. You can also use **Hide others** to see only the selected problem area.
   b. Use **Repair all** to attempt to repair all the issues listed in the panel. You could also select individual bodies and click **Repair** from the semi-halo to attempt to repair the issues related to the selected body.
   c. An out-of-date status will appear indicating that the issues listed in the panel should be re-checked. **Re-check geometry** to update the list of issues.
   d. You may **Create named selections** of the issues by selecting one or more messages in the panel and using the semi-halo.
   e. Click **Close** when you are finished.

11.6.1.1. Check Geometry Tool Options

The following options are available:

Repair all	Automatically repairs all geometry problems listed in the panel.
Repair	Automatically repairs the selected problem area.
Re-check geometry	Re-checks the selected geometry and updates the list in the panel.
Zoom to fit	Automatically zooms in on the problem area in the model.

Hide others	Hides all other geometry except for the selected problem area(s).
Create named selection	Creates a named selection for the problem geometry.

12: Preparing Designs for Analysis

The Discovery application can import a variety of native and neutral CAD formats, but you may need to prepare this data for export and for CAE. The **Prepare** tab includes tools you can use to prepare your designs for export and analysis.

This section contains the following topics:

12.1. Beams and Shells

Use the tools in the **Beams and Shells** group to create and edit beam objects. You can also create midsurfaces and extend or trim surfaces and merge them with nearby surfaces.

> **Note:** While beams and shells are available for setup and export to other tools, they are currently excluded from simulation in the Discovery application.

This section contains the following topics:

12.1.1. Beams

A beam is a 1D representation of a 3D solid in 3D space. In essence, a beam is a slender geometric object with a constant cross-section. When your design goals include rotation bending, using a beam element offers a computationally efficient solution in comparison to solid or surface bodies. Beams can be combined with surface bodies and volume bodies, or your model can comprise only of beams. In the Discovery application, you can:

- Assign to curves (lines or edges) or body edges an existing beam profile from a supported library to specify its appearance, dimensions, and material properties. You can filter based on code (AISC, EUROCODE) or beam shape (I, Channel, L, etc.). You can also create a new beam profile or edit an existing beam profile and save your beam profile to a local library.
- In addition to selecting curves or body edges, you can also define continuous beam paths along selected points by linking multiple vertices in sequence to create a curve. You can also create precise connections between structural elements by linking two vertex pairs to generate a curve.
- Rather than assigning an existing beam profile, you can also create a new beam profile or edit an existing beam profile and save your beam profile to a local library.
- Change the orientation of the Beam.
- Automatically connect beams to adjacent beams based on a tolerance distance.
- Convert existing constant-section 3D solids to beams and centrelines. You can then edit the beam properties, rename the profile, or assign the beam a profile from standard libraries instead.
- Split beams where one beam end lies in the middle of another beam.

> **Note:** Beams cannot be created in a component tab.

Note: Beams sections also include the material property definition, as these are typically defined by the code of practice and are typically manufactured and purchased from a third-party supplier. For more information on material properties, see Material Properties on page 741.

This section contains the following topics:

12.1.1.1. Assigning an Existing Beam Profile to a Line or Edge

To assign a beam profile:

1. From the ribbon in the **Prepare** tab, select **Assign** from the **Beams and Shells** group.
2. Select curves (lines or edges) or body edges in your model.

> **Note:** You can also define beam geometry by linking multiple vertices in sequence to create a curve or generating a curve between two vertex pairs.

3. Select a beam profile from the dropdown on the right side of the HUD.

   Beam profiles consist of pre-defined dimension, material, and cross-section properties. Click the filter **Show profile types** to narrow down the list of profiles. You can filter based on code (AISC, EUROCODE) or beam type (I, Channel, L, etc.).

   The Discovery application supports several standard libraries of beam profiles. You can specify the appearance, dimensions, and material properties of a segment by assigning a beam profile. Any beam profiles created when 3D solids are converted are also listed.

   If you've previously created a custom beam profile and saved it to a local library, it will also be available from the drop-down list.

4. Optionally, you can modify the beam properties, then, if desired, you can save the modified profile by selecting **Save to local library**. For more information on local libraries, see Saving a Custom Beam to a Local Library on page 574.

> **Note:** You can remove an assigned profile assignment from a beam. Right-click the beam assignment in the Physics Tree and select **Delete**.

This section contains the following topics:

12.1.1.1.1. Adding a Custom Beam Profile

You have the option of defining properties for a beam either by selecting a beam profile from a standard library, which you can then edit, or creating a custom beam profile.

To create a custom beam profile:

1. From the ribbon in the Prepare tab, select **Assign** from the **Beams and Shells** group.
2. Select a line segment in your model.
3. From the dropdown list of profiles, select **New Beam Profile**. The application then populates the beam profile with default properties for the selected type.
4. Specify the appearance, dimensions, and material properties of the selected segment.
5. Optionally, you can save the modified profile by selecting **Save to local library**. For more information on local libraries, see Saving a Custom Beam to a Local Library on page 574.

> **Note:** You can remove an assigned profile assignment from a beam. Right-click the beam assignment in the Physics Tree and select **Delete**.

This section contains the following topics:

12.1.1.1.2. Saving a Custom Beam to a Local Library

You can save new and modified beam profiles to a local library. The default local library folder is set in the Physics Settings.

To save a custom beam profile:

1. For **Name** under **User Defined Beam Library**, enter a beam name, or accept the default.
2. Select one or more libraries for the new profile. You can also create a new local library.
3. Click **Save to local library**.

To remove a beam profile from a local library, select it from the Assign HUD or physics tree and then deselect the library name(s) from which you want to remove the profile.

12.1.1.1.3. Assign Beam Tool Options

The following options are available. Note that not all options are available for all beam types.

Beam name	Create a new beam profile or select an existing profile from the list. Use Filter to narrow the list by code (AISC, EUROCODE) or beam type (I, Channel, L).
Show profile types	The filter narrows down the list of beam profiles. You can filter based on code (AISC, EUROCODE) or beam type (I, Channel, L, etc.).
Source	The code of practice defining the beam profile and material properties.
Profile	The general shape profile of the cross section (for example, "L" for angle sections).
Family	The family subclass of the section that fits within the profile (for example, Equal angle, Unequal angle).
Size classification	The declared size as defined by the code of practice. The actual dimensions may vary.
Treat as	The type of profile you want to create (Beam, Cable, Spring, etc.). This parameter is then passed on to the Ansys Mechanical application.
Material	The material used.
Grade	The grade of material used, as defined by the code of practice.

Dimensions

The following dimension properties are available. Not all properties are available for all beam types.

Section depth	The measured depth of the section.
Section width	The measure width of the section.
Web thickness	The thickness of the beam web.
Flange thickness	The thickness of the beam flanges.
Web radius	The radius of the beam web.
Flange radius	The radius of the beam flanges.
Kdet	The code specific dimension Kdet.

K1	The code specific dimension K1.

Profile Properties

The following profile properties are available.

Cross-section area	The cross-sectional area.
Ixx	The second moment of area about the sections primary (xx) axis of bending.
Iyy	The second moment of area about the sections secondary (yy) axis of bending.

User Defined Beam Data

The following options are available for saving your beam data to your local library.

Name	The name of the new beam profile to save to a local library.
Save to local library	Saves the new beam profile to the selected local library name(s). The location for local libraries is defined in the Physics Settings.
Local libraries	The defined local libraries in which you can save any new beam profiles.
Create a new library	Create a new local library in the default location specified in the Physics Settings.

12.1.1.2. Assigning a Beam Profile Using a Vertex Pair

You can define a curve as a beam by selecting two vertex pairs. This feature is used to create precise connections between structural elements by linking selected vertices. You can then assign an existing beam profile to the curve or create a new beam profile for the beam assignment.

1. From the ribbon in the **Prepare** tab, select **Assign** from the **Beams and Shells** group.
2. Select the **Create vertex pair** tool guide on the left side of the HUD.
3. Select two vertices to generate a curve and define it as a beam.
4. Select a beam profile from the dropdown on the right side of the HUD.

   Beam profiles consist of pre-defined dimension, material, and cross-section properties. Click the filter **Show profile types** to narrow down the list of profiles. You can filter based on code (AISC, EUROCODE) or beam type (I, Channel, L, etc.).

   The Discovery application supports several standard libraries of beam profiles. You can specify the appearance, dimensions, and material properties of a segment by assigning a beam profile. Any beam profiles created when 3D solids are converted are also listed.

   If you've previously created a custom beam profile and saved it to a local library, it will also be available from the drop-down list.

5. Optionally, you can modify the beam properties, then, if desired, you can save the modified profile by selecting **Save to local library**. For more information on local libraries, see Saving a Custom Beam to a Local Library on page 574.

> **Note:** You can remove an assigned profile assignment from a beam. Right-click the beam assignment in the Physics Tree and select **Delete**.

12.1.1.3. Assigning a Beam Profile Using a Vertex Chain

You can define a curve as a beam by linking multiple vertices in sequence. This feature is ideal for defining continuous beam paths along selected points. You can then assign an existing beam profile to the curve or create a new beam profile for the beam assignment.

1. From the ribbon in the **Prepare** tab, select **Assign** from the **Beams and Shells** group.
2. Select the **Create vertex pair chain** tool guide on the left side of the HUD.
3. Select a series of vertices to generate a curve and define your beam.
4. Select a beam profile from the dropdown on the right side of the HUD.

   Beam profiles consist of pre-defined dimension, material, and cross-section properties. Click the filter **Show profile types** to narrow down the list of profiles. You can filter based on code (AISC, EUROCODE) or beam type (I, Channel, L, etc.).

   The Discovery application supports several standard libraries of beam profiles. You can specify the appearance, dimensions, and material properties of a segment by assigning a beam profile. Any beam profiles created when 3D solids are converted are also listed.

   If you've previously created a custom beam profile and saved it to a local library, it will also be available from the drop-down list.

5. Optionally, you can modify the beam properties, then, if desired, you can save the modified profile by selecting **Save to local library**. For more information on local libraries, see Saving a Custom Beam to a Local Library on page 574.

> **Note:** You can remove an assigned profile assignment from a beam. Right-click the beam assignment in the Physics Tree and select **Delete**.

12.1.1.4. Extracting a Beam

To extract a beam:

1. From the ribbon in the **Prepare** tab, select **Extract**.

   To convert an existing 3D solid to a beam, the solid must have a constant cross-section. However, it can have tapered ends, notches, or cut-outs along its length.

2. Select the 3D solid.

   The Discovery application creates a beam profile and assignment for the new beam and lists it in the Physics tree. You can edit, rename, or delete the beam profile from there.

12.1.1.5. Changing a Beam Orientation

To change a beam's orientation:

1. From the ribbon in the **Prepare** tab, select **Orient**.

The Discovery application displays an **Orient** widget on the endpoint of the selected beam.

2. Change the orientation of the beam in the following ways:
   - Select and drag the blue arrow to rotate the beam.
   - Double click the arrow to rotate 90° clockwise.
   - Drag the green or red arrows to translate the Beam. You can translate away from the curve.
   - To allow for precise movements, press the space bar while dragging to set the focus to the dimension.
   - Orient the beam to an object using the **Orient to object** tool in the HUD.

12.1.1.6. Connecting a Beam

To connect a beam:

1. From the ribbon in the **Prepare** tab, select **Connect**.

   The Discovery application highlights endpoints and endpoint connections that can be extended or moved to connect to a nearby beam, within the specified tolerance distance.
2. In the right side of the HUD, enter the tolerance distance in the **Maximum distance** field.
3. Select **Free ends only** to ignore beam endpoints that already connect to a beam.
4. Connect the beams. Use **Select problem** to connect only the endpoints you select. Then select the highlighted beams you want to connect and exit the **Connect** tool without clicking **Complete**.
5. You can also optionally exclude beams by clicking **Exclude problem** and selecting the highlighted beams you want to exclude. Then click **Complete** to connect all other highlighted beams.

12.1.1.6.1. Connect Beam Tool Guides and Options

Connect Beam Tool Guides

The following tool guides are available:

Perform action on selected location	Connects only the endpoints you select. Then select the highlighted beams you want to connect and exit the **Connect** tool.
Select locations to ignore	Excludes endpoints from automatic connection.
Complete	Connects all endpoints that have not been excluded from automatic connection. Do not click if you have been selecting endpoints using the selection tool.

Connect Beam Tool Options

The following options are available:

Maximum distance	Enter the tolerance distance within which a gap is highlighted for connection.
Free ends only	Ignores beam endpoints that already connect to a beam.

12.1.1.7. Split Beams

The Split tool finds connections between solid (extracted) beams, where the end of one beam lies somewhere in the middle of another beam. You can choose to split all connections found, select specific connections, or exclude some connections.

For any connections found, the beam with the smaller cross section is split by the beam with the larger cross section. Beams with the same cross section are not highlighted.

When determining beams to split, the application disregards cross sections that are not the same but are of similar size. You can specify the percentage size difference used to consider any two beams to be similar.

You can also choose to ignore beams that meet at end points.

This section contains the following topics:

12.1.1.7.1. Splitting All Highlighted Beams

When you first activate the Split tool, eligible endpoint connections are highlighted. You can specify options to further clarify which connections should be highlighted, then accept the automatic selections.

1. From the ribbon in the Prepare tab, select **Split** from the **Beams and Shells** group. The application highlights any endpoint connections that can be split.
2. For **Ignore similar**, specify the percentage size difference used to consider two beams to be similar. When determining beam connections to split, the application uses this percentage to determine if it should disregard cross sections that are not the same but are of similar size.
3. Beams meeting at endpoints are disregarded. If you prefer that they're included, de-select the **Ignore end-end cases** option.
4. Click **Complete** to split all highlighted endpoint connections.

12.1.1.7.2. Selecting Beams to Split

When you first activate the Split tool, eligible endpoint connections are highlighted. You can specify options to further clarify which connections should be highlighted, then select which of the endpoint connections you want to split.

To select beams for splitting:

1. From the ribbon in the Prepare tab, select **Split** from the **Beams and Shells** group. The application highlights any endpoint connections that can be split.
2. For **Ignore similar**, specify the percentage size difference used to consider two beams to be similar. When determining beam connections to split, the application uses this percentage to determine if it should disregard cross sections that are not the same but are of similar size.
3. Beams meeting at endpoints are disregarded. If you prefer that they're included, de-select the **Ignore end-end cases** option.
4. Ensure the **Select** tool is activated, then choose the endpoint connections you want to split. You can either individually select endpoint connections or use box-select.

   > **Note:** If you individually select endpoint connections, they are split when selected, and you do not have to click Complete.

5. If using box-select, click **Complete** to split the selected endpoint connections.

12.1.1.7.3. Excluding Beams to Split

When you first activate the Split tool, eligible endpoint connections are highlighted. You can specify options to further clarify which connections should be highlighted, then select which of the highlighted endpoint connections you do not want to split. This feature is useful in a large design where there are many endpoint connections, and you want most of them split.

To exclude any highlighted beams from being split:

1. From the ribbon in the Prepare tab, select **Split** from the **Beams and Shells** group. The application highlights any endpoint connections that can be split.
2. For **Ignore similar**, specify the percentage size difference used to consider two beams to be similar. When determining beam connections to split, the application uses this percentage to determine if it should disregard cross sections that are not the same but are of similar size.
3. Beams meeting at endpoints are disregarded. If you prefer that they're included, de-select the **Ignore end-end cases** option.
4. Select the **Exclude** tool and select any endpoint connections you do not want to split.
5. Click **Complete** to split the endpoint connections that remain highlighted.

12.1.1.7.4. Split Beam Tool Guides and Options

Split Beam Tool Guides

The following tool guides are available:

Select endpoint connections to split	In the geometry, select endpoint connections you want to split. Beam connections that are highlighted but not selected will not be split.
Exclude endpoint connections	In the geometry, select endpoint connections you want to exclude. All other highlighted connections will be split.
Complete	Split all selected problem areas.

Split Beam Tool Options

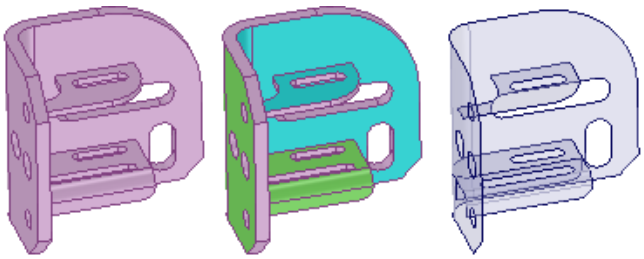
The following tool options are available:

Ignore similar	Directs the application, when determining endpoint connections to split, to disregard any cross sections that are not the same but are of similar size. You specify the percentage size difference used when considering beams to be similar.
Ignore end-end cases	Directs the application to disregard any beams that meet at end points.

12.1.2. Midsurface Tool

The **Midsurface** tool creates a surface midway between two offset faces. The midsurfaces are automatically extended or trimmed to adjacent faces, and the distance between the original faces is stored as a thickness property. You can use these midsurfaces for FE analysis.

Color highlighting shows you face pairs that have been selected, as shown below. The midsurface will be offset from the cyan faces. Green indicates that a face is paired with a cyan face. Unselected faces and faces without offsets are shown in the original color.



The thickness of the original model face offsets are stored as a property named **Thickness** in the **Midsurface** section of the **Properties** panel. This is a face property, so you must select the face in the Design window rather than in the model tree, even if it is a single face. You can change this property, and it is included in the Ansys data when it is sent out to Ansys via the Discovery application add-in.

The **Midsurface** tool detects and removes small faces of midsurfaces that are created when an edge is equal to half of the part thickness.

If the **Midsurface** tool finds missing faces because neither side can be offset, you will receive an error message in the error box that lists the faces. If the tool fails to create midsurface parts, the problem faces or edges are highlighted.

- Midsurfaces inherit the material properties of their parent components, but you can change the material properties for the midsurface object.
- Midsurfaces are shown in the model tree with a default name that includes the assigned thickness. When you rename the surface, the thickness is appended to the new name.
- You can change the thickness of a midsurface face, select the midsurface face in the Design window and change the **Thickness** property in the **Properties** panel.
- You can remove the Midsurface association by setting the **Thickness** to 0.
- When you right-click a Midsurface in the tree, you can select the source body using **Select** > **Midsurface Source**.

This section contains the following topics:

12.1.2.1. Making a Midsurface Body from a Regular Open Surface Body

1. Select the surface in the model tree.
2. Change the **Thickness** property in the **Properties** panel.

12.1.2.2. Creating Midsurface Faces by Selecting Offset Pairs

1. Click **Midsurface** in the Analysis group of the Prepare tab.
2. Select the **Use selected faces** option in the Options panel.

   This option automatically detects all offset face pairs on a body with one or more offset distances.
3. Select a face for which you want to create a midsurface.
4. Select a second face that makes an offset pair with the first face. All face pairs that have the same offset distance will be added to the selection.
5. Use the **Add/Remove Faces** tool guide to remove detected face pairs or to add undetected face pairs.

   To add or remove a detected face pair, click either face in the pair.

Note:

- If the face has an offset pair that was previously selected, the pair will be deselected.
- You may need to remove a face from selection if it is paired incorrectly.
- The midsurface distance will be offset from this face the same distance as adjacent faces, regardless of any potential offset pair for this face.
- Two faces may be detected as a pair because they have the same offset distance as the face pairs to be midsurfaced. Select the cyan face to remove the pair from selection.
- A face pair that should be midsurfaced may not be detected because its offset pair is not a perfect offset of the first face. Select the face you want offset to add it to the selection. Its midsurface will be offset using the thickness of neighboring detected face pairs.

6. (Optional) Click the **Selected Faces** tool guide or hold the **Ctrl** key and select additional face pairs with a different offset distance.

   When you add face pairs, all face pairs with the same offset distance will be added to the selection.

7. Click the **Complete** tool guide when you are finished selecting faces.

   When you successfully create a midsurface, the solid will become semi-transparent and the surface will be opaque until you select a different tool or clear your selection.

   Midsurface bodies are created in a component in the model tree, and are named using the name of the original object and appended with MidsurfaceN, where N is a unique number.

12.1.2.3. Creating Midsurface Faces Based on an Offset Range

1. Click **Midsurface** in the Define group on the Prepare tab.
2. Select **Use range** in the Options panel.

   The face pairs with offset distances within the range are automatically detected. If you have face pairs selected, the range will automatically change to include the offset distance.

3. Select the body for which you want to detect offset face pairs.
4. Change the **Minimum thickness** and **Maximum thickness** values in the Options panel as needed.

   Face pairs within this range will be selected, while face pairs outside this range will be removed from the selection.

5. Click the **Complete** tool guide when you are finished selecting faces.

12.1.2.4. Midsurface Tool Guides and Options

Midsurface Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Faces	The **Select Faces** tool guide is active by default. This tool guide allows you to select a pair of offset faces, and all other face pairs with the same offset distance are automatically detected.

Add/Remove Faces	The **Add/Remove Faces** tool guide allows you to select additional faces to offset or remove detected face pairs from the selection.
Swap Sides	The **Swap Sides** tool guide allows you to switch the face pairs. You may need to do this when you detect pairs with more than one offset distance, and the offset relationships are incorrectly detected.
Complete	The **Complete** tool guide creates the midsurface faces.

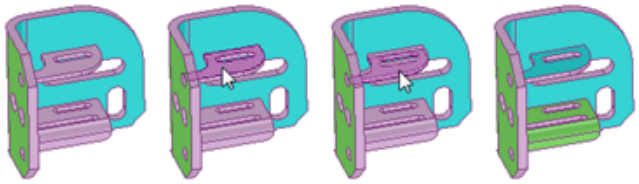
Midsurface Tool Options

The following options are available:

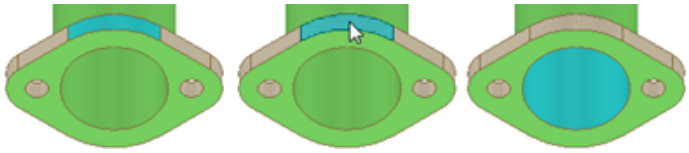
Use selected faces	Creates midsurfaces for only the faces you select.
Use range	Creates midsurfaces on all faces in the specified thickness range.
Thickness tolerance	Change the value of this option to detect offset spline faces with an offset value within the tolerance amount.
Create midsurfaces in	Select **Same component** to create the midsurfaces in the same component as the part you selected for midsurfacing. Select **Active component** to create the midsurfaces in the active component.
Create midsurfaces at	Select the location for the midsurface created. Choose from **Middle** (default), **Bottom** (green), or **Top** (cyan).
Group midsurfaces	Select this option to create midsurfaces in a new sub-component. Deselect the option to create the midsurface objects in the component you select in the option above (same component or active component).
Extend surfaces	Select this option to extend surfaces to another surface.
Trim surfaces	Select this option to trim surfaces to another surface.

12.1.2.5. Midsurfaces Examples

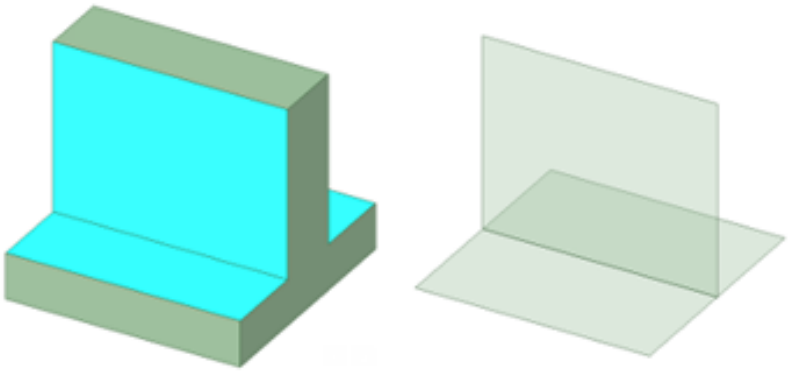
*Selecting an additional face pair with the **Use selected faces** option and the **Selected Faces** tool guide. All face pairs with the same offset are added to the selection.*



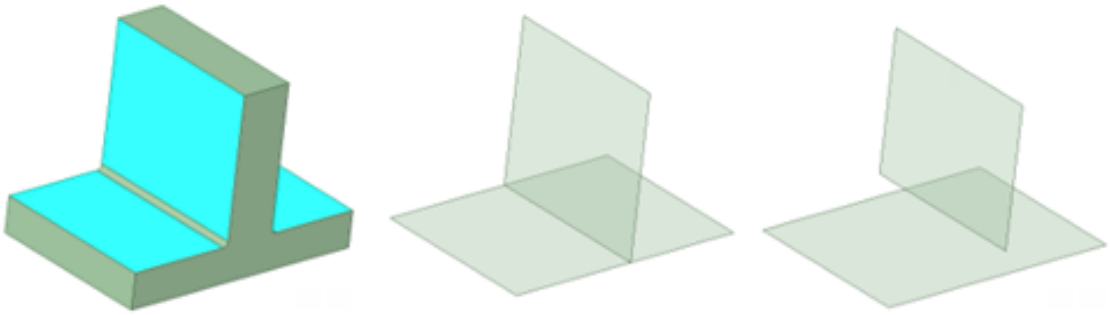
Removing a face that was automatically detected but not desired as a midsurface pair.



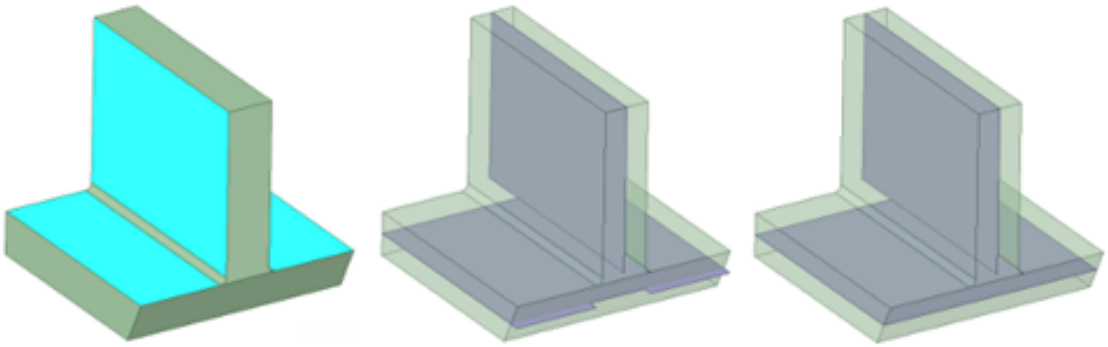
***Extend Surfaces** is **ON** by default. Because it is obvious in the above example, the surfaces meet even when **Extend Surfaces** is turned **OFF**.*



*When the edges are rounded, the surfaces meet with **Extend Surfaces** turned **ON**. They do not meet with **Extend Surfaces** turned **OFF**.*

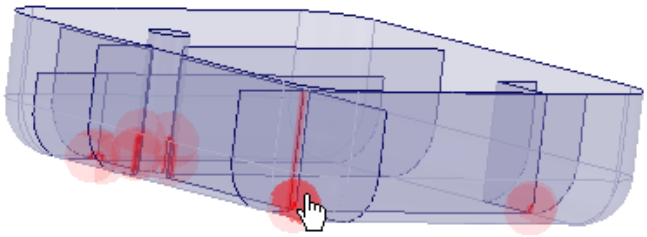


***Trim Surfaces** trims the surfaces to the extent of the original body. In the example above, the slope at the bottom shows two different results. In the second image with **Trim Surfaces** turned **OFF**, the midsurface will extend to the extent of the largest face of the pair. In the third image with **Trim Surfaces** turned **ON**, the midsurface is trimmed to the sloped face.*



12.1.3. Extend Tool

Use the **Extend** tool to extend or trim surfaces and merge them with nearby parts, or to extend or trim sketch curves. The tool automatically detects faces that can be extended or trimmed and highlights them, as shown below. Click a highlighted area to perform the extension or trimming. You can also select a surface that wasn't automatically detected and attempt to extend or trim it. This tool only works with surface parts and sketch curves.



You can also use the **Extend** tool to create a weld body between two surfaces. The edges of surface parts or sketch curves that can be used to create the weld body are highlighted automatically based on the Maximum distance specified. Select individual sites from the highlighted edges or curves to create welds manually, or choose to create welds at all sites found.

This section contains the following topics:

12.1.3.1. Extending or Trimming Adjacent Surfaces

1. Click **Extend** in the Prepare tab.

   The edges of surface parts or sketch curves that can be extended or trimmed are highlighted.

   Chains of tangent faces are automatically grouped to offset them together.
2. Change the following options:
   - **Maximum distance**: The maximum distance between surfaces. The tool will search for adjacent faces again if you change this value.
   - **Create weld body**: Creates a weld body between two surfaces. See Creating a Weld Body Between Surfaces on page 584 for details.
   - **Trim surfaces**: Controls whether or not surfaces may be trimmed in addition to extended.
   - **Partial intersections**: Controls whether or not faces that partially intersect are detected.
   - **Same body**: Allows a surface to be trimmed or extended by a face or edge on the same body.
   - **Extend to curves**: Controls whether or not surfaces are extended to curves, when possible.
   - **Merge after extend or trim**: Merges bodies, if possible, when you trim or extend an edge on one surface body up to a face or edge on another body.
3. If you want to extend or trim only some of the surfaces or sketch curves, click the highlighted spots.

   Hold **Ctrl** to select multiple edges.

   The surfaces or sketch curves are extended or trimmed when you click them.
4. Click **Complete** if you want to extend or trim all highlighted surfaces.

   Press **Esc** to cancel the operation.

12.1.3.2. Creating a Weld Body Between Surfaces

1. Select **Extend** in the Prepare tab.
2. Select **Create weld body** to create a weld body between two surfaces.

   The edges of surface parts or sketch curves that can be used to create the weld body are highlighted based on the **Maximum distance** specified. Chains of tangent faces are automatically grouped to offset them together.

3. If required, change the **Maximum distance** between surfaces. The tool will search for adjacent faces again if you change this value.
4. Select individual sites from the highlighted edges or curves, or click **Complete** to create welds at all sites found.

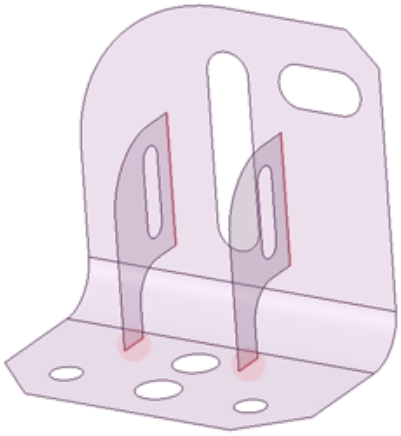
   To select individual sites manually:

   a) Use the **Select Edges** tool guide to manually select edges where the weld body will be created. You can Ctrl-select multiple edges or select chains or loops.
   b) (Optional) Use the **Select Target Faces** tool guide to select a target face where the weld body will attach.
   c) Click **Complete** to create the welds. Press Esc to cancel the operation.

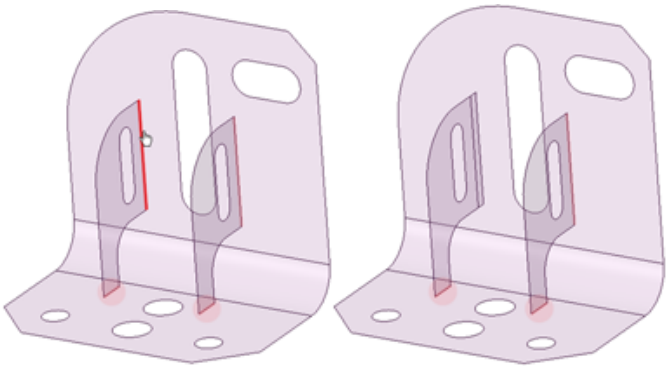
   The weld surface is added to the tree. You can add a thickness to a weld by selecting it and editing its Properties.

The following example shows how you can create welds.

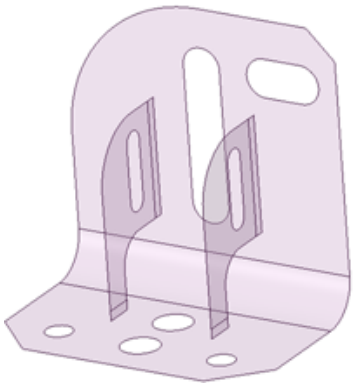
The edges of surface parts or sketch curves that can be used to create the weld body are highlighted based on the Maximum distance.



Select individual sites to create the welds.



Click Complete to create welds at all sites found.



12.1.3.3. Extend Tool Guides and Options

Extend Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Problem	The **Select Problem** tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.
Select Geometry	The **Select Geometry** tool guide allows you to select faces that were not automatically found. Hold Ctrl to select multiple objects or box select in the design area.
Exclude Problem	The **Exclude Problem** tool guide allows you to select problem areas for exclusion from selection or fixing.
Select Edges	The **Select Edges** tool guide allows you to manually select edges where the weld body will be created when the **Create weld body** option is selected.
Select Target Faces	The **Select Target Faces** tool guide allows you to select a target face where the weld body will be created when the **Create weld body** option is selected.
Complete	The **Complete** tool guide merges or trims the highlighted surfaces.

Extend Tool Options

The following options are available:

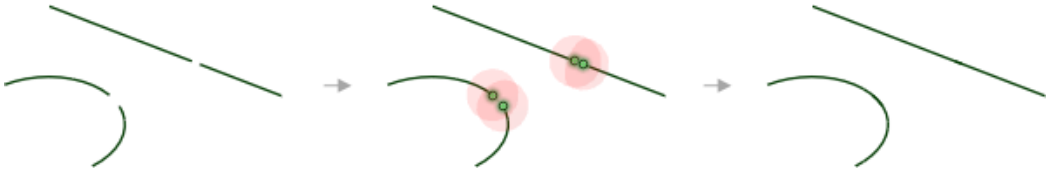
Maximum distance	The maximum distance between parts. The tool will search for adjacent faces again if you change this value.
Create weld body	Creates a weld body between two surfaces.
Merge after extend or trim	Merges bodies, if possible, when you trim or extend an edge on one surface body up to a face or edge on another body.
Trim surfaces	Controls whether or not surfaces may be trimmed in addition to extended.
Partial intersections	Controls whether or not faces that partially intersect are detected.
Same body	Allows a surface to be trimmed or extended by a face or edge on the same body.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

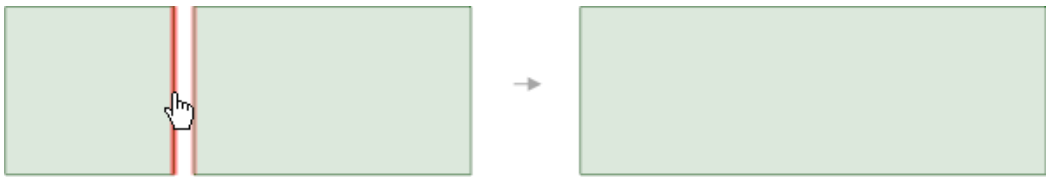
Extend to curves	Finds surfaces to extend to curves when the curve is in the same plane as the surface.

12.1.3.4. Extending/Trimming Examples

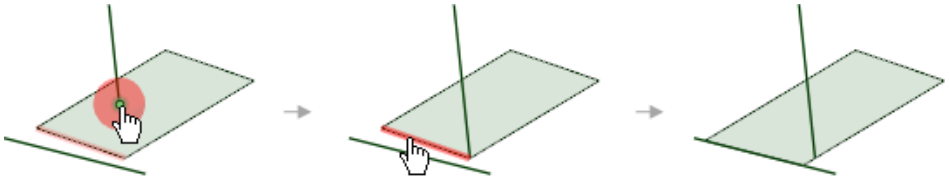
Curves are extended from endpoint to endpoint.



*The edges of separate surfaces are merged when you select the **Merge after extend or trim** option.*



*You can extend curves to surfaces and surfaces to curves when you select the **Extend to curves** option.*



12.2. Bolts

In a static stress analysis, you can define bolts to accurately account for the load transfer between parts in an assembly and therefore compute the stress and deformation of your assembly. For a natural frequency analysis, bolts account for the correct stiffness and mass of the bolted connection, computing the natural frequencies and mode shapes of your assembly including bolts. Note that the Bolts feature targets general assembly setup and is not intended for post-processing.

In the Discovery application, idealized bolts are represented by 3-D beam elements with a circular cross section, where the diameter of the cross section is equal to the shank diameter of the bolt. Idealized bolts can connect any number of solid parts, either using through holes or blind holes with mesh connections based on the head diameter automatically created between the ends of the idealized bolt and the associated solid geometry.

For solids connected with idealized bolts using through holes, the connection between the idealized bolt and the solid is based on creating a mesh connection between each end of the beam and the solid mesh on the top and bottom surfaces of the connection, with the mesh-based connection on the solid region extending out to the Head Diameter of the bolt. (The behavior of the mesh connection depends on the stage, where Explore uses a rigid connection and Refine uses a deformable connection.) Using the head diameter to create the connection distributes the resulting bolt forces over the area represented by the head diameter. Note that an idealized bolt is only connected to the solid geometry at the top and bottom of the bolt, so there is not any connection or interaction between the bolt shank and the solid surfaces representing the bolt hole.

For solids connected with idealized bolts using blind holes, the top connection is the same as a through hole, but the connection to the blind hole is created between the end of the idealized bolt and the inside surface of the blind

hole. (Similar to a through hole, the connection behavior depends on the stage, where Explore uses a rigid connection and Refine uses a deformable connection.)

In the Discovery application, you define a bolt by creating a new bolt type or by assigning an existing Bolt definition from a standard or user-defined library to two or more coaxial holes. If you define a tolerance for maximum hole coaxial offset, the application detects any bolts with misaligned geometry within that tolerance. A Bolt type specifies the grade and the bolt dimensions as well as material properties. When selecting an existing Bolt definition, click the **Filter** icon to narrow down the list of Bolts.

> **Note:** Bolts cannot be created in a component tab.

You can display bolts as 3D objects or as lines.

Once a bolt is defined, you can then add a Bolt Preload condition to model a bolt under pretension.

After solving, you can assess your bolts in the Connection Assessment Tool. This tool cycles through the bolts and shows whether they pass or fail the factor of safety.

> **Note:** For Topology Optimization, the cylindrical surfaces of bolts are considered protected regions, and bolt assignments are included in the simulation. However, bolt preloads are ignored.

This section contains the following topics:

12.2.1. Defining Bolts

1. From the ribbon in the **Prepare** tab, select **Assign** > **Bolts** from the **Connections** group.
2. Select a hole.

   The Discovery application automatically highlights all faces in line with that hole. If something is selected that you do not want included, control-click to de-select.

   You can select all similar bolt holes by clicking **Select similar bolt holes**.

   If you have separated holes that are coaxial, and you want to create bolts through those holes, click **Create bolts through separated coaxial holes**.
3. The application automatically selects a bolt type from a standard library based on the hole size. The bolt type specifies the grade and the bolt dimensions as well as material properties. However, you can also select a bolt type from the dropdown menu on the right side of the HUD or select **New Bolt** to define your own. When choosing from a library, click the **Show bolts with codes** icon to narrow down the list of bolts.
4. Optionally, under **Tolerance**, specify a tolerance for **Max hole coaxial offset** to ensure the application detects bolts with misaligned geometry within the specified tolerance. The default for this option is specified in the Physics Settings.
5. Optionally, you can save the new or modified bolt type by selecting **Save to local library**. For more information on local libraries, see Saving a Custom Bolt to a Local Library.
6. Click **Accept** to complete the bolt setup.

   Once a bolt is defined, you can add a Bolt Preload condition from the **Simulation** tab to model the bolt under pretension.

12.2.2. Bolt Tool Options

The following options are available:

Select a bolt	Select a Bolt type from a standard or user-defined library or select **New Bolt** to create a new bolt type. You can click the Filter icon to narrow down the list of bolt types when selecting from a standard library. Or, enter your bolt size in the field to assign a standard bolt size.
Show bolts with codes	Narrow down the list of bolts. You can filter based on standard libraries.
Create bolts through separated coaxial holes	Create bolts through coaxial holes, even if they're separated.
Select similar bolt holes	Automatically select all other bolt holes of a similar size to the one selected.

Dimensions

The following properties are available for defining your bolt dimensions.

Shank diameter	The diameter of the threadless part of the bolt's body.
Head diameter	The diameter of the bolt's head.
Head height	The height of the bolt's head.

Tolerance

The following option allows you to define a tolerance for detecting bolts.

Max hole coaxial offset	The tolerance for detecting bolts with misaligned geometry. The default for this option is specified in the Physics Settings.

User Defined Bolt Data

The following options are available for saving your defined beam to a local library.

Name	The name of the new bolt type to save to a local library.
Save to local library	Saves the new bolt type to the selected local library name(s). The location for local libraries is defined in the Physics Settings.
Local libraries	The defined local libraries in which you can save any new bolt types.
Create a new library	Create a new local library in the default location specified in the Physics Settings.

12.3. Fillet Welds

Fillet welds join two overlapping pieces (lap joint) or two pieces placed perpendicularly to each other (for example, a T or L-shaped joint). The application detects fillet welds on your model and enables you to specify properties for that weld. You can then save that weld definition to a local library.

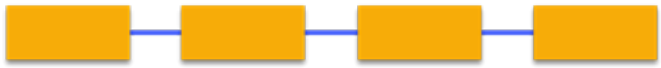
The Discovery application supports both continuous and intermittent welds. To enable intermittent welds to adjust automatically to geometry modifications, the application uses "minimum pitch" and "limit number" inputs, from which the actual pitch and number of welds is computed automatically. As a result, a single weld definition can be applied to multiple weld seams, resulting in a range of pitches and number of welds.

Note: Welds cannot be created in a component tab.

This section contains the following topics:

Limiting the Number of Welds

As the seam length increases, the number of welds is limited to ensure that no additional welds are introduced, even though they could be accommodated in the gaps between welds.

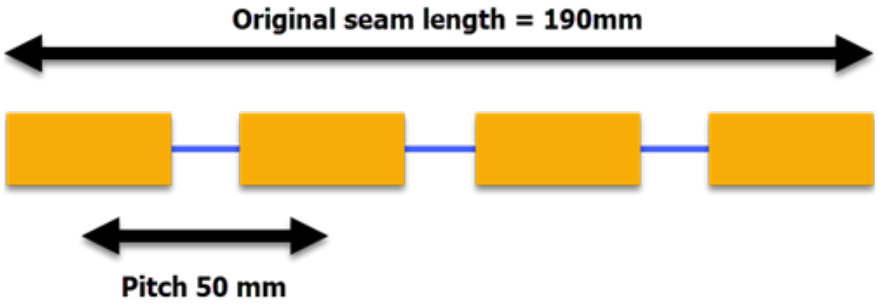


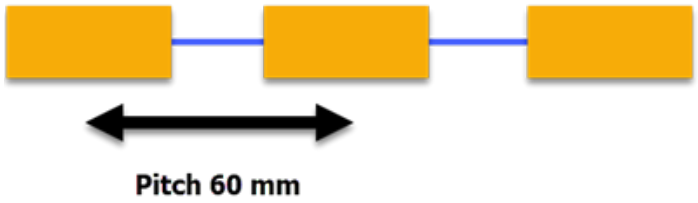


If a limit number is not defined, then the number of welds created is the most that fit while achieving the minimum pitch.

Ensure a Minimum Pitch along a Decreasing Seam Length

As the seam length decreases, ensure that a minimum pitch is achieved between welds.





As the length reduces, the minimum pitch of 40 mm can no longer be satisfied. As a result, the application automatically reduces the number of welds and increases the achieved pitch.

For intermittent welds, the minimum pitch, if defined, must be greater than the weld length. Otherwise, it defines a continuous weld. If minimum pitch isn't defined, it defaults to *1.5 x weld length*.

12.3.1. Defining Fillet Welds

The Assign Welds tool enables you to define properties for welds on a model. When you select geometry, the Discovery application detects fillet welds on that geometry. You can then select an existing fillet weld profile, or define a new one.

To define a new fillet weld profile:

1. From the ribbon in the **Prepare** tab, select **Assign** > **Welds** from the **Connections** group.
2. Click **Select generation locations** then select the geometry on which you want to define welds. You can select edges or edge loops, or bodies and faces. The application automatically detects all welds on that geometry based on the **Detection distance**.

   > **Note:** If the body and the face only overlap partially, the system can find only the edges that are in contact.

   You can also click **Select excluded locations** to select locations from which welds are excluded.
3. For **Size**, enter the size of the weld.
4. Select **Intermittent welds** if you want discontinuous welding. You can then define characteristics for your intermittent weld:
   a) For **Length**, enter the length of each weld in the intermittent weld.
   b) To specify a minimum distance between the welds, select **Minimum pitch** and enter the minimum distance between the center of one weld and the next. This value must be greater than the **Length**.
      If minimum pitch isn't defined, the minimum distance defaults to *1.5 x weld length*.
   c) To limit the number of welds, select **Limit number** and enter the maximum number of welds.

   Once the Length is defined, information about the intermittent weld(s) is displayed in the Pitch and Number fields.

   For more information on how the application uses minimum pitch and limit number values to automatically adjust intermittent welds to geometry modifications, see Fillet Welds on page 589.
5. To modify the default weld detection distance around selected geometry, expand the settings. Then, for **Detection distance**, enter **Min** and **Max** distance values.
6. Click **Save to local library** to save the weld definition to a local library for re-use.
   If the weld definition is already saved, and you want to remove it, select **Remove from local library**.
7. When you are finished, click **Accept**.

Once a weld profile has been created, you can modify the name of the Weld profile by entering a new name in the **Name** field. You can then apply this weld profile to other welds.

12.3.2. Fillet Welds Tool Guides and Options

Fillet Welds Tool Guides

The following tool guides are available:

Select generation locations	This tool enables you to select geometry on which you want to define welds. The Discovery application automatically detects all welds on that geometry.
Select excluded locations	This tool enables you to select locations from which welds are excluded.

Fillet Welds Tool Options

The following options are available:

Size	The size of the weld.
Intermittent welds	Select if you want discontinuous welding.
Length	The length of each weld in the intermittent weld.
Pitch	The distance between the center of one weld to the next. Display-only.
Number	The number of welds. Display-only.
Minimum pitch	Select to define the minimum distance between the center of one weld and the next. This value must be greater than the length. If minimum pitch isn't defined, the minimum distance defaults to *1.5 x weld length*. For more information on how the application uses minimum pitch and limit number values to automatically adjust intermittent welds to geometry modifications, see Fillet Welds on page 589.
Limit number	Select to limit the number of welds along the weld length. For more information on how the application uses minimum pitch and limit number values to automatically adjust intermittent welds to geometry modifications, see Fillet Welds on page 589.
Name	The name for the weld definition. You can enter this name after the weld definition is created.
Detection distance	Select to modify the default weld detection distance on selected geometry, then enter **Min** and **Max** distance values.
Save to local library	Select to save the weld definition to a local library for re-use.
Remove from local library	Select to remove the weld definition from a local library.

12.4. Changing the Prepare Tab Display

On the **Prepare** tab, you can use the **Show** dropdown to toggle the display of beam and shell profiles, bolts, and weld profiles in 3D. You can also display each material in the model in its own color.

12.5. Generate

Use the tools in the **Generate** group to create or subdivide bodies for analysis.

This section contains the following topics:

Volume Extract	Use the **Volume Extract** tool to create a solid based on an enclosed region within a part.
Enclosure	Use the **Enclosure** tool to detect coincident faces, edges, or vertices and imprint them to allow for mesh connections.
Split by Plane	Use the **Split by Plane** tool to split a part based on a plane.
Wrap	Use the **Wrap** tool to wrap geometry around a target model.
Imprint	Use the **Imprint** tool to imprint coincident faces.

12.5.1. Volume Extract Tool

Use the **Volume Extract** tool in the **Generate** group on the **Prepare** tab to create the volume that represents the internals of a part or assembly (including meshes). You can create multiple fluid zones at the same time if you have many seed faces selected.

> **Note:** Selecting a mix of solid and faceted entities is not supported.

A component named **Volume** containing the different fluid zone bodies named **Volume** is created in the model tree, and the bodies used to generate the volume are temporarily transparent when the volume is created.

> **Note:** If an edge will be removed when you use the **Volume Extract** tool, that edge will flash red and you will receive a warning message. This can happen when you select a capping edge that will be removed when it is merged with the rest of the model. The highlight identifies the problem so you can pick a different edge.

This section contains the following topics:

12.5.1.1. Creating a Volume Body

1. Click the **Volume Extract** tool in the **Generate** group of the **Prepare** tab.
2. If needed, click the **Select Faces** tool guide to select the faces that enclose the volume of the area. Alternatively, click the **Select Edges** tool guide to select the edge loops that enclose the volume of the area.

   Only select faces or edge loops that you intended to use to enclose the volume area. You can double-click to select more than one contiguous edge. You can also click a selected face or edge to deselect it. If you hover over a face that contains internal edge loops, the edge loops are highlighted. Click the face to select the highlighted edge loops.

   For faceted bodies, use **Shift+Double-Click** on a facet edge to select a faceted loop.
3. Click the **Select Seed Face** tool guide and select a face inside the volume area. The **Select Seed Face** tool guide is used to select a face to determine the inside of the enclosed volume. You need to use this tool guide to see a preview or to complete the selection for the **Volume Extract** tool to correctly identify the inside of the volume.
4. If necessary, click the **Select Cap Faces** tool guide to cap a face.
5. (Optional): Select **Preview inside faces** to activate the Preview slider. Your model immediately updates to paint a preview of the inner faces shown in red at 100%. Click the **Play** button to preview the faces in an animation, or use the slider to manually preview the faces. Move the slider to the left and right to see a progression of how the faces were captured.

   When previewing the faces, all Select tools and context menu Select tools, as well as all Volume Extract tool guides, are disabled until the Preview check box is turned off.
6. Click the **Complete** tool guide.

   The application creates a solid part based on the volume you defined.

12.5.1.1.1. Updating the Volume Body

Right-click the volume part (named **Volume** by default) in the model tree. Select one of the following commands:

- **Update Volume Body as Created** updates the volume body based on objects that were visible when the volume was created (regardless of their current visibility). Use this option for simple regeneration.

- **Update Volume Body in Context** updates the volume body based on objects that are currently visible in the design area. Use this option to remove bodies from the volume calculation.

12.5.1.2. Creating a Facet-Based Tolerant Volume Body

You can also create the volume for a single body with a leak or gaps or when there are gaps between the bodies in the model using the **Extract tolerant volume** option. Specify the **Maximum gap** allowed between bodies. The facet-based tolerant volume is extracted for bodies where the gaps are smaller than the **Maximum gap** size specified.

1. Click the **Volume Extract** tool in the **Generate** group of the **Prepare** tab.
2. Click the **Select Faces** tool guide to select the faces that enclose the volume of the area. Alternatively, click the **Select Edges** tool guide to select the edge loops that enclose the volume of the area.

   Only select faces or edge loops that you intended to use to enclose the volume area. You can double-click to select more than one contiguous edge. You can also click a selected face or edge to deselect it. If you hover over a face that contains internal edge loops, the edge loops are highlighted. Click the face to select the highlighted edge loops.

   For faceted bodies, use **Shift+Double-Click** on a facet edge to select a faceted loop.
3. Click the **Select Seed Face** tool guide and select a face inside the volume area. The **Select Seed Face** tool guide is used to select a face to determine the inside of the enclosed volume. You need to use this tool guide to see a preview or to complete the selection for the **Volume Extract** tool to correctly identify the inside of the volume.
4. If necessary, click the **Select Cap Faces** tool guide to cap a face.
5. Select **Extract tolerant volume** to extract the facet-based tolerant volume when the gaps are smaller than the maximum gap specified.

   Specify an appropriate value for the **Maximum gap**.

   The default **Maximum gap** tolerance is automatically computed based on the seed and capping face selections when the **Extract tolerant volume** option is selected.
6. (Optional): Select **Preview inside faces** to activate the Preview slider. Your model immediately updates to paint a preview of the inner faces shown in red at 100%. Click the **Play** button to preview the faces in an animation, or use the slider to manually preview the faces. Move the slider to the left and right to see a progression of how the faces were captured.

   When previewing the faces, all Select tools and context menu Select tools, as well as all Volume Extract tool guides, are disabled until the Preview check box is turned off.
7. Click the **Complete** tool guide.

   The application creates a faceted part based on the volume you defined.

12.5.1.3. Volume Extract Tool Guides and Options

Volume Extract Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Faces	The **Select Faces** tool guide activates by default when you select the Volume Extract tool, and selects faces whose edges seal an enclosed region. This is a shortcut to selecting all the edges that are detected in a face. You can **Ctrl+ Select** multiple seed faces, and then choose to select a different tool guide. Once you create a volume, the **Select Faces** tool guide resets by default.

Select Edges	The **Select Edges** tool guide selects edge loops that enclose a region. Edges are used to stop the propagation of face selection, emanating face-wise outward, starting at the selected seeds.
Select Cap Faces	The **Select Cap Faces** tool guide selects optional capping faces. This is important when an internal edge loop is either not simply fillable, or when you want some non-standard fill geometry to be created.
Select Seed Face	The **Select Seed Face** tool guide selects a face that lies within the volume you want to enclose. Use this tool guide to see a preview or to complete the selection for the **Volume Extract** tool to correctly identify the inside of the volume.
Complete	The **Complete** tool guide creates the volume solid based on the edges and seed face you select.

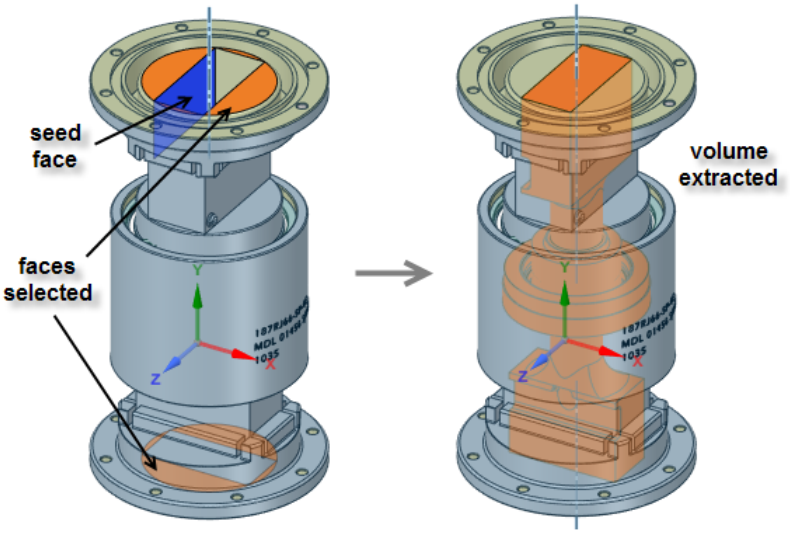
Volume Extract Tool Options

The following options are available:

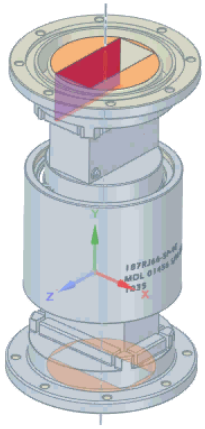
Merge created volume	Merges the created volume into the outer shell to effectively fill the inner cavity. This option is useful if the outer volume is of main interest.
Imprint capping edges	Leaves the capping faces intact so that the location of the inlets and outlets can be seen.
Create share topology	Creates share topology between the volume extracted and the input bodies.
Preview inside faces	Enables you to preview the selection of inner faces and detect leaks. Click the **Play** button to preview the faces in an animation, or use the slider to manually preview the faces.
Extract tolerant volume	Extracts the facet-based tolerant volume, when the gaps are smaller than the maximum gap specified.
Maximum gap	Specifies the maximum gap for extracting the facet-based tolerant volume.

12.5.1.4. Extracting Volume Examples

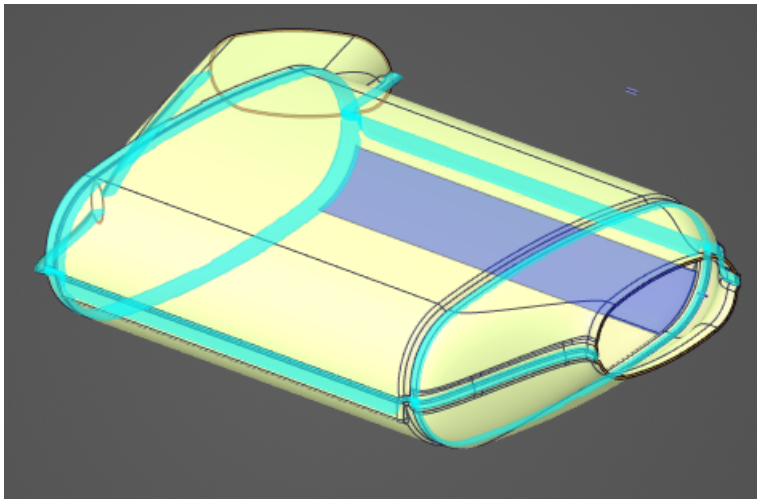
The faces enclosing the region of interest and a seed face are selected to extract the volume of interest.



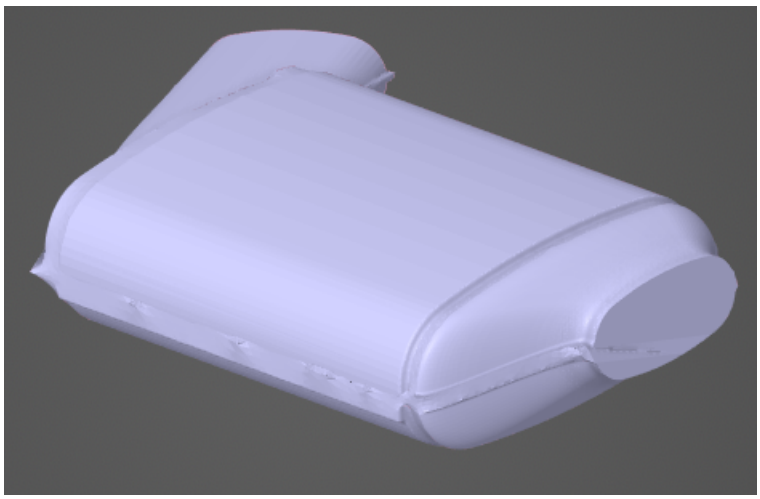
*Preview the selection of inner faces for the volume extraction and detect leaks. Click the **Play** button to preview the faces in an animation, or use the slider to manually preview the faces.*



This example shows the selected faces enclosing the region of interest and a seed face. The gaps between the bodies that are smaller than the Maximum gap specified are highlighted.



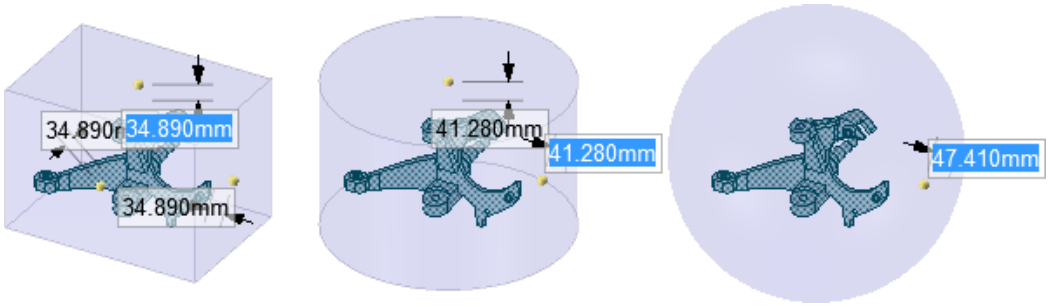
The extracted facet-based volume covers these gaps between the bodies.



12.5.2. Enclosure Tool

An enclosure is a solid around a body or bodies that has a cushion around the enclosed solid(s). The enclosure can be a box, cylinder, or sphere, as shown below.

Enclosures are used by analysis tools to simulate fluid.



This section contains the following topics:

12.5.2.1. Creating an Enclosure

1. Click **Enclosure** in the **Generate** section of the Prepare tab.
2. Select one or more solid or faceted bodies in the Design window or model tree.

   These bodies will be inside the enclosure.
3. Set the **Default cushion** amount.

   This is a percentage of the minimum enclosure size, and determines the initial distance between the enclosed object(s) and the closest point of the enclosure to the objects. You can adjust the distances by typing in the fields in the Design window.
4. Select the shape of the enclosure in the Options panel: **Box**, **Cylinder**, **Sphere**, or **Custom shape**.

   If you select **Custom shape**, you must use the **Custom shape** tool guide to select a solid to use as the enclosure shape.
5. (Optional) Click the **Set Orientation** tool guide and select a line, axis, or origin to change the orientation of the enclosure.

The enclosure is oriented with the world origin by default.

6. (Optional) Deselect the **Symmetric dimensions** option to set the cushion distance different for dimensions that are opposite each other.
7. (Optional) Type a value to adjust the cushion values, if necessary.

   Press **Tab** to move among the cushion values.
8. (Optional) Enable **Create share topology** to create share topology between the enclosed body and the input bodies.

   Subtract from enclosure is enabled by default. This will remove the selected geometry from the enclosure.
9. Click the **Complete** tool guide when you are finished.

   The enclosure is created in a sub-component of the active part.

12.5.2.2. Enclosure Tool Guides and Options

Enclosure Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Bodies	Use the **Select Bodies** tool guide to select the bodies that will be enclosed.
Set Orientation	Use the **Set Orientation** tool guide to change the orientation of the enclosure relative to your design or the axis of a coordinate system.
Custom Shape	Use the **Custom Shape** tool guide to select a solid to use as the custom shape when you set the enclosure type to **Custom** in the Options panel.
Complete	Click the **Complete** tool guide when you are finished.

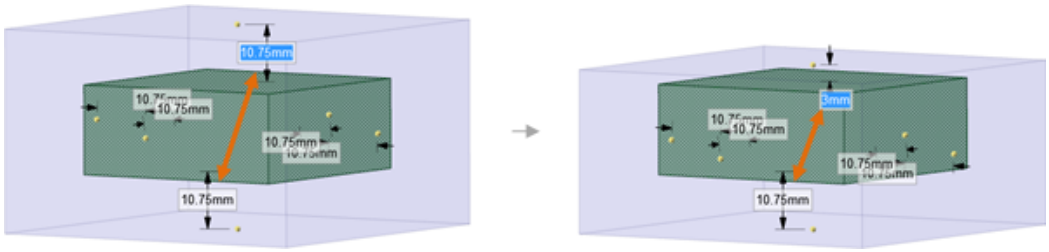
Enclosure Tool Options

The following options are available:

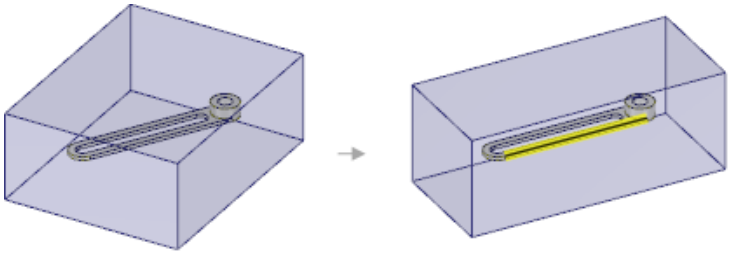
Default cushion	This is a percentage of the minimum enclosure size, and determines the distance between the enclosed object(s) and the closest point of the enclosure to the objects. You can change adjust the distances by typing in the fields in the Design window.
Enclosure type	Select an enclosure shape from the list. If you select **Custom Shape**, you must use the **Custom Shape** tool guide to select a solid to use as the enclosure shape.
Symmetric dimensions	Forces the dimensions to remain symmetric. Deselect this option if you want to enter values for dimensions and you don't want the opposite dimensions changed.
Create share topology	Creates share topology between the enclosed body and the input bodies.
Subtract from enclosure	Removes selected geometry form the enclosure.

12.5.2.3. Enclosures Examples

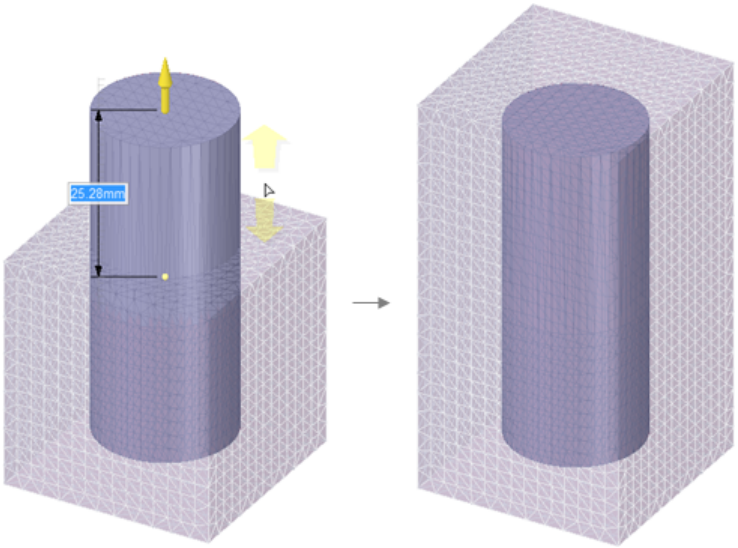
*Changing the cushion by typing in the field doesn't change the size of the opposite cushion. The **Symmetric dimensions** option must be disabled.*



Changing the orientation of the enclosure so it is aligned with the edge highlighted in yellow aligns the enclosure with that edge.



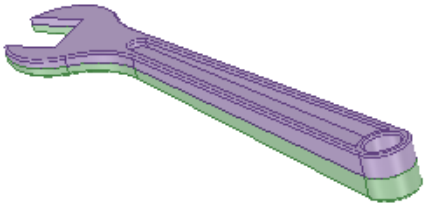
*If the associated body is modified, you can update the enclosure using the context menu item found by right-clicking on **Enclosure** in the model tree.*



12.5.3. Split by Plane Tool

The **Split by Plane** tool splits parts based on a plane. It is intended to be used to split symmetrical parts for analysis. This tool is similar to Split Solid, except Split by Plane allows you to select an axis, point, or edge, to use as splitting plane locations which are not allowed by Split Solid.

The part below has been split along the length of its handle. The purple and green parts are symmetrical, and can be analyzed faster than the whole part.



This section contains the following topics:

12.5.3.1. Splitting Objects by Plane

1. Click **Split by Plane** in the **Generate** group of the **Prepare** tab.
2. Select the object(s) you want to split.
3. Select a plane to split the object:
   - To split the part on the same plane as a face, edge, or point, click **Select Cutter** and select a reference face, edge, or point.
   - To define a temporary plane based on design elements, click **Build Cutting Plane** and select faces, edges, or points to define the plane. You can select more than one face, edge, or point, and the cutting plane will be created that bisects the elements. The wrench in the image above is tapered. Using the **Build Cutting Plane** tool guide, you can select the top and bottom of the wrench to create a plane that bisects the wrench along its plane of symmetry.
4. To delete objects created by the split, click the **Select Regions** tool guide and mouse over the parts created by the split. The regions will be highlighted. Click to delete the region.
5. Click the **Complete** tool guide.

   The application splits the solid along the plane.

12.5.3.2. Split by Plane Tool Guides and Options

Split by Plane Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Target	The **Select Target** tool guide is active by default. Use this tool guide to select the object that will be split.
Select Cutter	Use the **Select Cutter** tool guide to select a reference face, edge, or point with which to cut the part. Use Ctrl to select more than one object. The plane is previewed in gray before you select the reference object.
Build Cutting Plane	Use the **Build Cutting Plane** too guide to select faces, edges, or points to create a temporary plane with which to cut the object. The temporary plane(s) are shown with a dashed line. Select a temporary plane to complete the split.
Select Regions	Use the **Select Regions** tool guide to select regions that will be removed. You can move your mouse over regions that were created and highlight them before you click to delete.

Split by Plane Tool Options

The following options are available:

Merge when done	Select this option to merge all touching solids or surfaces when you exit the tool. Hidden objects are not merged. This saves you the extra step of selecting all the cut-up regions after you are done and manually merging them all back together.

12.5.4. Imprint Tool

The **Imprint** tool detects coincident faces, edges, and vertices between bodies and imprints them onto the coincident face. The contact regions will be the same shape, and the resulting mesh on each face will be similar. This can be helpful when analyzing stress between two parts.

> **Note:** Curves and Beams that intersect faces create imprinted Points that are converted to Pinned Datum Points.
>
> Sketch points coincident to faces create Pinned Datum Points when imprinted.

This section contains the following topics:

12.5.4.1. Imprinting a Part

1. Click **Imprint** in the **Generate** group of the Prepare tab. The tool will automatically detect and highlight coincident edges.
2. Specify the **Tolerance** for the distance between objects (faces, edges, or curves).
3. Select the **Type** of objects to be considered (faces, edges, beams, or curves).
4. For models with instanced parts, select the appropriate option for **Imprint by**. See Imprint options for the descriptions of the options available.
5. Select the object(s) you want to change:
   - Use the **Select Problem** tool guide to select highlighted areas that you want to change. The cursor will change to a hand when you move the mouse over a problem area. This tool guide is active by default.
   - Use the **Select Geometry** tool guide to select edges that were not automatically detected. You can use box, lasso, and paint select modes.
6. Click the **Complete** tool guide.

   The application imprints the selected edges of the coincident faces.

> **Note:** There is an internal limit to the number of imprint sites found by the tool. The limit is set to 1000 to optimize for performance.

12.5.4.2. Imprint Tool Guides and Options

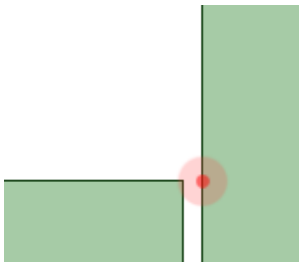
Imprint Tool Guides

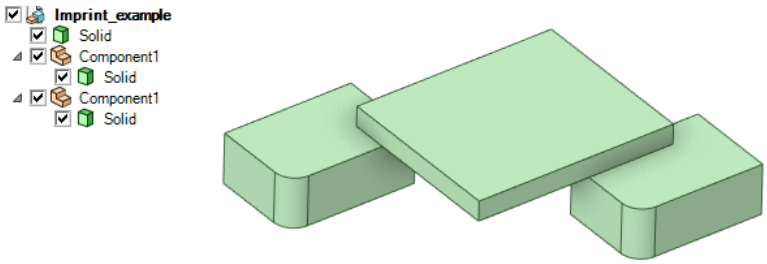
The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

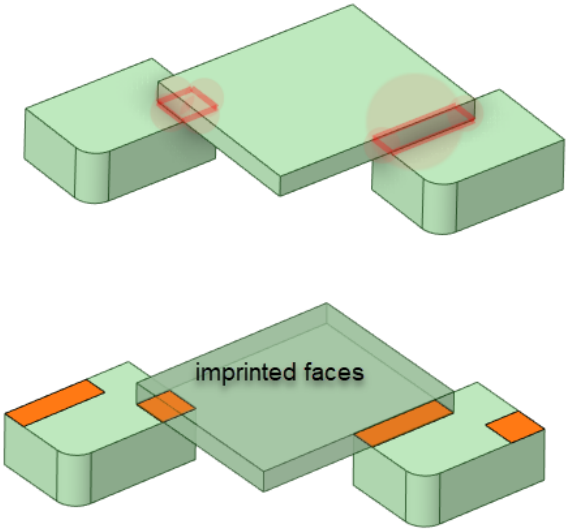
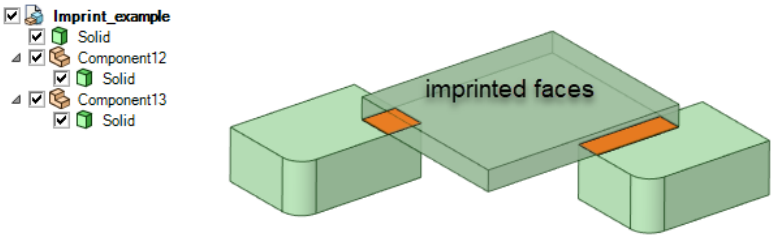
Select Problem	The **Select Problem** tool guide is active by default. This tool guide allows you to select problem areas that are automatically found by the tool.
Exclude Problem	The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.
Complete	The **Complete** tool guide imprints all the highlighted edges.

Imprint Tool Options

The following options are available:

Tolerance	If the distance between objects is less than or equal to the Tolerance value, they will be considered intersecting. In the example, the distance between the objects is 0.01, which is equal to the **Tolerance**. 

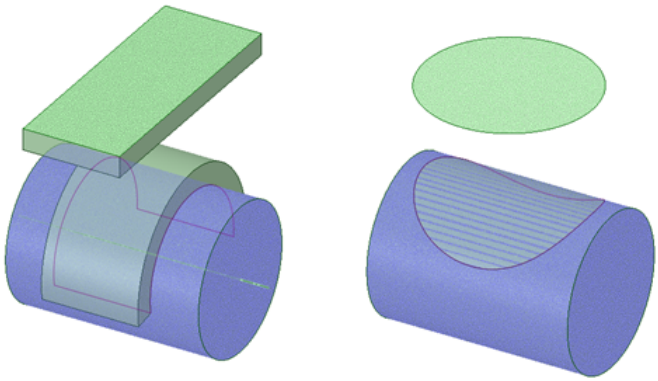
<table>
<tr><td rowspan="4">Imprint by</td><td>For models with instanced parts, select the appropriate option for Imprint by.</td></tr>
<tr><td>Ignore instances: Ignores instanced parts if the imprint would create a mismatch across the instances.
 

In this example, the imprint would create a mismatch across the instances, so the instanced parts are ignored.</td></tr>
<tr><td>Imprint across instances: Imprints across instanced parts. Additional imprints are created to preserve the matching across instanced parts.
 
</td></tr>
<tr><td>Imprint and break instances: Imprints across instanced parts. Instances that can no longer be identical to one another will be broken (see the model tree).
 
</td></tr>
<tr><td>Face</td><td>Finds intersections between two faces.</td></tr>
<tr><td>Edge</td><td>Finds intersections between two edges.</td></tr>
<tr><td>Beam/Curve</td><td>Finds intersections between curves/beams.</td></tr>
</table>

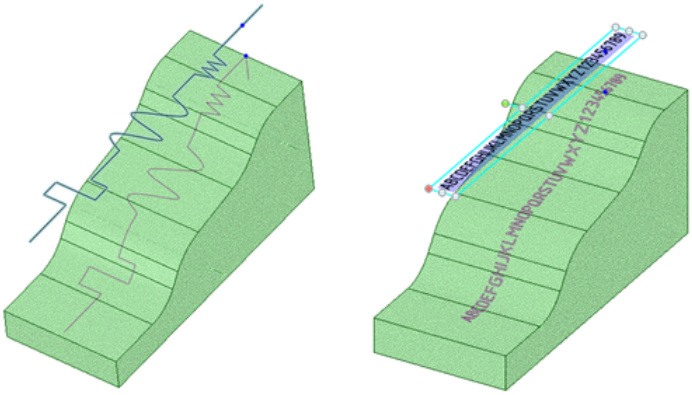
12.5.5. Wrap Tool

The **Wrap** tool lets you wrap geometry around a target model.

*The example below shows both a solid block and a circular surface wrapped onto a cylindrical target body using the **Wrap** tool.*



Sketched Curves and Notes can also be wrapped around a target body.



This section contains the following topics:

12.5.5.1. Wrapping Geometry and Text

1. Click the **Wrap** tool in the **Generate** group.
2. The **Select Target** tool guide is active by default.
   Select the target body.
3. The **Select Geometry** tool guide becomes active.
   Select the geometry, curves, points, point patterns, or notes to wrap around the target.
4. (Optional) Click the **Select Start** tool guide and select a Datum Point or Origin to use as a start point for the wrap.
   You can drag the source and target start points to reposition the wrap
5. (Optional) Set the Wrap Options.
   - Imprint as edges
   - Delete source geometry

6. Click **Complete** to wrap the selection.

12.5.5.2. Wrap Tool Guides and Options

Wrap Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

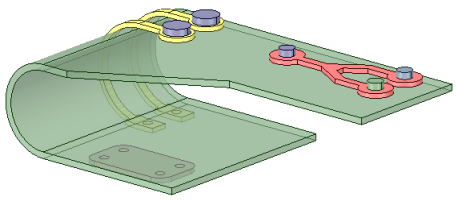
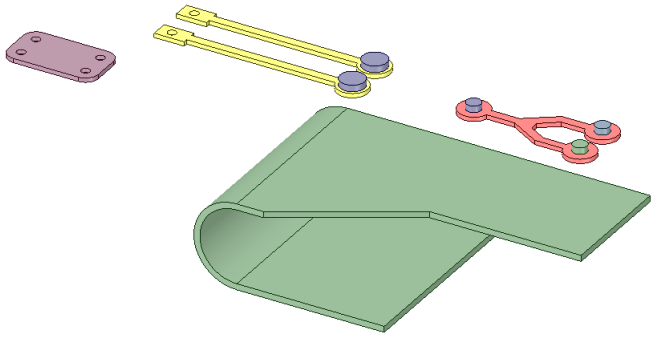
Select Target	The **Select Target** tool guide is used for selecting the target body you want to wrap geometry around.
Select Geometry	The **Select Geometry** tool guide is used for selecting the geometry to wrap around the target.
Select Start	The **Select Start** tool guide is used to select a datum or origin point to use as a start point for the wrap.
Complete	The **Complete** tool guide wraps the selection.

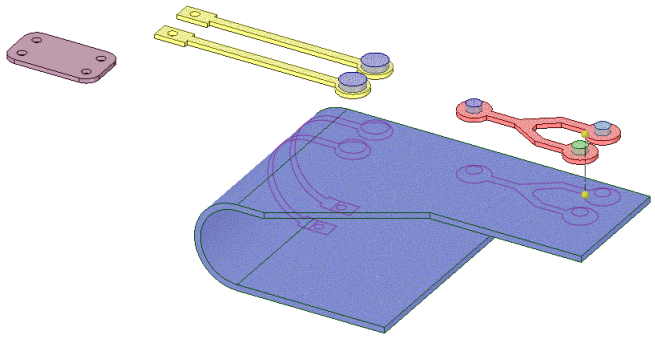
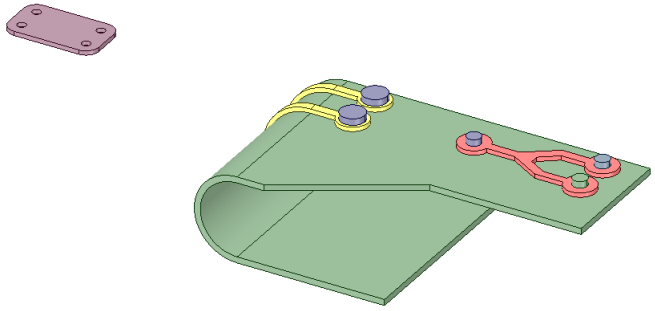
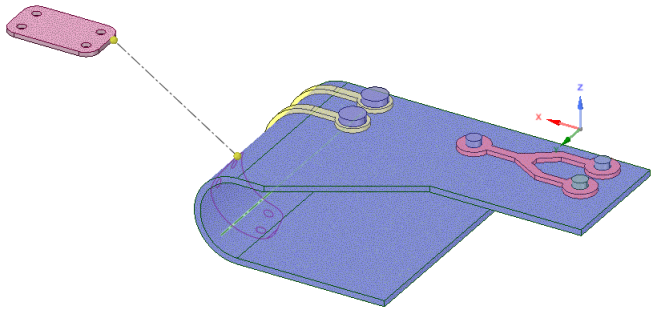
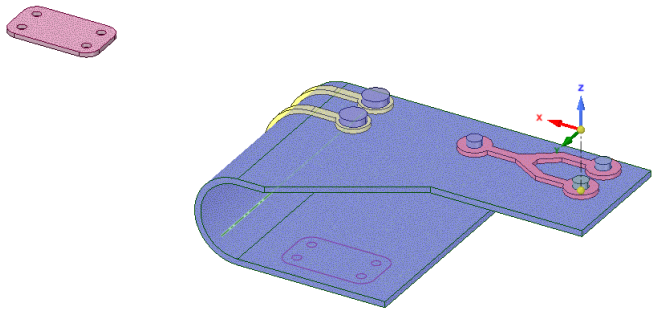
Wrap Tool Options

The following options are available:

Imprint as edges	When selected, no solid or surface geometry is wrapped. Solid and surface boundaries, and curves, are imprinted on the target.
Delete source geometry	The source geometry is deleted after it is wrapped.

12.5.5.3. Wrapping Examples

Layers of solid geometry can be wrapped. Consider a circuit board that is not flat, for example. Components layered onto the board need to follow the contours of the board. An example of the final circuit board is shown with transparency in the board to show a component underneath.	
The starting model for the circuit board is shown below. Notice that the components are at two different heights and they extend beyond the board. The spacing of the solids is based on their proximity to the *first* selected solid to be wrapped.	

Components are about to be wrapped onto the contoured board. The board is selected as the target and the first set of components is selected for wrapping. it includes a base layer of components and a second layer that rest on top of the first. Notice the starting point indicated by the yellow handles on the right. By default, the handle locations are set to the shortest distance between the Source and the Target. They can be moved to different locations if desired. In the image below, the default location is used for the first set of components. The wrapped locations are previewed in purple.	
The image shows the completed wrap for the first set of components.	
The last component gets wrapped underneath the board. Notice how the start point is initially at the shortest distance. This would distort its spacing relative to the first set of components.	
In order for the component to map too the correct location, an origin is selected as the start point. This will maintain the original spacing of the component relative to the other components.	

12.6. Remove

Tools in the **Remove** group help you easily remove elements to simplify your designs.

Interference	Use the **Interference** tool to merge parts that clash.
Suppression	Use the **Suppression** tool to suppress edges or vertices in your model.
Rounds	Use the **Rounds** tool to remove rounded edges from a design.
Faces	Use the **Faces** tool to remove features from a design.
Short Edges	Use the **Short Edges** tool to find edges shorter than a specified length that may impact meshing results.

This section contains the following topics:

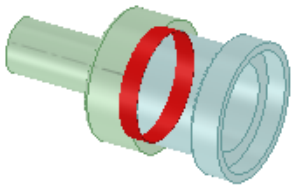
12.6.1. Interference Tool

The **Interference** tool detects and removes interference from clashing bodies. The interference is removed from the body with the largest volume.

The tool searches all visible bodies for interference. Bodies that are hidden (turned off in the model tree) are ignored.

If you want to remove an interfering region from one of the bodies, use the **Interference Volumes** tool on the **Measure** tab to create a solid of the interfering region. You can then use the **Combine** tool to remove this solid from one of the parts.

You can also specify the Interference tolerance to remove the interference between the bodies which cannot normally be removed with the default setting.



This section contains the following topics:

12.6.1.1. Removing Interference

1. Click **Interference** in the Remove group of the Prepare tab.
   The tool will automatically detect and highlight interference, as shown in the image above.
2. Specify the Interference tolerance, if applicable.
3. Select the object(s) you want to change:
   - Use the **Select Problem** tool guide to select highlighted areas that you want to change. The cursor will change to a hand when you move the mouse over a problem area. This tool guide is active by default.
   - Use the **Select Geometry** tool guide to select edges that were not automatically detected.
4. Click the **Complete** tool guide.
   The application removes the interference by merging the bodies into one part.

12.6.1.2. Interference Tool Guides and Options

Interference Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Problem	The **Select Problem** tool guide is active by default. This tool guide allows you to select problem areas that are automatically found by the tool.
Select Geometry	The **Select Geometry** tool guide allows you to select edges that were not automatically found. Hold Ctrl to select multiple edges or box select in the design area.
Complete	The **Complete** tool guide merges the clashing objects.

Interference Tool Options

The following options are available:

Interference tolerance	The default value is zero. Specify the Interference tolerance to remove the interference between the bodies which cannot normally be removed with the default setting.
Subtract from smaller body	When selected, the interference is subtracted from the smaller body. Otherwise the interference is removed from the body with the largest volume.

12.6.2. Suppression Tool

The **Suppression** tool enables you to suppress edges or vertices in your model. As an alternative method for cleaning up your model, edges or vertices can be marked as suppressed using this tool so that they will not be considered in downstream applications such as the Mechanical application. Edges or vertices that may result in problematic mesh elements or extra refinement can be suppressed using this tool.

Additionally, the repair tools **Split Edges**, **Extra Edges**, and **Small Faces** include an additional option to suppress topologies. Extra edges or vertices in the model can be marked as suppressed using this option.

The suppressed edges or vertices are highlighted to indicate them in the display when you are in the **Suppression** tool.

> **Note:** Suppressing vertices or edges does not affect simulations in the Discovery application.

This section contains the following topics:

12.6.2.1. Using the Suppression Tool

To suppress edges or vertices:

1. Select the **Suppression** tool in the **Prepare** tab.
2. Use the **Select** tool guide to select edges or vertices to suppress.

3. (Optional) Use the **Exclude selection** tool guide exclude edges or vertices for suppression.
4. Click **Complete** to suppress the edges or vertices.
   The suppressed edges or vertices are highlighted to indicate them in the display.

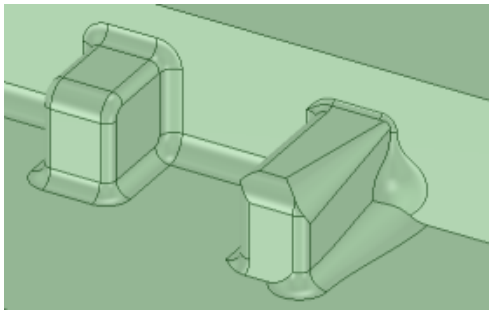
12.6.2.2. Suppression Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select	The **Select** tool guide is active by default. Use this tool guide to select edges or vertices for suppression.
Exclude selection	Use the **Exclude selection** tool guide to exclude previously suppressed edges or vertices from the selection.
Complete	Use the **Complete** tool guide to suppress the selected entities.

12.6.3. Rounds Tool

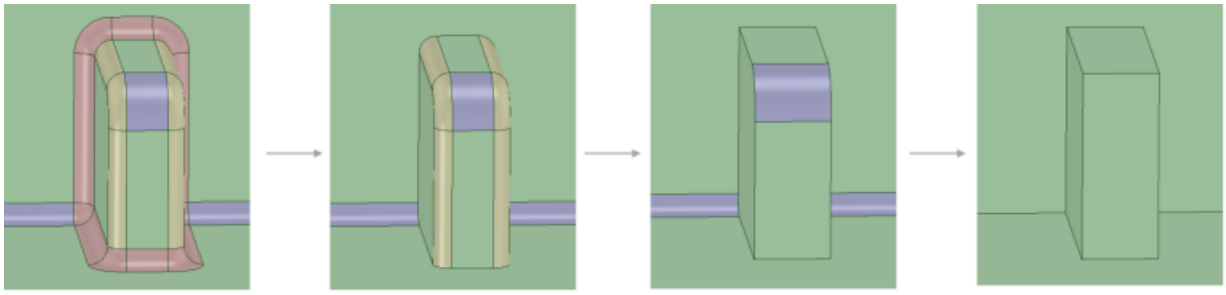
The image below shows an example of well-formed rounds, on the left, and poorly formed rounds, on the right.



Note:

- You will have a greater chance of success if you remove only a few rounds at a time. If you spend a lot of time removing rounds, you may want to review the application's tutorials or technical support information for advanced techniques you can use when removing rounds.
- Closely inspect your model from various angles and zoom levels, so you can best determine the areas of rounds you want to remove. Each area of a model is unique, and different areas may require different approaches to round removal.
- Select the round's or round group's shortest edge first.

To ensure accurate and complete round removal, remove the rounds in the reverse order in which they were created, as shown in the example below of a model with colored rounds.



This section contains the following topics:

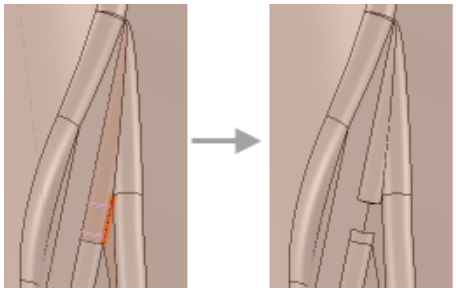
12.6.3.1. Removing Rounds

To split and partially remove a round:

1. Click **Rounds** from the **Remove** group on the **Prepare** tab.

   Avoid splitting into a round. Instead, split on either side of it. You may need to split and partially remove one or more rounds before adjacent rounds can be removed, especially when two or more rounded faces meet along an edge or vertex. The rounded face is split and filled in the middle. To select faces or capping faces, use the **Faces** tool.

2. Hover over one tangent edge of the rounded face to display the round's split lines. These lines show you a visual indication of where the round will split, as shown below. You can move the split lines by moving your mouse.

3. If necessary, from the **Options** panel, change the **Cap width** option to adjust the percentage of the round face that is removed.

   - Click the edge to split the rounded face.
   - Select the rounds you want to remove:
     - Click an object in the **Design** tree to select all rounds on the object.
     - Select a round face in the design area. You can hold **Ctrl** and select faces or draw a box to select multiple faces.

       Ctrl+click to deselect a face.

       From the context menu choose **Select** > **Using Box**. Move your mouse over the area you want to remove to identify the parts you want to select. Draw the selection box around the area. You can only select round faces while this tool is active.

4. To select faces or capping faces, use the **Faces** tool.
5. Click the **Complete** tool guide.

   The application removes the rounded faces you selected and replaces them with a sharp edge.

12.6.3.2. Rounds Tool Guides and Options

Round Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

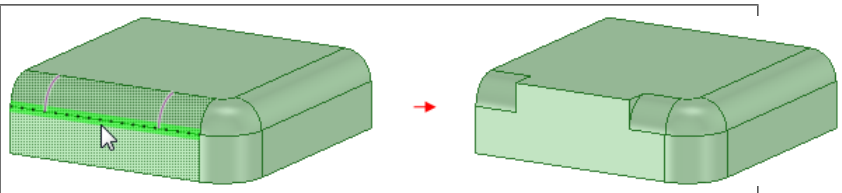
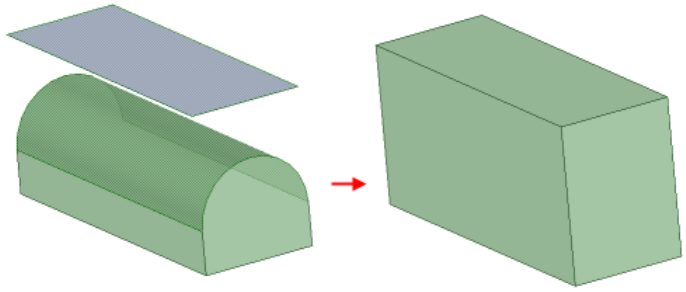
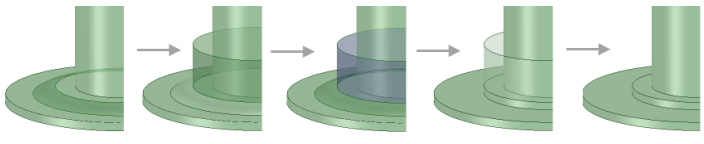
Select Rounds to Remove	The **Select Rounds to Remove** tool guide is active by default. You can also select by drawing a box in the design window. You can only select round faces while this tool is active.
Complete	The **Complete** tool guide finishes the operation. **F** or **Enter** are shortcuts for removing the rounds.

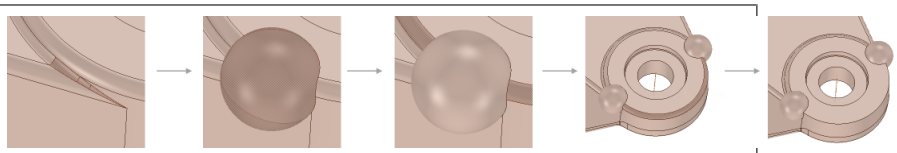
Round Tool Options

The following options are available:

Auto-shrink fill area	Select this option to split round faces where they can't be removed when you select a chain of rounds and it cannot be filled in its entirety. The rounds will be partially removed.
Cap width	The percentage of the width of a round that is removed when you click an edge of an existing round. The splits are previewed as the cursor is moved along the round tangent edges.

12.6.3.3. Removing Rounds Examples

Splitting and partially removing a round.	
Using other tools to remove a round.	
Progression using a face to remove a round.	

Using the Sphere tool to split and partially remove a round. • Try using the Sphere tool from the **Insert** group on the **Design** tab. Using this tool, you can create a round sphere and place it at difficult geometry junctions. Once in place you can use the sphere to split the junction in order to remove the round. In some cases, you may want to create a chain of two or more spheres to help with removing larger or more complex rounds areas. • To avoid trouble with removing spheres, it's good practice to fill the sphere immediately after you remove the rounds on either side of a sphere or sphere chain.	

12.6.4. Faces Tool

The **Faces** tool allows you to quickly remove faces from your design. Use it to simplify your design by removing holes, protrusions, etc.

1. Click **Faces** in the Remove group of the Prepare tab.
2. Select the faces you want to remove. Hold the Ctrl key to select more than one face.
3. Click the **Complete** tool guide.

   The application attempts to remove the selected faces by extending neighboring faces.

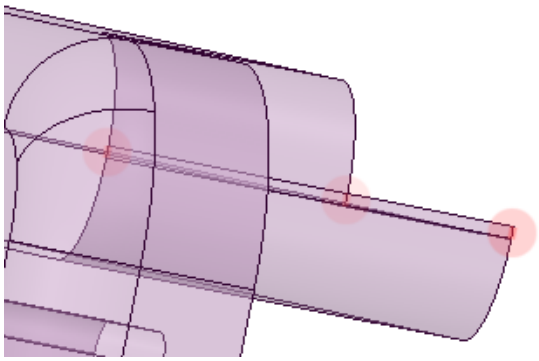
12.6.4.1. Faces Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Faces to Remove	The **Select Faces to Remove** tool guide is active by default. You can also select by drawing a box in the design window. You can only select faces while this tool is active. Edges and vertices will not be selected.
Complete	The **Complete** tool guide finishes the operation.

12.6.5. Short Edges Tool

The **Short Edges** tool finds any edges smaller than a specified length. If the design must be meshed for running simulations, short edges have an impact on the mesh quality.



1. Click the **Short Edges** tool in the Remove group of the Prepare tab.
2. Set the **Maximum length** option or keep the default value.
3. Edges within the maximum length tolerance are highlighted.
4. Click the problem areas to fix them or click the **Complete** tool guide.

12.6.5.1. Short Edges Tool Guides and Options

Short Edges Tool Guides

The following tool guides help you through the process.

Select Problem	The **Select Problem** tool guide is active by default. This tool guide allows you to select and fix problem areas that are automatically found by the tool.
Exclude Problem	The **Exclude Problem** tool guide allows you to exclude areas from selection and fixing.
Complete	The **Complete** tool fixes all remaining problem areas or those that are currently selected.

Short Edges Tool Options

The following options are available:

Maximum length	Set the maximum length to find edges less than the specified length.

12.7. Identify

The **Detect** tool in the **Identify** ribbon group consists of several tools that search and identify problems in your design.

Small Bodies	Use the **Small Bodies** tool to find small bodies less than or equal to the specified volume.
Surface Bodies	Use the **Surface Bodies** tool to find all surface bodies in the model.
Logos	Use the **Logos** tool to find protrusions or depressions.
Threads	Use the **Threads** tool to find threads less than or equal to a specified maximum diameter.

Holes	Use the **Holes** tool to find holes less than or equal to a specified maximum diameter.
Bad Faces	Use the **Bad Faces** tool to detect faces that have corrupt rendering.
Sharp Edges	Use the **Sharp Edges** tool to find sharp edges on faceted and solid bodies.
Clearance	Use the **Clearance** tool to find small gaps between parts.
Sweepable Bodies	Use the **Sweepable Bodies** tool to identify bodies that have free (unstructured) source and target faces opposite each other and mapped faces on the perpendicular sides between.
Fasteners	Use the **Fasteners** tool to identify and classify parts as bolts, nuts, and washers.

This section contains the following topics:

12.7.1. Detect Small Bodies Tool

The **Small Bodies** tool detects all small bodies less than or equal to a volume you specify.

Small bodies outside of the specified limit are highlighted in red.

This section contains the following topics:

12.7.1.1. Finding Small Bodies

1. Select **Small Bodies** from the dropdown menu of the **Detect** tool in the **Identify** group of the **Prepare** tab.
2. Enter a **Volume** in the options panel to set the search criterion.

   Alternatively, you can use **Select volume on model** in the Options to set the **Volume** limit.
3. To delete empty components after small bodies are detected and removed, select **Delete empty components**.
4. Click **Find matching small bodies** to search for and highlight all bodies with a volume less than or equal to the volume you specified.

   Alternatively, you can select **Automatically find matches** in the Options to detect matching small bodies automatically, although it may take a little longer to search and display the results on large models.
5. **Group named selection** is enabled by default and creates a single named selection for all the detected areas. Deselect this option to create a named selection per detected area.
6. Select the red highlighted object and choose to convert it to a named selection or a primary selection. Use **Delete detected area** to remove the identified object.

12.7.1.2. Small Bodies Tool Options

The following options are available in the options panel:

Volume	This is the volume limit to detect small bodies. Any bodies greater than the volume specified will be highlighted.
Automatically find matches	Searches and displays the results. It is an alternative to using **Find matching small bodies**.
Group named selections	Groups the named selection for the **Detect** tools.

Delete empty components	After small bodies are detected and removed, choose to delete any empty components.

12.7.2. Detect Surface Bodies Tool

The **Surface Bodies** tool detects all surface bodies in the model.

The surface bodies are highlighted in red.

This section contains the following topics:

12.7.2.1. Finding Surface Bodies

1. Select **Surface Bodies** from the dropdown menu of the **Detect** tool in the **Identify** group of the **Prepare** tab.
2. To delete empty components after small bodies are detected and removed, select **Delete empty components**.
3. Click **Find surface bodies** to search for and highlight all bodies with a volume less than or equal to the volume you specified.

   Alternatively, you can select **Automatically find matches** in the Options to detect surface bodies automatically, although it may take a little longer to search and display the results on large models.
4. **Group named selection** is enabled by default and creates a single named selection for all the detected areas. Deselect this option to create a named selection per detected area.
5. Select the red highlighted surface body and choose to convert it to a named selection or a primary selection. Use **Delete detected area** to remove the identified object.

12.7.2.2. Surface Bodies Tool Options

The following options are available in the options panel:

Automatically find matches	Searches and displays the results. It is an alternative to using **Find surface bodies**.
Group named selections	Groups the named selection for the **Detect** tools.
Delete empty components	After surface bodies are detected and highlighted, choose to delete any empty components.

12.7.3. Detect Logos Tool

The **Logos** tool detects and finds all protrusions or depressions less than or equal to a maximum height that you specify. Only logos with sharp edges at the part's endpoint are identified.

Logo protrusions or depressions exceeding the specified limit are highlighted in red.

This section contains the following topics:

12.7.3.1. Finding Logos

1. Select **Logos** from the dropdown menu of the **Detect** tool in the **Identify** group of the **Prepare** tab.
2. Enter a **Max height** in the options panel to set the search criterion.

3. Click **Find matching logos** to search for and highlight all objects with a maximum height less than or equal to the value you specified. Only logos with sharp edges at the part's endpoint are identified.

   Alternatively, you can select **Automatically find matches** in the Options to detect matching logos automatically, although it may take a little longer to search and display the results on large models.
4. **Group named selection** is enabled by default and creates a single named selection for all the detected areas. Deselect this option to create a named selection per detected area.
5. Select the red highlighted object and choose to convert it to a named selection or a primary selection. Use **Delete detected area** to remove the identified object.

12.7.3.2. Logos Tool Options

The following options are available in the options panel:

Max height	This is the maximum height to detect logos. Any protrusions or depressions with a height greater than that specified will be highlighted.
Group named selections	Groups the named selection for the **Detect** tools.
Automatically find matches	Searches and displays the results. It is an alternative to using **Find matching logos**.

12.7.4. Detect Threads Tool

The **Threads** tool detects and finds all threads less than or equal to a maximum diameter you specify.

Threads exceeding the specified limit are highlighted in red.

This section contains the following topics:

12.7.4.1. Finding Threads

1. Select **Threads** from the dropdown menu of the **Detect** tool in the **Identify** group of the **Prepare** tab.
2. Enter a **Max diameter** in the options panel to set the search criterion.
3. Click **Find matching threads** to search for and highlight all threads with a maximum diameter less than or equal to the value you specified.

   Alternatively, you can select **Automatically find matches** in the Options to detect matching threads automatically, although it may take a little longer to search and display the results on large models.
4. **Group named selection** is enabled by default and creates a single named selection for all the detected areas. Deselect this option to create a named selection per detected area.
5. Select the red highlighted object and choose to convert it to a named selection or a primary selection.

12.7.4.2. Threads Tool Options

The following options are available in the options panel:

Max diameter	This is the maximum diameter to detect threads. Any threads with a diameter greater than the diameter specified will be highlighted.
Group named selections	Groups the named selection for the **Detect** tools.

Automatically find matches	Searches and displays the results. It is an alternative to using **Find matching threads**.

12.7.5. Detect Holes Tool

The **Holes** tool searches an assembly for holes less than or equal to a maximum diameter you specify.

Holes exceeding the specified limit are highlighted in red.

This section contains the following topics:

12.7.5.1. Finding Holes

1. Select **Holes** from the dropdown menu of the **Detect** tool in the **Identify** group of the **Prepare** tab.
2. Enter a **Max diameter** in the options panel to set the search criterion.
3. Click **Find matching holes** to search for and highlight all holes with a maximum diameter less than or equal to the value you specified.

   Alternatively, you can select **Automatically find matches** in the Options to detect matching holes automatically, although it may take a little longer to search and display the results on large models.
4. **Group named selection** is enabled by default and creates a single named selection for all the detected areas. Deselect this option to create a named selection per detected area.
5. Select the red highlighted object and choose to convert it to a named selection or a primary selection. Use **Delete detected area** to remove the identified object.

12.7.5.2. Holes Tool Options

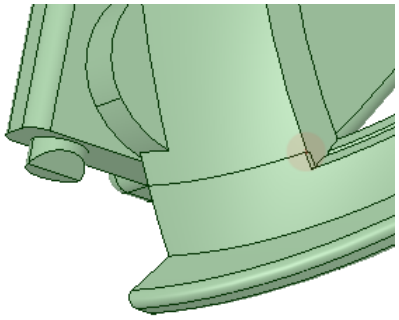
The following options are available in the options panel:

Max diameter	This is the maximum diameter to detect holes. Any holes with a diameter greater than the diameter specified will be highlighted.
Group named selections	Groups the named selection for the **Detect** tools.
Automatically find matches	Searches and displays the results. It is an alternative to using **Find matching holes**.

12.7.6. Detect Bad Faces Tool

The **Bad Faces** tool detects faces that have corrupt rendering. It detects bad edges that get flagged when faceting faces for graphics display.

A bad face is highlighted with a red circle.



This section contains the following topics:

12.7.6.1. Finding Bad Faces

1. Select **Bad Faces** from the dropdown menu of the **Detect** tool in the **Identify** group of the **Prepare** tab.
2. Click **Find matching bad faces** to search for and highlight faces with corrupt rendering. Those faces are highlighted with a red circle.

   Alternatively, you can select **Automatically find matches** in the Options to find bad faces automatically, although it may take a little longer to search and display the results on large models.
3. **Group named selection** is enabled by default and creates a single named selection for all the detected areas. Deselect this option to create a named selection per detected area.
4. Select the red highlighted face and choose to convert it to a named selection or a primary selection.

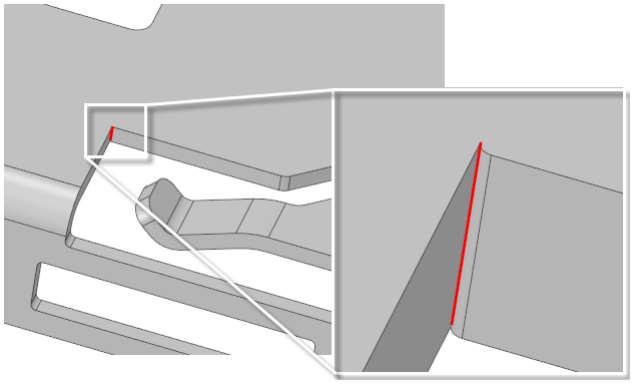
12.7.6.2. Bad Faces Tool Options

The following option is available for finding bad faces:

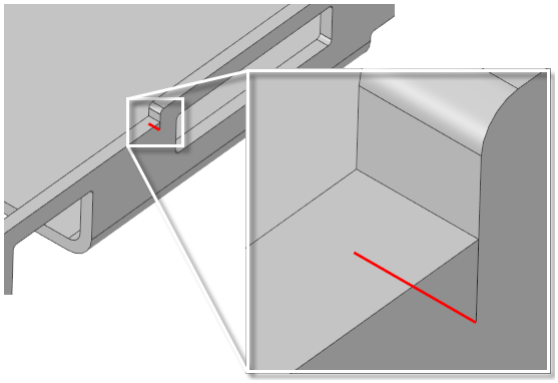
Automatically find matches	Searches and displays the results. It is an alternative to using **Bad Faces**.
Group named selections	Groups the named selection for the **Detect** tools.

12.7.7. Detect Sliver Faces Tool

The **Sliver Faces** tool detects faces within a specified angle range and lets you add them to a selection or convert them into a primary or named selection.



Edges on overlapping faces are also identified.



This section contains the following topics:

12.7.7.1. Finding Sliver Faces

1. Select **Sliver Faces** from the dropdown menu of the **Detect** tool in the **Identify** group of the **Prepare** tab.
2. Enter a **Max angle** in the **Options** panel to detect sliver faces with angles between 0° and the specified value.
3. Alternatively, enable **Automatically find matches** in the Options panel to have the application detect matching faces automatically. Searching and displaying results may take longer for large models.
4. Alternatively, enable **Group named selection** to create a single named selection for all the detected areas. Deselect this option to create a named selection per detected area.
5. Alternatively, select the red highlighted faces and choose to convert them to a named selection or a primary selection, individually.

12.7.7.2. Sliver Faces Tool Options

The following options are available for finding sharp edges:

Max angle	Enter a Max angle to detect sliver faces with angles between 0° and the specified value.
Automatically find matches	Automatically searches and displays matching results.
Group named selections	Groups the named selections.

12.7.8. Detect Clearance Tool

The **Clearance** tool detects small gaps between parts.

Gaps between faces that are equal to or less than the distance you specified are highlighted in red.

This section contains the following topics:

12.7.8.1. Finding Clearance

1. Select **Clearance** from the dropdown menu of the **Detect** tool in the **Identify** group of the **Prepare** tab.
2. Enter a **Max distance** in the options panel to check for near faces.
3. Click **Find matching clearances** to search for and highlight gaps between faces that are less than the specified distance.

Alternatively, you can select **Automatically find matches** in the Options to detect matching small bodies automatically, although it may take a little longer to search and display the results on large models.

You can use the **Pull** or **Move** tools to adjust any gaps as needed.

4. **Group named selection** is enabled by default and creates a single named selection for all the detected areas. Deselect this option to create a named selection per detected area.
5. Select the red highlighted object and choose to convert it to a named selection or a primary selection.

> **Note:** Verify that the detected results are accurate.

> **Note:** The minimum clearance tolerance value is set to a default of 1e-5m. Clearances between 0 and this value are not detected.

12.7.8.2. Clearance Tool Options

The following options are available in the Options panel:

Max distance	This is the maximum distance to detect for near faces. Gaps between faces that are equal to or less than the specified distance will be highlighted.
Automatically find matches	Searches and displays the results. It is an alternative to using **Find matching clearances**.
Group named selections	Groups the named selection for the **Detect** tools.

12.7.9. Detect Sweepable Bodies

The Show Sweepable Bodies feature displays bodies that can be swept. Sweepable bodies have free (unstructured) source and target faces opposite each other and mapped faces on the perpendicular sides between. They can be filled with an all hex and/or prism mesh that is interpolated between the source and target faces.

Once sweepable bodies are identified, you can select the identified bodies or create a named selection that includes them. You can also hide those bodies that are sweepable so you can edit any bodies not currently sweepable before transferring geometry to a downstream application.

This section contains the following topics:

12.7.9.1. Finding Sweepable Bodies

To find sweepable bodies:

1. Select **Sweepable Bodies** from the dropdown menu of the **Detect** tool in the **Identify** group of the **Prepare** tab.
2. Select **Find matching objects** to identify sweepable bodies.

   Alternatively, you can select **Automatically find matches** in the Options to detect sweepable bodies automatically, although it may take a little longer to search and display the results on large models.
3. You can then do one or more of the following:

Option	Description
Converted detected area to selection	Select to select all identified sweepable bodies.
Convert detected area to named selection	Select to convert all identified sweepable bodies to a named selection. The application creates a single named selection for all detected areas, but you can deselect **Group named selection** to create a named selection per detected area.
Hide detected area	Select to hide identified sweepable bodies so you can edit any bodies not currently sweepable.

12.7.9.2. Sweepable Bodies Tool Options

The following options are available in the options panel:

Hide detected area	Hide all identified sweepable bodies so that you can edit any bodies not currently sweepable.
Convert detected area to named selection	Convert all identified sweepable bodies to a named selection. The application creates a single named selection for all detected areas, but you can deselect **Group named selection** to create a named selection per detected area.
Convert detected area to selection	Select all identified sweepable bodies.
Automatically find matches	Detect sweepable bodies automatically. **Note:** The application may take a little longer to search and display the results on large models.
Group named selection	Creates a named selection per detected area when **Convert detected area to named selection** is clicked.

12.7.10. Detect Fasteners Tool

The **Fasteners** tool detects and classifies parts as bolts, nuts, and washers.

The fastener detection method uses a machine-learning–trained heuristic approach. You have the ability to reclassify fasteners into one of the other categories.

This section contains the following topics:

12.7.10.1. Finding Fasteners

1. Select **Fasteners** from the dropdown menu of the **Detect** tool in the **Identify** group of the **Prepare** tab.

   The tool automatically searches for bolts, nuts, and washers in the model. The quantity of each fastener type is displayed in the options panel.

2. Show or hide **Bolts**, **Nuts**, and **Washers** in the options panel.

   To clearly view and review detected areas, use **Hide others** to hide undetected bodies.

3. Review each of the detected fasteners using the next and previous arrows and determine if you need to **Reclassify as** a different fastener.
4. **Group named selection** is enabled by default and creates a single named selection for all the detected areas. Deselect this option to create a named selection per detected area.
5. Select the red highlighted object and choose to convert it to a named selection or a primary selection. Use **Delete detected area** to remove the identified object.

> **Note:** To control the search confidence of the different fastener types, expand the options panel. Move the **Bolts**, **Nuts**, and **Washers** slider to the right to remove items with lower confidence. This can remove misclassified items. Move the **Others** slider to the left to see other items with lower confidence. Other items with lower confidence can be potential bolts, nuts, or washers that the application missed.

12.7.10.2. Fasteners Tool Options

The following options are available in the options panel:

Review	Review each of the detected fasteners using the next and previous arrows. Use **Zoom to fit** to get a closer look at the part. Use **Reclassify as** to change the identified part to a different fastener.
Bolts, **Nuts**, **Washers**	Select to show or hide identified bolts, nuts, and washers. Move the **Bolts**, **Nuts**, and **Washers** slider to the right to remove items with lower confidence. This can remove misclassified items. Move the **Others** slider to the left to see other items with lower confidence. Other items with lower confidence can be potential bolts, nuts, or washers that the application missed.
Group named selection	Creates a named selection per detected area when **Convert detected area to named selection** is clicked.
Hide others	Hide undetected bodies to clearly view and review detected areas.
Convert detected area to named selection	Convert all identified fasteners to a named selection. The application creates a single named selection for all detected areas, but you can deselect **Group named selection** to create a named selection per detected area.
Convert detected area to selection	Converts identified fasteners to a selection.
Delete detected area	Delete identified fasteners.

12.8. Shared Topology

Use the tools in the **Shared Topology** group to prepare bodies for sharing topology.

Pre-check	Use the Pre-check tool to check bodies for any issues before sharing among bodies in a model.
Share	Use the Share tool to identify topology that can be shared among bodies in the model.
Unshare	Use the Unshare tool to unshare topologies.

Review	Use the Review tool to select, filter, and review Shared Topology.

This section contains the following topics:

12.8.1. Navigating Through Shareable Topology

In addition to stepping through identified problem areas, the navigation tools that allow you to step through all topology identified as shareable.

1. Click the **Share** tool and then click **Preview**. Sharing candidates are highlighted in red.
2. Click the **Next** or **Back** buttons to view each identified sharing candidate. The individual sharing candidates are identified in darker red.
3. (Optional) Select **Zoom to Fit** to zoom in and center on each sharing candidate when you click next or back.
4. (Optional) Select **Clip Volume** to show only the volume centered on the sharing candidate, hiding other regions of the model. You can then use the **Next** and **Back** buttons to step through the candidates.
5. Click the **Complete** tool to create shared topology at the selected candidate. Other candidates are not shared.
6. Repeat these steps until all required shared topology is created.

> **Note:** To cancel out of navigation mode, click the **Select Problem** tool guide. You may then create shared topology on all identified sharing candidates in one operation.

This is one of a few geometry repair and navigation tools included here for ease of access. Their behavior is the same as if accessed from the **Repair** ribbon.

12.8.2. Pre-check Tool

The **Pre-check** tool checks topology for any issues before sharing among bodies in a model. Find problem areas in your topology before sharing using the **Coincidence tolerance** as the threshold for finding short edges, slivers, duplicate edges, duplicate faces, self-intersections, and misalignments.

After checking the topology and fixing any problematic geometry, you can use the **Share** tool to share coincident topology.

This section contains the following topics:

12.8.2.1. Checking the Topology in Preparation for Sharing

1. Review the problem areas, if any, that the tool identifies and try to fix them.
   Problem areas will be highlighted in red.
2. Enter a **Coincidence tolerance** in the **Options** list.
   Objects that are within this distance from each other will be considered coincident.

> **Note:** The tolerance should be as small as possible to avoid collapsing small edges.

3. Select the type of topology you would like to find.

   Short edges, slivers, duplicate edges, and duplicate faces are selected by default. You can also select self-intersections and misalignments, however, they can take longer to find.

4. Select the highlighted object and choose to convert it to a selection.

12.8.2.2. Precheck Tool Options

The following options are available:

Convert to selection	The **Convert to selection** option converts the highlighted objects to the current selection.
Coincidence tolerance	The **Coincidence tolerance** sets the threshold for checking the topology and finding problematic geometry. The purpose of coincidence tolerance in precheck tool is to validate its value that is automatically predicted by software. If it collapses model features (face/edges) unnecessarily, you should opt for lower tolerance.
Short edges **Slivers** **Duplicate edges** **Duplicate faces** **Self-intersections** **Misalignments**	Select the type of topology you would like to find. Short edges, slivers, duplicate edges, and duplicate faces are selected by default. You can also select self-intersections and misalignments, however, they can take longer to find. **Short edges** refer to edges that have very small lengths compared to the overall size of the model. All edges that are less than coincidence tolerance get collapsed automatically by the Share tool (up to a limit of 1e-5m). While using higher tolerances, you should use this tool to do pre cleanup. **Slivers** are thin, elongated elements that have a much higher aspect ratio (length-to-width ratio) than desired. All sliver faces with width less than coincidence tolerance should be cleaned up using the **Small faces** tool. **Duplicate edges** are redundant geometric features that have the same geometric location in the model. All edges of a body that are coincident with in coincident tolerance should be cleaned up if they exist in the model. **Duplicate faces** are identical surface elements that occupy the same physical space. All faces of a body that are coincident with in coincident tolerance should be cleaned up if they exist in the model. **Self-intersections** occur when a part of the model intersects with itself. All self-intersecting entities are highlighted and should be cleaned up using **Check geometry repair**. **Misalignments** refer to geometric features that do not align properly, such as mismatched nodes or vertices along shared edges. All alignment issues in the model should resolved by repositioning bodies to match accurately and have better Share Topology.

12.8.3. Share Tool

The **Share** tool will identify topology that can be shared among bodies in the model. You can then choose to share all or some of the candidates identified.

Any shared topologies are added in the tree under **Connections** > **Contacts**.

- *Visible* bodies are searched for sharing candidates. Hidden bodies are ignored.

- Candidates are identified by imprinting bodies with each other.
- Potential candidates outside the coincidence tolerance are ignored.

> **Note:**
>
> - When you re-apply the **Share** tool, already-found shared examples are not re-detected.
> - The **Share** tool does not work on **Locked Bodies**. It treats them as if they are hidden.
> - Using **Instancing**, bodies with **Shared Topology** within a component have the **Shared Topology** propagated to other instances of the component. This only affects bodies with **Shared Topology** and does not affect bodies without it.
> - Component names are changed when they are made independent in the process of sharing.
> - If using geometry parameters for a design variation study, note that the shared topology will be removed if the parameterization results in changes to the shared faces or edges.

This section contains the following topics:

12.8.3.1. Using the Share Topology Tool

1. Click the **Share** tool.
2. (Optional) Enter a new **Coincidence tolerance** in the Options panel.
3. Click **Preview shared topology** to preview coincident topology to share based on the **Coincidence tolerance** specified.

   Each time you modify the options, click **Preview shared topology** to preview the coincident topology based on the changes.
4. Using the **Select** tool guide, select individual candidates to share them.

   Using **Select Geometry**, select additional faces or edges to share them.

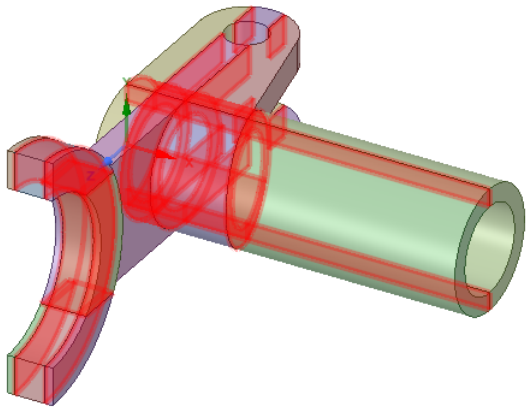
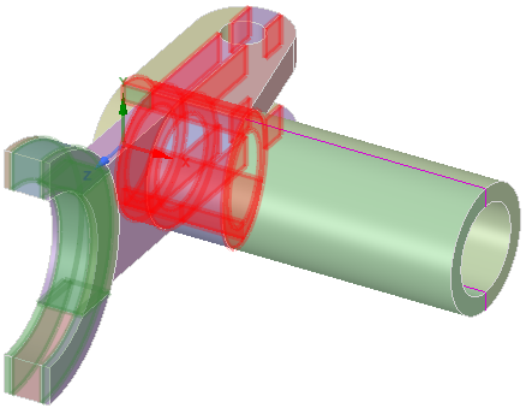
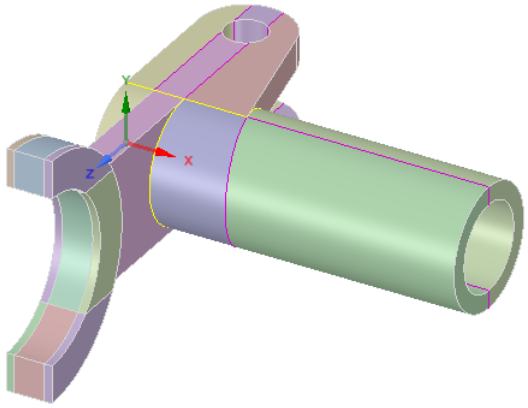
   You can also select **Exclude** and select individual candidates to exclude them.
5. Click the **Complete** tool guide to apply sharing to all remaining candidates.

   You can also click **Complete** to Share all coincident topology based on the options without previewing the coincident topology.

   Any shared topologies are added in the tree under **Connections** > **Contacts**.

   After sharing topology, you can select, filter, and review your results using the **Shared Topology Review** tool.

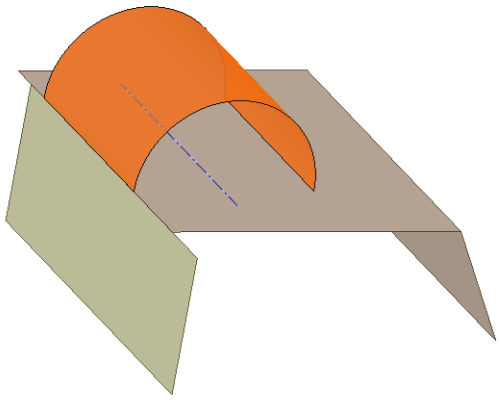
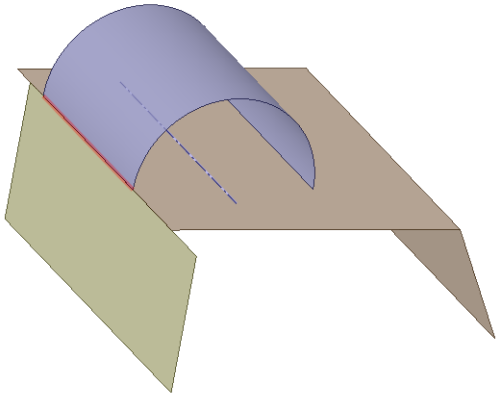
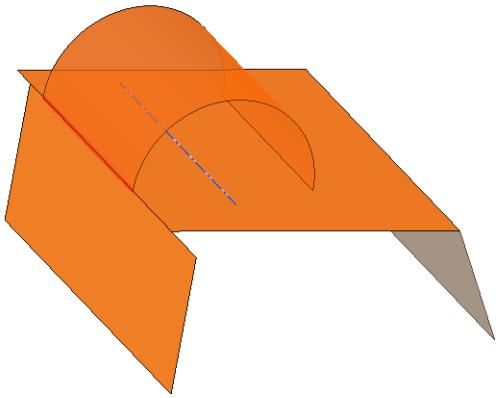
> **Note:** If using geometry parameters for a design variation study, note that the shared topology will be removed if the parameterization results in changes to the shared faces or edges.

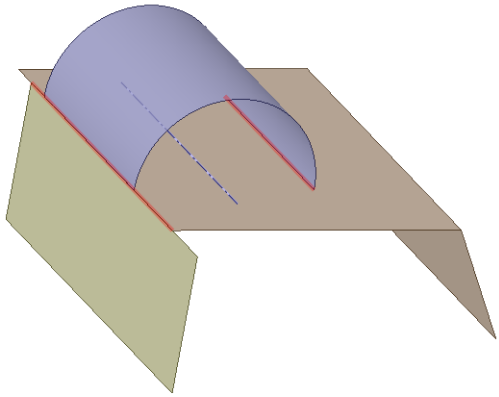
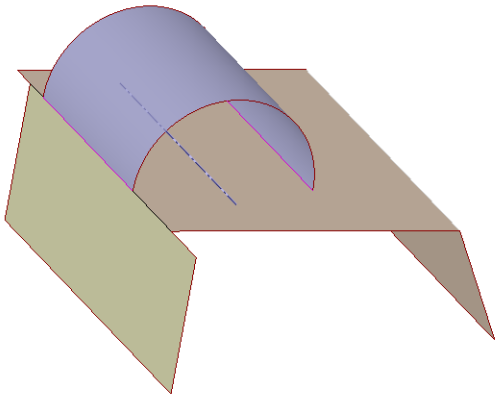
Click **Preview shared topology** to find sharing candidates and highlight them in red.	
Using the **Exclude Problem** tool guide, candidates in one area of the model are selected for exclusion. These candidates are now highlighted in green to indicate they are excluded.	
Click the **Complete** tool guide to share all remaining candidates.	

12.8.3.2. Manually Selecting Topologies to Share

You can also manually select groups of topologies for sharing.

- Faces and edges may be chosen
- Choose individual entities in pairs
- Ctrl-select additional entities if more than two entities need to be shared

Enter the Share tool. Click the **Select Geometry** tool guide. First click: Select the first surface.	
Second click: Select the edge.	
Third click: Hold Ctrl and select the surface body.	

After selections are complete, the possible connections are highlighted.	
After clicking **Complete**, the connections are made.	

12.8.3.3. Share Tool Guides and Options

Share Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select	The **Select** tool guide is active by default. This tool guide allows you to select coincident areas that are automatically found by the tool.
Select Geometry	The **Select Geometry** tool guide allows you to select faces and edges to share.
Exclude Problem	The **Exclude Problem** tool guide allows you to exclude areas sharing.
Complete	The **Complete** tool guide shares all highlighted coincident areas.

Share Tool Options

The following options are available:

Preview shared topology	Click to preview coincident topology to share based on the **Coincidence tolerance** specified. Each time you modify the options, click to preview the coincident topology based on the changes.

Coincidence tolerance	Enter a value in model units. Objects that are within this distance from each other will be considered coincident.
Preserve instances	Select this option to preserve instanced components. Contacts between instanced components and other geometries will be ignored.
Preview connectivity by color	Select this option to see the types of connections that will be generated after sharing. This option is off by default, but can add a lot of clarity to the preview.

12.8.4. Unshare Tool

The **Unshare** tool unshares topologies. Simply select shared entities in the graphics area to unshare. The tool's selection filter only allows selection of entities which are currently shared. Shared Faces display is enabled while the tool is active to assist in selection. Unshare also works with instances.

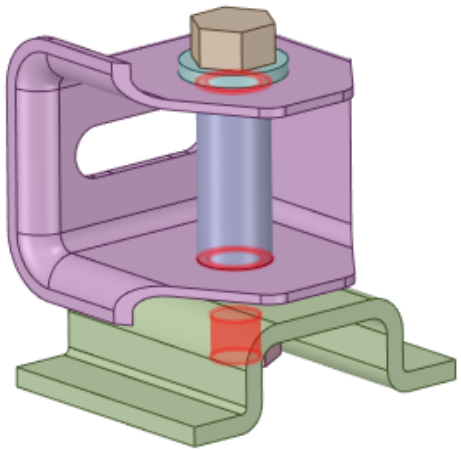
This section contains the following topics:

12.8.4.1. Using the Unshare Tool

1. Click the **Unshare** tool .

   Shared faces are displayed. You can enable the display of Shared Edges and/or Shared Vertices.
2. The **Include named selections** option is enabled by default for **Search Options**. Named selections created using the **Review** tool will also be searched for and unshared.
3. The **Include dependent topolgy** option is enabled by default for **Search Options**. Children of unshared entities will also be unshared. For example, unsharing a face will unshare the edges and/or vertices associated with it.

The figure shows the initial model with shared topology highlighted.



Three faces and their boundary edges are currently shared.

- The **Select** tool guide is active by default. Use this tool guide to select the entities that are automatically found by the tool.

  You can also use the **Select Geometry** tool guide to select faces and edges to unshare.
- Use the **Exclude Problem** tool guide to exclude entities for the unshare operation.
- Click **Complete** to unshare all highlighted shared entities.

12.8.4.2. Unshare Tool Guides and Options

Unshare Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select	The **Select** tool guide is active by default. This tool guide allows you to select previously shared entities that are automatically found by the tool.
Select Geometry	The **Select Geometry** tool guide allows you to select faces and edges to unshare.
Exclude Problem	The **Exclude Problem** tool guide allows you to exclude entities for the unshare operation.
Complete	The **Complete** tool guide unshares all highlighted shared entities.

Unshare Tool Options

The following options are available:

Include named selections	Named selections created using the **Review** tool will also be searched and unshared. This option is enabled by default.
Include dependent topology	Children of unshared entities will be unshared. For example, unsharing a face will unshare the edges and/or vertices associated with it. This option is enabled by default.

12.8.5. Share Topology Review Tool

The **Share Topology Review** tool allows you to select, filter, and review shared topology results.

This section contains the following topics:

12.8.5.1. Using the Share Topology Review Tool

1. Use **Select Bodies** to select bodies for review. Selecting an area outside of the model will deselect all selected bodies.
2. With a body selected or when reviewing the shared topology in the entire model:
   a) Select **Faces** to display shared and not shared faces. Select the display color for shared and not shared faces.
   b) Select **Edges shared** to display edges shared twice, three times, or more than three times. Select the display line thickness and color for the **Edges shared** options.
   c) Select **Free edges** to display free edges shared once or twice. Select the display line thickness and color for the **Free edges** options.
   d) For models with beams, select **Beam connection** to display beams according to the number of shared points. Select the display color for the beams based on the number of shared points.
   e) Select **Vertices** to display vertices in the model view.
   f) Select **Show obscured** to display entities that are hidden in the model view.

g) (Optional) Select **Keep visible** to enable persistent display of shared entities when using other tools.

3. (Optional) Select **Force share** to retry sharing for face pairs that are not shared based on the global **Coincidence tolerance** specified in the **Share** tool. For each face pair that is not shared, a dynamic local tolerance is automatically computed and sharing is retried.
4. (Optional) Select **Create named selections** to create one or more named selections per pair of not shared faces.
5. Click the **Complete** tool guide to apply sharing to all remaining candidates.

12.8.5.2. Share Topology Review Tool Guides and Options

Share Topology Review Tool Guides

Select bodies	Use the **Select bodies** tool guide to select bodies to review for Share Topology.

Share Topology Review Tool Options

The following options are available:

Force share	Select **Force share** to retry sharing for face pairs that are not shared based on the global **Coincidence tolerance** specified in the **Share** tool. For each face pair that is not shared, a dynamic local tolerance is automatically computed and sharing is retried.
Create named selections	Select **Create named selections** to create one or more named selections per pair of not shared faces.
Display	When reviewing the shared topology in the entire model or with a body selected, select entities to display and set the display properties. Choose from: **Faces** - **Shared** or **Not shared** **Edges shared** **Free edges** **Beam connection** **Vertices** **Show obscured** **Keep visible**

12.8.5.3. Share Topology Display Options

Confirming that topology sharing is present can be done by enabling the display options in the **Shared Topology Review** tool. The following choices are available for shared and free display, which are independent and may be enabled separately.

- **Faces**: Shared faces are highlighted such that they can be seen through the model.
  - **Shared**:

    Green indicates shared faces.

- **Not shared**:

  Red indicates faces which are not shared.

- **Edges**: Edges shared among bodies are highlighted in unique colors according to the number of edges shared. Shared edges can be set to **Thick** or **Thin** for display purposes.
  - **Double**:

    Dark blue indicates edges shared twice.
  - **Triple**:

    Violet indicates edges shared three times.
  - **Multiple**:

    Yellow indicates edges shared multiple (more than three) times.
- **Vertices**: Shared vertices are highlighted in unique colors according to the number of connected edges.
- **Beam connection**: Free beams are highlighted in unique colors according to the number of shared points.
  - **No Point**:

    Red indicates a beam sharing no points.
  - **One Point**:

    Gold indicates beams sharing one point.
  - **Two Point**:

    Green indicates beams sharing two points.
- **Free edges**: Free edges are highlighted in unique colors according to the number of edges shown.
  - **Single**:

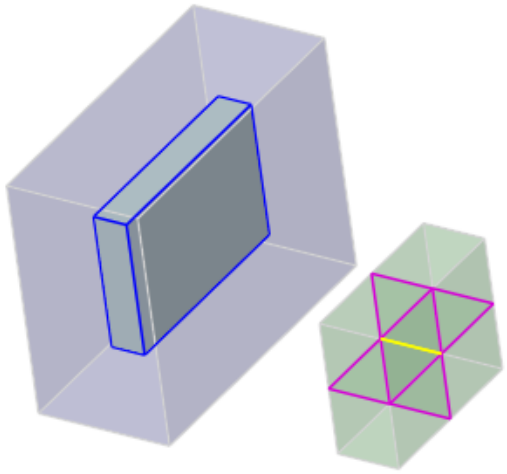
    

    Red for edges shown once.
  - **Double**:

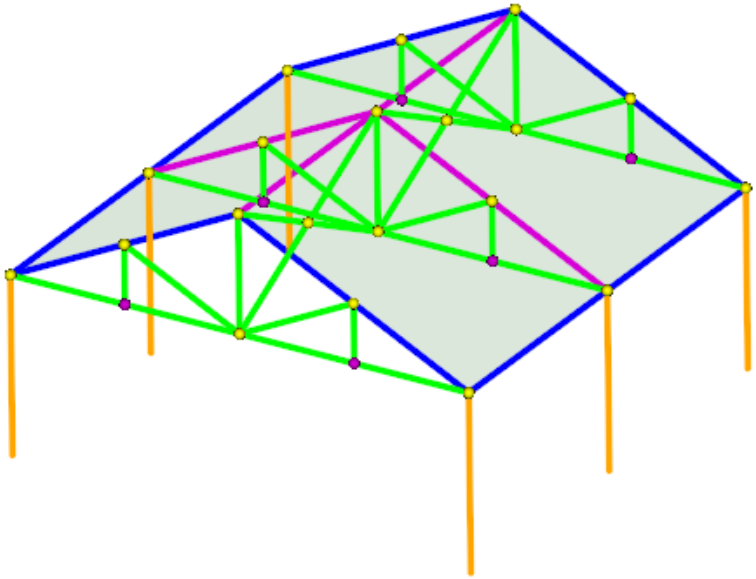
    

    White for edges shown twice.

This image shows a simple model with shared edges.



This image shows a beam and shell structure with shared edges and vertices displayed.



12.9. Ansys Transfer

The **Ansys Transfer** group contains options for transferring geometry to Ansys applications.

Update Parameters	Use **Update Parameters** to update parameter values in the Workbench parameter set.
Workbench	Use **Workbench** to start the **Ansys Workbench** application with a geometry system.
Mechanical	Use **Mechanical** to start the **Ansys Mechanical** application with the geometry imported.
Fluent	Use the workflow options in the dropdown to start the **Ansys Fluent** application. Select the appropriate options in the **Fluent Launcher**.

This section contains the following topics:

12.9.1. Fluent Workflow Options

Select the appropriate workflow option in the dropdown to start the **Ansys Fluent** application.

- Select **Watertight Geometry Workflow** to transfer the geometry to the **Ansys Fluent** application with the Watertight Geometry workflow selected. See the Ansys Fluent User's Guide for more information about Using the Watertight Geometry Workflow.
- Select **Custom New Workflow** to transfer the geometry to **the Ansys Fluent application**. See the Ansys Fluent User's Guide for more information about Customizing Workflows.
- Select **Export PMDB** to save Ansys Part Manager Files (*.pmdb).

> **Note:** To improve performance and robustness, support for all load managers (LSF, SLURM, SGE, PBSPro) except Remote Solve Manager (RSM) is removed.

13: Icepak

The **Icepak** group contains tools for identifying and simplifying bodies for conversion to Icepak object geometry. It also has tools that can create Icepak openings, fans, and grilles. The table below summarizes the tools.

Identify Objects	Use the **Identify Objects** tool to identify bodies that can be converted to Icepak objects.
Icepak Simplify	Use the **Icepak Simplify** tool to simplify bodies to make them compatible with Icepak object geometry.
Openings	Use the **Openings** tool to create surface bodies to represent an opening in a body.
Fan	Use the **Fan** tool to convert existing fans to Icepak fans or to add Icepak fans to specified bodies. Both the hub and shroud of user-identified fans will be deleted upon conversion to an Icepak fan.
Grille	Use the **Grille** tool to create an Icepak Grille on a selected face.
Enclosure	Use the **Enclosure** tool to convert an object to an Icepak Enclosure. The Enclosure tool approximates a box-shaped object as an open box based on any open geometry found by the tool.

This section contains the following topics:

13.1. Identify Objects Tool

The **Identify Objects** tool identifies bodies that can be converted to Icepak objects.

Icepak object types are compatible with certain types of geometry. By default the **Identify Objects** tool looks for the "Block" type and highlights any bodies whose geometry is compatible with Icepak blocks. The workflow for this tool is similar to other Find and Fix tools.

> **Note:** Surface bodies may be made up of multiple surfaces. In these cases, right-click on the surface node in the design tree and choose **Detach All**. This will separate the surfaces so they can be individually considered as Icepak objects.

This section contains the following topics:

13.1.1. Converting Bodies to Icepak Objects

1. Open a model that has bodies to convert.
2. Select the **Identify Objects** tool.

   The tool finds all bodies that can be converted to the Icepak Block type.
3. (Optional) Enable **Hide Icepak bodies** in the **Options** panel to hide bodies after they are converted.
4. If some of the found bodies do not need to be converted, click the **Exclude Problem** tool guide and select those bodies.
5. Click **Complete** to convert the highlighted bodies.

6. To convert bodies to different Icepak object types, click the **Select Bodies** tool guide.
7. Change the **Icepak Object Type** in the **Options** panel.
8. Select the bodies to convert and click **Complete** to convert the selected bodies.

   Only bodies with geometry compatible with the **Icepak Object Type** will be converted. If any of the selected bodies has incompatible geometry, a warning message is displayed.

   Repeat the steps for other object types.

As bodies are converted, their icon in the design tree changes to reflect that they are now Icepak bodies.

Use the **Show** menu to toggle the display of the following bodies.

- Icepak Bodies
- Non-Icepak Bodies
- All Bodies

> **Note:** Since Icepak object types depend on the geometry, if a converted body's geometry changes, it can become incompatible with the object type. In this case (rounded edges, etc.) the body is converted back to a normal solid body.

13.1.2. Identify Objects Tool Guides and Options

Identify Objects Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Problem	Use the **Select Problem** tool guide to convert individual bodies found by the tool.
Select Bodies	Use the **Select Bodies** tool guide to manually choose which bodies to convert. When this tool guide is active, you can change the Icepak Object Type in the Options panel.
Exclude problem	Use the **Exclude problem** tool guide to select bodies that you do not want to convert.
Complete	The **Complete** tool guide completes the conversion.

Identify Objects Tool Options

The following options are available:

Icepak Object Type	When the Select Bodies tool guide is active, you can change the type of Icepak object to one of the following types: • Block • Plate • Opening • Source • PCB • Resistance • Wall • Grille • None
Hide Icepak bodies	Enable this option to hide the Icepak objects after they are converted. This makes it easy to see which bodies still need to be converted.

13.2. Icepak Simplify Tool

The **Icepak Simplify** tool simplifies bodies to make them compatible with Icepak object geometry.

This section contains the following topics:

13.2.1. Simplifying Bodies

1. Open a model that has bodies to convert to Icepak objects.
2. Select the **Icepak Simplify** tool.
3. Choose a Simplification **Type** in the **Options** panel.
   - **Level 0** - Bounding box

     Each body is fit within its simple bounding box.
   - **Level 1** - Cuboid, cylinder fit
     - Bodies are simplified to simple solid or annular cylinders and cones.
     - Bodies already in this state are not simplified further.
     - Bodies with additional features may be simplified to bounding boxes if they cannot be resolved as cylinders or cones.
     - Bodies can be split into multiple polygons if required.
   - **Level 2** - Polygon fit (Default)
     - Bodies are analyzed along a specified direction and fit within polygonal shapes
     - Bodies can be split into multiple polygons if desired
   - **Level 3** - CAD object

     Bodies are converted to faceted bodies
4. Depending on the **Type**, set additional options.
   - **Level 0** - No options needed.
   - **Level 1**:

- Use **Cleanup** to remove details such as rounds before attempting to simplify, making it easier to recognize basic shapes.
- Use **Allow splitting** to split a body into multiple simplified shapes if possible.

  Use **Randomize color** to randomly assign colors to additional objects created by splitting.

- **Level 2**:
  - Use **Enforce axis** to choose an axis along which the simplification will be based.

    Select **Automatic** to have the system choose an alignment programmatically, or select **X axis**, **Y axis**, or **Z axis** to align polygons along the selected axis.

  - Use **Points on arc** to set the number of interior points to use for representing curved edges.
  - Use **Length threshold** to specifiy the threshold value for representing curved edges using interior points. The number of points is specified by the **Points on Arc** setting. If a curved edge length, as a percentage of the largest edge in the polygon profile, is higher than the **Length Threshold** percentage, then interior points will be added. Otherwise, the curve is considered too short and will be represented as a straight line.
  - Use **Clean up** to remove details such as rounds before attempting to simplify, making it easier to recognize basic shapes.
  - Use **Allow splitting** to split a body into multiple simplified shapes if possible.

    Use **Randomize color** to randomly assign colors to additional objects created by splitting.

  - Use **Preserve original** to preserve the original body when a body is simplified.

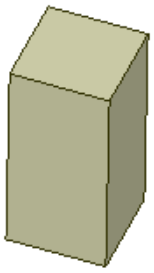
- **Level 3**:

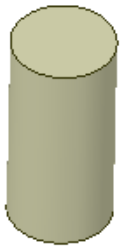
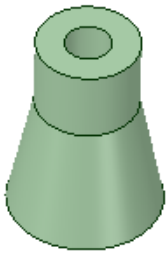
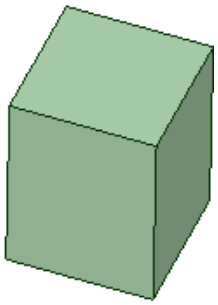
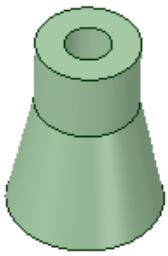
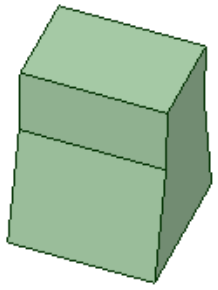
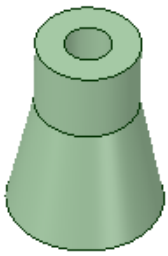
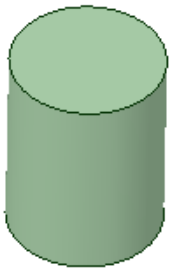
  Use **Facet Quality** to control the fineness of the facets. The facets preview on the object and remain visible until you exit the tool.

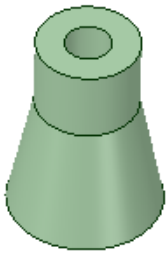
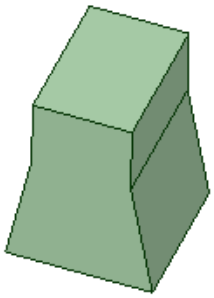
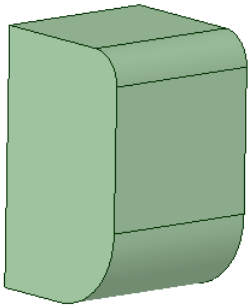
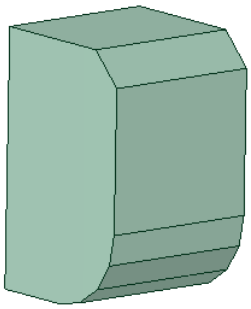
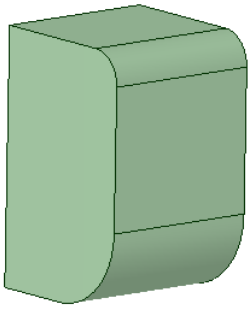
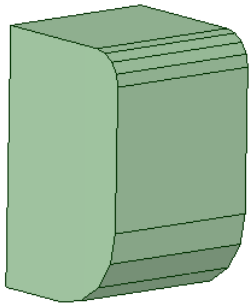
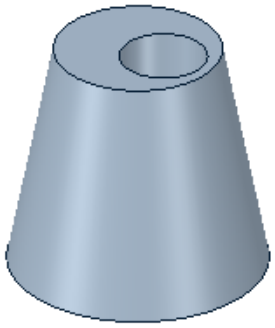
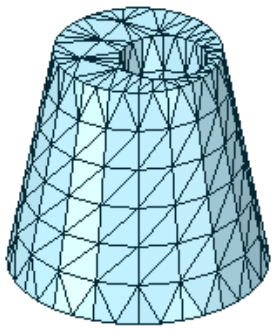
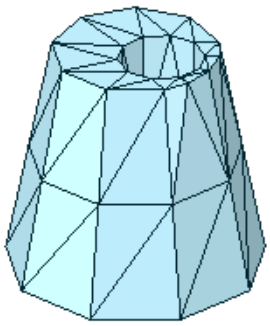
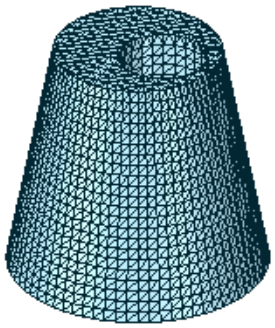
  > **Note:** If you want the highest level of faceting possible, and can accept the performance cost, choose the finest setting in Level 3.

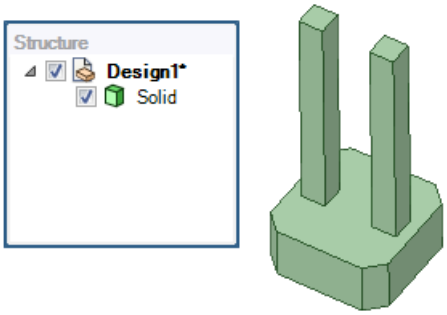
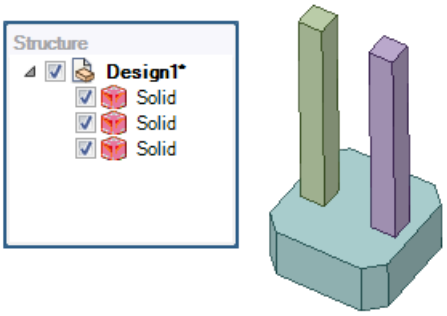
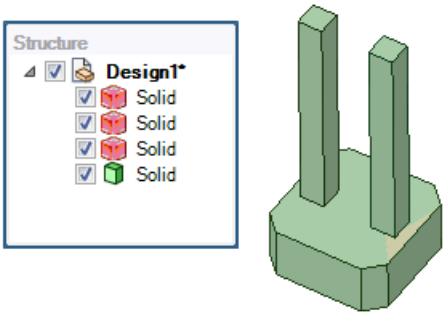
5. Select bodies to simplify. Box-selection is allowed.

13.2.2. Icepak Simplify Tool Examples

Level and Options	Original Body	Simplified Body
Level 0		

Level 1 Cleanup = No		
Level 1 Cleanup = Yes		
Level 2 Enforce Axis = Automatic		
Level 2 Enforce Axis = X axis		
Level 2 Enforce Axis = Y axis		

Level 2 Enforce Axis = Z axis		
Level 2 Enforce Axis = Automatic Points on arc = 3 Length threshold = 20% The larger round uses Points on Arc for faceting.		
Level 2 Points on arc = 3 Length threshold = 10% Decreasing the Length Threshold decreases the round size that uses Points on Arc for faceting.		
Level 3 Facet Quality = Default		
Level 3 Left Image: Facet Quality = Decrease Right Image: Facet Quality = Increase Decreasing Facet Quality results in coarser faceting. Increasing Facet Quality results in finer faceting.		

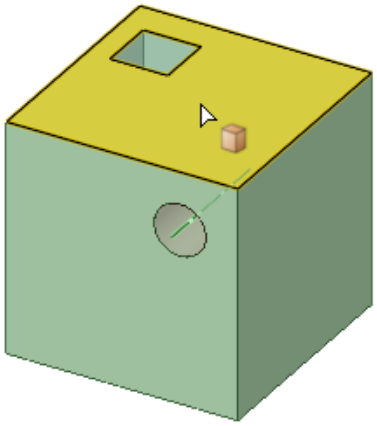
This single body is to be simplified.	
Icepak Simplify deletes the original body by default and, because **Allow splitting** is *ON*, it creates three Icepak objects.	
With **Preserve original** body *ON*, the original solid is preserved.	

13.3. Opening Tool

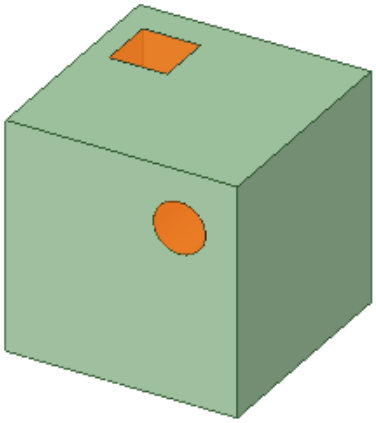
The **Opening** tool creates surface bodies to represent an opening in a body.

13.3.1. Creating an Opening Surface Body

1. Select the **Openings** tool.
2. Select a face with an opening. In the example, there is a rectangular cut through the block and a cylindrical hole partially through the block.



3. The opening surface body is created automatically. In the example, after selecting both the top and the front face, two openings are created.



13.4. Fan Tool

The **Fan** tool converts bodies to Icepak fans.

This tool has two modes.

- One functions like the Midsurface tool, in that pairs of coaxial and opposite-facing faces are Ctrl-selected. On the second selection, the fan is created.
- In the second mode, either the hub (inside) or shroud (outside) face is selected and then the Complete tool guide is clicked to create the fan.

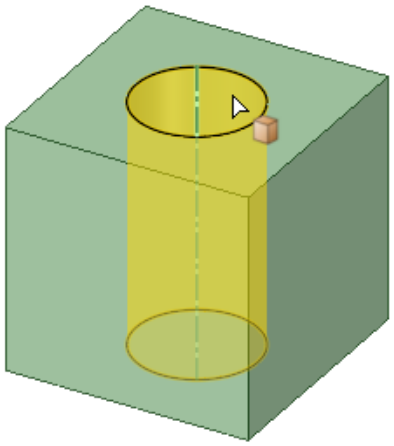
> **Note:** As fans are created, the original geometry is deleted and replaced with simpler bodies, so it is easier to see which fans have been converted. If the case used to create the fan is not square, the tool adjusts it to be square.

This section contains the following topics:

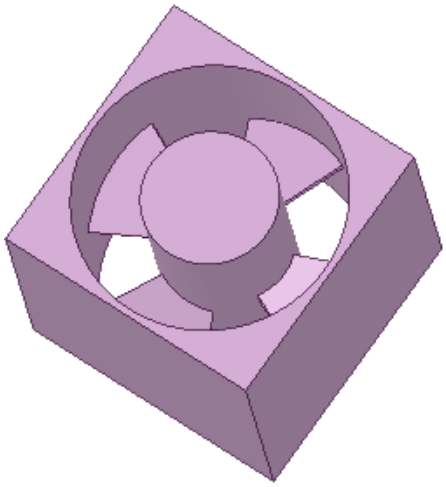
13.4.1. Creating a Fan

1. Select the **Fan** tool.

2. The case (outside) is assumed by rule. This allows for very fast and repeated fan creation.

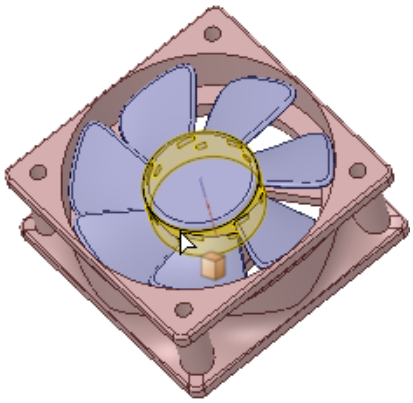


3. Click **Complete** .

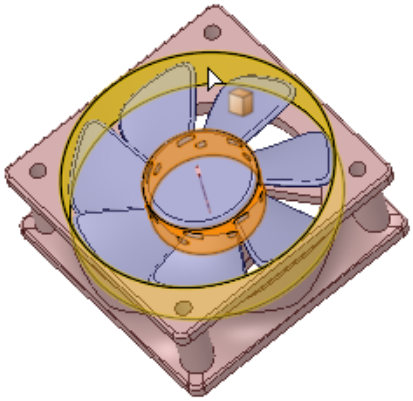


13.4.2. Creating a Fan From a Hub and a Case

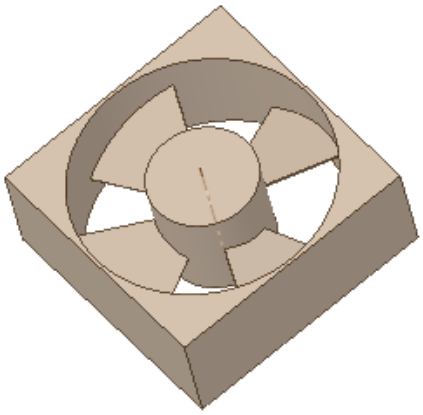
1. Select the **Fan** tool.
2. Select the hub.



3. Press the Ctrl key and select the shroud.



4. The Fan is created automatically. As fans are created, the original geometry is deleted and replaced with simpler bodies so it is easier to see which fans have been converted. If the shroud used to create the Fan is not square, the tool adjusts it to be square.



13.4.3. Fan Properties

Icepak body properties are read-only and include the sizes of the fans, including **Height**, as determined by the geometry or rule. When you create or add an Icepak fan, the following properties display.

Object Type	For fans, the object type is fan.
Shape	For fans, the shape is fan.
Fan Hub Radius	Radius of the fan hub.
Fan Shroud Radius	Radius of the fan shroud.
Fan Case Height	Height of the fan case.

13.5. Grille Tool

The **Grille** tool creates an Icepak Grille on a selected face.

There are several **Grille** types available in the **Options** panel.

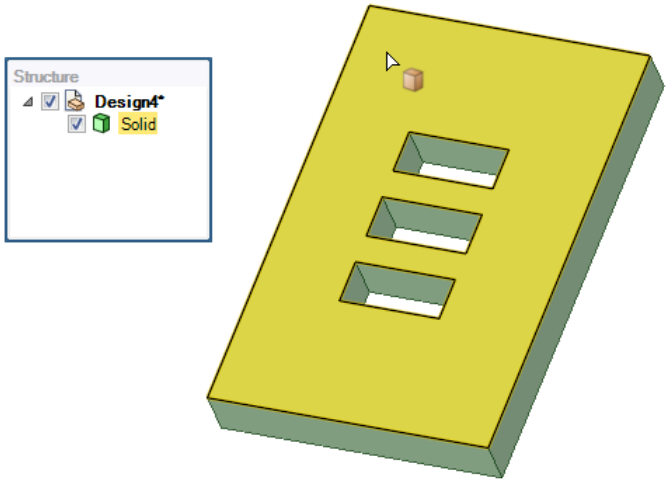
- **Rectangular**
- **Circular**

- **Polygonal**
- **Inclined Rectangular**
- **Automatic**
- **Preserve openings** allows the original openings to be preserved when a grille is created.

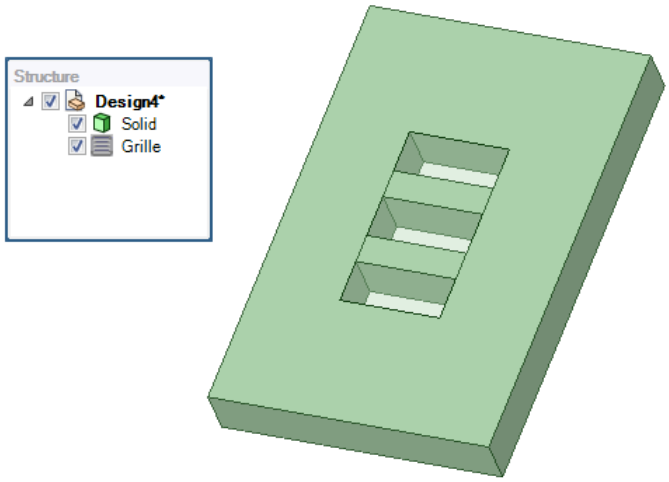
This section contains the following topics:

13.5.1. Creating a Rectangular Grille

1. Select the **Grille** tool.
2. Select **Rectangular** in the Options panel.
   By default, the **Select Opening Bodies** tool guide is active.
3. Click the **Select Face** tool guide and select the face on which to create the Grille.

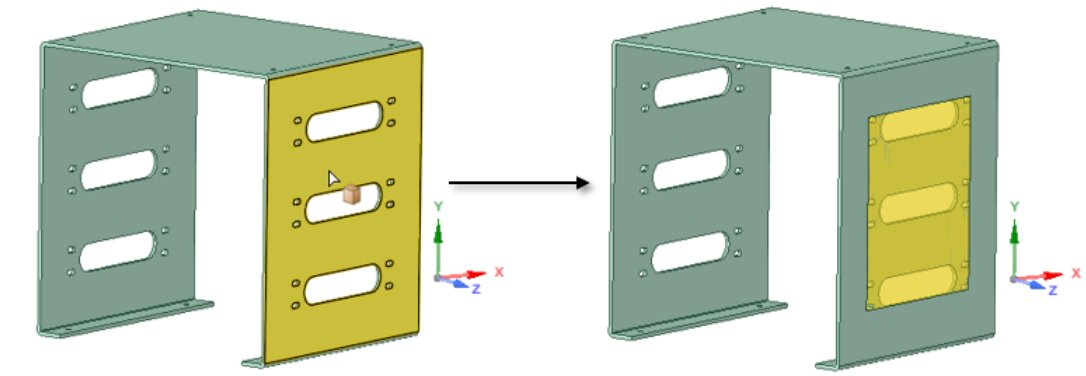


4. Click **Complete** to create the Grille.

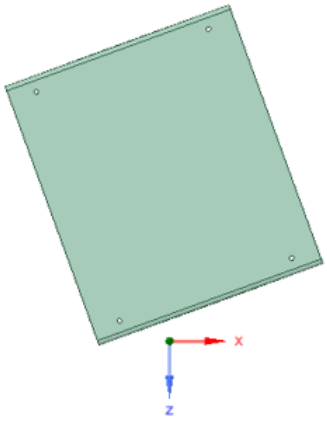


Inclined Grilles are also supported for surface bodies. Inclined means that the openings are not aligned with the axes of the World Origin. The workflow is the same as for Rectangular Grilles.

The following example shows openings aligned with the World Origin Z-axis.

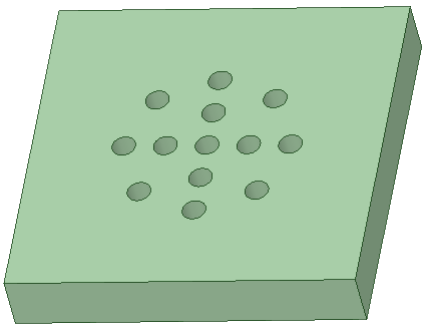


When viewed from the top, it is clear the part is not aligned with the World Origin axes.

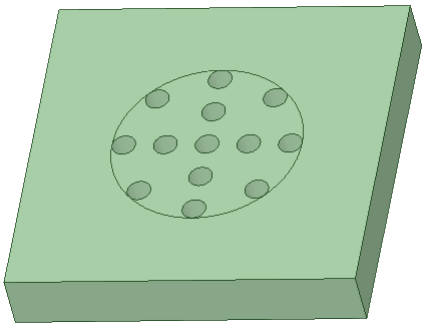


13.5.2. Creating a Circular Grille

1. Select the **Grille** tool.
2. Select **Circular** in the Options panel.
   By default, the **Select Opening Bodies** tool guide is active.
3. Click the **Select Face** tool guide and select the face on which to create the Grille.

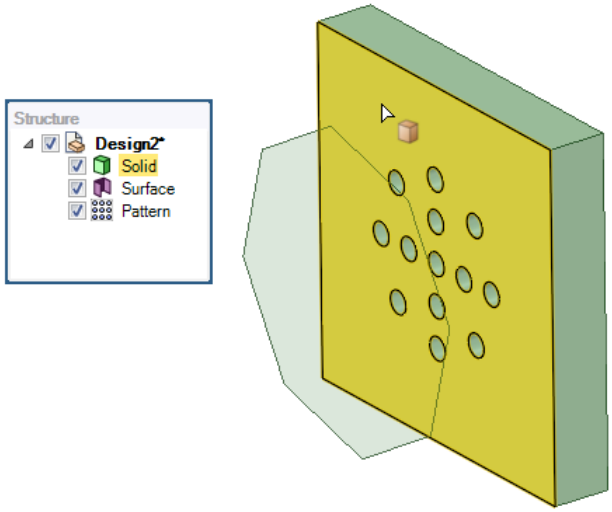


4. Click **Complete** to create the Grille.

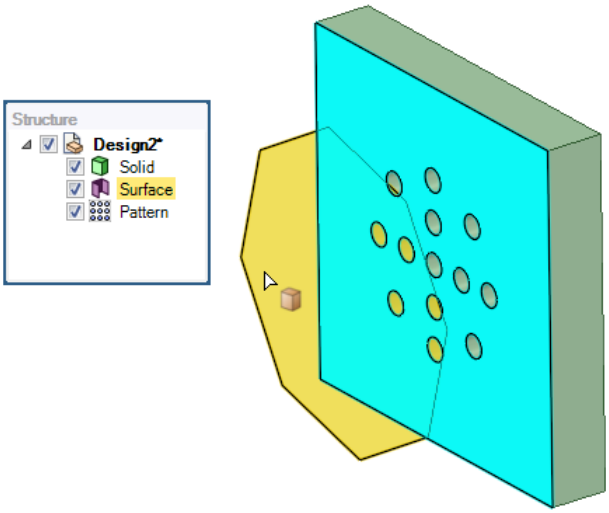


13.5.3. Creating a Polygonal Grille

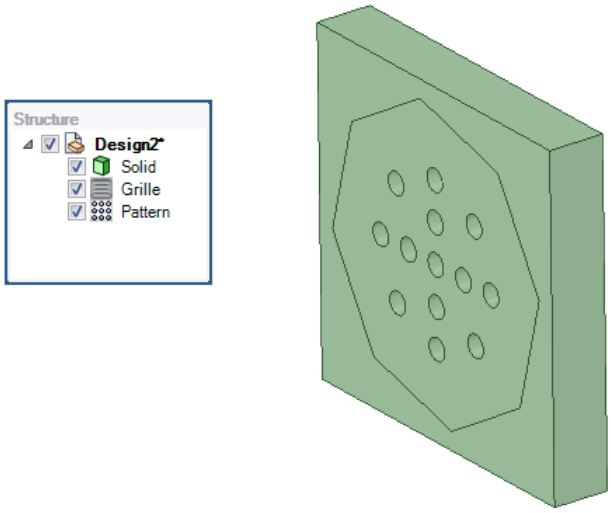
1. Select the **Grille** tool.
2. Select **Polygonal** in the Options panel.
3. By default, the **Select Opening Bodies** tool guide is active.
4. Click **Select Face** and select the face on which to create the Grille.



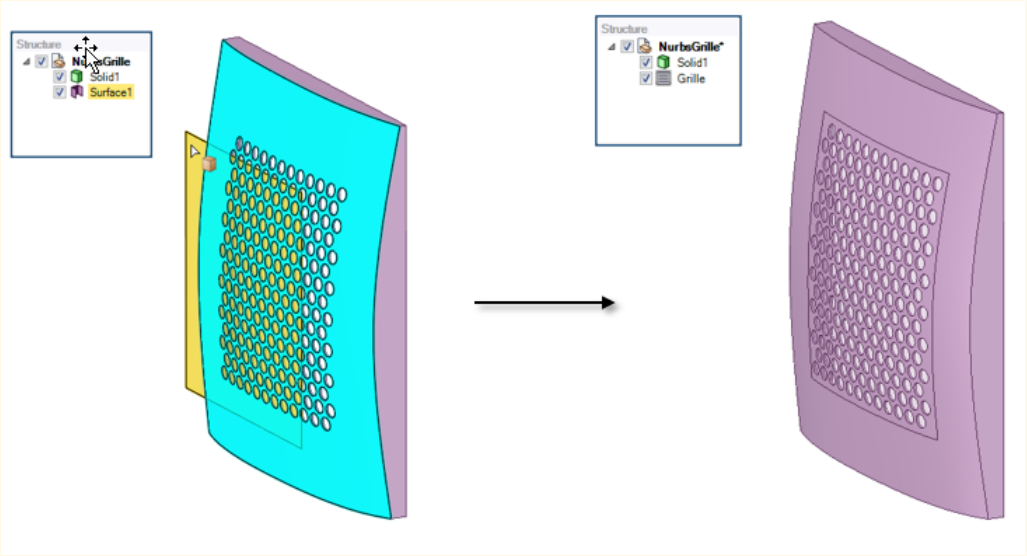
5. The **Select Polygon** tool guide is activated.
   Select a body to serve as the polygon.



6. Click **Complete** to create the Grille.



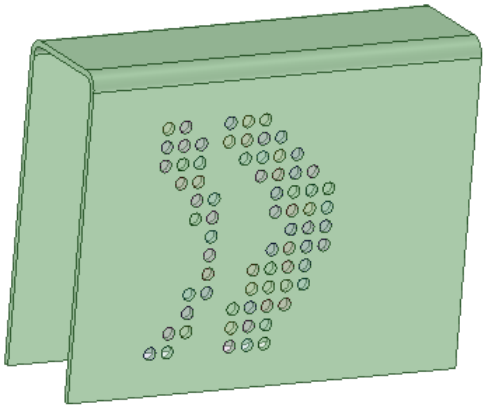
Note: The polygonal shape can also be projected onto curved surfaces.

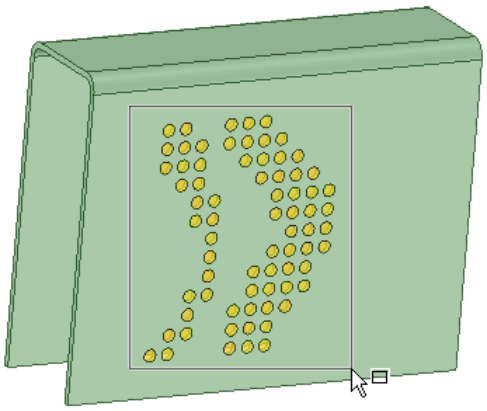


13.5.4. Creating an Automatic Grille

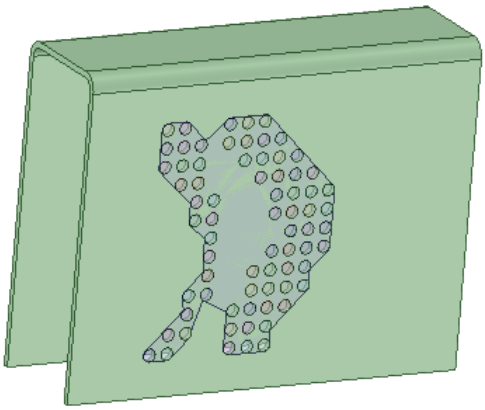
1. Select the **Grille** tool.
2. Choose **Automatic** in the Options panel
3. Select all of the openings.

> **Note:** This part has several holes that were converted to Openings using the **Opening** tool.





The Grille is created with a best-fit polygon surrounding the Openings.



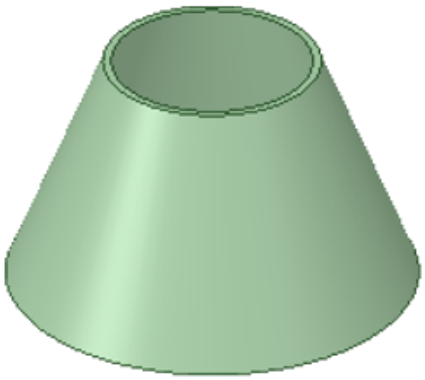
13.6. Enclosure Tool

The **Enclosure** tool converts objects to Icepak enclosures. It approximates a box-shaped object as an open box based on any open geometry found by the tool.

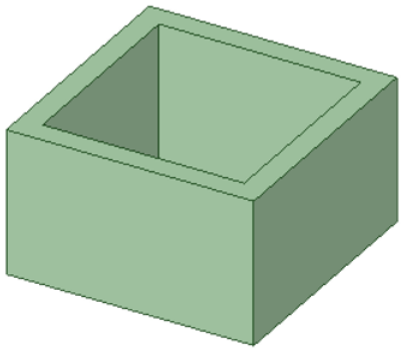
This section contains the following topics:

13.6.1. Creating an Enclosure From a Box-Shaped Object

1. Select the **Enclosure** tool.
2. Select a box-shaped object.



3. The Enclosure is created automatically.



After conversion, the solid's icon changes in the design tree.

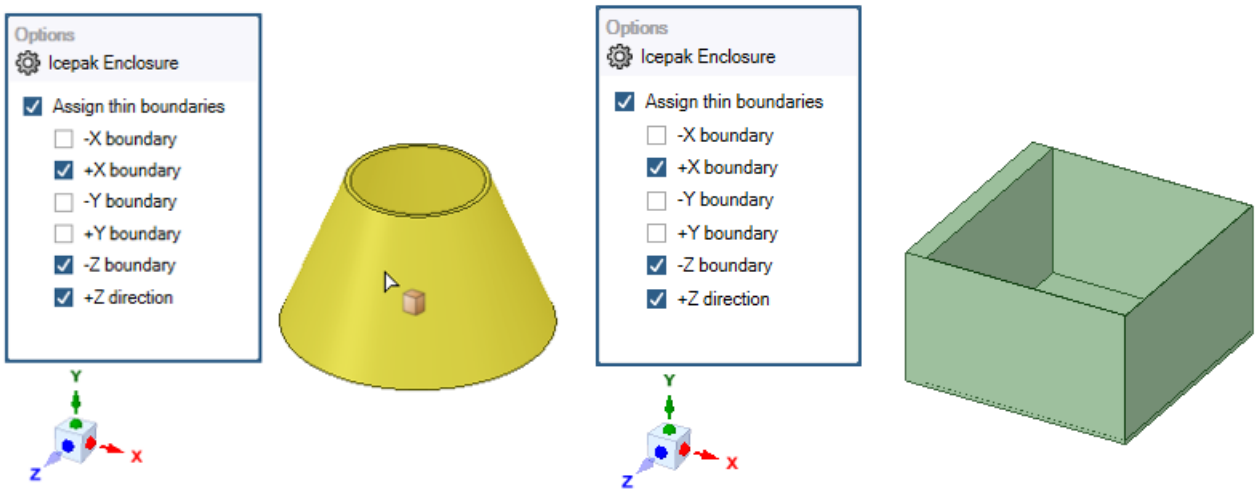


13.6.2. Enclosure Tool Options

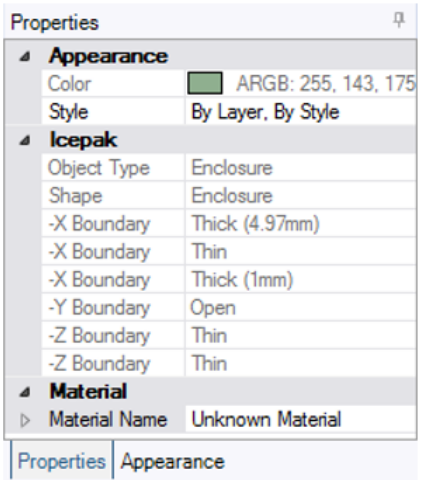
The **Enclosure** tool options panel has an **Assign thin boundaries** option, which will make any checked boundaries thin.

Assign thin boundaries	Check to allow selection of individual boundary options.
-X boundary	Check to assign -X as a thin boundary.
+X boundary	Check to assign +X as a thin boundary.
-Y boundary	Check to assign -Y as a thin boundary.
+Y boundary	Check to assign +Y as a thin boundary.
-Z boundary	Check to assign -Z as a thin boundary.
+Z boundary	Check to assign +Z as a thin boundary.
Preserve original	Check to keep original object shape along with new enclosure.

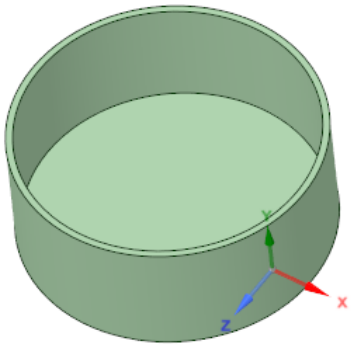
In the example below, the XYZ Direction gizmo illustrates the meaning of the boundary labels.

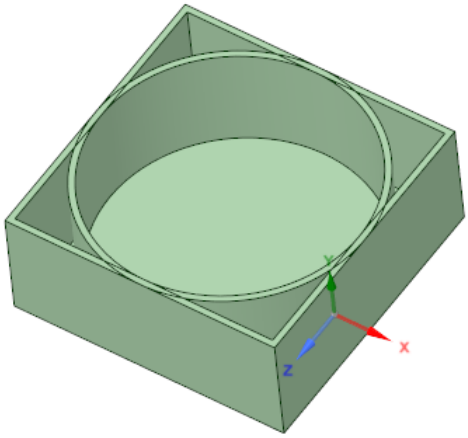


Read-only displays of the boundaries are included in the enclosure's **Properties** panel.



Enable **Preserve original** in **Options** to keep the original shape along with the new enclosure.




Y
X
Z

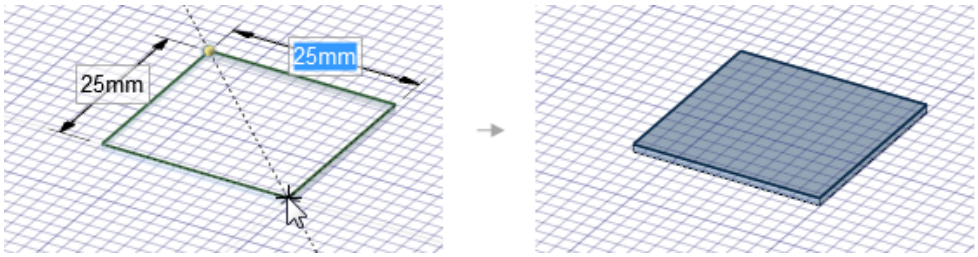
14: Sheet Metal

14.1. Creating and Editing Sheet Metal

Create a new sheet metal design using the tools in the **Sheet Metal** tab. See Converting a design to sheet metal if you have already modeled the part.

To create a new sheet metal design:

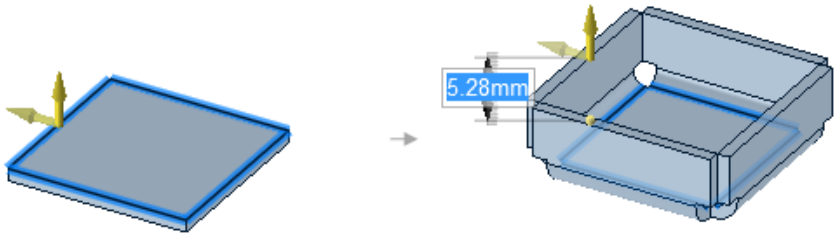
1. Click the **Sheet Metal** tab.
2. Select a tool in the **Sketch** group.
3. Sketch the first wall of your sheet metal part:



You can see a preview of the wall's thickness as you sketch.

See Sketching sheet metal parts for detailed information and instructions.

4. Select **Pull** in the **Sheet Metal** tab.
5. Select an edge or edges and pull side walls on your part:



Junctions, corner reliefs, and bends are automatically created as needed.

See Pulling sheet metal parts for detailed information and instructions.

14.1.1. Highlighting Objects in a Sheet Metal Design

Sheet metal objects such as junctions, reliefs, notches, and others are highlighted using specific colors.

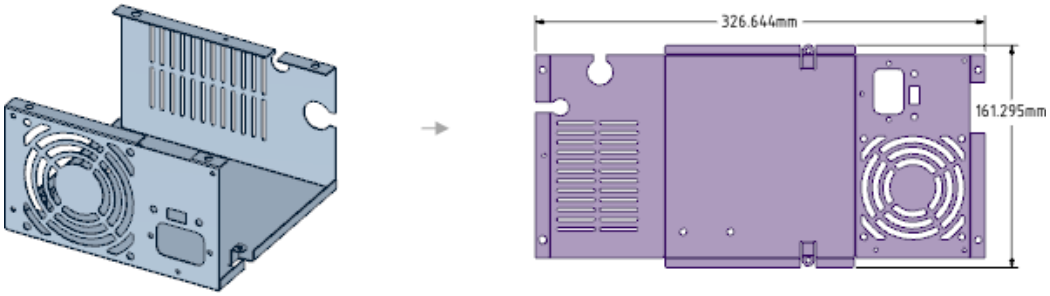
You can customize the highlighting colors in the Sheet Metal Options.

14.2. Working with Sheet Metal

You can use the Discovery application to create sheet metal designs and components.

The application recognizes a component as sheet metal if it consists of only planar, cylindrical, and conical faces, and when faces are offsets of the same size.

Sheet metal parts are displayed in azure (light blue) by default. Unfolded parts are displayed in purple, as shown in the image below.



Unfolded parts have the following default layers:

- Dimension
- Bend Lines Up
- Bend Lines Down
- Bend Dimensions
- Forms
- Markers

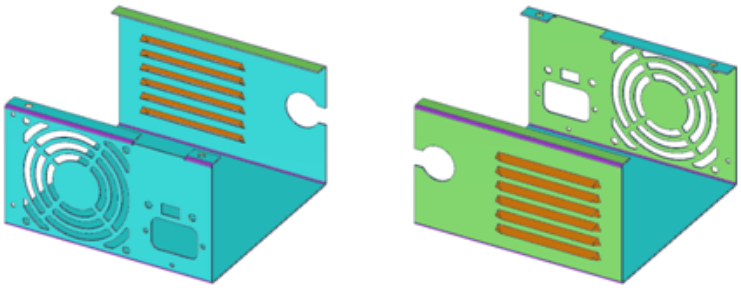
14.3. Importing Sheet Metal Designs

14.3.1. Convert Tool

Use the **Convert** tool in the **Import** ribbon group on the **Sheet Metal** tab to convert an existing design to sheet metal within the Discovery application. You may need to convert if you imported regular geometry or if you moved a part to another component.

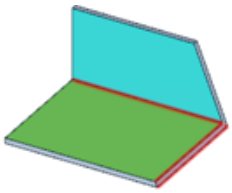
When you click the **Convert** tool and select a surface body of a design, the surface body automatically thickens into a sheet metal body at the default sheet metal thickness. This enhancement eliminates the need to redo a surface you may have sketched prior to converting the design to sheet metal. After you convert the body, you can use the Identify tool to call out forms, joggles, hems, etc.

Walls, bends, junctions (only created, default-sized junctions, not imported junctions), and forms that were made in Sheet Metal are identified by color: Faces are blue, junctions are purple, forms are orange, partial bends are yellow, and edges of end faces that are not square are red. Identification is important because it allows the element to be unbent or flattened when you unfold the part.



14.3.1.1. Converting a Solid to Sheet Metal

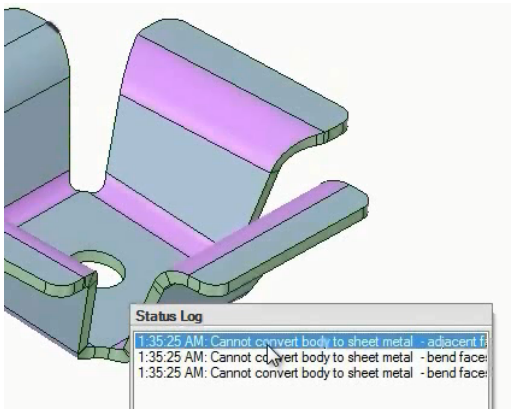
1. Select **Convert** in the **Import** group on the **Sheet Metal** tab.
   The **Select Bodies** tool guide is enabled by default.
2. Select the body you want to convert.
   You can use **Ctrl+click** to select multiple bodies.
   Walls, bends, and junctions (only default-sized junctions you created as sheet metal, not imported junctions) are automatically detected and identified by color. Faces that are not separated by the default thickness are not highlighted. Edges shown in red indicate end faces that are not square, as shown in the following figure.



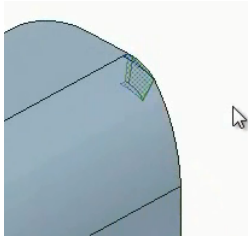
3. The **Assign Objects** tool guide is enabled for you to convert hard edges to junctions:
   a) Select an option from the **Junction** tool in the **Create** group on the **Sheet Metal** tab.
   b) Select the hard edges that you want to convert to the current junction type.
   Clicking an edge shown in red creates a junction or squares up a sharp face. The edge you click to square a face determines the final length of the sheet metal wall.

When you convert to sheet metal, faces that have problems are identified with error messages in the Status log. Selecting each message highlights the problem.

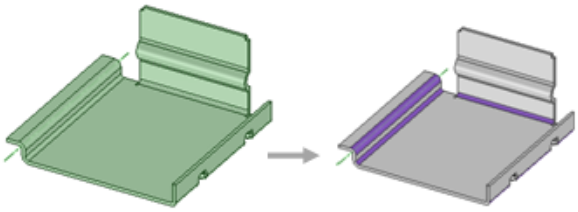
In the example below, the message is selected and the problem is highlighted in the upper left corner of the image.



A detailed view of the problem geometry is shown below.



This example shows the converting of a sheet metal design with bends automatically detected.



14.3.1.2. Convert Tool Guides

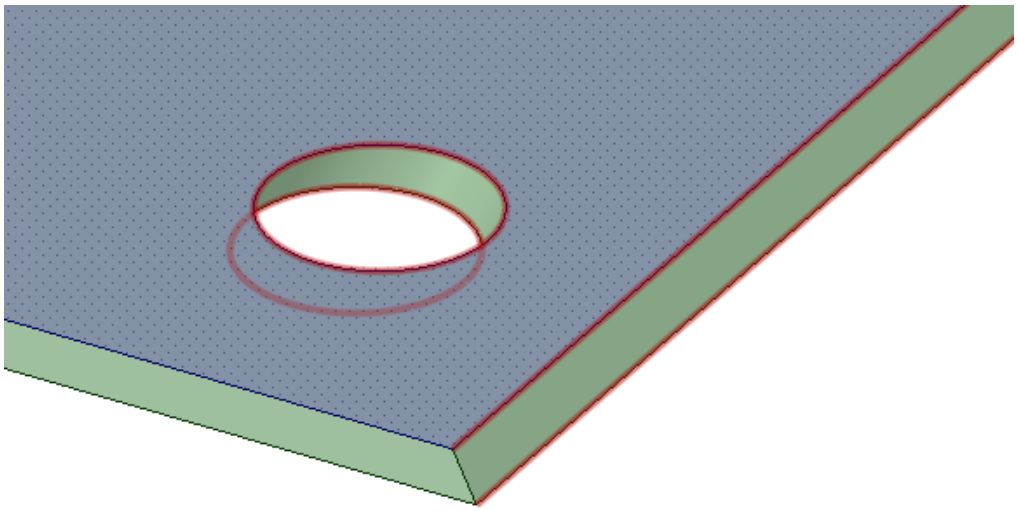
The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Bodies	Use the **Select Bodies** tool guide to select the part(s) you want to convert to sheet metal.
Assign Objects	Use the **Assign Objects** tool guide to assign or change the junction types, reliefs, and notches. You can only select geometry that is appropriate for the tool guide.

14.3.2. Square-Up Tool

After you have converted a design to sheet metal, you can use the **Square-Up** tool in the Import ribbon group on the **Sheet Metal** tab to find and fix edges and faces that are not square to the Sheet Metal sheet.

This image shows a simple example. The circular hole and right thickness face are not perpendicular to the sheet. The Square-Up tool finds this geometry and highlights it.



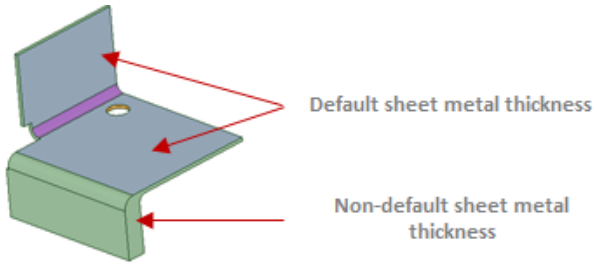
14.3.2.1. Squaring up Sheet Metal Geometry

1. Select **Square-Up** in the **Import** group in the **Sheet Metal** tab.
   Geometry that can be squared-up is highlighted in red.
2. Select the highlighted geometry to fix:
   - If you select an edge, that edge remains fixed and the opposite edge is moved to line up with the selected edge
   - If you select a face, all edges misaligned on the opposite face are moved.

14.3.3. Identify Tool

When you use the Convert tool to convert a body to sheet metal, or, are working with an existing sheet metal body, only the sheet metal faces that are separated by the default sheet metal thickness are shown in blue.

In the example, sheet metal that varies in thickness from the default does not display in blue.



After you have converted a design to sheet metal, you can use the **Identify** tool in the **Import** ribbon group in the **Sheet Metal** tab to call out sheet metal geometry such as forms, hems, joggles, beads, notches, and gussets.

You can also use **Select** > **Using Box** to identify sheet metal geometry. Box selecting selects front and back faces of the pre-identified sheet metal geometry.

Some geometry is automatically identified and other geometry must be manually assigned. As the geometry is identified, the highlighting color changes to indicate.the sheet metal objects.

Objects in a converted sheet metal part must be identified before they can be unfolded.

14.3.3.1. Identifying Sheet Metal Geometry

1. Select **Identify** in the **Import** group in the **Sheet Metal** tab.
2. Identify each of the following objects as needed by clicking the appropriate tool guide. Click **Complete** when finished.
   - **Beads**: Select highlighted beads or select the faces to identify as a bead.
   - **Forms**: Select highlighted forms or select the faces to identify as a form.
   - **Formed Bends**: Select highlighted formed bends or select the faces to identify as a formed bend.
   - **Gussets**: Select highlighted gussets or select the faces to identify as a gusset.
   - **Hems**: Select highlighted hems or select the faces to identify as a hem.
   - **Joggles**: Select the faces to identify as a joggle.

- **Notches**: Select highlighted notches or select the faces to identify as a notch.
- **Tabs**: Select highlighted tabs or select the faces to identify as a tab.
- **Reliefs**: Select highlighted reliefs or select the faces to identify as a relief.

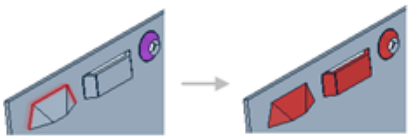
14.3.3.2. Identify Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

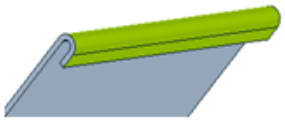
Beads	Use the **Beads** tool guide to find and identify beads.
Forms	Use the **Forms** tool guide to find and identify formed sheet metal details on a part.
Formed Bends	Use the **Formed Bends** tool guide to find and identify formed bends.
Gussets	Use the **Gussets** tool guide to find and identify gussets.
Hems	Use the **Hems** tool guide to find and identify hems.
Joggles	Use the **Joggles** tool guide to select faces to identify as joggles.
Notches	Use the **Notches** tool guide to find and identify notches.
Tabs	Use the **Tabs** tool guide to identify tabs.
Reliefs	Use the **Reliefs** tool guide to identify reliefs.
Complete	Use the **Complete** tool guide to complete each identification.

14.3.3.3. Identify Tool Examples

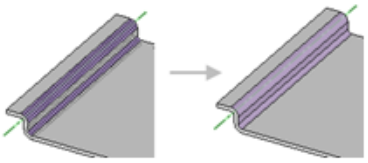
Automatically identifying forms.



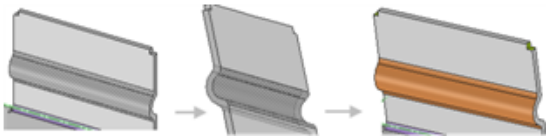
Automatically identifying a hem.



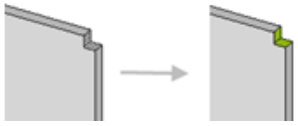
Selecting two bends to identify a joggle.



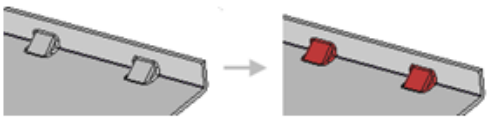
Selecting the front faces and back faces to identify a bead.



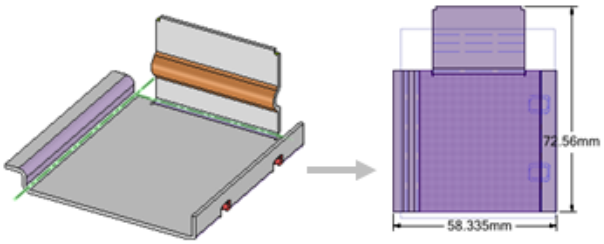
Selecting the thickness edges to identify a notch.



Automatically identifying gussets.



Unfolding a sheet metal design after identifying the geometry.



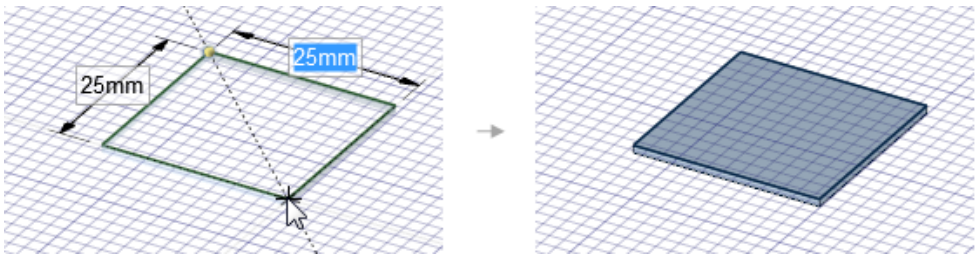
14.4. Sketching Sheet Metal Parts

The tools in the **Sketch** group in the **Sheet Metal** tab are a little different from normal sketch tools:

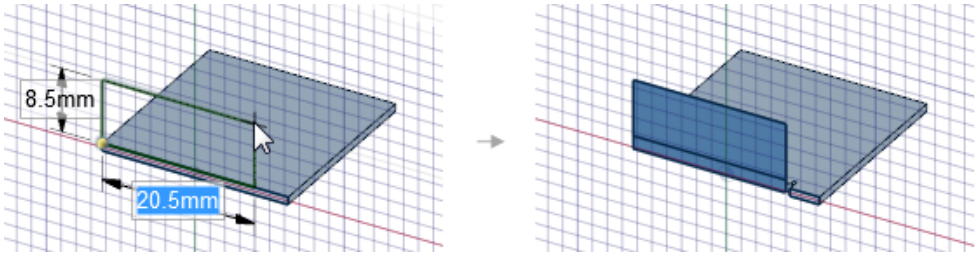
Rectangles, circles, polygons, and ellipses are automatically extruded to the thickness of a sheet metal wall as you sketch. See Sheet Metal Options to change the default wall thickness. Lines, splines, and arcs are extruded when they form a closed profile.

Sketching in empty space creates new walls.

A preview is displayed as you sketch that shows you the wall thickness for each sketch curve:

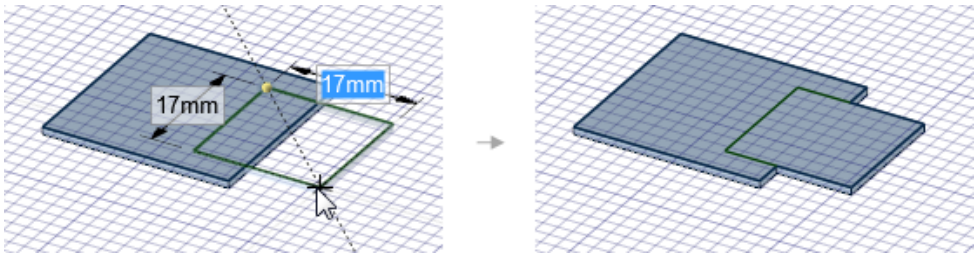


Sketching perpendicular to an existing wall creates a new wall with a bend between the walls with corner or rip reliefs where needed. The bend radius is created inside when the sketch is connected to the top edge and outside when the sketch is connected to the bottom edge:

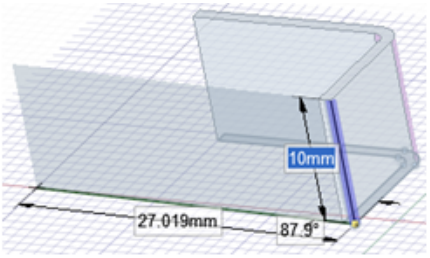


Sketching on an existing wall creates sketch curves on the wall that you can use with the **Bend** and **Split** tools.

Sketching adjacent to or overlapping an existing wall adds material to the wall and may imprint the overlapping curves on the wall:



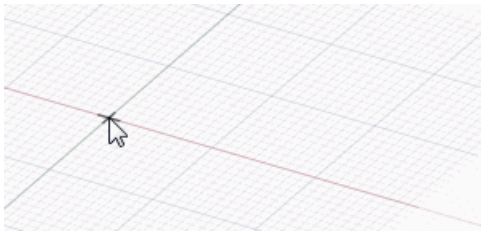
In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch:



14.4.1. Line Tool

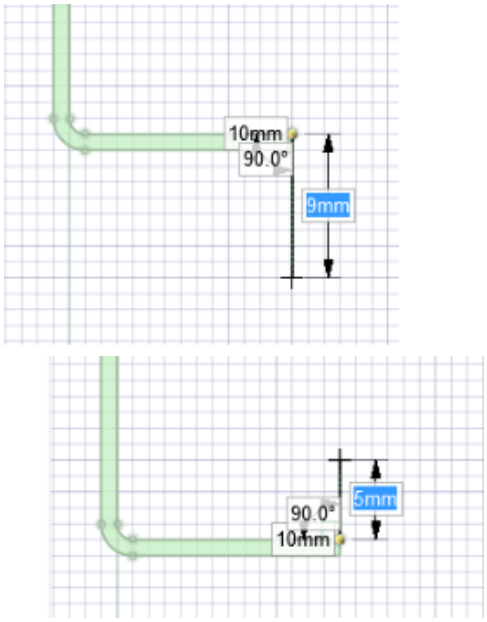
The **Sheet Metal Line** tool is primarily used to sketch freeform shapes and to draw lines that will become bends. See Bending a sheet metal wall.

Lines, splines, and arcs are extruded to the thickness of a sheet metal wall when they form a closed profile. You can change the default wall thickness in the sheet metal properties.



In addition, you can use the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

When sketching lines for a profile wall, the attachment edge for determining inside or outside radius follows the same paradigm as flat-sketched walls. In the images below, both lines start at the top corner. Sketching vertically downward creates an outside wall, while sketching vertically upward creates an inside wall.



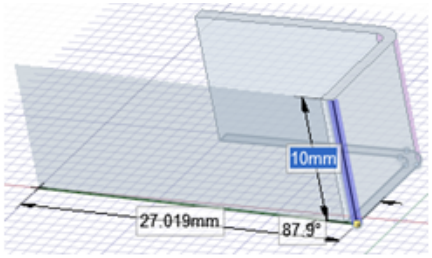
14.4.1.1. Sketching a Line in Sheet Metal

1. Select **Line** in the **Sketch** group or press **L**.
2. (Optional) Click the **Profile** option if you want to create walls as you sketch.
3. Select the first point of the line.

   You can dimension the points relative to other sketch objects.
4. Select the next points of the line.

   If you want any section of the line to be an arc, right-click and select **Switch to Arc**, then click to set the radius. Right-click and select **Switch to Line** to return to drawing straight lines between points. You must create at least one line segment before using this option.
5. End the line:
   - Double-click to set the end point of the line.
   - Right-click and select **Finish Line**.
   - Press **Esc**.
   - Connect the end point to the start point.
   - Click any tool (except the Clipboard and Orient tools).

To do it faster, click and drag to draw one straight line.



This example shows the sketching of a polyline using the **Profile** Sheet metal option.

14.4.1.2. Line Tool Options

The following additional options are available:

Draw from center	Select this option to sketch a line from the center. Click to define the center of the line, then click again to set the length. You can also drag to draw the line.

The following options are available for every sketch tool:

Snap to grid	Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See Units options to change the minor grid spacing.
Snap to angle	Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Curve Fitter Options	If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves. • **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points. • **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated. • **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

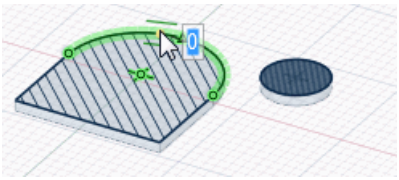
The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
Flat	Any closed sketch creates a flat wall.
Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
Inside	Maintains the length of the sheet's inside surface when bending.
Outside	Maintains the length of the sheet's outside surface when bending.

14.4.2. Tangent Line Tool

Use the **Sheet Metal Tangent Line** tool to sketch lines that are tangent to curves in your design.

Lines, splines, and arcs are extruded to the thickness of a sheet metal wall when they form a closed profile. You can change the default wall thickness in the sheet metal properties.



In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

14.4.2.1. Sketching a Tangent Line in Sheet Metal

1. Select **Tangent Line** in the **Sketch** group.

   Curves are highlighted as you hover over them, and the tangency indicator appears at your cursor location. If there are no curves in your design, this tool is disabled. You must add a curve to create a tangent line.
2. (Optional) Click the **Profile** option if you want to create walls as you sketch.
3. Click the curve you want to draw tangent to.

   As you move the mouse, the start point moves so that the line remains tangent to the curve.
4. (Optional) Dimension the line with a length.

   You cannot dimension from another sketch object when creating a tangent line.
5. Click to set the end point of the line.

   If you move your mouse over another curve, the line snaps so that it is tangent to the second curve. Hold **Alt** to stop your cursor from snapping to curves.

14.4.2.2. Tangent Line Tool Options

The following options are available for every sketch tool:

Snap to grid	Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See Units options to change the minor grid spacing.
Snap to angle	Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Curve Fitter Options	If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves. • **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points. • **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated. • **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

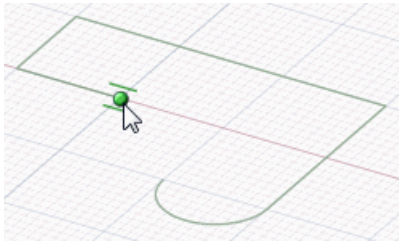
The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
Flat	Any closed sketch creates a flat wall.
Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
Inside	Maintains the length of the sheet's inside surface when bending.
Outside	Maintains the length of the sheet's outside surface when bending.

14.4.3. Tangent Arc Tool

Use the **Sheet Metal Tangent Arc** tool to sketch an arc that is tangent to a curve or line in your design.

Lines, splines, and arcs are extruded to the thickness of a sheet metal wall when they form a closed profile. You can change the default wall thickness in the sheet metal properties.



In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

14.4.3.1. Sketching a Tangent Arc in Sheet Metal

1. Select **Tangent Arc** in the **Sketch** group.
   The tool is disabled if there are no curves or lines in the sketch plane.
2. (Optional) Click the **Profile** option if you want to create walls as you sketch.
3. Click the line or curve you want to draw tangent to.
   This is usually done at the end point of a line, arc, or spline, but it can be on the line. If two lines share an end point, use the scroll wheel to set tangency to the other line.
4. (Optional) Dimension the radius and chord angle.
5. Click to set the end point of the arc.
   The arc cannot end on its start point to make a circle or end on the same line as its start point.

14.4.3.2. Tangent Arc Tool Options

The following options are available for every sketch tool:

Snap to grid	Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See Units options to change the minor grid spacing.

Snap to angle	Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Curve Fitter Options	If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves. • **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points. • **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated. • **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

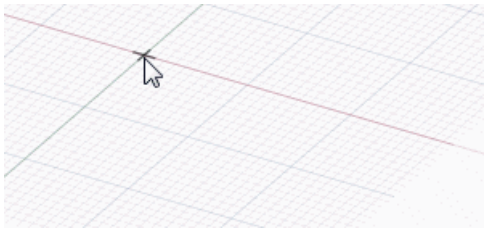
The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
Flat	Any closed sketch creates a flat wall.
Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
Inside	Maintains the length of the sheet's inside surface when bending.
Outside	Maintains the length of the sheet's outside surface when bending.

14.4.4. Rectangle Tool

The **Sheet Metal Rectangle** tool is most often used to draw the base of a sheet metal part. After you sketch a base, you can use the Pull tool to create walls perpendicular to the rectangle.

Rectangles, circles, polygons, and ellipses are automatically extruded to the thickness of a sheet metal wall as you sketch. You can change the default wall thickness in the sheet metal properties.



In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

14.4.4.1. Sketching a Rectangle in Sheet Metal

1. Select **Rectangle** in the Sketch group or press **R**.

2. (Optional) Select **Draw from center** from the Options to start drawing the rectangle by clicking the point for its center instead of a corner.
3. (Optional) Click the **Profile** option if you want to create walls as you sketch.
4. Click to set the first corner.

   Move your mouse over the sketch grid to preview the rectangle. Dashed lines appear when you create a square or golden rectangle.

   You can dimension the points relative to other sketch objects.
5. Click to set the opposite corner of the rectangle.

14.4.4.2. Rectangle Tool Options

The following additional options are available:

Draw from center	Select this option to sketch rectangles from their centers. Click to define the center of the rectangle, then click again to set the length of the sides. You can also drag to draw the rectangle. Hold the Alt key while drawing a rectangle to toggle this option on-the-fly.

The following options are available for every sketch tool:

Snap to grid	Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See Units options to change the minor grid spacing.
Snap to angle	Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Curve Fitter Options	If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves. • **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points. • **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated. • **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

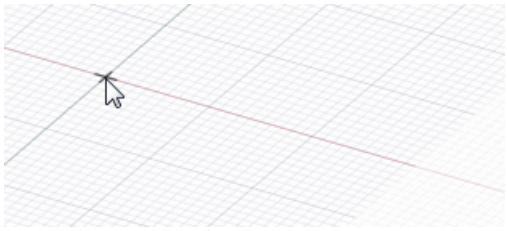
Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
Flat	Any closed sketch creates a flat wall.
Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
Inside	Maintains the length of the sheet's inside surface when bending.

Outside	Maintains the length of the sheet's outside surface when bending.

14.4.5. Three-Point Rectangle Tool

The **Sheet Metal Three-Point Rectangle** tool is used to draw a rectangle at any angle on the sketch plane.

Rectangles, circles, polygons, and ellipses are automatically extruded to the thickness of a sheet metal wall as you sketch. You can change the default wall thickness in the sheet metal properties.



In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

14.4.5.1. Sketching a Three-Point Rectangle in Sheet Metal

1. Select **Three-Point Rectangle** in the **Sketch** group.
2. (Optional) Select **Draw from center** from the Options to start drawing the rectangle by clicking the point for its center instead of a corner.
3. (Optional) Click the **Profile** option if you want to create walls as you sketch.
4. Click to set the first corner of the rectangle.

   Move your mouse over the sketch grid to preview the rectangle. Dashed lines appear when you create a square or golden rectangle.

   You can dimension the points relative to other sketch objects.
5. Click to set the length of the second side.

> **Note:** Click and drag to draw the first side, then click to set the length of the second side.

14.4.5.2. Three-Point Rectangle Tool Options

The following additional options are available:

Draw from center	Select this option to sketch rectangles from their centers. Click to define the center of the rectangle, then click again to set the length of the sides. You can also drag to draw the rectangle. Hold the Alt key while drawing a rectangle to toggle this option on-the-fly.

The following options are available for every sketch tool:

Snap to grid	Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See Units options to change the minor grid spacing.

Snap to angle	Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Curve Fitter Options	If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves. • **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points. • **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated. • **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

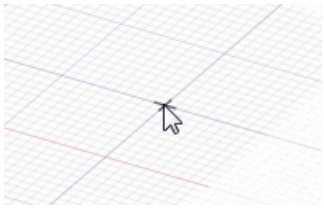
The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
Flat	Any closed sketch creates a flat wall.
Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
Inside	Maintains the length of the sheet's inside surface when bending.
Outside	Maintains the length of the sheet's outside surface when bending.

14.4.6. Ellipse Tool

Use the **Sheet Metal Ellipse** tool to draw ellipses in sheet metal.

Rectangles, circles, polygons, and ellipses are automatically extruded to the thickness of a sheet metal wall as you sketch. You can change the default wall thickness in the sheet metal properties.



In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

14.4.6.1. Sketching an Ellipse in Sheet Metal

1. Select **Ellipse** in the **Sketch** group.
2. (Optional) Click the **Profile** option if you want to create walls as you sketch.

3. Click to set the center of the ellipse.
   You can dimension the points relative to other sketch objects.
4. Click to set the overall length and angular orientation of the first axis.
5. Click to set the length of the second axis.

14.4.6.2. Ellipse Tool Options

The following options are available for every sketch tool:

Snap to grid	Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See Units options to change the minor grid spacing.
Snap to angle	Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Curve Fitter Options	If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves. • **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points. • **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated. • **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

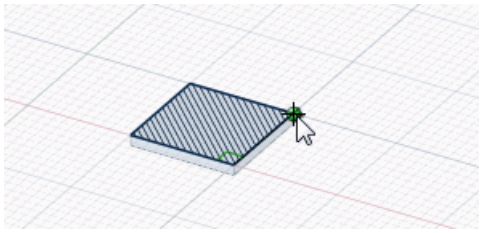
The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
Flat	Any closed sketch creates a flat wall.
Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
Inside	Maintains the length of the sheet's inside surface when bending.
Outside	Maintains the length of the sheet's outside surface when bending.

14.4.7. Sweep Arc Tool

Use the **Sheet Metal Sweep Arc** tool to sketch an arc when you know the location of its center and end points.

Lines, splines, and arcs are extruded to the thickness of a sheet metal wall when they form a closed profile. You can change the default wall thickness in the sheet metal properties.



In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

14.4.7.1. Sketching a Sweep Arc in Sheet Metal

1. Select **Sweep Arc** in the **Sketch** group.
2. Click to set the center of the arc.
   You can dimension the point relative to other sketch objects.
3. (Optional) Click the **Profile** option if you want to create walls as you sketch.
4. Click to set the start point and radius of the sweep circle.
   You can dimension the chord angle or offset the angular dimension.
5. Click to set the end point of the arc.

14.4.7.2. Sweep Arc Tool Options

The following options are available for every sketch tool:

Snap to grid	Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See Units options to change the minor grid spacing.
Snap to angle	Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Curve Fitter Options	If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves. • **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points. • **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated. • **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

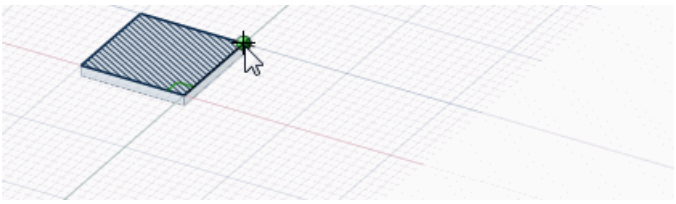
Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
Flat	Any closed sketch creates a flat wall.

Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
Inside	Maintains the length of the sheet's inside surface when bending.
Outside	Maintains the length of the sheet's outside surface when bending.

14.4.8. Three-Point Circle Tool

Use the **Sheet Metal Three-Point Circle** tool to draw circles when you know the location of the edge of the circle but you don't know the location for its center, or when you want to draw a circle that is tangent to two objects.

Rectangles, circles, polygons, and ellipses are automatically extruded to the thickness of a sheet metal wall as you sketch. You can change the default wall thickness in the sheet metal properties.



In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

14.4.8.1. Sketching a Three-Point Circle in Sheet Metal

1. Select **Three-Point Circle** in the **Sketch** group.
2. (Optional) Select **Three-point circle segment** from the Options to create an arc that is a segment of a three-point circle.
3. (Optional) Click the **Profile** option if you want to create walls as you sketch.
4. Click to set the first point on the circle's edge.

   If you click a curve or line, the circle will be drawn tangent to the curve or line, unless you click the midpoint or vertex.

   You can dimension the points relative to other sketch objects.
5. Click to set the second point on the circle's edge.

   If the circle disappears as you move your mouse over the sketch grid, then the cursor location cannot be included in any circle drawn through the first two points and the current point. If you click a curve or line, the circle will be drawn tangent to the curve or line unless you click the midpoint or vertex.
6. Click to set the last point on the circle's edge.

14.4.8.2. Three-Point Circle Tool Options

The following additional options are available in the **Options** panel:

Three-point circle segment: Check this option to create an arc that is a segment of a three-point circle. To create an arc with this option, click to set the first point, click to set the second point, then enter the diameter or click to set the final point.

The following options are available for every sketch tool:

Snap to grid	Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See Units options to change the minor grid spacing.
Snap to angle	Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Curve Fitter Options	If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves. • **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points. • **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated. • **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

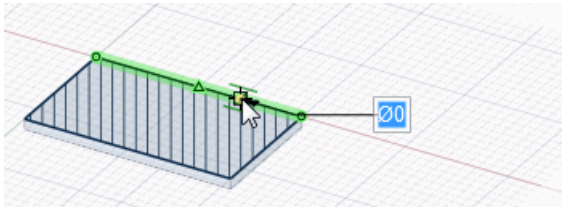
The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
Flat	Any closed sketch creates a flat wall.
Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
Inside	Maintains the length of the sheet's inside surface when bending.
Outside	Maintains the length of the sheet's outside surface when bending.

14.4.9. Circle Tool

Use the **Sheet Metal Circle** tool to draw circles in sheet metal.

Rectangles, circles, polygons, and ellipses are automatically extruded to the thickness of a sheet metal wall as you sketch. You can change the default wall thickness in the sheet metal properties.



In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

14.4.9.1. Sketching a Circle in Sheet Metal

1. Select **Circle** in the **Sketch** group or press **C**.
2. (Optional) Click the **Profile** option if you want to create walls as you sketch.
3. Click to set the circle's center.

   You can dimension the points relative to other sketch objects.
4. Click to set the circle's diameter.
5. The circle will snap to existing sketches or determined circles and arcs in the plane of the sketch.

   If you sketch two circles that are tangent to each other, and then change the diameter of one circle by editing its dimension, tangency with the other circle is maintained.

   If you drag the center of a circle that is tangent to another circle, the radius of the other circle changes to maintain tangency.

14.4.9.2. Circle Tool Options

The following options are available for every sketch tool:

Snap to grid	Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See Units options to change the minor grid spacing.
Snap to angle	Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Curve Fitter Options	If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves. • **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points. • **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated. • **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

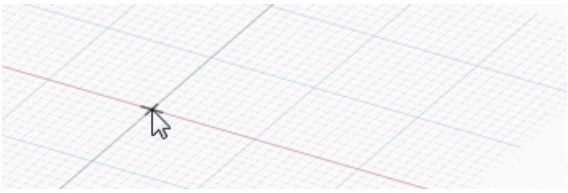
The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
Flat	Any closed sketch creates a flat wall.
Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
Inside	Maintains the length of the sheet's inside surface when bending.
Outside	Maintains the length of the sheet's outside surface when bending.

14.4.10. Polygon Tool

Use the **Sheet Metal Polygon** tool to draw a polygon with between 3 and 64 sides.

Rectangles, circles, polygons, and ellipses are automatically extruded to the thickness of a sheet metal wall as you sketch. You can change the default wall thickness in the sheet metal properties.



In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

14.4.10.1. Sketching a Polygon in Sheet Metal

1. Select **Polygon** in the **Sketch** group.
2. (Optional) Select **Use internal radius** from the Options to dimension the polygon based on the diameter of a circle inscribed within the polygon. Uncheck the option to dimension the polygon based on a circumscribed circle.
3. (Optional) Click the **Profile** option if you want to create walls as you sketch.
4. Click to set the center of the polygon.

   You can dimension the points relative to other sketch objects.
5. Drag the mouse to draw the polygon and change its orientation.

   The orientation is the polygon's angle relative to the X and Y axis.

   You can press **Tab** and type a number to change the diameter, orientation, or number of sides.
6. Click to complete the polygon.

   The sides of the polygon are all related, and act as one object. When pulled in 3D, the edges and faces of the polygonal solid will also maintain this relationship.

> **Tip:** If you trim a polygon sketch with the Trim Away tool, you can drag the original sides of the polygon with the Select tool to recreate the polygon.

14.4.10.2. Polygon Tool Options

The following additional options are available in the **Options** panel:

Use internal radius: Select this option to dimension the polygon based on the diameter of a circle inscribed within the polygon. Uncheck the option to dimension the polygon based on a circumscribed circle. In the image below, the blue circle is inscribed within the polygon and the orange circle is circumscribed around it.

The following options are available for every sketch tool:

Snap to grid	Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See Units options to change the minor grid spacing.

Snap to angle	Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Curve Fitter Options	If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves. • **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points. • **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated. • **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

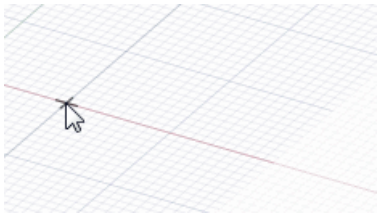
The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
Flat	Any closed sketch creates a flat wall.
Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
Inside	Maintains the length of the sheet's inside surface when bending.
Outside	Maintains the length of the sheet's outside surface when bending.

14.4.11. Spline Tool

Use the **Sheet Metal Spline** tool to draw curvy lines in sheet metal.

Lines, splines, and arcs are extruded to the thickness of a sheet metal wall when they form a closed profile. You can change the default wall thickness in the sheet metal properties.



In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

14.4.11.1. Sketching a Spline in Sheet Metal

1. Select **Spline** in the **Sketch** group.
2. (Optional) Click the **Profile** option if you want to create walls as you sketch.

3. Select the first point of the spline.
4. Select the next points of the spline.

   You can dimension spline points by entering the coordinate distance from the start point to each point, or dimension each point relative to another sketch object.
5. End the spline:
   - Double-click to set the end point of the spline.
   - Right-click and select **Finish Spline**.
   - Press **Esc**.
   - Connect the end point to the start point.
   - Click any other tool (except the **Clipboard** and **Orient** tools).

You can create a closed spline in the following ways:

- When drawing a spline, end it on the start point.
- When editing the spline, drag one end point on top of the other point.

Once you have created a closed spline, you cannot edit it into an open spline.

14.4.11.2. Spline Tool Options

The following options are available for every sketch tool:

Snap to grid	Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See Units options to change the minor grid spacing.
Snap to angle	Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.
Curve Fitter Options	If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves. • **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points. • **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated. • **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

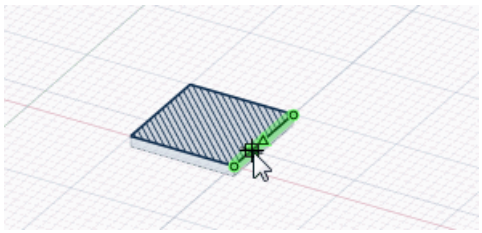
Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
Flat	Any closed sketch creates a flat wall.

Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
Inside	Maintains the length of the sheet's inside surface when bending.
Outside	Maintains the length of the sheet's outside surface when bending.

14.4.12. Three-Point Arc Tool

Use the **Sheet Metal Three-Point Arc** tool to sketch an arc when you know its chord angle and the location of its start and end points.

Lines, splines, and arcs are extruded to the thickness of a sheet metal wall when they form a closed profile. You can change the default wall thickness in the sheet metal properties.



In addition, you can choose the **Profile** option to create a wall that is perpendicular to the sketch plane as you sketch.

14.4.12.1. Sketching a Three-Point Arc in Sheet Metal

1. Select **Three-Point Arc** in the **Sketch** group.
2. (Optional) Click the **Profile** option if you want to create walls as you sketch.
3. Select the start point of the arc.
4. If the start point is on another line, arc, or spline, then the arc will initially be drawn tangent to that line, arc, or spline.

   You can dimension the point relative to other sketch objects.
5. Select the end point of the arc.
6. Set the arc's radius.

   You can dimension the chord angle or radius.

   The arc will snap to tangent objects.

14.4.12.2. Three-Point Arc Tool Options

The following options are available for every sketch tool:

Snap to grid	Select this option turn snapping on or off while sketching. The cursor will snap to the minor grid spacing increment while you sketch. The defaults are 1mm for Metric and 0.125in for Imperial units. See Units options to change the minor grid spacing.
Snap to angle	Select this option to turn angle snapping on or off while sketching. The cursor will snap to the angular snap increment while you sketch. The default is 15 degrees.

Curve Fitter Options	If the Sketch plane passes through a Mesh object, the system will fit curves through the facet points. Lines are displayed green and arcs are displayed blue. The following options apply to the system-generated curves. • **Fit curves** - Uncheck this option if you do not want the system to fit curves through the points. • **Tolerance** - Determines how many points will be found, which also determines how many curves will be created. The smaller the tolerance, the more points will be found and the curves will be generated. • **Auto-merge** - When checked On, the system will merge lines and arcs to form splines. Splines are displayed pink.

The following Sheet Metal sketch options allow you to choose between sketching a flat wall or a wall that is perpendicular to the sketch plane:

Auto	When sketching an open profile (series of lines), a perpendicular wall is created. When sketching a closed profile (square, circle, etc.), a flat wall is created.
Flat	Any closed sketch creates a flat wall.
Profile	When sketching an open profile (series of lines), a perpendicular wall is created.
Automatic	Use the starting location of the sketch to determine whether the inside or the outside of the sheet is maintained when bending.
Inside	Maintains the length of the sheet's inside surface when bending.
Outside	Maintains the length of the sheet's outside surface when bending.

14.5. Editing Sheet Metal Parts

14.5.1. Selecting Sheet Metal Objects

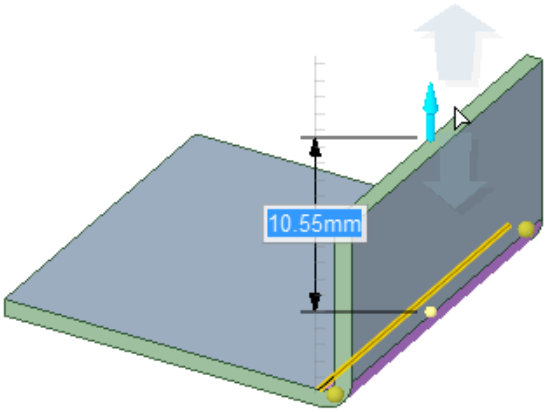
The **Select** tool on the **Sheet Metal** tab works much like the standard Select tool, with the following differences:

- The tools on the **Sheet Metal** tab recognize junctions, bends, bend reliefs, and forms first, then edges and points when you scroll the mouse wheel to select the underlying entities.
- Your selection is now automatically converted from faces to junctions, bend reliefs, or forms when you switch to the **Sheet Metal** tab and the **Pull**, **Select**, or **Move** tool is active. The selection is converted back to faces when you switch to a different tab.
- You can box-select sheet metal objects. For example, to select a corner relief, box-select all of the faces in the corner relief. If you box-select any of the faces from right to left, the entire sheet metal object is also selected, even if you have not fully enclosed the object within the selection box.
- Only the tool guides and options that are appropriate for sheet metal are displayed.

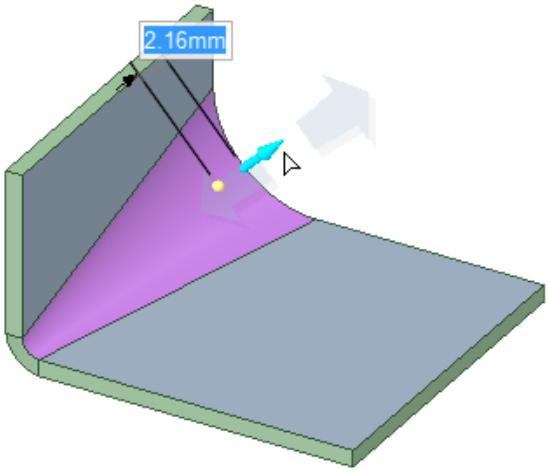
Shortcut keys work for all of the tools on both tabs. For example, you can press **S** to activate the **Select** tool from either the **Design** tab or the **Sheet Metal** tab.

14.5.2. Sheet Metal Pull Tool

Use the **Pull** tool in the **Edit** ribbon group in the **Sheet Metal** tab to change sheet metal parts.



You can create conical bends by pulling one edge of a bend junction.



14.5.2.1. Pulling a Sheet Metal Part

1. Select the **Pull** tool in the **Edit** group in the **Sheet Metal** tab.

   Shortcut keys work for all of the tools on both tabs. For example, you can press **P** to activate the Pull tool from either the Design tab or the Sheet Metal tab.

2. Change the Junction and Relief tool settings in the Modify group, if necessary.

   The settings of these tools are used when you pull. For example, if the Junction tool is set to Sharp, then a sharp edge is created when you pull. The tools' icons change to represent their current settings, so you can see them at a glance.

3. (Optional) Change the bend behavior in the Options panel:

   The bend options control where bend radii are created in relation to adjacent walls.

   - **Use Edge Location**: When you pull a sheet metal edge, the location of the bend radius depends on the edge you select and the direction you pull.



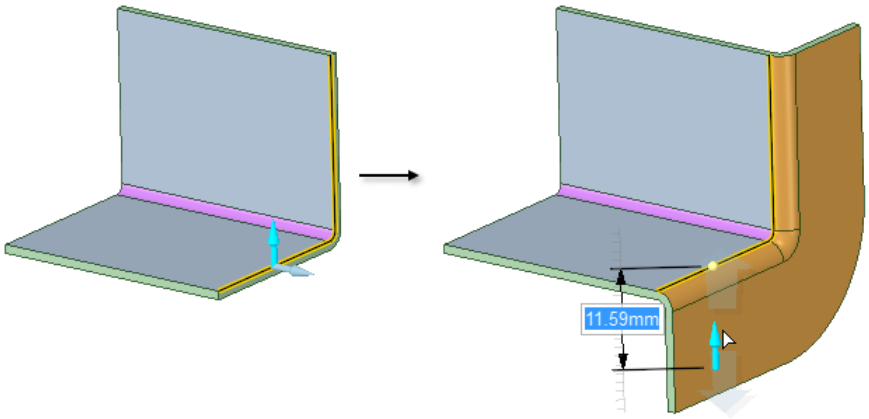
When you pull *away* from the other edge (1), the selected edge becomes an inside corner. When you pull *across* the other edge (2), the selected edge becomes an outside corner.

- **Inside**: The length of the inside surface of the wall is maintained, like 1 in the illustration above.
- **Outside**: The length of the outside surface of the wall is maintained, like 2 in the illustration above.
- **Offset**: The amount of space to allow for the bend when pulling.
- **Bend angle**: The angle between the two walls.
- **Bend radius**: The inside radius of the bend.

4. Select one of the following:
   - An edge to create a new wall with the default sheet thickness.
   - A thickness face to extend an existing wall.
   - A wall face to offset (or move) the wall in the direction of the Pull arrow.
   - A thickness edge to round or chamfer the edge. You cannot create a hem on a non-thickness edge.
5. (Optional) Click the **Pull Direction** tool guide or hold **Alt** and select a reference element to set the direction of the pull.
6. (Optional) Click the **Up To** tool guide or press **U** and select a face, edge, or point to pull up to.
7. Click a yellow Pull arrow and drag in the direction of the arrow.

14.5.2.2. Creating Panel Bends

Select a chain of edges to create a Panel Bend.



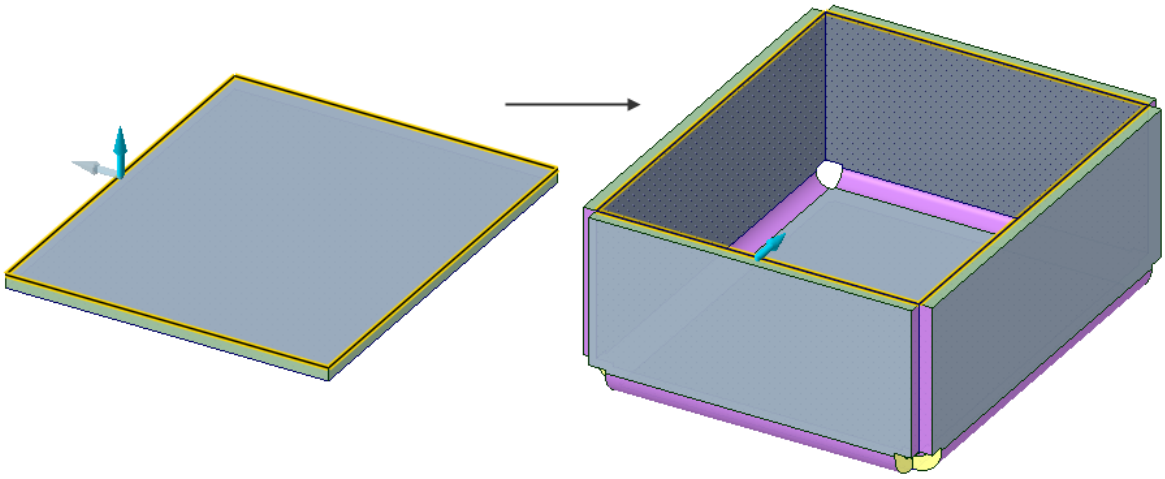
You can change the height of the Panel Bend (11.59 mm in the image above) after the bend is created. Select the Panel Bend and change the **Height** property in the Properties panel.

You can also change the **Flat Length** property to change how the Panel Bend unfolds. See the Unfold topic for more information.

14.5.2.3. Creating Walls

Junctions and bend reliefs are automatically created and are determined by the current settings of the Junction and Relief tools. Pulling an edge on an inside contour creates a wall with a gap on both sides, if a circular or square bend relief is chosen at the standard bend relief width. If a Rip relief type is selected, then the gap is equal to the rip width.

You can select an edge loop to pull multiple walls simultaneously, as shown below.

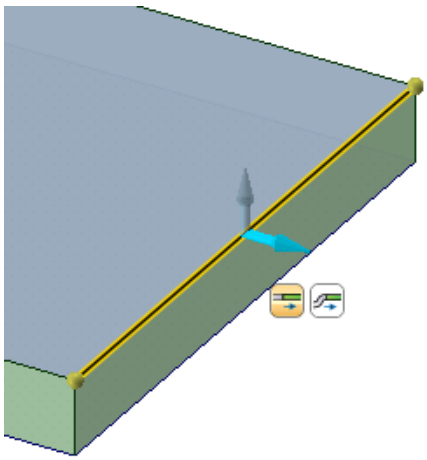


When you pull walls up on a design, the side edges of the new wall will follow the angles of its neighbors. This angle is usually 90°, but can be almost any angle. If the neighboring walls are obtuse angles, the neighbor wall must be less than 135° for the new wall to follow along the neighbors.

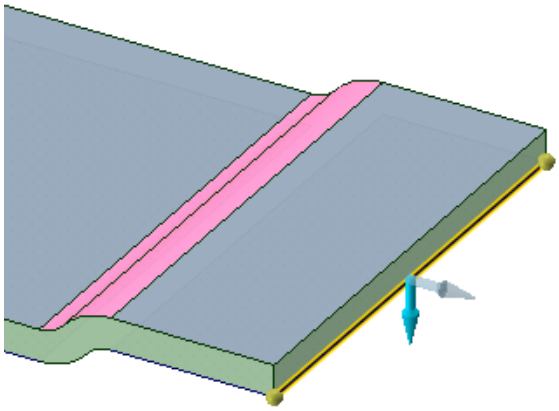
When pulling side edges with neighboring bend junctions, you can snap the resulting round radius to the radius of the junction. Hold the **Shift** key while pulling to snap to the bend radius, inner or outer, whichever is appropriate.

14.5.2.4. Creating Joggles

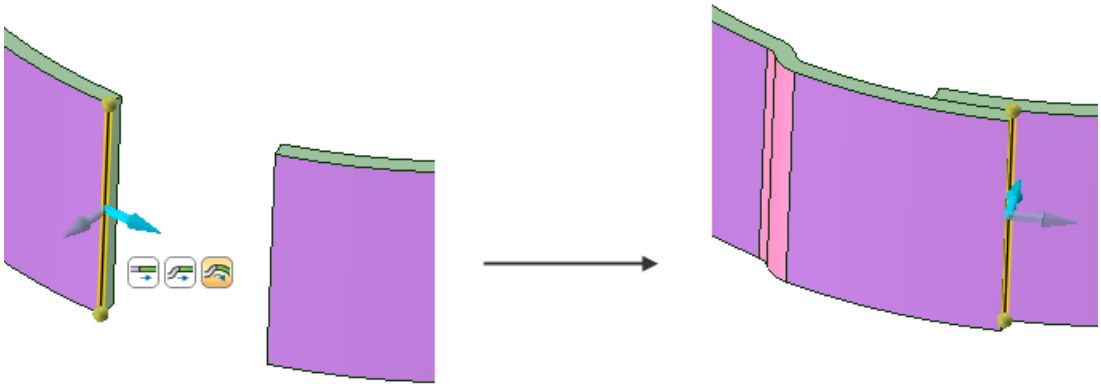
Since Sheet Metal thickness faces are thin, and you are likely zoomed out, selecting a non-thickness edge allows both edge and face Pull options. The image below shows that you can choose to pull the edge (dimmed Pull arrow) to create a new wall, or pull the face (active Pull arrow) to extend the wall. When you extend the wall, you have the option of creating a Joggle by clicking the Joggle icon in the mini-toolbar.



If you choose Joggle in the mini-toolbar  , pulling the edge will create a Joggle.

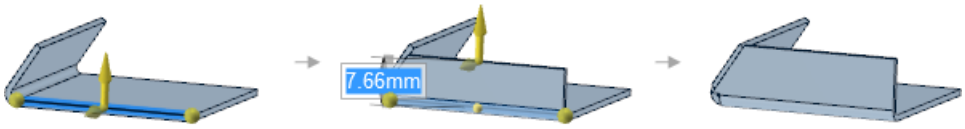


On cylindrical walls, the mini-toolbar has an option to create a Curved Joggle  .

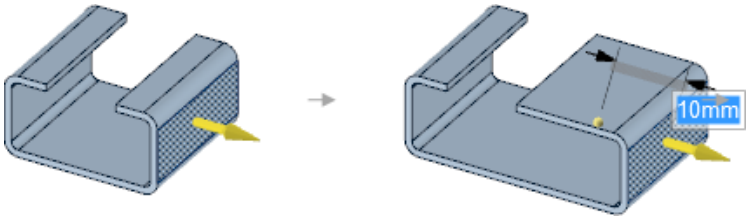


14.5.2.5. Pulling a Sheet Metal Parts Examples

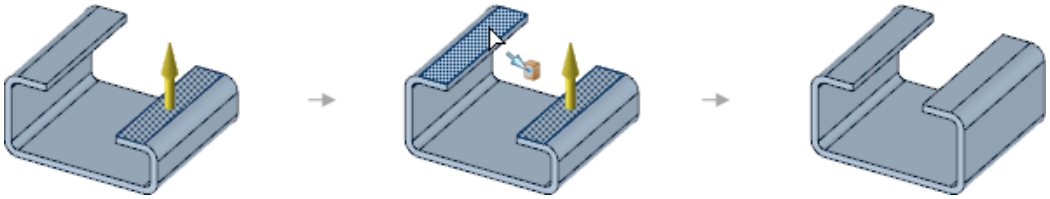
Pulling a wall with an adjacent wall creates a matching angle in the end of the new wall.



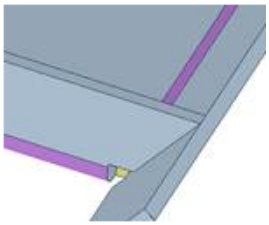
Pulling one side of a wall moves the wall and changes adjacent walls.



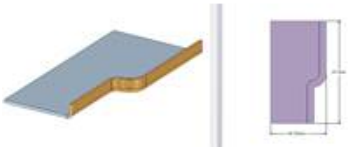
*Moving walls using the **Up To** tool guide.*



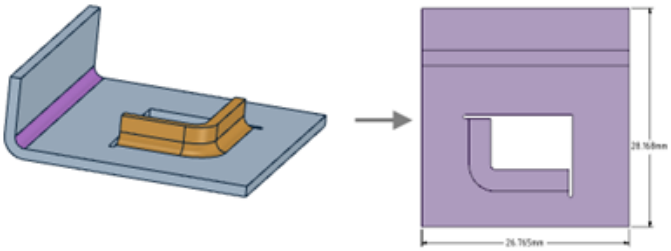
Pulling a wall up to an angled wall. The wall merges to the walls from which it extends. The extension and angled wall do not merge, a small gap is maintained.



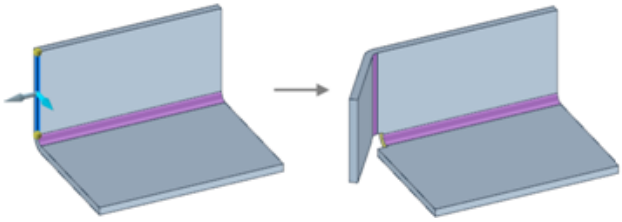
Creating swept sheet metal walls by selecting a tangent chain of edges and pulling them perpendicular to the chain (left). Swept sheet metal walls can be unfolded (right).



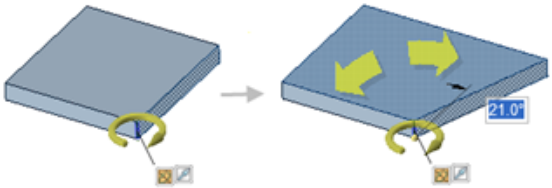
Pulling the edge of an inner loop to create a flange form. Edge reliefs are created automatically and can be unfolded. (Reliefs are not highlighted in the figure because they are not yet classified as edge reliefs).



Pulling an angled sheet metal wall. The neighboring wall is modified so that it does not maintain the junction.

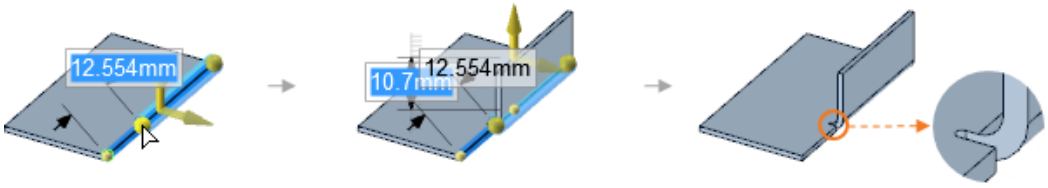


*Using the **Revolve** tool guide to make an angled end face.*



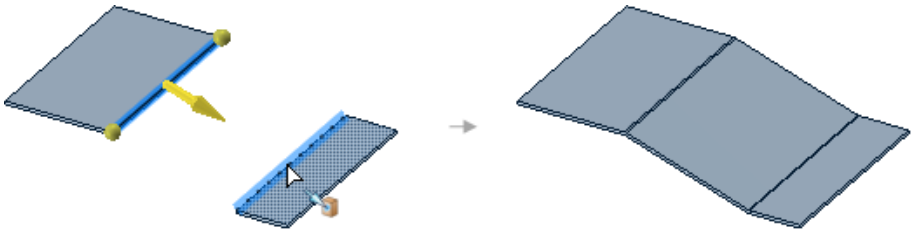
14.5.2.6. Pulling a Partial Wall

1. Click the **Pull** tool.
2. Change the Relief tool setting in the **Create** group, if necessary.
3. Select an edge on the wall.
4. Drag one or both of the yellow balls to the desired width.
   Type a number to dimension the location from the end of the edge.
5. Click a yellow Pull arrow and drag in the direction of the arrow.
6. Reliefs are created as necessary. The shape and size of the relief is determined by the Edge Relief tool and your Sheet Metal Options.



14.5.2.7. Pulling a Wall Between Two Parallel Walls

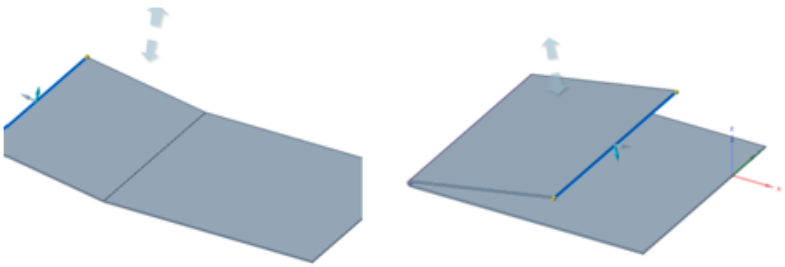
1. Select **Pull** in the **Create** group in the **Sheet Metal** tab.
2. Select an edge on the first wall.
3. Click the **Up To** tool guide or press **U**.
4. Select an edge on the second wall.
   The Pull handle previews the direction of the pull direction when you move your mouse over an edge.
   Bends are automatically added as needed.



14.5.2.8. Controlling the Bend Angle of a Pull

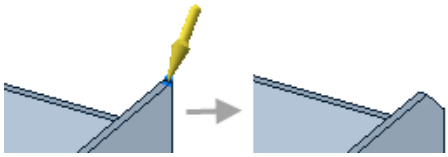
Enter a value in the **Bend angle** option before pulling.

The angle is measured from the flat state of the sheet. The following image shows a 10° (left) and 170° (right) bend



14.5.2.9. Creating a Round or Chamfer

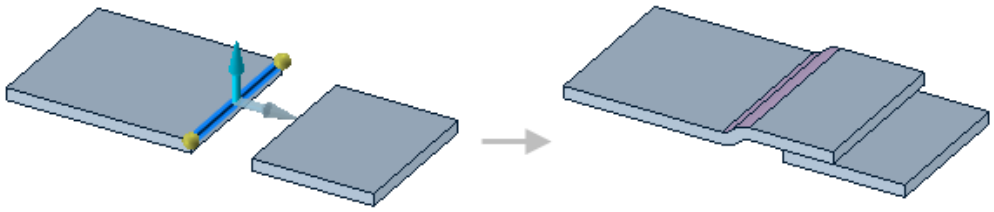
1. Click the **Pull** tool.
2. Select one or more thickness edges.
3. Click **Round** or **Chamfer** in the Pull options panel.
   You can also select **Round** or **Chamfer** from the mini-toolbar.
4. Click a yellow Pull arrow and drag in the direction of the arrow.



14.5.2.10. Creating a Jog or S Junction

1. Select the **Pull** tool in the **Create** group in the **Sheet Metal** tab.
2. Select the edge of a sheet metal wall.
3. Select a **Bend** option in the **Sheet Metal Options** panel.
4. Pull along the direction of the wall.

   The wall will jog up one wall thickness and continue in the same direction, as shown. The bend faces become joggles.



14.5.2.11. Sheet Metal Pull Tool Guides and Options

Sheet Metal Pull Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select	The **Select** tool guide is active by default. This tool guide allows you to select an object to pull.
Revolve	The **Revolve** tool guide allows you to revolve instead of pulling in a direction.
Pull Direction	The **Pull Direction** tool guide allows you to set the direction to pull.
Up To	The **Up To** tool guide pulls the object up to another object. When using the **Up To** tool guide to move a face, form, or bead to a new location on a part, such as an edge or face, select a direction with a Move tool handle and click the Up To tool to move the face, form, or bead to the new location. You can also use this tool guide with a sheet metal model in flattened mode.

Pull Tool Options

The following **Pull** tool options are available:

- **Add**: Only add material when you pull. If you pull in a subtractive direction, no change will occur. You can combine this option with other Pull options.
- **Cut**: Only remove material when you pull. If you pull in an additive direction, no change will occur. You can combine this option with other Pull options.
- **No merge**: Pulls without merging into other objects even when the object pulled intersects with an existing object.
- **Round**: Create a rounded corner (fillet) on a thickness edge while pulling.
- **Chamfer**: Create a chamfer on a thickness edge while pulling.

The following sheet metal options are available:

- **Use Edge Location**: When you pull a sheet metal edge, the location of the bend radius depends on the edge you select and the direction you pull.

  When you pull *away* from the other edge, the selected edge becomes an inside corner. When you pull *across* the other edge, the selected edge becomes an outside corner.

- **Inside**: The length of the inside surface of the wall is maintained, like 1 in the illustration above.
- **Outside**: The length of the outside surface of the wall is maintained, like 2 in the illustration above.
- **Offset**: The amount of space needed to offset the sheet, in order to allow space for pulling.
- **Dimension to Selected Edge**: Uses the selected edge to automatically calculate the dimension reference based on the subsequent pull direction.
- **Dimension to Inside Edge**: Uses the inside edge as the dimension reference regardless of the pull direction.
- **Dimension to Outside Edge**: Uses the outside edge as the dimension reference regardless of the pull direction.
- **Bend angle**: Angle between the two walls.
- **Bend radius**: Inside radius of the bend.

14.5.3. Sheet Metal Move Tool

- The tools in the **Sheet Metal** tab recognize junctions, bends, bend reliefs, and forms first, then edges and points when you scroll the mouse wheel to select the underlying entities.
- You can select a form with one click and move or rotate the form using the **Sheet Metal Move** tool. The 2D Move handle is displayed for a form, and it is lined up with the frame of the form.
- Your selection is automatically converted from faces to junctions, bend reliefs, or forms when you switch to the **Sheet Metal** tab and the **Pull**, **Select**, or **Move** tool is active. The selection is converted back to faces when you switch to a different tab.
- Only the tool guides and options that are appropriate for sheet metal are displayed.
- The side of the wall you select determines how a bend changes when you rotate a wall. See Rotating sheet metal walls. When you move a wall, any associated sheet metal objects associated with that wall, such as junctions, edges, reliefs, corner reliefs, and bends, also move with the wall.

Shortcut keys work for all of the tools on both tabs. For example, you can press **M** to activate the Move tool from either the Design tab or the Sheet Metal tab.

Faces which have beads can be moved with the **Move** tool.

14.5.3.1. Moving an Edge Relief

1. Select the **Move** tool in the **Create** group in the **Sheet Metal** tab.
2. Select the edge relief.
3. Click an axis and drag in that direction.

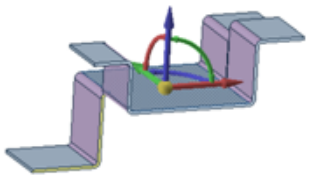
14.5.3.2. Moving a Wall Between Joggles

1. Click the **Move** tool.
2. Select a wall between joggles.
3. Click an axis and drag in that direction.
   The wall is translated between the joggles.

> **Note:** You can move a joggle along a bend. If you have trouble moving a joggle, use the sheet metal **Select** tool to select the joggle, then usethe **Move** tool in the **Edit** ribbon group of the **Design** tab to move it.

14.5.3.3. Sheet Metal Move Tool Example

The example shows translating a wall between four joggles.



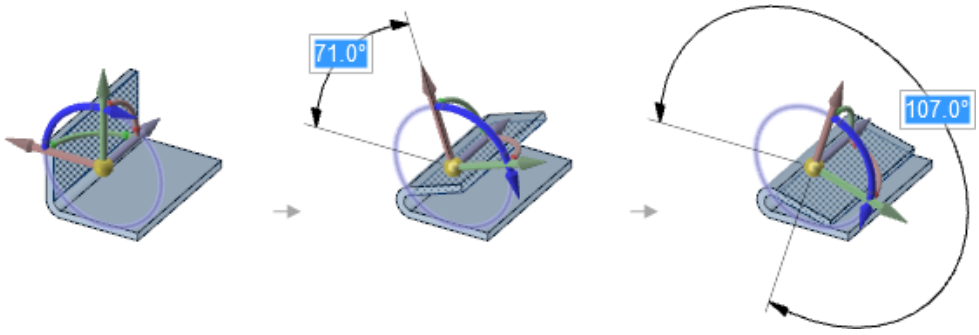
14.5.3.4. Rotating with the Move Tool

Select one face of the sheet metal wall and rotate it with the **Move** handle. If you select the inside of a face to rotate around the inside bend center, or select the outside face to rotate around its outer mold line

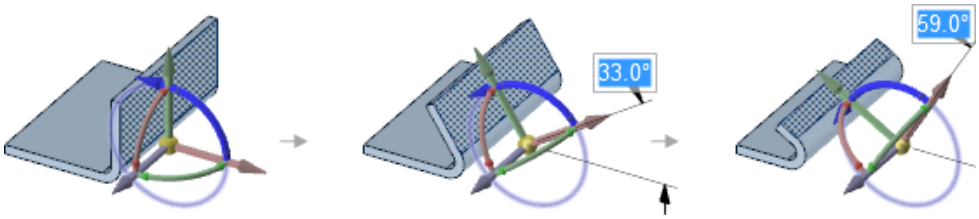
A default junction to rotate around is chosen based on which junction causes the smaller portion of the part to rotate. To rotate around the other junction, drag the Move handle anchor (the center ball) to an edge on the other junction.

14.5.3.4.1. Rotating a Sheet Metal Wall

1. Click  **Move** in the **Edit** group on the **Design** tab.
2. Select a face of the wall you want to rotate.
   The face you select determines the rotation point:
   Select the inner face to rotate around the inside of the bend:

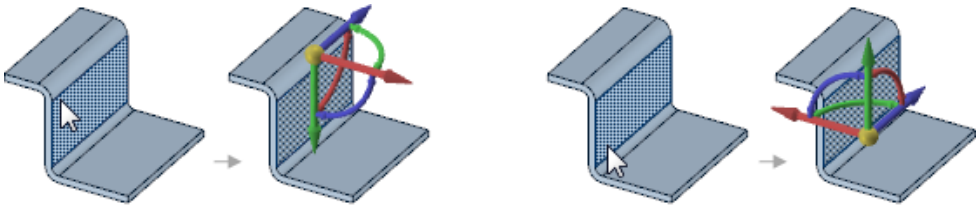


Select the outer face to rotate around the outside of the bend (the outer mold line):



When a wall is rotated around the outer mold line, the size of the walls change. This type of rotation is useful because the mold line is a measurable point for inspection.

If the face is connected to more than one junction, then the junction closest to the point where you click is selected:



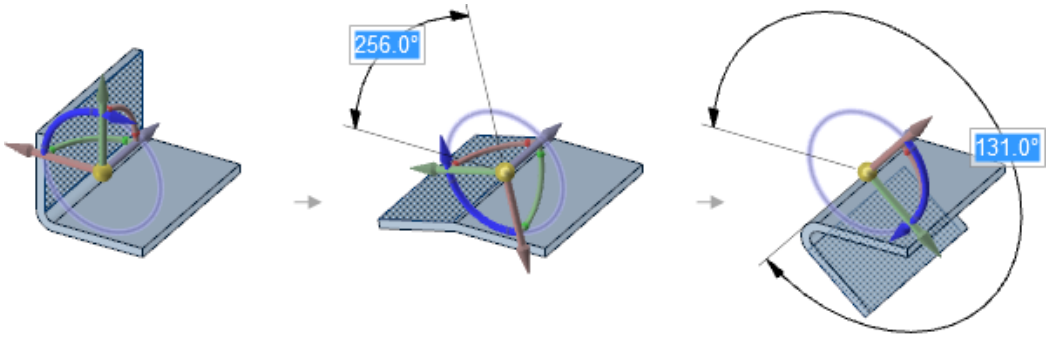
3. Select the blue rotation axis and rotate the wall.

   The Move tool will stop when the wall touches another wall.

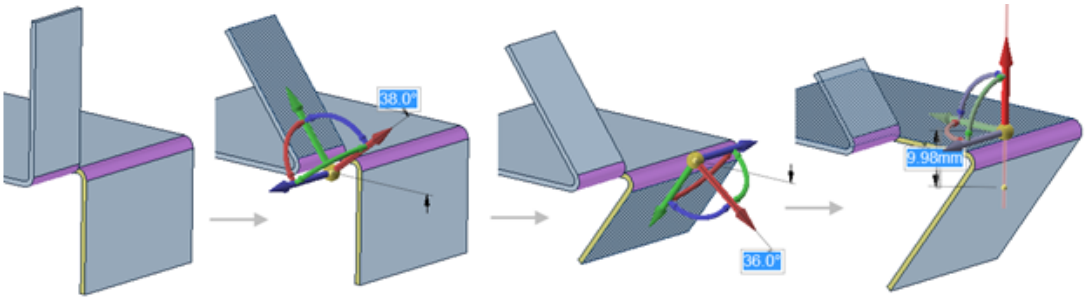
   The bend will reverse direction if you rotate the wall so its angle is greater than 180°.

14.5.3.4.2. Rotating with Sheet Metal Move Tool Examples

Rotating the inside wall so the bend is reversed.



Rotating walls that are split on an edge around a bend junction and then translating tha base face shared by the split walls.



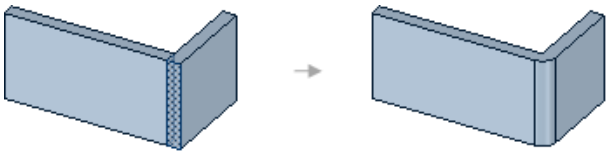
14.6. Modifying Sheet Metal Parts

The Discovery application automatically creates reliefs, junctions, and other sheet metal features as you create a sheet metal design or convert a design to sheet metal.

Use the following tools in the **Modify** ribbon group on the **Sheet Metal** tab to change the type of existing sheet metal features or to create additional features.

This section contains the following topics:

14.6.1. Junction Tool



The icon on the **Junction** option tool changes to reflect the junction type. This setting is saved until you close the Discovery application. The junction type can also be found in the **Sheet Metal** section of the **Properties** panel when you select a junction.

14.6.1.1. Changing a Junction on a Sheet Metal Design

1. Select an edge or face of a junction.

   You can Ctrl+click to select multiple junctions.

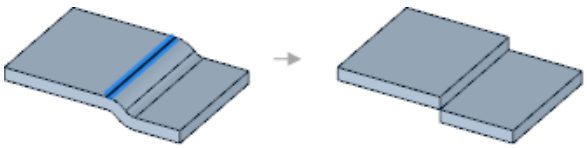
2. Select an option from the Junction tool.

   You can also select an option from the mini-toolbar or change the **Junction type** value in the Sheet Metal section in the Properties panel.

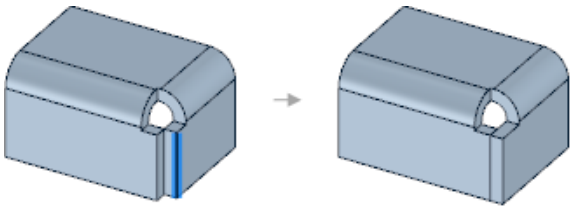
This section contains the following topics:

14.6.1.1.1. Junction Tool Examples

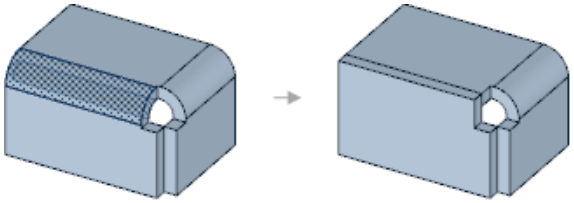
*Selecting an edge to change an S Bend junction to a **No Overlap** junction*



*Selecting the edge of a corner junction to change the junction type from **No Overlap** to **Full Overlap***



*Changing a Bend to a **Full Overlap** junction*



14.6.1.2. Removing a Junction

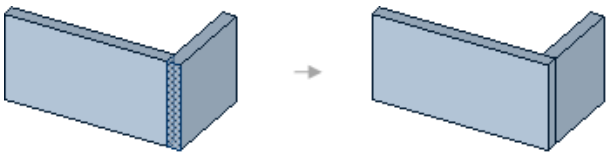
1. Select an edge or face that belongs to one or more Full overlap, Partial overlap, or No overlap bend junctions.
2. Ctrl+click to elect multiple edges.
3. Select the **None** option from the Junction option tool.

   You can also right-click the edge or face and select **None** as the **Junction type** in the Sheet Metal section of the Properties panel.

   The geometry at the junction will not change, but the junction will no longer be recognized by the Discovery application. You may need to do this when you want to make changes that can't be made to a recognized junction, such as creating a partial flange.

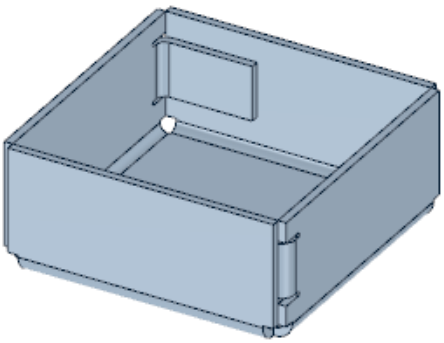
14.6.1.3. Reversing an Overlap Junction

Right-click a partial overlap or full overlap junction and select **Reverse** from the context menu.



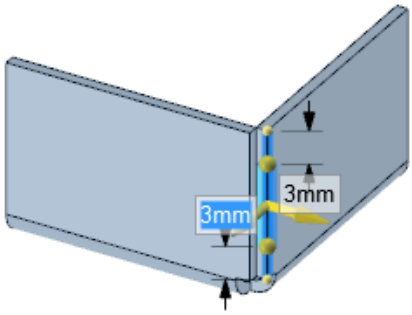
14.6.1.4. Creating a Partial Flange Junction

The part shown below has partial flange junctions on the front and back corners:

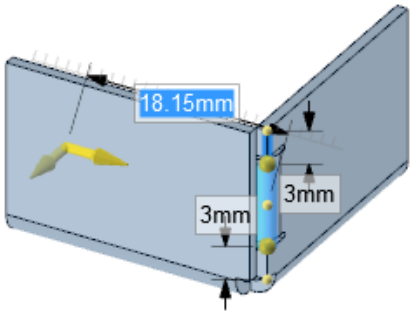


To create a partial flange:

1. Select an edge or face that belongs to a junction.
2. Change the junction type to **No Overlap**, if necessary.
3. Change the **Junction type** parameter in the Sheet Metal section of the Properties panel to **None**.
4. Select **Pull** in the **Edit** group on the **Sheet Metal** tab.
5. Select the outside edge of one wall of the junction.
6. Drag the yellow balls in to the points where you want the flange to end, as shown below.

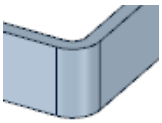


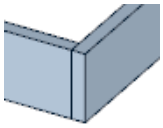
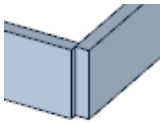
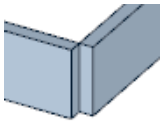
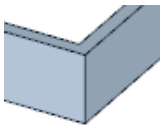
7. Pull in the direction that is parallel with the adjacent wall, as shown below.



14.6.1.5. Junction Tool Options

Select an option from the **Junction** tool in the **Create** group on the **Sheet Metal** tab:

Bend	A traditional, circular bend between two sheet metal walls.	

Full Overlap	One wall overlaps the other.	
Partial Overlap	One wall overlaps the other halfway.	
No Overlap	The walls meet but do not overlap.	
Sharp	Removes rips and bends from the geometry and returns it to a shell, where there is no inside or outside radii. (You can also rotate a 180° "hem" to create a 90° flange.)	
Flange	One side of the junction is bent and the other is straight. Use **Reverse** to change the bent and straight sides.	
None	The junction will no longer be recognized by the Discovery application.	

The junction type you select is saved as the default for the session, so any junctions created by **Pull** will be this junction type.

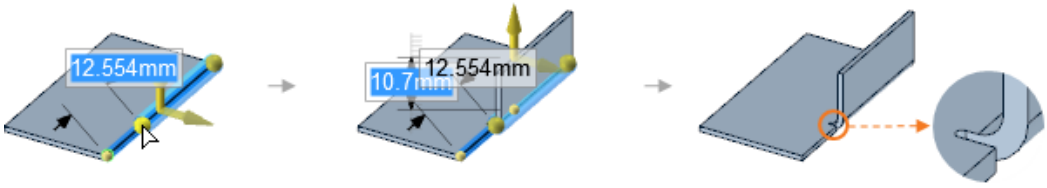
14.6.2. Edge Relief Tool

Edge reliefs are automatically created when you pull a partial wall.

The edge relief type is determined by the Edge Relief option tool in the Modify group on the Sheet Metal tab. The default is round, but you can change it to another type. The icon on the Edge Relief tool changes to reflect the relief type. This setting is saved until you close the Discovery application.

The shape of the end of a partial split is also determined by the relief type you have set for the Edge Relief tool. See Splitting a sheet metal face.

The dimensions of a relief are determined by the relief settings in Sheet Metal Options.



Dragging a yellow ball end point to pull a partial wall creates an edge relief.

A message appears in the status bar if a relief cannot be created, such as when it is too close to the side.

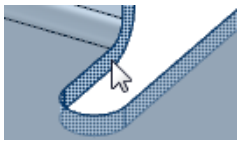
After you create a bend that has an edge relief, you can select the relief and change the type using the Edge Relief tool, or you can change the type, depth, and width in the Properties panel. You can also move an edge relief along a sheet metal wall.

Edge relief type
Round
Square
Rip
Circular
Smooth

Note: Smooth and Circular edge reliefs do not change the 3D geometry. Their effect is seen in the unfolded part.

14.6.2.1. Changing an Edge Relief Type and Dimensions

1. Select an edge or face of a relief.

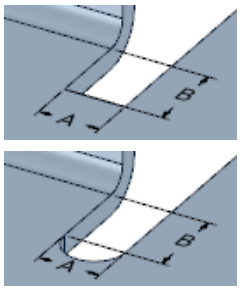
   

   You can **Ctrl+click** to select multiple reliefs.

2. Select an option from the Edge Relief tool in the Modify ribbon group on the Sheet Metal tab.

   You can also select a relief type from the mini-toolbar or modify the **Bend Relief Type** in the Sheet Metal section in the Properties panel. Depending on the relief type, you can also modify:

   - **Bend Relief Width**: The width of the relief is A in the images below.
   - **Bend Relief Depth**: The depth of the relief is B in the images below.

14.6.3. Corner Relief Tool

Corner reliefs are created automatically when you create a sheet metal design or convert a design to sheet metal.

The corner relief type is determined by the Corner Relief option tool in the Modify group on the Sheet Metal tab. The default type is square, but you can change it to another type. The icon on the Corner Relief option tool changes to reflect the corner relief type. This setting is saved until you close the Discovery application.

You can select a corner relief and view its properties in the Sheet Metal section of the Properties panel.

Note: You may need to create a corner relief on a sheet metal part, for example, when you import a shelled part that doesn't have any reliefs, or when you want to convert a watertight corner to a regular corner. When you change a sharp edge to another junction type or when you split a wall at a corner, corner reliefs are automatically added.

14.6.3.1. Adding a Corner Relief

1. Select an option from the **Corner Relief** tool in the **Modify** group on the Sheet Metal tab.
2. Select one or more points where you want a corner relief added.

14.6.3.2. Changing the Type of a Corner Relief

1. Select one or more corner reliefs by clicking any face in the corner relief.
2. Select an option from the **Corner Relief** tool in the **Create** group on the **Sheet Metal** tab:

Corner relief type	Application
Default	Any corner
Filled	Any corner
Smooth	Corners between bends or angles
Square	Any corner
Circular	Any corner
Punch	Any corner
Triangular	Any corner
Rectangular	Any corner
Oblong	Any corner
Diagonal	Corners between bends of unequal radius or angles
Laser Edge	Any corner
Laser Symmetric	Any corner
None	Any corner

3. You can also select a corner relief type from the mini-toolbar or modify the following values in the Sheet Metal section of the Properties panel:
   - **Relief Type**: Select a corner relief type.
   - **Reference**: Select Corner or Middle based on how you want the corner to be positioned.
   - **Diameter, Offset, Width, Height**: Enter values for the selected relief type.

   Note: You may wish to rotate a wall. To achieve a solid rotation and retain correct geometry, before rotating a sheet metal wall face, you can automatically convert the corner reliefs to a default corner

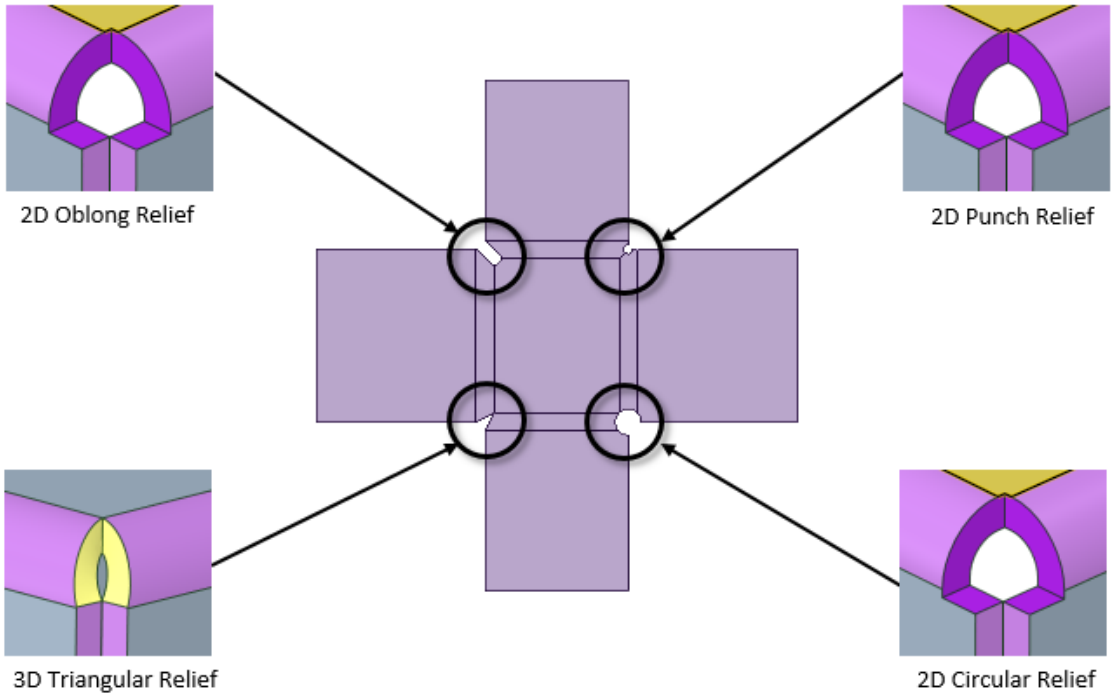
relief (which is purposely a bit over-sized). When finished with the rotation, the corner reliefs will automatically restore to their original relief type.

14.6.3.3. Two-Dimensional (2D) and Three-Dimensional (3D) Corner Reliefs

In the image below, the 3D corner relief is visible in the folded (3D) model, but, since the other three were created as 2D corners, they show default geometry in the folded model and their actual geometry in the flat (2D) model. 2D corners show as default corner reliefs in folded models, but show their actual shape in flat models.

- **2D Oblong Relief** - Created as 2D and shows default geometry in the folded and actual geometry in the flat model.
- **3D Triangular Relief** - Created as 3D and shows actual geometry in the folded and flat models.
- **2D Punch Relief** - Created as 2D and shows default geometry in the folded and actual geometry in the flat model.
- **2D Circular Relief** - Created as 2D and shows default geometry in the folded and actual geometry in the flat model.

Note that each 2D relief is shown as a default relief. In the **Unfolded** part, each relief shows the actual shape.



14.6.3.4. Two-Dimensional (2D) and Three-Dimensional (3D) Corner Relief Options

In **Discovery Settings** > **Sheet Metal**, you can set all corners to 2D.

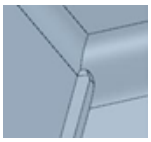
Individual corners can also be changed to 2D by updating their properties in the Options panel. You can change from 3D to 2D and 2D to 3D, as possible with respect to geometry. When selecting 2D-Only, the unfolded model will show the actual selected corners even when the 3D model can only show the default corners.

14.6.3.5. Examples of Corner Relief Styles

Changing a corner relief type to **Laser symmetric**.



Selecting the **Smooth** option from the Corner Relief tool for unequal radius bends.



Selecting the **Smooth** option for a non-90-degree corner.



The **Punch** is centered on the intersection of the Bend Lines. The **Punch** diameter can be changed in the Properties panel. The images show the diameter changed from 1 mm to 3 mm.

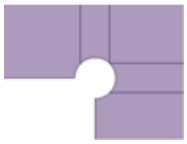




The following examples show the effect of the Reference corner relief property in an unfolded sheet metal part.

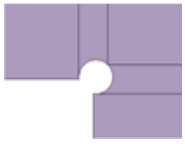
Corner relief type: **Circular**

Reference: Corner



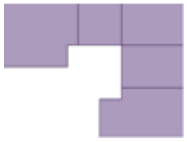
Corner relief type: **Circular**

Reference: Middle



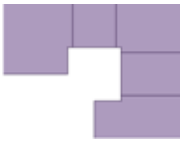
Corner relief type: **Square**

Reference: Corner

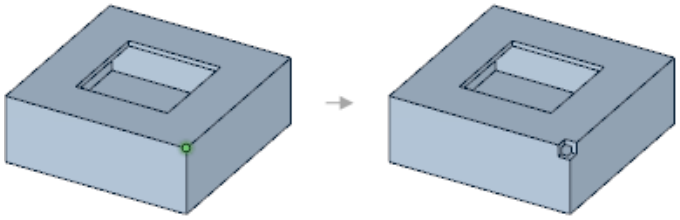


Corner relief type: **Circular**

Reference: Middle



14.6.3.6. Corner Relief Added to a Shelled Part Converted to Sheet Metal



Adding a corner relief to a shelled part that was previously converted to sheet metal. This part will need junctions and split faces before it can be unfolded.

14.6.3.7. Changing a Corner Relief Type Back to Default

1. Select one or more corner reliefs.
   You can select a Filled corner by selecting the vertex where the corner relief would be.
2. Select the **Default** option from the Corner Relief tool.
   You can also modify the **Relief Type** value in the Sheet Metal section of the Properties panel.

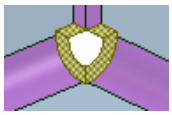
14.6.3.8. Creating a Smooth Corner Relief Between Equal Bends

1. Ctrl + click to select equal bends neighboring a corner relief.
2. Select the **Smooth** option from the Corner Relief tool.

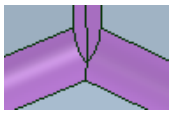
14.6.3.9. Watertight Corner on a No-Overlap Junction

Watertight corners are sheet metal corners that don't have reliefs so, to make them, you simply remove the reliefs. Parts with watertight corners can be unfolded and modified just like any other sheet metal part.

1. Select the faces of the corner.

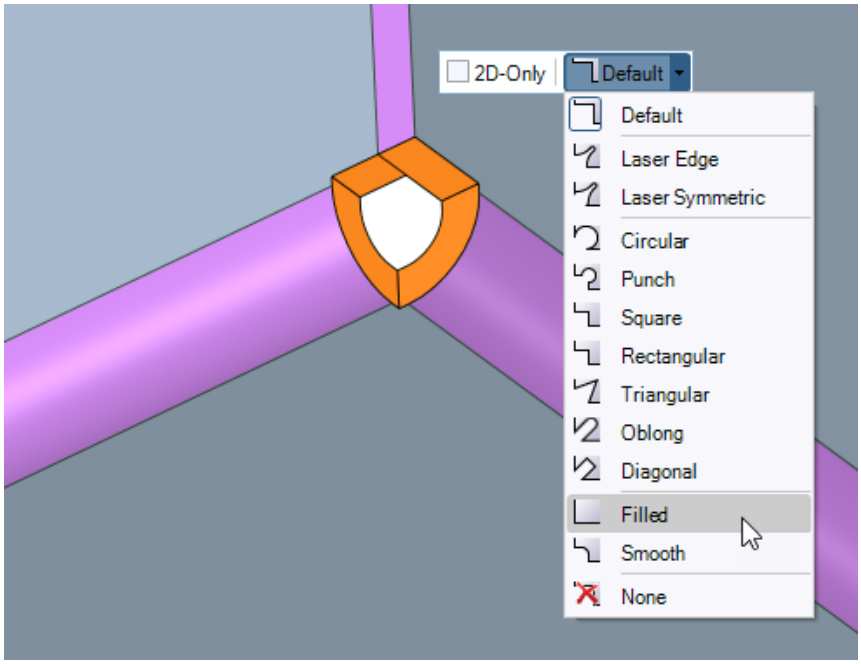


2. Select **Filled** in the mini-toolbar, or in the **Corner Relief** dropdown menu.
   The corner is now watertight.

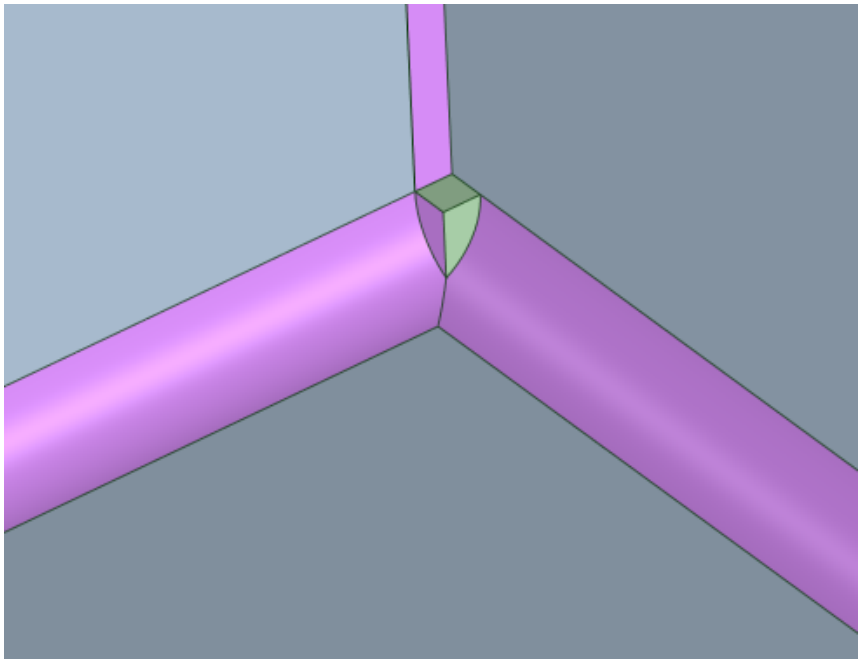


14.6.3.10. Watertight Corner on a Full Overlap Junction

1. Select the faces of the corner, and choose a **Filled** relief from the **Corner Relief** dropdown menu.

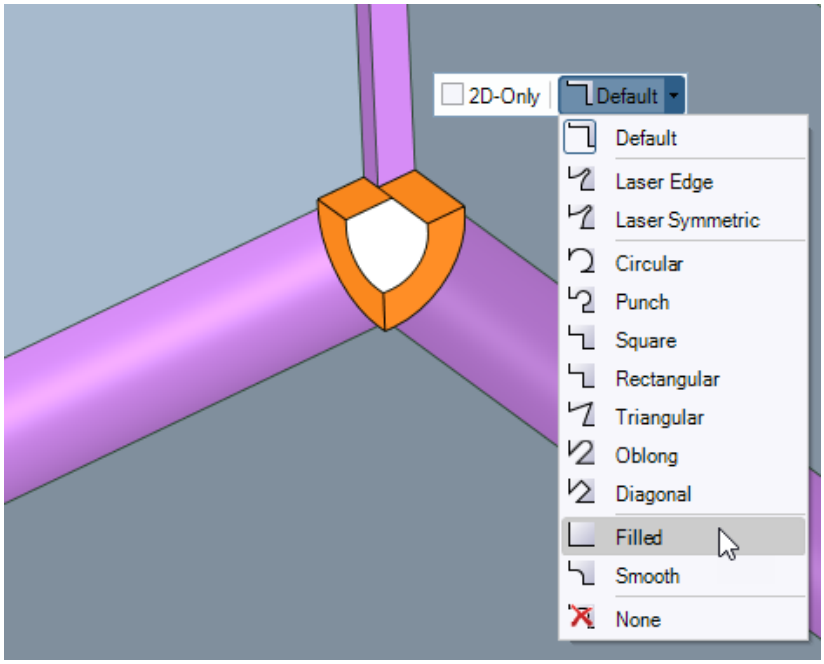


2. The watertight corner is created.

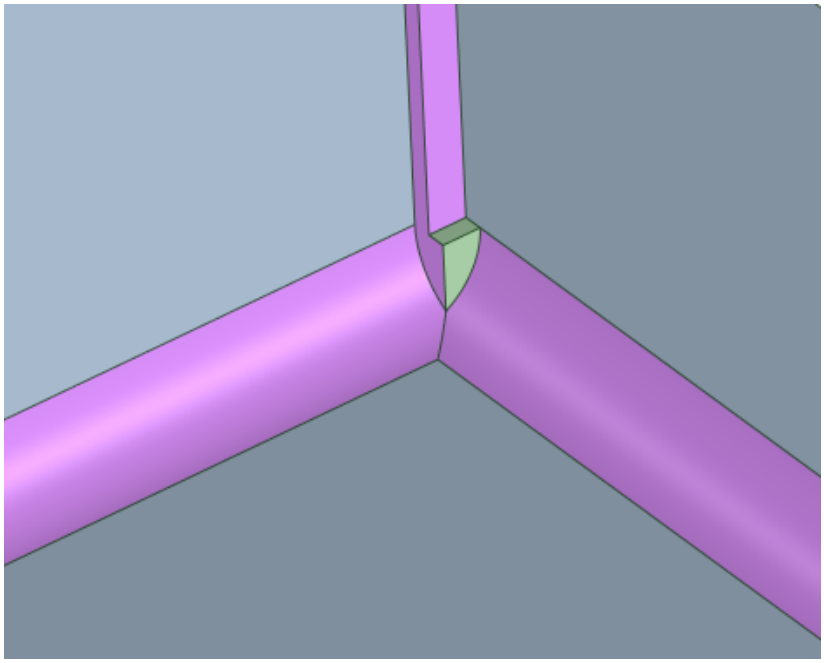


14.6.3.11. Watertight Corner on a Partial Overlap Junction

1. Select the faces of the corner, and choose a Filled relief from the Corner Relief dropdown menu in the Sheet Metal tab.

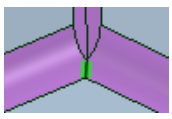


2. The watertight corner is created.

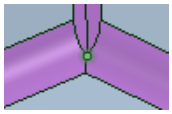


14.6.3.12. Converting a Watertight Corner to a Corner with Reliefs

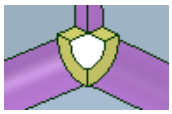
1. Select the edge or point where the bends meet, as shown below.



or



2. Select the **Corner relief** tool in the Modify group on the Sheet Metal tab and choose a Relief type.
   The corner now has a relief.



14.6.3.13. Removing a Corner Relief

1. Select one or more corner reliefs.
2. The corner relief object is removed. The geometry remains.

14.6.4. Notch Tool

Use the **Notch** option tool in the Modify ribbon group on the Sheet Metal tab to create a notch in an edge of a sheet metal wall. The notch type is also determined by this tool. The icon on the **Notch** tool changes to reflect the type.

When you convert a part that includes a notch, and then click the Identify tool and select the Notches tool guide, notches are clearly identified by the cyan color in the sheet metal part.

14.6.4.1. Creating a Notch



1. Select a vertex or thickness edge.
2. Select an option from the Notch tool in the Create ribbon group on the Sheet Metal tab to create the notch.

Notch type
Square
Rectangular
45-Degree Bevel
XY Bevel
Radius

14.6.4.2. Modifying a Notch

Select one or more notches and do any of the following:

- Select an option from the Notch tool.
- Modify the **Notch Type** in the Sheet Metal section in the Properties panel. Depending which Notch Type you select, you can modify the **Width**, **L1**, **L2**, and **Radius** values.

14.6.4.3. Removing a Notch but Leaving the Geometry

1. Select the notch.
2. The notch object is removed. Highlighting Notches no longer highlights the object.
   The geometry remains and can now be manipulated.

14.6.4.4. Removing a Notch by Filling

1. Select the notch.
2. Click the **Fill** tool in the Edit section of the Design tab.

14.6.5. Hem Tool

The following table shows the available Hem types.

Option
Simple
Open
Teardrop
Rolled
None

Hems can be created in two basic ways.

- Applied to a selected edge
- Pulling a selected edge

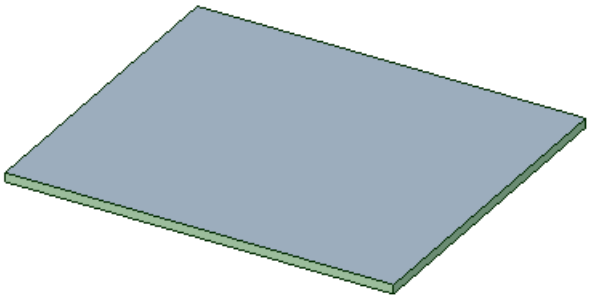
If you move a wall with a hem, the hem moves with the wall.

Click the **Hems** option in the Highlight group to highlight all the hems in your sheet metal design.

14.6.5.1. Creating a Hem Using the Hem Tool

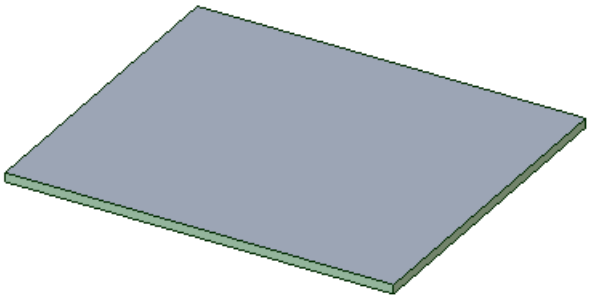
1. Select an edge
   - You can only create Hems on straight edges that have squared straight thickness edges.
   - You can not create hems on non-thickness edges, such as those created with chamfers.
2. Choose a Hem type from the Hem tool dropdown list
3. The Hem is created
   - Simple hems are offset by one wall thickness.

14.6.5.2. Creating a Hem Using the Pull Tool



1. Use the **Hem** tool dropdown menu to set the default Hem type
2. Enter the **Pull** tool and select an edge.
3. Choose the inline Pull arrow (not the arrow at 90 degrees from the face).
4. Choose **Joggle** or **Hem** from the mini-toolbar
5. Pull back over the face to create the Hem.

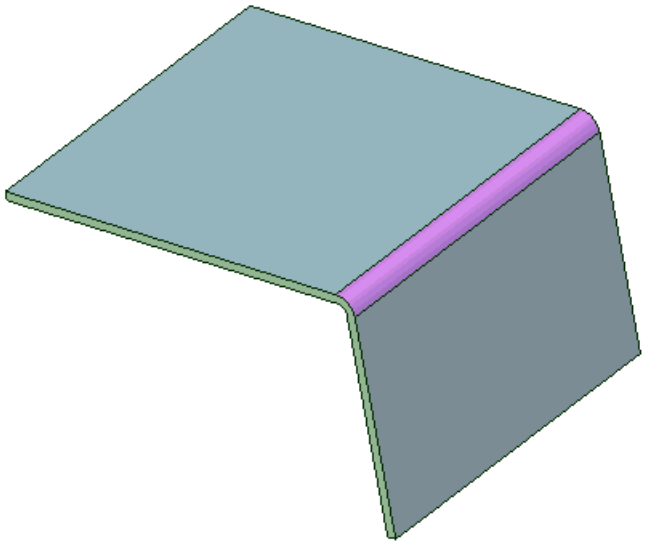
14.6.5.3. Creating a Partial Edge Hem



1. Use the **Hem** tool dropdown menu to set the default Hem type.
2. Enter the **Pull** tool and select an edge.
3. Drag the edge end handles (yellow balls) to size the length of the Hem.
4. Choose the inline Pull arrow (not the arrow at 90 degrees from the face).
5. Choose **Joggle** or **Hem** from the mini-toolbar.
6. Pull back over the face to create the Hem.

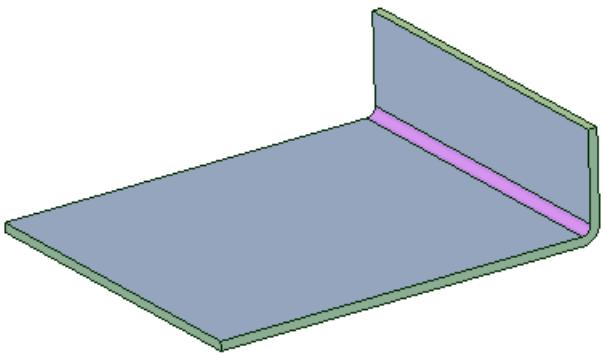
Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

14.6.5.4. Creating a Hem on Multiple Edges



1. Use the **Hem** tool dropdown menu to set the default Hem type
2. Enter the **Pull** tool and double-click an edge to select an entire edge chain
3. Choose the inline Pull arrow (not the arrow at 90 degrees from the face).
4. Choose **Joggle** or **Hem** from the mini-toolbar.
5. Pull back over the face to create the Hem.

14.6.5.5. Creating a Hem Using the Move Tool



1. Enter the **Move** tool
2. Select the *inside* wall of the bend.
   Turn the mouse wheel to select a face that is hidden by another face, if necessary.

3. Rotate the wall back on its neighboring face, as shown below.

   A 0.0001 mm gap is created automatically.

14.6.5.6. Modifying a Hem

1. Select one or more hems.
2. Select an option from the Hem tool.

   You can also select a hem type from the mini-toolbar or modify the **Hem Type** in the Sheet Metal section in the Properties panel. Depending which hem type you select, you can modify the **Length**, **Height**, **Gap**, and **Diameter** values.

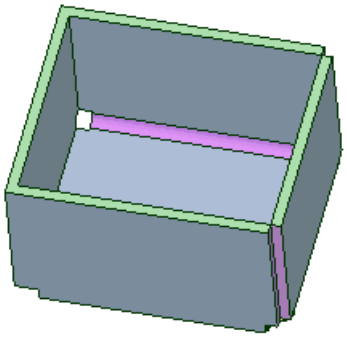
   With a hem selected, you can also modify the hem's **Inner Radius**, **Bend Steps**, **Bend Allowance** (BA) and **Bend Deduction** (BD) parameters in the Sheet Metal section of the Properties panel. When you change a hem's BA, the BD parameter also automatically updates, depending on the hem's other parameters, such as **Inner Radius**, **Height**, or **Angle**.

14.6.5.7. Removing a Hem Object but Leaving the Geometry

1. Select the hem.
2. The hem object is removed. Highlighting Hems no longer highlights the object. The geometry remains and can now be manipulated.

   To remove a hem, you can also select **Fill** from the **Edit** group in the **Design** tab, or right-click and select **Delete** from the context menu.

14.6.6. Miter Tool



When you pull two Sheet Metal edges that form a corner, a Miter will be created. Miters are Sheet Metal objects like Hems Notches, Reliefs, and Junctions.

There are two types of Miters.

- Bisector
- Perpendicular

14.6.6.1. Creating a Miter

1. Select two edges that form a corner.
2. Enter the **Pull** tool.
3. Pull the edges.
4. The miter is created as you pull.

You can also enter the **Pull** tool and then select the edges.

14.6.6.2. Modifying a Miter

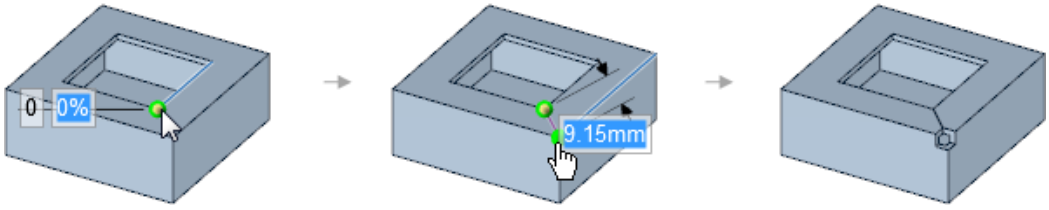
1. Select a miter.
2. Choose one of the following options from the mini-toolbar.
   a) **Bisector**: If the selected miter is Perpendicular, you can change it to Bisector.
   b) **Perpendicular**: If the selected miter is Bisector, you can change it to Perpendicular.
   c) **None**: Remove the miter
   d) **Reverse**: If the selected miter is Perpendicular, you can reverse the direction.

14.7. Creating Sheet Metal Parts

14.7.1. Split Tool

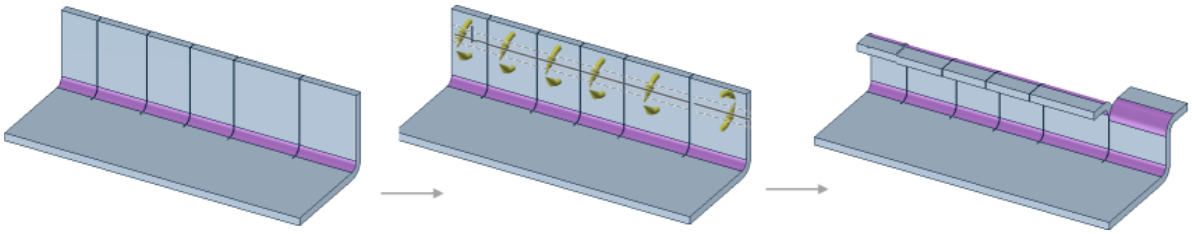
Use the **Split** tool in the **Create** group on the **Sheet Metal** tab to divide sheet metal faces and create partial rips.

You can use the **Split** tool and the **Select Two Cutter Points** tool guide and select a corner for either or both points to make a miter junction, as shown below. Corner reliefs are automatically created where needed.



You can only select a face (not a thickness edge) with this tool. You cannot split a bend face.

When creating multiple splits along a sheet metal part, regardless of whether you use one or two cutter points or bend direction, the resulting bend spans the length of the sheet metal part across all splits, and, on both sides of the split(s):

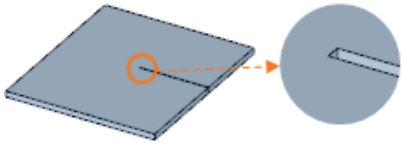


The **Split face gap option** in Sheet Metal Options controls the width of a gap when you use the **Split** tool.

14.7.1.1. Creating a Partial Split or Rip

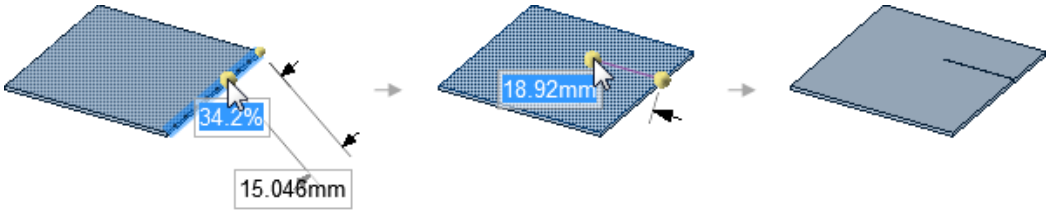
1. Select **Split** in the **Create** group on the Sheet Metal tab.
2. Select the **Select Two Cutter Points** tool guide.
3. Change the selection for the **Relief** tool, if necessary.

   The shape of the end of a partial split is determined by the type of relief selected for the **Relief** tool. The **Relief** tool is set to **Square** in the example below.



4. Select a point on an edge.
5. You can enter a dimension for the percentage of the distance along the edge or the distance from the end of the edge.

   The yellow ball turns green when the endpoint snaps to a point.
6. Click the point for the end of the split or rip.



14.7.1.2. Splitting Using an Imprinted Edge or Sketch Curve

Click **Split** and then select an imprinted edge or sketch curve.

14.7.1.3. Split Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Cutter Point	The **Select Cutter Point** tool guide is active by default. This tool guide splits a face perpendicular to an edge at the point you select.
Select Two Cutter Points	The **Select Two Cutter Points** tool guide allows you to select two points and a split is made between the points. The snap radius from Options > Snap is used.

14.7.2. Bead Tool

Use the **Bead** tool in the **Create** ribbon group on the Sheet Metal tab to create a bead on a face of a sheet metal part.

You can create a bead using a line, spline, arc, tangent curve chain, layout curve, imprinted edge, or other curve on a face. The curve must be completely on the face and must not self-intersect. The curve can extend from an edge of the face or it can extend from edge to edge to create a full bead.

A dotted line previews the bead. If the preview extends beyond the edge of the face, or if the curve is too tight, the bead will not be created. Faces which have beads can be moved with the **Move** tool.

The bead unfolds in the same way as a form unfolds.

To relocate beads on a face, select the Move tool.

14.7.2.1. Adding a Bead to Your Design

1. Click the **Bead** tool in the **Create** group on the **Sheet Metal** tab.
   The **Select Path** tool guide is enabled.
2. Select one or more curves to use to create the bead.
3. (Optional) Modify the **Bead radius** value in the Options panel.
4. (Optional) Modify the **Round radius** value in the Options panel.
5. Click the **Create Bead** tool guide.

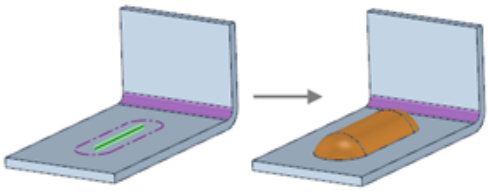
> **Tip:** To do this faster, select the curve then click the **Bead** tool.
>
> You can also create a bead using the sheet metal Bend tool.

14.7.2.2. Bead Tool Options

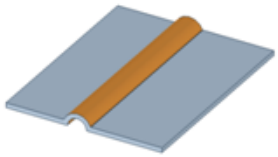
Bead radius	The radius of the bead, which is also the height of the bead. The minimum bead radius is 0.25 x T (sheet metal thickness).
Round radius	Radius of the round at the base of the bead where it intersects with the surface.

14.7.2.3. Bead Tool Examples

Create a bead from a line on a sheet metal face.



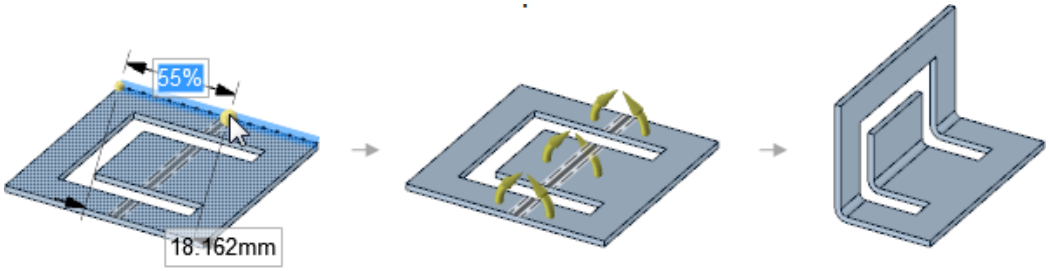
Creating a bead using a line that extends across a face.



14.7.3. Bend Tool

The Sheet Metal **Bend** tool works similar to **Split**, and is used to create bends in a sheet metal flat or unfolded part. You can bend along a sketch curve on the sheet metal face, bend perpendicular to an edge, or bend between two points.

You can also place multiple bend lines along a surface. Existing in-progress bends, that are not yet bent (flipped) remain de-selected as you place one or more new, additional bend lines along the sheet metal surface.



14.7.3.1. Bending Along a Line

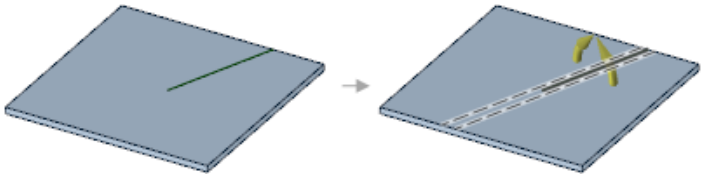
1. Click the **Bend** tool in the Create group on the Sheet Metal tab.
   The **Create Bend** option is enabled in the Bend Options panel.
2. (Optional) Modify the following values in the Bend Options panel:
   - **Bend angle**: The angle of the bend.

     When selecting on an edge to create a bend angle, modifying the Bend Angle option propagates to the pull edge handles immediately.

   - **Bend radius**: The inner radius of the bend.

     When a Bend Table is applied to the part, **Bend radius** is changed to **Tool radius** and the input becomes a dropdown list of available radius values from the table.

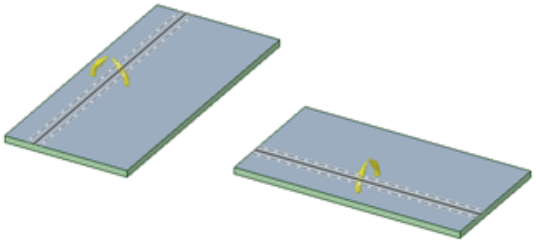
   - **Die width**: The width of the die. This changes to a dropdown list of available widths when a Bend Table is applied to the part.
   - **Bend Allowance:** Value to use for calculating the flat length of the bend.
   - **Bend Deduction:** Value to use for calculating the flat length of the bend.
3. Click a sketch curve on the sheet metal face. The bend will be made across the full length of the face, regardless of the length of the sketch curve.



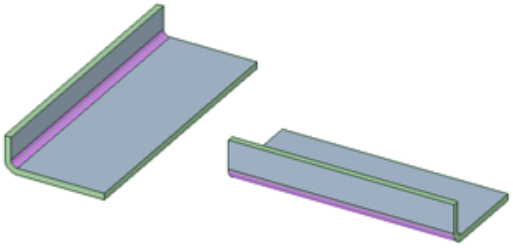
4. Click the **Complete** tool guide to create the bend.

14.7.3.2. Bending in Multiple Components at Once

1. Follow the same steps you would use for creating a single bend.
2. Select curves in multiple components. The curves can belong to one component but lie on a face of another component. For example, curve 1 belongs to component 1, but is sketched on the face of component 2.

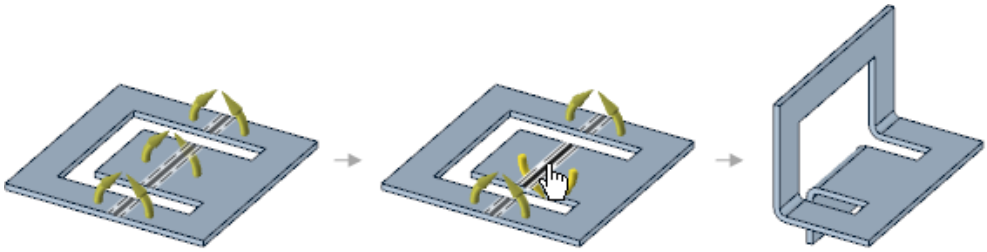


3. Click the **Complete** tool guide to create the bend.



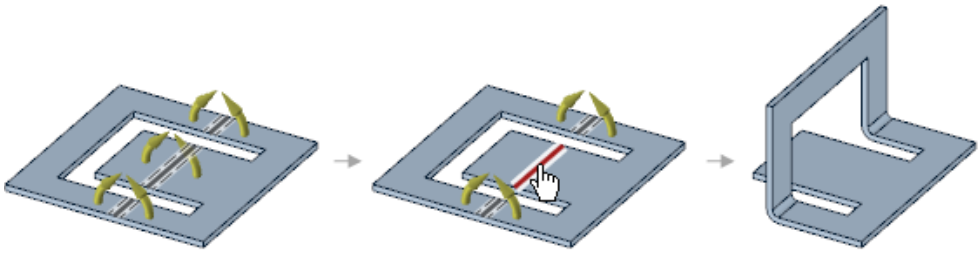
14.7.3.3. Reversing Bend Direction

Click the bend line to cycle through bending up and bending down.



14.7.3.4. Removing Bend Segments

Clicking a bend segment until the bend line turns red removes the segment from the bend.

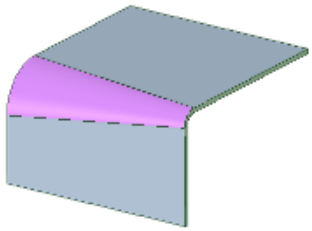


The dashed lines to either side of the bend line show the extents of the bend allowance area, based on the radius set by default or overridden by the user in the options area.

If necessary, you can click an empty location in the Design window while a tool guide is active to clear the bend line and activate the **Select Cutter Point** tool guide.

14.7.3.5. Cutting Holes in Flattened Conical Bends

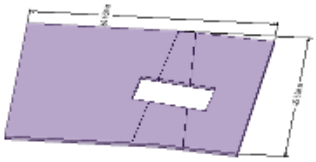
1. You have a conical bend where you need to add a hole.



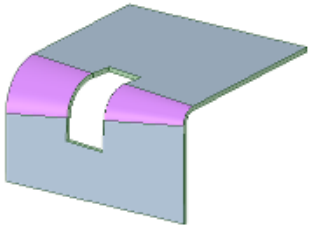
2. Flatten the part and sketch on or across the flattened conical face.



3. Pull the sketch to cut through the part.



4. The cut shows up in the folded part.

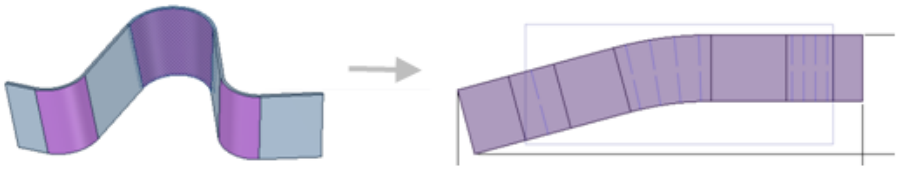


14.7.3.6. Adding Bend Steps

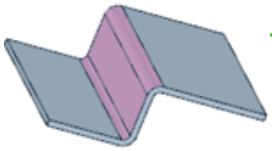
1. Select a bend.
2. Modify the **Bend Steps** value in the **Sheet Metal** section of the **Properties** panel.

   Adding bend steps creates a bumped bend, which instructs the operator and machine to create a large radius bend by bumping instead of "rolling" smoothly. The bend is hit with a sharp tool, creating closely spaced sharp bends that form a large bend.

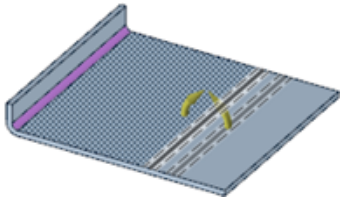
   To display the bend steps and bend dimension properties of the bend in an unfolded sheet metal part, turn on the visibility in the Bends and Bend Dimensions layers in the Layers panel or in the Style ribbon group on the Display tab.



14.7.3.7. Creating a Joggle



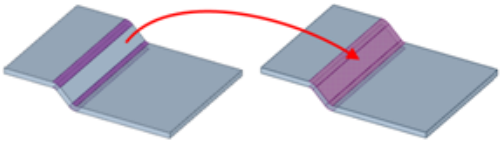
1. Click the **Bend** tool in the **Create** group on the **Sheet Metal** tab.
2. Select the **Create Joggle** option in the Bend Options panel.
3. (Optional) Modify the following values in the Bend Options panel:
   - **Bend angle**: the angle for both bends of the joggle
   - **Bend radius**: the inside radius for both bends of the joggle
   - **Joggle height**: the measurement between the top of the sheet going into the bottom bend and the top of the sheet coming out of the top bend.
   - **Bend Allowance:** Value to use for calculating the flat length of the joggle.
   - **Bend Deduction:** Value to use for calculating the flat length of the joggle.
4. Use the **Select Cutter Point** or **Select Two Cutter Points** tool guide to create the bottom bend of the joggle.
5. Click to select the second bend of the joggle, which is previewed as you mouse over the sheet metal face, as shown below.

   If necessary, you can click an empty space in the Design window to clear your selections and activate the **Select Cutter Point** tool guide.
6. Use the **Select Anchor Point** tool guide to select the face to remain fixed when the rest of the body is bent to create the joggle.
7. Click the **Complete** tool guide to create the joggle.

   You can move the walls between joggles.

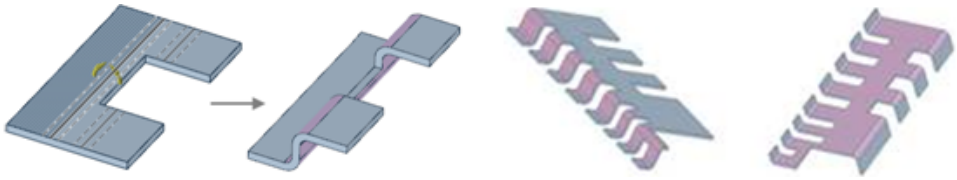
14.7.3.8. Creating a Joggle from Existing Bends



1. Select two bends with a face between them.

2. Right-click and select **Create Joggle From Bends** from the context menu.
   The face between the two bends is added to the joggle. The highlighting color changes to indicate the joggle.

Joggles created using the Bend tool.



To return a joggle to independent bends, right-click the joggle and select **Explode Joggle** from the context menu.

The independent bends remain selected. If you want to return the bends to a joggle, right-click and select **Create Joggle From Bends**.

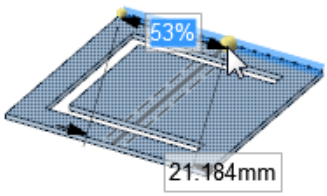
14.7.3.9. Creating a Bead Using the Bend Tool

1. Click the **Bend** tool in the **Create** group on the **Sheet Metal** tab.
2. Select the **Create Bead** option in the Bend Options panel.
3. (Optional) Modify the following values in the Bend Options panel:
   - **Bead radius**: The radius of the bead, which is also the height of the bead. The minimum bead radius is 0.25xT (sheet metal thickness).
   - **Round radius**: Radius of the round at the base of the bead that intersects with the surface.
4. Click a curve on the sheet metal face, or select a tool guide and select bend points based on the tool guide you use.
5. Click the **Create Bend** tool guide to complete the bead.

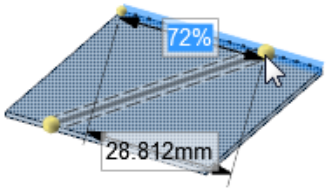
> **Note:** You can also create a bead using the Bead tool.

14.7.3.10. Using the Bend Tool Guides

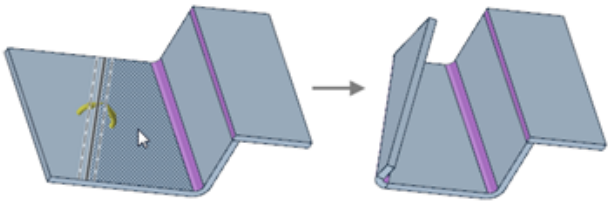
The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Cutter Point: Click a point on an edge to create a bend that is perpendicular to the edge at the point. Press **Tab** to enter a value for the percentage or distance from the bend to the endpoint along the edge.	

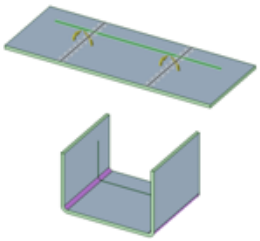
Select Two Cutter Points: Click a point on one edge and a point on another edge to create a bend between the points. You can see a preview of the line as it extends to the second edge before you click it. Press **Tab** to enter a value for the percentage or distance from the bend to the endpoint along the edge.



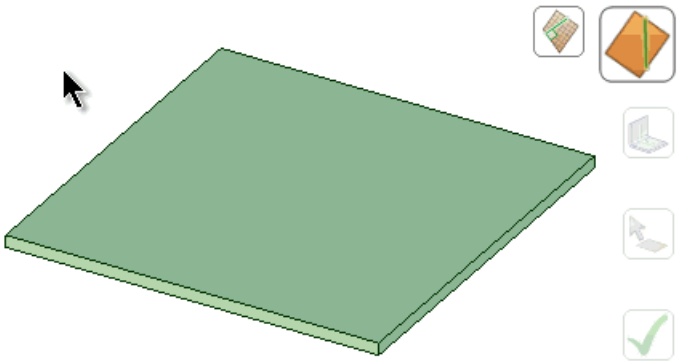
Select Anchor Point: Select the face you want to remain fixed when the body is bent:



Select Curves: Select curves to be wrapped around bends.



14.7.3.11. Using the Cross Break Option



Cross breaks are a cost effective method for strengthening a thin piece of sheet metal. Typically, this is done in an X shape, however two lines crossing at most angles will add strength.

14.7.3.11.1. Adding a Cross Break

1. Enter the **Bend** tool.
2. In the Options panel, click **Create Cross Break**.

3. **Select Cutter Point** is the default tool guide. When you mouse over an edge, you see a preview of the bend line and its location. Click anywhere along the edge to place a bend line perpendicular to the edge across the entire face.
4. The **Select Two Cutter Points** tool guide lets you select both bend line endpoints to produce different angles.
5. Click **Complete** once you have created two crossing bend lines.
6. The Discovery application shows the cross break as triangular shaded areas on the face.

14.7.3.12. Bend Tool Guides and Options

Bend Tool Guides

The following tool guides help step you through the process. Use the Tab key to cycle through them. As soon as the tool guide switches, the cursor may change to reflect the active guide.

Select Cutter Point	The **Select Cutter Point** tool guide is active by default. This tool guide creates a bend that is perpendicular to the edge at the point you select.
Select Two Cutter Points	The **Select Two Cutter Points** tool guide allows you to select a point on one edge and a point on another edge to create a bend between the points.
Select Anchor Point	The **Select Anchor Point** tool guide allows you to select the face that will be fixed when the body is bent.
Complete	Click the **Complete** tool guide to create the bend.

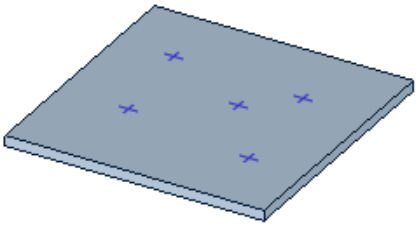
Bend Tool Options

The following options are available:

Create Bend	Use the **Create Bend** option to bend a sheet metal face along a line.
Create Joggle	Use the **Create Joggle** option to create a joggle bend in a face.
Create Bead	Use the **Create Bead** option to create a bead along a curve on a face.
Create Cross Break	Use the **Create Cross Break** option to create a cross break for strengthening a thin piece of sheet metal.

14.7.4. Marker Tool

Use the **Marker** tool to create one or more blue ‘cross-hair' placeholder markers on planar sheets or side walls (flat surfaces) of your sheet metal design.



14.7.4.1. Creating a Marker

1. Select **Marker** in the **Create** group on the **Sheet Metal** tab.
2. Select the **Place the marker** tool guide to activate the tool. The tool remains active, enabling you to place multiple markers.
3. Move your cursor over the face where you want to place a marker.
4. Click to place the marker.

14.7.4.2. Moving a Marker

When moving a marker, the only Move handles that display are those that allow you to move the marker along the design's grid. Markers move along with the walls on which they are initially placed.

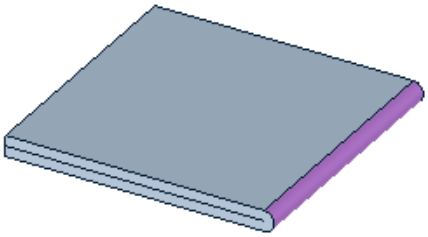
1. Click **Move** in the **Edit** group on the Sheet Metal tab.
2. Click the marker that you want to move.

   Hold **Ctrl** and click to select multiple markers. Markers you select with **Ctrl** will move as a group. You can also select one marker, then right-click and choose **Select All** to select all markers. To move a marker, you can also enter a new location value.
3. Move the marker to its new position.

> **Note:** Markers that are added to a sheet metal part remain in place if you bend the part. When you unfold a sheet metal part, markers that appeared on the part will also display on the unfolded part.

14.7.5. Double Wall Tool

Use the **Double wall** tool in the **Create** group on the Sheet Metal tab to create a folded full or partial double wall in your sheet metal design, for example, in use with creating pipes, ducts and fittings.

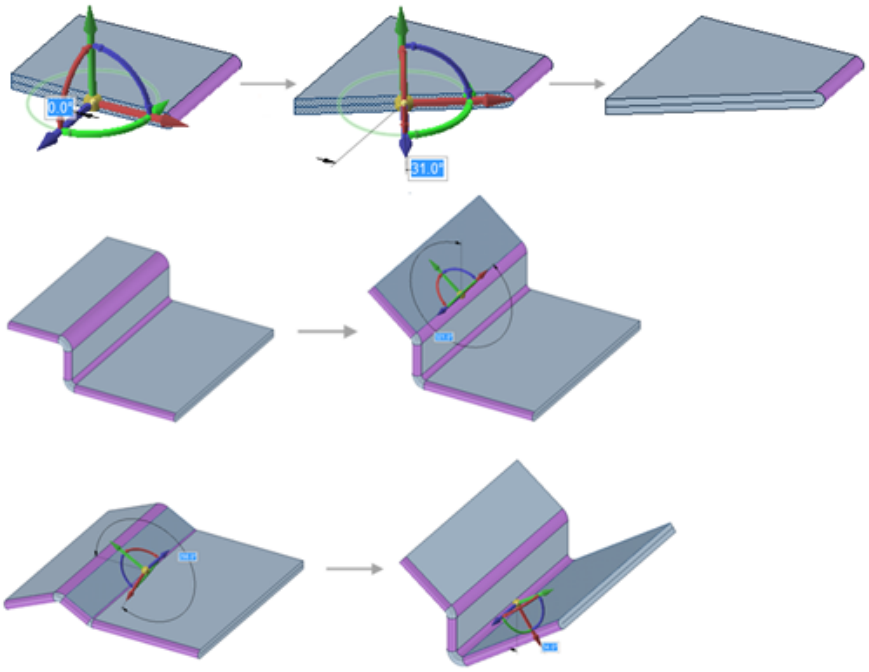


14.7.5.1. Creating a Double Wall

1. Select **Double wall** in the **Create** group on the Sheet Metal tab.
2. Select the **Select faces** tool guide to activate the tool.
3. Click the face where you want to place the double wall.
4. Click the **Select Edges** tool guide, and click the edge where you want to place the bend.
5. Click the **Complete** tool guide to create the double wall.

14.7.5.2. Double Wall Tool Examples

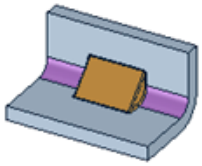
You can not only rotate double walls around a bend axis, but you can also click anywhere on the model, such as a face, and rotate the double wall to change the model's geometry. How the double wall rotates depends on which face is selected, as shown in the examples below:



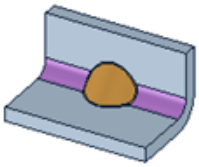
Note: When you move a double wall, the wall and any objects associated with it, such as junctions and reliefs, also move..

14.7.6. Gusset Tool

Create a flat or cylindrical gusset on a sheet metal bend.



Flat gusset



Cylindrical gusset

14.7.6.1. Creating a Gusset

1. Select **Gusset** in the **Create** group on the Sheet Metal tab.
2. From the Options panel, select a gusset type (**Flat** or **Cylindrical**), and then choose **Angle**, **Depth**, **Width**, and **Round radius** values for the gusset.
3. Select the bend where you want to place the gusset. A blue outline of the gusset appears to indicate where the gusset will be located on the bend.
4. Select a location to create the gusset.

14.7.6.2. Moving a Gusset

When moving a gusset, the only Move handle that displays is simply the handle that allows you to move the gussets along the bend axis on which the gusset is defined.

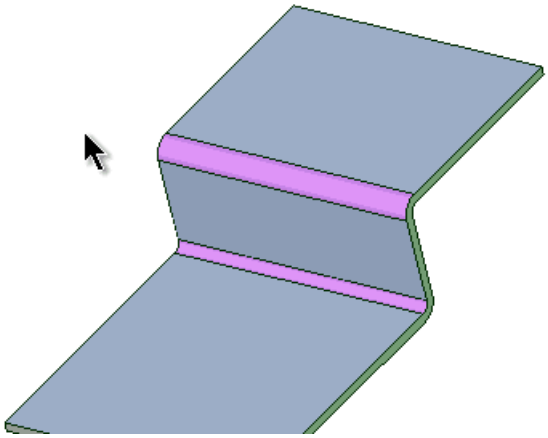
1. Click Move in the Edit group on the Sheet Metal tab.
2. Select the gusset that you want to move.
3. Move the gusset along the bend axis to relocate the gusset to a new position.

   You can also enter a new location value for the gusset.

> **Note:** When you use the Fill tool to fill a gusset, the bend geometry on which the gusset is defined remains intact.

14.7.7. Tab Tool

Create a tab on a junction or split in the sheet metal design.



14.7.7.1. Creating a Tab

1. In your design, select the junction where you want to place the tab.
2. Select the **Tab** tool in the Sheet Metal tab Create group. The **Select Edges** tool guide activates. The tab highlights in preview mode (blue) so you can see the tab's values and placement.
3. (Optional): Modify the options in the Options panel
   - **Length**
   - **Spacing**
   - **Clearance**
   - **Height**
   - Edge options:
     - **Sharp Edges**
     - **Rounds**
     - **Chamfers**
   - **Flip:** Checkbox to alternate the tabs in the opposite direction along the edge used to create the hem.
4. Click **Complete** to create the tab.

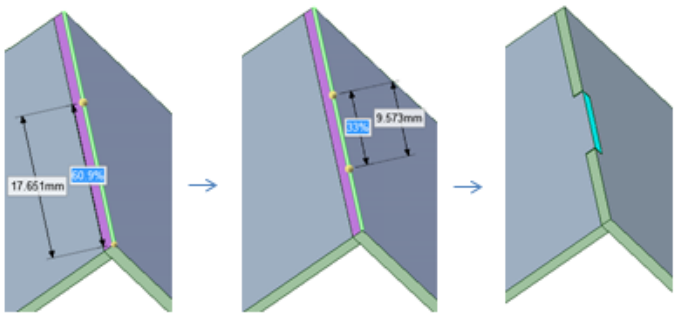
With a Hem feature selected, the RMB has two Hem menu options.

- **Flip:** Alternate the tabs in the opposite direction along the edge used to create the hem.
- **Reverse:** Switches the edge to the other body to create the hem.

14.7.7.2. Creating a Single Tab

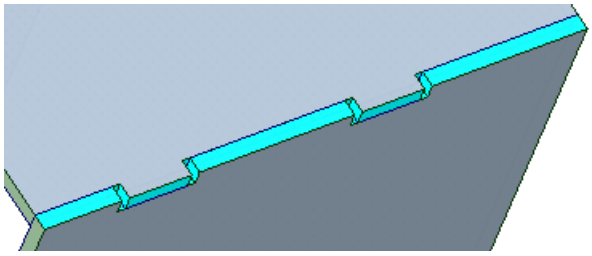
1. In your design, select the junction where you want to place the tab.
2. Click the **Tab** tool in the Sheet Metal tab **Create** group.
3. Click the **Select Points** tool guide.
4. (Optional): Modify the options in the Options panel.
   - **Clearance1**
   - **Clearance 2**
   - **Height**
   - Edge options:
     - **Sharp Edges**
     - **Rounds**
     - **Chamfers**
5. Hover over the edge you want for the tab.
6. Click once to locate the first point. Dimensions display to enter an exact location.
7. Click the edge a second time to locate the end of the tab. Dimensions display to enter an exact tab length.
8. Click **Complete** to create the tab.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

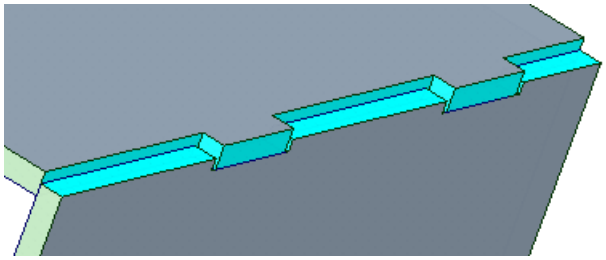


14.7.7.3. Tab Tool Options

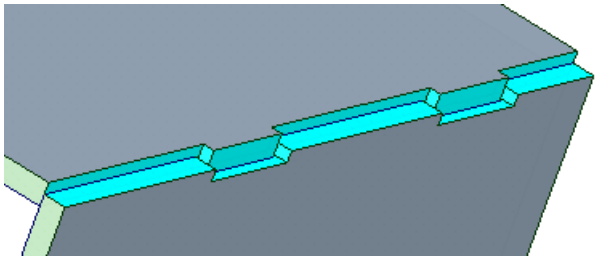
Overlap Full



Overlap 1-sided partial

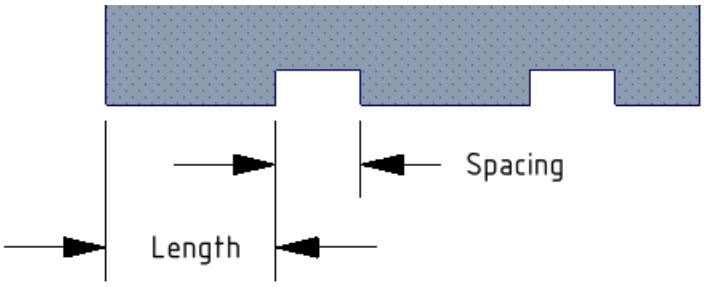


Overlap 2-sided partial



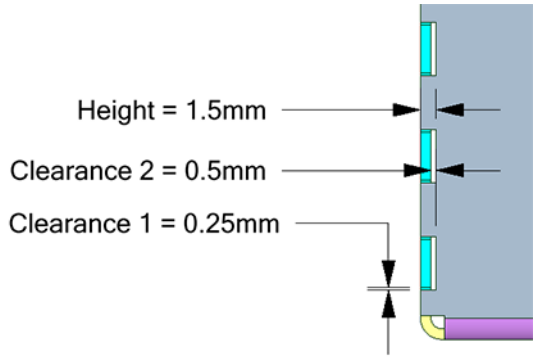
Length

Spacing



Clearance

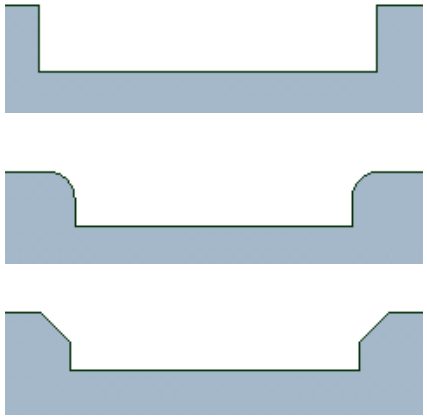
Height



Sharp Edges

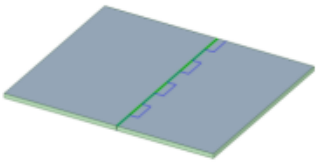
Rounds

Chamfers



14.7.7.4. Tab Tool Examples

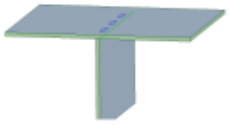
Tabs created on a sheet metal junction.



Tabs created on a split sheet metal partial edge.

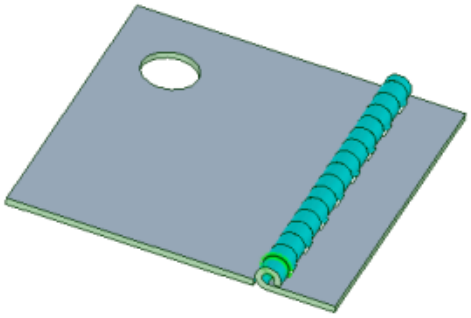


Tabs created at a T junction where one sheet is perpendicular to another.



14.7.8. Hinge Tool

Create a hinge between sheet metal walls at a junction or split location.

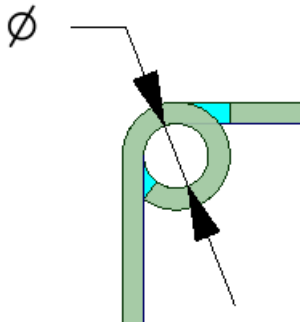


14.7.8.1. Creating a Sheet Metal Hinge

1. Create one of the three overlap junctions, or create a split across a flat wall.
2. Select the junction where you want to place the hinge.
3. Select **Hinge** in the Sheet Metal tab **Create** group.
4. The **Select Edges** tool guide activates and the hinge highlights in preview mode to show the hinge's placement and size.
5. (Optional): Modify the Hinge options in the Options panel:
   - **Pin diameter**
   - **Pitch**
   - **Knuckle length**
   - **End play**
   - **Paint clearance**
   - Hinge direction can be flipped using the **Flip** check box.
6. Click **Complete** to create the hinge.

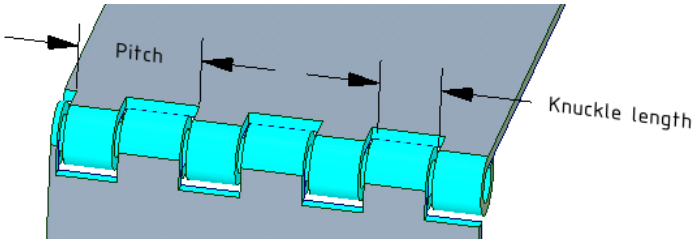
14.7.8.2. Hinge Tool Options

Pin diameter

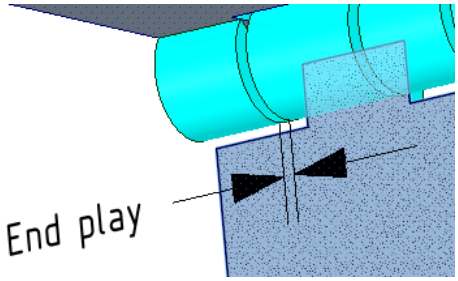


Pitch

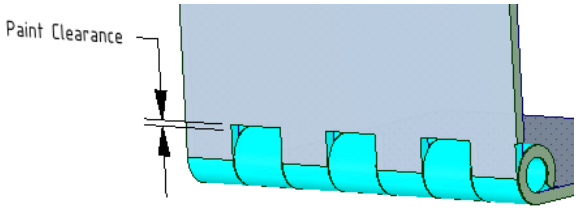
Knuckle length



End play

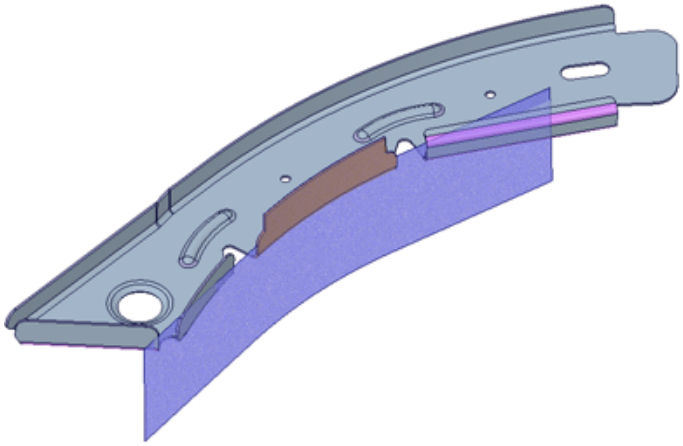


Paint clearance



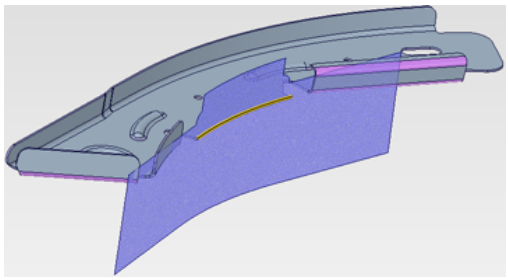
14.7.9. Flange Tool

Use the **Flange** tool in the **Create** group on the Sheet Metal tab to create a complex flange form. Flange forms get their shape from a separate, existing surface.

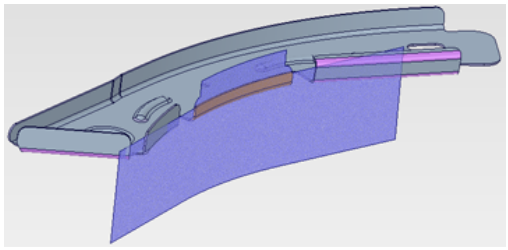


14.7.9.1. Creating a Flange

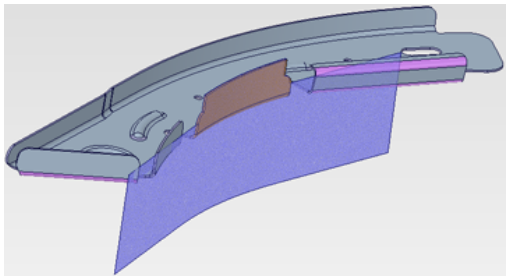
1. Create a surface that has the shape desired for the Flange.
2. Click the **Flange** tool.
3. Select edges where the Flange will be created.
4. Click the **Select Faces** tool guide.
5. Select the Face to define the Flange.



1. Select the Face to define the Flange.
2. Enter a **Flange Radius**, **Flange Height**, or **Bend Allowance** in the Options panel.
3. Click **Complete**.



1. Click **Complete**.
2. Enter a new Flange Height. Notice how it follows the shape of the selected face.



14.7.9.2. Flange Tool Options

The Flange tool has following options:

- **Flange Radius**: Use this option to change the size of the Radius at the base of the Flange.
- **Flange Height**: Use this option to change the Flange wall height.
- **Bend Allowance**: Use this option to set the Bend Allowance for the Flange.

14.7.10. Engraved or Cut-out Text in Sheet Metal

A note is text that is engraved or cut out of the sheet metal part when it is fabricated. The special sheet metal note can only be created, edited, moved, or deleted in the flattened state, but you can see it in the folded state. The text can cross bends, as shown in the example below.

You can only create lightweight notes, which means they are rendered simply and don't require as many resources as normal rendering. Their faces cannot be edited directly; you can only change the text by changing the note.

14.7.10.1. Adding Engraved or Cut-out Text to Your Design

1. Unfold the sheet metal part.

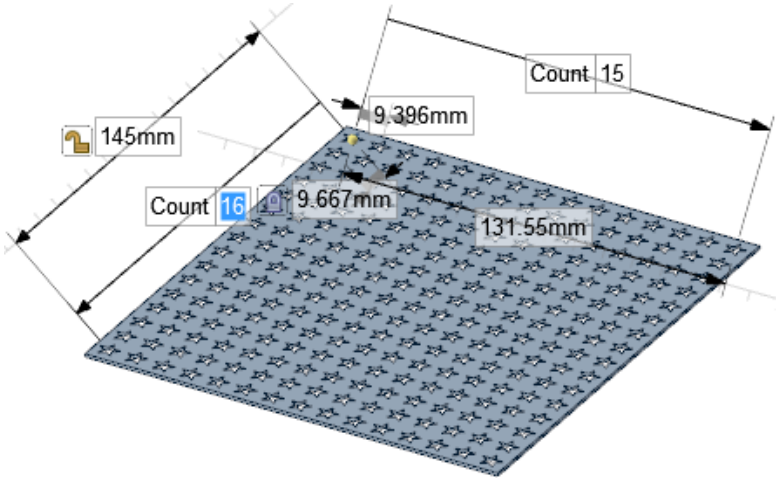
2. Click **Note** in the Annotation group on the Detail tab.
3. Click the face where you want to place the note.
4. Click again to place the note.
5. Type the note text.
6. Right-click the note text and select **Convert To Lightweight** > **Cutout** or **Convert To Lightweight** > **Engraving**.
   You can change the note text like a normal annotation note.

14.7.10.2. Converting a Sheet Metal Note to a Regular Annotation Note

Right-click the note text and select **Convert To Note**.

14.7.11. Patterns in Sheet Metal

Lightweight patterns allow you to create patterns in sheet metal that contain hundreds of thousands of members. They are displayed nearly as fast as one member because only the original member's graphics facets are calculated and then translated into graphics according to the pattern dimensions.



You can't select the graphics-only members. All changes are driven from the pattern leader and all pattern members move when you move the leader.

When you toggle from a real to a lightweight pattern, the hole that is selected as the pattern leader remains selected. This allows you to convert a lightweight pattern to a regular pattern, change the center of modification or the leader position, then convert the pattern back to lightweight for speed purposes.

Lightweight patterns are automatically converted to regular patterns when you export a sheet metal design.

> **Note:** You can only make a lightweight pattern using a pattern of holes.

14.7.11.1. Making a Lightweight Pattern

1. Follow the instructions in Creating a pattern to create a pattern of holes.
   You can make any type of pattern, including linear, circular, radial, etc.

2. Right-click the pattern leader and select **Lightweight Pattern**.

14.7.11.2. Converting a Lightweight Pattern to a Normal Pattern

Right-click the pattern leader and select **Toggle Lightweight Pattern**.

14.8. Unfolding Sheet Metal Components

You can unfold a sheet metal design and changes appear simultaneously in the unfolded component and in the original design.

Unfolded parts are not exported with the solid when you export a sheet metal part; however, you can explicitly export the unfolded part.

You can unfold the following types of walls, but you must right-click a planar face and not a cylinder or a cone:

- Planar walls
- Cylindrical walls, including large diameter cylinders (roll bends)
- Most conical walls, such as a part with variable radius round between arcs that was shelled and converted to a sheet metal part. Parts that are made with blends between radii also unfold reliably.
- Z bends and S bends, even when they neighbor a cylindrical or conical wall.
- Walls that includes holes that touch a bend or are included in an object that includes a bend.
- Double-walled parts that include a complex unbending edge.

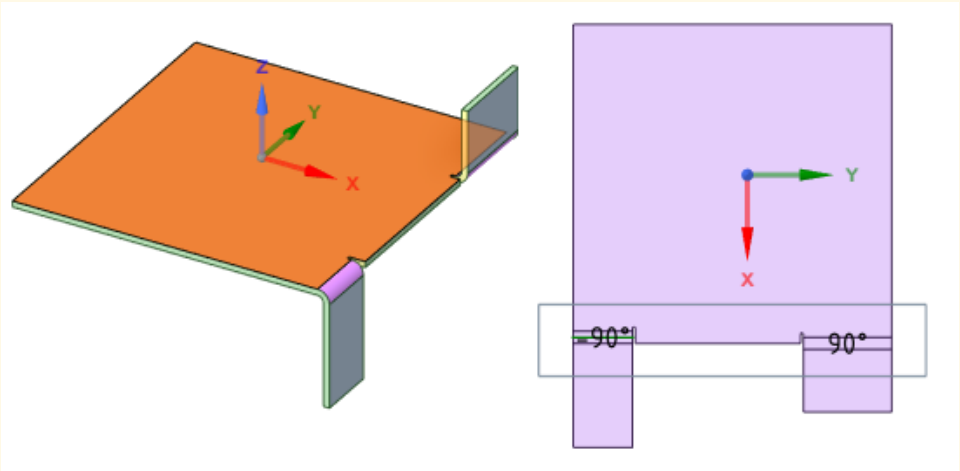
14.8.1. Unfolding a Component

1. Select a face of a sheet metal component with at least one bend junction.

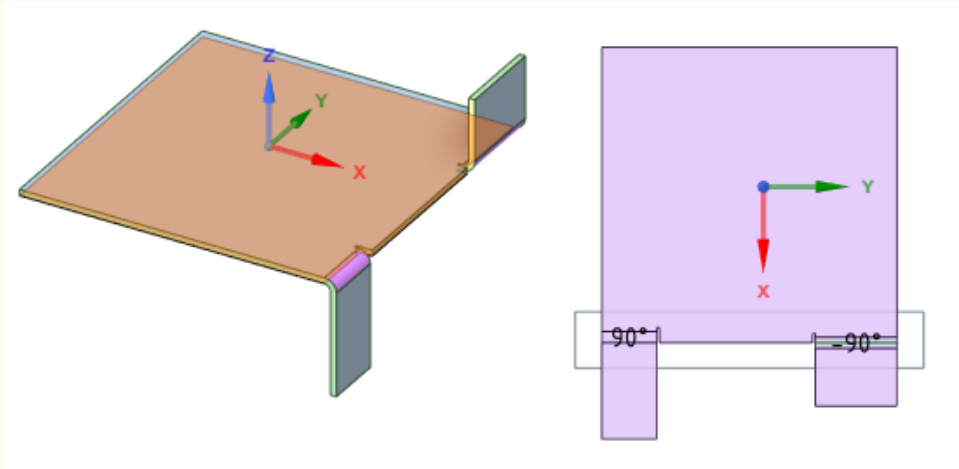
   When you select a single face of a sheet metal part, the **Unfold** tool is enabled. You can also right-click and select **Unfold Part** from the context menu.

Note: The selected face sets the orientation of the unfolded design. Bend angle signs are related to the referenced face selected for unfolding.

In this example, when the top face is selected, the bend angle is positive for the upward bends.



If the bottom face is selected, the bend angle is negative for the upward bends.

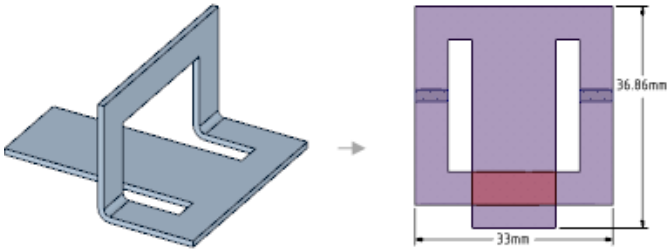


As the unfolded parts can be viewed from the top or bottom direction in drawings or unfolded views, the bend annotations in the unfolded view may or may not be in the same direction as the drawing view direction. Verify the bend angle with the referenced face to ensure proper bend orientations.

2. Click **Unfold** in the **Flat** group on the **Sheet Metal** tab.

Note: Conical bends that are not exactly conical will flatten *approximately*.

An unfolded version of the component is displayed in a new Design window as a top view with its overall dimensions, as shown in the figure below. It also appears in the Structure tree as an unfolded part
.
The visibility of the unfolded part in the original design is initially set to Off in the Structure tree. The unfolded part is saved as part of your design.

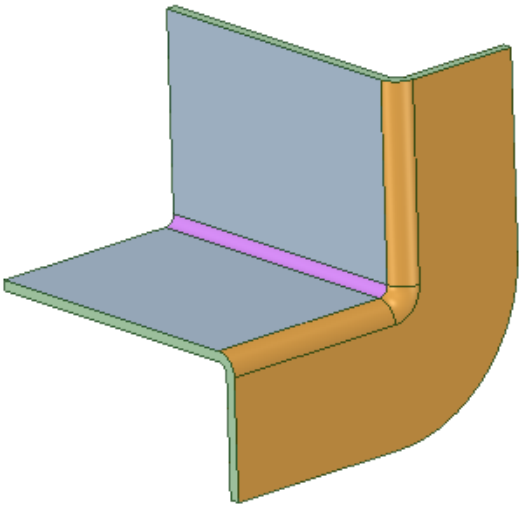


- Overall dimensions on an unfolded part are measured based on the orientation of the sketch grid. The inside angle of a bend is used to calculate dimensions. The two flat extent dimensions are automatically displayed.
- Bends are listed as objects in the Structure tree for the unfolded part. Bend lines and bend dimensions are placed on a Bends layer with the visibility off. Turn the visibility on in the Layers panel to view the bend lines.
- If a chamfer is recognized as a form, it unfolds based on the **Flatten form as** value in the Sheet Metal section of the Properties panel for the design.
- If an unfolded component has conflicting geometry, the conflicting geometry is made a separate surface in the Structure tree and highlighted in red, as shown in the figure below. The edge that prevents the unfold is also highlighted.

- If you save an unfolded sheet metal part within a component, opening that component in an assembly displays a check box and icon for the unfolded part in the Structure tree. You can toggle the visibility of the unfolded sheet metal part in the unfolded part design window.

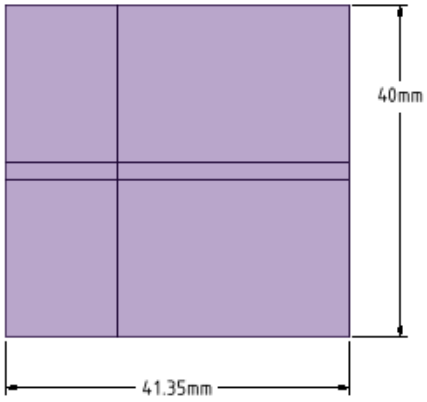
14.8.2. Unfolding a Panel Bend

Panel Bends have a **Flat Length** property that you can change to set the length of the unfolded bend.

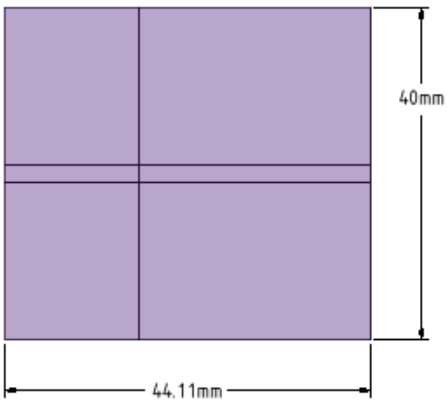


The Panel Bend above has a default Flat Length of 13.59 mm, which results in an unfolded dimension of 41.35 mm. Setting the Flat Length property to 16.35 mm results in and unfolded length of 44.11 mm as shown below.

Flat Length = 13.59 mm

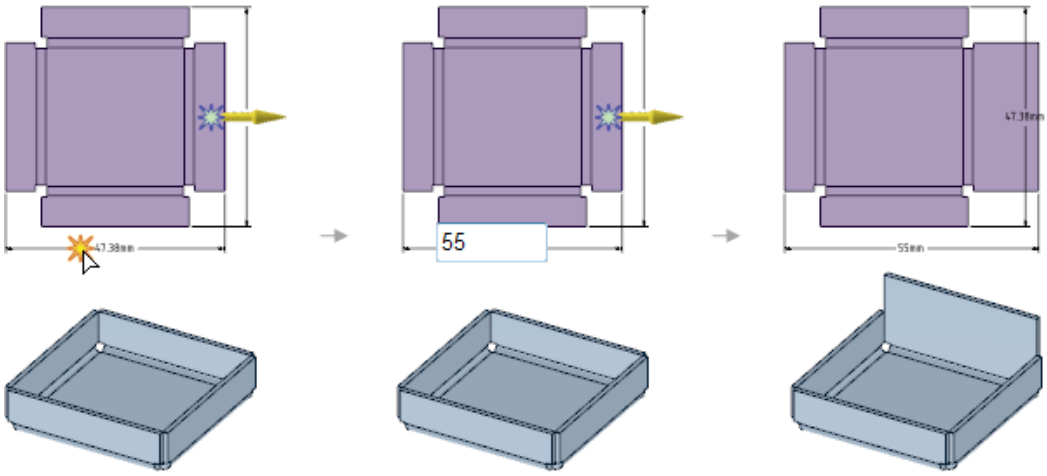


Flat Length = 16.35 mm

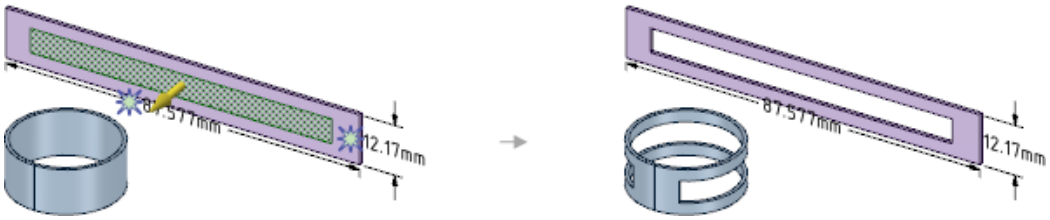


14.8.3. Unfolding Sheet Metal Component Examples

Using the Pull tool and annotation dimensions to change the height of a wall on the unfolded part (on top); the changes are reflected in the folded part (on the bottom).



Using the Pull tool to put a hole in an unfolded cylinder also creates the hole in the folded part.

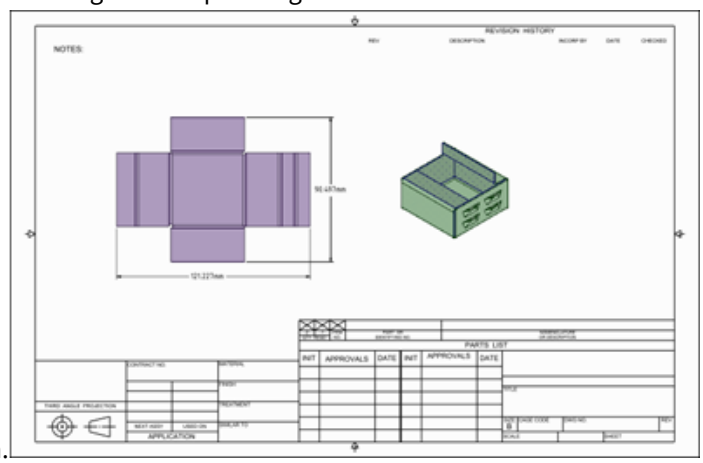


14.8.4. Creating a View of an Unfolded Part in a Drawing Sheet

To create a view of an unfolded part in a drawing sheet:

1. Unfold the sheet metal part in the drawing sheet.
   Move the mouse to preview the unfolded part.

2. Click the drawing sheet to place a general view at that location.



14.8.5. Adding Bend Dimensions to an Unfolded View

To add bend dimensions to an unfolded view:

1. Select the view.
2. Select **Show Bend Dimensions** from the context menu.

   RMB > **Show Bend Dimensions**
3. Make sure the **Bend Dimensions** layer is turned on.

   With this option enabled, any changes made to the model will automatically update the drawing sheet bend dimensions.

14.8.6. Exporting an Unfolded Part as .dxf

1. Place the desired dimensions and notes on the unfolded view using the tools in the Detail tab.
2. Set the Graphics Style in the Display tab to **Wireframe**.

   This step prepares the model for dxf export and shows you what the resulting image will look like.
3. Select **Save As** from the Discovery application menu and specify AutoCAD files (*.dxf) as the export type.

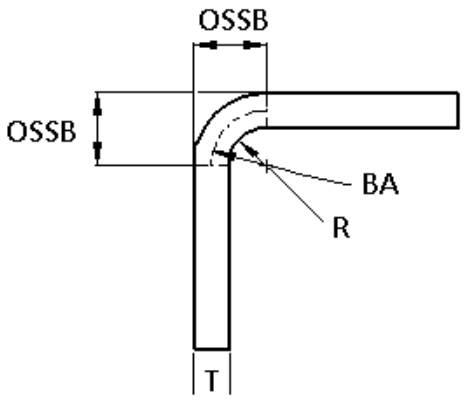
   The unfolded model is saved as a 2D dxf file with the included annotations and can be used in manufacturing.

14.8.7. Calculating Unfolded Lengths

The Discovery application can use sheet metal bend deduction to calculate unfolded lengths. You can set the default Sheet Metal Options in the Options window.

You can change the default bend properties for a specific component or bend by selecting the component or bend and modifying the values in the Properties panel:

- The **Bend Allowance** property contains the length of the arc through the bend at the neutral axis of the sheet metal wall being bent. Adding the Bend Allowance and the lengths of the two sides give the unfolded length.
- The **Bend Deduction** value for the selected bend represents 2 times the Outside Set Back (OSSB) minus the bend allowance.

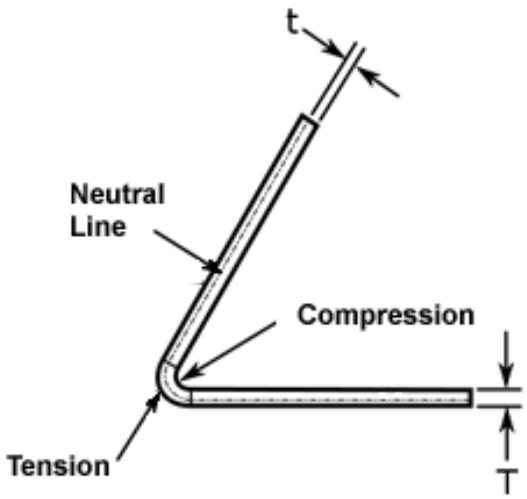


Bend Allowance (BA) and Bend Deduction (BD) are linked. Modifying one value causes the other to be automatically recalculated as follows.

BD = 2 x OSSB - BA

14.8.8. Calculating Bend Allowances

Bend allowance calculations use what is known as a **K-factor**. This is the ratio of the location of the neutral line (**t** in the image below) to the material thickness (**T**). When metal is bent, the material at the inside radius is put in a compressive state while the material at the outside radius is put in tension. The neutral line is the point of zero stress where the material transitions from compression to tension.

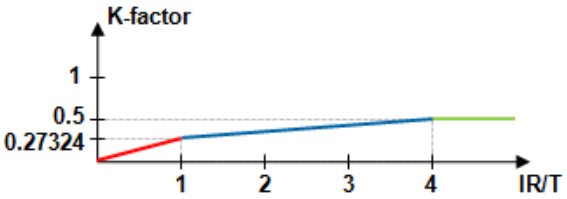


K-factor = t/T

The K-factor is a geometric calculation and does not account for physical factors for a given bend process (material type, bend operation type, tools, etc.). Because of this, the only way to know the actual K-factor for a given setup is to do a reverse calculation from an actual bend (that is bend the metal, measure the result, and calculate the K-factor).

The Discovery application determines the correct K-factor curve so you have a more accurate result, without changing the K-factor or frequently change bend tables. The default bend allowance produces an accurate result for normal bends made with normal tools and a normal press brake. For basic parts, you should use the default allowances. For special cases, you can use a bend allowance table.

The curve for the default values is shown below.



The graph shows three segments: red, blue and green.

- The red segment represents bend radii less than the material thickness. Because of the physical limit of the steel compressibility, it is difficult to obtain bends with a radius less than the thickness in an air bend process.
- The blue segment shows that, as the inner radius increases, the K factor **is not constant**. It increases up to approximately 0.5 when the inner radiu is about four times the material thickness. It does not increase beyond that because the material is no longer stretched. A K-factor greater than 0.5 is not possible.
- The green segment is constant at 0.5, which is the neutral axis for bends with a radius greater than four times material thickness.

The point where the red and blue segments meet is important. It represents a bend radius equal to the material thickness. For normal bends, a reverse calculation finds the K-factor to be:

K = (4-PI) / PI = 0.27324

This can be confirmed by press brake manufacturers. It is correct for normal bends because it is based on the actual physical result.

You can build bend allowance tables for your materials and processes. When your tables are filled (which is a one-time task), you can unfold with the tables, or use the default value.

With a sheet metal design selected at the top level of the Structure tree, the **K-Factor Type**option displays in the **Sheet Metal** group of the **Properties** panel. By default, the K-factor type is set to **Variable**, but you can also set a constant K-factor.

The **Constant** K-Factor type option allows you to enter a numerical value for the part. Using this option, you can map the design to other mainstream CAD functionality, or, adjust the design to a manufacturer's unique standards or tolerance levels to achieve various unfolding results.

> **Note:** You should only change the K-factor so it corresponds to the working properties of the metal.

14.8.9. Using a Bend Deduction Table

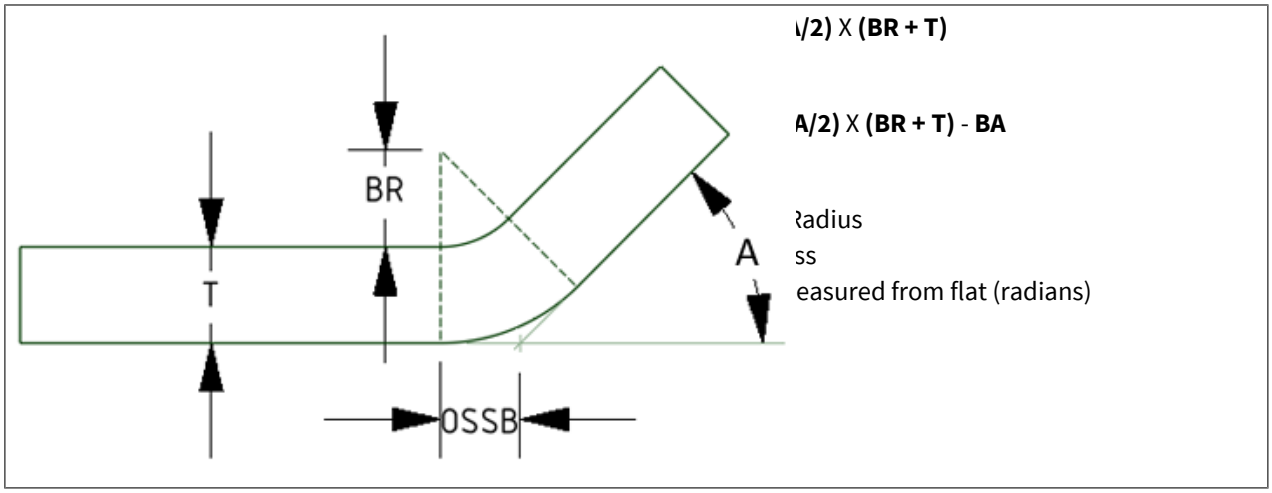
Bend deduction tables are used to calculate flat pattern layouts. The tables are based on the Bend Deduction (BD) formula, which calculates the flat line length to produce a specific bend angle.

BD = 2 X OSSB - BA

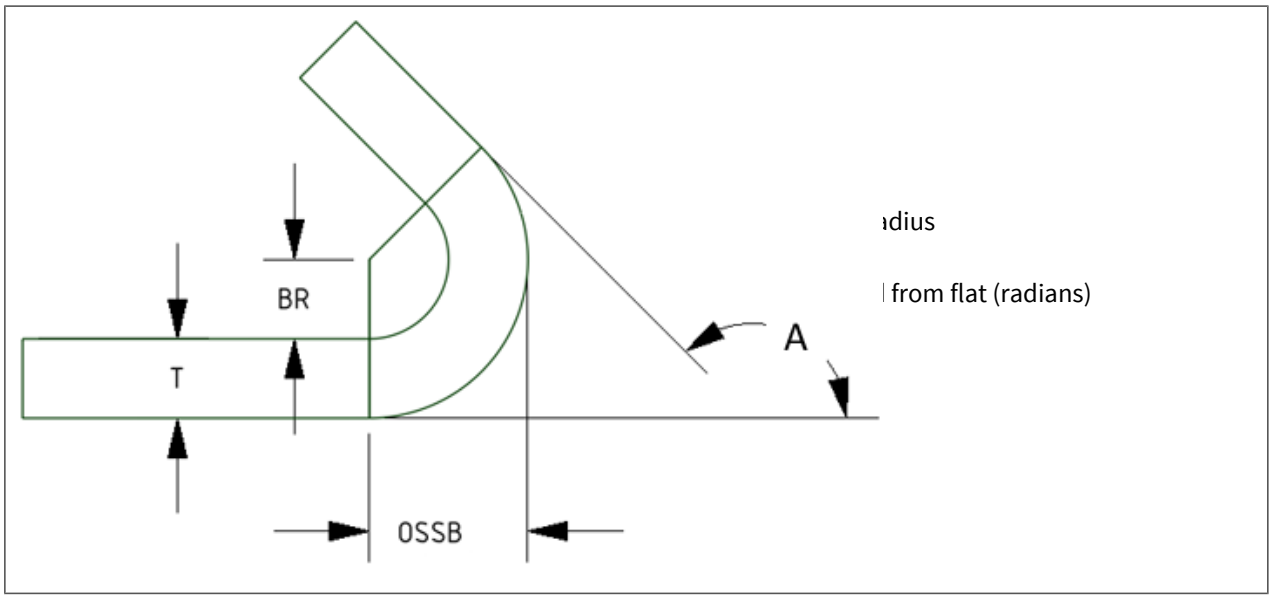
Where:

- **BD** is Bend Deduction
- **BA** is the Bend Allowance that equals the arc length of the bend along the *neutral line*. The *neutral line* is the radius where the material transitions from compressive stress (inside) to tensile stress (outside)
- **OSSB** is the Outside Set Back. Depending on the bend angle, the Discovery application uses two different definitions of OSSB as explained below.

For bend angles less than 90-degrees, OSSB is measured from the beginning of the bend to the Outside Mold Line, which is the intersection of the planar faces. See the image below.



For bend angles greater than, or equal to, 90-degrees, the OSSB is measured from the beginning of the bend to a plane tangent to the outside bend face. This means that the OSSB is the same as the outside radius of the bend. See the image below.



Note:

- If a bend table is not assigned to a part, the Bend Allowance is calculated from the default K-factor found in the Sheet Metal Options.
- If a bend table is assigned to a part, the Bend Allowance is obtained from the table and the assigned bend table file must specify tables for all possible combinations of part thickness and Vee Die width.
- If some bends in the part use Vee Die and other bends in the part do Not, the assigned bend table file must specify a table with no Vee Die Width entry.
- If you enter values in the Properties panel for a Bend Allowance or Bend Deduction, it will override the values obtained from the bend table or computed using K-factor.
- You can set the bend deduction to a negative value to correctly represent the corresponding bend allowance.

A bend table can also be assigned to the part. All values may then be obtained from the bend table instead of being calculated. Any values not obtained from the bend table may result in an error when you unfold the design. You can clear the sheet metal bend table assignment from a component by selecting the blank value from the **Bend Table** property drop-down.

The table below shows a simple bend table as it would appear in a spreadsheet editor. Keywords are shown in bold. Angle Values are measured from the flat.

	A	B	C	D	E	F	G
1	**Version**	2					
2	**Type**	BendDeductionTangent					
3	**Units**	mm					
4							
5	**Thickness**	1					
6	**AngleValues**		90	60	45	120	135
7	**Radius**	1	2.1	1.043	0.707	1.467	1.15
8	**Radius**	1.2	2.16				

When you unfold a sheet metal design, the thickness, inner bend radius, and angle from the design are used to determine the value in the selected table.

Once you select a bend table, you can edit the **Vee Die Width** property in the Properties panel. Vee die width sets the width of the tooling that produces the bend.

To use a sheet metal bend table to calculate unfolded lengths, you can use an existing file or create a custom file. To use an existing file, select a sheet metal component and change the bend table property in the properties panel. To use a custom file, do the following:

1. Copy the `Sample.csv` file in the *installation directory*>*/Library/Bends* folder where *installation directory*>is where you installed the software or into a Discovery Support File directory.
2. Paste it into the same directory and give it a new name.
3. Edit the file as described below in the **Comma-separated value file** section.
4. Select the component and change its **Bend Table** property for the new custom file.

14.8.9.1. Comma-Separated Value File

Bend table files are stored as comma-separated value (CSV) files, which can be edited. The file below shows the same data as in the previous table, but as it appears when stored in the comma-separated value file.

```
1  Version,2
2  Type,BendDeductionTangent
3  Units,mm
4
5  Thickness,1
6  AngleValues,,90,60,45,120,135
7  Radius,1,2.1,,0.65,5.5,6.6
8  Radius,1.2,1.9,,,,
```

Tables can have two types of data. In the example above, the data type is **DataTypeCompensation**, which is the default. The table below is an example where data type is set to **DataTypeRadius**.

	A	B	C	D	E	F	G
1	**Version**	3					
2	**Type**	BendAllowance					
3	**Units**	mm					
4							
5	**Thickness**	1					
6	**VeeDieWidth**	20					
7	**DataType**	Radius					
8	**AngleValues**		45	60	90	120	135
9	**Radius**	1	1	1	1.1	1.2	1.3
10	**Radius**	2	2	2	2.2	2.5	3
11	**Radius**	3	3	3.1	3.2	3.3	3.5

In this table, each combination of **Thickness** and **VeeDieWidth** represents a table that provides the actual radius based on tool radius and angle. In the example above the green cells are the tool radius, blue cells are the angle, and pink cells are the actual radius applied to the geometry.

The radius table only affects bend creation. Once the bend is created the actual (table lookup) radius is stored in the bend. You can change the value later or disconnect the table without affecting the bend.

When a radius table is applied, **Bend** creation changes slightly. In the **Sheet Metal Options** panel, **Die width** is a dropdown list. The Bend radius text box label is changed to Tool radius and the input dynamically changes to a dropdown list if the following conditions are met.

- One or more edges of a sheet metal part are selected
- The selected edge is in a part that references a bend table file
- The bend table file contains a table that matches the part's thickness and selected vee die width

14.8.9.2. Keyword Description Parameters for Bend Tables

Keyword	Description	Parameters
Version,<version-number>	Specifies the file format version.	The file format version number is provided. The current version is 3.00. Versions 1.00 and 2.00 are also supported.

Keyword	Description	Parameters
Type,<type-name>	The type of data contained in the file.	A single string representing the type of data contained in the tables. Supported values are file format version dependent. • Version 1.0 ° **Bend Allowance** ° **BendDeduction:** Outside setback is measured from the bend outside mold line for all bend angles. • Version 2.0 ° **Bend Allowance** ° **BendDeductionOml:** Outside setback is measured from the bend outside mold line for all bend angles. ° **BendDeductionTangent:** Outside setback is bend angle dependent: Bend angle < 90 degrees: outside setback is measured from the bend outside mold line. Bend angle >= 90 degrees: outside setback is measured from tangent plane to bend. • Version 3.0 ° **Bend Allowance** ° **BendDeductionOml:** Outside setback is measured from the bend outside mold line for all bend angles. ° **BendDeductionTangent:** Outside setback is bend angle dependent: Bend angle < 90 degrees: outside setback is measured from the bend outside mold line. Bend angle >= 90 degrees: outside setback is measured from tangent plane to bend.
Units,<units-type>	The default units are used for numeric values which do not have units provided explicitly.	Must be in, mm, or cm.
Thickness,<value>	Begins a bend table definition.	A single parameter specifying the thickness for which the bend deductions in the table are defined.
VeeDieWidth,<value>	Optional. Specifies an optional tooling parameter, allowing multiple sets of bend deduction values for a given thickness. Must occur after a Thickness keyword.	A single parameter specifying the width of the die tool.

Keyword	Description	Parameters
DataType, <datatype-name>	Optional. Specifies whether bend radii will be specified manually or obtained from a Bend Table. Inputs in the Bend tool switch to dropdown lists of values contained in the table.	Two types are supported in Version 3.0. • DataTypeCompensation: This is the default and was the only type for Versions 1.0 and 2.0. Allows for radius values to be manually entered. • DataTypeRadius: New in version 3.0. Limits bend radii and VeeDieWidths to those contained in the table.
AngleValues,<value>, ...	Defines the columns of the bend deduction table. Note the extra empty cell to aid in table alignment. The values defined by the table are associated with the preceding Thickness and VeeDieWidth keywords.	A list of angles, specified in degrees.
Radius,<radius>,<deduction value>, ...	Defines the body of the bend deduction table. Individual deduction values may be omitted. The total number of deduction values provided cannot exceed the number of values provided above for the AngleValues keyword.	The first parameter (radius) provides the inner bend radius for the table row. The following parameters all provide the deduction values for each combination of angle and radius.
KFactorFallback <True/False>	Controls what happens when an invalid VeeDieWidth is encountered.	**True**: K-Factor is used **False**: The Bend fails. This is the default.

14.9. Changing Sheet Metal Part Properties

You can modify the thickness, inner radius, and K-factor of sheet metal components in the Properties panel. Modifying the component's properties affects all the objects in the component. Modified property values appear in bold. You can set the default values for sheet metal components by setting Sheet Metal Options. Delete a modified property value to return it to the default value.

The thickness of sheet metal is maintained with an offset relationship. A sheet metal component retains its properties when you move it into a non-sheet metal component.

1. Select the sheet metal component in the model tree.
2. Change the following in the Properties panel:
   - **Bend Table**: The bend table to use for this sheet metal part.
   - **Flatten form as**: How forms should be represented in the unfolded part. You can choose from the following values:
     - **No flatten**: The form is shown as-is in the unfolded part.
     - **Outline**: A dotted line is shown for the extents curve and center cross-hair. The outline is added to the Bends layer on the unfolded part, which is hidden by default. This is the default value.
     - **Remove**: The form is not shown on the unfolded part.

- **Inner Radius**: How the inside radius of a bend is calculated. This field shows you what you have set in the two fields below.
  - **Radius**: The radius of a bend.
  - **Type**: Select **Thickness Ratio** or **Absolute**.

> **Note:** When the Bend Radius in Sheet Metal Options is set to **Thickness ratio** and you change the sheet metal part thickness for the part, the value for the **Inner Radius** now changes in the Properties panel. Hence, changing the part thickness property will not change all the bend radii, but the displayed bend radius changes to show that there is a problem for you to fix. You can fix them by selecting the desired bends and changing their properties manually.

- **K-Factor**: The K-Factor is used to calculate the bend radius, and is determined by the material, the type of bend, and the ratio of the bend radius to the thickness of the metal.
- **Material**: You can set the gage for a material used in a sheet metal part. Assigning a gage enables you to control the thickness of a sheet metal part at a more granular level. Gage, or, 'gauge', is the thickness of the metal organized by numbers: the smaller the number the thinner the metal. A sheet metal design or component of a design must be assigned a material in order for the gage drop-down to display. You cannot select a gage for designs or components that are assigned the 'Unknown Material' option.

  To assign a material, from the Structure tree select the design at the top level, or select the component for which you want to assign a material, and then click the Material Name drop-down from the Material group in the Properties panel. From the Thickness group in the Properties panel, click the gage drop-down, and select a gage.
- **Sheet Metal**: True or false. Determines whether or not the component is a sheet metal part. All sheet metal properties and parameters are removed from a part when you turn off the Sheet Metal property in the Properties panel. This includes things like junctions, reliefs, forms, etc.
- **Thickness**: The thickness of the sheet metal part.

  You may need to change the inside radius of bends, change the bend or corner reliefs, or alter the geometry after you change the thickness.

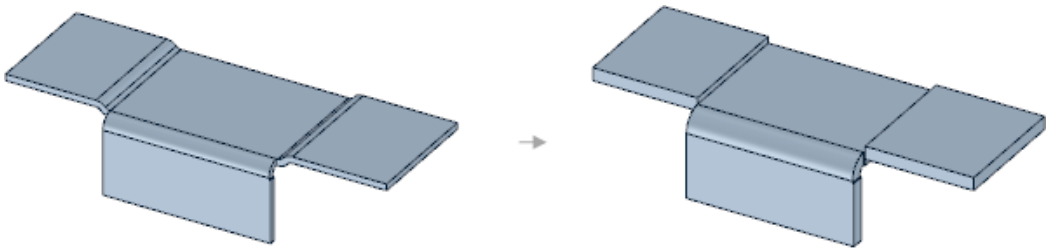
14.9.1. Exporting Sheet Metal Part Properties

With a Sheet Metal part selected, the RMB now has an **Export Sheet Metal Properties** command. It opens the Export Sheet Metal Properties dialog for writing a CSV file. The file has columns for the following properties.

- Thickness
- Area of the flattened part
- Perimeter of the flattened part
- Maximum X and Y extents of the flattened part
- Number of holes
- Number of bends

You must first unfold the part to have the flat properties export.

14.9.2. Changing Part Properties Example



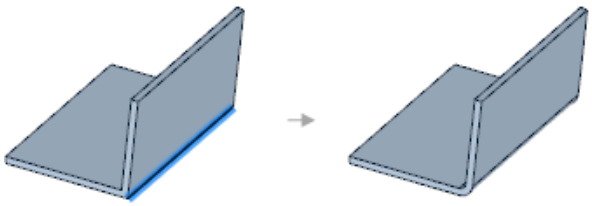
Changing the thickness of parts caused the S junctions to become no-overlap junctions because the radius was too small for the new thickness. You would need to change the inside radius to convert the junctions back to S junctions on these parts.

14.10. Correcting a Sheet Metal Component

The Discovery application 's sheet metal functionality can be used to fix components that are intended to be made from sheet metal but which do not conform to the sheet metal standards required for production. There are a number of typical errors in sheet metal creation that can be easily remedied in the application.

1. Fix joints that are not bendable.

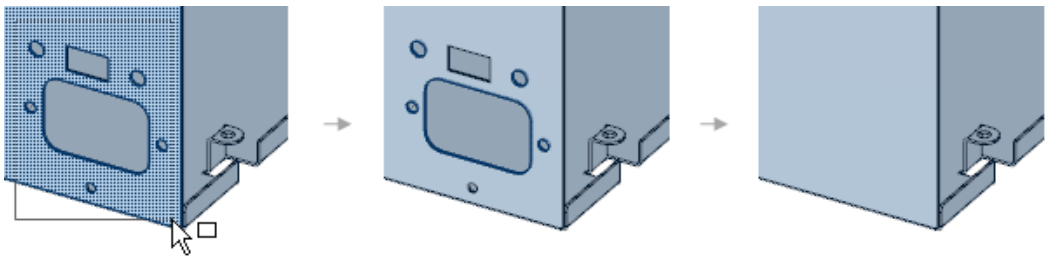
   Select the edge of an improper joint, click Junction, and select a junction type. Once a proper joint has been created, it will update with changes to part thickness, bend radius, and K-factor as well as show the joint's bend allowance on the unfolded design. An example of this change is shown below.



2. Fill in extraneous geometry.

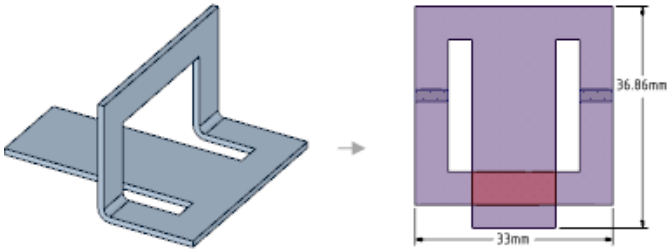
   A sheet metal component often has a number of details (such as punches, louvers, and vents) that may not be relevant to each person involved in the manufacturing process. The application allows these details to be quickly and easily filled in order to simplify parts and retain focus on only the elements necessary for each step of the manufacturing process.

   To fill in geometry, use Select to box select desired features and then Fill to remove them. An example is shown below.



14.10.1. Identifying Problems When Unfolding Parts

If an unfolded component has conflicting geometry, the conflicting geometry is made a separate surface in the model tree and highlighted in red, as shown in the figure below. The edge that prevents the unfold is also highlighted.



15: Materials

This section contains the following topics:

When you create a geometry or import one without a material definition in the CAD model, the Discovery application automatically assigns structural steel as the default material. Then, if a fluid condition is applied, the application applies the default fluid material of water.

> **Note:** You can select your material library and default solid and fluid materials in **File Menu** > **Settings** > **Physics**, under **Material Defaults**. You must restart the application or create a new simulation for the new defaults to take effect. Furthermore, you can access additional materials from the Granta MI materials databases under **Granta MI Materials** (a restart is required).

> **Note:** When you sign in to the cloud during a Discovery application session, you can import Granta cloud materials. Click **Materials** > **Import from Granta Cloud Materials**. The **Granta Material Picker** is launched. This allows you to choose materials to add to your local Discovery application library, which will then be available for simulations.

You can assign different materials to your model by selecting specific bodies and then choosing the material. The **Materials** dropdown menu in the **Simulation** tab, or the **Assign** dropdown menu from the **Materials** group in the **Prepare** tab show the materials already contained in the model or recently used materials.

You can select **Other** to search the full list of materials. You can also add materials to your favorites by clicking the star next to each one. Your favorite materials are placed at the top where they are easy to find. To remove a material from your favorites, click the star next to it again.

You can also change the assigned material by selecting the material from the **Physics** tree and double-clicking it to edit it in the HUD. If you select specific bodies within a material assignment, and then make changes to the selected material, assembled temperature, or edit the material properties, a new material assignment will be automatically created. A single material assignment will be used to reference all bodies with that material and, in the case of solid materials in structural simulations, the same assembled temperature. Any assignments of the same material and assembled temperature will be combined into a single assignment automatically.

You can filter materials based on physics compatibility or the selected library. Note that the **Local library** is where any edited materials that are saved are stored, so if this is disabled you will not see any user saved materials. If you are accessing additional materials using the Granta Selector or EduPack and exporting them to your local or shared library locations, a separate license is required.

Once you have selected a material, you can modify the properties or **Appearance** for that material. You can also obtain a detailed description under the **Information** section of the material options.

15.1. Material Properties

Here is a list of material properties available in the Discovery application. For some properties with temperature dependence, tabular data is available.

Note: Temperature-dependent material properties are based on the computed temperature for all physics except in the following cases:

- Modal and random vibration simulations use the body temperature, which you can set in the physics tree.
- Structural without thermal conditions and topology optimization simulations use the assembled body temperature, which you can set in the **Materials** HUD. Thermal stress is calculated based on the difference between the assembled temperature and the default body temperature specified in the tree.
- Fluid simulations, including fluid-solid (conjugate) heat transfer, use the initial fluid temperature (specified in the physics tree) to define material properties for both fluids and solids.

Note: The default body temperature can be adjusted in **File Menu > Settings > Physics > Physics Defualts**, under **Working temperature**.

Density	The density of a material is the mass per unit volume. It is essentially a measurement of how tightly matter is packed together in a volume.
Molar mass	The average mass of a substance per mole of molecules. The base units are entered as [mass mole^-1], for example, g/mol.
Young's modulus	Young's modulus measures the stiffness of a solid material. It describes a material's strain response (proportional deformation) to uniaxial stress (force per unit area) in the direction of this stress. For example, if you were to pull on the ends of a wire, the wire would get longer. If you were to put a weight on top of a column, the column loses height.
Poisson's ratio	Poisson's Ratio measures the ratio of the proportional decrease in the lateral measurement of a material to the proportional increase in length when it is elastically stretched. So, when a material is stretched, it tends to contract in the directions transverse to the direction of stretching. For example, when a rubber band is stretched, it becomes noticeably thinner. The Poisson ratio, then, is the ratio of relative contraction to relative expansion. In certain rare cases, a material will actually shrink in the transverse direction when compressed (or expand when stretched) which will yield a negative value of the Poisson ratio. Most materials have Poisson's ratio values ranging between 0.0 and 0.5. A perfectly incompressible material deformed elastically at small strains would have a Poisson's ratio of exactly 0.5.
Shear modulus	Shear Modulus measures the ratio of shear stress to shear strain. It describes a material's response to shear stress, such as cutting it with blunt scissors. A large shear modulus value indicates a solid is highly rigid. In other words, a large force is required to produce deformation. A small shear modulus value indicates a solid is soft or flexible and little force is needed to deform it.
Bulk modulus	The Bulk Modulus of a substance is a measure of how resistant to compression that substance is. It describes the material's response to (uniform) hydrostatic pressure, such as the pressure at the bottom of the ocean or a deep swimming pool. Bulk Modulus is defined as the ratio of the infinitesimal pressure increase to the resulting relative decrease of the volume.
Tensile yield strength	Tensile Yield Strength is defined as the point on the Stress-Strain curve where the material starts to deform plastically, or, to put it another way, the stress a material can withstand without permanent deformation.

Tensile ultimate strength	Tensile Ultimate Strength is defined as the maximum stress a material can withstand.
Thermal expansion coefficient	The thermal expansion of a material is the volumetric response of the material to a change in temperature. Thermal expansion is assumed to be isotropic in behavior and so expands or contracts equally in all directions.
Thermal conductivity	For solid thermal and thermal structural simulations, this is the Isotropic thermal conductivity of a material, and it is the ability of the material to conduct heat equally in all directions. For fluid-solid heat transfer simulations, you can specify the **Directionality** as **Isotropic** or **Orthotropic**. Orthotropic thermal conductivity is the material's ability to conduct heat differently in each of the X, Y, and Z directions. Choose the reference frame to be in the global or a user-defined coordinate system. **Note:** When specifying orthotropic thermal conductivity, defining very low values in one or two of the three directions may lead to convergence difficulties. In **Explore**, you can add a local fidelity to the orthotropic solid region or solve the model in **Refine**. In **Refine**, a local fidelity may be required to ensure a high quality mesh for the orthotropic region. **Note:** Thermal conductance at a contact or fluid-solid interface is calculated using the isotropic value when the material has orthotropic thermal conductivity. You must also define the isotropic value.
Specific heat	The specific heat capacity of a material is the heat required to raise the unit mass of a substance by one degree of temperature.
Viscosity	The viscosity is the dynamic viscosity of a material and is the resistance to flow. The dynamic viscosity must be defined when modeling fluid flow.
Electrical conductivity	Electrical conductivity is the measure of a material's ability to conduct an electric current. A high conductivity indicates a material that readily allows the movement of electrical charge.
Relative permittivity	Relative permittivity is the ratio of the permittivity of a material to the permittivity of free space or vacuum.
Electric loss tangent	Electrical loss tangent represents a dielectric material's ability to dissipate the power of a high-frequency electric field. The smaller the loss tangent, the less lossy the material.
Relative permeability	Relative permeability is the ratio of the permeability of a material to the permeability of free space or vacuum.
Electrical Conductivity	Electrical Conductivity is a material property, measured in units of S/m (Siemens per meter).

15.1.1. Tabular Material Properties

Some material properties listed in this table provide constant values as well as tabular data. For example, density may be a constant value displayed or entered in the numerical entry field, or it may consist of tabulated temperature-dependent values.

To view tabulated data for some of the material properties, click the table button next to the property value.

Entries in the first column of the table are sorted in ascending order. New data can be added to the empty fields at the bottom of the table. Note that newly added data is automatically sorted when the table is refreshed or reopened.

You can import data to the table or export the entire table to a CSV file format. You can delete rows from the table and copy an entire table to the clipboard and paste it elsewhere.

15.2. Environmental Properties

Here is a list of environmental properties available in the Discovery application.

> **Note:** Click **Environment** on the Material HUD options panel to expand the environmental properties.

Climate change (CO_2-eq)	This value is the CO_2-equivalent of 1 kg of the material from a combination of its ores and some recycled content. Climate change refers to a change in the state of the climate that can be identified by shifts in the mean and/or the variability of its properties, and that persists for an extended period, typically decades or longer. It is calculated based on the global warming potential (GWP), developed by the Intergovernmental Panel on Climate Change. The GWP model allows comparisons of the global warming impacts of different gases, reported as carbon dioxide equivalents (denoted as CO_2-eq).
Embodied energy	This value is the embodied energy of the production of 1 kg of the material from a combination of its ores or feedstocks and some recycled content. It represents both the direct and indirect energy use throughout the life cycle of a good or a service, and it is based on the cumulative energy demand method developed by ecoinvent. It includes renewable and non-renewable energy resources (biomass, fossil, geothermal, nuclear, primary forest, solar, water, and wind), and the value reported is the sum of all of these resources.
Potential to recycle	This value indicates whether, when the material reaches its end of life, it can be recycled, regardless of the recyclate's quality. A material is designated as recyclable if it can be reprocessed, by generally available and economically viable means. It includes processing of both new and old scrap: new scrap is material that is generated during the manufacturing process, and old or post-consumer scrap is that which has been used by a consumer.

15.3. Changing the Appearance of Materials

You can make a material appear more realistic by applying color and opacity. For any default material that already exists in the simulation, the appearance properties are preset for optimum visualization. Any newly created materials will not have any preset appearance properties.

Note: These changes will apply wherever this material is assigned in this simulation.

1. Select the body in your geometry for which you want to change its appearance.
2. Select **Materials** in the ribbon.
3. Under the material options, view all material properties, then click **Appearance** to expand.
4. Under **Appearance**, you can modify any or all the categories listed below.

Option	Description
Color	Select a color for the assigned material.
Opacity	Adjust the slider to the desired value of transparency.

15.4. User-Defined Material Libraries

You can define your own materials and specify its properties by modifying the properties of an existing material in the list. This user defined material will be available in the material list for a project if it is saved. If you click **Save to local library** under the material options, your new materials will be saved to the local material library and available anytime you start the Discovery application. A unique material name must be used to avoid overwriting any existing material.

When you first save a newly defined material to the local library, the application creates a file named `MyLocalMaterialLibrary.json`. This is usually a hidden file, but you can find it by typing `%localappdata%\Ansys_Inc\Discovery\LocalMaterialLibrary` in your Windows File Explorer. The `LocalMaterialLibrary` folder does not exist until you save your first material.

Anytime you create a new material, it's saved to `MyLocalMaterialLibrary.json`. If you rename the file in the `LocalMaterialLibrary` folder, new attempts to save the material to the local library saves the new file with the original default name: `MyLocalMaterialLibrary.json`.

When you start the application, it will attempt to read all files in the `LocalMaterialLibrary` folder. After it successfully parses the materials, they are added to the local material library. Note that only files using the default file name `MyLocalMaterialLibrary.json` provide the option to **Remove from local library** those user defined materials. If a material is loaded from a file other than `MyLocalMaterialLibrary.json` in the `LocalMaterialLibrary` folder, this option will not be available.

If you load a project that contains a material in the local library, but this material does not exist in your local library, the application will convert the material to a user defined material when it starts up.

15.5. Accessing Additional Granta MI Materials

The Discovery application allows you to access additional materials by connecting to a Granta MI materials database. Granta MI databases are licensed separately.

To connect and load materials from Granta MI:

1. Go to **File menu** > **Settings** > **Physics**.
   a) Under **Material Defaults**, select **Granta MI materials database.**
      Additional settings appear under the **Granta MI Materials** section.
   b) Under **Granta MI Materials**, specify the URL of your Granta MI application server.

> **Note:** Contact your Granta MI administrator to find out the address of your Granta MI application server. Your access will be validated based on your Windows credentials.

2. Choose the set of materials you want to load by selecting the **Database**, **Table**, and **Subset**.
   Granta MI groups similar kinds of data into different categories, known as a **Table**. **Subset** provides a more focused view or selection of the data within a **Table**.
3. Restart the application for the changes to take effect.
   Select **Materials** in the ribbon of the **Simulation** tab to access the additional materials from the list.

> **Note:** To see the materials loaded in from the Granta MI application server, disable **Physics compatibility** under the filter in the right side of the **Material** HUD.

> **Note:** For information about Granta Materials Data for Simulation (MDS), which is a built-in database of 700+ predefined materials included with the Discovery application, see **Granta Materials Data for Simulation Licensing**.

15.6. Compressible Flows

Compressibility effects are encountered in gas flows at high velocity and/or in which there are large pressure variations. When the flow velocity approaches or exceeds the speed of sound in the gas or when the pressure change in the system ($\Delta p/p$) is large, the variation of the gas density with pressure has a significant impact on the flow velocity, pressure, and temperature.

Compressible flows can be characterized by the value of the Mach number ($M \equiv u/c$, where c is the speed of sound in the gas). When the Mach number is less than 1.0, the flow is termed subsonic. At Mach numbers much less than 1.0 (M 0.1 or so), compressibility effects are negligible and the variation of the gas density with pressure can safely be ignored in your flow modeling. As the Mach number approaches 1.0 (which is referred to as the transonic flow regime), compressibility effects become important. When the Mach number exceeds 1.0, the flow is termed supersonic, and may contain shocks and expansion fans which can impact the flow pattern significantly.

Compressible flows are described by the standard continuity and momentum equations and the treatment of density as detailed below.

The compressible form of the gas law (the ideal gas law) is written in the following form:

$$\rho = (p_{op} + p)/(RT/M_w)$$

Where p_{op} is the operating pressure, p is the local static pressure relative to the operating pressure, R is the universal gas constant, and M_w is the molecular weight. The temperature, T, will be computed from the energy equation.

Guidelines and Best Practices

- The Discovery application provides compressible flow modeling capabilities for subsonic internal flows. However, there may be cases where the flow becomes transonic or supersonic due to a flow constriction in the fluid domain. You can use the application to solve internal flow applications such as high-speed gas flows or gas flows with large temperature variations, flows through orifices or other flow constrictions, and exhaust systems with high

temperature ranges. External aerodynamics applications, supersonic and hypersonic flows, and compressible liquids should be avoided.

- For simulations including multiple fluids, all gases in the simulation will be treated as compressible using variable density with the ideal gas law. All liquids in a simulation with multiple fluids are treated as incompressible, regardless of whether compressibility is turned on.
- Buoyancy is not compatible with compressible flow. Turn off the **Include buoyancy** selection in the **Gravity HUD** to solve a compressible flow simulation.
- Mass flow rate specification at a flow outlet is not supported in compressible flow simulations in **Explore** and the LiveGX solver in **Refine**. You can specify pressure at the flow outlet or use the **Refine** stage without LiveGX instead.
- Velocity specification at a flow outlet is not supported in compressible flow simulations. You can specify pressure at the flow outlet or use the **Refine** stage (without LiveGX) to specify mass flow rate at the outlet.
- In **Refine**, if a domain with a compressible fluid is excluded from the simulation, after subsequent inclusion it is incompressible.

16: Scripting

Scripting in the Discovery application allows users to automate tasks, customize workflows, and extend functionality across various areas of the application. Whether automating repetitive tasks or creating advanced custom scripts, scripting enhances efficiency and flexibility within the simulation environment.

For more details on scripting, refer to the help documentation at https://developer.ansys.com/docs/discovery.

This section contains the following topics:

16.1. Script Editor

To launch the **Script Editor**, choose **Script Editor** from the **File Menu** dropdown menu.

> **Note:** The **Script Editor** cannot be opened while **History Tracking** is on.

You can work with the Discovery application programmatically using Python. A built-in script editor allows you to create and run scripts to perform repetitive tasks.

Write your script in the main Script Window. You can also enter single commands in the command line. You can also drag and drop an existing script file into the **Script Editor** to load it directly for editing.

For reference, API documentation is located in the installation directory: `[local drive]\ANSYS Inc\[version]\Discovery\Discovery.Api.[version]`.

This section contains the following topics:

16.1.1. Inserting a Selection

1. Click **Insert Selection** to paste the current selection as a script into the script window. You have three options for pasting: **Smart Variable**, **Index**, and **Ray**.
2. (Optional) Click the dropdown arrow to select another option and then click **Insert Selection**.

Index	The internal index of each selected entity.
Ray	The internal ray fire point and direction.
Smart Variable	Combines the other methods to make selections better persist when replaying scripts, by storing more information. This makes replay more robust in matching the selected entity during geometry changes.

16.1.2. Recording Actions for a Script

Click the **Record** to record scriptable actions in the Discovery application, which are then written as code in the Script Editor. The type of selection that is chosen will affect how the script is recorded.

Note: The easiest way to learn the scripting API is to use the record option while performing your operations and then modify the recorded script as appropriate.

16.1.3. Writing a Script

Intellisense

Intellisense, or intelligent code completion, is included in the editor to assist with entering commands. It is a context-aware code completion feature that improves the process of coding applications by reducing typos and other common mistakes. The code completion and related tools serve as documentation and disambiguation for variable names, functions, and methods using reflection.

When you type in the Script Editor, you will see suggestions for commands, helper functions, and variables (created or with Smart Variable Selection).

Auto complete features add parentheses when writing functions in the Script Editor. For-loops can be created automatically.

You can also search for existing APIs from the Script Editor and select it to automatically add the code to import it into the script.

- The example shows the import of the **CustomProperty API**. Start by typing in the Script Editor and click the **All** tab to see all suggestions.

```
# Python Script, API Version = V21

    Custom
    CustomForm
    CustomFormOptions
    CustomHelper
    CustomLineCap
    CustomMethods
    CustomObject
    CustomPartProperty
    CustomPartPropertyDictionary
    CustomProperty
    CustomPropertyDictionary
```

- Select **CustomProperty** from the list. The API is then imported into the script as shown.

```
from SpaceClaim.Api.V21 import CustomProperty
CustomProperty
```

- Use **Cast** and **CastCheck** with expressions or variables.
- **Cast** enables you add a type to an expression. For example, you can declare a variable and set its type in the Scripting Tool.

  Note that when you cast a variable inside of a method, it is only cast to that type inside the method.
- **CastCheck** enables you to check the type of expression by surrounding it with an if-statement. Once a variable is cast, you can choose to add a **CastCheck**, which adds an if-statement to the code. This if-statement allows you to confirm the variable's type before performing additional calculations or modifications.

Selection Converting and Filtering

To improve selection in scripting, created selections can be converted or filtered based on size, geometry, or other requirements.

For each of the selections, you can choose from the Filter and Convert methods available. For example, you can choose to filter faces, edges, parts, bodies, or components. You can convert selections to types (such as faces or edges), or convert by shape, size, etc.

For example, you can get the root part of the assembly, convert that selection to edges, and then filter by size:

```
omp = Selection.Create(GetRootPart())

smalledges = comp.ConvertToEdges().FilterByLength(MM(0), MM(.05))
```

Power Selection and Smart Variables

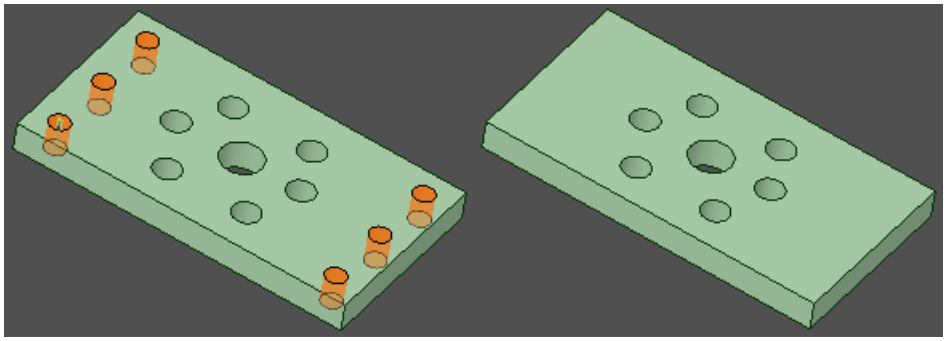
Power Selection is recorded and, when using Smart Variables, can be used to perform similar operations on multiple bodies.

```
selection = PowerSelection.Faces.BySurfaceHoleRadius(Selection.Create(Face1),

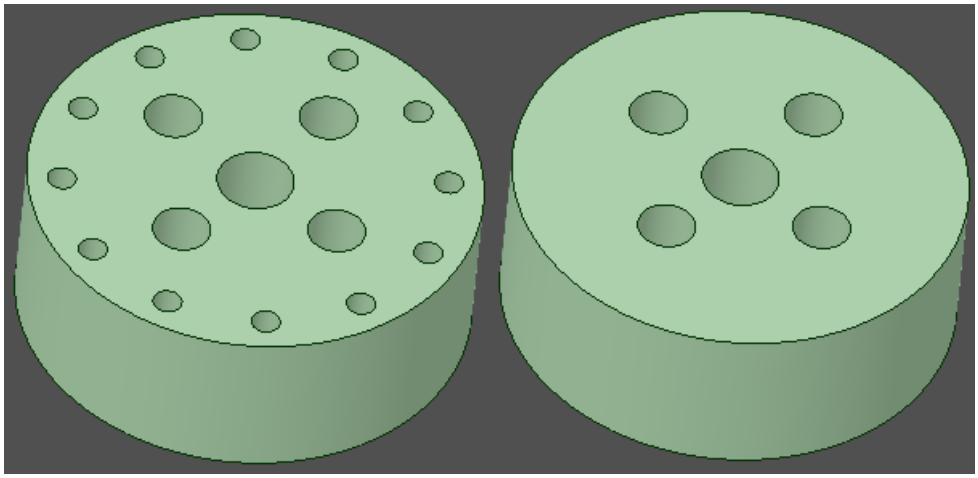
    PowerSelectOptions(False),
    SearchCriteria.SizeComparison.SmallerOrEqual)
Fill.Execute(selection)
# EndCommand
```

For example, you can record the power-selection of holes with a certain diameter, and then the filling of those holes with the Fill tool. That script can then be used to fill holes of the same size in other models, even if the number of holes is different.

The example below shows six holes power-selected, filled and recorded.



The script is then run on a different model and 12 holes with the same diameter are filled.



Note: Refer to the Class Library for a detailed description of the API. A copy of the Class Library compiled help file (*.chm) is included in the installation. The path will be similar to the paths shown below and will change according to the version you are using and where Discovery is installed. For example: `C:\Program Files\ANSYS Inc\[version]\Discovery\Discovery.Api.[version].`

16.1.4. Publishing Scripts to the Ribbon

You can publish a script to create a tool button in the ribbon so that it can be used in any project.

To publish a script:

1. Write a script.
2. Click **Publish Script**. The **Publish Script** dialog appears.
3. Enter a **Name** and **Tooltip**.
4. (Optional) Enter **Tab name** and **Group name** for script location in the ribbon. **Note:** Scripts cannot be added to existing Discovery application tabs so existing tab names are not available.
5. For **Icon path**, enter a path to the icon, or browse to select the icon.
6. (Optional) Choose **Password-protected copy** if you want to hide the script contents.
7. Click **OK**.

The tool is added to the chosen **Tab** and **Group** on the **Ribbon**. Once a script is published, it can be edited or removed by selecting the **Edit Script** or **Remove** option on the **Ribbon** if it is not protected.

When publishing scripts, you can choose **Password-protected copy** to protect the script. When using this option, two files will be saved: a .dscript file in the **Published** directory, which is editable, and a .pdscript file in the **Published\secure** directory, which is not editable. The **Published** directory can be found at `<local-drive>\AppData\Roaming\Discovery\Published Scripts` and **Published\secure** is found at `<local-drive>\AppData\Roaming\Discovery\Published Scripts\secure`.

Protected scripts allow their creators to share the script with others. To use a shared script (either .dscript or .pdscript) you can add it to the **Published** directory.

Note: Once you publish a script, it will appear on the ribbon in any instance of the Discovery application on your computer.

Note: Published scripts run in **Debug** mode when open in the Script editor and **Run** mode otherwise. Close any published scripts that are open in the Script editor to have them run in **Run** mode.

16.1.5. Embedding Scripts

Scripts can be embedded into the Discovery project.

To embed a script:

1. Write a script.
2. Click **Embed Script** . The Name Embedded Script dialog appears.
3. Enter a **Name**.
4. Click **OK**.

The script is saved and stored in the project.

Note: You can embed a single script into the Discovery project. Once a script is embedded, it remains embedded when you **Save**/**Resume** the model.

16.1.6. Debug versus Run

Scripts are executed in Debug mode by default. This enables a script writer to get feedback on their script while creating it, as well as allowing Debugging tools. An option to execute scripts in Run mode is also available. Using Run mode is faster than Debug mode since the script is executed on a single thread.

Debugging the Script

To debug your script, click the drop-down arrow next to the green play button and select **Debug**. Then click the play button to start debug mode. Script execution stops the first time it reaches a breakpoint line. To continue, click the play button again to go to the next breakpoint line.

- **Step Into** - The debugger executes the function call statement and then pauses execution at the first line of the function called.
- **Step Over** - The debugger executes the function call and then pauses after the function returns.
- **Step Out** - Only applies if execution is first stopped inside a function (by a breakpoint inside function or when Step In is activated). The debugger executes the rest of the function without pausing, and then returns to the line after the function call and pauses.
- **Breakpoints** - You can left or middle click the grey column to the left of the line numbers to create a breakpoint, which is displayed by a red circle.

  When you click the play button in Debug mode, script execution stops the first time it reaches the breakpoint line. Clicking either button again forces it to run the breakpoint line and continue to the next breakpoint line.

  In Run mode, breakpoints are inactive. They are shown as red circles with black centers.

Note: **Step Over** and **Step Into** will work even if breakpoints are not defined.

Note: These options only apply to functions defined in the script, not native Discovery application function calls.

Running the Script

To execute your script in Run mode, click the drop-down arrow next to the green play button and select **Run**. Then click the icon again to run your script.

16.1.7. Using the DiscoveryInputHelper

The DiscoveryInputHelper enables you to pause the execution of the script to get various inputs from interactive tools, before continuing to run the script. You can set up custom workflows using the DiscoveryInputHelper functions, including creating widgets and tool guides for user input. The DiscoveryInputHelper can only be used in Debug mode.

Note: The InputHelper is still available for migrating from Ansys Discovery SpaceClaim scripts.

- You can create tool guides and specify selections which can also be filtered based on type. For example, you can choose to filter faces, edges, or bodies. You can also specify if the selections are mandatory or optional.
- Script variables can be added to a custom HUD as buttons, check boxes, dropdowns, or groups of such controls. You can also add tool tips for the controls created.
- Quantities in the DiscoveryInputHelper allow you to specify values with relevant quantities, to be used in other objects such as physics conditions.

Example - Creating Tool Guides for Selection

This example shows how you can select a body to be moved and select edges to be rounded.

```
# Python Script, API Version = V21

s1 = InputHelper.CreateSelection('Body (Required)', SelectionFilter.Body, True)
s2 = InputHelper.CreateSelection('Edge (2 Required)', SelectionFilter.Edge,
2, [True])
s3 = InputHelper.CreateSelection('Edge or face (optional)',
SelectionFilter.Edge|SelectionFilter.Face)

InputHelper.PauseAndGetInput('Select body to move. Select edges for rounding.',
 s1, s2, s3)

# Translate Along X Handle
selection = s1.Value
direction = Direction.DirX
options = MoveOptions()
result = Move.Translate(selection, direction, MM(30), options)
# EndBlock
```

```
# Create 1 Round
selection = s2.Value
options = ConstantRoundOptions()
result = ConstantRound.Execute(selection, MM(1), options)
# EndBlock
```

The script pauses to start the interactive tool, where you can select a body to be moved and two edges to be rounded.



When you finish selecting the entities, you can click **Complete** or **Abort** and resume the script. Abort doesn't necessarily act as you might expect, as it doesn't end the script execution. You will need to add some extra validation to stop the script (such as "`if click.Success == True`")

Example – Creating Quantity Inputs

This example shows how you can add multiply quantity inputs and receive a value and a unit for each. Once a value is received, it can be added to any condition that can accept that value.

```
# Python Script, API Version = V22
# Create Box
result = BlockBody.Create(Point.Create(MM(-10), MM(10), MM(0)),
Point.Create(MM(10), MM(10), MM(20)), ExtrudeType.ForceAdd)
# EndBlock

forceInputExternal = DiscoveryInputHelper.CreateQuantityInput("External",
ForceUnit.Newton)
forceInputInternal = DiscoveryInputHelper.CreateQuantityInput("Internal",
ForceUnit.Newton)
temperatureInput = DiscoveryInputHelper.CreateQuantityInput("Temperature",
TemperatureUnit.DegreeCelsius)

DiscoveryInputHelper.PauseAndGetInput("Get Input", forceInputExternal,
forceInputInternal, temperatureInput)

force1 =
Conditions.Force.Create(FaceSelection.Create(GetRootPart().Bodies[0].Faces[4]),
 forceInputExternal.Value)
temperature =
Conditions.Temperature.Create(FaceSelection.Create(GetRootPart().Bodies[0].Faces[0]),
 temperatureInput.Value)
```

16.1.8. Known Limitations

- Script recording of actions in the tree may not properly include selection. As a workaround, record the appropriate selection and edit the recorded script command.
- You may receive an error if you execute a script in Run mode which includes a Reset command and encounters an execution error. This will make the Discovery application unstable, and a restart is needed. Executing your script in Debug mode first will ensure that any issues are addressed.
- Scripts recorded in the SpaceClaim application with Index Selections may have different indices when played in the Discovery application. Updating to new Indices may be required.

- Running a script in Run mode that does multiple Refine solves with different geometries will not work. The workaround is to use Debug mode.
- Script parameters may be combined with other parameter types in the Variations table. However, script methods "ClearAll" and "ClearAllForParameters" may be problematic when used with non-script parameters and should be avoided.

17: Extension Builder

The Discovery application's **Extension Builder** is a powerful tool for creating custom extensions. With this tool, you can easily add buttons and tools to groups and tabs on the ribbon. By importing or writing scripts, you can then use your custom tools and buttons to perform a wide range of tasks within the application.

Launch the **Extension Builder** by selecting it from the **File Menu** dropdown.

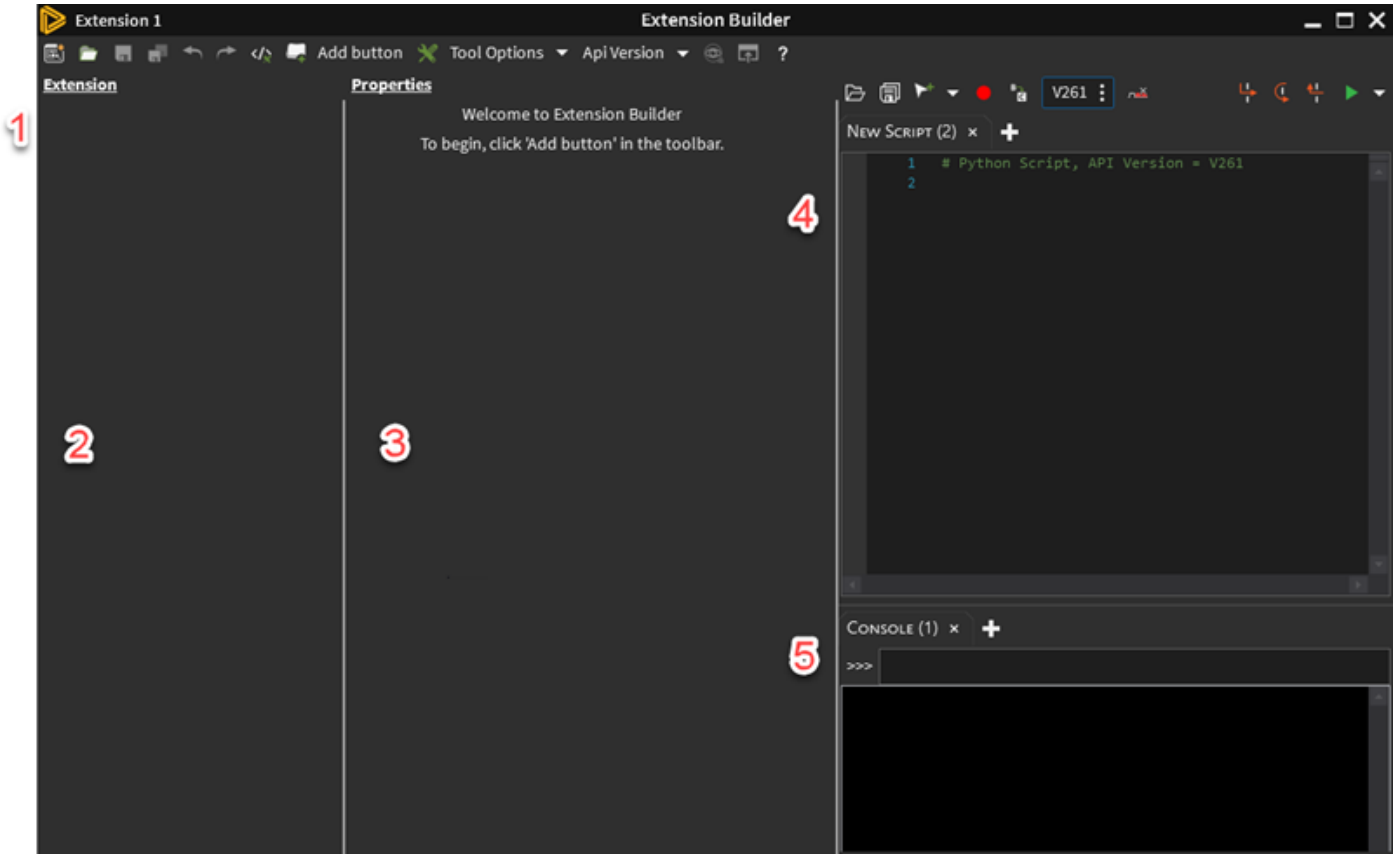
> **Note:** The **Extension Builder** cannot be opened if **History Tracking** or the stand-alone **Script Editor** is currently running.

For more information about **Scripting** and the **Script Editor**, see Script Editor on page 748.

This section contains the following topics:

17.1. User Interface

To use the **Extension Builder** efficiently, you need to learn its interface. This section explains its components and functions, helping you navigate and access tools for your tasks.



1. You can use the **Extension Builder Toolbar** to create, add, preview, save, and publish your extensions.
2. The **Extension Panel** displays the elements within your extension.

3. Using the **Properties Panel**, you can edit the properties of the selected extension element.
4. The **Script Editor** lets you write and associate scripts with your extension.
5. When you run a script in the **Script Editor**, the **Console Window** will display messages related to your scripts and extensions.

17.1.1. Extension Builder Toolbar

In the **Extension Builder**, you can access a set of buttons located at the upper left corner of the interface.

With these buttons, you can create new extensions, add elements to existing ones, preview your work, save your progress, and publish your extensions. Each button represents a specific action, making it easy for you to access the tools you need without having to navigate through different menus or options.

17.1.1.1. Adding a Reference Script

Reference scripts only include functions that you intend to call from within your extension. Reference scripts run when a function from another script is called.

1. Click **Add reference script** to add the script to your extension. The reference script will appear at the top of the **Extension Panel**.
2. (Optional) As needed, call functions in the reference script from your extension. Any code that is not inside a function may execute when the reference script is added to an extension.

17.1.1.2. Adding a Button

Click **Add button** to add a button to execute a script. It can be designed to accept user input, but this is not required.

17.1.1.3. Adding Tool Options

1. Select a tool in the **Extension Panel**.
2. Click the **Extension Panel Toolbar** or right-click the tool in the **Extension Tree** and choose the tool option to add: **Add text box**, **Add toggle**, **Add color palette**, **Add slider**, **Add combo box**, **Add list box**, **Add tool button**, or **Add sub options**.

17.1.1.4. Previewing an Extension

Clicking the **Preview Extension** toggle allows you to preview your extension within the current session of the Discovery application.

17.1.1.5. Publishing an Extension

1. Save your extension to enable the **Publish Extension** button.
2. Click **Publish Extension** to publish your extension to the Discovery application.
3. (Optional) See Save and Publish on page 763 for more information.

17.1.2. Extension Panel

Reference Scripts

After you load a reference script to your extension project using the **Add reference script** button, it will appear at the top of this section. Reference scripts run when a function from another script is called. Note that reference scripts should only include functions that you intend to call from other scripts within your extension.

> **Note:** Any code that is not inside a function may execute when the reference script is added to an extension.

Extension Tree

The **Extension Tree** displays the grouping and categorization of elements within your **Extension**.

17.1.2.1. Editing Nodes

1. Click a node in the **Extension Tree** to edit it. This will populate the **Properties Panel** and show any associated script in the **Script Editor**.
2. Modify the desired properties, as needed.
3. Once you are done, select **Apply** to update the information.

17.1.2.2. Deleting Nodes

Right-click the desired node in the **Extension Tree**and select **Delete**. This action will also delete all descendant nodes, and it cannot be undone.

17.1.2.3. Adding Buttons to Groups

1. Right-click a **Group** in the **Extension Tree** and select **Add New Button**.
2. Enter properties for the new button and associated tools in the **Properties Panel**.
3. Click **Create** to add the new button to the selected **Group**.

17.1.2.4. Adding Tool Options

1. To add a tool option to your **Tool**, right-click the **Tool** in the **Extension Tree**.
2. Choose **Add text box**, **Add toggle**, **Add color palette**, **Add slider**, **Add combo box**, **Add list box**, **Add tool button**, or **Add sub options**. See Tools on page 759 for more information.

17.1.2.5. Adding Tool Options Sections

1. To add a tool option section to your **Tool**, right-click the **Tool** in the **Extension Tree**.
2. Choose **Add primary input**, **Add options panel**, or **Add expanded**. See Tools on page 759 for more information.

17.1.2.6. Drag and Drop Options

1. Select and option in the **Extension Tree**.
2. Drag the option to a new location within the **Extension Tree**. Options can only be dragged onto nodes that allow them.

17.1.3. Properties Panel

The **Properties Panel** provides detailed information about the object being created or viewed and is located centrally in the interface.

Ensure that any icon, script, or file you select from your local machine remains in its original location. This is necessary for your extension to access it correctly.

17.1.3.1. Editing Properties

1. Select an element in the **Extension Tree** to display its corresponding properties in the **Properties Panel**.
2. Edit the desired properties to customize the element's appearance, behavior, and functionality. The **Properties Panel** enables precise control over these aspects of the extension's design and functionality. See Tools on page 759 for more information.

17.1.4. Script Editor

On the right side of the **Extension Builder**, you'll find the **Script Editor**. This is where you can create and modify scripts to extend the functionality of your extension. The **Script Editor** provides a range of features to help you write code, such as syntax highlighting, autocompletion, and debugging tools. You can also drag and drop an existing script file into the **Script Editor** to load it directly for editing.

Scripting is a powerful tool that can help you automate complex tasks, customize your workflow, and integrate with other extensions. For more information about **Scripting** and the **Script Editor**, see API on page 762, Intellisense on page 763, and Script Editor on page 748.

17.1.4.1. Managing Script Tabs

- (Optional) Click **+** to add a new tab for a new script.
- (Optional) Click **x** to close an existing tab.
- (Optional) Click  to close all script tabs, leaving only active scripts and those associated with highlighted tree items open.

17.1.5. Console Window

The **Console Window** presents text-based data that pertains to the operation of your extension, which includes status messages, error messages, debugging information, and other relevant details. By using the **Console Window**, you can monitor and troubleshoot your extension, making it an invaluable tool for identifying issues and comprehending how your extension is performing.

17.2. Tools

The **Extension Builder**'s tools are its driving force, as they can be customized with a range of options to perform various functions.

17.2.1. Adding Tools

1. Click the **Add button** in the **Extension Builder Toolbar** to open the **Properties Panel**.

2. Add **Tab name**, **Group name**, **Button name**, and **Tool name**, as well as icons and help text for these elements. Items denoted with an asterisk are required for this tool creation process. The recommended maximum size for icon images is 32 x 32 pixels.
3. (Optional) If your extension requires user input, you can select the **User input required** option to add a tool, and specify selection filters such as **Body**, **Face**, and **Edge**, to enable users to select these elements while using the tool.
4. (Optional) You have the option to add a **Complete button** and can select **Close tool on complete**, which closes the tool automatically after you finish your task.
5. (Optional) You can set the initial state of elements to **Visible**, **Hidden**, or **Disabled**, and include scripts to change their states to enhance the functionality of your extension.
6. (Optional) You can expand **Advanced Options** and choose to create a tree object, organize it into a hierarchy under the extension entry object, and add a tooltip. For more information about **Tree objects**, see Managing Tree Objects on page 760.

17.2.2. Managing Tree Objects

This feature allows you to organize customizable objects from multiple tools within a single tree, streamlining referencing and seamless integration with your tools. Key advantages of this functionality include the ability to edit customizable object values and enable the same tool to create multiple objects of the same type.

1. In the **Properties Panel**, expand **Advanced Options** and select **Create tree object** to add a tree object for your tool.
2. In **Tree hierarchy**, enter strings separated by semicolons to define the structure of your tree hierarchy.
3. Add tooltips by entering content into **Tree object tooltip**.
4. Click **Preview Extension** to see your changes in the Discovery application. The tree object will appear at the bottom of the **Extension tree** on the left side of the screen.
5. (Optional) To create another tree object, click **Complete** in your tool. This adds a new tree object to the tree.
6. (Optional) To edit an existing tree object, double-click it in the tree. The object will be highlighted in solid orange. Once highlighted, click **Complete** to make updates to the selected tree object for your tool.

17.2.3. Tool Options

Text Box	Create a text input to read user input values.
Toggle	Enable users to switch various features on or off.
Color Palette	Add display flexibility and highlights to your models.
Slider	Adjust displays by sliding up or down your chosen scale.
Combo Box	Provide users with options in a dropdown menu.
List Box	Display information based on different settings.
Tool Button	Execute tasks based on associated scripts. Alternatively, you have the option to change your **Tool Button** into a **Browse button**. Upon selection, you can specify the filter you wish to associate with your **Browse button** within your extension. Clicking on the browse feature in your extension will prompt an Open window that includes your specified filter. Note that the filter will be preceded with "*."
Sub Options	Functions like a **Combo Box** on the left side of the HUD that provides users with options in a dropdown menu. Note that a single **Sub Options** object is possible.

17.2.3.1. Adding Tool Options from Toolbar

From the **Extension Builder Toolbar** select an option from the **Tool Options**dropdown menu: **Text Box**, **Toggle**, **Color Palette**, **Slider**, **List Box**, **Combo Box**, **Tool Button**, or **Sub Options**. When adding a **Combo Box**, **List Box**, or **Sub Options** separate the input values with semicolons for proper formatting. By referencing these tool options in scripts, you can significantly enhance the functionality of your tools.

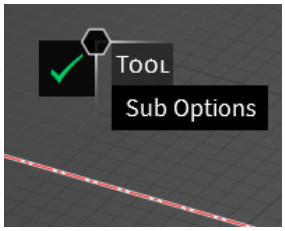
17.2.3.2. Adding Tool Options from Tree

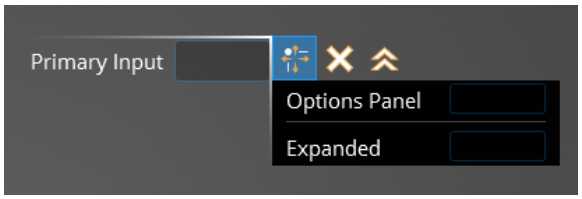
To add tool options to a **Tool** in the **Extension Tree**, right-click the **Tool** and choose from one of the options.

17.2.3.3. Adding Text box Value With Units

1. Right-click a tool in the **Extension Tree** and choose **Add text box**.
2. In the **Properties panel**, enter a **Text box name** and choose **Value with Units** as your **Text box type**.
3. As desired, enter a value for **Initial value unit**, choose a **Quantity type**, and select an **Initial value unit**.
4. Based on the entered **Initial value** and other options, enter the desired **Set min. value** and **Set max. value**.
5. As needed, choose a **Hud position** and **Initial state** for your text box, or check to make it **Read only**.
6. Click **Create** to add the text box to your extension. During preview or publication of your extension, any values entered outside of the specified range will prompt an error message.

17.2.4. Tool Options Sections





- The **Primary Input** is on the inside of the right side of the HUD.
- The **Options Panel** is on the outside of the right side of the HUD.
- **Expanded** adds another option panel on the outside of the right side of the HUD. This panel allows you to provide options to a user only when they choose to view the **Expanded** options menu by clicking the  button.
- The **Sub Options** are on the inside of the left side of the HUD.

17.2.4.1. Adding Tool Options Sections

1. From the **Extension Tree**, right-click your **Tool** to add sections where options can be added: **Primary Input**, **Options Panel**, **Expanded**, and **Sub Options**.

2. From each of these new sections, you can add a **Text Box**, **Toggle**, **Color Palette**, **Slider**, **Combo Box**, **List Box**, or **Tool Button**. You can write scripts to reference tool options in each section, adding to the flexibility and functionality of your tool design.

17.2.5. Copy Script Reference

Allows you to select a **Tool** within the **Extension Panel** and copy code to reference it. You can then paste the code into the **Script Editor**.

17.2.5.1. Copying a Script Reference

1. Right-click a **Tool** in the **Extension Tree**.
2. Choose **Copy script reference** to copy it to your clipboard.
3. Right-click within the **Script Editor** and paste the code reference into your script.

17.3. API

API (Application Programming Interfaces) in the **Extension Builder** offer developers tools and functions to create custom scripts or automate tasks. They can be used to define methods and properties for interacting with the application's environment, manipulating data, and performing actions. They can enable developers to extend the functionality of their extensions, automate tasks, and build customized solutions.

You can look for existing **APIs** from the **Script Editor** and choose to incorporate them into the script by automatically adding the necessary import code.

Note that there are multiple ways for code to reference elements within the Discovery application, providing developers with a range of coding options. The following example code demonstrates several methods for referencing a **Combo Box** and a **Tool**.

```
ScriptApi.GetTool("myTool").ToolOptions[0]
ScriptApi.GetTool("myTool").ComboBoxes[0]
ScriptApi.GetTool("myTool").GetComboBox("myCombo")

ScriptApi.GetRibbonButton("myButton").Tool.ToolOptions[0]
ScriptApi.GetRibbonButton("myButton").Tool.ComboBoxes[0]
ScriptApi.GetRibbonButton("myButton").Tool.GetComboBox("myCombo")
```

The tool options **Text Box**, **Toggle**, **Color Palette**, **Slider**, **List Box**, **Combo Box**, and **Sub Options** all have .Name, .Visible, and .Enabled properties.

The **Text Box** value can be a double, quantity, or string. It should be noted that the .GetQuantity property was added to facilitate intellisense, but it's not necessary. The **Toggle** control has a .Toggled property that's a boolean representing the toggle's state. The **Slider** control has .MaximumValue and .MinimumValue settings. The **List Box** and **Combo Box** tool options have two properties: .Value and .Items. .Value gets or sets the currently selected value, while .Items gets all the available values.

17.4. Intellisense

The **Script Editor** includes **Intellisense**, which provides intelligent code completion assistance to simplify command entry. This context-aware code completion feature enhances the coding process by minimizing common errors and typos. The code completion tools can help to identify variable names, functions, and methods.

While entering commands in the **Script Editor**, the suggestions for commands, helper functions, and variables will appear. The auto-complete functionality will add parentheses when writing functions in the **Script Editor**, and for-loops can be created automatically.

17.5. Preview

Preview allows you to test how your extension elements and scripts work together within the Discovery application.

17.5.1. Previewing an Extension

1. Click the **Preview Extension** button.
2. (Optional) If you need to make updates or changes to your extension, close the preview by clicking the button again.
3. (Optional) After making the necessary changes, such as updating properties in the **Properties Panel** or modifying a script, you can preview your extension again to see the impact of your modifications.

17.5.2. Error Handling

If there are any errors in your script or unexpected inputs that the script cannot handle, error messages will be displayed to help you identify and resolve the issue. This feedback is particularly useful when you're using the **Preview Extension** feature to test your script before publishing it.

In addition to error messages, you can also include print statements in your code associated with buttons and tools to help with debugging. These statements will print out specific values or messages to the **Console Window** (or in a dialog box) as the script is executed, allowing you to track the progress of your code and identify any issues that arise. By using error messages and print statements together, you can fine-tune your script and ensure that it functions correctly in a variety of scenarios.

17.6. Save and Publish

Saving will allow you to store your project on your computer so you can return to it later and continue editing it.

Publishing will allow you to share your project without allowing others to modify it.

17.6.1. Saving your Extension

When you're finished working on your extension, you can choose to save it by clicking either the **Save** or **Save As** button.

17.6.2. Publishing your Extension

1. Save your extension.

2. Click the **Publish Extension** option in the **Extension Builder Toolbar**. This will create a package containing your project files that others can copy into their install and use but not modify.
3. The published extension will be saved to this directory on your machine: [%ProgramData%]\Discovery\AddIns\Scripted[MyExtension.dstool].
4. To share your project, send the .dstool file to another Discovery user and ask them to put it in the same directory on their machine.
5. Instruct them to restart the application, and the new extension will appear in the tab for your tool.

17.7. Help

Using the **Extension Builder**, you can add tooltips and help content to your extension to provide guidance to other users.

17.7.1. Adding Help

1. To add help, access the **Properties Panel** while creating a tool or button and locate the relevant element.
2. After naming the element, click the **F1** button to add the help content that should appear in the Discovery application when you hover your mouse over the element.
3. (Optional) To access this help content in the application, hover your mouse over the element and click **F1** to launch the help and view the desired information.

17.7.2. Adding Tool Tips

1. While you are creating a button or tool within your extension, locate items that have the option to add tooltips.
2. Enter text into the field with a "tooltip" label, providing the appropriate information for the button or tool.
3. (Optional) You can view your tooltip in the Discovery application when you hover your mouse over the corresponding element in the user interface.

17.8. Examples

The **Extension Builder** can create extensions that can perform various tasks within the Discovery application.

This section contains the following topics:

17.8.1. Example 1: Creating a Box

1. To add a ribbon button and start creating an extension, click **Add button** in the **Extension Builder Toolbar**.
2. In the **Properties** panel, enter your **Tab name**, **Group name** and **Button name**. For the purposes of this exercise, enter "MyTab," "MyGroup," and "MyButton," respectively.
3. (Optional) To add help for your **Tab** name and **Button** name, click **F1**. Additionally, you can enter a tooltip for your button. To access **F1** help, hover over the extension elements in the Discovery application and press **F1**. The **Button** tooltip will appear when you hover over the button in the application. See Help on page 764 for more information.
4. You can either enter the directory path to an icon for your button or click the folder icon to locate and add your icon.
5. Set **Initial state** to "Visible."
6. Check **User input required** so you can add a tool for your ribbon button.

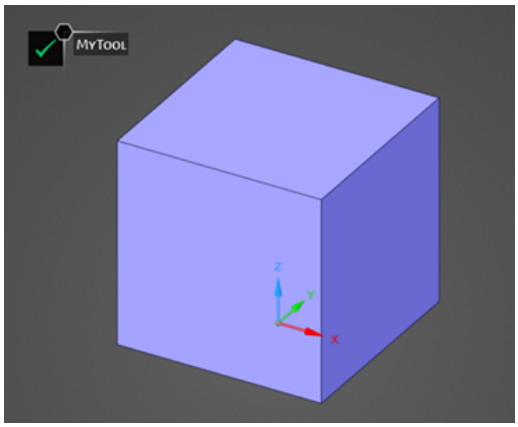
7. For your **Tool** name, enter 'MyTool'.
8. Check **Add complete button** to add a complete button and script for your tool. Optionally, you can check **Close tool on complete** to close your tool after it's executed.
9. In the **Script editor**, enter this script for your complete button. Note that when copying and pasting the code below, it should be on a single line, and any formatting or spacing issues should be removed. Comments should remain on separate lines.

```
# Create Box
result = BlockBody.Create(Point.Create(MM(-10), MM(-10), MM(0)),
Point.Create(MM(10), MM(10), MM(20)), ExtrudeType.ForceAdd)
# EndBlock
```

10. After you have finished writing your script, click **Create** to apply it to your **Complete** button.
11. Click **Preview Extension** to preview your work in the application.
12. After clicking to **Preview Extension**, you should see that "MyTab," "MyGroup," and "MyButton" have been added to the application.

    To launch **MyTool**, click **MyButton**.

13. Click **Complete** to activate your script and create a box.



17.8.2. Example 2: Creating a Box or Sphere

Make sure you have completed the Example 1: Creating a Box on page 764 example before beginning this example. Additionally, remember to exit the preview mode of the previous example by clicking on the **Preview Extension** button before starting this exercise.

1. In the extension tree, right-click **MyTool** and choose "Add combo box." Alternatively, with your tool selected, you can select **Tool Options** from the **Extension Builder Toolbar** and choose "Add combo box."
2. In the **Properties** panel, enter the following: **Combo box name**: "Box or Sphere"; **Combo box values**: "Box;Sphere"; **Hud position**: "Primary input"; **Initial state**: "Visible". Note that the **Combo box values** should be separated with semicolons.
3. Click **Create** to add the combo box to your tool.
4. In the **Extension Tree**, right-click **MyTool** and choose "Add text box." Alternatively, with your tool selected, you can select **Tool Options** from the **Extension Builder Toolbar** and choose "Add text box."
5. In the **Properties** panel, enter the following information: **Text box name**: "Size"; **Text box type**: "Value(double)"; **Initial Value**: "0" (zero); **Hud position**: "Primary input"; and **Initial state**: "Visible".
6. Click **Create** to add the text box to your tool.

7. In the **Script Editor**, click **+** to add a new script. Copy and paste the following code into the new script. Remove any formatting or spacing errors.

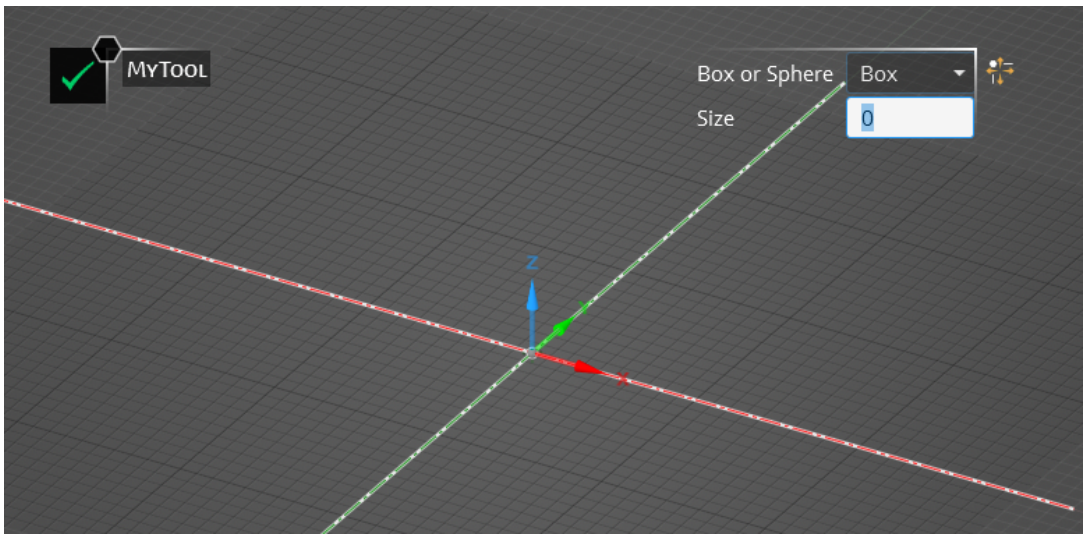
```
# Python Script, API Version = V232
def chooseShape(choice,shapeSize):
if choice == "Box":
BlockBody.Create(Point.Create(MM(-shapeSize), MM(-shapeSize), MM(0)),
Point.Create(MM(shapeSize), MM(shapeSize), MM(2*shapeSize)),
ExtrudeType.ForceAdd)
elif choice =="Sphere":
SphereBody.Create(Point.Create(MM(0), MM(0), MM(0)),
Point.Create(MM(shapeSize),
MM(0), MM(0)), ExtrudeType.None)
```

8. In the **Script Editor**, click **Save As**, name the script "chooseShapeReferenceScript," and choose a location to save your script.
9. To reference the 'chooseShapeReferenceScript' script that you created for this tool, click **Add reference script** in the **Extension Builder Toolbar** located at the upper-left corner, locate the script, and click **Open**.
10. Click "MyTool" in the extension tree.
11. (Optional) In the **Properties Panel**, uncheck **Close tool on complete** to enable your tool to run multiple times. Only do this if you had previously selected this option in **Example 1**.
12. Replace the 'MyTool' script in the **Script editor** with the following. Remove any formatting or spacing errors while copying and pasting the code below.

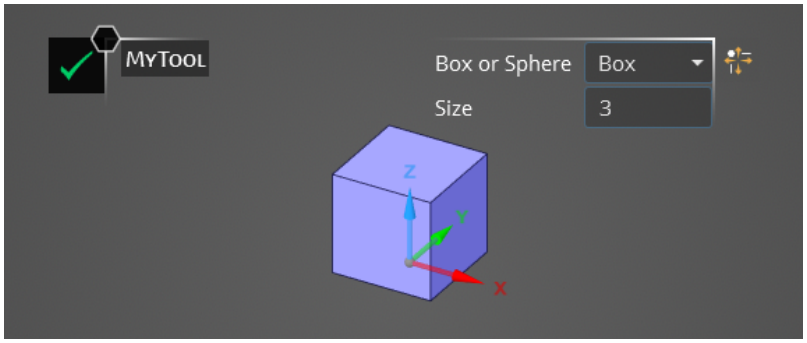
```
state = ScriptApi.GetTool("MyTool").ToolOptions[0].Value
shapeSize = ScriptApi.GetTool("MyTool").ToolOptions[1].Value

chooseShape(state,shapeSize)
```

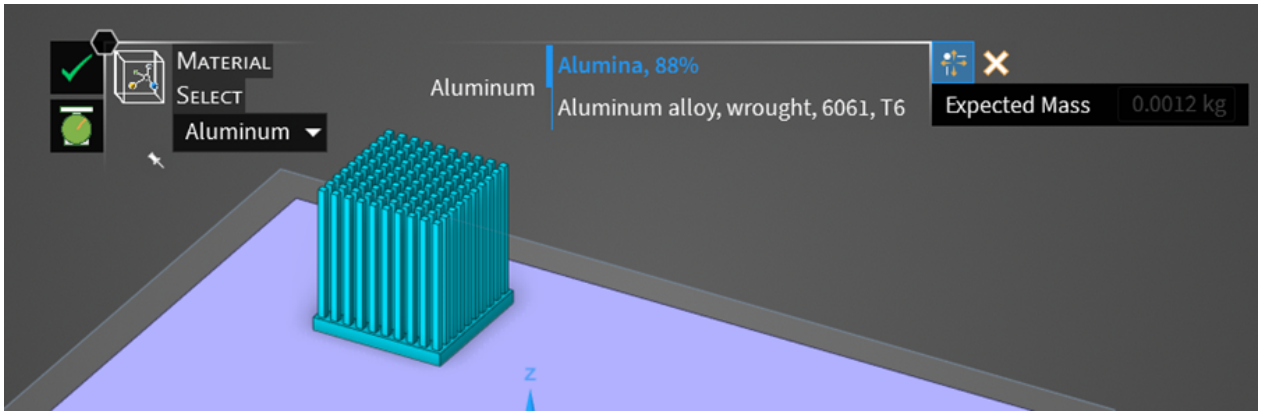
13. Click **Apply**.
14. Click **Preview Extension**.
15. In **Preview Extension**, **MyTool** has been updated with the combo and text boxes.



Choose "Box" with size "3" and click **Complete** .



17.8.3. Example 3: Creating a Heat Sink



The **Heat Sink Example** is accessible for download from the Ansys Discovery Space. Once downloaded, you can open this extension within **Discovery** and use it to gain a comprehensive understanding of the **Extension Builder**'s functionality.

Unlike the first two examples, it is a fully operational extension that helps you design a basic concept heat sink. A good heat sink design is essential for effective heat dissipation in electronic devices and other applications where thermal management is critical.

Once opened in the **Extension Builder**, click **Preview Extension** to launch it within the Discovery application.

This section contains the following topics:

17.8.3.1. Example 3: Working with Foundation

1. The **Foundation** button initiates the creation of a base environment for generating heat sink examples. This button illustrates the execution of a script within the **Extension Builder** to accomplish a specific task and it does not require any user input. Note that once the **Foundation** button has been executed, it cannot be rerun within the current session and is disabled.
2. In the **Extension Panel**, you can click **Foundation** to view the associated script. The **Foundation** script utilizes functions from the **GeomHelper.py** reference script, accessible by double-clicking it at the top of the **Extension Panel**. Additionally, a second reference script, **Calculations.py**, is included to provide calculations used by the extension.

17.8.3.2. Example 3: Working with Heat Sink Options

1. The **Heat Sink Options** button activates the **Heat Sink Options** tool in the Discovery application, enabling you to interact and input your preferences for creating a heat sink.
2. In the **HUD**, the **Heat Sink Options** tool features a **Complete** button, which becomes active as soon as you make changes to the **Width** or **Diameter** text boxes in the **4. Primary Input** section, enabling you to finalize your design.

   > **Note:** You can control this functionality in the code, as the **Complete** button is enabled once this tool is active: **heat_sink_tool.CompleteButton.Enabled = True**.

3. The **Selection Set**, included in the **Sub Options** group and functions like a **Combo Box**, allows you to choose which of the four areas on the foundation you will create and design the heat sink.
4. **Primary Input**

   The **Primary Input** section includes **Width** or **Diameter** and **X-Spacing**, **Y-Spacing**, and **Height** text box inputs. These inputs are used to control the dimensions and spacing for your heat sink. **Width** or **Diameter** are displayed based on your selection of **Square** or **Circle** in the **Fin Shape List Box** tool located in the **4. Options Panel**.

5. **Options Panel**

   The **Options Panel** includes the **Fin Shape** and **Hollow Center** settings, and the **Hollow Diameter** or **Hollow Width** setting. **Fin Shape** is an example of a **List Box** and allows you to choose between square or circle designs. **Hollow Center** is a combo box, allowing you to choose between a square, circle or no design. **Hollow Diameter** and **Hollow Width** are text box examples, allowing you to enter the desired measurement depending on the selection for **Hollow Center**. Note that entering a value of zero for width or diameter will return an error message.

   When you first activate the **Heat Sink Options** tool, you'll notice that the **Hollow Center** option is visible. After selecting **Square** or **Circle** from the combo box, **Hollow Diameter** or **Hollow Width** inputs become available.

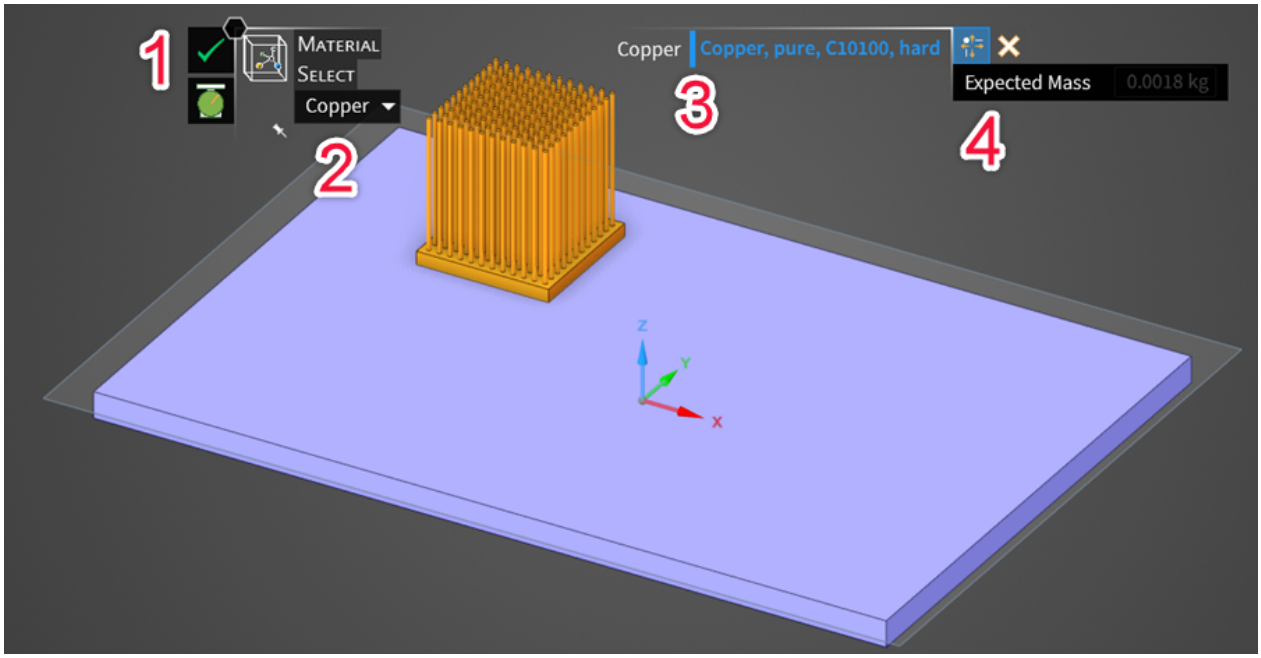
   You can see how this is controlled by looking at the **Initial state** field for the **Hollow Diameter** and **Hollow Width** tool options, and by reviewing the script associated with **Hollow Center**.

6. **Expanded**

   The **Expanded** section is available when pressing the  button. In this section, you can choose the **Fin Color**, as well as choose to use **Fin Taper**. If selected, you can choose the taper percentage. **Fin Taper** is an example of a **Toggle**, **Fin Color** is an example of a **Color Palette**, and **Taper %** is a **Slider**. Note that these have not been implemented in the **Heat Sink Example** extension, but you can add code to give them the desired functionality.

17.8.3.3. Example 3: Working with Materials

1. The **Material** section enables you to modify the materials used in designing your heat sink, illustrating how the tools adapt and change in response to your choices.



2. **Material** tool

   The **Material** tool consists of a **Complete** button to finish the task and a **Volume** button. Clicking the **Volume** button triggers a message that shows the volume of your heat sink.

3. **Sub Options**

   Select lets you pick either **Aluminum** or **Copper** as the material for your heat sink design. This choice automatically updates the **4. Primary Input** display options to align with your selected material.

4. **Primary Input**

   Aluminum and **Copper** are displayed, depending on the option chosen in **Sub Options**.

5. **Options Panel**

   Displays the **Expected Mass** of the heat sink design, which updates based on the chosen materials.

18: Physics Conditions

The Discovery application currently supports the following physics types: Structural, Thermal, Fluids and Electromagnetics.

- A structural simulation that can be used to simulate and evaluate structural displacements, stresses, strains, forces, and fatigue. Structural simulations currently support both steady-state and modal structural simulations.
- A thermal simulation enables you to model steady-state or time-dependent thermal effects in a solid and/or a fluid.
- A fluids simulation can be used to simulate fluid motion and, optionally, heat transfer in and around structures or components.
- An electromagnetics simulation is used to evaluate the S-parameters, gain and directivity of antennas and similar devices when driven by a coaxial terminal or external circuit.
- An electric simulation models the application of an electric potential or electric current.

The application enables you to apply physical conditions to your model (including faceted geometries in **Explore**) and specify them in your simulation. The conditions vary depending on your physics.

In addition, several conditions apply to more than one physics type. The application adds a gravity condition to your analysis by default, which includes the simulation of self weight and/or buoyancy effects, for structural and fluid physics respectively. With the symmetry condition, applicable to fluid, solid-thermal, and structural physics regions, as well as topology optimization simulations, you can enforce a mirrored symmetry on one or more faces in the same plane.

When you set up physical conditions, the application captures the selections to ensure they persist better when you make changes or update your model using the **Update From New CAD File...** option.

- **Physics AutoUpdate** updates the physics tree as you make modifications to the model, keeping the model and physics trees current with each other. To improve performance when making significant model changes, consider pausing the automatic update by selecting the **AutoUpdate** icon at the top of the physics tree and then turning it back on once model changes are complete to re-synch the trees, keeping the model and physics trees current with each other.
- For physics conditions set up using regular selections, or including power selections or patterns, conditions that can be automatically updated and assigned are indicated in the physics tree. Right-click the condition and select **AutoUpdate Assignment** to have the condition update when you update the model.
- For physics conditions set up using Named Selections, the conditions are automatically reassigned when you replace the Named Selections, or when you update the model.

Review the simulation setup to ensure the automatically assigned conditions are appropriate before proceeding.

> **Note:** To edit an existing physics condition from the tree, hover over the condition and click the pencil or double-click the condition. Either action opens a condition-specific HUD and the options displayed are narrowed to those relevant to the condition. This reduced HUD makes complex modifications easier including reassigning locations or changing input values.

This section contains the following topics:

18.1. QuickScope

When setting up a large, highly complex, multiple-body geometry, the **QuickScope** tool is a visual way to pick the bodies to include in the simulation. It provides options to quickly include or exclude all bodies into the simulation, highlight bodies, create named selections of included bodies and to get visual feedback on your progress. Further control of QuickScope behavior is available from the Simulation Options and in the Scoping options in the Physics defaults, available in the Discovery settings.

This section contains the following topics:

18.1.1. Including Bodies with QuickScope

Before using the QuickScope tool, set the tool options, available from the Simulation Options and in the Scoping options in the Physics defaults, available in the Discovery settings.

Then, to pick the bodies to include in the simulation:

1. If necessary, select options from **Include** to specify how QuickScope selection should include or exclude bodies, the visual progress effects, and what to do when finished.

   These selections override the default Simulations Options and are useful if, for example, your organizational process requires specific physics models or convergence criteria to be used for all (or most) simulations, which don't match the Ansys defaults.

2. Expand the **Include** options if you want to **Create named selections for included bodies** or **Hide excluded bodies**.
3. Select bodies in the model to include in the simulation, or click the **Scope** icon next to the body in the model tree.

If you've opted to automatically create a named selection group, the bodies included during the QuickScope session are added to a named selection group and appear in **Advanced Selection** > **Groups** > **Named Selection** list with the **QuickScope Group** default name. Hover over the tree to highlight the objects in the group and click the group to select them. .

18.1.2. QuickScope Tool Guides and Options

QuickScope Tool Guides

The following tool guide is available for QuickScope:

Select multiple bodies	Allows you to select multiple bodies without holding down the Ctrl key.

QuickScope Tool Options

The following tool options are available for QuickScope:

Include	Specifies how QuickScope selection should include or exclude bodies. These selections override the default Simulations Options and are useful if, for example, your organizational process requires specific physics models or convergence criteria to be used for all (or most) simulations, which don't match the Ansys defaults. Available options are: • **All**: Include All bodies in simulation • **None**: Exclude all bodies in simulation • **Custom**: Select when adding bodies from the model or tree. Automatically selected when a subset of bodies is included (All=On) or excluded (None=Off).
Invert	Reverses the selection of included/excluded bodies.
Invert appearance	Reverse the appearance of included and excluded bodies. This selection overrides the default Simulation Option. See previous entry.

Create named selection for included bodies	Creates a named selection for the bodies included in the simulation. This option is available when the options are expanded.
Hide excluded bodies	Hides the bodies excluded from the simulation. This option is available when the options are expanded.

18.2. Physics on Faceted Geometry

Physics conditions can be added to faceted geometries in **Explore**. These geometries may consist of multiple bodies with interfaces and contacts, where some or all bodies are faceted. For example, heat exchanger applications have complex shapes, such as gyroids, and often necessitate the use of faceted geometry.

To add a physics condition to a faceted model, select a set of triangles for the condition location. You can also right-click and choose **Add Facet Region** to specify a faceted region, which creates a virtual face you can then use as the location for physics conditions, monitors, and mesh sizes. In addition, you can perform a volume extract on a faceted body.

Known Limitations

When preparing your model for analysis, the following tools in the **Prepare** tab cannot be used with faceted bodies:

- **Welds**
- Beam **Extract**
- **Midsurface**
- **Bolts**

There are other considerations while setting up a model with faceted geometry:

- **Monitors** on construction geometry are not evaluated for facets.
- For simulations with multiple fluid facet bodies in contact with a solid thermal fact body, create monitors on the solid side rather than the fluid side of the fluid-solid interface.
- For simulations with multiple facet bodies, monitors on facets with overlaps may produce zero values on one side. Resolve by removing overlaps or create a monitor on the other side.
- Mouse-over does not work for cutter bodies.
- Facet-NURBS interfaces on body contained in another need to be on a continuous surface between entrance and exit on face(s) of containing body.
- Adding a new faceted region to a faceted body will result in the simulation being out of date (including adding a new monitor).
- **QuickScope** cannot be used with faceted bodies.
- Faceted regions cannot be generated using scripts.
- The **Internal Flow** tool cannot be used with faceted bodies.
- To apply a **Symmetry** condition, a user-defined faceted region must first be created.
- When using the **Contact Review** tool, contacts between facet bodies with no regions defined will always show the entire body rather then actual contact faces. Excluding these contacts will exclude contacts for the entire body.
- When working with facets in fluid-solid heat transfer simulations, use the **Interface Review** tool to reduce the **Face overlap allowance**. This will yield better results when searching for interfaces and contacts between faceted bodies with no regions defined.
- When using the **Connection Assessment** tool, contacts between facet bodies and facet regions are not supported.
- Faceted bodies cannot be selected for topology optimization.

18.3. General Physics Conditions

Conditions that apply to Structural, Thermal and Fluids include:

- Gravity, which includes the simulation of self weight and/or buoyancy effects for structural and fluid physics respectively.
- Symmetry, which enforces a mirrored symmetry on one or more faces in the same plane.

This section contains the following topics:

18.3.1. Gravity

By default, the Discovery application adds a gravity condition to your analysis to simulate self weight, added mass, and/or buoyancy effects, for structural and fluid physics respectively. If neither of these are selected, then the gravity specification will have no effect on your simulation. You can review the **Gravity** settings in the **Physics** tree or double-click it to access the full settings.

When enabled, self weight is simulated in the specified direction of gravity. Including self weight depends on the density of the material. If the self weight is negligible in comparison with other forces, you can turn self weight off by clearing the selection of **Include self weight**. Turning on self weight also means that the gravitational acceleration is used in combination with any added mass inputs.

When enabled, buoyancy is simulated in a fluid flow analysis. For fluids, buoyant flow is the most important effect of gravity. Because the force of gravity is proportional to fluid density, buoyant flow can arise whenever the fluid density varies due to the temperature or material specification. The addition of buoyancy simulates natural or free convection, where the fluid density varies in temperature.

For many applications involving buoyancy, it is sufficient to assume a constant fluid density when the change in density over the expected range of conditions is relatively small. This is often true for many liquids. When the fluid density is not a function of pressure or temperature, the Boussinesq model is applied.

The Boussinesq model uses a constant density fluid model, but applies a local gravitational body force throughout the fluid that is a linear function of fluid thermal expansivity, and the local temperature difference with reference to the **Fluid Initial Temperature**.

In general, it is good practice to specify the initial temperature as an approximate average value of the expected domain temperature for steady-state problems.

If the effects of buoyancy are not important in your simulation (for example, where the velocity of the fluid is such that buoyancy has little impact), you can turn it off by clearing the selection of **Include buoyancy**.

18.3.2. Symmetry

With the symmetry condition, you can enforce a mirrored symmetry on one or more faces in the same plane.

- For fluid physics regions, the symmetry condition creates a free slip, insulated wall to mirror the expected pattern of the flow on either side of it.
- For solid-thermal physics regions, the symmetry condition creates an insulated wall to mirror the expected pattern of the thermal solution on either side of it.
- For a structural physics region, the symmetry condition creates a support normal to the face.
- For a topology optimization simulation, the symmetry condition ensures that the model is dimensionally reduced based on a mirror plane. Note that the protected distance on the symmetric surface is set to zero.

Symmetry is currently only applicable to flat faces. You can apply symmetry to multiple faces on the same plane, even if the selected faces are on bodies with different physics applied.

18.4. Structural Conditions

Loads and constraints for structural physics include:

- Bolt preload
- Zero and nonzero displacements
- Externally applied forces, pressures, masses, and moments
- Supports
- Connections such as contacts, joints, and springs

This section contains the following topics:

18.4.1. Support

A support enables you to prevent translational or rotational movement for the selected part of the model. Unless your forces are perfectly in balance, you'll need to provide supports. A fixed support is often used, as it prevents both translational and rotational movement; however, a number of other support types are available.

This section contains the following topics:

18.4.1.1. Setting a Support

To set up a support:

Select the faces to set as supports.

1. Select **Support** from the dropdown menu of the **Structural** tool in the **Physics** group of the **Simulation** tab, or by selecting the **Support** tool in the **Structural** condition HUD.
2. Select the faces to set as supports.

   If your model contains Named Selections, you can click **Advanced filters**, select **Link named selections**, and choose one from the list.

3. Under **Structural Support**, select the most appropriate description for your support.

   Unless your forces are perfectly in balance, you'll need to provide supports. A fixed support is often used, as it prevents both translational and rotational movement, however, a number of other support types are available.

   - **Fixed**: Movement is prevented in any direction, including rotations.
   - **Sliding**: Rotation is prevented except for normal to the planar surfaces where it is free to rotate about the normal to the plane.
   - **Hinged**: Only applicable to cylindrical surfaces. With this support type, the support is free to rotate about that axis, which is defined as along the line of the cylinder.
   - **Ball**: Only applicable to spherical surfaces. Prevents translational degrees of freedom.
   - **Displaced**: Allows you to set the possible directions for moving and rotating. For each axis, you can specify whether it's fixed or free to move or rotate.

4. For the Displaced support type:
   a) Enter a displacement value.
   b) Define the rotation and/or translation of the support. Specify 0 to prevent any motion in a given direction. The available directions will depend on the support type.
5. Click **Accept** to complete the setup.

18.4.1.2. Support Tool Guides and Options

Support Tool Guides

The following tool guides are available:

Direction tool	The **Direction tool** defines the direction of the applied condition. When this tool is active, select an entity (a geometry or direction in the local coordinate system) to align the direction with that entity.

Support Types

The following support types are available:

Fixed	Movement is prevented in any direction, including rotations.
Sliding	Rotation is prevented except for normal to the planar surfaces where it is free to rotate about the normal to the plane.
Hinged	Only applicable to cylindrical surfaces. With this support type, the support is free to rotate about that axis, which is defined as along the line of the cylinder.
Ball	Only applicable to spherical surfaces. Prevents translational degrees of freedom.
Displaced	Allows you to set the possible directions for moving and rotating. For each axis, you can specify whether it's fixed or free to move or rotate.

Support Tool Options

The following options are available for the Displaced support type:

Displacement	Enter a displacement value.
dx, dy, dz	Define the rotation of the support. Specify 0 to prevent any motion in a given direction. The available directions will depend on the support type.
0x, 0y, 0z	Define the translation of the support. Specify 0 to prevent any motion in a given direction. The available directions will depend on the support type.

18.4.2. Force

When you apply a force load, the Discovery application determines a normal direction based on the selected location, but you can also use the Direction tool to set the force direction. By default, the force is distributed across the selected geometry. You can decide if the force varies depending on the area of a surface to which it is applied.

Alternatively, you can specify a Torque force and then enter the rotational forces.

You can also apply a distributed load in a specific direction and apply the force load from a remote point.

> **Note:** The application uses a flexible stiffness behavior for remote points.

This section contains the following topics:

18.4.2.1. Specifying a Distributed Force Load

By default, a force is distributed across the selected geometry. You can decide if the force varies depending on the area of a surface to which it is applied. To specify a distributed force:

1. Select **Force** from the dropdown menu of the **Structural** tool in the **Physics** group of the **Simulation** tab, or by selecting the **Force** tool in the **Structural** condition HUD.
2. Select a face.

   If your model contains Named Selections, you can click **Advanced filters**, select **Link named selections**, and choose one from the list.

3. Leave the default option of **Distributed**. This distributes the force across the selected geometry.
4. Specify if the force is defined as a total or by unit area.

   > **Note:** Select **Force per unit area** if the force is dependent on the geometry, for example, a load due to snow or wind.

5. Specify the force magnitude and press Enter.

   You can also select **Use force components** to specify the X, Y, Z components of the force.

6. The Discovery application determines a normal direction based on the selected location. Distributed forces will be applied in that direction.

   You can also define the direction. Use the **Direction tool** or Alt-click to select an entity (a geometry or direction in the local coordinate system) and align the direction with that entity.

7. Select **Remote point** if you want to apply the force from an arbitrary point in space.

   The application automatically chooses a point in space, shown by an arrow, but you can also enter the coordinates.

   Or you can define the point's origin by clicking **Set remote point location** and selecting a geometry. If you select an edge, face or body, the remote point is located at the centroid of the selection.

18.4.2.2. Specifying a Torque Force Load

You can specify a Torque force, or a rotational force. To specify a torque force:

1. Select **Force** from the dropdown menu of the **Structural** tool in the **Physics** group of the **Simulation** tab, or by selecting the **Force** tool in the **Structural** condition HUD.
2. Select a face to apply a force.

   If your model contains Named Selections, you can click **Advanced filters**, select **Link named selections**, and choose one from the list.

3. Select **Torque** to define a Torque force.
4. Specify the torque magnitude and press Enter.

   You can also select **Use force components** to specify the X, Y, Z components of the force.

5. The Discovery application determines a normal direction based on the selected location. Torque forces will be applied about that direction.

You can also define the direction. Use the **Direction tool** or Alt-click to select an entity (a geometry or direction in the local coordinate system) and align the direction with that entity.

18.4.2.3. Force Tool Guides and Options

Force Tool Guides

The following tool guides are available.

Direction tool	The **Direction tool** defines the direction of the applied condition. When this tool is active, select an entity (a geometry or direction in the local coordinate system) to align the direction with that entity.
Set remote point location	This tool enables you to define the remote point location by selecting a geometry. If you select an edge, face or body, the remote point is located at the centroid of the selection.

Force Types

The following Force types are available:

Distributed	Distributes the force across the selected geometry. You can decide in the Tool Options if the force varies depending on the area of a surface to which it is applied.
Torque	Specifies a force that causes an object to rotate.

Force Tool Options

The following options are available for the Distributed force type:

Force	Enter the force magnitude.
Total force	Select to define the force as a total force.
Force per unit area	Select to define the force per unit area. This option is useful if the force is dependent on the geometry. For example, a load due to snow or wind.
Use force components	Select to specify the force as X, Y, Z components.
Use remote point	Select if you want to apply the force from an arbitrary point in space. the Discovery application automatically chooses a point in space, shown by an arrow, but you can also enter the coordinates. You can also define the point's origin using **Set remote point location**.

The following options are available for the Torque force type:

Force	Enter the force magnitude.
Use force components	Select to specify the force as X, Y, Z components.

18.4.3. Pressure

A Pressure load applies a constant normal pressure on one or more flat or curved faces. A positive value for pressure acts into the face, tending to compress the solid body.

Pressure is a scalar load and can only be applied normal to a face. If you require a distributed pressure-like load to be applied in other directions, for example, to model snow load on a pitched roof, you should use Force per unit area.

18.4.3.1. Specifying a Pressure Load

To specify a pressure load:

1. Select **Pressure** from the dropdown menu of the **Structural** tool in the **Physics** group of the **Simulation** tab, or by selecting the **Pressure** tool in the **Structural** condition HUD.
2. Select one or more faces to apply a pressure.

   If your model contains Named Selections, you can click **Advanced filters**, select **Link named selections**, and choose one from the list.

3. Specify the constant pressure magnitude and press Enter. The force will be applied normal to the surface, including for curved faces.

   To apply a pressure in a specific direction, use Force per unit area as an alternative.

18.4.4. Displacement

A displacement constraint specifies that a portion of the model displaces relative to its original location. Displacement is defined in terms of translation, the difference between the initial and final positions along the straight line of a specified axis; and/or rotation, the distance rotated around a circle based on the specified axis. You can also apply displacement from a remote point.

> **Note:** Stiffness behavior for remote displacements is handled differently in Explore and Refine modes. In the Explore stage, the Discovery application uses a rigid behavior. However, the application uses a flexible behavior in the Refine stage.

This section contains the following topics:

18.4.4.1. Specifying a Displacement Constraint

To displace a portion of the model relative to its original location:

1. Select **Displacement** from the dropdown menu of the **Structural** tool in the **Physics** group of the **Simulation** tab, or by selecting the **Displacement** tool in the **Structural** condition HUD.
2. Select the faces to displace.

   If your model contains Named Selections, you can click **Advanced filters**, select **Link named selections**, and choose one from the list.

3. Select if the prescribed displacement is to be only in **Translational** direction, **Rotational** direction, or a **Combined** rotation and translation.
4. Specify the **Translation** distance and/or **Rotation** that the location is to be moved by and press Enter.

Optionally, specify components to keep specific directions as free to move, or if fixed, specify how far they are to be moved.

After being moved, the location will be fixed in the new position. If you enter a zero value, then it will act like a support.

The values entered are in a local coordinate system, determined from the location selection, with *x* normal to a plane, or along a cylinder, with the centroid located at the center.

5. Select **Remote point** if you want to apply the displacement from an arbitrary point in space.

   The application automatically chooses a point in space, shown by an arrow, but you can also enter the coordinates.

   Or you can define the point's origin by clicking **Set remote point location** and selecting a geometry. If you select an edge, face or body, the remote point is located at the centroid of the selection.

6. The Discovery application determines a normal direction based on the selected locations.

   You can also define the direction. Use the **Direction tool** or Alt-click to select an entity (a geometry or direction in the local coordinate system) and align the direction with that entity.

18.4.4.2. Displacement Tool Guides and Options

Displacement Tool Guides

The following tool guides are available:

Direction tool	The **Direction tool** defines the direction of the applied condition. When this tool is active, select an entity (a geometry or direction in the local coordinate system) to align the direction with that entity.
Set remote point location	This tool enables you to define the remote point location by selecting a geometry. If you select an edge, face or body, the remote point is located at the centroid of the selection.

Displacement Types

The following displacement types are available:

Translation	Displaces the selected faces a specified distance normal to a plane.
Rotation	Rotates the selected faces a specified distance along a cylinder.
Combined	Displaces the selected faces normal to a plane and then rotates them.

Displacement Tool Options

The following options are available:

Translation	Enter the distance the selected face(s) is to be moved by. Available for the Translation and Combined displacement types.
Rotation	Enter the rotated distance the selected face(s) are to be moved by. Available for the Rotation and Combined displacement types.

dx, dy, dz	Specify the displacement translation as X, Y, Z components. You can keep specific directions as free to move, or if fixed, specify how far they are to be moved. Available for the Translation and Combined displacement types.
0x, 0y, 0z	Specify the displacement rotation as X, Y, Z components. You can keep specific directions as free to move, or if fixed, specify how far they are to be moved. Available for the Rotation and Combined displacement types.
Use remote point	Select if you want to apply the displacement from an arbitrary point in space. the Discovery application automatically chooses a point in space, shown by an arrow, but you can also enter the coordinates as X, Y, and Z values. You can also define the point's origin using **Set remote point location**

18.4.5. Moment

This vector-based load distributes a moment "about" (the vector of) an axis across one or more flat or curved faces, or about one or more edges or vertices. Use the right-hand rule to determine the sense of the moment.

You can apply the moment load by defining the components of that vector in either a direction relative to the selected geometry, or to the model as a whole. When defining by the local coordinate system, the z direction is normal to a planar face or the axis of a cylindrical face.

This section contains the following topics:

18.4.5.1. Specifying a Moment

To specify a moment:

1. Select **Moment** from the dropdown menu of the **Structural** tool in the **Physics** group of the **Simulation** tab, or by selecting the **Moment** tool in the **Structural** condition HUD.
2. Select one or more faces.

   If your model contains Named Selections, you can click **Advanced filters**, select **Link named selections**, and choose one from the list.
3. Select the coordinate system you want to specify the moment components in, either **Local** or **Global**.
4. Specify the applied moment components in the given coordinate system and then press Enter.
5. Align the direction of the moment with any entity (a geometry or direction in the local coordinate system) by clicking the **Direction tool** and then selecting the entity. The origin and orientation of a local coordinate system can be defined in the Design tab.

18.4.5.2. Moment Tool Guides and Options

Moment Tool Guides

The following tool guides are available:

Direction tool	The **Direction tool** defines the direction of the applied condition. When this tool is active, select an entity (a geometry or direction in the local coordinate system) to align the direction with that entity.

Moment Tool Options

The following options are available:

Mx, My, Mz	Specify the applied moment components in the given coordinate system.
Global coordinate system	Select to define the moment components in the global coordinate system.
Local coordinate system	Select to define the moment components in the local coordinate system.

18.4.6. Mass

You can idealize the inertial effects from a body using the **Mass** condition. Applications include applying a force with an acceleration or any other inertial load, or adding inertial mass to a structure, which affects modal solutions.

You can specify a mass on a geometry or using a remote point. When applied to geometry, the mass constraint uses the centroid of the selected geometry as the origin.

> **Note:** The Discovery application uses a flexible stiffness behavior for remote points.

This section contains the following topics:

18.4.6.1. Specifying a Mass

To specify a mass:

1. Select **Mass** from the dropdown menu of the **Structural** tool in the **Physics** group of the **Simulation** tab, or by selecting the **Mass** tool in the **Structural** condition HUD.
2. Select the geometry to define the location; the mass uses the centroid of the selected geometry as the origin.

   If your model contains Named Selections, you can click **Advanced filters**, select **Link named selections**, and choose one from the list.
3. To define a mass as originating on an arbitrary point in space, select **Use remote point**.

   The application automatically chooses a point in space, shown by an arrow, but you can also enter the coordinates.

   Or you can define the point's origin by clicking **Set remote point location** and selecting a geometry. If you select an edge, face or body, the remote point is located at the centroid of the selection.
4. Specify the **Mass** value and press Enter.

18.4.6.2. Mass Tool Guides and Options

Mass Tool Guides

The following tool guides are available:

Set remote point location	This tool enables you to define the remote point location by selecting a geometry. If you select an edge, face or body, the remote point is located at the centroid of the selection.

Mass Tool Options

The following options are available:

Mass	Specify the Mass value.
Use remote point	Select if you want to apply the mass from an arbitrary point in space. the Discovery application automatically chooses a point in space, shown by an arrow, but you can also enter the coordinates as X, Y, and Z values. You can also define the point's origin using **Set remote point location**.

18.4.7. Bearing Load

The Bearing Load condition simulates a load applied to bearings in either the radial or axial (thrust) direction, or both. Apply bearing load to a complete cylinder defined in a single face.

The radial component of the bearing load is applied as non-uniform distribution of force across the bearing surface and the axial (thrust) component is applied as a uniform distribution of force.

> **Note:** When analyzing more than one cylinder, be sure that you assign each cylinder its own bearing load condition. Assigning a single bearing load to multiple cylinders divides the load among the multiple cylindrical faces by area ratio. For the single bearing load applied to the two cylinders, the reactions are proportional to each cylinder's area as a fraction of the total load area.

> **Note:** The 0 degree angle for the radial component of a bearing load is determined by the projection of the gravity direction onto the plane with the cylinder axis as its normal. If gravity is not selected, an arbitrary -z direction is used as a nominal gravity direction. When the cylinder axis is parallel to the z axis, no projection can be found. In this case, a fall-back -y direction is used instead.

This section contains the following topics:

18.4.7.1. Specifying a Bearing Load

To set up a bearing load:

1. Select **Bearing load** from the dropdown menu of the **Structural** tool in the **Physics** group of the **Simulation** tab, or by selecting the **Bearing load** tool in the **Structural** condition HUD.
2. Select a cylindrical face on which to apply the bearing load.

   If your model contains Named Selections, you can click **Advanced filters**, select **Link named selections**, and choose one from the list.

   > **Note:** When analyzing more than one cylinder, be sure that you assign each cylinder its own bearing load condition.

3. For **Radial force**, specify the value for the load acting radially, if applicable.

   Radial (*R*) loads act at right angles to the bearing's axis of rotation. Its direction defaults to that of gravity.

4. For **Thrust force**, specify the value for the load acting axially, if applicable.
   The Axial or thrust (*T*) loads act parallel to the axis of rotation.
5. Optionally, specify an **Angle** value if you want to alter the direction the radial load is applied and press Enter.

18.4.7.2. Bearing Load Tool Options

The following options are available:

Radial force	Specify the value for the load acting radially. Radial (*R*) loads act at right angles to the bearing's axis of rotation. Its direction defaults to that of gravity.
Thrust force	Specify the value for the load acting axially. The Axial or thrust (*T*) loads act parallel to the axis of rotation.
Angle	Specify an angle value if you want to alter the direction that the radial load is applied.

18.4.8. Bolt Preload

The Bolt Preload condition in the Refine stage applies a pretension load to a cylindrical face -- typically to model a bolt under pretension. For example, you can use the bolt pretension to model a bolt which is holding together two bodies by applying a known value of clamp force, grip length adjustment, or torque (to simulate the tightening of a nut or bolt). In addition, in both Explore and Refine stages, you can assign a bolt preload to a bolt assignment. Bolt assignments are currently not supported for Named Selections or faceted bodies.

> **Note:** Prior to defining a Bolt Preload in the Explore stage, ensure you have defined your bolt assignments in the Prepare tab. For more information on assigning bolts, see Bolts.

When a Bolt Preload is applied to a cylinder face, the cylinder axis becomes the bolt axis direction. Any bodies that the bolt passes through will be bonded to it by default. If you wish to exclude them from being bonded, then select the faces of the body that the bolt passes through with the secondary selection.

> **Note:** Diameter on Bolt Preload is ignored in the Refine stage when the preload is assigned to an idealized bolt. In addition, Bolt Preload parameterization is not supported. Bolt Preloads are ignored during Topology Optimization.

If you have multiple bolts with the same load value and axial direction, the Discovery application applies the same preload value to all bolts selected.

> **Note:** When a bolt pretension load is applied to a model, the application automatically performs a two-step analysis to simulate bolt pretension and locking. In the first simulation step the bolt pretension load is applied. In the second simulation step the shortened grip length of the bolt is locked and the other structural loads are applied.

This section contains the following topics:

18.4.8.1. Specifying a Bolt Preload in Explore

Prior to defining a Bolt Preload in the Explore stage, ensure you have defined your bolt assignments in the Prepare tab.

To specify a bolt preload on a bolt in the Explore stage:

1. Select **Bolt preload** from the dropdown menu of the **Structural** tool in the **Physics** group of the **Simulation** tab, or by selecting the **Bolt preload** tool in the **Structural** condition HUD.
2. Select one or more bolt axes, defined in a bolt assignment, to apply a bolt preload.

> **Note:** Named selections do not currently support bolt assignments.

3. Select an option for defining the bolt:

Option	Description
Clamp force	Specify a known value of axial load to apply to the length of the bolt.
Grip length adjustment	Specify a known value of adjustment to apply to the length of the bolt.
Torque	**a.** Specify the torque to which the bolt will be tightened. **b.** You can optionally change the default **Torque Coefficient** value, which varies based on the material. **c.** Bolt diameter is detected automatically, but you can select **Specify bolt diameter** if you want to specify a **Shank diameter** value. If you do not specify the diameter, the Discovery application uses the geometric diameter (determined per-bolt location).

> **Note:** If you have multiple bolts with the same load value and axial direction, the same preload value is applied to all bolts selected.

18.4.8.2. Specifying a Bolt Preload in Refine

To specify a bolt preload on a bolt in the Refine stage:

1. Select **Bolt preload** from the dropdown menu of the **Structural** tool in the **Physics** group of the **Simulation** tab, or by selecting the **Bolt preload** tool in the **Structural** condition HUD.
2. Select one or more cylindrical faces to apply a bolt preload. You can also select one or more bolt axes, which were defined in a bolt assignment in the Prepare tab.

   Any bodies that the bolt passes through will be bonded to it by default. If you wish to exclude them from being bonded, then select the faces of the body that the bolt passes through with the secondary selection.

   If your model contains Named Selections, you can click **Advanced filters**, select **Link named selections**, and choose one from the list.

> **Note:** Named selections do not currently support bolt assignments.

3. Select an option for defining the bolt:

Option	Description
Clamp force	Specify a known value of axial load to apply to the length of the bolt.
Grip length adjustment	Specify a known value of adjustment to apply to the length of the bolt.
Torque	**a.** Specify the torque to which the bolt will be tightened. **b.** You can optionally change the default **Torque Coefficient** value, which varies based on the material. **c.** Bolt diameter is detected automatically, but you can select **Specify bolt diameter** if you want to specify a **Shank diameter** value. If you do not specify the diameter, the Discovery application uses the geometric diameter (determined per-bolt location).

Note: If you have multiple bolts with the same load value and axial direction, the same preload value is applied to all bolts selected.

18.4.8.3. Bolt Preload Tool Options

The following options are available:

Clamp force	Specify a known value of axial load to apply to the length of the bolt.
Grip length adjustment	Specify a known value of adjustment to apply to the length of the bolt.
Torque	Specify the torque to which the bolt will be tightened.
Torque coefficient, K	Optionally, change the torque coefficient, which varies based on the material.
Specify bolt diameter	Bolt diameter is detected automatically, but you can select this option if you want to specify a value. If you do not specify the diameter, the Discovery application uses the geometric diameter (determined per-bolt location).
Shank diameter	Specify the bolt diameter.

18.4.9. Acceleration

An acceleration constraint defines a linear acceleration of a portion of a model in each of the global Cartesian axis directions. In essence, loads (forces) are developed in the bodies in the direction of the specified accelerations. Currently translational acceleration is supported.

This section contains the following topics:

18.4.9.1. Setting an Acceleration Constraint

To specify an acceleration constraint:

1. Select **Acceleration** from the dropdown menu of the **Structural** tool in the **Physics** group of the **Simulation** tab, or by selecting the **Acceleration** tool in the **Structural** condition HUD.
2. Select one or more bodies.

If your model contains Named Selections, you can click **Advanced filters**, select **Link named selections**, and choose one from the list.

3. Define the direction for the acceleration. Use the **Direction tool** or Alt-click to select a face or edge.
4. Specify at which **Acceleration** the selected bodies are to be moved by and press Enter. The values entered are in a local coordinate system, determined from the location selection, with x normal to a plane, or along a cylinder, with the centroid located at the center.

   Optionally, keep specific directions as free to move, or if fixed, specify at which acceleration they are to be moved.

18.4.9.2. Acceleration Tool Guides and Options

Acceleration Tool Guides

The following tool guides are available:

Direction tool	The **Direction tool** defines the direction of the applied condition. When this tool is active, select an entity (a geometry or direction in the local coordinate system) to align the direction with that entity.

Acceleration Tool Options

The following options are available:

Acceleration	Specify at which acceleration the selected bodies are to be moved by.
X, Y, Z	Specify acceleration values in a local coordinate system, determined from the location selection, with x normal to a plane, or along a cylinder, with the centroid located at the center. You can keep specific directions as free to move, or if fixed, specify at which acceleration they are to be moved.

18.4.10. Velocity

A velocity constraint specifies that a portion of the model rotates with a specified velocity relative to its original position. You define a velocity in terms of the number of turns/revolutions of rotation per minute around a specified axis.

> **Note:** If a rotational velocity is applied to a cyclically symmetric structure, unsymmetric deformation may result in the Explore stage from the discretization of the model. Use a local fidelity or increase the fidelity slider to improve your results.

This section contains the following topics:

18.4.10.1. Specifying a Velocity Constraint

To set a velocity:

1. Select **Velocity** from the dropdown menu of the **Structural** tool in the **Physics** group of the **Simulation** tab, or by selecting the **Velocity** tool in the **Structural** condition HUD.

2. Select one or more bodies.

   If your model contains Named Selections, you can click **Advanced filters**, select **Link named selections**, and choose one from the list.

3. Set the cylindrical frame that defines the origin of the rotation. Use the **Direction tool** or Alt-click to select a face or edge. If you do not have geometry to set the origin of the rotational axis, select **Override rotation axis origin** and define the origin.
4. Specify the rotational **Velocity** at which the selected bodies are to be rotated and them press Enter. The values entered are in a local coordinate system, determined from the location selection, with x normal to a plane, or along a cylinder, with the centroid located at the center.

   Optionally, keep specific directions as free to move, or if the velocity is fixed, specify how quickly they are to be moved.

18.4.10.2. Velocity Tool Guides and Options

Velocity Tool Guides

The following tool guides are available:

Direction tool	The **Direction tool** set the cylindrical frame that defines the origin of the rotation. When this tool is active, select a face or edge to align the direction with that entity.

Velocity Tool Options

The following options are available:

Velocity	Specify the rotational velocity at which the selected bodies are to be rotated..
0X, 0Y, 0Z	Specify rotational velocity values in a local coordinate system, determined from the location selection, with x normal to a plane, or along a cylinder, with the centroid located at the center. You can keep specific directions as free to move, or if fixed, specify how quickly they are to be moved.
Override rotation axis origin	Select if you do not have geometry to set the origin for the rotational axis, then define the specific directions for the axis.
X, Y, Z	Specify the directions for the origin of the rotational axis.

18.5. Structural Connections

The Discovery application supports a number of connection types which can be added as structural conditions, including contacts, joints, and springs.

In addition to the ability to manually add a contact, the application automatically detects and defines bonded contacts when you specify a Structural simulation. These bonded contacts can be reviewed in the **Bonded Contacts (default)** review tool.

In addition, once you've run a simulation, you can run the Connection Assessment Tool. The Connection Assessment Tool cycles through the connections and shows whether they pass or fail the factor of safety for the connection based on specified shear and tensile limit values, or whether they’re borderline.

This section contains the following topics:

18.5.1. Contacts

The Contact condition enables you to manually define one or more contacts, a type of connection between surface bodies.

By default, the Discovery application automatically generates contacts and sets them to bonded. You can review these generated contact pairs in the Bonded Contacts (default) review tool. From there, you can change the contact behavior or convert to a joint or sliding contact.

Guidelines

For both manually and automatically defined contacts, the application assumes "assembly contact", a small sliding contact solution where the surfaces in contact are near each another and the amount of physical sliding is small. The application supports assembly contact applications with load transfer across parts, such as bolted connections or interference fits. However, large sliding contact applications, such as snap-fit insertions or compression of O-ring seals, are not supported.

For automatically generated bonded contacts in either the Explore or Refine stage, the contacts are bonded if the contact surfaces are separated by a distance less than or equal to the **Maximum detection distance** shown in the Bonded Contact Review tool. This method ensures that any automatically detected bonded contacts are bonded during the solution.

Specifying either frictionless or frictional contact results in a nonlinear solution, which results in longer solution times and a possibility that the solution will not converge. A common reason for nonconvergence is rigid body motion. If the solution fails to converge, running a Natural Frequency simulation of your setup may help determine if the contacts and supports are set up correctly. In the Refine stage setting the **Adjust gap or overlap** for frictionless or frictional contacts to **Make just touching** can be useful for overcoming nonconvergence due to rigid body motion.

Contact stress results on curved surfaces with frictionless contact may be inaccurate in the Explore stage depending on the geometry and solution fidelity. Use the Refine stage for improved stress results.

For kinematic assembly conditions, you can reduce solution time by using hinge or a spherical joints rather than frictionless or frictional contact.

18.5.1.1. Defining Contacts

To manually define contacts:

1. Select **Contact** from the dropdown menu of the **Structural** tool in the **Physics** group of the **Simulation** tab, or by selecting the **Contact** tool in the **Structural** condition HUD.
2. Select your contact type:
   - Select **Bonded** to specify that no sliding or separation between faces or edges is allowed. Think of the region as glued. This type of contact allows for a linear solution since the contact length/area will not change during the application of the load. Any gaps will be closed and any initial penetration will be ignored.
   - Select **Sliding** to define contact where the surfaces can freely slide in the tangential direction relative to each other.
   - Select **Excluded** if the bodies are to ignore each other.
3. For bonded contacts in a structural simulation, select **Strength limits** to specify the shear and tensile stress limits of the bond during service conditions if the contacts are not perfectly bonded.

   After solving, you can assess your contacts in the Connection Assessment Tool. This tool cycles through the connections and shows whether they pass or fail the factor of safety based on these specified shear and tensile limit values.

4. For bonded contacts in a structural-thermal simulation, under **Thermal Conductance Definition**, select a method for defining conductance.

   - Select **Perfect conductor** to transfer heat between the two surfaces without any resistance.
   - Select **Perfect insulator** to define near zero conduction of heat transfer between two surfaces.
   - Select **Specified conductance** to specify the conductive heat transfer as a value.
   - Select **Thickness and material** where the Discovery application then uses them to compute the conductance.
   - Select **Thickness and conductivity** where the Discovery application then uses them to compute the conductance.

   If contact occurs, a small value of thermal contact conductance yields a measured amount of imperfect contact and a temperature discontinuity across the contact surfaces. A small thermal contact conductance can also be used to represent a thin layer of material with a different conductivity, such as a gasket.

   For large values of thermal contact conductance, the resulting temperature discontinuity tends to vanish and perfect thermal contact is approached. With a zero value, however, the solver assumes that no heat is transferred across the contacting surfaces.

   > **Note:** While the application can solve with a value of 0, which specifies no heat transfer between the faces, a project with a 0 value for thermal conductance is invalid when exported to the Workbench application. You must then replace zero conductance with some small value.

5. For sliding contacts, you can further refine your contact behavior by specifying the following:

Idealized	Allows frictionless sliding but no separation, which enforces linear behavior.
Frictional	Allows for sliding with separation. You can then enter a **Friction coefficient**. **Note:** Friction coefficient values are ignored in the Explore stage, but separation is allowed.
Friction coefficient	The coefficient of friction. Entering zero results in frictionless sliding.
Adjust gap or overlap	Defines how the contact interface is treated with **Adjust gap or overlap**. Select **Make just touching** to direct that any initial gaps are closed and any initial penetration is ignored, creating an initial stress-free state. Or you can model true contact gap/penetration plus the offset value you specify with **Define specific offset**.
Stiffness factor	Modifies the default factor for **Stiffness factor** to control the amount of penetration between contact and target surfaces.
Detection radius factor	Modifies the default factor for **Detection radius factor** to specify the radius of contact detection to be used during solve in order to obtain a good convergence.

When you select one or more surface bodies for the contact condition, any contact pairs that use that surface body will use the newly defined contact type rather than the default contact type automatically applied.

18.5.1.2. Contact Tool Guides and Options

Contact Tool Guides

The following tool guides are available:

Select secondary surface	When you select one or more faces for a contact, any additional faces that contact those selected faces will also be defined as the contact type you specify. If you want to narrow it down to a specific pairing of faces, select the **Select secondary surface** tool before selecting the second face. You can also use Alt-click.

Contact Types

The following contact types are available:

Bonded	Select **Bonded** to specify that no sliding or separation between faces or edges is allowed. Think of the region as glued. This type of contact allows for a linear solution since the contact length/area will not change during the application of the load. Any gaps will be closed and any initial penetration will be ignored.
Sliding	Select **Sliding** to define contact where the surfaces can freely slide in the tangential direction relative to each other.
Excluded	Select **Excluded** if the bodies are to ignore each other.

Bonded Contact Tool Options

The following tool options are available for the Bonded contact type in a structural analysis:

Strength limits	Select to specify the **Shear** and **Tensile** stress limits of the bond during service conditions if the contacts are not perfectly bonded. After solving, you can assess your contacts in the Connection Assessment Tool. This tool cycles through the connections and shows whether they pass or fail the factor of safety based on these specified shear and tensile limit values.
Shear stress	Specify the shear stress limit of the bond during service conditions. This is used when monitoring the factor of safety of the contact. Contact failure is not simulated.
Tensile stress	Specify the tensile stress limit of the bond during service conditions. This is used when monitoring the factor of safety of the contact. Contact failure is not simulated.

Thermal Conductance Definition	In a structural-thermal simulation, choose how to define the conductive heat transfer between contact surfaces. You can specify the thermal **Conductance** as a value, or you can specify a **Thickness** along with a **Material** or **Conductivity**, which the Discovery application then uses to compute the thermal conductance. Choose **Perfect conductor** if there is no resistance to heat transfer between the surfaces or **Perfect insulator** for a near zero conductance of heat. If contact occurs, a small value of thermal contact conductance yields a measured amount of imperfect contact and a temperature discontinuity across the contact surfaces. A small thermal contact conductance can also be used to represent a thin layer of material with a different conductivity, such as a gasket. For large values of thermal contact conductance, the resulting temperature discontinuity tends to vanish and perfect thermal contact is approached. With a zero value, however, the solver assumes that no heat is transferred across the contacting surfaces. **Note:** While the application can solve with a value of 0, which specifies no heat transfer between the faces, a project with a 0 value for thermal conductance is invalid when exported to the Workbench application. You must then replace zero conductance with some small value.

Sliding Contact Tool Options

The following tool options are available for a sliding contact type:

Idealized	Allows frictionless sliding but no separation, which enforces linear behavior.
Frictional	Allows for sliding with separation. You can then enter a **Friction coefficient**. **Note:** Friction coefficient values are ignored in the Explore stage, but separation is allowed.
Friction coefficient	The coefficient of friction. Entering zero results in frictionless sliding.
Adjust gap or overlap	Defines how the contact interface is treated with **Adjust gap or overlap**. Select **Make just touching** to direct that any initial gaps are closed and any initial penetration is ignored, creating an initial stress-free state. Or you can model true contact gap/penetration plus the offset value you specify with **Define specific offset**. This option is only available in the **Refine** stage.
Stiffness factor	Modifies the default factor for **Stiffness factor** to control the amount of penetration between contact and target surfaces. This option is only available in the **Refine** stage.
Detection radius factor	Modifies the default factor for **Detection radius factor** to specify the radius of contact detection to be used during solve in order to obtain a good convergence. This option is only available in the **Refine** stage.

18.5.2. Automatically Generated Contacts

The Discovery application automatically detects and defines bonded contacts when you set a Structural simulation. These bonded contacts can be reviewed by selecting the **Bonded Contacts (default)** review tool from the physics tree. You can view and filter contacts then convert the contact or contact group to sliding or specify the contact behavior. You can also:

- convert the contact or contact group to a joint or a permanently bonded contact
- exclude the contact or contact group from contact detection
- revert converted contacts back to the default group
- split a contact group into individual per-body contacts or a single contact into individual face pairs

Contact detection behavior in the **Explore** stage is determined by the **Contact Detection** setting from **File** > **Settings** > **Physics**. You can choose to prioritize speed, accuracy, or a balance of the two.

> **Note:** Friction coefficient values are ignored in the Explore stage, but separation is allowed.

This section contains the following topics:

18.5.2.1. Selecting and Reviewing Automatically Generated Contacts

To review contacts, select the **Bonded Contacts (default)** group, or any contact group, in the physics tree. Each contact pair is shown in the model as a single dot, or marker.

This section contains the following topics:

Selecting Contact Markers

If you hover over one of these graphical identifiers, the Discovery application highlights the two surfaces of the contact pair. Selecting the identifier shows the contact # and total number of contacts. If markers overlap, press Ctrl and scroll the mouse wheel to select them.

> **Note:** Selecting any additional contact pairs does not deselect previously selected contact pairs.

Click **Select markers for pairs** to select markers for individual contacts to be converted.

Filtering Contact Markers

Filter to show all contact pairs on selected locations by selecting the **Select a primary body** or **Select a primary face** filter and then selecting one or more locations.

Narrow the review further by selecting the **Select a secondary body** filter or **Select a secondary face** filter and a location to highlight and select only the contacts between this location and the primary locations.

You can also ALT-click the secondary locations.

18.5.2.2. Converting Contacts in Contact Review

When reviewing contacts, you can convert a contact to permanently bonded or exclude the contact so it will no longer be automatically included in the simulation. Additionally, you can convert the automatically generated

bonded pair(s) to a sliding contact or to a joint. The Discovery application then removes that contact from the current contact group and adds it as an individual contact, contact group, or joint in the physics tree. If multiple contact pairs are selected for conversion to a sliding contact, they are grouped into a new contact group in the physics tree.

> **Note:** You can always revert one or more contacts back to the default type.

1. Select one or more contacts you want to convert to manual contacts. The HUD displays the number of bonded contacts selected.
2. Do one of the following:
   - Click **Convert to sliding** to convert the contact to a sliding contact. If only one contact is selected, an individual contact is added to the physics tree. If more than one is selected, a new contact group is added to the physics tree.
   - Click **Exclude from contact detection** to ensure the contact is no longer automatically detected.
   - Click **Convert to joint** to convert the contact to a joint. Each contact is converted and added as an individual joint in the physics tree.
   - For contact groups other than the Bonded Contacts (default) group, click **Convert to permanently bonded** to ensure the contact is always recognized as a bonded contact.

18.5.2.3. Reverting Contacts to Default Type

If you have converted contacts to a different type, or set them to excluded, you can always return those contacts to the default contact group.

1. In the physics tree, select the contact group containing the contacts you want to return to the default group.
2. Select the markers for the individual contact pairs you want to remove from this group.
3. Select **Remove contact pairs from the group**.

   The selected contacts will be removed from the contact group and returned to the default contact group.

18.5.2.4. Modifying the Default Contact Behavior

Automatically detected contacts default to a bonded behavior, but you can modify the properties for the selected contact or contact group in the **Tool Options** under **Group Properties**. You can specify how the contacts are grouped, specify a different contact behavior, and specify linear and non-linear properties. These changes apply to **all** contact pairs in the current contact group. Individually selecting contact pairs applies only to conversion options.

> **Note:** These options apply only to the Default Bonded Contacts group. Once a group is converted to a permanently bonded group or a sliding contact, additional options are available. For more information on these, see Defining Contacts on page 788 and Contact Tool Guides and Options on page 790.

1. For **Detection distance**, specify the **Min** and **Max** radius of contact detection that the Discovery application will use to obtain a good convergence during solve.
2. To modify the grouping behavior, select an option under **Contact Face Grouping**:
   - Select **Group faces by body** to generate the contact based on grouped bodies.
   - Select **Group tangent faces** to generate the contact based on grouped topologies, such as groups of tangent faces.
   - Select **No grouping** to generate the contact on all separate faces.

3. For an electromagnetic simulation, under **Electric Conductance Definition**, select a method for defining current conduction between two contacting surfaces.

   - Select **Perfect conductor** to transfer current between the two surfaces without any resistance.
   - Select **Perfect insulator** to prevent the flow of current between two surfaces.
   - Select **Specified conductance** to specify the electrical conduction as a value.
   - Select **Thickness and material** where the Discovery application then uses them to compute the conductance.
   - Select **Thickness and conductivity** where the Discovery application then uses them to compute the conductance.

   Specifying a value for electric **Conductance** (the real constant electric contact conductance) models current flow between two contacting surfaces in electromagnetic physics by accounting for surface interaction.

   > **Note:** While the application can solve with a value of 0, which specifies no electric transfer between the faces, a project with a 0 value for electric conductance is invalid when exported to the Workbench application. You must then replace zero conductance with some small value.

4. For a structural-thermal simulation, under **Thermal Conductance Definition**, select a method for defining conductance.

   - Select **Perfect conductor** to transfer heat between the two surfaces without any resistance.
   - Select **Perfect insulator** to define near zero conduction of heat transfer between two surfaces.
   - Select **Specified conductance** to specify the conductive heat transfer as a value.
   - Select **Thickness and material** where the Discovery application then uses them to compute the conductance.
   - Select **Thickness and conductivity** where the Discovery application then uses them to compute the conductance.

   Specifying thermal **Conductance** refers to heat transfer between contact surfaces (thermal contact conductance). A low value indicates imperfect contact with a temperature gap and can also represent a thin layer of different material, like a gasket.

   For large values of thermal contact conductance, the resulting temperature discontinuity tends to vanish and perfect thermal contact is approached.

   > **Note:** While the application can solve with a value of 0, which specifies no heat transfer between the faces, a project with a 0 value for thermal conductance is invalid when exported to the Workbench application. You must then replace zero conductance with some small value.

   > **CAUTION:** When a contact is excluded or removed due to the suppression of a body, then the thermal conductance is not retained if the action is undone.

18.5.2.5. Splitting Contacts and Contact Groups

When you convert multiple automatically-generated contacts to sliding or excluded contacts from the Bonded Contacts (default) tool, they become a new contact group in the Physics Tree. From there, you can additionally split the group into individual per-body contacts, or if a single contact references multiple face pairs, split it into individual face pairs.

To split a contact group into individual per-body contacts or a contact into individual face pairs:

1. Select the contact or contact group in the physics tree.

   Note: This action does not apply to the Bonded Contacts (default) group.

2. Right-click and select **Split**.

18.5.2.6. Default Contacts Tool Guides and Options

Default Contacts Tool Guides

The following tool guides are available for default (Bonded or Sliding) Contact groups:

Select a primary body	Narrow the selection of contacts shown to show only contacts on the selected body.
Select a secondary body	Narrow the selection of contacts shown even further to only the contacts between this secondary body and the primary body.
Select a primary face	Narrow the selection of contacts shown to show only contacts on the selected face.
Select a secondary face	Narrow the selection of contacts shown even further to only the contacts between this secondary face and the primary face.
Select markers for pairs	Manually select markers for individual contacts to be converted.

Default Contacts Tool Options

The following tool options are available for default Contact groups:

Convert to sliding	Select to convert the contact to a sliding contact. If only one contact is selected, an individual contact is added to the physics tree. If more than one is selected, a new contact group is added to the physics tree.
Convert to joint	Select to convert the contact to a joint. Each contact is converted and added as an individual joint in the physics tree.
Convert to permanently bonded	For contact groups other than the Bonded Contacts (default) group, select to ensure the contact is always recognized as a bonded contact.
Exclude from contact detection	Select to ensure the contact is no longer automatically detected.

The following additional options are available for the default contact group when expanded:

Detection distance	Sets the **Min** and **Max** radius of contact detection that the Discovery application will use to obtain a good convergence during solve.
Contact Face Grouping	To modify the grouping behavior, select an option: • Select **Group faces by body** to generate the contact based on grouped bodies • Select **Group tangent faces** to generate the contact based on grouped topologies, such as groups of tangent faces. • Select **No grouping** to generate the contact on all separate faces.

Electric Conductance Definition	In an electromagnetic simulation, choose how to define the electric conductance between the two sides of a contact. Electric contact conductance can be used to include a resistance to the flow of electric current. You can specify the electric conductance as a value, or you can specify a thickness and a material or conductivity, which the solver uses to compute the electric conductance. Choose **Perfect conductor** if there is no resistance between the surfaces or **Perfect insulator** for a near zero conductance of charge. **Note:** While the application can solve with a value of 0, which specifies no electric transfer between the faces, a project with a 0 value for electric conductance is invalid when exported to the Workbench application. You must then replace zero conductance with some small value.
Thermal Conductance Definition	In a structural-thermal simulation, choose how to define the conductive heat transfer between contact surfaces. You can specify the thermal **Conductance** as a value, or you can specify a **Thickness** along with a **Material** or **Conductivity**, which the Discovery application then uses to compute the thermal conductance. Choose **Perfect conductor** if there is no resistance to heat transfer between the surfaces or **Perfect insulator** for a near zero conductance of heat. If contact occurs, a small value of thermal contact conductance yields a measured amount of imperfect contact and a temperature discontinuity across the contact surfaces. A small thermal contact conductance can also be used to represent a thin layer of material with a different conductivity, such as a gasket. For large values of thermal contact conductance, the resulting temperature discontinuity tends to vanish and perfect thermal contact is approached. With a zero value, however, the solver assumes that no heat is transferred across the contacting surfaces. **Note:** While the application can solve with a value of 0, which specifies no heat transfer between the faces, a project with a 0 value for thermal conductance is invalid when exported to the Workbench application. You must then replace zero conductance with some small value.
Strength limits	Select to specify the **Shear** and **Tensile** stress limits of the bond during service conditions if the contacts are not perfectly bonded. After solving, you can assess your contacts in the Connection Assessment Tool. This tool cycles through the connections and shows whether they pass or fail the factor of safety based on these specified shear and tensile limit values.
Shear stress	Specify the shear stress limit of the bond during service conditions. This is used when monitoring the factor of safety of the contact. Contact failure is not simulated.
Tensile stress	Specify the tensile stress limit of the bond during service conditions. This is used when monitoring the factor of safety of the contact. Contact failure is not simulated.

> **Note:** Once a contact group is converted to a permanently bonded or sliding contact, contact behavior options are consistent with the Structural Contact Condition options.

18.5.3. Joints

A joint typically serves as a junction where bodies are joined together. Joint types are characterized by their rotational and translational degrees of freedom as being fixed or free.

Joints support fixed, spherical, and hinged types, but you can also define additional types using a support.

This section contains the following topics:

18.5.3.1. Setting Up a Joint

To set up a joint:

1. Select **Joint** from the dropdown menu of the **Structural** tool in the **Physics** group of the **Simulation** tab, or by selecting the **Joint** tool in the **Structural** condition HUD.
2. Select the appropriate geometry depending on the joint type. For cylindrical or spherical joints you will need to select appropriately shaped geometry.

   If your model contains Named Selections, you can click **Advanced filters**, select **Link named selections**, and choose one from the list.

3. Select the geometry representing both sides that will need to be connected.

   The second side will be automatically suggested, but you can choose to make a different selection if needed. Click **Select secondary surface** and select the second face.

   If you only have the geometry of one side modeled, then define a Support instead.

4. Select a joint type:

   - **Fixed**: Movement is prevented in any direction, including rotations.
   - **Hinged**: Only applicable to cylindrical or flat surfaces. For cylindrical surfaces, the axis is along the line of the cylinder. For flat surfaces, you specify an axis that lies in the plane of the selected surface passing through the centroid of the surface. The support is free to rotate about that axis.
   - **Spherical**: Only applicable to spherical surfaces. Translational movement is prevented.

   > **Note:** In addition, you can also define additional types using a support.

5. Click **Accept** to complete the setup.

18.5.3.2. Joints Tool Guides and Types

Joints Tool Guides

The following tool guides are available:

Select a secondary surface	The second side will be automatically suggested, but you can choose to make a different selection if needed. Select this tool and select the second face.

Joint Types

The following joint types are available:

Fixed	Movement is prevented in any direction, including rotations.
Hinged	Only applicable to cylindrical or flat surfaces. For cylindrical surfaces, the axis is along the line of the cylinder. For flat surfaces, you specify an axis that lies in the plane of the selected surface passing through the centroid of the surface. The support is free to rotate about that axis.
Spherical	Only applicable to spherical surfaces. Translational movement is prevented.

18.5.4. Springs

A spring is an elastic element used to store mechanical energy which retains its original shape after a force is removed. Springs connect two bodies together and transfer the load from one body to the other. Springs are typically defined in a stress free or "unloaded" state. This means that no loading conditions exist unless preloading is specified, which creates an initial "loaded" state. Longitudinal or torsional springs generate a force that depends on displacement or rotation, respectively.

This section contains the following topics:

18.5.4.1. Specifying a Longitudinal Spring

Longitudinal springs generate a force that displaces along the axis. To define a longitudinal spring:

1. Select **Spring** from the dropdown menu of the **Structural** tool in the **Physics** group of the **Simulation** tab, or by selecting the **Spring** tool in the **Structural** condition HUD.
2. Define the spring by selecting two faces or a face and a point. Click **Select secondary surface** or Alt-click to select your second location.

   If your model contains Named Selections, you can click **Advanced filters**, select **Link named selections**, and choose one from the list.
3. Select a **Longitudinal** spring type.
4. Specify a **Stiffness** value and press Enter.
5. Enter the spring **Length**, defined as the distance between the center of the two selected faces or between the center of the selected face and a point.
6. Additionally, in the **Refine** stage, you can optionally select a **Preload type**.
   - Select **Free length** to define a preload as a displacement length, then enter a non-zero value for the displacement length. The actual length is calculated using the spring end points from the defined locations.
   - Select **Load** to define the preload as a force load, then enter a non-zero value for the load. A positive value creates tension and a negative value creates compression.

18.5.4.2. Specifying a Torsional Spring

Torsional springs generate a force that rotates about the axis. To define a torsional spring:

1. Select **Spring** from the dropdown menu of the **Structural** tool in the **Physics** group of the **Simulation** tab, or by selecting the **Spring** tool in the **Structural** condition HUD.
2. Define the spring by selecting two faces or a face and a point. Click **Select secondary surface** or Alt-click to select your second location.

   If your model contains Named Selections, you can click **Advanced filters**, select **Link named selections**, and choose one from the list.
3. Select a **Torsional** spring type.
4. Enter a **Stiffness** value.
5. Enter the spring **Length**, defined as the distance between the center of the two selected faces or between the center of the selected face and a point.
6. Additionally, in the **Refine** stage, you can optionally select a **Preload type**.
   - Select **Torque** to define the preload as a force in the opposite direction, then enter a non-zero value for the torque.
   - Select **Rotation** to define the preload as a rotational displacement, then enter the displacement as a non-zero value.

18.5.4.3. Springs Tool Guides and Options

Springs Tool Guides

The following tool guides are available:

Select secondary surface	Click or Alt-click to select your second location.

Spring Types

The following spring types are available:

Longitudinal	Longitudinal springs generate a force that displaces along the axis.
Torsional	Torsional springs generate a force that rotates about the axis.

Springs Tool Options

The following options are available:

Stiffness	Specify the stiffness value.
Length	Specify a value for the spring length, defined as the distance between the center of the two selected faces or between the center of the selected face and a point.

Preload	For the longitudinal spring type, you can optionally select from one of the following preload types: • Select **Free length** to define a preload as a displacement length, then enter a non-zero value for the **Free length**. The actual length is calculated using the spring end points from the defined locations. • Select **Load** to define the preload as a force load, then enter a non-zero value for the **Load force**. A positive value creates tension and a negative value creates compression. For the torsional spring type, you can optionally select from one of the following preload types: • Select **Torque** to define the preload as a force in the opposite direction, then enter a non-zero value for the **Torque**. • Select **Rotation** to define the preload as a rotational displacement, then enter the displacement as a non-zero value.

18.5.5. Connection Assessment

The Connection Assessment Tool cycles through the connections and shows whether they pass or fail the factor of safety for the connection based on specified shear and tensile limit values, or whether they're borderline. All connections in the tool are shown by default, colored by status.

Before using the Connection Assessment tool, you must first specify the shear stress and tensile stress limits of the bond and set the range for connections to be highlighted as borderline.

Using filters, you can narrow down the connections you want to view.

This section contains the following topics:

18.5.5.1. Setting up Connection Assessment

To set up Connection Assessment:

1. Specify the shear stress and tensile stress limits of the bond, used to determine the factor of safety when setting the contact properties, either in the Bonded Contact Review tool (when set to permanently bonded) or when setting up the Structural Contact condition.
2. In the Physics Settings, set the range for connections to be highlighted as borderline.

18.5.5.2. Using Connection Assessment

Before using the Connection Assessment tool, you must first specify the shear stress and tensile stress limits of the bond and set the range for connections to be highlighted as borderline. For more information, see Setting up Connection Assessment on page 800.

To use the Connection Assessment Tool:

1. Ensure you have solved your simulation.
2. Click the **Connection Assessment** tool in the ribbon.
3. Cycle through the connections using the arrow icons. The tool displays the Shear and Tensile values for the selected connection and whether it's Passed, Failed, or Borderline. The tensile value may also show that it's in compression.

4. By default, **Show all** is selected and all connections are shown, but filtering options are available to help narrow down the connections for review. You can filter by location or by connection type, or you can show only the failed connections.
   - To show only connections on one or more bodies, select the **Show for selected bodies** filter and one or more bodies. To show only connections between the selected bodies and an additional body, select the **Show only between selected bodies** filter and one or more bodies, or ALT-select the secondary bodies.
   - To show only connections on one or more faces, select the **Show for selected faces** filter and one or more faces. To show only connections between the selected faces and another face, select the **Show only between selected faces** filter and one or more faces, or ALT-select the secondary face(s).
   - Show only Bolt connections by selecting the **Show only bolt connections** filter.
   - Show only Contact connections by selecting the **Show only contact connections** filter.
   - Show only failed connections by selecting the **Show only failed connections** filter.
5. Select **Automatic zoom** to automatically zoom to the connection currently shown.
6. Additionally, you can save the connection pass/fail information to a comma-separated values (CSV) text file by selecting **Connection Assessment Tool** > **Export to CSV** in the ribbon.

18.5.5.3. Connection Assessment Tool Guides and Options

Connection Assessment Tool Guides

The following tool guides are available for Connection Assessment:

Show for selected bodies	Narrow the selection of connections shown to show only connections on the selected body.
Show only between selected bodies	Narrow the selection of connections shown even further to only the connections between this secondary body and the primary body.
Show for selected faces	Narrow the selection of connections shown to show only connections on the selected face.
Show only between selected faces	Narrow the selection of connections shown even further to only the connections between this secondary face and the primary face.
Show only bolt connections	Filter to show only the bolt connections.
Show only contact connections	Filter to show only the contact connections.
Show only failed connections	Filter to show only the connections that failed.

Connection Assessment Tool Options

The following tool options are available for Connection Assessment:

Note: The Discovery application displays whether the currently selected connection Passed, Failed, or was Borderline and the Critical FOS value.

Shear FOS	Displays the Shear factor of safety values for the selected connection.
Tensile FOS	Displays the Tensile factor of safety values for the selected connection.
Automatic zoom	Automatically zooms to the connection currently shown.
Show all	Shows all connections available for assessment.

18.6. Fluid Flow Conditions

Fluid flow conditions are used to define bodies where flow effects are simulated, optionally including effects, such as buoyancy and wall friction. The fluid flow setup should represent sensible engineering assumptions of the true conditions and therefore should be considered carefully when defining the simulation. You can define your problem by explicitly setting the following fluid condition types:

Flow	An opening where fluid is entering or exiting the simulation.
Fans	A two-dimensional intake and exhaust fan used to model the flow of fluid into or out of a system.
Walls	The representation of a solid or other barrier that prevents any fluid flow. It can be used without defining the solid geometry, or where the geometry is modeled. You can define the interaction of the fluid with this barrier.
Porous	A porous medium, such as a filter or packed bed, modeled as a momentum loss, the magnitude of which involves resistance coefficients.
Rotating Fluid Zones on page 818	A fluid zone that is rotating as a result of moving parts (such as rotating blades, impellers, and similar types of moving surfaces).
Internal Fans on page 819	A three-dimensional internal fan used to simulate the effect of an axial fan by applying a distributed momentum source in a fluid volume.

To apply a fluid flow condition to your model, select **Fluid Flow** in the ribbon of the **Simulation** tab and choose the condition type.

For applications that encounter compressibility effects, see Compressible Flows on page 746.

To parameterize fluid flow conditions, see Physics Available as Parameters on page 884.

This section contains the following topics:

18.6.1. Inlets and Outlets

Flow conditions are used to define where flow is entering or exiting the solution domain.

Determining the type of flow specifications depends on the available flow information. The table below helps you determine the type of flow specification to apply for flow entering the domain.

Upstream conditions (Flow In)	Flow specification	Additional information
A plenum or device delivering flow.	Pressure	Solves the velocity profile and mass flow. Gauge total pressure is the absolute total pressure minus the operating pressure.
A positive displacement device delivering a certain mass or volume flow rate.	Mass flow rate	Solves for pressure. Velocity is derived based on local density. The mass flow rate is capable of overcoming the back pressure of the system being modeled.
	Velocity	Solves for pressure. Mass flow is derived based on the local density. You can specify a velocity profile and the flow direction. If the fluid flow into the domain has rotating or twisting movement as it enters a flow inlet, select **Swirling flow** and specify the speed of fluid rotation.

To include the distribution of temperature within the fluid, you will need to specify the in-flow **Temperature**. For inlets, you will specify the total temperature (also known as the stagnation temperature) rather than the static temperature for compressible flow. For incompressible flow, specify the static temperature. The total temperature represents the temperature of the fluid in a plenum at rest upstream. The local static temperature will be lower, depending on the velocity of the fluid.

> **Note:** Velocity inlets cannot be used with compressible flow because fluid density may vary at that location.

The table below helps you determine the type of flow specification to apply for flow exiting the domain.

Downstream conditions (Flow Out)	Flow specification	Additional information
Fluid is exiting the domain into a large volume where the static pressure is known.	Pressure	A constant static (gauge) pressure will be applied over the entire face. The velocity and mass flow are functions of the solution.
The average velocity is known at the outlet.	Velocity	The velocity is constrained over the face according to the value provided. Pressure is calculated at the outlet as a function of the upstream solution.
When the mass flow through the location is known or expected and at least one pressure condition has been specified.	Mass flow rate	The pressure and velocity are allowed to vary naturally, while maintaining a net mass flow.

For pressure outlets, you can specify the temperature of the re-entering fluid, also known as the backflow temperature.

> **Note:** Your setup must make sense from a physics perspective. For example, for steady-state flow, you can't specify 10kg of fluid flowing into your model and only 2 kg leaving. For this reason, it is highly recommended that one of your flow conditions is defined by pressure.

This section contains the following topics:

18.6.1.1. Setting Up Inlets and Outlets

Define fluid flowing into or out of the model at a specified location.

1. Click the **Fluid Flow** dropdown menu in the **Simulation** tab ribbon and choose **Flow**.
2. Select one or more faces to apply a flow condition.

   If your model contains named selections, click **Advanced filters** and use **Link named selections** to choose one from the list.
3. In the **Fluid Flow** HUD, specify the direction of flow to be in or out by selecting **Inlet** or **Outlet**.
4. Set the fluid flow condition type to be a velocity, mass flow rate, or pressure inlet or outlet.
   - For **Velocity**, enter the velocity magnitude at which fluid is flowing into or out of the domain.

     For velocity inlets, describe the velocity profile of the flow to be **Plug** or **Fully developed** and specify its direction.

     > **Note:**
     > - **Fully developed** flow is available for circular faces in **Explore** and for **Refine** with the LiveGX solver.
     > - If the simulation is transferred to the Fluent application, the fully developed velocity profile will not be transferred.
     > - Fully developed flow direction can only be specified in the normal direction.

     If the fluid flow into the domain has rotating or twisting movement as it enters a flow inlet, select **Swirling flow** and specify the speed of fluid rotation. See Velocity Inlet Options on page 805 for more information.

     > **Note:** Where the location of a velocity inlet is curved rather than planar, **Explore** implements the condition as mass flow inlet.

   - For **Mass flow rate**, this is where fluid is delivered by a positive displacement device upstream and is capable of overcoming the back pressure of the system being modeled.
   - For **Pressure**, enter the gauge total pressure, which represents the pressure obtained by bringing the flow isentropically to rest (for example, upstream plenum pressure). It is measured relative to the operating pressure.

     > **Note:** Your setup must make sense from a physics perspective. So, for example, you can't specify 10kg of fluid flowing into your model and only 2 kg leaving. For this reason, it is highly recommended that one of your flow conditions is defined by pressure.

5. To include the distribution of temperature within the fluid, you will need to specify the in-flow **Temperature**. You will specify the total temperature (also known as the stagnation temperature) rather than the static temperature. The total temperature represents the temperature of the fluid in a plenum at rest upstream. The local static temperature will be lower, depending on the velocity of the fluid.

   For pressure outlets, you can specify the temperature of the re-entering fluid, also known as the backflow temperature.

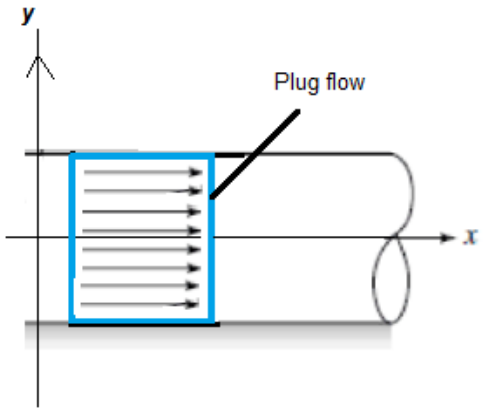
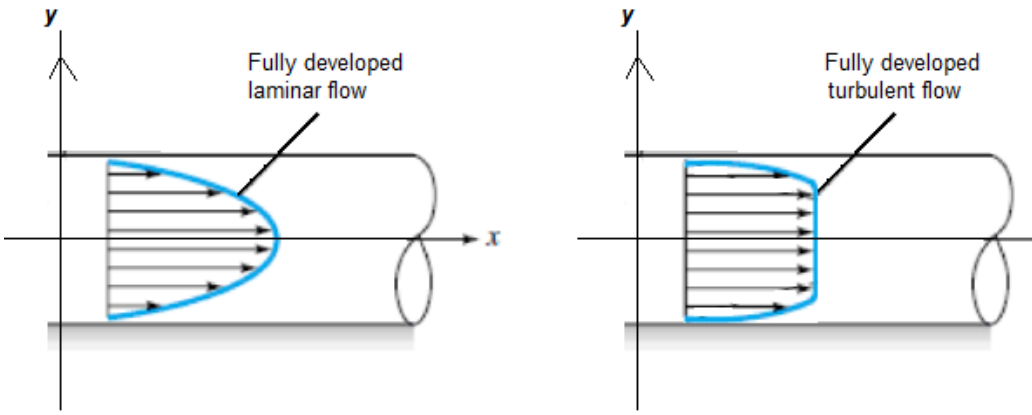
18.6.1.2. Inlet and Outlet Tools

The following options are available:

Velocity	Enter the velocity magnitude at which fluid is flowing into or out of the domain. The specified flow velocity at this location is maintained regardless of any resistance. Additional options for velocity inlet are listed in the table below.
Mass flow rate	Enter the mass flow rate. This is where fluid is delivered by a positive displacement device upstream and is capable of overcoming the back pressure of the system being modeled. The specified flow rate at this location is maintained regardless of any resistance.
Pressure	Enter the gauge total pressure, which represents the pressure obtained by bringing the flow isentropically to rest (for example, upstream plenum pressure). It is measured relative to the operating pressure.

Velocity Inlet Options

The following velocity inlet options are available:

Velocity profile	By default, the velocity profile is considered **Plug** or uniform. That is, the fluid's velocity is constant across any cross-section of the pipe perpendicular to its axis.  Select **Fully developed** if the velocity profile is no longer changing in the direction of flow. This typically occurs after the fluid has traveled a certain distance from the inlet or past an obstruction, allowing the effects of viscosity to fully develop across the pipe or channel's cross-section. the Discovery application will pick the profile based on the turbulence model used in your simulation. This option is available for circular faces in **Explore** and for **Refine** with the LiveGX solver. 
Direction	Specify the direction of flow to be in the normal direction or per the specified X, Y, and Z components for plug flow. With fully developed flow, the direction of flow can only be specified in the normal direction.
Swirling flow	Include **Swirling flow** if the fluid has rotating or twisting movement. Enter the rotational speed if known. You can reverse the direction of rotation by entering a negative rotational speed or double-clicking the arrow showing the rotation direction.

18.6.2. Intake and Exhaust Fans

Intake and exhaust fans are used to model the flow of fluid into or out of a system. In the Discovery application, two-dimensional intake fans are inlets that allow fluid to flow into the system and exhaust fans are modeled as outlets that allow fluid to flow out of the system.

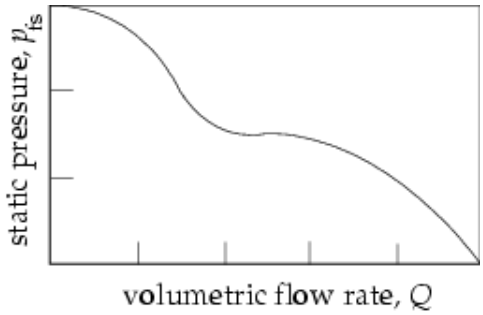
This section contains the following topics:

18.6.2.1. Fan Curves

When modeling fans, fan curves are used to select the appropriate size and type of fan for a particular application. A fan curve, which is usually supplied by the fan manufacturer, is a graph that represents the relationship between volumetric flow rate and the pressure rise across the fan (static pressure). Fan curves are also used to determine the fan pressure rise (also known as the operating point of a fan) based on the computed volumetric flow rate. The fan operating point represents the fan pressure rise and volumetric flow rate where the fan is in stable equilibrium with the system.

> **Note:** If intake or exhaust fans are present in the model, the default pressure drop monitor does not account for the fan pressure rise when calculating the default pressure drop. To obtain accurate pressure drop values in models with intake or exhaust fans, create custom monitors for fan pressure rise.

The figure below shows a curve for a common tube-axial fan. The total volumetric flow rate, Q, is plotted against the fan static pressure, p_{fs}.



The accuracy of the fan flow rate used by the Discovery application is directly related to the accuracy with which the fan static pressure is computed. This, in turn, depends on how accurately pressure losses in the entire system are modeled. Therefore, care should be taken to model all features of the system that contribute to the overall nature of the pressure distribution in the system.

18.6.2.2. Adding a Fan

To add a two-dimensional fan condition to a face or a group of faces that represent a single fan location:

1. Click the **Fluid Flow** dropdown menu in the ribbon of the **Simulation** tab and choose **Fan**.
2. Select a face or a group of faces that represent a single fan location and specify as an intake fan or an exhaust fan.

   If your model contains named selections, click **Advanced filters** and use **Link named selections** to choose one from the list.
3. Define the pressure rise due to the fan.

   In the HUD's options panel, three methods are available:

- Load a pre-defined fan curve or specify the fan curve table in terms of **Volume flow rate** versus pressure rise (static pressure).
- Specify the fan curve table in terms of **Velocity** versus pressure rise (static pressure).
- Specify a Constant pressure rise across the fan. For an intake fan the constant pressure rise is the total pressure and for an exhaust fan it is the static pressure.

4. Enter the **Gauge pressure**, upstream of an intake fan (total pressure) or downstream of an exhaust fan (static pressure).
5. Click **Accept**.

18.6.2.3. Loading a Pre-Defined Fan Model

The Discovery application supports local and Icepak libraries of fan curve data from several manufacturers. Icepak fan libraries only support gases. For the **Volume flow rate** option, select a pre-defined fan model from the library dropdown in the HUD. The fan models consist of tabular data of volumetric flow rate versus pressure rise (static pressure).

You can filter and sort fans to refine your search for the best fan curve fit. You can find information about the selected fan, such as manufacturer, model number, and radius specifications in the HUD's options panel.

You can add new user-defined fans to the library and remove them. The user-defined fans will appear in the fan library dropdown and will be available for use in other simulations.

18.6.2.4. Specifying the Curve for a Fan

You can define a fan curve based on **Velocity** or **Volume flow rate**. Enter pairs of the velocity versus pressure rise (static pressure) or volume flow rate versus pressure rise (static pressure).

The table must contain more than two rows of values, otherwise it will be implemented as a constant pressure rise. You can modify the table as needed and export values from it.

For any fans you've defined (and after accepting the fan settings), enter the name of the fan and click **Save to local library** for use in other simulations. You can also **Remove from local library** any added user-defined fans.

18.6.2.5. Fan Tools

The following options are available:

Volume flow rate	Define the fan curve by entering pairs of volume flow rate and pressure rise (static pressure) values in the table. The table must contain more than two rows of values, otherwise it will be implemented as a constant pressure rise. You can modify the table as needed and export values from it.
Velocity	Define the fan curve by entering pairs of velocity and pressure rise (static pressure) values in the table. The table must contain more than two rows of values, otherwise it will be implemented as a constant pressure rise. You can modify the table as needed and export values from it.
Constant pressure rise	Enter the pressure rise across the fan. For an intake fan the constant pressure rise is the total pressure and for an exhaust fan it is the static pressure.
Gauge pressure	Enter the gauge pressure. This is the pressure upstream of an intake fan (total pressure) or downstream of an exhaust fan (static pressure).
Fan name	Enter the name of the newly defined fan model.

Save to local library	Click to save the new fan curve to the local library.
Remove from local library	Click to remove a user defined fan from the local library. It will remain in the list. If saved with your project, it will be available when the project is resumed. For new projects it will no longer be available in the list.

18.6.3. Walls

Wall conditions assigned to surfaces of a fluid region prevent flow through those surfaces. A no slip wall is created by default for all fluid flows.

The conditions that best describe the forces applied to a wall are summarized below.

Condition at the wall	Wall type	Additional information
The fluid sticks to the wall. If the wall is moving, the fluid moves with the same velocity as the wall.	Non-slip	Specify the wall velocity to be stationary, rotational, or translational. For rotating and translating walls, the direction of motion must be tangential to the face.
Where the viscosity or velocity of the fluid flow means that the friction at the interface with a solid wall has a negligible impact and can be ignored.	Free slip	Components of velocity normal to a free slip wall are zero. Free slip walls are used to: • approximate the free surface of a liquid, for example where the surface of water meets air. • provide a symmetry boundary by modeling an insulated free-slip wall. • indicate the extent of the simulation for external fluid flow.

If your simulation includes the heat distribution within the fluid, choose how heat transfers through the wall, via the thermal condition option that best matches your known information.

This section contains the following topics:

18.6.3.1. Setting Up a Wall Condition

Walls assigned to surfaces of a fluid are typically used to mimic the presence of a solid body and prevent flow through those surfaces. Walls can also be used to define thermal conditions at the extremity of the fluid flow.

1. Click the **Fluid Flow** dropdown menu in the ribbon of the **Simulation** tab and choose **Wall**.
2. Select one or more faces to apply a wall condition. Use Ctrl + click to select multiple faces.

   If your model contains named selections, click **Advanced filters** and use **Link named selections** to choose one from the list.
3. From the dropdown menu on the left side of the HUD, specify the motion of the fluid in contact with the wall.

   By default, the wall is defined to be non-slip, that is the fluid sticks to the wall. The non-slip wall can be stationary or moving. If the wall is moving, the fluid moves with the same velocity as the wall. Moving walls can be **Rotating** or **Translating** and the direction of motion must be tangential to the face. Non-slip, insulated walls will automatically be assumed on any external surface of bodies that represent liquid or gas and do not have flow conditions applied.

Choose **Free slip** when viscosity or velocity of the fluid flow means that the friction at the interface with a solid wall has a negligible impact and can be ignored. Free slip walls are automatically assumed on any walls that indicate the extent of the simulation region for external fluid flow.

4. Optionally, on the right side of the HUD, click **Define a thermal condition** to include thermal conditions and specify how heat is transferred at the wall. Additionally, click **Include internal radiation** to include internal radiation and specify the internal emissivity.

18.6.3.2. Thermal Conditions for Walls

The table below provides descriptions of the thermal conditions for walls.

Thermal Condition	**Description**	**Heat flux at the wall** q_w **calculated from the thermal condition**
Insulated	The wall is perfectly insulated (adiabatic).	$q_w = 0$
Heat flux	The heat flux (energy transfer rate per unit area [W/m^2 or equivalent]) is specified. Entering a value of 0 is equivalent to a perfectly insulated boundary.	$q_w = q_{specified}$
Heat flow	The heat flow (energy transfer rate [W or equivalent]) is specified.	$q_w = Q_{specified} / A$ $Q_{specified}$ is the specified heat flow A is the area
Temperature	The wall temperature is specified.	The solver discretizes q_w based on the local temperature gradient adjacent to the wall.

Thermal Condition	Description	Heat flux at the wall q_w calculated from the thermal condition
Convection	Heat transfer through the wall occurs through external convection. The **Convection coefficient** and **External temperature** are specified.	$q_w = h\left(T_{convection} - T_w\right)$ h is the specified convection coefficient $T_{convection}$ is the specified external temperature T_w is the wall temperature as computed by the solver
Convection and **External Radiation**	Heat transfer through the wall boundary occurs through both external convection and external radiation exchange with a blackbody having a specified **External temperature** and **Convection coefficient**.	$q_w = h\left(T_{convection} - T_w\right) + \varepsilon\sigma\left(T_r^4 - T_w^4\right)$ where radiation is defined as $q_w = \varepsilon\sigma\left(T_r^4 - T_w^4\right)$ ε is the external (wall) emissivity σ is the Stefan-Boltzmann constant T_r is the specified external temperature

18.6.3.3. Internal Radiation

Heating or cooling of surfaces and/or heat sources or sinks due to radiation within the fluid can be included in your model. You should include radiative heat transfer in your simulation when the radiant heat flux, $Q_{rad}=\sigma(T^4_{max}-T^4_{min})$, is important to consider compared to the heat transfer rate due to convection or conduction. Typically, this will occur at high temperatures, where the fourth-order dependence of the radiative heat flux on temperature implies that radiation will dominate.

In the Discovery application, the non-diffracting fluid (also known as surface-to-surface (S2S)) method is used to model radiation. It is used to account for the radiation exchange in an enclosure of gray-diffuse surfaces. The energy exchange between two surfaces depends in part on their size, separation distance, and orientation. These parameters are accounted for by a geometric function called a "view factor".

The S2S method is typically used when the gas medium does not participate significantly in radiative heat transfer. It assumes that any absorption, emission, or scattering of radiation can be ignored, therefore, only "surface-to-surface"

radiation is considered. This option is appropriate for applications such as solar collector systems, radiative heaters, electronics cooling, automotive underhood cooling systems, or spacecraft heat rejection systems.

S2S is computationally very expensive when you calculate the radiation and view factors for a large number of surfaces. To reduce the computational time as well as the storage requirement, the number of surfaces is reduced by creating surface "clusters". The surface clusters are made by starting from a face and adding its neighbors and their neighbors until a specified number of faces per surface cluster is collected.

By default, view factors are calculated using a face-to-face basis, in which clustering is used in a limited way only. The boundary faces act as the surfaces for the view factor calculation, and then a cluster view factor is obtained by taking the area-weighted average of the view factors of the faces within the cluster.

> **Note:** If you transfer a file to the Fluent application to perform further analysis, a view factor file will not be included in the transfer. You must read or compute a view factor file in the Fluent application.

18.6.3.4. Wall Types and Options

By default, the wall is defined to be no slip. For such cases, the fluid adjacent to the wall moves at the same speed as the wall. The following options are available:

Free slip	The motion of the wall does not impact the flow. Typically used to represent the extent of the fluid region adjacent to a frictionless wall. Free slip walls are automatically assumed on any walls that indicate the extent of the simulation region for external fluid flow.
Stationary	The wall is not moving or rotating. This option is available with non-slip walls.
Rotating	The wall is rotating and specify its rotational speed This option is available with non-slip walls.
Translating	The wall is moving in a translational direction. Specify its velocity. The direction of motion for the translating wall must be tangential to the face. This option is available with non-slip walls.
Thermal condition	Thermal conditions at the extremity of the fluid flow. See Thermal Conditions for Walls on page 810 for details.
Internal radiation	Internal radiation is used to model surface-to-surface radiation heat transfer within the fluid region. See Internal Radiation on page 811 for more information.
Internal emissivity	Internal emissivity is the measure of a material's ability to emit heat by radiation from within itself, or from a surface that is surrounded by the same material. It is a dimensionless quantity that ranges from 0 to 1. A value of 0 means radiation is reflected and 1 means radiation is absorbed. Blackbody surfaces have an emissivity of 1.
External emissivity	External emissivity is the measure of a material's ability to emit heat by radiation to its surroundings, or to receive radiative heat from its surroundings. It is a dimensionless quantity that ranges from 0 to 1. A value of 0 means radiation is reflected and 1 means radiation is absorbed. Blackbody surfaces have an emissivity of 1.

18.6.4. Porous Conditions

Porous conditions can be used to model a wide variety of applications, such as fluid flow through filters, packed beds, perforated plates, flow distributors, and tube banks. The porous medium is modeled as a momentum loss, the magnitude of which involves resistance coefficients.

The resistance coefficients control the momentum loss. In general, the loss consists of two parts, a linear term which models the resistance due to viscous losses, and a quadratic term which models the resistance due to inertial losses. The loss is expressed as:

$S_i = -(C_1 \mu v_i + \frac{1}{2} C_2 \rho |v| v_i)$

where

S_i is the momentum loss

C_1 is the viscous resistance coefficient

C_2 is the inertial resistance coefficient

μ is the fluid viscosity

ρ is the fluid density

$|v|$ is the velocity magnitude

v_i is the i^{th} component of the velocity vector

For low-Reynolds number flows, the viscous loss typically dominates, and the inertial resistance coefficient can often be set to 0. For turbulent flows, the inertial loss typically dominates, and the viscous resistance coefficient can often be set to 0.

The values you select for the viscous resistance coefficient and inertial resistance coefficient typically depend on the application and what data you know. For instance:

- If you are modeling flow through a porous medium where Darcy's law applies, set the viscous resistance coefficient, C_1, to $1/\alpha$, where α is the permeability. (If the viscous resistance is large compared to other terms in the momentum equation, the governing equations will automatically recover Darcy's law.)
- If you are modeling flow in a packed bed, the Ergun equation is very similar to the momentum loss expression. The resistance coefficients can be derived from the Ergun equation parameters.
- The inertial resistance coefficient can be interpreted as a loss coefficient per unit length. So, if you are modeling a device such as a perforated plate, filter, or tube bank with a known loss coefficient K_L and thickness Δx, you can select $C_2 = K_L/\Delta x$
- The momentum loss equates to a pressure drop. So if you have experimental tabulated data of pressure drop as a function of velocity for a porous medium of known thickness, you can derive the resistance coefficients.

By default, the resistance coefficients are the same in all directions, isotropic. Alternatively, you can specify different coefficients in the streamwise and crossflow directions, bidirectional, or set a different value for each of the three directions, orthotropic. For the bidirectional and orthotropic options, the reference frame can be Cartesian or cylindrical, depending on the geometry selected using the Direction tool guide. See Specifying Bidirectional and Orthotropic Porous Media on page 815 to specify the direction in other ways.

This section contains the following topics:

18.6.4.1. Setting Up a Porous Condition

To add a porous condition to simulate filters and packed beds:

1. Click the **Fluid Flow** dropdown menu in the ribbon of the **Simulation** tab and choose **Porous**.

2. Select one or more bodies to represent the location of the porous medium (such as a filter or packed bed). Use Ctrl + click to select multiple bodies.

   If your model contains named selections, click **Advanced filters** and use **Link named selections** to choose one from the list.

   > **Note:** If a body not designated as a body of influence overlaps a fluid region, the simulation will arbitrarily assign the overlapping region as either fluid or solid. This can result in unexpected or erroneous outcomes. Check for body interference before solving.

   If using bidirectional or orthotropic media, the Discovery application will try to determine the corresponding directions automatically. Alternatively, you can use the **Direction** tool in the HUD (or Alt-select) to set the direction by selecting a face, edge, plane, line, axis, coordinate system or any of the coordinate system's axes. See Specifying Bidirectional and Orthotropic Porous Media on page 815 for more information on how you can define the streamwise and crossflow directions of the porous medium.

3. Optionally, in the right side of the HUD, select **Define a thermal condition** to define a thermal condition in the porous medium and switch to a thermal flow simulation. If thermal effects are included, the porosity and material properties of the solid material must be specified under **Solid Properties** in the options panel.
4. Specify the viscous and inertial resistance coefficients in the options panel. If **Directionality** is set to **Bidirectional** or **Orthotropic**, specify the streamwise and crossflow viscous and inertial resistance coefficients. For conical or cylindrical orthotropic media, you will need to specify the radial, circumferential, and tangential resistance coefficients.

   > **Tip:**
   > - Use **Isotropic** if the porous loss coefficients are the same in all directions. Porous media such as packed beds are typically isotropic.
   > - Use **Bidirectional** if the porous loss coefficients are specified in the streamwise and crossflow directions. Filters and perforated plates are often porous media that are dependent on direction. Bidirectional is also used for cylindrical or conical filters.
   > - Use **Orthotropic** where the porous loss is different in each of the three directions.

5. Press Enter or click **Accept** ✓ to complete the setup.

   For bidirectional and orthotropic porous media, markers will be displayed on the model corresponding to the streamwise and crossflow resistance coefficients you specified. If a cylindrical or conical face or a cylindrical coordinate system is selected to set the direction, the markers will reflect that set of resistance coefficients.

18.6.4.2. Deriving the Porous Coefficients Based on Experimental Pressure and Velocity Data

Experimental data for pressure drop across a porous medium, of thickness Δx, against velocity through it can be fitted to a curve in the form of $\Delta P/\Delta x = Av^2 + Bv$ to determine the resistance coefficients.

An example set is shown in the table below:

Velocity (m/s)	**Pressure Drop (Pa)**
20.0	197.8

Velocity (m/s)	Pressure Drop (Pa)
50.0	948.1
80.0	2102.5
110.0	3832.9

Plotting a curve through these points yields $\Delta P/\Delta x = 0.27394v^2 + 4.68819v$. In this example, based on a porous media thickness, Δx, assumed to be 1 m, the viscous and inertial resistance coefficients are calculated as follows:

Viscous Resistance Coefficient, C_1

From this curve the viscous resistance factor, C_1, can be determined by $C_1 = B/\mu$, where B is from the fitted curve, and μ is the fluid viscosity. In this example, B = 4.68816 kg/(m^3 s) and if $\mu = 1.7894 \times 10^{-5}$, then $C_1 = 261996$ 1/m^2

Inertial Resistance Coefficient, C_2

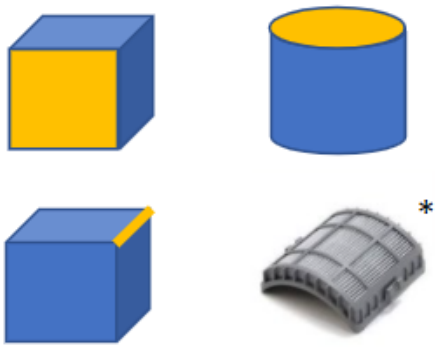
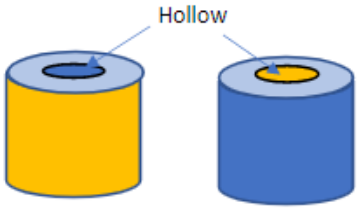
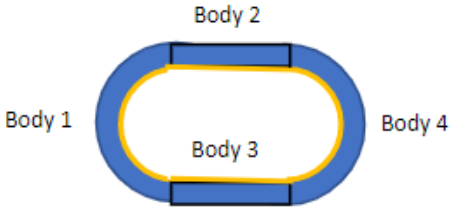
From this curve the inertial resistance factor, C_2, can be determined by $C_2 = 2A/\rho$, where A is from the fitted curve, and ρ is the density of the fluid. In this example, A = 0.27394 kg/m^4 and if $\rho = 1.225$ kg/m^3, this leads to $C_2 = 0.4472$ 1/m.

> **Note:** If you derive negative porous resistance coefficients when fitting experimental pressure-velocity data, this indicates a non-physical result, where the porous region would promote flow rather than resist it. The Discovery application does not allow negative coefficients. Review and modify the pressure-velocity curve fitting to ensure that the resulting coefficients are positive. If negative resistances persist after curve adjustment, set those negative resistance values to zero, so only positive resistances are considered in the simulation. This ensures the model remains physically meaningful and avoids solver errors or non-physical pressure rises.

18.6.4.3. Specifying Bidirectional and Orthotropic Porous Media

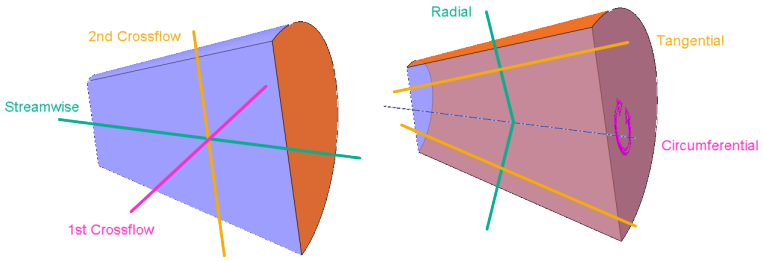
When the porous medium is not isotropic, the Discovery application offers the option to specify viscous and inertial resistance coefficients in the streamwise and crossflow directions.

To specify the direction, you must first select **Porous** from the **Fluid Flow** dropdown menu in the **Simulation** tab ribbon. Select one or more fluid bodies and set the **Directionality** option to **Bidirectional** or **Orthotropic**. Use the **Direction** tool guide to set the primary direction by selecting a face, edge, line, plane, axis, coordinate system or any of the coordinate system's axes. The following table summarizes how directionality is defined based on different filter shapes:

Directionality	Example	Description
Linear		Any selection of a straight edge, plane, planar face will define the streamwise direction. A surface or edge need not be part of the location of the porous condition. When using a Cartesian coordinate system to set the direction, the X-axis is considered the streamwise direction, while the Y and Z axes are the crossflow directions, respectively. Selecting an individual axis of a coordinate system sets the streamwise direction parallel to it, while the next direction in the coordinate system becomes the first crossflow direction. *Some shapes can be assumed linear bidirectional or could be idealized as such.
Cylindrical and conical		Selecting either the inner or outer cylindrical surface, or a circular edge, will define it with a cylindrical coordinate system, with streamwise being normal to the selected face. Cones are also supported if a face is selected (angle detected from face). The surface or edge does not have to be on the medium. Therefore, if you have an irregular shape (typically rotationally symmetric) with which you want a cylindrical cross-section to be used, choose another axially aligned cylinder or circular line (such as an inlet pipe), or use a cylindrical coordinate system. When using a cylindrical coordinate system to define the location, the central axis of the porous medium is set at the system's origin.
Composite		For ovals or other shaped filters, use multiple filter bodies, and a combination of cylindrical and linear bidirectional definitions.

Tip: When the directionality of a porous condition is defined by a local coordinate system, you can ensure that the location of the condition and the coordinate system are moved together by putting them in a single component. When the component is moved, all its contents are moved.

The following images explain the different directions of the viscous and inertial resistance coefficients using the selected geometry (highlighted in orange) as a reference to set the primary direction.



18.6.4.4. Porous Tool Guides and Options

Porous Tool Guides

The following tool guide is available:

Direction tool	The **Direction tool** defines the direction of the bidirectional or orthotropic porous medium. Select a face, edge, line, plane, axis, coordinate system or any of the coordinate system's axes to align the direction with that entity.

Porous Tool Options

The following options are available:

Directionality	Select **Isotropic** if the porous loss coefficients are the same in all directions. Porous media such as packed beds are typically isotropic. Select **Bidirectional** if the porous loss coefficients are specified in the streamwise and crossflow directions. Filters and perforated plates are often porous media that are dependent on direction. Bidirectional is also used for cylindrical or conical filters. **Orthotropic** is available for filters where the porous loss is different in all three directions.
Viscous	Set the viscous resistance coefficient that controls the momentum loss. It is typically obtained by correlating the pressure drop to velocity using experimental data, when the loss is dominated by viscous effects. It is also equal to the inverse permeability, if that is known. For **Bidirectional** and **Orthotropic**, you will need to set the resistance coefficients in the **Streamwise** and **Crossflow** directions.
Inertial	Set the inertial resistance coefficient that controls the momentum loss. It is typically obtained by correlating the pressure drop to velocity using experimental data, when the loss is dominated by inertial effects. For **Bidirectional** and **Orthotropic**, you will need to set the resistance coefficients in the **Streamwise** and **Crossflow** directions.
Porosity	Enter a value between 0 and 1. A porosity of 1 represents a fully open medium. Porosity is used to compute the effective thermal properties of the porous media.
Density	Enter the density of the solid material.
Isotropic thermal conductivity	Enter the Isotropic thermal conductivity of the solid material. This is the material's ability to conduct heat equally in all directions.

Specific heat	Enter the specific heat capacity of the solid material. It is the heat required to raise the unit mass of a substance by one degree of temperature.

18.6.5. Rotating Fluid Zones

The Discovery application generally solves the equations of fluid flow and heat transfer in a stationary (or inertial) reference frame. However, there are many problems where the equations might need to be solved in a moving (or non-inertial) reference frame. For example, fans and mixing tanks with their rotating blades, impellers, and similar objects (comprising a cylindrical body and blades) cause the fluid around those parts to flow with some rotation. In most cases, the moving parts render the problem unsteady when viewed from the stationary frame. However, the flow around the moving part can (with certain restrictions) be modeled as a steady-state problem with respect to the moving frame.

For simulations involving moving parts, the application uses the moving reference frame model with a steady speed and modifies the equations of motion to incorporate the additional acceleration terms that occur due to the transformation from the stationary to the moving reference frame. By solving these equations in a steady-state manner, the flow around the moving parts can be modeled.

Guidelines and Best Practices

- The location of the rotating fluid zone must be a revolved body with cylindrical or conical outer faces.
- A rotating fluid zone and porous zone cannot be specified on the same body.
- A rotating fluid zone cannot share a fluid-fluid interface with another rotating fluid zone.
- Only **Wall** condition options **Rotating with the fluid zone** or **Stationary** are allowed. The default wall behavior for a rotating fluid zone is **Rotating with the fluid zone**.
  - User specified moving walls are not permitted on a rotating fluid zone.
- Rotating fluid zones are not supported in compressible flow simulations.
- Rotating fluid zones are not supported in time-dependent fluid flow simulations.
- For some simulations involving solid-body rotation, there could be some discrepancies between solutions in **Explore** and **Refine**.
- Results Variables on page 917 for rotating fluids zones are shown in the absolute reference frame.
- For closed domain cases, you might run into convergence discrepancies.
- In **Refine**, it might be necessary to reduce the target residual and increase the maximum number of iterations. Monitor plots should be used to help assess convergence.
- In a fluid-solid heat transfer simulation, the effect of rotation of a solid thermal region on the fluid will be included only where a face of that region is in contact with a rotating fluid zone.
- Fluid flow conditions cannot be created on faces representing the rotating device or outer cylindrical/conical faces of a rotating fluid zone.
- Non-coaxial bodies cannot be created in a single rotating fluid zone.
- If a face on the solid thermal region is in contact with faces on both a rotating and a stationary fluid zone, you must imprint the face to obtain heat transfer between the solid thermal region and both fluid zones.
- Rotating geometry consisting of thin blades may be difficult to resolve in **Explore**, which in turn leads to poor results. Use a higher fidelity setting for better results.
- In **Explore**, all flow conditions located on a rotating fluid zone are defined relative to the absolute frame.
  - For example, if a rotating fluid zone includes a swirl inlet, the rotational velocity is to be entered in absolute frame and not with respect to the rotating fluid zone.
- In **Refine,** all flow conditions located on a rotating fluid zone are defined relative to the absolute frame except for a mass flow outlet. Flow through a mass flow outlet takes the direction of the upstream flow.

18.6.5.1. Setting Up a Rotating Fluid Zone

To add a rotating fluid zone to simulate flow around moving parts:

1. Click the **Fluid Flow** dropdown menu in the ribbon of the **Simulation** tab and choose **Rotating Fluid Zone**.
2. Select a revolved body with cylindrical or conical outer faces to represent the location of the rotating fluid zone.

   The Discovery application creates a default rotating wall for the zone. The axis of rotation is automatically defined by the cylindrical or conical geometry.

   If your model contains named selections, click **Advanced filters** and use **Link named selections** to choose one from the list.

   > **Note:** If a body not designated as a body of influence overlaps a fluid region, the simulation will arbitrarily assign the overlapping region as either fluid or solid. This can result in unexpected or erroneous outcomes. Check for body interference before solving.

3. Specify the rotational speed and press Enter.

18.6.6. Internal Fans

Internal (or 3D) fans simulate the effect of an axial fan by applying a distributed momentum source to a body within a fluid region. The source is defined by the following equation:

$S_a=\Delta P(Q)/h$

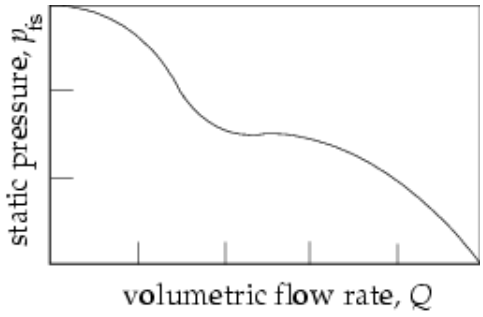
Where ΔP is the pressure rise across the fan for a given axial flow rate, Q, and h is the thickness of the fan body.

This section contains the following topics:

18.6.6.1. Fan Curves

When modeling fans, fan curves are used to select the appropriate size and type of fan for a particular application. A fan curve, which is usually supplied by the fan manufacturer, is a graph that represents the relationship between volumetric flow rate and the pressure rise across the fan (static pressure). Fan curves are also used to determine the fan pressure rise (also known as the operating point of a fan) based on the computed volumetric flow rate. The fan operating point represents the fan pressure rise and volumetric flow rate where the fan is in stable equilibrium with the system.

The figure below shows a curve for a common tube-axial fan. The total volumetric flow rate, Q, is plotted against the fan static pressure, p_{fs}.



The accuracy of the fan flow rate used by the Discovery application is directly related to the accuracy with which the fan static pressure is computed. This, in turn, depends on how accurately pressure losses in the entire system are modeled. Therefore, care should be taken to model all features of the system that contribute to the overall nature of the pressure distribution in the system.

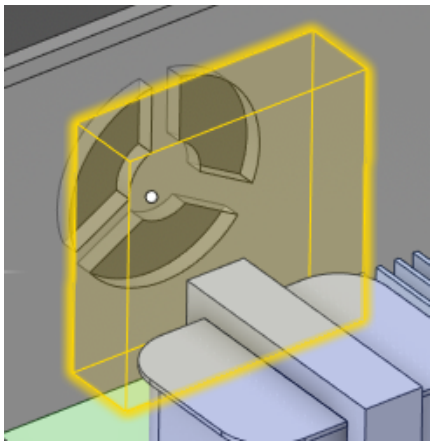
18.6.6.2. Adding an Internal Fan

To add a three-dimensional internal fan condition to a body that represents the fan location:

1. Click the **Fluid Flow** dropdown menu in the ribbon of the **Simulation** tab and choose **Internal Fan**.
2. Select a single fluid body that represents the location of the fluid at the internal fan.

> **Note:** You are not selecting the actual solid fan, but rather a volumetric fluid body that you create, which is representative of the fan region. The internal fan region is only an approximation of the actual flow, so it can be any shape with constant cross section that represents the fan region where the flow enters and exits.

Here is an example of a rectangular volumetric fluid body representing the location at the internal fan.



The application automatically detects the inlet face of the fan.

If your model contains named selections, click **Advanced filters** and use **Link named selections** to choose one from the list.

> **Note:** If a body not designated as a body of influence overlaps a fluid region, the simulation will arbitrarily assign the overlapping region as either fluid or solid. This can result in unexpected or erroneous outcomes. Check for body interference before solving.

3. To override the automatic selection of inlet face, use the Select inlet face tool guide  to define the inlet of the fan.

> **Note:** The inlet and outlet of the fan must be located on single faces that span the entire cross section of the body. The inlet and outlet faces of the fan cannot be external.

4. Define the pressure rise due to the fan.

In the HUD's options panel, two methods are available:

- Load a pre-defined fan curve or specify the fan curve table in terms of **Volume flow rate** versus pressure rise (static pressure).
- Specify a **Constant pressure rise** across the fan.

5. In the options panel, **Shrouded** is selected by default. A shrouded fan channels fluid through the fan so that the fluid is pulled through the fan system. Creating a shrouded fan creates internal fluid walls on the exterior faces of the fan, which force the fluid to flow through the fan. This maximizes the cooling efficiency of the fan.

   With this option off, it creates a fluid-fluid interface between the fan fluid volume and the surrounding fluid volume, allowing fluid to flow through the interface.
6. In the options panel, you can include a heat source, select **Specify total heat**. Enter the total heat value representing the heat dissipation of the fan.
7. In the options panel, specify the **Hub radius** and **Tip radius** of the blade. This value will not affect the fluid flow and will only be used for fan curve filtering.
8. Click **Accept**.

> **Note:**
>
> - Flow inlets and outlets cannot be located on the fan body.
> - If the simulation is transferred to the Fluent application, it will not include internal fan data.
> - Internal fans are not supported in **Refine** when LiveGX is switched off. Turn on **Use LiveGX solver** under **Simulation Options**, or switch to the **Explore** stage.

18.6.6.3. Loading a Pre-Defined Fan Model

The Discovery application supports local and Icepak libraries of fan curve data from several manufacturers. Icepak fan libraries only support gases. For the **Volume flow rate** option, select a pre-defined fan model from the library dropdown in the HUD. The fan models consist of tabular data of volumetric flow rate versus pressure rise (static pressure).

You can filter and sort fans to refine your search for the best fan curve fit. You can find information about the selected fan, such as manufacturer, model number, and radius specifications in the HUD’s options panel.

You can add new user-defined fans to the library and remove them. The user-defined fans will appear in the fan library dropdown and will be available for use in other simulations.

18.6.6.4. Specifying the Curve for a Fan

You can define a fan curve based on **Volume flow rate**. Enter pairs of the volume flow rate versus pressure rise (static pressure).

The table must contain more than two rows of values, otherwise it will be implemented as a constant pressure rise. You can modify the table as needed and export values from it.

For any fans you've defined (and after accepting the fan settings), enter the name of the fan and click **Save to local library** for use in other simulations. You can also **Remove from local library** any added user-defined fans.

18.6.6.5. Internal Fan Tool Guides and Options

Internal Fan Tool Guides

The following tool guide is available:

Select inlet face	Select an alternative face to the Discovery application's automatic detection of the inlet fan face.

Internal Fan Tool Options

The following options are available:

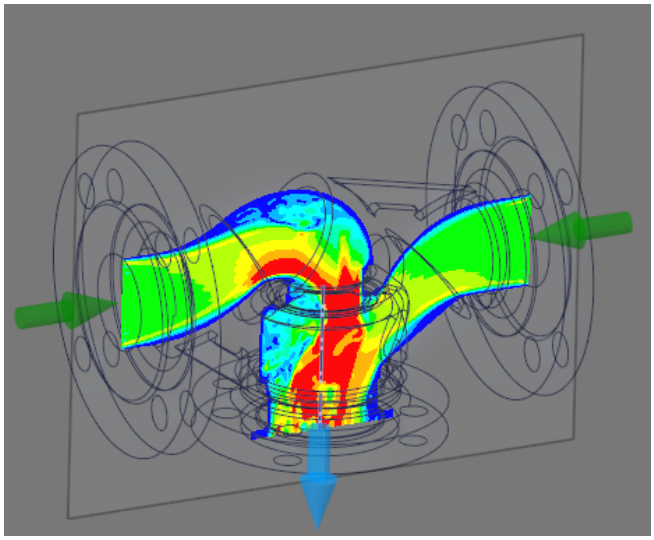
Volume flow rate	Define the fan curve by entering pairs of volume flow rate and pressure rise (static pressure) values in the table. The table must contain more than two rows of values, otherwise it will be implemented as a constant pressure rise. You can modify the table as needed and export values from it.
Constant pressure rise	Enter the pressure rise as a constant value.
Shrouded	Channel fluid through the fan blades so that the fluid is pulled through the fan system rather than around it. This maximizes the cooling efficiency of the fan.
Specify total heat	Enter the total heat value representing the heat dissipation of the fan.
Hub radius	Define the radius of the fan hub. This is the inner radius of the blade-swept volume on which the fan's distributed momentum sources are applied. This value will not affect the fluid flow and will only be used for fan curve filtering.
Tip radius	Define the radius of the blade tips of the fan. This is the outer radius of the blade-swept volume on which the fan's distributed momentum sources are applied. This value will not affect the fluid flow and will only be used for fan curve filtering.
Internal Fan name	Enter the name of the newly defined fan model.
Save to local library	Click to save the new fan curve to the local library.
Remove from local library	Click to remove a user defined fan from the local library. It will remain in the list. If saved with your project, it will be available when the project is resumed. For new projects it will no longer be available in the list.

18.6.7. Internal and External Flow

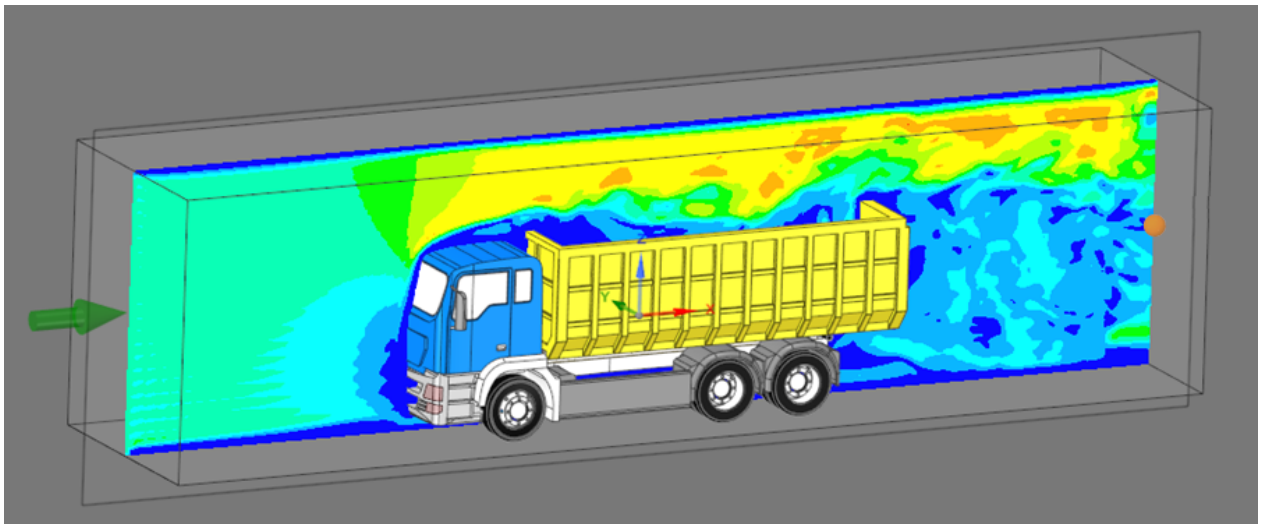
In a fluid flow simulation, the fluid flow region comprises one or more volume bodies representing a space around and between solid objects through which fluid may flow; that space is filled with a fluid material. There are two types of fluid flow simulations:

- A simulation that represents internal flow.
- A simulation that represents external flow.

For internal flow simulations, the motion of the fluid material is constrained by physics conditions to the region by walls representing the bounding solid bodies, and may enter or exit the region through inlets, outlets, and openings that represent conditions outside of the modeled region. Examples of internal flow are fluid flowing through compressors, pumps, fans, and pipes.



In external flow simulations, the fluid material surrounds the body. Examples of external flow are air flow around an airfoil, vehicle, or any object. When simulating external flow around an object, your geometry might not contain a fully defined external flow boundary. You will create an external flow boundary by defining a bounding box that surrounds the relevant aspects of your geometry, such as around a vehicle.



Note: You can set one or more solid bodies as cutter bodies to automatically block the fluid flow, allowing you to experiment with the positions of solid objects within the flow path without needing to geometrically extract overlapping bodies from the fluid.

To model internal or external flow, use one of the **QuickStart** options in the ribbon of the **Simulation** tab:

- **Internal Flow**: After the internal flow region is defined by the inlets, outlets, and (if relevant) flow caps, the flow volume is extracted automatically. Values for the flow inlet and outlet conditions are shown in the physics tree.
- **External Flow**: The enclosure volume is automatically created and values for the flow inlet and outlet conditions can be found in the physics tree. Friction will be included on the ground and model surfaces. No friction will be included for the remaining enclosure surfaces.

> **Note:** The **External Flow** quick start option is only available when in the **Explore** stage. It cannot be transferred to the Fluent application from the **Explore** stage. To model external flow in **Explore** and **Refine**, use the **Enclosure tool** (accessible from the ribbon of the **Prepare** tab). In general, the **Enclosure and Volume Extract tools** (also in the **Prepare** tab) can be used for more complex geometries and applications to create flow volumes.

This section contains the following topics:

18.6.7.1. Extracting an Internal Flow Volume

To extract an internal flow volume:

1. Prior to setting up your flow volumes, you can specify solid bodies as cutter bodies to automatically block the fluid flow without having to extract the overlapping bodies from the fluid. This enables you to quickly experiment with the positions of solid objects within the flow path.

   To set a solid body as a cutter body, right-click the body in the geometry or model tree and select **Cutter Body** from the context menu.

   For overlapping bodies, right-click the body in the geometry or model tree and select:

   - **Overlapping Bodies** > **Set as Cutter Bodies** to set the body as a cutter body
   - **Overlapping Bodies** > **Exclude from Simulation** to exclude the body from the simulation entirely
   - **Overlapping Bodies** > **Identify** to launch the Interference tool, which detects and removes interference from clashing bodies

2. Click the **Inlet** tool then select the face that identifies where the fluid enters the interior flow volume. Use Ctrl + click to select more faces for additional inlets.

   After the first face is selected, you will automatically be switched to Outlet. Pressing the Ctrl key returns you to Inlet.

3. Click the **Outlet** tool then select all the faces that identify where the fluid exits the interior flow volume.
4. Optionally, click the **Cap** tool to select a face that identifies where the interior flow volume is capped so that the fluid does not enter or exit. Use Alt + click to select more faces for additional flow caps.

After the internal flow region is defined by the inlets, outlets, and (if relevant) flow caps, the flow volume is extracted automatically. Values for the flow inlet and outlet conditions are shown in the physics tree.

18.6.7.2. Creating an External Flow Volume

1. Prior to setting up your flow volumes, you can specify solid bodies as cutter bodies to automatically block the fluid flow without having to extract the overlapping bodies from the fluid. This enables you to quickly experiment with the positions of solid objects within the flow path.

   To set a solid body as a cutter body, right-click the body in the geometry or model tree and select **Cutter Body** from the context menu.

   For overlapping bodies, right-click the body in the geometry or model tree and select:

   - **Overlapping Bodies** > **Set as Cutter Bodies** to set the body as a cutter body
   - **Overlapping Bodies** > **Exclude from Simulation** to exclude the body from the simulation entirely
   - **Overlapping Bodies** > **Identify** to launch the Interference tool, which detects and removes interference from clashing bodies

2. Click the **Inlet direction** tool then select the green arrow that indicates where the flow enters.

3. Click the **Ground plane** tool then select the enclosure face that acts as the ground.

The enclosure volume is automatically created and values for the flow inlet and outlet conditions can be found in the physics tree. Friction will be included on the ground and model surfaces. No friction will be included for the remaining enclosure surfaces.

18.6.7.3. Internal Flow Tool Guides

The following tool guides are available for the internal flow:

Inlet	Defines the face that identifies where the fluid enters the interior flow volume. Use Ctrl + click to select more faces for additional inlets. After the first face is selected, you will automatically be switched to the Outlet tool guide. Pressing the Ctrl key returns you to Inlet.
Outlet	Defines the faces that identify where the fluid exits the interior flow volume.
Cap	Defines the face that identifies where the interior flow volume is capped so that the fluid does not enter or exit. Use Alt + click to select more faces for additional flow caps.

18.6.7.4. External Flow Tool Options

The following tool options are available for an external flow:

Inlet direction	Click and then select the green arrow to indicate where the flow enters.
Ground plane	Click and then select the enclosure face that acts as the ground.

18.6.8. Fluid-Fluid Interfaces

Fluid-fluid interfaces exist when your simulation contains different fluid regions in contact with one another or a single fluid region that comprises more than one body. For example, the Discovery application automatically creates a fluid-fluid interface if one of the fluid regions is a rotating zone or in simulations containing additional bodies as locations for porous conditions. Additionally, multiple bodies might be used to allow for the local refinement of the mesh in **Refine**.

Reviewing the fluid interface allows you to control the detection distance and visually diagnose the overlapping interfaces or conversely check that the interface does exist where it is expected. Decreasing the detection distance enforces a tighter tolerance for detecting whether faces are touching or not. Using a smaller detection distance usually removes any spurious faces that may be overlapping with another condition or interface.

You can convert selected interface pairs to virtual walls, where you can then specify thermal conduction properties across these thin surfaces. Using surfaces that are coincident (surfaces that are not separated by a gap), when defining virtual walls, typically results in more accurate results.

This section contains the following topics:

18.6.8.1. Reviewing Fluid-Fluid Interfaces

The application automatically detects and creates fluid-fluid interfaces in simulations with multi-body fluid regions. Reviewing the interfaces allows you to control the detection distance and visually diagnose the overlapping faces. Decreasing the detection distance enforces a tighter tolerance for detecting whether faces are touching or not. Using

a smaller detection distance usually removes any spurious faces that may be overlapping with another condition or interface. Additionally, you can convert any of the interfaces to virtual walls, where you can then specify thermal conduction properties. These fluid-fluid interfaces can be reviewed by double-clicking the automatically created condition under **Fluid-Fluid Interfaces** in the physics tree. Alternatively, under **Review** on the **Simulation** tab, select **Default Fluid-Fluid Interfaces** to load the **Interface Review** tool. Each interface pair is shown in the model as a single dot.

1. Review the automatically generated interfaces between the fluid regions. The total number of generated interface pairs appears on the right side of the HUD. Each interface pair is shown in the model as a single small marker.
2. Hover over one of the spherical identifiers, the application highlights the surface of the interface pair. Select an identifier to show the interface pair and total number of interfaces.

   > **Note:** Selecting an additional marker does not deselect previously selected pairs.

3. Select the interface pair you want to review.

   You can filter to show all interface pairs on selected locations by selecting the **Select a primary body** or **Select a primary face** filter and then selecting one or more locations.

   You can narrow the review further by selecting the **Select a secondary body** filter or **Select a secondary face** filter and a location to highlight and select only the interface pairs between this location and the primary locations. You can also ALT-select the secondary locations.

4. Optionally, convert selected interface pairs to virtual walls, where you can then specify thermal conduction properties across these thin surfaces.

   Using surfaces that are coincident (surfaces that are not separated by a gap) typically results in more accurate results.

5. Optionally, specify the maximum distance of interface detection to be used in the **Max** field for **Detection distance**.
6. By default, interface faces are grouped by body. You can choose to group them by other topologies, like tangent faces, or to apply no grouping so that interfaces are created on all interface faces.

18.6.8.2. Fluid-Fluid Interfaces Tool Guides and Options

Fluid-Fluid Interfaces Tool Guides

The following tool guides are available:

Select a face	Select a face for which you want to review all interface pairs on the primary face.
Select a body	Select a body for which you want to review all interface pairs on the primary body.
Select a secondary face	Select a secondary face to review only interface pairs between the secondary face and any primary face.
Select a secondary body	Select a secondary body to review only interface pairs between the secondary body and any primary body.
Select markers	Select markers to highlight individual interface pairs.

Fluid-Fluid Interfaces Tool Options

The following options are available:

Create virtual wall	Create a virtual wall from the selected interface.
Detection distance	Specify the maximum distance of interface detection to be used. For example, for a rotating fluid zone inside a stationary fluid zone, where a narrow gap exists between a face in the fluid-fluid interface and a fluid wall on the stationary zone or a fluid-solid interface, the detection distance should be small enough to avoid the detection of the outer face.
Face overlap allowance	Specify the percentage that faces are allowed to overlap during interface detection. The default for this field is set in the Physics Defaults on page 76. This option is particularly useful when you have very small contact points, especially near curved edges.
Group faces by body	View interfaces grouped by the body and optionally create a virtual wall per grouped body.
Group tangent faces	View interfaces grouped by tangent faces and optionally create a virtual wall per grouped tangent face.
No grouping	View individual interface pairs and optionally create a virtual wall for the desired separate face pairs.

18.6.8.3. Defining Virtual Walls

In simulations where fluid-fluid interface pairs exist, you have the option to convert any interface pair to a virtual wall. Creation of a virtual wall modifies the fluid-fluid interface pair to influence the flow in a similar way to a stationary wall condition and block flow across it.

> **Note:** Although virtual walls are available only with the LiveGX solver, you can still transfer data to the Fluent application.

1. From the physics tree, double-click **Virtual Wall** to edit it.

   > **Note:** You can specify thermal conduction properties across these thin surfaces. The heat transfer is only between the sides of a virtual wall and not along it.

2. Choose **Perfect conductor** if there is no resistance to heat transfer between the fluid-fluid regions or **Perfect insulator** to block conduction of heat across the virtual wall.

   > **Note:** Using surfaces that are coincident (surfaces that are not separated by a gap) typically results in more accurate results.

18.7. Solid Thermal Conditions

This section contains the following topics:

18.7.1. Temperature

The Temperature condition simulates maintaining the defined temperature over the selected geometry. A temperature condition will either add or remove heat from a body depending on the other thermal conditions applied to the simulation model. Basically, the face will be maintained at the specified temperature and the amount of heat transferred through the face will adjust accordingly.

This section contains the following topics:

18.7.1.1. Specifying a Temperature Condition

To specify a temperature:

1. Select **Temperature** from the dropdown menu of the **Thermal** tool in the **Physics** group of the **Simulation** tab, or by selecting the **Temperature** tool in the **Thermal** condition HUD.
2. Select one or more faces or bodies to define the temperature.

   To filter the selection by face, all faces of body, construction geometry, body, or physics region, click the **Advanced filters** tool. Only valid geometric entities will be available for selection. If your model contains Named Selections, click **Link named selections** to choose one from the list.

3. Specify the **Temperature** to maintain and press Enter.

The face or body will be maintained at the specified temperature and the amount of heat transferred through the face will adjust accordingly.

18.7.1.2. Temperature Tool Guides and Options

Temperature Tool Guides and Options

The following tool guides are available for temperature:

Select	Select a face to add a thermal condition. This is the default filtering option.
Advanced filters	Filter the selection by face, all faces of body, construction geometry, body, physics region, or named selection. Only valid geometric entities will be available for selection.

Temperature Tool Options

The following tool options are available for temperature:

Temperature	The temperature at which the face or body will be maintained. The amount of heat transferred through the face will adjust accordingly.

18.7.2. Convection

The **Convection** condition causes convective heat transfer to occur through one or more flat or curved faces (in contact with a fluid).

To define a convection load, specify a constant value for the heat transfer coefficient (also called the film coefficient). This value is based on the composition of the fluid in contact with the face, the geometry of the face, and the hydrodynamics of the fluid flow past the face. Then enter the external temperature, which is the temperature of the surrounding fluid.

This section contains the following topics:

18.7.2.1. Specifying a Convection Condition

To specify convection:

1. Select **Convection** from the dropdown menu of the **Thermal** tool in the **Physics** group of the **Simulation** tab, or by selecting the **Convection** tool in the **Thermal** condition HUD.
2. Select faces that are on the exterior of the model.

   If your model contains Named Selections, you can click **Advanced filters**, select **Link named selections**, and choose one from the list.

3. Specify the **Convection Coefficient** (or heat transfer coefficient).

   This value is based on the composition of the fluid in contact with the face, the geometry of the face, and the hydrodynamics of the fluid flow past the face.

4. Specify the **External temperature** of the surrounding fluid and press Enter.

18.7.2.2. Convection Tool Guides and Options

Convection Tool Guides

The following tool guides are available for the convection condition:

Select	Select a face to add a thermal condition. This is the default filtering option.
Advanced filters > **Link named selections**	To apply this condition to a named selection, select this option and then choose a named selection from the list.

Convection Tool Options

The following tool options are available for the convection condition:

Convection coefficient	The convection coefficient (or heat transfer coefficient). This value is based on the composition of the fluid in contact with the face, the geometry of the face, and the hydrodynamics of the fluid flow past the face.
Convection temperature	The external temperature of the surrounding fluid.

18.7.3. Heat

The Heat condition simulates the transmission across faces or bodies of a specified amount of heat energy. A positive value of heat will add energy to a body, whereas a negative value of heat will remove energy from a body.

You can apply heat to either a face or body as a total heat (in Watts), a heat per unit area (Watts/m^2), or heat per unit volume (Watts/m^3), depending on the type of geometry selected. For example, if you select a face, you can apply a total heat or heat per unit area, whereas if you select a body, you can apply a total heat or heat per unit volume.

When you specify a heat per unit area, the input value is multiplied by the area of the faces to determine the total heat. If you select multiple bodies, the input value is divided by the total volume of the bodies to get the total heat. When you apply heat per unit volume to multiple selected bodies, then the input value is multiplied by the volume of the bodies to determine the total heat.

This section contains the following topics:

18.7.3.1. Specifying a Heat Condition

To specify a heat condition:

1. Select **Heat** from the dropdown menu of the **Thermal** tool in the **Physics** group of the **Simulation** tab, or by selecting the **Heat** tool in the **Thermal** condition HUD.
2. Select one or more faces or bodies. A combination of faces and bodies is not permitted.

   To filter the selection by face, all faces of body, construction geometry, body, or physics region, click the **Advanced filters** tool. Only valid geometric entities will be available for selection. If your model contains Named Selections, click **Link named selections** to choose one from the list.

3. Specify the value for heat and press Enter. A positive value will transfer heat into the model (a heat source), where as a negative value will transfer heat out of the model (a heat sink).
   - Select **Total heat** to apply heat transfer to either a surface or body as a total heat (in Watts).
   - Select **Heat per unit area** to apply heat transfer to selected faces as a heat per unit area (W/m^2).
   - Select **Heat per unit volume** to apply heat transfer to selected bodies as a heat per unit volume (W/m^3).

18.7.3.2. Heat Tool Guides and Options

Heat Tool Guides

The following tool guides are available for the heat condition:

Select	Select a face to add a thermal condition. This is the default filtering option.
Advanced filters	Filter the selection by face, all faces of body, construction geometry, body, physics region, or named selection. Only valid geometric entities will be available for selection.

Heat Tool Options

The following options are available for the heat condition:

Total heat	Defines the heat transfer to either a surface or body as a **Total heat** (in Watts).
Heat per unit area	Defines the heat transfer to selected faces as a **Heat per unit area** (W/m^2).

Heat per unit volume	Defines the heat transfer to selected bodies as a **Heat per unit volume** (W/m^3).

18.7.4. Insulated

An insulated condition defines a perfectly insulated thermal condition where the heat flux across the boundary is zero. Basically, no heat will be gained or lost through the selected faces.

To define an insulated condition, select a face to define as insulated.

This section contains the following topics:

18.7.4.1. Specifying an Insulated Condition

To specify an insulated condition:

1. Select **Insulated** from the dropdown menu of the **Thermal** tool in the **Physics** group of the **Simulation** tab, or by selecting the **Insulated** tool in the **Thermal** condition HUD.
2. Select a face to define as insulated.

   An insulated thermal condition is where no heat will be gained or lost through the selected faces.

   If your model contains Named Selections, you can click **Advanced filters**, select **Link named selections**, and choose one from the list.
3. Click **Accept** to complete the setup.

18.7.4.2. Insulated Tool Guides and Options

Insulated Tool Guides

The following tool guides are available for the insulated condition:

Select	Select a face to add a thermal condition. This is the default filtering option.
Advanced filters > **Link named selections**	To apply this condition to a named selection, select this option and then choose a named selection from the list.

18.8. Fluid-Solid Heat Transfer

Fluid-solid heat transfer simulations model heat transfer within and between fluids and solids to allow analysis of temperature distribution in both regions. You can also combine fluid-solid heat transfer with electric conduction.

Example applications of fluid-solid heat transfer (also known as conjugate heat transfer) are simulations of:

- Heat exchangers
- Thermal management (engines, motors, electronic components, exhaust systems, and manifolds)
- Boilers, process heaters
- Free or forced convection simulation of busbars, fuses, conductors, electronic traces, etc.

The models can comprise multiple solid materials and separate flow paths with different fluid materials.

In fluid-solid heat transfer simulations, bodies are automatically identified as being solid or fluid based on the conditions or materials applied. Interfaces are automatically detected between fluid and solid bodies. These interfaces enable heat to be perfectly conducted between the fluid and solid bodies. The fluid side of the interface is treated as a non-slip wall, that is, zero velocity. Bonded contacts are detected between solid bodies, enabling conduction of heat between them. The effect of insulation between solid bodies can be modeled by converting a bonded to a excluded contact. Sliding contacts and other structural conditions cannot be modeled in combination with a fluid-solid heat transfer simulation.

There may be times when you run into issues while modeling fluid-solid heat transfer. The Discovery application issues warnings and messages to help you resolve the problems. The table below lists possible issues you might face and suggestions on how to resolve them.

Issue	**Resolution**
A fluid body is not included in the simulation, but it does not need to have any flow conditions applied. How can I add it to the physics?	Assign a fluid material to the body, then add it to the physics. If the whole fluid region is closed, that is, there are no flow conditions on any of its external faces, apply a wall boundary to a face. In this way convection due only to buoyancy can be simulated.
The interfaces and contacts are not created and there is no heat transfer between the bodies.	Check the geometry of the model for interference, such as overlapping volumes or gaps. Use **Show** > **Body Interference** in the **Display** tab to find areas of interference. Interference can also occur if the model has been made overly complex by the creation of multiple artificial bodies. To detect and fix overlapping volumes, use the **Interference tool**.
A fluid-solid interface includes a face shared between fluid bodies and there is no flow or the simulation cannot be solved.	The fluid face contacts faces of both solid and fluid bodies. Use the **Imprint tool** to split the face.
A face appears in both a fluid-solid interface and a contact and there is no heat transfer between the bodies.	The solid face contacts faces of both fluid and solid bodies. Use the **Imprint tool** to split the face.
Contact is detected between bodies, but the amount that they overlap is small, and the heat loss or gain could be better defined manually.	If imprinting or splitting the surface is not possible or appropriate, then exclude contact between the surfaces, and apply a solid-thermal condition to better model the heat transfer.
Contact is detected between bodies, but in reality there is a small insulating layer between them.	Convert the contact in this location to be excluded.
My model comprises layers of solid lining the inside of a pipe, but the fluid interface is detected by both the lining and the pipe.	Change the **Detection distance** for the fluid-solid interface. Click **>>** on the right side of the **Interface Review** HUD to see this option.
My model has many thin parts. How can I find out what parts are captured in the simulation?	Use **Resolution** in the **Simulation** tab ribbon to find and visualize the areas in the geometry that are not being captured in the simulation. Use in conjunction with **Size Preview**. Identified areas appear in red. Increase fidelity to include more geometry. This is available in **Explore**.

Issue	Resolution
The solid material in my model specifies an orthotropic thermal conductivity. The thermal conductance computed from a material does not recognize the orthotropic thermal conductivity value.	Thermal **Conductance** in the **Interface Review** tool will be calculated using the isotropic value of the material when specifying a **Thickness** and **Material**.
When modifying body positions inside an enclosure for fluid-solid heat transfer simulations, the application creates an unintended extra cavity.	Check for body interference before solving. This would help detect overlapping solid and fluid bodies before running the simulation.
When a body that is not designated as a cutter body or body of influence overlaps a fluid region, the simulation will arbitrarily assign the overlapping region as either fluid or solid. This can result in unexpected or erroneous outcomes.	Check for body interference before solving. This would help detect overlapping solid and fluid bodies before running the simulation.

18.8.1. Modeling Fluid-Solid Heat Transfer

To simulate fluid flow and heat transfer effects in and around structures:

1. Set up Fluid Flow Conditions on page 802 to define the inlets and outlets. Specify the temperature of the fluid flowing in.
2. The fluid material can be assigned to the body or bodies before applying the fluid conditions.
3. Set up Solid Thermal Conditions on page 828, such as Insulated or Temperature, to external surfaces of the solid. A Heat condition can be applied to a surface or a volume. Alternatively, simply select bodies to Include in Simulation if all the external surfaces are going to be convecting, the default Solid Thermal condition.

   All other inputs will be created automatically, these will show as default conditions in the tree, which you can review and adjust as needed.

   > **Note:** In **Explore**, if you define solid-thermal conditions at an interface, the interface conditions will take precedence.

4. Optionally, set up Current and Voltage conditions.

   > **Note:** In **Explore**, if you define electric conditions at an interface, the interface conditions will take precedence.

5. Optionally, set one or more solid bodies as cutter bodies to automatically block fluid flow.

   Right-click the body in the geometry or model tree and select **Cutter Body** (only available in **Explore**) from the context menu.

   For overlapping bodies, right-click the body in the geometry or model tree and select:

   - **Overlapping Bodies** > **Set as Cutter Bodies** to set the body as a cutter body
   - **Overlapping Bodies** > **Exclude from Simulation** to exclude the body from the simulation entirely
   - **Overlapping Bodies** > **Identify** to launch the Interference tool, which detects and removes interference from clashing bodies

> **Note:** If a body not designated as a cutter body or body of influence overlaps a fluid region, the simulation will arbitrarily assign the overlapping region as either fluid or solid. This can result in unexpected or erroneous outcomes. Check for body interference before solving. This would help detect overlapping solid and fluid bodies before running the simulation.

18.8.2. Reviewing Fluid-Solid Interfaces

The Discovery application automatically detects and defines fluid-solid interfaces when you simulate heat transfer within and between fluids and solids. These fluid-solid interfaces can be reviewed by selecting the automatically created condition under **Fluid-Solid Interfaces** in the physics tree. Double-click the interface in the physics tree to load the **Interface Review** tool. Each interface pair is shown in the model as a single dot.

1. Review the automatically generated interfaces between the fluid and solid regions. The total number of generated interface pairs appears on the right side of the HUD. Each interface pair is shown in the model as a single small marker.
2. Hover over one of the spherical identifiers, the application highlights the surface of the interface pair. Select an identifier to show the interface # and total number of interfaces.

   > **Note:** Selecting an additional marker does not deselect previously selected pairs.

3. Select the interface pair you want to review.

   You can filter to show all interface pairs on selected locations by selecting the **Select a primary body** or **Select a primary face** filter and then selecting one or more locations.

   You can narrow the review further by selecting the **Select a secondary body** filter or **Select a secondary face** filter and a location to highlight and select only the interface pairs between this location and the primary locations. You can also ALT-select the secondary locations.

4. (Optional) To exclude an interface from automatic detection, select the dot representing the interface and click **Exclude from interface detection**.

   Excluded interfaces will be treated as insulated. If you wish to apply a different condition, you must apply conditions to both sides of the interface.

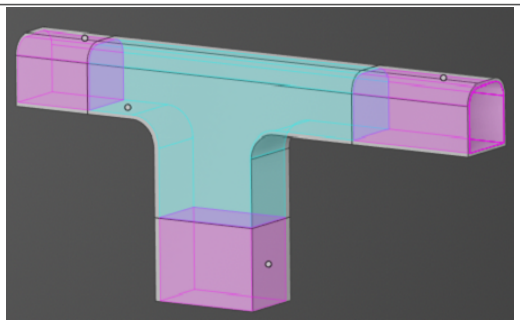
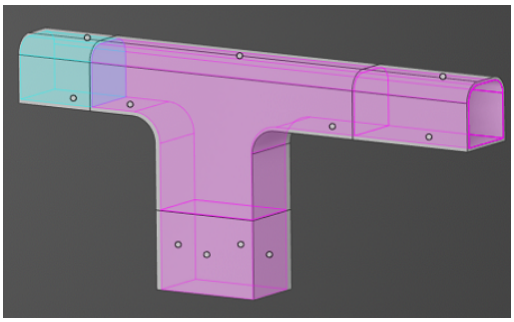
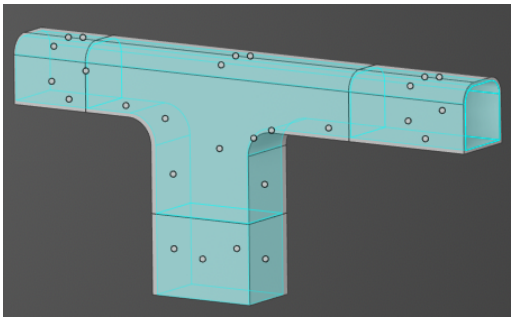
5. (Optional) Specify the maximum distance of interface detection to be used in the **Max** field for **Detection distance**. For example, for a thin-walled pipe, you could set this distance to ensure that interfaces are not inadvertently included between the contained fluid and the outer pipe surface.
6. Optional) Specify in **Face overlap allowance** the percentage that faces are allowed to overlap during interface detection. The default for this field is set in the Physics Defaults.

   This option is particularly useful when you have very small contact points, especially near curved edges.

7. By default, interface faces are grouped by body. You can choose to group them by other topologies, like tangent faces, or to apply no grouping so that interfaces are created on all interface faces.

   Here is an example of a model consisting of three bodies showing the different interface face groupings:

Option	Description
Group faces by body	*Four (4) interface pairs*

Option	Description
	
Group tangent faces	*Eleven (11) interface pairs* 
No grouping	*Twenty-seven (27) interface pairs* 

8. Additionally, under **Thermal Conductance Definition**, select a method for defining conductance between the fluid and solid region.
   - Select **Perfect conductor** to transfer heat between the two surfaces without any resistance.
   - Select **Perfect insulator** to define near zero conduction of heat transfer between two surfaces.
   - Select **Specified conductance** to specify the conductive heat transfer as a value.
   - Select **Thickness and material** where the Discovery application then uses them to compute the conductance.
   - Select **Thickness and conductivity** where the Discovery application then uses them to compute the conductance.

   > **Note:** You can set only one value of thermal conductance between a particular pair of fluid and solid regions. However, different values can be specified for separate flow paths.

   For example, you might specify thermal conductance to represent reduction in heat transfer due to fouling.

CAUTION: When an interface is excluded or removed by the suppression of the solid thermal physics region, then the thermal conductance is not retained if the action is undone.

Tip:

- When reviewing fluid-solid interfaces, if you find interface pairs that overlap with contacts, you may need to use the imprint tool to split the faces, or to merge the solid bodies to ensure that the fluid-solid heat transfer is correctly modeled.
- In a fluid-solid heat transfer simulation, the different flow paths are represented by different colors. The color is based on the defined inflow fluid temperature, with the hottest being red, and the coolest blue.

9. Optionally, include internal radiation if you want to include the effects of radiative heat transfer and specify the internal emissivity.

18.8.3. Fluid-Solid Interfaces Tool Guides and Options

Fluid-Solid Interfaces Tool Guides

The following tool guides are available:

Select a face	Select a face for which you want to review all interface pairs on the primary face.
Select a body	Select a body for which you want to review all interface pairs on the primary body.
Select a secondary face	Select a secondary face to review only interface pairs between the secondary face and any primary face.
Select a secondary body	Select a secondary body to review only interface pairs between the secondary body and any primary body.
Select markers	Select markers to highlight individual interface pairs.

Fluid-Solid Interfaces Tool Options

The following options are available:

Exclude from interface detection	Exclude the interface pairs from interface detection.
Include internal radiation	Switch internal radiation on or off.
Detection distance	Specify the maximum distance of interface detection to be used. For example, for a thin-walled pipe, you could set this distance to ensure that interfaces are not inadvertently included between the contained fluid and the outer pipe surface.

Face overlap allowance	Specify the percentage that faces are allowed to overlap during interface detection. The default for this field is set in the Physics Defaults. This option is particularly useful when you have very small contact points, especially near curved edges.
Group faces by body	View interfaces grouped by the body and optionally create an excluded interface per grouped body.
Group tangent faces	View interfaces grouped by tangent faces and optionally create a excluded interface per grouped topologies, such as groups of tangent faces.
No grouping	View individual interface pairs and optionally create an excluded interface for the desired separate face pairs.
Internal emissivity	Internal emissivity is the measure of a material's ability to emit heat by radiation from within itself, or from a surface that is surrounded by the same material. It is a dimensionless quantity that ranges from 0 to 1. A value of 0 means radiation is reflected and 1 means radiation is absorbed. Blackbody surfaces have an emissivity of 1.
Define Thermal Conductance	Choose how to define heat transfer between the fluid and solid region. You can specify the thermal **Conductance** as a value, or you can specify a **Material** along with the material **Thickness** or **Conductivity**, which the Discovery application then uses to compute the thermal **Conductance**. Choose **Perfect conductor** if all the heat is transferred between the fluid and solid region or **Perfect insulator** to create a near zero conductance of the material.

18.9. Electromagnetics Conditions

Conditions for electromagnetics physics include:

- **Circuit Port** - A circuit port consists of current source along with a user-defined parallel impedance load.
- **Mode Port** - Mode ports are intended for transmission lines with a disc-like surface. These represent connecting conductors across the cross-section of the end of a transmission line. The cross-section can be closed (like a coaxial cable with concentric circles) or open (like a microstrip or trace) but either case should have air or vacuum on one side.

If circuit port or mode port are created, then an electromagnetic simulation can be run. Voltage and current conditions are disabled.

Electromagnetics conditions can only be set up and run in the Refine stage, To apply an electromagnetics condition to your model:

- Select **Electromagnetics** in the ribbon of the **Simulation** tab and choose the condition type–either circuit port or mode port.
- For circuit port or mode port: Select the first edge to create the port. As you hover over, a large yellow arrow appears. This shows the direction towards the next allowable edge. Alt-select if you want to choose a different second edge. Press Enter.

Impedance, magnitude and phase properties can be parameterized for **Circuit Port** and **Mode Port**. To parameterize **Impedance** in the physics tree, click the **Add as a parameter** button. A blue triangle on top implies that the value is parameterized. To parameterize **Magnitude** and **Phase**, click inside the input field and then click the **Parameterize** button. A highlighted parametrize button on top implies that the value is parameterized. All parameters can be viewed in the **Parameters** table.

Use the **Size Preview** option to visually review any user-defined or automatically generated local fidelities. This makes it easier to preview grid size controls before running the simulation.

This section contains the following topics:

18.9.1. Creating a Circuit Port

1. Select the first edge to create the port.

   As you hover over, a large yellow arrow appears. This shows the direction towards the next allowable edge.
2. Alt-select if you want to choose a different second edge.
3. Accept the default impedance value or modify it and press Enter.

   An electromagnetic region is created. See the physics tree for more details. The edges must lie in the same Cartesian plane. The face in between the edges can lie on the boundary of two bodies, or it can cut through an unrelated solid. However, it cannot lie along or cut through a conductor.

18.9.2. Creating a Mode Port

1. Select the first edge to create the port.

   As you hover over, a large yellow arrow appears. This shows the direction towards the next allowable edge.
2. Alt-select if you want to choose a different second edge. Either one or both edges can be interior to the face.
3. Accept the default impedance value or modify it and press Enter.

   An electromagnetic region is created. See the physics tree for more details. The edges must lie in the same Cartesian plane. The face in between the edges can lie on the boundary of two bodies, or it can cut through an unrelated solid. However, it cannot lie along or cut through a conductor.

18.9.3. Electromagnetics Tools and Options

The following tools are available for frequency-response (FDTD) electromagnetics simulations:

Select a circuit port	Select the conducting edges to define the circuit port.
Select a mode port	Select the conducting edges to define the mode port.
Select secondary face	Select a secondary face to determine impedance.
Select markers	Select markers to highlight individual ports.

18.9.4. Electromagnetics Best Practices

Best practices for setting the values for the frequency-response (FDTD) electromagnetics physics include:

- **Minimum size** for **Global** fidelity should be the size of the metallic features that you would like to resolve throughout the simulation region. For example, for traces that are 1mm in width or a helical antenna made from a wire with a diameter of 1mm, set this field to 1mm at the largest, and most safely to about 0.8mm (80% width or diameter). The global feature size should be used if all the metal geometry in the simulation has a similar feature size.
- **Local Fidelity** can be defined for metal geometry with small features requiring resolution in certain regions. You can use the same rules you set for global **Minimum size** but you only need to apply them to the required geometry. Using local fidelity can make the simulation run faster and consume less memory than setting a smaller global

Minimum size. For example, if you have a patch antenna with traces that are 1mm in width, but one small region has 0.25mm wide traces, then it is best practice to use 1mm **Minimum size** and 0.25mm **Local Fidelity**.

- Thin, axis-aligned traces ideally should be modeled as 2D surface bodies. However, if you choose to use 3D geometry, you do not need to reduce the minimum feature size (global or local) to the physical thickness of the trace, but only consider the width of the traces (and the proximity of different traces) when setting the minimum feature size.
- Thin, non-planar or non-axis aligned structures are currently implemented as axis-aligned 2D structures. Therefore, if the structure is not axis-aligned, you must treat it as 3D geometry and must set a local fidelity equal to 80-100% of the thickness of the structure. It is assumed that the thickness is the smallest feature of the structure.
- Circuit ports do not need local fidelities unless those ports have a width less than the global **Minimum size**. In such cases, higher accuracy is obtained by setting local fidelity to be the same as the width of the port.
- Scaling of simulation time with the number of ports is optimal for antenna structures with a small number of ports, since the total time to solve all ports scales with the number of ports.
  - For devices with a high degree of symmetry, where the performance can be characterized only by considering one column (or a few columns) of the S-matrix, only one port (or a subset of ports) should be excited during initial exploration of the design, which can be achieved by making some of the ports passive. After initial exploration, the full S-matrix can be calculated.
- **Maximum simulation duration** should be the largest of $50/f_{min}$ and $2Q/f_{res}$ where:
  - f_{min} is the minimum frequency specified in the properties of your electromagnetic region,
  - Q is the quality factor, and
  - f_{res} is the resonant frequency of any resonant mode of your device.

  Since Q and f_{res} may not be known prior to simulation, envision Q as the maximum value you want resolved in the results.
- **Target numerical convergence** should rarely be changed from the default of 1e-5. The only exception is for very high-quality factor resonances in a broadband simulation, where it may be as low as 1e-8.

18.10. Electric Conditions

Conditions for electric physics include:

- **Voltage** - Voltage condition can only be applied to faces and only allows for primary selection. It simulates the application of an electric potential. Circuit and mode port are disabled when voltage is selected.
- **Current** - Current condition can only be applied to faces and only allows for primary selection. It simulates the application of an electric current. Circuit and mode port are disabled when current is selected.

Electric simulations can proceed when voltage or current are established. Electric conditions are supported with thermal conditions or in a fluid-solid heat transfer simulation but are not supported in combination with structural conditions.

To apply an electric condition to your model:

- Select **Electromagnetics** in the ribbon of the **Simulation** tab and choose the condition type–either current or voltage.
- Select one or more faces to create the voltage or current source. Specify a value and press Enter.

18.10.1. Creating a Voltage or Current Source

1. Select one or more faces to apply the voltage or current.
2. Specify a value and press Enter.

An electrical simulation is set up. It is considered valid only if a minimum of two electrical parameters are present, and voltage must be one of them.

Possible cases can involve :

- A pair of unique voltage levels forming a voltage differential, which drives the flow of current.
- A combined state of voltage and current, prompting flow of current due to these conditions.

19: Simulation Options

The Discovery application automatically adjusts solution options and applies appropriate modeling methods based on the physics inputs.

You can specify the solution options for a simulation or explicitly set the modeling method by clicking **Simulation Options** in the ribbon of the **Simulation** tab.

- **Units settings** sets the units. Any change to these settings applies to the current session for all simulations. Changes to these units do not affect units from the status bar or those under **File > Settings > Units and Display Precision**.
- **Include newly created bodies** includes in the simulation by default all new bodies created in the model. Switch off this option if you want to suppress physics.
- **Additional [Physics] Options** displays the simulation options for each physics type.

You can also set up a natural frequency or modal analysis, which enables you to model the undamped, free vibration characteristics of a structure by determining natural frequencies and mode shapes, or a topology optimization analysis, which you can use to compute an optimal structural design for a selected region of your model.

For information about problems and error messages that you may encounter, see **Troubleshooting**.

> **Note:** Many options are not available in the **Explore** stage. You must be in **Refine** to see all the available options.

> **Note:** Electromagnetics simulation options are for port analysis only. They are not available for simulations with voltage and current applied.

This section contains the following topics:

Structural Simulation Options

Specify modeling method	Choose between setting the modeling method to be **Non-linear**, in which case large deformations will be captured, or setting the method to be **Linear**, in which case sliding contact will be idealized and large deformation will not be included.
Specify solution progression	Specify the substep progression in a nonlinear simulation.
Constant damping ratio	For a random vibration simulation, you can change the default value for the Constant damping ratio, which specifies a consistent level of damping throughout the structure's response.
Mesh Quality	Select any mesh quality criteria you want to monitor, then specify upper and lower limits. After meshing, you are notified if any selected meshing criteria are not met. Meshing criteria available include **Element quality**, **Jacobian ratio**, and **Aspect ratio**. **Jacobian ratio** is on by default, but you can choose different defaults in the **Physics Settings**.

Fluid Simulation Options

Specify calculation type	Choose between performing a **Static or steady-state** or **Time-dependent** calculation. Steady-state is used to determine the long-term behavior of a system that does not change over time. Time-dependent is used to follow behavior over a specified duration.
Stop on	Choose **Engineering convergence**, **Numerical convergence**, **Monitored value convergence** or **Solution stopping criterion** to determine the stopping criteria for when the problem has converged.
Use LiveGX solver	On by default when **Enable Fluent solver option** is turned on in the **Physics Settings**. Use LiveGX solver to leverage GPU solver technology combined with body-fitted meshing to provide both fast and accurate fluid solutions. Switch off **Use LiveGX solver** to run the simulation using the Fluent solver.
LiveGX solver mode	Choose between the **GPU** and **CPU** solvers. Use the GPU solver for a quick solution. If the demands of a simulation exceed the graphics card resources available, select CPU instead to access the system RAM, which is generally larger than the GPU memory.
Fluids solver	**Program Controlled** is the default selection, allowing the application to determine the most appropriate pressure-based solver for your simulation setup. Otherwise, you can choose **SIMPLE** (a segregated algorithm) for steady-state, incompressible simulations or **Coupled** for time-dependent, compressible flows.
Specify modeling method	Choose one of the turbulence models if modeling turbulent flow. Alternatively, if the flow is not turbulent, set the fluid flow modeling method to Laminar. Learn more about the different turbulence models.
Specify convergence settings	Specify the criteria for convergence. This is the point at which further iterations are no longer necessary. An important part of determining whether your solution is appropriate is to judge whether the solution has converged to an acceptable level.
Specify meshing method	Choose between two meshing methods, **Tetrahedra mesh** and **Polyhedra mesh**. Your selection should take into consideration the complexity of your model, the solution accuracy you would like to achieve, and the computational expense.
Specify capture sharp edges (Explore only)	**Program Controlled** is the default selection, allowing the application to determine how to best capture sharp edges on your model. You can change the setting to capture **All sharp edges**. This option attempts to capture all sharp edges in the mesh leading to increased accuracy. This can increase computation time and decrease solution stability. Otherwise, choose **None** to avoid capturing sharp edges. **Note: Program Controlled** and **All sharp edges** do not apply to faceted geometries; instead, **None** is used. Additionally, if you select **Program Controlled** and **All sharp edges**, you will not be able to use fidelity **Resolution** to display areas in the geometry not covered by the simulation.
Auto-adjust quality in low fidelity areas	The application automatically improves mesh quality where needed by repositioning nodes.
Mass flow-weighted average for Monitors	The application generally uses an area-weighted averaging calculation for all results monitors. Use **Mass flow-weighted averaging** instead for some results variables.
Reference pressure	Enter a reference pressure to be used in the calculation of the force on a wall. You can assign a different default value in the **Physics Settings**.

Mesh Quality	Select any mesh quality criteria you want to monitor, then specify upper and lower limits. After meshing, you are notified if any selected meshing criteria are not met. Meshing criteria available include **Orthogonal quality**, **Skewness**, and **Aspect ratio**. **Orthogonal quality** is on by default, but you can choose different defaults in the **Physics Settings**.
Capture thin structures (Explore only)	Accurately represents mesh areas with thicknesses smaller than the local size, resulting in anisotropic cells in these thin regions and reducing the total cell count. Stretched (anisotropic) cells are created along thin directions, which can help lower the cell count compared to isotropic refinement. If this option is not enabled, isotropic cells are generated and very thin regions are merged or overlooked, possibly leading to a loss of detail.

Solid Thermal Simulation Options

Specify calculation type	Choose between performing a **Static or steady-state** or **Time-dependent** calculation. Steady-state is used to determine the long-term behavior of a system that does not change over time. Time-dependent is used to follow behavior over a specified duration.
Specify modeling method	Choose between setting the modeling method to be **Non-linear**, in which case large deformations will be captured, or setting the method to be **Linear**, in which case sliding contact will be idealized and large deformation will not be included.
Specify solution progression	Specify the substep progression in a nonlinear simulation.
Mesh Quality	Select any mesh quality criteria you want to monitor, then specify upper and lower limits. After meshing, you are notified if any selected meshing criteria are not met. Meshing criteria available include **Element quality**, **Jacobian ratio**, and **Aspect ratio**. **Jacobian ratio** is on by default, but you can choose different defaults in the **Physics Settings**.

Electromagnetics Simulation Options

Modify frequency points	Specify the **Far-field**, **Near-field** and **S-parameter** frequency points.
Specify loss settings	Specify whether **Dielectric** losses should be **Based on materials** or if they can be **Ignored**. Specify the **Perfect conductivity threshold** value to set the material threshold for electrical conductivity.
Specify convergence settings	Specify the **Maximum simulation duration** for each simulation. The simulation is considered converged once the volume-integrated electric field intensity drops below a ratio of its peak value. Enter the **Target numerical convergence** to specify this threshold.

19.1. Changing Simulation Options Default Settings

The Discovery application allows the default simulation options to be overridden, without being specified for each simulation. This is useful if, for example, your organizational process requires specific physics models or convergence criteria to be used for all (or most) simulations, which don't match the Ansys defaults.

To define different default simulation options for all future simulations, go to **File menu** > **Settings** > **Physics** and make the changes under **Simulation Options Defaults**. These defaults will apply after the next update to the solution,

however, any defaults for user specified simulation options will be set for newly created simulations. Any changes made to the default settings can be reverted to the original default values using the **Reset Simulation Options Settings** button.

19.2. Calculation Types

Determine how calculations are performed in your simulation. By default, the Discovery application will automatically determine the calculation type based on the physics specifications in your simulation. If you prefer to explicitly set the calculation type, go to **Simulation Options** in the ribbon of the **Simulation** tab and click **Specify calculation type**. Choose between performing a **Static or steady-state** or **Time-dependent** calculation.

Select **Static or steady-state** to determine the long-term behavior of a system that does not change over time.

Select **Time-dependent** to follow behavior over a specified duration (also called time-history analysis or transient simulation). You can then specify the simulation duration in seconds. If you do not specify a duration, the simulation will continue until interrupted. A time-dependent analysis, by definition, involves conditions that are a function of time.

The following table provides a description of different steady-state (static) and time-dependent physics applications.

Physics	Calculation Type	Description
Structural	Static	Used to determine the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed. That is, the loads and the structure's response are assumed to vary slowly with respect to time.
Thermal	Steady-state	Used to determine temperatures, thermal gradients, heat flow rates, and heat fluxes in an object that are caused by thermal loads that do not vary over time. A steady-state thermal analysis calculates the effects of steady thermal loads on a system or component. Engineers often perform a steady-state analysis before performing a time-dependent thermal analysis, to help establish initial conditions.
Fluid flow	Steady-state	Used to determine fluid velocities, pressures and temperatures caused by fluid and thermal conditions that do not vary over time. A steady-state fluid analysis calculates the effects of steady fluid and thermal conditions for either internal or external fluid domains. Engineers often perform a steady-state fluid flow analysis to determine flow characteristics (velocity, temperature and pressure) that do not vary with time.

Physics	Calculation Type	Description
Thermal	Time-dependent	Used to determine temperatures and other thermal quantities that vary over time. Many applications may involve the variation of temperature distribution over time, such as with cooling of electronic packages or a quenching analysis for heat treatment. Also of interest are the temperature distribution results in thermal stresses that can cause failure. In such cases the temperatures from a time-dependent thermal analysis can be used as inputs to a structural analysis for thermal stress evaluations. Many heat transfer applications such as heat treatment problems, electronic package design, nozzles, engine blocks, pressure vessels, fluid-structure interaction problems, and so on involve time-dependent thermal analyses.
Fluid flow	Time-dependent	Used to simulate systems where the flow behavior varies over time. Enabling time dependence is sometimes useful when attempting to solve steady-state problems that tend toward instability (for example, natural convection problems in which the Rayleigh number is close to the transition region). It is possible in many cases to reach a steady-state solution by integrating the time-dependent equations.
Electric	Steady-state	Used to determine the electric potential within a conducting body when external voltage or current loads are applied. Based on the solution, additional results such as current density, voltage, and joule heating can be calculated.
Electromagnetic	Steady-state	Used in antenna design, with near-field and far-field patterns shown for radiated electric fields, gain or other antenna parameters.

Physics Combinations and Calculation Types on page 37 provide a summary of the physics available based on the calculation type and simulation stage.

In addition, for a structural simulation, a natural frequency or modal analysis is available.

19.3. Structural Linear and Non-Linear Modeling

By default, the Discovery application automatically determines the structural modeling method for you based on the physics inputs. If frictional or frictionless contacts are present in the model, the application automatically invokes a large strain solution. Otherwise, the application uses a small strain solution. You can explicitly set the modeling method by going to **Simulation Options** in the ribbon of the **Simulation** tab and modifying the **Additional Structural Options**. This option only affects solutions when in the **Refine** stage.

Select **Specify modeling method** to override the method set by the application. If you set the modeling method to **Non-linear**, it activates a large strain solution, which captures large strains, large deformations and rotations. If you set the method to **Linear**, it activates a small strain solution, and frictional and frictionless contacts are idealized as linear contacts.

> **Note:** For a static structural analysis, a large strain solution includes large strain, large deflection, and large rotation. Stress stiffening effects are also captured by a large strain solution.

For a pre-stressed modal analysis, the application computes the static pre-stress condition based on a small strain solution and includes both stress stiffening and spin softening effects for the computation of the pre-stressed modes.

19.4. Modal Structural Simulations

Modal analysis enables you to model the undamped, free vibration characteristics of a structure by determining natural frequencies and mode shapes. You can then design your structure to avoid resonant frequencies or to vibrate at a specific frequency. Note that, although the results of a modal analysis include displacement results, you should only use these displacements to visualize the mode shape. The natural frequency and mode shape results are based on unspecified dynamic loads, so the results do not include absolute displacement values.

You can also include the pre-stress effects of a static analysis in your modal analysis. A typical example is the large tensile stress induced in a turbine blade under a centrifugal load that can be captured by a static structural analysis. Because this causes significant stiffening of the blade, including this pre-stress effect will result in much higher, realistic natural frequencies in a modal analysis.

Note: The Discovery application assumes a linear pre-stress condition for pre-stressed modal analysis. All non-linear behavior is treated as linear for the pre-stress condition. For the purpose of computing the pre-stress condition, sliding frictional contacts are assumed to be bonded, and sliding frictionless contacts are assumed to be idealized sliding.

Note: Modal (Natural Frequency) analysis is also used as the basis for several of the Topology Optimization goals (Maximize Natural Frequency, Balance Stiffness and Frequency, and Target Natural Frequency).

To specify a modal analysis:

1. Select the **Natural Frequency** in the ribbon of the **Simulation** tab.
2. Specify the **Number of Modes** in the physics tree. You can specify up to 12 modes.
3. Set up any constraints, including forces, supports, displacements, and mass conditions.

The specified number of mode shapes are calculated, with Von Mises Stress and Displacement Magnitude results calculated for each mode. Each is shown as a separate result. The results for mode 1 are displayed by default, but you can select a different mode from the legend. You can calculate results for each frequency of a selected mode shape or for all modes at once.

Note: Stress and strain results are not available by default for modal analyses. You can turn on those results in the Results Settings.

19.5. Topology Optimization

With a Topology Optimization analysis, you can compute an optimal structural design for a selected region of your model with specific design goals and constraints. Topology optimization is a physics driven optimization that is based on a set of loads and constraints.

Currently, you can use optimization to either maximize the strength of your topology, minimize the response to free vibration, or both. You can also target a specified frequency, minimize volume, or minimize stress.

> **Note:** For Topology Optimization, the cylindrical surfaces of bolts are considered protected regions, and bolt assignments are included in the simulation. However, bolt preloads are ignored.

> **Attention:** The Discovery solver uses an elemental method to calculate factor of safety for the Minimize Volume optimization goal. However, the monitor factor of safety uses a nodal calculation. As a result, the resulting monitor factor of safety may appear to be lower than the value for the minimum factor of safety. To ensure that the factor of safety constraint has been met, view the `%TEMP%/Discovery/{sessionId}/Logs/LiveTopologyOptimization_{sessionId}` file. If the complete optimization run is successful, SOLUTION FOUND FOR THE OPTIMIZATION PROBLEM appears at the end of the file.

You can find a course about topology optimization on the Ansys Innovation Space.

This section contains the following topics:

19.5.1. Optimizing for Strength or Response to Free Vibration

You can use optimization to either maximize the strength of your topology, minimize the response to free vibration, or both.

1. In Explore mode, set up any structural conditions. Possible recommended conditions when optimizing to maximize strength include forces, moments, pressures, supports, and displacements. Possible recommended conditions when optimizing to minimize the response to free vibration include supports.
2. In the ribbon, click **Topology Optimization**.
3. In the physics tree, double-click **Optimization**.
4. In the HUD, you can optionally select bodies to optimize. All bodies are optimized by default. To select specific bodies:
   a) Click the **Select bodies to optimize** tool.
   b) Select one or more bodies you want to optimize.
   c) Click **Accept**.

   You can default back to including all bodies in your optimization by clicking the **Optimize all simulation bodies** tool.
5. Select the goal for your optimization. You can:
   - **Maximize Stiffness**: This option uses iterative static structural analysis to maximize stiffness and minimize compliance. It reduces the volume or mass but maintains the maximum stiffness possible for the remaining structure.
   - **Maximize Natural Frequency**: This option uses iterative natural frequency analysis to minimize the response to frequency vibration.
   - **Balance Stiffness and Frequency**: This option uses a combination of static structural and natural frequency analysis to both maximize stiffness and minimize the response to frequency vibration.
6. In the tool options, for **Volume reduction**, specify the percentage of distance surrounding applied conditions to be protected during optimization.

   Additionally, you can also select **Reduce by volume** and enter the **Volume reduction** as a distance value.

7. Additionally, you can specify the following optional optimization conditions:

Option	Description
Min Thickness	Select to specify the minimum size of connected parts. Enter the **Minimum thickness** value in the physics tree.
Max thickness	Select to specify the maximum size of connected parts. Enter the **Maximum thickness** value in the physics tree.
Pull direction	For mold-based manufacturing processes, enable this option to define the direction that the optimized product will be removed from the mold. Then define the pull direction by double-clicking **Pull direction** in the physics tree.
Table direction	For a multi-axis milling process, enable this option if you want to define the direction from which approach to the machine is blocked. Then define the table direction by double-clicking **Table direction** in the physics tree.
Overhang prevention	For a 3D printing process, enable this option if you want to define the growing direction of the manufacturing process. Then define overhang prevention by double-clicking **Overhang prevention** in the Physics Tree.

8. Specify the distance surrounding applied conditions to be protected during optimization, as well as a protected distance for any specific faces. You can also select any additional geometry you want protected.
9. Optionally, set a Symmetry constraint to ensure that the model is dimensionally reduced based on a mirror plane.
   a) Click **Symmetry** in the ribbon.
   b) Select one or more flat faces in the same plane. You can select multiple faces, even if they are on bodies with different physics applied.
   c) Click **Accept** to complete your setup.
10. Optionally, define multiple load cases.
   a) Add physics conditions as parameters by clicking the **Add as a parameter** tool next to the condition in the Physics Tree.
   b) View the Design Variations table by clicking the **Design Variations** icon on the SID.
   a) In the Load Cases tab of the Design Variations table, click **Add** to define load cases.

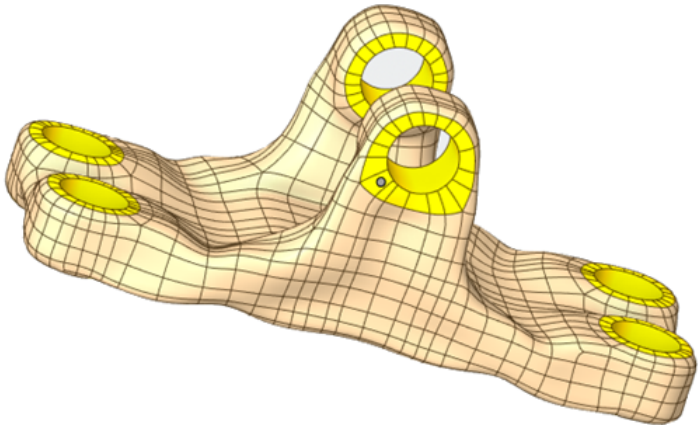
> **Note:** Any design variations set up in Refine mode are overwritten when Test Cases are set in Topology Optimization.

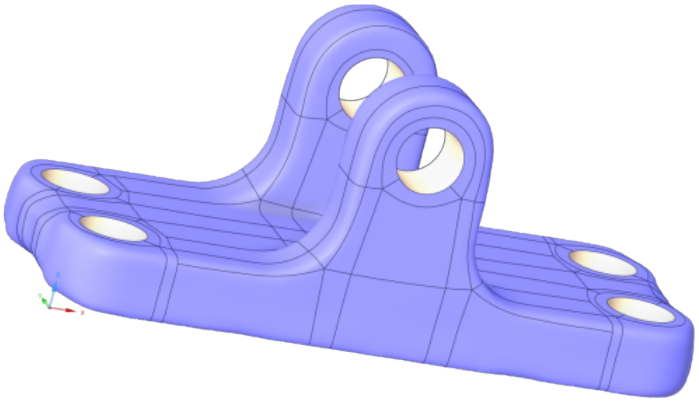
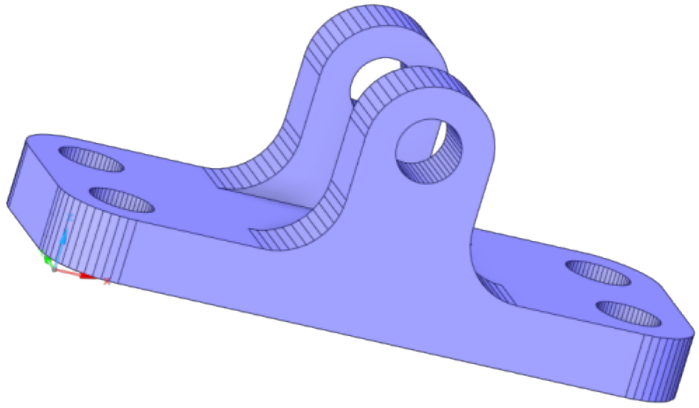
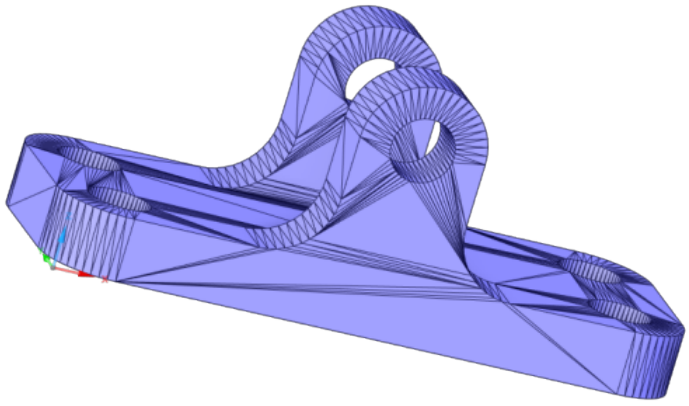
11. Automatic mesh refinement is off by default, but you can turn it on to automatically refine the mesh during optimization.
   a) From the **Simulation** tab ribbon, select the **Global** dropdown menu. The Global fidelity options appear.
   b) For **Automatic mesh refinement**, click **Off** and select **Program controlled** instead.
12. Solve. You can create several different sets of conditions and solve several times.

> **CAUTION:** If you edit the topology optimization conditions during the solve, the optimization tries to adapt the current shape to meet the new conditions. While this can be significantly faster than restarting from the initial geometry, it may result in a different optimized shape than the one calculated directly from the initial geometry. If you prefer to optimize from the initial geometry instead, edit the condition values as desired, then click **Reset** in the Results Arc to restart the simulation. Note that if you edit the

geometry, structural conditions, or simulation conditions, the Discovery application always restarts the optimization from the initial geometry.

13. Once you have a faceted geometry result you want to save, click **Add optimized body to model** in the Results Arc to save it to the model. The new faceted geometry is now listed in the Model tree.
14. Right-click the faceted body in the Model tree to perform any of the following actions:

Option	Description
Create Validation Simulation	Once the topo opt simulation is complete, and you have created a faceted body of the resulting optimized shape, select this option to create a new simulation that includes all of the physics conditions from the topology optimization simulation applied to the faceted body. This could be a single test or multiple test cases depending on the topo opt setup. This option automatically creates a validation simulation of the faceted geometry created from the topology optimization simulation.
Smooth Facets	Select this option to smooth your shape. A new faceted model appears in the Model tree.
Convert to SubD hybrid	Select this option to convert your faceted shape to a hybrid model and add it to the Model tree. The hybrid model contains both subdivision surfaces for regions of the model where you want to allow for flexible and freeform editing and traditional surfaces such as planes or cylinders for regions requiring precise topology.  Once you have created the hybrid model of the resulting optimized shape, you can right-click the hybrid model in the Model tree and select the **Create Validation Simulation** option to create a validation simulation that includes all of the physics conditions from the topology optimization simulation applied to the hybrid model.
Convert to Solid > Fit Surfaces	Select this option to smooth the facets, fit surfaces onto the facets, then create a solid body by stitching those patches together whenever possible. The patched body is then added to the Model tree.

Option	Description
	
Convert to Solid > Mostly Tessellated	Select this option to convert your faceted shape to a solid model and add it to the Model tree, with the facets in flat areas merged together. As seen in this image, that means that curved areas will still have a faceted shape. 
Convert to Solid > Fully Tessellated	Select this option to convert your faceted shape to a solid model and add it to the Model tree, with the facets converted to faces on the solid. **Note:** The more facets there are on the shape, the longer the conversion will take. 

19.5.2. Optimizing for a Specified Frequency

The Target Natural Frequency option uses iterative natural frequency analysis to optimize at a target specified frequency.

1. In Explore mode, set up any structural conditions. Possible recommended conditions when optimizing to maximize strength include forces, moments, pressures, supports, and displacements. Possible recommended conditions when optimizing to minimize the response to free vibration include supports.
2. In the ribbon, click **Topology Optimization**.
3. In the physics tree, double-click **Optimization**.
4. In the HUD, you can optionally select bodies to optimize. All bodies are optimized by default. To select specific bodies:
   a) Click the **Select bodies to optimize** tool.
   b) Select one or more bodies you want to optimize.
   c) Click **Accept**.

   You can default back to including all bodies in your optimization by clicking the **Optimize all simulation bodies** tool.
5. Select **Maximize Natural Frequency** as the goal for your optimization.
6. In the tool options, enter the **Target frequency**.
7. Additionally, you can specify the following optional optimization conditions:

Option	Description
Min Thickness	Select to specify the minimum size of connected parts. Enter the **Minimum thickness** value in the physics tree.
Max thickness	Select to specify the maximum size of connected parts. Enter the **Maximum thickness** value in the physics tree.
Pull direction	For mold-based manufacturing processes, enable this option to define the direction that the optimized product will be removed from the mold. Then define the pull direction by double-clicking **Pull direction** in the physics tree.
Table direction	For a multi-axis milling process, enable this option if you want to define the direction from which approach to the machine is blocked. Then define the table direction by double-clicking **Table direction** in the physics tree.
Overhang prevention	For a 3D printing process, enable this option if you want to define the growing direction of the manufacturing process. Then define overhang prevention by double-clicking **Overhang prevention** in the Physics Tree.

8. Specify the distance surrounding applied conditions to be protected during optimization, as well as a protected distance for any specific faces. You can also select any additional geometry you want protected.
9. Optionally, set a Symmetry constraint to ensure that the model is dimensionally reduced based on a mirror plane.
   a) Click **Symmetry** in the ribbon.
   b) Select one or more flat faces in the same plane. You can select multiple faces, even if they are on bodies with different physics applied.
   c) Click **Accept** to complete your setup.
10. Optionally, define multiple load cases.
    a) Add physics conditions as parameters by clicking the **Add as a parameter** tool next to the condition in the Physics Tree.
    b) View the Design Variations table by clicking the **Design Variations** icon on the SID.

a) In the Load Cases tab of the Design Variations table, click **Add** to define load cases.

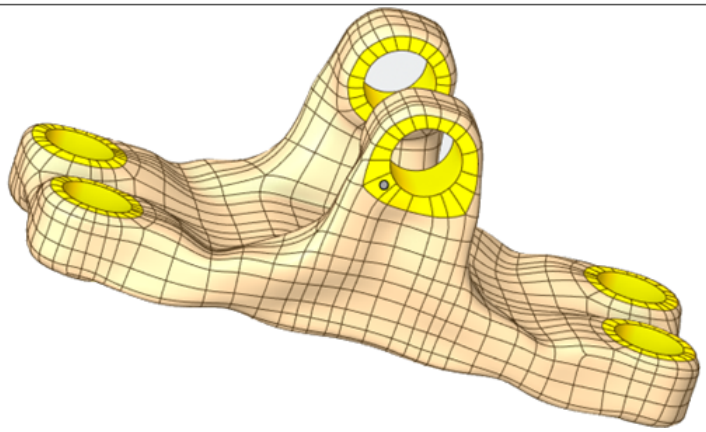
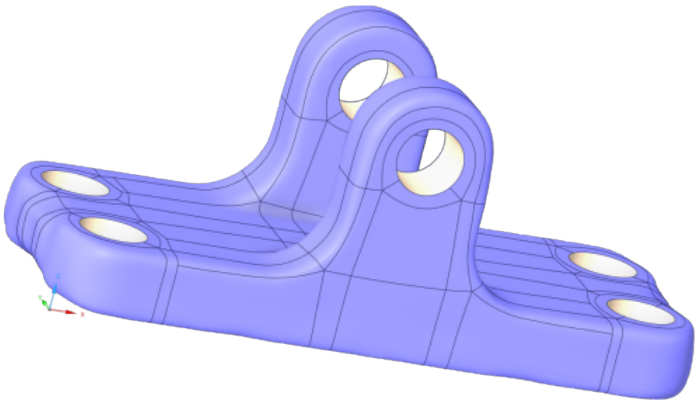
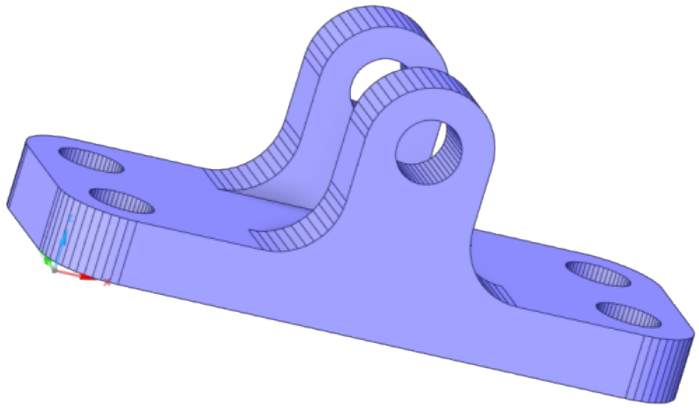
> **Note:** Any design variations set up in Refine mode are overwritten when Test Cases are set in Topology Optimization.

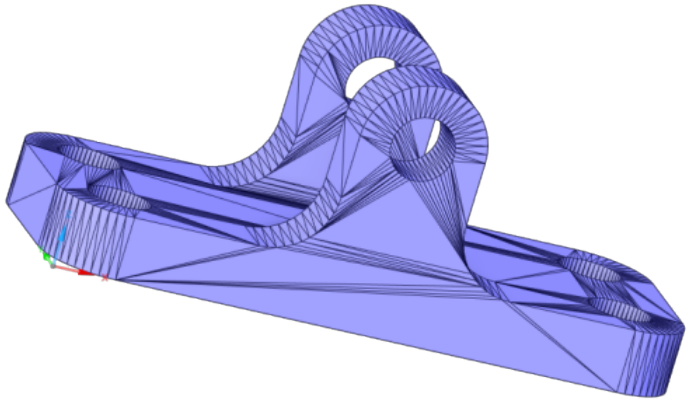
11. Automatic mesh refinement is off by default, but you can turn it on to automatically refine the mesh during optimization.
    a) From the **Simulation** tab ribbon, select the **Global** dropdown menu. The Global fidelity options appear.
    b) For **Automatic mesh refinement**, click **Off** and select **Program controlled** instead.
12. Solve. You can create several different sets of conditions and solve several times.

> **CAUTION:** If you edit the topology optimization conditions during the solve, the optimization tries to adapt the current shape to meet the new conditions. While this can be significantly faster than restarting from the initial geometry, it may result in a different optimized shape than the one calculated directly from the initial geometry. If you prefer to optimize from the initial geometry instead, edit the condition values as desired, then click **Reset** in the Results Arc to restart the simulation. Note that if you edit the geometry, structural conditions, or simulation conditions, the Discovery application always restarts the optimization from the initial geometry.

13. Once you have a faceted geometry result you want to save, click **Add optimized body to model** in the Results Arc to save it to the model. The new faceted geometry is now listed in the Model tree.
14. Right-click the faceted body in the Model tree to perform any of the following actions:

Option	Description
Create Validation Simulation	Once the topo opt simulation is complete, and you have created a faceted body of the resulting optimized shape, select this option to create a new simulation that includes all of the physics conditions from the topology optimization simulation applied to the faceted body. This could be a single test or multiple test cases depending on the topo opt setup. This option automatically creates a validation simulation of the faceted geometry created from the topology optimization simulation.
Smooth Facets	Select this option to smooth your shape. A new faceted model appears in the Model tree.
Convert to SubD hybrid	Select this option to convert your faceted shape to a hybrid model and add it to the Model tree. The hybrid model contains both subdivision surfaces for regions of the model where you want to allow for flexible and freeform editing and traditional surfaces such as planes or cylinders for regions requiring precise topology.

Option	Description
	 Once you have created the hybrid model of the resulting optimized shape, you can right-click the hybrid model in the Model tree and select the **Create Validation Simulation** option to create a validation simulation that includes all of the physics conditions from the topology optimization simulation applied to the hybrid model.
Convert to Solid > Fit Surfaces	Select this option to smooth the facets, fit surfaces onto the facets, then create a solid body by stitching those patches together whenever possible. The patched body is then added to the Model tree. 
Convert to Solid > Mostly Tessellated	Select this option to convert your faceted shape to a solid model and add it to the Model tree, with the facets in flat areas merged together. As seen in this image, that means that curved areas will still have a faceted shape. 

Option	Description
Convert to Solid > Fully Tessellated	Select this option to convert your faceted shape to a solid model and add it to the Model tree, with the facets converted to faces on the solid. **Note:** The more facets there are on the shape, the longer the conversion will take. 

19.5.3. Optimizing to Minimize Stress

The Minimize Stress goal enables you to reduce by percent or volume, but also select whether that reduction is based on von Mises stress or Principal stress.

1. In Explore mode, set up any structural conditions.
2. In the ribbon, click **Topology Optimization**.
3. In the physics tree, double-click **Optimization**.
4. In the HUD, you can optionally select bodies to optimize. All bodies are optimized by default. To select specific bodies:
   a) Click the **Select bodies to optimize** tool.
   b) Select one or more bodies you want to optimize.
   c) Click **Accept**.

   You can default back to including all bodies in your optimization by clicking the **Optimize all simulation bodies** tool.
5. Select **Minimize Stress** as the goal for your optimization.
6. In the tool options, for **Volume reduction**, specify the percentage of distance surrounding applied conditions to be protected during optimization.

   Additionally, you can also select **Reduce by volume** and enter the **Volume reduction** as a distance value.
7. Select the type of stress to minimize.

Option	Description
Minimize von Mises stress	Select to minimize equivalent stress.
Minimize principal stress	Select to minimize principal stress.

8. Additionally, you can specify the following optional optimization conditions:

Option	Description
Min Thickness	Select to specify the minimum size of connected parts. Enter the **Minimum thickness** value in the physics tree.
Max thickness	Select to specify the maximum size of connected parts. Enter the **Maximum thickness** value in the physics tree.
Pull direction	For mold-based manufacturing processes, enable this option to define the direction that the optimized product will be removed from the mold. Then define the pull direction by double-clicking **Pull direction** in the physics tree.
Table direction	For a multi-axis milling process, enable this option if you want to define the direction from which approach to the machine is blocked. Then define the table direction by double-clicking **Table direction** in the physics tree.
Overhang prevention	For a 3D printing process, enable this option if you want to define the growing direction of the manufacturing process. Then define overhang prevention by double-clicking **Overhang prevention** in the Physics Tree.

9. Specify the distance surrounding applied conditions to be protected during optimization, as well as a protected distance for any specific faces. You can also select any additional geometry you want protected.
10. Optionally, set a Symmetry constraint to ensure that the model is dimensionally reduced based on a mirror plane.
    a) Click **Symmetry** in the ribbon.
    b) Select one or more flat faces in the same plane. You can select multiple faces, even if they are on bodies with different physics applied.
    c) Click **Accept** to complete your setup.
11. Optionally, define multiple load cases.
    a) Add physics conditions as parameters by clicking the **Add as a parameter** tool next to the condition in the Physics Tree.
    b) View the Design Variations table by clicking the **Design Variations** icon on the SID.
    a) In the Load Cases tab of the Design Variations table, click **Add** to define load cases.

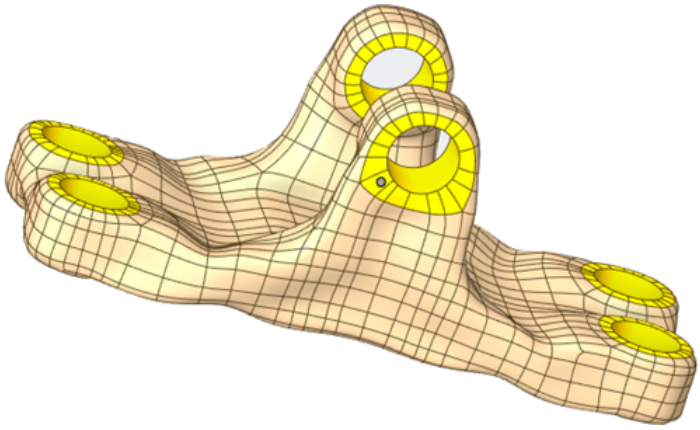
> **Note:** Any design variations set up in Refine mode are overwritten when Test Cases are set in Topology Optimization.

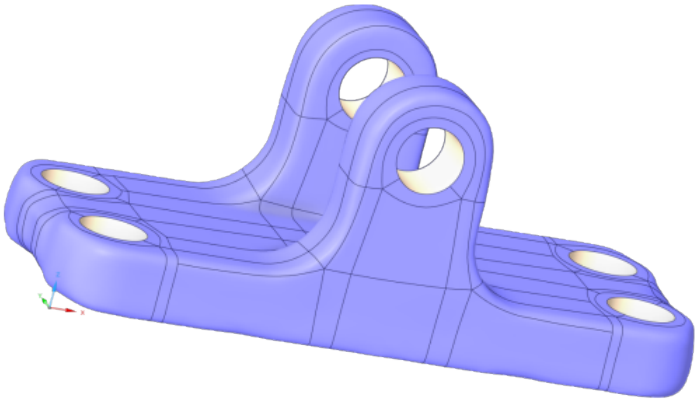
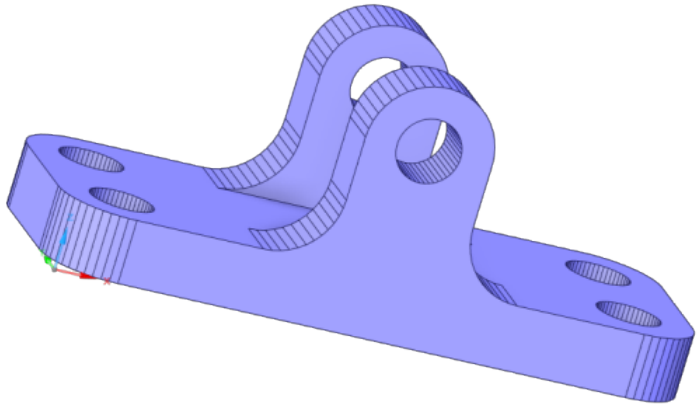
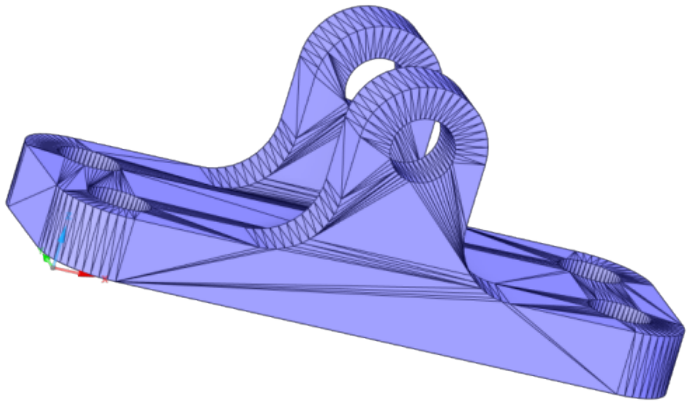
12. Automatic mesh refinement is off by default, but you can turn it on to automatically refine the mesh during optimization.
    a) From the **Simulation** tab ribbon, select the **Global** dropdown menu. The Global fidelity options appear.
    b) For **Automatic mesh refinement**, click **Off** and select **Program controlled** instead.
13. Solve. You can create several different sets of conditions and solve several times.

> **CAUTION:** If you edit the topology optimization conditions during the solve, the optimization tries to adapt the current shape to meet the new conditions. While this can be significantly faster than restarting from the initial geometry, it may result in a different optimized shape than the one calculated directly from the initial geometry. If you prefer to optimize from the initial geometry instead, edit the condition values as desired, then click **Reset** in the Results Arc to restart the simulation. Note that if you edit the

geometry, structural conditions, or simulation conditions, the Discovery application always restarts the optimization from the initial geometry.

14. Once you have a faceted geometry result you want to save, click **Add optimized body to model** in the Results Arc to save it to the model. The new faceted geometry is now listed in the Model tree.
15. Right-click the faceted body in the Model tree to perform any of the following actions:

Option	Description
Create Validation Simulation	Once the topo opt simulation is complete, and you have created a faceted body of the resulting optimized shape, select this option to create a new simulation that includes all of the physics conditions from the topology optimization simulation applied to the faceted body. This could be a single test or multiple test cases depending on the topo opt setup. This option automatically creates a validation simulation of the faceted geometry created from the topology optimization simulation.
Smooth Facets	Select this option to smooth your shape. A new faceted model appears in the Model tree.
Convert to SubD hybrid	Select this option to convert your faceted shape to a hybrid model and add it to the Model tree. The hybrid model contains both subdivision surfaces for regions of the model where you want to allow for flexible and freeform editing and traditional surfaces such as planes or cylinders for regions requiring precise topology.  Once you have created the hybrid model of the resulting optimized shape, you can right-click the hybrid model in the Model tree and select the **Create Validation Simulation** option to create a validation simulation that includes all of the physics conditions from the topology optimization simulation applied to the hybrid model.
Convert to Solid > Fit Surfaces	Select this option to smooth the facets, fit surfaces onto the facets, then create a solid body by stitching those patches together whenever possible. The patched body is then added to the Model tree.

Option	Description
	
Convert to Solid > Mostly Tessellated	Select this option to convert your faceted shape to a solid model and add it to the Model tree, with the facets in flat areas merged together. As seen in this image, that means that curved areas will still have a faceted shape. 
Convert to Solid > Fully Tessellated	Select this option to convert your faceted shape to a solid model and add it to the Model tree, with the facets converted to faces on the solid. **Note:** The more facets there are on the shape, the longer the conversion will take. 

19.5.4. Optimizing to Minimize Volume

The Minimize Volume option removes material from the topology based on a target stress, factor of safety or aggressiveness.

1. In Explore mode, set up any structural conditions.
2. In the ribbon, click **Topology Optimization**.
3. In the physics tree, double-click **Optimization**.
4. In the HUD, you can optionally select bodies to optimize. All bodies are optimized by default. To select specific bodies:
   a) Click the **Select bodies to optimize** tool.
   b) Select one or more bodies you want to optimize.
   c) Click **Accept**.

   You can default back to including all bodies in your optimization by clicking the **Optimize all simulation bodies** tool.
5. Select **Minimize Volume** as the goal for your optimization.
6. In the tool options, select what you want to target when minimizing volume.

Option	Description
Von Mises stress	Select to minimize volume based on a maximum equivalent stress, then specify the maximum stress value. **Note:** While this value is used to guide the optimizer, the Von Mises stress at convergence may differ slightly.
Principal stress	Select to minimize volume based on a maximum principal stress, then specify the maximum stress value.
Factor of safety	Select to minimize volume based on factor of safety, then specify the factor of safety value. **Attention:** The Discovery solver uses an elemental method to calculate factor of safety. However, the monitor factor of safety uses a nodal calculation. As a result, the resulting monitor factor of safety may appear to be lower than the value for the minimum factor of safety. To ensure that the factor of safety constraint has been met, view the %TEMP%/Discovery/{sessionId}/logs/LiveTopologyOptimization_{sessionId} file. If the complete optimization run is successful, SOLUTION FOUND FOR THE OPTIMIZATION PROBLEM appears at the end of the file.

Option	Description
Stiffness reduction	Select to minimize volume based on stiffness reduction, then enter a value for how aggressively the solver refines your shape.

7. Additionally, you can specify the following optional optimization conditions:

Option	Description
Min Thickness	Select to specify the minimum size of connected parts. Enter the **Minimum thickness** value in the physics tree.
Max thickness	Select to specify the maximum size of connected parts. Enter the **Maximum thickness** value in the physics tree.
Pull direction	For mold-based manufacturing processes, enable this option to define the direction that the optimized product will be removed from the mold. Then define the pull direction by double-clicking **Pull direction** in the physics tree.
Table direction	For a multi-axis milling process, enable this option if you want to define the direction from which approach to the machine is blocked. Then define the table direction by double-clicking **Table direction** in the physics tree.
Overhang prevention	For a 3D printing process, enable this option if you want to define the growing direction of the manufacturing process. Then define overhang prevention by double-clicking **Overhang prevention** in the Physics Tree.

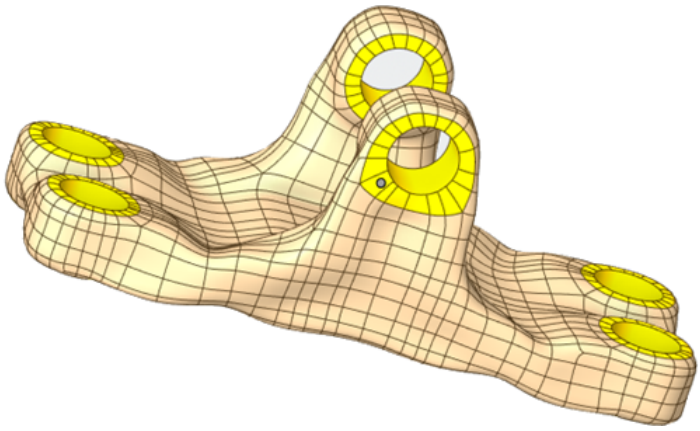
8. Specify the distance surrounding applied conditions to be protected during optimization, as well as a protected distance for any specific faces. You can also select any additional geometry you want protected.
9. Optionally, set a Symmetry constraint to ensure that the model is dimensionally reduced based on a mirror plane.
   a) Click **Symmetry** in the ribbon.
   b) Select one or more flat faces in the same plane. You can select multiple faces, even if they are on bodies with different physics applied.
   c) Click **Accept** to complete your setup.
10. Optionally, define multiple load cases.
   a) Add physics conditions as parameters by clicking the **Add as a parameter** tool next to the condition in the Physics Tree.
   b) View the Design Variations table by clicking the **Design Variations** icon on the SID.
   a) In the Load Cases tab of the Design Variations table, click **Add** to define load cases.

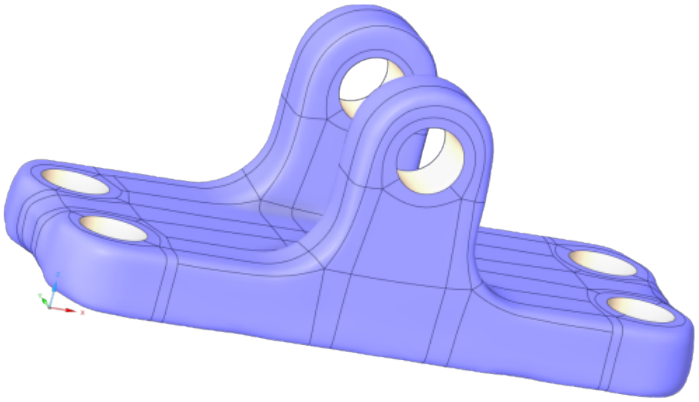
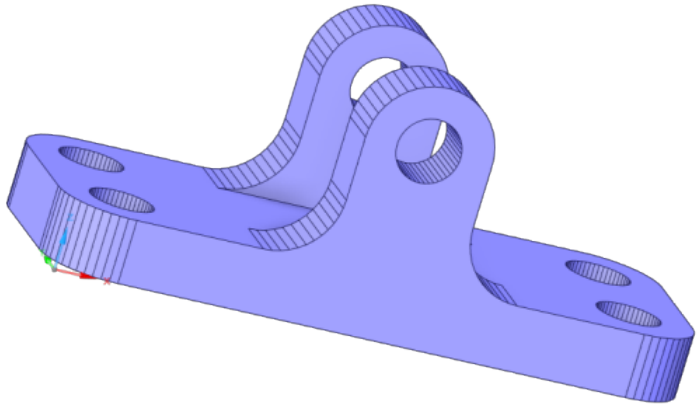
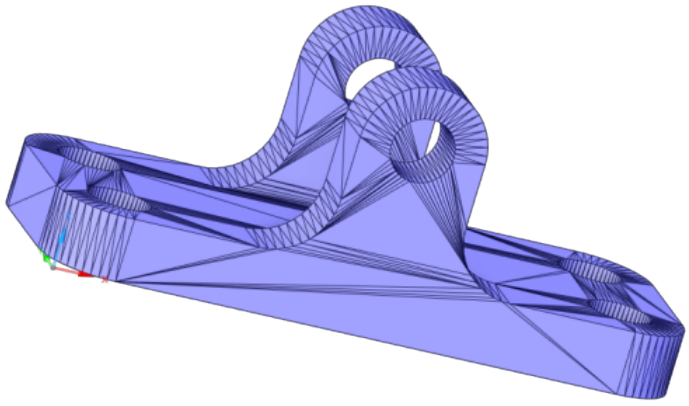
> **Note:** Any design variations set up in Refine mode are overwritten when Test Cases are set in Topology Optimization.

11. Automatic mesh refinement is off by default, but you can turn it on to automatically refine the mesh during optimization.
   a) From the **Simulation** tab ribbon, select the **Global** dropdown menu. The Global fidelity options appear.
   b) For **Automatic mesh refinement**, click **Off** and select **Program controlled** instead.
12. Solve. You can create several different sets of conditions and solve several times.

CAUTION: If you edit the topology optimization conditions during the solve, the optimization tries to adapt the current shape to meet the new conditions. While this can be significantly faster than restarting from the initial geometry, it may result in a different optimized shape than the one calculated directly from the initial geometry. If you prefer to optimize from the initial geometry instead, edit the condition values as desired, then click **Reset** in the Results Arc to restart the simulation. Note that if you edit the geometry, structural conditions, or simulation conditions, the Discovery application always restarts the optimization from the initial geometry.

13. Once you have a faceted geometry result you want to save, click **Add optimized body to model** in the Results Arc to save it to the model. The new faceted geometry is now listed in the Model tree.
14. Right-click the faceted body in the Model tree to perform any of the following actions:

Option	Description
Create Validation Simulation	Once the topo opt simulation is complete, and you have created a faceted body of the resulting optimized shape, select this option to create a new simulation that includes all of the physics conditions from the topology optimization simulation applied to the faceted body. This could be a single test or multiple test cases depending on the topo opt setup. This option automatically creates a validation simulation of the faceted geometry created from the topology optimization simulation.
Smooth Facets	Select this option to smooth your shape. A new faceted model appears in the Model tree.
Convert to SubD hybrid	Select this option to convert your faceted shape to a hybrid model and add it to the Model tree. The hybrid model contains both subdivision surfaces for regions of the model where you want to allow for flexible and freeform editing and traditional surfaces such as planes or cylinders for regions requiring precise topology.  Once you have created the hybrid model of the resulting optimized shape, you can right-click the hybrid model in the Model tree and select the **Create Validation Simulation** option to create a validation simulation that includes all of the physics conditions from the topology optimization simulation applied to the hybrid model.
Convert to Solid > Fit Surfaces	Select this option to smooth the facets, fit surfaces onto the facets, then create a solid body by stitching those patches together whenever possible. The patched body is then added to the Model tree.

Option	Description
	
Convert to Solid > Mostly Tessellated	Select this option to convert your faceted shape to a solid model and add it to the Model tree, with the facets in flat areas merged together. As seen in this image, that means that curved areas will still have a faceted shape. 
Convert to Solid > Fully Tessellated	Select this option to convert your faceted shape to a solid model and add it to the Model tree, with the facets converted to faces on the solid. **Note:** The more facets there are on the shape, the longer the conversion will take. 

19.5.5. Topology Optimization Goals and Tool Options

Topology Optimization Goals

The following topology optimization goals are available:

Maximize Stiffness	This option uses iterative static structural analysis to maximize stiffness and minimize compliance. It reduces the volume or mass but maintains the maximum stiffness possible for the remaining structure.
Maximize Natural Frequency	This option uses iterative natural frequency analysis to minimize the response to frequency vibration.
Balance Stiffness and Frequency	This option uses a combination of static structural and natural frequency analysis to both maximize stiffness and minimize the response to frequency vibration.
Target Natural Frequency	This option uses iterative natural frequency analysis to optimize at a target specified frequency.
Minimize Volume	This option removes material from the topology based on a target stress, factor of safety or aggressiveness.
Minimize Stress	This option enables you to reduce by percent or volume, but also select whether that reduction is based on von Mises stress or Principal stress.

Topology Optimization Tool Options

The following tool options, available under **Manufacturing Constraints**, are common to all goals:

Min thickness	Select to specify the minimum size of connected parts. Enter the **Minimum thickness** value in the physics tree.
Max thickness	Select to specify the maximum size of connected parts. Enter the **Maximum thickness** value in the physics tree.
Pull direction	For mold-based manufacturing processes, enable this option to define the direction that the optimized product will be removed from the mold. Then by double-clicking **Pull direction** in the physics tree.
Table direction	For a multi-axis milling process, enable this option if you want to define the direction from which approach to the machine is blocked. Then by double-clicking **Table direction** in the physics tree.
Overhang prevention	For a 3D printing process, enable this option if you want to define the growing direction of the manufacturing process. Then by double-clicking **Overhang prevention** in the Physics Tree.

The remaining tables discuss options specific to each goal:

- Maximize Stiffness
- Maximize Natural Frequency
- Balance Stiffness and Frequency
- Target Natural Frequency
- Minimize Volume
- Minimize Stress

Maximize Stiffness Tool Options

The following tool options are specific to the **Maximize Stiffness** goal.

Volume reduction	This option is dependent on the **Reduce** setting. • If reducing by percentage, the percentage of distance surrounding applied conditions to be protected during optimization. • If reducing by volume, a distance value.
Reduce by percent	This option directs the optimization to reduce depending on a specified percentage of distance surrounding applied conditions that should be protected during optimization.
Reduce by volume	This option directs the optimization to reduce by a distance value.

Maximize Natural Frequency Tool Options

The following tool options are specific to the **Maximize Natural Frequency** goal.

Volume reduction	This option is dependent on the **Reduce** setting. • If reducing by percentage, the percentage of distance surrounding applied conditions to be protected during optimization. • If reducing by volume, a distance value.
Reduce by percent	This option directs the optimization to reduce depending on a specified percentage of distance surrounding applied conditions that should be protected during optimization.
Reduce by volume	This option directs the optimization to reduce by a distance value.

Balance Stiffness and Frequency Tool Options

The following tool options are specific to the **Balance Stiffness and Frequency** goal.

Volume reduction	This option is dependent on the **Reduce** setting. • If reducing by percentage, the percentage of distance surrounding applied conditions to be protected during optimization. • If reducing by volume, a distance value.
Reduce by percent	This option directs the optimization to reduce depending on a specified percentage of distance surrounding applied conditions that should be protected during optimization.
Reduce by volume	This option directs the optimization to reduce by a distance value.

Target Natural Frequency Tool Options

The following tool options are specific to the **Target Natural Frequency** goal.

Target frequency	The targeted natural frequency value.

Minimize Volume Tool Options

The following tool options are specific to the **Minimize Volume** goal.

Maximum stress	The maximum stress value upon which volume minimization is based. Available if **Von Mises stress** or **Principal stress** are selected.
Von Mises stress	Select to minimize volume based on a maximum equivalent stress. **Note:** While this value is used to guide the optimizer, the Von Mises stress at convergence may differ slightly.
Principal stress	Select to minimize volume based on a maximum principal stress.
Factor of safety (selection)	Select to minimize volume based on a specified factor of safety.
Factor of safety (value)	The factor of safety value. Available if **Factor of safety** is enabled.
Stiffness reduction	Select to minimize volume based on stiffness reduction.
Aggressiveness	The value that determines how aggressively the solver refines the shape.

Minimize Stress Tool Options

The following tool options are specific to the **Minimize Stress** goal.

Volume reduction	This option is dependent on the **Reduce** setting. • If reducing by percentage, the percentage of distance surrounding applied conditions to be protected during optimization. • If reducing by volume, a distance value.
Reduce by percent	This option directs the optimization to reduce depending on a specified percentage of distance surrounding applied conditions that should be protected during optimization.
Reduce by volume	This option directs the optimization to reduce by a distance value.
Minimize von Mises stress	Select to minimize equivalent stress.
Minimize principal stress	Select to minimize principal stress.

19.5.6. Manufacturing Constraints

You can activate optional manufacturing constraints in the **Topology Optimization** HUD. These include:

- Min Thickness
- Max thickness
- Pull direction
- Table direction
- Overhang prevention

Once you've activated a manufacturing constraint, it will be visible in the physics tree. For constraints that need only a value, you can enter the value in the tree. To specify any additional information, double-click the manufacturing constraint to view the HUD.

This section contains the following topics:

19.5.6.1. Setting the Pull Direction

For mold-based manufacturing processes, the pull direction defines the direction that the optimized product will be removed from the mold. A bidirectional pull direction is specified by default, but you can turn this option off. To set the pull direction:

1. In the Optimization HUD, expand **Manufacturing Constraints** and enable **Pull direction**.
2. Double-click **Pull direction** under **Optimization** in the physics tree to define the direction.
3. In the Pull Direction HUD, enter **X**, **Y**, and **Z** values or select a location in the geometry to set the Pull direction, which aligns with the selected face normal as indicated by the arrow. Double-click the arrow to reverse the direction.
4. Select **Bidirectional** to indicate the optimized product can be removed from the mold from both sides. This option is on by default.

19.5.6.1.1. Pull Direction Tool Guides and Options

Pull Direction Tool Guides

The following tool guides are available for pull direction:

Direction tool	Defines the direction that the optimized product will be removed from the mold. To specify the direction, click this option and select a location in the geometry, which aligns with the selected face normal as indicated by the arrow. The direction can also be defined using the **X, Y, Z** values.

Pull Direction Tool Options

The following tool options are available for pull direction:

X, Y, Z	Enter values for the direction that the optimized product will be removed from the mold. The direction can also be defined using the **Direction tool**.
Bidirectional	Enable if the optimized product can be removed from the mold from both sides. This option is on by default, but you can select it to turn it off.

19.5.6.2. Specifying the Table Direction

For a multi-axis milling process, you can define the direction from which approach to the machine is blocked. To set the table direction:

1. In the Optimization HUD, expand **Manufacturing Constraints** and enable **Table direction**.
2. Double-click **Table direction** under **Optimization** in the physics tree to define the direction.
3. In the Table Direction HUD, enter **X**, **Y**, and **Z** values or select a location in the geometry to set the Table direction, which aligns with the selected face normal as indicated by the arrow. Double-click the arrow to reverse the direction.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

19.5.6.2.1. Table Direction Tool Guides and Options

Table Direction Tool Guides

The following tool guides are available for table direction:

Direction tool	Defines the direction from which approach to the part being milled is blocked. To specify the direction, click this option and select a location in the geometry, which aligns with the selected face normal as indicated by the arrow. You can also double-click the arrow to reverse the direction. The direction can also be defined using the **X, Y, Z** values.

Table Direction Tool Options

The following tool options are available for table direction:

X, Y, Z	Enter values for the direction from which approach to the part being milled is blocked. The direction can also be defined using the **Direction tool**. This option is only available when you select **>>** in the HUD.

19.5.6.3. Specifying Overhang Prevention

For a 3D printing process, you can define the growing direction of the manufacturing process. To specify the overhang prevention:

1. In the Optimization HUD, expand **Manufacturing Constraints** and enable **Overhang prevention**.
2. Double-click **Overhang prevention** under **Optimization** in the physics tree to define the direction.
3. In the Overhang Prevention tool, enter **X**, **Y**, and **Z** values or select a location in the geometry to set the direction, which aligns with the selected face normal as indicated by the arrow. Double-click the arrow to reverse the direction.
4. For **Maximum overhang**, enter a value for the maximum allowable inclination angle. You can also enter this value directly in the physics tree.

19.5.6.3.1. Overhang Protection Tool Guides and Options

Overhang Protection Tool Guides

The following tool guides are available for overhang protection:

Direction tool	Defines the direction of growth. To specify the direction, click this tool and select a location in the geometry, which aligns with the selected face normal as indicated by the arrow. You can also double-click the arrow to reverse the direction. The direction can also be defined using the **X, Y, Z** values.

Overhang Protection Tool Options

The following options are available for overhang protection:

Maximum overhang	The maximum allowable inclination angle.
X, Y, Z	Enter values for the direction of the growth during the manufacturing process. The direction can also be defined using the **Direction tool**.

19.5.6.4. Setting the Protected Depth

You can specify the distance surrounding applied conditions to be protected during optimization, as well as set a protected distance for any specific faces. You can also select any additional geometry you want protected.

1. From the ribbon, select **Topology Optimization** > **Protected Depth**, or double-click **Protected depth** under **Optimization** in the physics tree.
2. For **Protected depth** in the Protected depth HUD, enter the distance surrounding applied conditions to be protected during optimization. You can also enter this value directly in the physics tree.
3. Use click and control-click to select any additional faces you want protected, and/or any faces already protected for which you want to change the protected depth value.
4. Enter a protected distance for those faces and click **Accept** to complete the setup.

19.5.6.4.1. Protected Depth Tool Guides and Options

Protected Depth Tool Guides

The following tool guides are available for protected depth:

Accept	Click **Accept** after selecting additional faces to be protected during optimization and entering a protected distance value. You do not have to select this option to define the default distance around which loads and supports are protected; that value is applied in the Physics tree and is also available when you activate the Protected Depth HUD.

Protected Depth Tool Options

The following tool options are available for protected depth:

Protected depth	By default, this option defines the distance surrounding applied conditions to be protected during optimization. This default distance is displayed in the Physics tree. When you select additional faces you want protected, enter a value for the protected distance surrounding those faces.

19.6. Random Vibration Analysis

A Random Vibration analysis enables you to determine the response of structures to vibration loads that are random in nature. An example would be the response of a sensitive electronic component mounted in a car subjected to the vibration from the engine, pavement roughness, and acoustic pressure.

For a random vibration analysis, you specify a constant damping ratio and define a base excitation Power Spectral Density (PSD) for all fixed supports. You can include up to 20 modes, post-process 1, 2 or 3-sigma displacement, stress, velocity and acceleration results, and view PSD output charts. You can also transfer setup to the Mechanical application.

Note: When specifying your modes, ensure that the range of frequencies sufficiently covers the range of the base excitation. This can be accomplished by running a modal simulation to ensure that the highest frequency extracted is 1.5x greater than the last frequency point in the PSD load.

19.6.1. Defining a Random Vibration Analysis

To define a random vibration analysis:

1. If desired, change the default option for the **Constant damping ratio** by clicking **Simulation Options** in the ribbon of the **Simulation** tab and specifying a different value.

   A constant damping ratio specifies a consistent level of damping throughout the structure's response.
2. From the **Natural Frequency** dropdown in the ribbon of the **Simulation** tab, select **Random Vibration**.
3. Double-click **PSD Acceleration (default)** in the Physics tree to open the PSD Load HUD.
4. From the tool guides, select whether you want a base excitation of **Acceleration PSD**, **Velocity PSD**, or **Displacement PSD**.
5. In the tool options, click the chart and define your frequency vs PSD input. This table describes how the excitation is distributed in terms of spectral value vs. frequency.

You can then solve and view Reaction force monitors and PSD Charts. In the Refine stage, you can also choose to view results in 1σ, 2σ, or 3σ.

19.7. Structural Solution Progression

By default, the Discovery application will automatically determine the substep progression in a nonlinear simulation. You can explicitly set the modeling method by going to **Simulation Options** in the ribbon of the **Simulation** tab and modifying the **Additional Structural Options**. This option only affects solutions when in **Refine**.

To set the substep progression:

1. Select **Specify solution progression** to override the values set by the application.
2. Specify the **Initial number of substeps** that the solver will use initially.
3. Specify the **Minimum number of substeps**; if the application determines fewer substeps are needed, then it will not reduce the substeps below this number.
4. Specify the **Maximum number of substeps**; if the application determines more substeps are needed, then it will not increase the substeps above this number.

19.8. Fluid Flow LiveGX Solver

The LiveGX solver is the default solver for fluid and fluid-thermal simulations in **Refine**. The LiveGX solver in **Refine** leverages GPU solver technology (currently available in **Explore**) combined with body-fitted meshing to provide both fast and accurate quantitative fluid results such as mass flow and pressure drop.

Depending on your system's available GPU resources, you have the option to choose between the GPU and CPU solvers. Use the GPU solver for a quick solution. If the demands of a simulation exceed the graphics card resources available, select CPU instead to access the system RAM, which is generally larger than the GPU memory. The CPU solver allows you to run larger cases, but it will take more time. Solve time can be improved by utilizing more CPU cores (HPC license required) by changing the Max number of processors. Whereas, the GPU solver will be faster, but the case size will be limited by the size of the GPU. If using the GPU solver, an approximate guideline is that 1 GB of

GPU memory is required for every 1 million elements in the model. You can see your GPU usage in the status bar when solving.

Shared memory fallback (implemented by NVIDIA for stable diffusion for some of the drivers) allows an application to use shared memory if GPU memory is exhausted. Applications that previously crashed when running out of memory can now continue to run, but at lower speeds. Simulation may run slower when solving a model that exceeds the GPU size.

To specify the CPU or GPU solver, go to **Simulation Options** in the **Simulation** tab ribbon. Click **Additional Fluid Flow Options (Refine)** to see the options in the expanded panel. Select **GPU** or **CPU** from **LiveGX solver mode**.

> **Note:** The LiveGX solver uses a different wall distance algorithm when run using **CPU** or **GPU**. This difference may result in slight variations in convergence behavior and results between a CPU and a GPU solution.

Along with the LiveGX solver, tetrahedral and polyhedral meshing options can be used to efficiently capture a wide range of physical features to both optimize GPU memory usage and accurately simulate fluid flow and heat transfer. Additionally, automatic boundary layer definition and local and global fidelity controls can aid in solution accuracy.

> **Note:** To change your default solver settings, visit the physics **Settings** panel and make your selections under **Solver Settings** and **Simulation Options Defaults**. You can override the default settings in the **Simulation Options** drop-down menu in the **Simulation** tab.

Known Limitations

- Results variable **Vortices Lambda 2** is not supported.
- The **Spalart-Allmaras** turbulence model is not supported.
- Compressible flows:
  - The Discovery application provides compressible flow modeling capabilities for subsonic internal flows. However, there may be cases where the flow becomes transonic or supersonic due to a flow constriction in the fluid domain. External aerodynamics applications, supersonic and hypersonic flows, and compressible liquids should be avoided.
  - For simulations including multiple fluids, all gases in the simulation will be treated as compressible using variable density with the ideal gas law. All liquids in a simulation with multiple fluids are treated as incompressible, regardless of whether compressibility is turned on.
  - Buoyancy is not compatible with compressible flow. Turn off the **Include buoyancy** selection in the **Gravity** HUD to solve a compressible flow simulation.
  - Mass flow rate specification at a flow outlet is not supported in compressible flow simulations. You can specify pressure at the flow outlet.
  - Velocity specification at a flow outlet is not supported in compressible flow simulations. You can specify pressure at the flow outlet.
  - In **Refine**, if a domain with a compressible fluid is excluded from the simulation, after subsequent inclusion it is incompressible.

19.9. Fluid Flow Solver

The application's fluid flow solver is a pressure-based solver that uses a pressure-velocity coupling scheme to solve flow problems in either a segregated or coupled manner. Using the coupled approach offers some advantages over the non-coupled or segregated approach.

From the **Simulation Options** dropdown menu in the **Simulation** tab ribbon, click **Additional Fluid Flow Options** to see the options in the expanded panel. The default **Fluids solver** is **Program Controlled**, allowing the application to determine the most appropriate pressure-based solver for your simulation setup. Otherwise, you can choose the solver algorithm:

- Use **SIMPLE** (a segregated algorithm) for steady-state, incompressible, and moderately turbulent flows, where computational efficiency is a priority.
- Use **Coupled** for time-dependent, compressible or swirling flows, where high accuracy and better handling of the strong coupling between pressure and velocity are necessary, despite the increased computational cost.

> **Note:** The SIMPLE solver uses less GPU memory than the coupled solver for the same mesh.

SIMPLE Segregated Algorithm

The SIMPLE algorithm (Semi-Implicit Method for Pressure-Linked Equations) is an iterative method for solving Navier-Stokes equations for incompressible flows. It uses a relationship between velocity and pressure corrections to enforce mass conservation and to obtain the pressure field. The SIMPLE solver is used to guess the pressure field, solve the momentum equations to get velocity, correct both based on mass conservation (continuity equation), and iterate the process until convergence.

The pressure-based segregated algorithm solves the momentum equation and pressure correction equations separately. This semi-implicit solution method results in slow convergence. Details about the equations used in the SIMPLE solver can be found in the **Fluent Theory Guide**.

Coupled Algorithm

The coupled scheme obtains a robust and efficient single-phase implementation for steady-state flows, with superior performance compared to the segregated solution schemes. This pressure-based coupled algorithm offers an alternative to the density-based and pressure-based segregated algorithm with SIMPLE-type pressure-velocity coupling. For time-dependent flows, using the coupled algorithm is necessary when the quality of the mesh is poor, or if large time steps are used.

The coupled algorithm solves the momentum and pressure-based continuity equations together. The full implicit coupling is achieved through an implicit discretization of pressure gradient terms in the momentum equations, and an implicit discretization of the face mass flux, including the Rhie-Chow pressure dissipation terms. Details about the equations used in the coupled solver can be found in the **Fluent Theory Guide**.

19.10. Fluid Flow Turbulence Modeling

A vast majority of industrial flows are turbulent. While conditions at boundaries may be steady, shear action near walls and other velocity profiles lead to time-dependent pressure and velocity fluctuations transported by and affecting the mean path of the fluid. These "turbulent" fluctuations span a wide range of length and timescales. As a result, computational demands for exact solutions of industrial turbulent flows far exceed available computing resources.

Reynolds-Averaged Navier-Stokes (RANS) models provide an economic approach to solving these complex, turbulent flows by modeling turbulence as fluctuations within a steady, average velocity field. These models are not exact but have been tuned to provide reasonable accuracy on appropriately scaled meshes. RANS models solve additional transport equations for turbulence and introduce an eddy viscosity (also known as turbulent viscosity) to the simulation to mimic the effect of turbulence, a modeling approach that is suitable for many engineering applications.

In the Discovery application, the k-omega SST turbulence model is appropriate for most cases and is the default for both time-dependent and steady-state simulations. Additional turbulence models are available and can be modified based on your simulation requirements.

19.10.1. Turbulence Model Options

By default, the Discovery application will automatically determine the fluid flow modeling method for you based on the physics inputs. You can explicitly set the modeling method by going to **Simulation Options** in the ribbon of the **Simulation** tab and make modifications under **Additional Fluid Flow Options**.

Select **Specify modeling method** to override the method set by the application. Choose one of the turbulence models if modeling turbulent flow. Alternatively, if the flow is not turbulent, set the fluid flow modeling method to **Laminar**.

The table below provides information about each of the turbulence options.

k-omega SST	A turbulence model that allows more accurate resolution of boundary layer behavior as the near wall resolution is refined. This model is useful also for free-stream flows, in particular, flows with adverse pressure gradients, flows around airfoils, etc. This model is appropriate for most industrial flows, and is therefore the default model. Where the near wall mesh resolution is coarse, the SST model assumes the boundary layer is fully developed. This is not always the case, especially under adverse pressure gradients when flows decelerate. Under these conditions, the SST model allows you to refine the near wall mesh and capture boundary layer instabilities which lead to separation. If this occurs, then you should switch to the k-omega standard model. k-omega models are typically better in predicting flow separation, and this is one reason why the k-omega SST model is among the most widely used models for aerodynamic flows.
k-epsilon standard	A general industrial model that has historically been used heavily in fluid flow simulations. The wall treatment for this model assumes a fully developed boundary layer, which is not often valid. The omega-based models offer a more consistent wall treatment. This model is available in **Refine**.
k-omega standard	A general industrial model that offers a reasonable compromise of accuracy and robustness and is generally conservative on separation prediction. This model is available in **Refine**.

k-epsilon realizable	A general industrial model that is a fair compromise between accuracy and robustness for many industrial applications. The k-omega-based SST and standard models are a better alternative and offer a more consistent wall treatment. Other, newer models, such as the k-omega SST model, offer various improvements over the historic realizable k-epsilon model and are normally favored. For cases where the flow separates under adverse pressure gradients from smooth surfaces (airfoils, etc.), k-epsilon models are generally not recommended as they can lead to overly optimistic design evaluations. This is one reason why k-epsilon models are not widely used in external aerodynamics. For cases where the flow separates under adverse pressure gradients from smooth surfaces (airfoils, etc.), k-epsilon models are generally not recommended as they can lead to overly optimistic design evaluations. This model is available in **Refine**.
Spalart-Allmaras	A computationally efficient one equation model designed for aerodynamics/aerospace applications. This model is available in **Refine** (not with the LiveGX solver).
Smagorinsky (LES)	A turbulence model that solves for large-scale fluctuating motions and uses "sub-grid" scale turbulence models for the small-scale motion. Time-dependent equations are solved for the turbulent motion with equations filtered in some way to remove very fine time and length scales. This model is available for time-dependent simulations in **Explore**.

19.11. Fluid Flow Meshing Methods

In the Discovery application, the meshing method you choose depends on the complexity of your model, the solution accuracy you would like to achieve, and the computational expense. Two meshing methods are available, tetrahedral meshing and polyhedral meshing. Tetrahedral meshes are composed of only tetrahedron-shaped elements, while polyhedral meshes can be composed of multiple shapes, including triangles, quadrilaterals, and polygons. Polyhedral meshes are more adaptable than tetrahedral meshes, as they can conform to the shape of the object they are modeling more closely. This in turn means polyhedral meshes have a lower cell count than the equivalent original tetrahedral mesh, and therefore solution convergence will generally be faster, possibly saving some computational expense.

From the **Simulation Options** dropdown menu in the **Simulation** tab ribbon, click **Additional Fluid Flow Options (Refine)** to see the options in the expanded panel. Click **Specify meshing method** to choose **Tetrahedra mesh** or **Polyhedra mesh**.

> **Note:** From the **Simulation Options** (Refine), you can also toggle on any mesh quality criteria you want to monitor, then specify upper and lower limits. For more information, see Simulations Options.

Polyhedra Meshing Limitations

- Edge refinement is not supported.
- Node count mesh statistics on a user selected location is not supported.

19.12. Mass Flow-Weighted Average for Monitors

The Discovery application generally uses area-weighted averaging for all results monitors. However, you can use mass flow-weighted averaging instead for the following results variables:

- Temperature
- Temperature uniformity
- Total pressure
- Pressure drop

Under **Simulation Options**, on the **Simulation** tab, click **Additional Fluid Flow Options** to see the options in the expanded panel. Click **Mass flow-weighted average for monitors** if you want monitors calculated using mass flow-weighted averaging.

You can select inlets, outlets, fluid-fluid interfaces, or planes crossing the flow region. Mass flow-weighted averaging is also available for surface bodies only in **Explore**.

An area-weighted average is calculated by integrating the scalar variable times the area divided by the total area over the region.

$$\frac{1}{A}\int \phi dA = \frac{1}{A}\sum_{i=1}^{n} \phi_i |A_i|$$

A mass flow-weighted average is calculated by integrating the scalar variable times mass flow divided by total mass flow over the region.

$$\frac{\int \phi\rho |\vec{v}\cdot d\vec{A}|}{\int \rho |\vec{v}\cdot d\vec{A}|} = \frac{\sum_{i=1}^{n} \phi_i \rho_i |\vec{v}_i \cdot \vec{A}_i|}{\sum_{i=1}^{n} \rho_i |\vec{v}_i \cdot \vec{A}_i|}$$

Note:

- **Mass flow-weighted average for monitors** is on by default for new simulations. This setting might be off in simulations created in previous versions of the application.
- If a mass flow-weighted average monitor is created using a location that includes both a solid and a fluid surface, the monitor value will ignore the solid region and return the mass flow-weighted average for the fluid region only.

19.13. Fluid Flow Convergence Criteria

The nonlinear equations governing fluid flow are solved using an iterative solution method. Convergence refers to the point at which further iterations are no longer necessary. An important part of determining whether your solution is appropriate is to judge whether the solution has converged to an acceptable level.

By default, the Discovery application will automatically use a set of numerical convergence settings that are suitable in most cases. You can explicitly set the convergence criteria by going to **Simulation Options** in the ribbon of the

Simulation tab and make modifications under **Additional Fluid Flow Options**. The **Refine** stage contains additional convergence settings that affect the solution.

In **Refine**, you can choose to determine the stopping criteria for when the problem has converged by selecting one of three options from the **Stop on** menu:

- **Engineering convergence**: The engineering quantities of pressure, velocity, and temperature are monitored for each flow condition, and temperature and pressure are monitored for each wall condition. When these monitored quantities have stabilized, the solution will be determined as complete. This option is typically useful if these values are the primary focus of the solution and are unaffected by local areas of turbulent flow. If convergence is not achieved, the solution will stop when the **Maximum number of iterations** are reached, which can be adjusted in the **Specify convergence settings**, if required. Additionally, you can modify the **Target engineering convergence** tolerance to specify the level of convergence that you require.
- **Numerical convergence**: Convergence is determined by examining differences throughout the fluid volume. This is useful if you are interested in the flow within the volume. To override the default values, select **Specify convergence settings**. Enter the **Maximum number of iterations**, if convergence is not met after this number of iterations, then the simulation will stop. Enter the **Target numerical convergence** to specify the difference for residual measurements (except energy) allowed between iterations, if the difference is below this level, it will be considered as converged. Enter the **Target numerical convergence, energy** to specify the difference for energy allowed between iterations, if the difference is below this level, it will be considered as converged.
- **Monitored value convergence**: This is similar to engineering convergence but will only use monitors of average values on the surfaces that you have defined. Under **Specify convergence settings** you can also modify **Target convergence, monitors** to specify the level of convergence that you require.
- **Solution stopping criterion**: Convergence is determined when the **Target stopping criteria** tolerance is met. A default value of 0.01 (or 1%) is used. When the percent change in solution is below the tolerance you specify in **Target stopping criteria** (under **Specify convergence settings**), it will be considered as converged.

> **Note:** This option is only available with the LiveGX solver.

In **Explore**, you can select **Specify convergence settings** and enter the **Target stopping criteria**. A default value of 0.01 (or 1%) is used. When the percent change in solution is below the tolerance you specify in **Target stopping criteria**, it will be considered as converged.

> **Note:** For a steady-state solution that is slowly converging, try reducing the **Target stopping criteria**. Oscillations may indicate unsteady flow phenomena. Review monitor details to assess the results. If necessary, switch to a time-dependent simulation.

19.14. Electromagnetics Settings and Convergence Criteria

Convergence refers to the point at which further iterations are no longer necessary. An important part of determining whether your solution is appropriate is to judge whether the solution has converged to an acceptable level.

By default, the Discovery application will automatically use a set of numerical convergence settings that are suitable in most cases. You can explicitly set the convergence criteria by going to **Simulation Options** in the ribbon of the **Simulation** tab and make modifications under **Electromagnetics Options**. This option only affects solutions when in the **Refine** stage.

- **Modify frequency points** to specify the **Far-field**, **Near-field** and **S-parameters** frequency points.
- **Specify loss settings**:
  - determine whether **Dielectric** losses should be based on materials or if they can be ignored.

- specify the **Perfect conductivity threshold** value to set the material threshold for electrical conductivity.

- **Specify convergence settings**:
  - specify the **Maximum simulation duration** for each simulation. The simulation is considered converged once the volume-integrated electric field intensity drops below a ratio of its peak value.
  - enter the **Target numerical convergence** to specify this threshold. The simulation will run longer for smaller values.

19.15. Simulation Mesh Quality

You can set the mesh metrics monitored during the solve and specify limits for those metrics. After meshing, you are notified if any selected meshing criteria are not met.

For fluids simulations, those include:

- Orthogonal quality (on by default) – lower limit (default 0.1)
- Skewness (off by default) – upper limit (default 0.95)
- Aspect ratio (off by default) – upper limit (default 20)

For structural simulations, those include:

- Element quality (off by default) – lower limit (default 0.05)
- Jacobian ratio (on by default) – upper limit (default 30)
- Aspect ratio (off by default) – upper limit (default 20)

Default mesh metrics are set in the in the **Physics Settings**, but you can also specify mesh metrics for this particular simulation under **Simulation Options** on the **Simulation** tab. Click **Fluid Mesh Quality** or **Structural Mesh Quality** to see the options in the expanded panel.

You can then enable mesh plots highlighting poor quality elements.

> **Note:** Mesh metrics are only available in **Refine**.

19.16. Troubleshooting Stalled Solutions

This troubleshooting guide addresses common issues that may cause the solution to stall or converge slowly. By understanding and identifying these issues, you can take appropriate actions to resolve these problems and obtain a solution.

Problem	Resolution
The steady-state solution is stalling because the problem requires a time-dependent solution.	Check for oscillating behavior in the solution monitors and switch to a time-dependent solution.

Problem	Resolution
The steady-state solution may stall because the fluid region is not adequately discretized and/or the mesh quality is poor.	Review the velocity field and the mesh, and look for areas where there may be fluid recirculation regions or high velocity gradients where the mesh is not fine enough to capture the fluid behavior. Additionally, add a local fidelity or increase the solution fidelity to improve the convergence behavior. **Note:** You cannot check the mesh quality in **Explore**.
The steady-state solution is slow to converge.	Check for slowly converging solution behavior in the solution monitors. You may need to specify a smaller target stopping criteria.

20: Simulation Optimization

This section contains the following topics:

20.1. Optimizing Performance with Solution Fidelity

The fidelity of the solution is controlled primarily using the slider in the stage navigator. This acts differently depending on if you are in **Explore** or **Refine**. In both cases an increased fidelity will capture additional detail but will take longer to produce a result. Additionally, higher fidelity may result in an increase in GPU memory usage. You can see how much GPU memory is being used in the status bar when solving. Monitoring GPU memory usage can be helpful to determine if your current simulation exceeds your GPU's available memory, likely leading to instability or dramatically slower solve times due to paging to system memory. Adjusting the fidelity slider will affect memory usage.

In **Explore**, the fidelity controls the solution grid density. With higher densities, additional geometric details can be captured, such as thinner parts and gaps. In **Refine**, the fidelity controls the solution mesh density. The mesh is generated automatically as part of the solution process and adapts automatically to the applied physics conditions.

For further control, you can choose the desired level of your solution accuracy or adjust the sizing methodology as described in Global Solution Fidelity . Additionally, you can make local fidelity adjustments to capture specific effects.

During topology optimization, you can automatically refine the mesh.

> **Note:** When opening a project on a different machine, the fidelity control in **Explore** may change due to GPU memory system differences. However, the obtained solution will remain consistent.

This section contains the following topics:

20.1.1. Global Solution Fidelity

Global fidelity allows you to choose the desired level of your solution accuracy or sizing methodology.

In **Explore**, under **Global**, on the **Simulation** tab, select a **Fidelity approach**. By default, **Standard** is the recommended setting. Although **Aggressive** or **Extreme** capture more geometric details and yield higher-fidelity results than the **Standard** setting, they run the risk of running out of GPU memory when solving on lower specification graphics cards. You can define the fidelity level using **Custom**. Specifying a value over 1, combined with a high fidelity slider position, may cause the simulation to run out of GPU memory.

In **Refine**, the size function controls how the mesh size is distributed on a surface or within a volume. Under **Global**, on the **Simulation** tab, you can choose to **Determine sizing automatically** for your simulation or you can specify the curvature, proximity, or fixed size functions to provide more precise control over mesh size distribution.

This section contains the following topics:

20.1.1.1. Curvature Size Function

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

The following properties define the curvature size function.

Minimum size	The global minimum size to be used in curvature size function mesh calculations. Some element sizes may be smaller than this based on local feature sizes or other geometric anomalies. Specify a value greater than 0 or accept the predefined setting.
Maximum face size	The maximum size to be used for surface meshing. Element faces may be larger than this size based on hard edge sizes or floating point arithmetic. Specify a value greater than 0 or accept the predefined setting.
Maximum size	The maximum size to be used in curvature size function mesh calculations. Specify a value greater than 0 or accept the predefined setting.
Growth rate	Represents the increase in element edge length with each successive 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 predefined setting. The predefined setting is 1.2.
Curvature normal angle	The maximum permitted angle that one element edge may span. Specify a positive value of up to 3.14 radians (180 degrees), or accept the predefined setting. The predefined setting is 0.31416 radians (or 18 degrees).
Maximum boundary layers	Boundary layers improve the mesh to resolve high gradients of velocity or temperature near the wall for fluid flow simulations. For fluid flow and fluid-solid heat transfer simulations, the default maximum boundary layers are 3. You can modify the number of boundary layers, increasing the number of layers improves accuracy and increases the solution time.

20.1.1.2. Proximity Size Function

The proximity size function is the minimum number of element layers created in regions that constitute "gaps" in the model. For the purposes of specifying a proximity size function, 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

The following properties define the proximity size function.

Minimum size	The global minimum size to be used in proximity size function calculations. Some element sizes may be smaller than this based on local feature sizes or other geometric anomalies. You can accept the predefined setting or specify a value greater than 0.
Maximum face size	The maximum size to be used for surface meshing. Element faces may be larger than this size based on hard edge sizes or floating point arithmetic. Specify a value greater than 0 or accept the predefined setting.

Maximum size	The maximum size to be used in proximity size function mesh calculations. Specify a value greater than 0 or accept the predefined setting.
Growth rate	Represents the increase in element edge length with each successive 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 predefined setting. The predefined setting is 1.2.
Proximity size function sources	Determines whether regions of proximity between faces and/or edges are considered when proximity size function calculations are performed. Choose one of the following: • **Edges**: Considers edge-edge proximity. Face-face and face-edge proximity are not considered. • **Faces**: This is the predefined setting. 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). • **Faces and edges**: Considers face-face and edge-edge proximity. Face-edge proximity is not considered.
Number of cells across gap	The minimum number of layers of elements to be generated in the gaps. You can specify a value greater than 0 to 100 or accept the predefined setting. The predefined setting is 1. The value of **Number of cells across gap** is an approximation: • The value may not be satisfied exactly in every gap. When sweeping, interval assignment may change the number of divisions (that is, elements or cells) in a gap. • You must define a minimum size such that it allows the specified **Number of cells across gap** to be achieved. For example, if you have a gap of 1 mm, the minimum size is 0.5 mm, and the value of **Number of cells across gap** is 3, only 2 cells can be generated in the gap. • For thin regions, to control the aspect ratio of the cells in the gap, you can use a value 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 enter 0.25 for **Number of cells across gap**. A value of 0 is not acceptable. • **Faces and edges**: Considers face-face and edge-edge proximity. Face-edge proximity is not considered.
Maximum boundary layers	Boundary layers improve the mesh to resolve high gradients of velocity or temperature near the wall for fluid flow simulations. For fluid flow and fluid-solid heat transfer simulations, the default maximum boundary layers are 3. You can modify the number of boundary layers, increasing the number of layers improves accuracy and increases the solution time.

20.1.1.3. Curvature and Proximity Size Function

This size function controls how the mesh size is distributed on a surface or within a volume.

The following properties define the curvature size function.

Minimum size	The global minimum size to be used in curvature and proximity size function mesh calculations. Some element sizes may be smaller than this based on local feature sizes or other geometric anomalies. Specify a value greater than 0 or accept the predefined setting.
Maximum face size	The maximum size to be used for surface meshing. Element faces may be larger than this size based on hard edge sizes or floating point arithmetic. Specify a value greater than 0 or accept the predefined setting.
Maximum size	The maximum size to be used in curvature and proximity size function mesh calculations. Specify a value greater than 0 or accept the predefined setting.
Growth rate	Represents the increase in element edge length with each successive 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 predefined setting. The predefined setting is 1.2.
Curvature normal angle	The maximum permitted angle that one element edge may span. Specify a positive value of up to 3.14 radians (180 degrees), or accept the predefined setting. The predefined setting is 0.31416 radians (or 18 degrees).
Proximity size function sources	Determines whether regions of proximity between faces and/or edges are considered when proximity size function calculations are performed. Choose one of the following: • **Edges**: Considers edge-edge proximity. Face-face and face-edge proximity are not considered. • **Faces**: This is the predefined setting. 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). • **Faces and edges**: Considers face-face and edge-edge proximity. Face-edge proximity is not considered.
Number of cells across gap	The minimum number of layers of elements to be generated in the gaps. You can specify a value greater than 0 to 100 or accept the predefined setting. The predefined setting is 1. The value of **Number of cells across gap** is an approximation: • The value may not be satisfied exactly in every gap. When sweeping, interval assignment may change the number of divisions (that is, elements or cells) in a gap. • You must define minimum size such that it allows the specified **Number of cells across gap** to be achieved. For example, if you have a gap of 1 mm, the minimum size is 0.5 mm, and the value of **Number of cells across gap** is 3, only 2 cells can be generated in the gap. • For thin regions, to control the aspect ratio of the cells in the gap, you can use a value 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 enter 0.25 for **Number of cells across gap**. A value of 0 is not acceptable. • **Faces and edges**: Considers face-face and edge-edge proximity. Face-edge proximity is not considered.

Maximum boundary layers	Boundary layers improve the mesh to resolve high gradients of velocity or temperature near the wall for fluid flow simulations. For fluid flow and fluid-solid heat transfer simulations, the default maximum boundary layers are 3. You can modify the number of boundary layers, increasing the number of layers improves accuracy and increases the solution time.

20.1.1.4. Fixed Size Function

The fixed size function controls how the mesh size is distributed on a surface or within a volume. Since it does not refine the mesh based on curvature or proximity, instead you specify minimum and maximum sizes and gradation is provided between sizes based on the specified growth rate.

The following properties define the fixed size function.

Minimum size	The minimum size to be used in fixed size function mesh calculations. Some element sizes may be smaller than this based on local feature sizes or other geometric anomalies. You can accept the predefined setting or specify a value greater than 0.
Maximum face size	The maximum size to be used in fixed size function mesh calculations. Element faces may be larger than this size based on hard edge sizes or floating point arithmetic. Specify a value greater than 0 or accept the predefined setting.
Maximum size	The maximum size to be used in fixed size function mesh calculations. Specify a value greater than 0 or accept the predefined setting.
Growth rate	Represents the increase in element edge length with each successive 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 predefined setting. The predefined setting is 1.2.
Maximum boundary layers	Boundary layers improve the mesh to resolve high gradients of velocity or temperature near the wall for fluid flow simulations. For fluid flow and fluid-solid heat transfer simulations, the default maximum boundary layers are 3. You can modify the number of boundary layers, increasing the number of layers improves accuracy and increases the solution time.

20.1.2. Local Fidelity

Using **Local** on the **Simulation** tab, you can manually set lower or higher result resolutions. A smaller value will capture more geometric details, leading to higher fidelity results. All user specified local fidelity values are treated as "soft" size controls during mesh generation, which means that the mesher can decide to override any user specified local fidelity to improve the overall quality of the mesh.

In fluid flow simulations, a local fidelity can be applied to a Body of Influence (BOI) to refine the mesh locally within a volume defined by a body that fully or partially overlaps the physics region or regions.

Local Fidelity Considerations

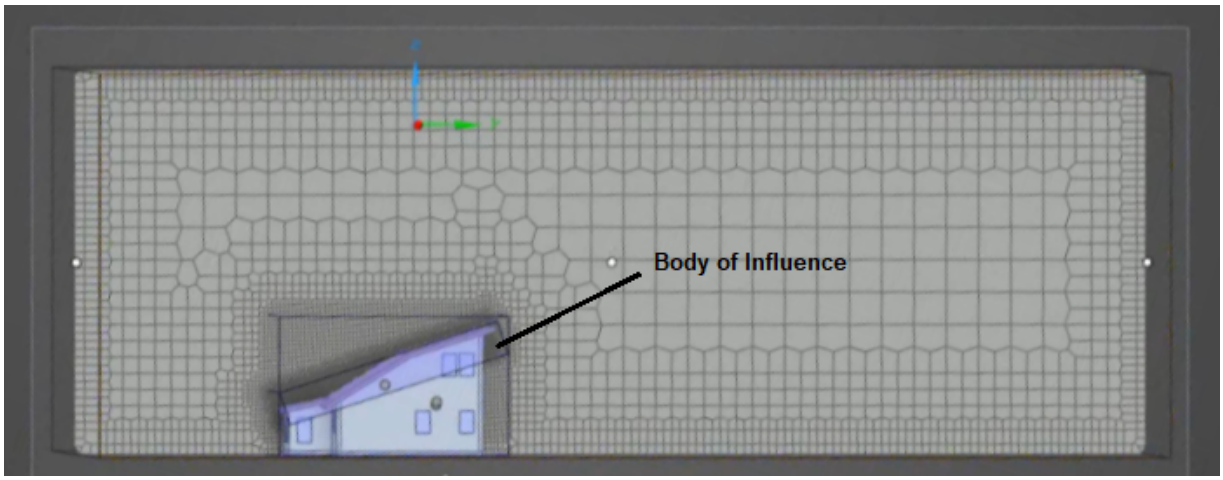
- In **Explore**, for structural and solid thermal simulations, local fidelity can only be assigned to bodies and for fluid flow simulations it can be assigned to faces and bodies.
- In **Refine**, you can apply local fidelity to bodies, faces, and edges for all physics types.

- The Discovery application might not apply the exact fidelity size you select. Increasing the fidelity slider setting might not change your mesh refinement. To view the applied local fidelity, review the mesh in **Refine**. In **Explore**, display the mesh or use the **Size Preview** tool to view the actual fidelity applied to a body. You can also use the Resolution plot to determine if the refinement is sufficient to capture all the salient geometric features.
- If you apply a local fidelity to one side of an interface, then the same fidelity sizing will be applied to the second side of the interface.
- If you specify different local fidelities at each interface of a fluid-solid, fluid-fluid, or solid-solid interface, the lower value will be applied to both sides of the interface.
- In **Explore**, you can assign local fidelity to suppressed bodies for body of influence meshing in fluid flow simulations. The fluid region will adopt the local fidelity sizing in areas that overlap with the suppressed fidelity body. Although the fidelity information is conveyed to the solver, the suppressed body itself is not. If multiple body of influence mesh regions overlap, the intersecting region will adopt the smaller mesh size.
- In **Explore**, when solving on a very fine mesh, systems using a 4GB graphics card could run out of memory. You can try decreasing the global or local refinement, or both. Additionally, make sure you only have one session of the application running, or simplify your model.

20.1.2.1. Body of Influence

In **Explore**, a Body of Influence (BOI) can be used in fluid flow simulations to refine the mesh locally within a volume defined by a body that fully or partially overlaps the physics region or regions.

A BOI to increase resolution of the flow solution might be used, for example, downstream of an object to avoid artificial dissipation of the wake due to a coarse mesh or to capture the large gradients in a jet of fluid. The body itself, which can be of any shape, must be excluded from the simulation so that it is not part of the computational model. Since a BOI is independent of the geometry defining the physics region, it allows for easy adjustment of the mesh refinement, by modification of size, shape or position of the body. Multiple BOIs can be created. If they overlap with one another, the smaller mesh size will be used on the intersection.



20.1.3. Fidelity Auto Generation for Electromagnetics

For electromagnetic simulations, select **Auto Generate**, on the **Simulation** tab, to set the geometry to apply automatic mesh sizing and generation of local fidelities. Select the geometry where you want to create local fidelity. You can select bodies, faces or edges. But fidelities are only created on conductors if the resulting size is less than the **Minimum size** under **Global**. View and edit the resulting local fidelities in the Physics Tree. Hover over to highlight the geometry and right-click to modify each local sizing object. You can replace location, edit value, delete and rename the local fidelity. You can also add a note.

Note: For all selected faces of conducting materials, the edges are sorted into contiguous collections. If there are multiple collections, the minimum separation between the collections is applied as a constraint to the face. For each edge in the selection region, if its length is less than the **Minimum size** (specified under **Global** on the **Simulation** tab), the edge length is applied as a constraint to the edge.

Note: For each face in the selection region, a local fidelity is created with size as the minimum distance between the face's loops or the minimum distance between edges for a single loop.

20.1.4. Specifying Automatic Mesh Refinement for Topology Optimization

Automatic mesh refinement is off by default, but you can turn it on to automatically refine the mesh during optimization. Note that the mesh refinement depends on the target fidelity set. When automatic mesh refinement is on, topology optimization begins with a coarser resolution, then refines the mesh during optimization to reach the specified fidelity at convergence. If a low target fidelity is set, refinement may not occur because there is no advantage in doing so, in terms of the computational time saved.

1. From the **Simulation** tab ribbon, select the **Global** dropdown menu. The Global fidelity options appear.
2. For **Automatic mesh refinement**, click **Off** and select **Program controlled** instead.

20.2. Designing with Parameters in Defined Variations

Explore and evaluate alternatives using parameters. By defining physics and geometric parameters, you can automate the simulation and exploration of the variations while retaining your original design. This allows you to investigate parameterized geometry changes or changes to physics conditions.

A defined variation (DV*n*) is an instance of the simulation that uses a set of values for all defined parameters. Output parameter values are calculated for each variation that is solved. Create new variations (manually or generate automatically) to solve all or selected variations, and then explore the solved variations in the variations chart.

In the Discovery application, you can make interactive modifications to the geometry and perform interactive solves, or you can set the current defined variations to revert to a set of defined parameters. The defined variations toolbar, in the defined variations table, gives you complete control over any of your variations to optimize your simulation.

You may see the following tabs in the defined variations table:

- Script Parameters: Modify parameter values of parameters defined in scripts
- Parameters: Manage parameterized values such as physics parameters and geometric driving dimensions
- Test Cases: Create test cases by grouping physics parameters
- Variations: Design alternatives using defined parameters

Note: When working with time-dependent simulations, specify a time duration (**Simulation** > **Physics** > **Simulation Options**) before solving for all variations.

This section contains the following topics:

20.2.1. Defined Variations Parameters

A *parameter* is a characteristic of a model or a simulation that you can quantify and vary to determine its effect on the results of the simulation. A simulation can have input parameters and output parameters:

- *Input parameters* define the values to analyze for the model under investigation. Inputs eligible for parameterization display the **Add as a parameter** icon. This icon may appear in the tree to the right of the condition, semi-halo above the condition's HUD, or to the right of a value input box. Input parameters may include:
  - geometric dimensions
  - physics conditions
  - local fidelity
  - topology optimization options
  - materials
  - script parameters
- *Output parameters* are the response outputs from the analysis and are defined with monitors. They may include any measurable quantity from the simulation or model, such as: volume, mass, frequency, stress, velocities, pressures, forces, and heat flux. They may also include environmental properties for some materials. The value of an output parameter is based on the current state.

Parameters are added to the defined variations table where you can manage parameterized values and design alternatives using defined parameters.

This section contains the following topics:

20.2.1.1. Geometry Available as Parameters

Define geometric parameters in the following ways using the **Pull** and **Move** tool options:

- Use Ruler to drive the dimension and click **Add as a parameter** next to the object's measured dimension.
- Use **Add a geometric parameter** to enter the **Pull - Parameterize** tool in the **Pull** and **Move** tools.

In the **Measure** tab, you can create output parameters for the following tools:

- **Measure tool**
- **Mass Properties tool**
- **Environmental Properties tool**

Parameters appear in the defined variations table.

20.2.1.2. Physics Available as Parameters

Some physics conditions can be parameterized by selecting the **Add as a parameter** icon. This icon may appear in the tree to the right of the condition and in the semi-halo above the condition's HUD. In addition, you can parameterize **Local fidelity** to consider mesh sizing and lower or higher result resolutions.

All physics conditions are listed in the **Manage Physics Parameters** panel where you control whether a condition is (or is not) parameterized through simple selection. You can access this panel in the **Simulation** tab, in the **Study** group of the ribbon. From the **Parameters** dropdown menu, select **Manage Physics Parameters**.

Fluid Flow Parameters

Condition	Constraints
Inlet	temperature, velocity, mass flow rate, pressure

Condition	Constraints
Outlet	pressure, velocity, mass flow rate
Walls	convection temperature and coefficient, heat flow, heat flux, temperature

Solid Thermal Parameters

Condition	Constraints
Convection	convection coefficient, external temperature
Heat	total heat volume, total heat area
Temperature	

Structural Parameters

Condition	Constraints
Acceleration	magnitude, components
Bearing load:	radial force, thrust
Bolt preload	
Displaced support:	displacement
Displacement	rotation, translation, remote point location (X/Y/Z)
Force, distributed	total force, force per unit area, remote point location (X/Y/Z)
Force, torque	magnitude
Pressure	magnitude
Mass	magnitude, remote point location (X/Y/Z)
Moment	
Spring	
Velocity	magnitude, components

Electromagnetics Parameters

Condition	Constraints
Circuit port	impedance, magnitude, phase
Mode port	impedance, magnitude, phase

20.2.1.3. Topology Optimization Options Available as Parameters

The following topology optimization options are available as parameters:

For the Maximum Stiffness, Maximum Natural Frequency, Balance Stiffness & Frequency, Minimize Stress objectives:

- Volume reduction (either % or by volume)

- Target Natural Frequency objective
- Target Frequency

For the Minimize Volume objective, with the Stiffness Reduction option: Aggressiveness

For the Minimize Volume objective, with Von Mises Stress or Principal stress option: Maximum stress

For the Minimize Volume objective, with Factor of Safety: Factor of safety

> **Note:** When solving a topology optimization, geometric output properties are not updated (including environment parameters).

20.2.1.4. Script Parameters

When creating a script intended for use with defined variations, limit the script to setup only and not include a solve step. Once you embed a script, the **Script Parameter** tab appears in the **Parameter Study** window and you can use script parameters as input to control variations.

Defined variations using input parameters can be updated and solved locally or remotely on the cloud.

Toolbar functionality includes:

- **Add** a script parameter to the table
- **Autorun** parameter script to run the script continuously, as you make parameter updates and gather results. ON by default.
- **Open Script Editor** opens the editor
- **Delete** selected rows

You can interactively:

- rename a parameter
- modify the value
- select from quantity types: Unitless, Length or Angle

> **Note:** Script parameters may be combined with other parameter types in the Variations table. However, script methods "ClearAll" and "ClearAllForParameters" may be problematic when used with non-script parameters and should be avoided.

> **Note:** Defined variations do not support using input script parameters in combination with other input parameters.

20.2.2. Defined Variations Table

Geometry properties and conditions that you add as parameters appear in the defined variations table. Access the table by clicking the defined variations hex in the SID, or click **Parameters** in the **Simulation** tab ribbon.

The defined variations table allows the selection of parameters and/or test cases that are used together to create a defined variation. Parameter values can be entered per variation and multiple variations solved. The current configuration can also be selected to update the geometry.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

The current variation can be changed to see the simulation input (geometry configuration and physics inputs).

> **Important:** Changes may cause some output parameters to go out-of-date. These will require an update by solving. The out-of-date icon may appear in the table or the chart and will resolve once the variation is solved.

Best Practice: When adding parameters to a variation table, there should be no dependencies between parameters.

Use the defined variations table to create, manage and solve defined variations. Each row in the table represents a defined variation and can be selected using a checkmark or star. All functionality resides in the toolbar:

- **Add** a variation manually
- **Generate defined variations** automatically based on an algorithm or manually based on a range of values
- **Set as current** a checked row
- **Update** current, starred or all variations whether local or remote
- **Show/hide columns** in the table
- **Duplicate** a variation
- **Show/hide chart**
- **Results comparison dashboard**
- **Export to CSV**
- **Sensitivity analysis** to identify better input parameter values
- **Optimization analysis** to identify variations that meet defined critera
- **Delete** a variation

This section contains the following topics:

20.2.2.1. Defined Variations Generator

The generator creates variations based on:

- one or more parameters that you've identified,
- the Latin Hypercube Sampling (LHS) algorithm or a range of values that you've defined and either the interval for each variation within that range, or the number of variations you want to pull out of that range
- the defined variation that will be used for unmodified parameter values
- select test cases

As you identify the parameter and define value ranges, the generator gives you immediate feedback on the number of variations that will be created. Once created, these newly generated variations populate to the bottom of the defined variations table.

The **Latin Hypercube Sampling** (LHS) is a stochastic method commonly used in computer experiments and simulations to generate random samples from multivariate and multidimensional distributions and is the bases to start making predictions based on simulations results and analyze them. In a way that ensures that each dimension is represented evenly across the sample. LHS is a type of stratified sampling that divides the range of each input variable into equally probable intervals and then selects one and only one sample in each interval for each variable. The result is a set of samples that covers the entire range of each input variable and that has a relatively low correlation between samples. This helps to ensure that the samples are representative of the underlying distribution and that they provide accurate estimates of the model output.

> **Note:** See Methods for Parametric Design Optimization for a deeper understanding of the LHS algorithm.

20.2.2.2. Defined Variations Chart

Solved defined variations are plotted as points on the chart. This visual representation makes it easier to compare different combinations of parameters (inputs) and computed values (outputs). Visual representation includes color and shapes of the points. Hover over a point to see the associated values and the variation it represents. Right-click to access more options such as **Export to CSV**.

A third input/output can be plotted onto the chart using the highlight dropdown. This colors the points based on the selected input/output on a color scale from red (maximum value) to green (minimum value).

For any target values you set in your monitor charts, those limits will be displayed on the Defined Variations chart as red highlighted sections.

You can generate a project report showing details about your project. Output parameter monitor data for the current defined variation is displayed in the report.

20.2.3. Defined Variations Test Cases and Generator

A test (or load) case is a collection of physics parameters that can be used with multiple defined variations.

Use the Test Cases tab to create and manage test cases. Each row in the table represents a test case and can be selected using a checkmark. All functionality resides in the toolbar:

- **Add** a test case manually
- **Generate test cases** for a range of values automatically
- **Show/hide columns** in the table
- **Duplicate** a test case
- **Delete** a test case
- **Manage physics conditions**

The Test Cases Generator creates test cases based on:

- one or more identified parameters that you choose,
- a range of values that you've defined and either the interval for each within that range, or the number of test cases you want to pull out of that range

As you identify the parameter and defined value ranges, the generator gives you immediate feedback on the number of test cases that will be created.

20.2.4. Defined Variations and Ansys Cloud (Explore)

Remote solving in Ansys Cloud provides a way to accelerate the computation of large numbers of defined variations by performing the simulations in parallel on Ansys cloud infrastructure instead of serially on your local hardware. Ansys cloud resources are always available and can solve hundreds of design variations in a fraction of the time it would take to solve locally. The potential time savings is larger for simulations that take longer to run, such as fluids and topology optimization scenarios.

To solve remotely in Ansys Cloud, your company must subscribe to an Ansys Cloud plan. Each plan includes a pool of cloud compute credits and provides cloud access to a limited number of users. Ansys Cloud credits are consumed based on the compute time for each simulation. Ansys Cloud calculates and reports the estimated cost for solving all submitted design variations based on physics and model fidelity. Actual credit consumption will be based on total solving time and may not match the estimate.

Use

- **Ansys Engineering Portal** (https://portal.ansys.com) to view and manage simulation jobs that you submitted to Ansys Cloud. For more information, see the [broken link to ansportal_guide] .

- **Ansys Admin Portal** (https://admin.ansys.com/) to manage your account. For more information, refer to the [broken link to ansadmin_guide] .

> **Note:** The following are known limitations:
>
> - Transient simulations (fluids and thermal) without an end time are not supported.
> - Estimates may vary from actual run times.

This section contains the following topics:

20.2.4.1. Activating Cloud Capability

To activate Ansys Cloud services within the Discovery application once you have satisfied the cloud prerequisites (see Requirements for Cloud Burst Computing):

1. Select **Remote solve in Explore (restart required)** from the **Remote Services** tab in the **Settings** panel.
2. Restart the application.
3. **Sign In** from the top right. If you do not have an Ansys ID, you are prompted to create one.

20.2.4.2. Remote Solving of Defined Variations (Explore)

Remotely solved simulations use the same fidelity as if solving locally although you can set a fidelity higher than what would successfully solve on your local machine by setting the **Fidelity approach** option in the **Global** dropdown list in the Simulation ribbon tab.

To solve defined variations remotely in the Explore stage:

1. Select the variations in the **Variations** table that you want to solve.
2. Select **Remote** from the **Variations** > **Update** dropdown list in the toolbar.
3. Review the **Submission Details**, select the billing **Account** from the dropdown list and **Submit**.

Each defined variation is generated using its defined input parameter values and sent to the Ansys Cloud to simulate. Changes should not be made to the model while defined variations are being generated. As a remote session completes, monitor values within the design variations table will update with calculated values. No graphical results are returned from remote solving runs.

Use

- **Ansys Engineering Portal** (https://portal.ansys.com) to view and manage simulation jobs that you submitted to Ansys Cloud. For more information, see the [broken link to ansportal_guide] .
- **Ansys Admin Portal** (https://admin.ansys.com/) to manage your account. For more information, refer to the [broken link to ansadmin_guide] .

> **Note:** Estimates shown in the submission details are approximations based on usage and will improve over time.

To cancel a remote solution session, select **Stop variations update**. All queued or running simulations will be cancelled and no additional credits will be consumed.

You can follow the progress of updates:

- A status bar on the bottom of the **Variations** table shows the state of updates from queued to solved, and various states in between.
- A progress bar in the Discovery blue footer displays the update progress.
- Operational messages and temporary status messages inform the state of the remote results download.

> **Attention:** Save your Discovery project while remote solving is in progress as results may not be recoverable if you close without saving. Remote solving continues in the cloud even when the application is closed so saving gives a measure of certainty that your results will download next time you launch the application.

20.3. Sensitivity Analysis and Predicting Quality Results

Defined variations can be analyzed and evaluated using sensitivity analysis to analyze the predictive approximation quality and input parameter sensitivities. A sensitivity analysis results in a sensitivity matrix, also known as a Coefficient of Prognosis (**CoP Matrix**). This analysis is a visualization of the effect of parameterized input on outputs.

Once the sensitivity analysis is complete, the panel is refreshed to display the CoP matrix and its corresponding 3D interactive response surface plot. The predictive nature of the matrix helps you in deciding whether to continue with the current model or whether additional sampling data or inputs are needed for improvement. The matrix is comprised of:

- Individual columns for each parameterized input containing single CoP values so the most significant input parameters can be identified.
- A summary column quantifying the overall prediction quality of the underlying approximation model (total CoP), where a high value indicates greater reliability of the single CoP estimates.

A **Total CoP** of 95% or greater indicates that the quality of the predictive model is sufficient to make design decisions. Less than 95% may indicate a need to add additional variations, remove outlier variations or improve the simulation setup to ensure answers are consistently converged.

The **Response Surface 3D Plot** plot displayed in **Sensitivity Analysis** visualizes the predicted response based on inputs. The response surface can display any combination of two input parameters on the x and y axes. If there are more than two input parameters, any additional input parameters are approximated by calculating the midpoint of the range indicated in the defined variations at the time of submission.

20.3.1. Running a Sensitivity Analysis

To run a sensitivity analysis, begin with some number of defined variations, where more variations is the better starting point.

1. In the Variations table, select **Update All** to solve the variations.
2. Once all the variations are updated, select **Sensitivity analysis** from the **Variations** toolbar to open the **Sensitivity Analysis** panel.
3. Select **Run Analysis**. This sends the variations data to the optimization engine and returns the results, refreshing the panel with the forecast quality results and 3D response surface plot.

   Probe the surface to see values and change axes to modify orientation.

If the analysis results appear out of date, re-run the analysis by selecting **Run Analysis** again.

Should you want more post-processing options, **Transfer to optiSLang** transfers and automatically opens the **Postprocessing Window** containing the data from the newly run analysis.

21: Mesh Evaluation

Mesh diagnostics help you evaluate mesh quality. In the Discovery application, you can obtain statistical information about your mesh, such as node and element counts. You can also get information about the quality of the mesh, which is determined by several quality metrics. A good quality mesh guarantees the best analysis results for the problem, minimizes the need for additional analysis runs, and improves your predictive capabilities.

You have several options for evaluating your mesh. You can set up monitors to monitor mesh quality and metrics. In addition, you can also specify mesh metrics that are monitored during the solve.

> **Note:** Mesh statistics, quality, and metrics are available only in **Refine**.

This section contains the following topics:

21.1. Monitoring Mesh Statistics

To report on the number of elements and nodes in the generated mesh:

1. From the **Monitors** menu in the ribbon of the **Simulation** tab, select **Custom**.
2. Select **Mesh Statistics** from the variable list in the **Monitors** HUD.

> **Note:** For structural simulations, statistical information pertaining to the mesh is available only in **Refine**.

3. Select the geometry for which you want mesh information.
4. Choose **Element count** or **Node count** in the options panel to obtain a report on the number of nodes or elements in the generated mesh.
5. Click the green check mark to accept the settings.

Once the mesh is generated, the number of nodes and elements are reported in the Notification hex of the SID as well as the monitor charts.

21.2. Monitoring Mesh Quality

Diagnostic information pertaining to the quality of the mesh is available only in the **Refine** stage.

To report on the quality of the generated mesh:

1. From the **Monitors** menu in the ribbon of the **Simulation** tab, select **Custom**.
2. Select **Mesh Quality** from the variable list in the **Monitors** HUD.
3. Select the geometry for which you want mesh information.
4. Choose one of the following metric types to evaluate the quality of the generated mesh:
   - Element quality
   - Aspect ratio
   - Jacobian ratio
   - Warping factor

- Parallel deviation
- Maximum corner angle
- Skewness
- Orthogonal quality

The minimum, maximum, average, and standard deviation values are reported for the selected metric type.

5. Click the green check mark to accept the settings.

Once the mesh is generated, the selected quality metric is reported in the monitor chart.

21.2.1. 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:

$$Quality = C\left(area \Big/ \sum (EdgeLength)^2\right)$$

- For three-dimensional brick elements:

$$Quality = C\left[volume \Big/ \sqrt{\left[\Sigma (Edge\ length)^2\right]^3}\right]$$

For each type of element, the following table lists the value of *C:*

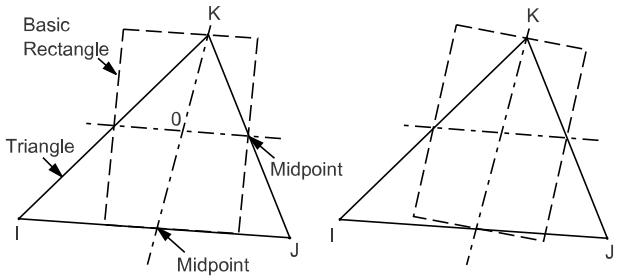
Element	**Value of *C***
Triangle	6.92820323
Quadrangle	4.0
Tetrahedron	124.70765802
Hexagon	41.56921938
Wedge	62.35382905
Pyramid	96

21.2.2. Aspect Ratio

Calculation for Triangles

The aspect ratio for a triangle is computed in the following manner, using only the corner nodes of the element:

Triangle aspect ratio calculation



1. A line is constructed from one node of the element to the midpoint of the opposite edge, and another through the midpoints of the other two edges. In general, these lines are not perpendicular to each other or to any of the element edges.
2. Rectangles are constructed centered about each of these two lines, with edges passing through the element edge midpoints and the triangle apex.
3. These constructions are repeated using each of the other two corners as the apex.
4. The aspect ratio of the triangle is the ratio of the longer side to the shorter side of whichever of the six rectangles is most stretched, divided by the square root of 3.

The best possible aspect ratio, for an equilateral triangle, is 1. A triangle having an aspect ratio of 20 is shown in the figure below.

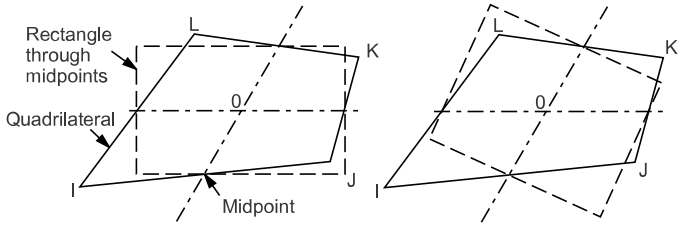
Aspect ratio for triangles



Calculation for Quadrilaterals

The aspect ratio for a quadrilateral is computed by the following steps, using only the corner nodes of the element:

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 two 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.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

The best possible aspect ratio, for a square, is 1. A quadrilateral having an aspect ratio of 20 is shown in the figure below.

Aspect ratio for quadrilaterals



21.2.3. Jacobian Ratio

Jacobian ratio is computed and tested for all elements except triangles and tetrahedra that (a) are linear (have no midside nodes) or (b) have perfectly centered midside nodes. A high ratio indicates that the mapping between element space and real space is becoming computationally unreliable.

Jacobian Ratio Calculation

An element's Jacobian ratio is computed by the following steps, using the full set of nodes for the element:

1. 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.

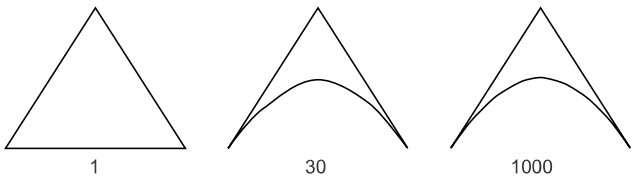
Element Shape	**Element Shape**
Element Shape	R_J Sampling Locations
10-node tetrahedra	corner nodes
5-node or 13-node pyramids	base corner nodes and near apex node (apex R_J factored so that a pyramid having all edges the same length will produce a Jacobian ratio of 1)
8-node quadrilaterals	corner nodes and centroid
20-node bricks	all nodes and centroid
all other elements	corner nodes

2. The Jacobian ratio of the element is the ratio of the maximum to the minimum sampled value of R_J. 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 four corner nodes. 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.
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 2D or 3D 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 degrees.

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 otherwise 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 increase the Jacobian ratio. Eventually, even very slight further movement will break the element. Refer to the

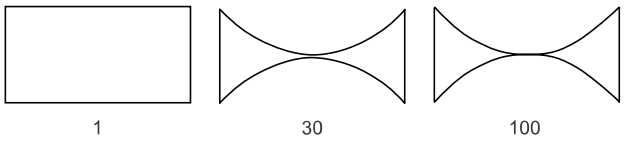
figure below. This is described as "breaking" the element because it suddenly changes from acceptable to unacceptable (that is, "broken").

Jacobian ratios for triangles



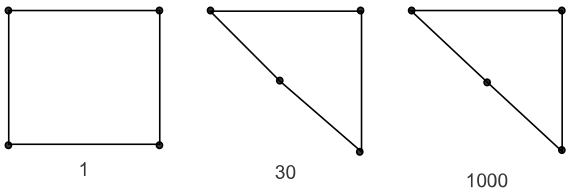
Any rectangle or rectangular parallelepiped having no midside nodes, or having midside nodes at the midpoints of its edges, has a Jacobian ratio of 1. Moving midside nodes toward or away from each other can increase the Jacobian ratio. Eventually, even very slight further movement will break the element. Refer to the figure below.

Jacobian rations for rectangles



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 climbs. Eventually, any further movement will break the element. Refer to the figure below.

Jacobian ratios for quadrilaterals



21.2.4. 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.

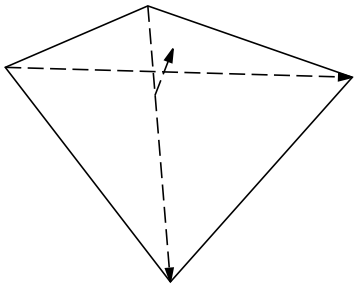
Warping Factor Calculation for Quadrilateral Shell Elements

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 Discovery application assumes 0 thickness for the shell.

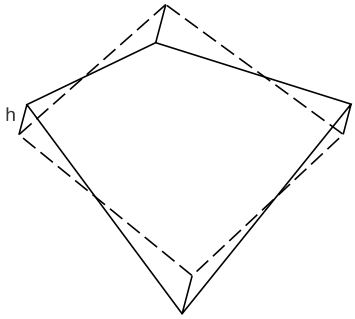
1. An average element normal is computed as the vector (cross) product of the two diagonals.

Shell average normal calculation



2. The projected area of the element is computed on a plane through the average normal (the dotted outline in the figure below).
3. The difference in height of the ends of an element edge is computed, parallel to the average normal. In the figure below, 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.

Shell element projected onto a plane



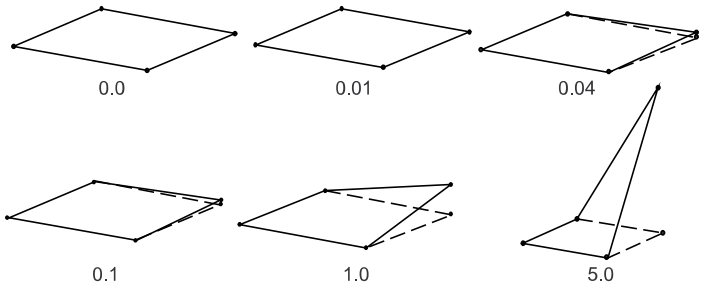
4. The "area warping factor" (F_a^w) for the element is computed as the edge height difference divided by the square root of the projected area.
5. For all shells except those in the "membrane stiffness only" group, if the thickness is available, the "thickness warping factor" is computed as the edge 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.

The figure below 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.

When the upper element is warped by a factor of 0.04, it just begins to visibly separate from the flat one.

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.

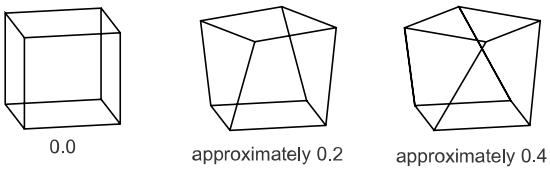
Warping beyond 1.0 would appear to be obviously unacceptable, however, SHELL181 permits even this much distortion. Furthermore, the warping factor calculation seems to peak at about 7.0. Moving the node further off the original plane, even by much larger distances than shown here, does not further increase 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 3D Solid Elements

The warping factor for a 3D solid element face is computed as though the four 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 six quadrilateral faces of a brick, three quadrilateral faces of a wedge, or one quadrilateral face of a pyramid. Any brick element having all flat faces has a warping factor of zero. Refer to the figure below.

Warping factor for bricks



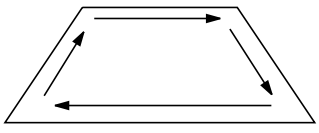
Twisting the top face of a unit cube by 22.5 degrees and 45 degrees relative to the base produces warping factors of about 0.2 and 0.4, respectively.

21.2.5. Parallel Deviation

Parallel deviation is computed using the following steps:

1. Ignoring midside nodes, unit vectors are constructed in 3D space along each element edge, adjusted for consistent direction, as demonstrated in the figure below.

   Parallel deviation unit vectors

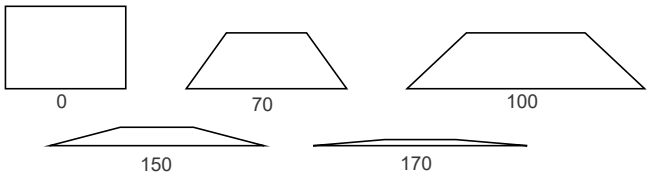
   

2. For each pair of opposite edges, the dot product of the unit vectors is computed, then the angle (in degrees) whose cosine is that dot product is computed. The parallel deviation is the larger of these two angles. (In the

illustration above, the dot product of the two horizontal unit vectors is 1, and acos (1) = 0 degrees. The dot product of the two vertical vectors is 0.342, and acos (0.342) = 70 degrees. Therefore, this element's parallel deviation is 70 degrees.)

3. The best possible deviation, for a flat rectangle, is 0 degrees. The figure below shows quadrilaterals having deviations of 0 degrees, 70 degrees, 100 degrees, 150 degrees, and 170 degrees.

Parallel deviations for quadrilaterals

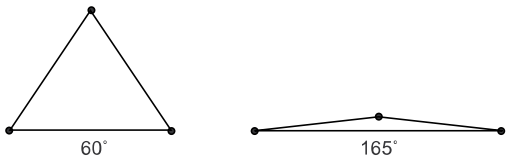


21.2.6. Maximum Corner Angle

The maximum corner angle is computed as follows.

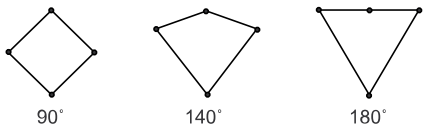
The maximum angle between adjacent edges is computed using corner node positions in 3D space. (Midside nodes, if any, are ignored.) The best possible triangle maximum angle, for an equilateral triangle, is 60 degrees. The figure below shows triangles having maximum corner angles of 60 degrees and 165 degrees.

Maximum corner angles for triangles



The best possible quadrilateral maximum angle, for a flat rectangle, is 90 degrees. The figure below shows quadrilaterals having maximum corner angles of 90 degrees, 140 degrees, and 180 degrees.

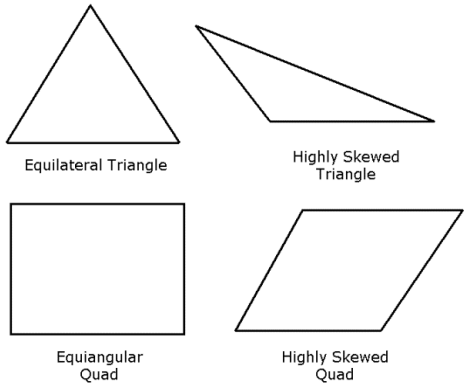
Maximum corner angles for quadrilaterals



21.2.7. Skewness

Skewness is one of the primary quality measures for a mesh. Skewness determines how close to ideal (that is, equilateral or equiangular) a face or cell is.

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 to < 1	bad (sliver)
0.75 to 0.9	poor
0.5 to 0.75	fair
0.25 to 0.5	good
> 0 to 0.25	excellent
0	equilateral

According to the definition of skewness, a value of 0 indicates an equilateral cell (best) and a value of 1 indicates a completely degenerate cell (worst). Degenerate cells (slivers) are characterized by nodes that are nearly coplanar (collinear 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, for example, pyramids and prisms.

Equilateral-Volume-Based Skewness

In the equilateral volume deviation method, skewness is defined as

$$\text{Skewness} = \frac{\text{Optimal Cell Size} - \text{Cell Size}}{\text{Optimal Cell Size}}$$

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.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

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 software computes both Equilateral-Volume-Based Skewness metrics (for the triangular faces) *and* Normalized Equiangular Skewness metrics (for the quadrilateral faces and 3D 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

$$\max\left[\frac{\theta_{max}-\theta_e}{180-\theta_e}, \frac{\theta_e-\theta_{min}}{\theta_e}\right]$$

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 (for example, 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 four 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.

21.2.8. 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,

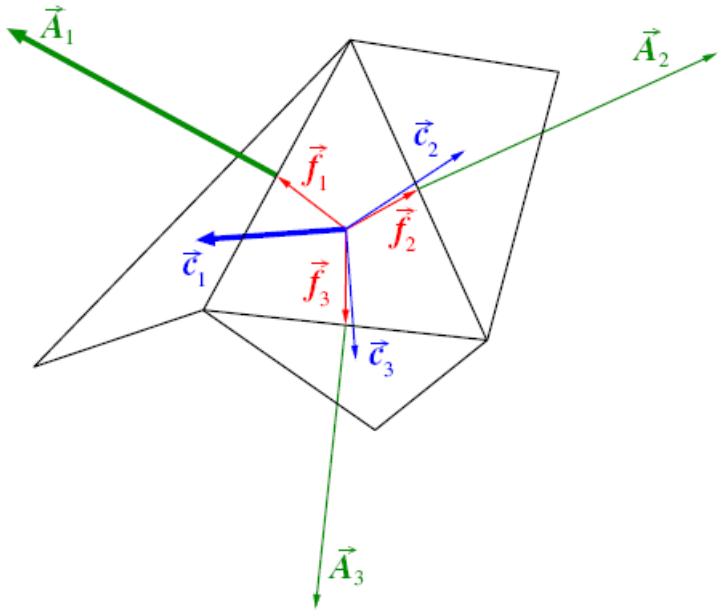
$\vec{c}_i$

; and the vector from the cell centroid to each of the faces,

$\vec{f}_i$

. The figure below illustrates the vectors used to determine the orthogonal quality for a cell.

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:

- 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 (for example, 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 relationship, 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.

In a similar way, orthogonal quality for faces is computed as the smallest cosine of the angle between the edge normal vector,

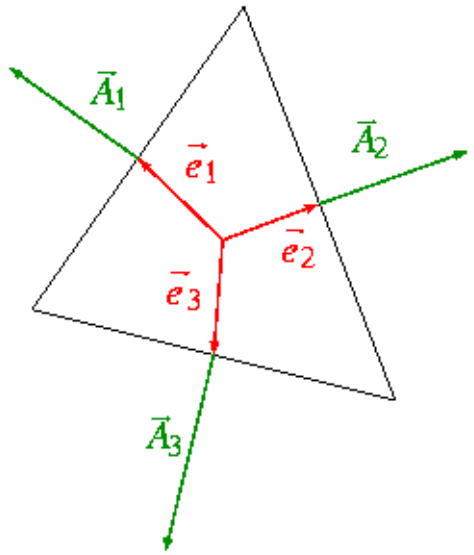
$\vec{A}_i$

, for each edge and the vector from the face centroid to the centroid of each edge,

$\vec{e}_i$

. The figure below illustrates the vectors used to determine the orthogonal quality for a face.

Vectors used to compute orthogonal quality for a face



21.3. Mesh Metrics

You can set the mesh metrics monitored during the solve and specify limits for those metrics. After meshing, you are notified if any selected meshing criteria are not met.

For fluids simulations, those include:

- Orthogonal quality (on by default) – lower limit (default 0.1)
- Skewness (off by default) – upper limit (default 0.95)
- Aspect ratio (off by default) – upper limit (default 20)

For structural simulations, those include:

- Element quality (off by default) – lower limit (default 0.05)

- Jacobian ratio (on by default) – upper limit (default 30)
- Aspect ratio (off by default) – upper limit (default 20)

Default mesh metrics are set in the in the **Physics Settings**, but you can also specify mesh metrics for this particular simulation under **Simulation Options** on the **Simulation** tab. Click **Fluid Mesh Quality** or **Structural Mesh Quality** to see the options in the expanded panel.

You can then enable mesh plots highlighting poor quality elements.

> **Note:** Mesh metrics are only available in **Refine**.

21.3.1. Viewing Mesh Metrics for the Simulation

Default mesh metrics can be set in the in the **Physics Settings**, but you can also specify the mesh metrics and their limits for this particular simulation under **Simulation Options** on the **Simulation** tab. Click **Fluid Mesh Quality** or **Structural Mesh Quality** to see the options in the expanded panel. After meshing, you are notified if any selected meshing criteria are not met.

Then, after simulation, you can view mesh plots that highlight poor quality elements.

> **Note:** Mesh metrics are only available in **Refine**.

1. Select the mesh metric you want to view from the **Resolution → Structural/Fluid options** dropdown list in the **Fidelity** group of the **Simulation** tab.
2. Click **Resolution** to view.

21.4. Mesh Quality Best Practices

Mesh quality plays a significant role in the accuracy and stability of numerical computations. While the Discovery mesh size defaults are intended to provide a reasonable mesh quality based on the defined physics, checking the quality of your mesh is essential to ensure an accurate solution. The following are indicators of mesh quality along with best practices.

Structural Mesh Metrics

Mesh Quality Indicator	Description	Best Practice
Element quality	It provides a composite quality metric that ranges between 0 and 1. A value of 1 indicates a perfectly shaped element while a value of 0 indicates that the element has a zero or negative volume. Learn more about Element Quality.	The optimal Element quality is 1.0, and a high-quality mesh typically avoids elements with an Element quality less than 0.05. Elements with element quality less than 0.05 should be avoided in regions with large stress gradients but may be acceptable if the primary result of interest is displacement.

Mesh Quality Indicator	Description	Best Practice
Jacobian ratio	It compares the shape of a given element to that of an ideal element. If an element has a poor Jacobian ratio, the element may not map well from element space to real space, thereby making computations based on the element shape less reliable. Learn more about Jacobian Ratio.	The optimal Jacobian ratio is 1.0, and a high-quality mesh typically has a Jacobian ratio ranging from 1 to 10 for most of its elements. Elements with large Jacobian ratios (>30) should be avoided in regions with large stress gradients but may be acceptable if the primary result of interest is displacement.
Aspect ratio	A measure of the stretching of an element. It is a measure of the ratio of the longest dimension to the shortest dimension of an element. Learn more about Aspect Ratio.	An aspect ratio of 1 is ideal, and values less than 5 are acceptable. Elements with large aspect ratios (>20) should be avoided in regions with large stress gradients but may be acceptable in areas of low stress gradients or if the primary result of interest is displacement.
Mesh element distribution	The density and arrangement of mesh elements within a computational domain.	The proper resolution of the mesh depends on the desired structural results. For accurate displacement values, a coarser mesh may be acceptable, whereas accurate stress and strain results require more mesh refinement. The mesh should be fine enough to minimize the change stress and strain from element to element especially in regions of stress concentrations.

Fluid Flow Mesh Metrics

Mesh Quality Indicator	Description	Best Practice
Orthogonality	A measure of how close the worst cells are to an ideal orthogonal state. Learn more about Orthogonal Quality.	The worst cells will have an orthogonal quality closer to 0, with the best cells closer to 1. The minimum orthogonal quality for all types of cells should be more than 0.01, with an average value that is significantly higher. Cells with an orthogonal quality of less than 0.1 can affect the rate of solution convergence.
Skewness	The difference between the shape of the cell and the shape of an equilateral cell of equivalent volume. Highly skewed cells can decrease accuracy and destabilize the solution. Learn more about Skewness.	Keep the maximum skewness for a tetrahedral mesh below 0.95, with an average value that is significantly lower. A maximum value above 0.95 may lead to convergence difficulties and may require changing the solver options. Highly skewed cells can decrease accuracy and destabilize the solution.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

Mesh Quality Indicator	Description	Best Practice
Aspect ratio	A measure of the stretching of an element. It is a measure of the ratio of the longest dimension to the shortest dimension of an element. Learn more about Aspect Ratio.	Avoid sudden and large changes in cell aspect ratios in areas where the flow field exhibits large changes or strong gradients. High aspect ratios are not an issue if the direction of the stretch is in the direction of the flow.
Mesh element distribution	The density and arrangement of mesh elements within a computational domain.	Ensure proper resolution of the mesh for both laminar and turbulent flows. The mesh should be fine enough to minimize the change in flow variables from cell to cell, especially in regions of large gradients. No flow passage should be represented by fewer than 5 cells.
Smoothness	The smoothness of the change in cell size during mesh generation or refinement. An abrupt change in cell size can result in solution instability and inaccuracy.	Improve the smoothness of the mesh by refining it based on the change in cell volume or the gradient of cell volume. Rapid changes in cell volume between adjacent cells translate into larger truncation errors. The application uses a default growth ratio of 1.2 for fluid meshing. It ensures that the change in size between cells is less than 20%.

22: Results

This section contains the following topics:

22.1. Displaying Results

Generating results to understand the behavior of an analyzed model is fundamental to any simulation analysis. The Discovery application offers a variety of result display types and tools to present the results you need. Those displayed results update instantly as you interact with your model.

When in **Refine**, you can choose to display the mesh with any evaluated results. Additionally, you can probe the solution anywhere on your model by mousing over the contour or iso-surface in **Refine**, or any result type in **Explore**. A value displays in the legend and changes as you move to a different location in your model.

Displayed results fall into the following categories:

- Scalar result presentation methods:
  - Contours: Visualize as a colored contour to assess the range and rate of change for scalar variables including scalar components of vector or tensor variables. Includes options to display volumetrically.
  - Iso-surface: Display surfaces of user-defined constant values.
- Directional result presentation methods:
  - Vectors: Distribute a number of arrows to show the direction and magnitude of vector variables on a surface. The arrows can be colored and sized by the magnitude of the vector. For fluid flow simulations, curved vectors provide a way to visualize the global field flow characteristics and explore the flow by velocity or circulation.
  - Streamlines: Show pathlines following the vectors of least turbulence from a seed location. The lines are colored by the selected variable in the results legend.
  - Particles: Show virtual or actual particles traveling along the path lines. These particles can be from a seed location or by filling a volume and are colored by the selected variable in the results legend.
  - Direction field: Display directionality on a surface based on the vector.

> **Note:** Result plots for structural and fluids simulations are combined. Structural simulations display **Displacement** as a vector variable oriented by deformation and then colored by the selected variable. **Reaction Force** and **Reaction Resultant** use their own variable for orientation, rather than displacement. These variables are not available for the **Direction field** plot. Fluids simulations use **Velocity** as the vector variable, which is then colored by the selected variable. Thermal variables are oriented by **Heat Flux**.

- Electromagnetics result presentation methods:
  - Near-field pattern: Display the electric field intensity within the simulation region. Visualize the results as a colored contour to assess the range and rate of change for electromagnetic variables.
  - Far-field pattern: Display the radiated electric field, gain and other antenna parameters versus angle, as observed on an infinite sphere.
  - Voltage Current density and Joule heat for electric simulations.

Results Variables on page 917 lists all commonly used variables based on the physics used in your simulation.

22.1.1. Displaying Vectors

Vectors show the direction and magnitude of variables on a volume. Vector results display either vector variables, geometry direction or legend magnitude, on a collection of seed points defined on a volume.

1. In the Results arc, click **Vectors**.

   You can display the following vectors based on the physics in your simulation:

   - For structural simulations, display **Displacement**, **Reaction Force**, or **Reaction Resultant**.
   - For thermal simulations, display **Heat Flux**.
   - For thermal-structural simulations, display **Displacement**, **Reaction Force**, **Reaction Resultant**, or **Heat Flux**.
   - For fluid flow simulations, display **Velocity**. You can also display curved vectors to visualize the fluid recirculation.
   - For electrical simulations, display **Current Density**.
   - For thermal-electric simulations, display **Current Density**, or **Heat Flux**.

2. Select a variable from the Results legend list to color your vector results by a different variable.

   For example, for a fluid flow simulation, you can display **Velocity** vectors and have them colored by **Temperature** (if that is the variable you selected from the list in the legend).

3. Analyze and display your results in different ways.

   For fluid flow simulations, the following options are available:

Curved	Selected by default for fluid flow simulations to display curved vectors. Curved vectors provide a way to visualize the global field flow characteristics, and explore the flow by velocity or circulation.
Thickness	Controls the width of the vector.
Tail length	Trims away some of the history of where vectors have been.
Count	Increases or decreases the number of vectors displayed.
Filter by	Displays values based on either **Magnitude** (value of) or **Gradient** (rate of change). Use the slider to change the maximum and minimum threshold of curved vectors to be displayed.
Invert range	Flips the maximum and minimum thresholds.
Reset	Resets the vector to its original range.

   For all other physics types, the following options are available:

Number	Controls the number of displayed arrows.
Arrow size	Controls the size of the displayed arrows.
Constant size	Sizes all arrows equally. If constant size is not checked (the default), the arrow size is based on the magnitude of the variable at that point.

22.1.2. Displaying Contours

Contours help visualize and assess gradients for variables displayed on a surface or volume. The regions between the contour lines are filled with a color that denotes the value of the variable in that region.

Note: Fluid and fluid-solid thermal contour results in the Explore stage, when viewed without the mesh display, show cell based values, which display the near-wall result values and may not reflect an assigned condition. However, contour results, when viewed with a mesh display, show the nodal results and reflect the values of assigned conditions.

1. In the Results arc, click **Contours**.
2. Select a variable from the Results legend list to display contour results.
   - For structural simulations, display contours for quantities such as **Displacement**, **Stress**, **Strain**, **Reaction Force**, or **Reaction Resultant**.
   - For thermal simulations, display **Temperature** or **Heat Flux**.
   - For thermal-structural simulations, display **Displacement**, **Reaction Force**, **Reaction Resultant**, or **Heat Flux**.
   - For fluid flow simulations, display **Velocity**, **Pressure**, **Temperature**, or **Vortices Lambda**. **Temperature** is only available if you have added a temperature condition, and **Vortices Lambda** is available only in **Explore**.
   - For electrical simulations, display **Voltage**, **Current Density**, or **Joule Heat**.
   - For thermal-electric simulations, display **Voltage**, **Current Density**, **Joule Heat** or **Heat Flux**.
3. Analyze and display your results in different ways using the contours options.

Location	**All Faces** displays surface contours and **All Bodies** displays volume contours. When volume contours are displayed on bodies, a range clipper is visible alongside the colormap. Slide the ends of the clipper to set the range of values to display.
Surface display priority	Sets the **Inner**, **Outer**, **Highest Value** or **Lowest Value** for surface contours.

Tip: In the Results arc, use **Cut plane** to create cross sections of your model where you can view results at the different locations within the model. Drag the plane anywhere in the volume to slice the results. Hover over the plane border to get a drag arrow for moving the plane. You can select the **Move** tool (in the **Design** tab ribbon or press M) to rotate the plane.

22.1.3. Displaying Iso-Surfaces

Iso-surfaces display a surface of constant value for a specified variable.

1. In the Results arc, click **Iso-Surface**.
2. Select a variable from the Results legend list to display iso-surface results.
   - For structural simulations, display contours for quantities such as **Displacement**, **Stress**, **Strain**, **Reaction Force**, or **Reaction Resultant**.
   - For thermal simulations, display **Temperature** or **Heat Flux**.
   - For thermal-structural simulations, display **Displacement**, **Reaction Force**, **Reaction Resultant**, or **Heat Flux**.
   - For fluid flow simulations, display **Velocity**, **Pressure**, **Temperature**, or **Vortices Lambda**. **Temperature** is only available if you have added a temperature condition, and **Vortices Lambda** is available only in **Explore**.
   - For electrical simulations, display **Voltage**, **Current Density**, or **Joule Heat**.
   - For thermal-electric simulations, display **Voltage**, **Current Density**, **Joule Heat** or **Heat Flux**.
3. Enter **Iso-Value 1** to get the magnitude of a specified variable on any surface.

In **Refine** you can have up to five iso-values. Use the **Number** slider to add more iso-values. Additional options are available:

Automatic	Automatically generates iso-surfaces based on the **Number** of iso-values you specify. They will be evenly spaced between the minimum and maximum values for the specified variable.
Manual	The specified iso-values are used to display for the selected variable.

22.1.4. Displaying Streamlines

For fluid flow simulations, streamlines help you visualize the lines traveled by massless particles that are released from all inlets in the physics region. You can also use streamlines to identify the source of a particle for which you know the final destination.

1. In the Results arc, click **Streamlines** to display the pathlines of the fluid flow.
2. To emit streamlines from an inlet, click the **Emitter control**. An emitter, represented by a spherical marker with a bounding ring, is added to the first inlet where streamlines will travel from that seed location.

   To emit streamlines from a different location, click and drag the emitter to the desired location.
3. To include additional emitters, click **Add emitter**. The new emitters will be added to subsequent inlets and finally offset from one another at the last inlet. A maximum of 10 emitters can be added.

   You can add emitters to multiple selected geometries.

   To place an emitter anywhere on your model, select any geometry (except bodies) as the location and right-click to choose **Add a Streamline Emitter** from the context menu. Depending on the geometry type, the emitter bounding ring might be elliptical rather than circular.

   > **Note:** When adding an emitter to a line, the Discovery application uses the length of the line as the radius and the normal to be the starting point.

4. Select a variable from the Results legend list to color your streamlines by a different variable.

   You can display streamlines colored by **Velocity**, **Pressure**, **Temperature**, or **Vortices Lambda**. **Temperature** is only available if you have added a temperature condition, and **Vortices Lambda** is available only in **Explore**.
5. Analyze and display your results in different ways using the streamlines options.

Tubes	By default, streamlines are displayed as **Tubes**. In **Refine**, you can display them as **Ribbons** to see the bends and curves of streamlines.
Arrows	Determines the direction of the streamline. Available in **Refine**.
Thickness	Adjusts the thickness of the streamlines.
Tail length	Modifies the length of each streamline by trimming away some of the history of where streamlines have been. Available in **Refine**.
Count	Increases or decreases the number of streamlines displayed.
Direction	Reverses the direction of the streamlines or traces streamlines in the forward and reverse direction when using Bidirectional.

6. Modify the streamlines by clicking the marker and using the **Move** handles to rotate it and move it along the axes. Click and drag the bounding ring to change its size and in turn the density of streamlines displayed.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

An emitter can be **Reset to the default emitter** to relocate it to its original position. Additional emitters will be removed.

To remove emitters, click the spherical marker and right-click to select **Remove Emitter** from the context menu.

22.1.5. Displaying Particles

For fluid flow simulations, particles show individual particles flowing through the volume. Unlike streamlines, by default, particles are uniformly seeded across the entire volume.

1. In the Results arc, click **Particles** to display how particles flow through the fluid.
2. To emit particles from an inlet, click the **Emitter control**. An emitter, represented by a spherical marker with a bounding ring, is added to the first inlet where particles will travel from that seed location.

   To emit particles from a different location, click and drag the emitter to the desired location.
3. To include additional emitters, click **Add emitter**. The new emitters will be added to subsequent inlets and finally offset from one another at the last inlet. A maximum of 10 emitters can be added.

   You can add emitters to multiple selected geometries.

   To place an emitter anywhere on your model, select any geometry (except bodies) as the location and right-click to choose **Add a Particle Emitter** from the context menu.

   > **Note:** When adding an emitter to a line, the Discovery application uses the length of the line as the radius and the normal to be the starting point.

4. Select a variable from the Results legend list to color your particles by a different variable.

   You can display particles colored by **Velocity**, **Pressure**, **Temperature**, or **Vortices Lambda**. **Temperature** is only available if you have added a temperature condition, and **Vortices Lambda** is available only in **Explore**.
5. Analyze and display your results in different ways using the particles options.

Count	Changes the particle count for better visibility.
Range	Clips the displayed results when you move both ends of the slider. Particles with data outside the range are not displayed. This option is available when emitters are not active in the model.
Number	Changes the number of emitted particles when the emitter is included.
Size	Changes the particle size.
Animation speed	Speeds up or slows down the animation of the particle display.
Color	Turns on or off particle colors.

6. Modify the particles by clicking the marker and using the **Move** handles to move it along the axes. Click and drag the bounding ring to change its size and in turn the density of particles displayed.

   Restart the emitters as needed and **Reset to the default emitter** to relocate emitters to their original position. Additional emitters will be removed.

   To remove emitters, click the spherical marker and right-click to select **Remove Emitter** from the context menu.

22.1.6. Displaying Direction Fields

Direction field displays directionality on a surface based on the vector.

1. In the Results arc, click **Direction Field**.
2. Select a variable from the Results legend list to display direction field results.

   A spherical marker with a rectangular boundary appears.
3. Modify the direction field by clicking the marker and dragging it along the plane. Use the Move handles to rotate it and move it along the axes. Click and drag the bounding rectangle to change its size.

   For fluids, direction field is a more advanced display than streamlines. For example, where streamlines produce lines from particle flow, direction field produces a whole image at every step. For thermal results, it shows the heat flux vector field, while for structural results, it displays the displacement vector information.

22.1.7. Displaying Near-Field Patterns

Near-field patterns display the electric field intensity within the simulation region. Visualize the results as a colored contour to assess the range and rate of change for electromagnetic variables. The regions are filled with a color that denotes the value of electric or magnetic field intensity in linear terms or in dB.

1. In the Results arc, click **Near-Field Pattern**.
2. Display your results in different ways using the near-field pattern options. Specify the **Surface display priority** to be **Outer**, **Highest Value**, or **Lowest Value**.
3. Select a variable and specify the frequency and other options in the Results legend.
   a) Select the electromagnetic frequency.
   b) Select a variable and units from the Results Legend list to display near-field results. You can select from **E** (electric field vector in V/m) or **H** (magnetic field strength in A/m).
   c) Select the variable option to display. You can select from **Magnitude**, **Real**, or **Imaginary**.

> **Note:** dB scaling is turned on by default for the magnitude plots, the near-field colors change when dB scaling is toggled. When using dB scaled results, the plot re-scales itself to adjust to the new legend range.

> **Tip:** In the Results arc, use **Cut plane** to create cross sections of your model where you can view results at the different locations within the model. Drag the plane anywhere in the volume to slice the results. Hover over the plane border to get a drag arrow for moving the plane. You can select the **Move** tool (in the **Design** tab ribbon or press M) to rotate the plane.

22.1.8. Displaying Far-Field Patterns

Far-field patterns display the radiated electric field, gain and other antenna parameters versus angle, as observed on an infinite sphere. Specifically, the far field radiation pattern of an antenna is used to quantify how an antenna preferentially emits electromagnetic radiation in some directions over others.

1. In the Results arc, click **Far-Field Pattern**.
2. Use the slider to change the maximum and minimum threshold of far-fields to be displayed.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

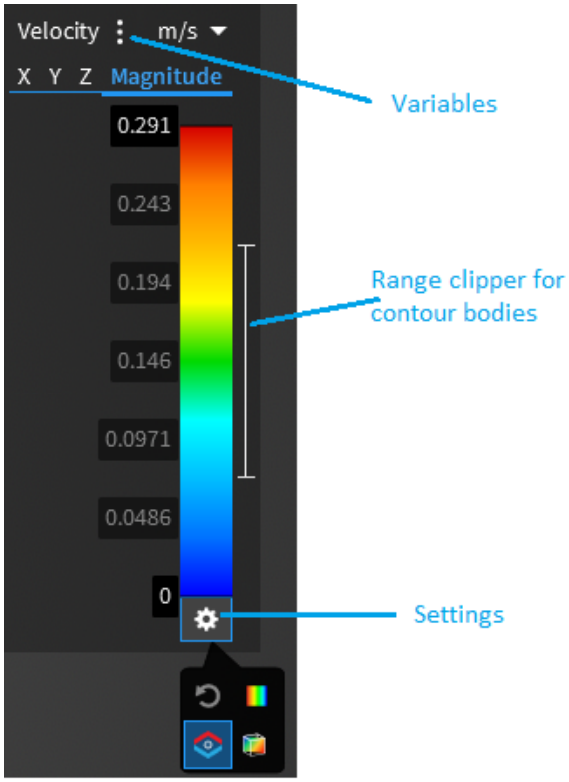
3. Select a variable and specify the frequency and other options in the Results legend.
   a) Select the electromagnetic frequency.
   b) Select a variable from the Results Legend list to display far-field results. You can select from **Gain**, **Directivity**, **Realized Gain**, **rE**, or **Axial Ratio**.
   c) Select the variable option to display. You can select from **Total**, **RHCP**, **LHCP**, **Theta**, **Phi**, **L3X**, **L3Y**, **X**, **Y**, or **Z**.
4. Select the dB scaling to display.

> **Note:** dB scaling is turned on by default. The far-field colors and shape change when dB scaling is toggled.

> **Tip:** Click **Show mesh** in the Results arc to display the mesh. You can see the FDTD grid. Optionally, export a 3D scene. You must turn off the mesh display and only show the far-field pattern.

22.1.9. Results Legend

The results legend displays the numerical range of the results, as well as color bands representing those range of values. The legend is visible after solving your simulation and displaying the results. Click and drag the legend to place it anywhere in the graphics window.



The legend shows the minimum and maximum values of the displayed results for the selected variable. Change the range of values to display by sliding the top border and bottom border of the legend up and down. Alternatively, you can overwrite the minimum and maximum values.

If you've added local coordinate systems for a structural simulation, you can also select a coordinate system for displaying your results. This option is available at the top of the legend.

You can probe the solution anywhere on your model by mousing over the contour or iso-surface in **Refine**, or any result type in **Explore**. A value is displayed in the legend and changes as you move to a different location in your model.

Additional results legend settings are available to help you visualize and analyze your results.

Show min/max locations 	Shows or hides the minimum and maximum value locations on your model.
Reset user defined ranges 	Resets the legend to the default values.
Toggle gradient style 	Switches between displaying results as distinct or smooth color bands.
Display out-of-range results as muted colors 	Switches the color display for out-of-range results. Results that fall outside the maximum and minimum values you set on the results legend will have muted colors.

Changes to the legend settings are saved along with your project and are available when you resume the simulation.

22.2. Working with Results

Once you have displayed your results, there are additional tools to aid you in working with those results.

This section contains the following topics:

22.2.1. Inserting a Cut Plane

To enable **Cut plane**, turn on **Contours**, **Iso-Surface**, **Mesh**, or **Near-Field** and **Far-Field Pattern** for electromagnetic simulations.

Select the **Cut plane** tool to insert moveable cut planes that show results in real time. To insert a cut plane:

1. In the Results arc, click **Cut plane**.
2. Hover over the cut plane in the geometry and an arrow appears, enabling you to move the cut plane.

   If you've added planes from the **Design** tab and you've turned on **Contours**, you can also view results on those planes. Select **Show results** next to the plane in the Model tree or right-click the plane in the tree and select **Show results**. To view the results only for that plane, right-click and select **Show Results on Selection Only**.

> **Note:** The cut plane in the Model tree already has results shown, therefore only user-defined planes will have the option to **Show results** when you've turned on **Contours**. If you are showing results on a user-defined plane in **Explore**, the cut plane results are disabled as is the **Cut plane** tool in the Results arc. Additionally, results on bodies in **Explore** are disabled when showing results on user-defined planes.

22.2.2. Displaying Deformations

You can use the **Deformation** tool in the Results arc to view the structural deflection more easily. This option is available when surface contours of the displacement variable are displayed, as well as when iso-surfaces and vectors are displayed in **Refine**. You can animate the deformation by selecting the **Show animation** button.

> **Note:** Deformations are available for all variables and plots except contours of highest value, lowest value and body plots. Deformations are not available for fluid flow simulations.

To view deformations:

In the Results arc, click **Deformation** once for scaled deformations and a second time for true deformations.

22.2.3. Animating Results

You can use the **Show animation** tool to play back the displacement due to structural deformation. This option is available when **Deformation** results are displayed.

To view animations:

1. View deformation results.
2. In the Results arc, click **Show animation**.

22.2.4. Restarting the Solver

The **Reset** tool enables you to restart the solver. This feature is only available in **Explore**.

To restart the solver:

In the Results arc, click **Reset**.

22.2.5. Adding the Optimized Body to the Model

Once you have a faceted geometry result in a topology optimization, you can save the optimized body to the model. The new faceted geometry is then listed in the Model tree.

For more information on topology optimization workflows, see Topology Optimization. Topology Optimization is only available in the Explore stage.

In the Results arc, click **Add optimized body to model**.

22.2.6. Displaying Charts

Once you have set up charts, you can then view them.

1. In the Results arc, click **Show chart**.
2. Select **Show chart** again to hide the charts.

22.2.7. Displaying the Mesh with Local Fidelity for Fluids

In **Explore**, you can display the mesh on the cut plane and other locations.

1. In the Results arc, click **Mesh**.
   The mesh is displayed on the cut plane by default.
2. Display the mesh on other locations in your geometry.
   - Display the mesh on user-defined planes by showing the results on those planes.
   - Display the mesh on surface bodies by turning off **Cut plane**.
   - Display the mesh on contours with a **Surface display priority** of **Inner** or **Outer**.

22.2.8. Generating a Mesh

In **Refine**, you can generate a mesh then, after it's generated, toggle the mesh view on and off.

More information on evaluating mesh quality is available in Mesh Evaluation.

1. In the Results arc, click **Generate mesh**.
2. After a mesh is generated, toggle the tool to show or hide the mesh.

22.3. Results Tools

The following tools are available from the Results arc.

Direction field	Display directionality on a surface based on the vector.
Particles	For fluid flow simulations, show the individual virtual or actual particles travelling along the path lines and flowing through the volume. The particles can be from a seed location or by filling a volume and are colored by the selected variable in the results legend. Unlike streamlines, particles are uniformly seeded across the entire volume by default.
Streamlines	For fluid flow simulations, visualize the lines traveled by massless particles that are released from an inlet in the physics region. You can also use streamlines to identify the source of a particle for which you know the final destination. Streamlines show the pathlines that follow the vectors of least turbulence from a seed location. The lines are colored by the selected variable in the results legend.
Iso-surfaces	Display a surface of constant value for a specified variable.
Contours	Visualize and assess gradients for variables displayed on a surface or volume. The regions between the contour lines are filled with a color denoting the variable's value in that region. These help you assess the range and rate of change for scalar variables including scalar components of vector or tensor variables. Contours include options to display volumetrically.
Vectors	Show the direction and magnitude of variables on a volume. Vector results display either vector variables, geometry direction or legend magnitude, on a collection of seed points defined on a volume. Vectors are displayed as a number of arrows to show the direction and magnitude of vector variables on a surface. The arrows can be colored and sized by the magnitude of the vector. For fluid flow simulations, curved vectors provide a way to visualize the global field flow characteristics and explore the flow by velocity or circulation.

Cut plane	Insert moveable cut planes that show results in real time. **Note: Cut plane** is available for **Contours**, **Iso-Surface**, **Mesh**, or **Near-Field** and **Far-Field Pattern**.
Deformation	View the structural deflection. This option is available when surface contours of the displacement variable are displayed, as well as when iso-surfaces and vectors are displayed in the Refine stage. Toggle the **Deformations** tool once for scaled deformations and a second time for true deformations. **Note:** Deformations are available for all variables and plots except contours of highest value, lowest value and body plots. Deformations are not available for Fluids simulations.
Show animation	Play back the displacement due to structural deformation. This option is available when Deformation results are displayed.
Reset	Restart the solver. This feature is only available in the Explore stage.
Add optimized body to model	Add the optimized body to the model once you have a faceted geometry result you want to save in a topology optimization. The new faceted geometry is then listed in the Model tree. For more information on topology optimization workflows, see Topology Optimization. Topology Optimization is only available in the Explore stage.
Show chart	Turn on or off the display of charts. See Charts on page 924 for more information.
Mesh	Display the mesh on cut planes. • To view the mesh on user-defined planes, show the results on those planes. • To view the mesh on the surface bodies, turn off **Cut plane**. • To display the mesh on contours, specify **Inner** and **Outer** as the **Surface display priority** This option is available in **Explore** when you specify a local fidelity for fluid flow simulations.
Generate mesh	The Discovery application creates the mesh. After a mesh is generated, click to show or hide the mesh. More information on evaluating mesh quality is available in Mesh Evaluation. This option is available in **Refine**.
Solve	Start or pause the solver.

22.4. Monitoring of Results

You can monitor the results of your simulation by performing quantitative calculations at specific locations on your model. The calculated value of the monitored result allows you to evaluate a single-value quantity. You can use these values, along with other results objects, to evaluate your design in more detail.

This section contains the following topics:

22.4.1. Results Variables

Several variables are available for you to monitor or display results. The list of commonly used monitors or results variables is based on the physics used in your simulation.

The following tables list the variables per physics type.

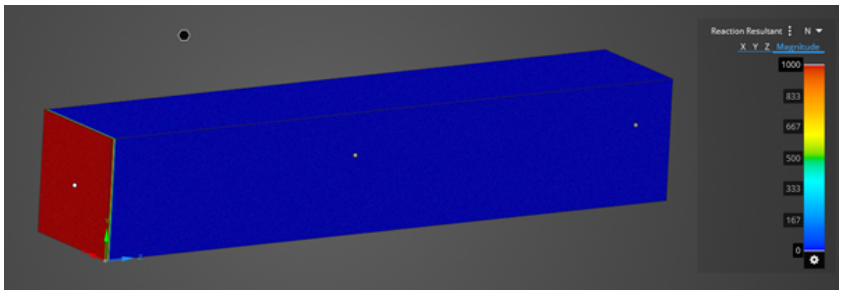
Generic Variables

Mass	The volume of each simulated body multiplied by their respective densities due to the assigned material.
Volume	The amount of space that the bodies in the simulation occupy.
Mesh Statistics	The number of elements and nodes in the generated mesh. For structural simulations, this option is available in **Refine**. See Monitoring Mesh Statistics on page 891 for more information.
Mesh Quality	Diagnostic information pertaining to the quality of the generated mesh. This option is only available in the **Refine** stage. See Monitoring Mesh Quality on page 891 for more information.

Note: There can be slight changes between the mass and volume between stages, this is expected due to the different methodologies. As fidelity increases, the differences will converge.

Structural Variables

Displacement	These are physical deformations that are calculated on and inside a model. Fixed supports prevent deformation; locations without a fixed support usually experience deformation relative to the original location. Deformations are calculated relative to the specified coordinate system.
Von Mises Stress	The first, second and third Von Mises stress (also known as equivalent stress) for the assessment of yield in ductile materials, such as metals.
Stress	A general three-dimensional stress state is calculated in terms of three normal (X, Y, Z) and three shear (XY, YZ, XZ) stress components aligned to the specified coordinate system.
Principal Stress	From elasticity theory, an infinitesimal volume of material at an arbitrary point on or inside the solid body can be rotated such that only normal stresses remain and all shear stresses are zero. The three normal stresses that remain are called the principal stresses. The principal stresses are always ordered such that σ1>σ2> σ3. The principal stresses and maximum shear stress are called invariants, meaning their value does not depend on the orientation of the part or assembly with respect to its specified coordinate system.
Maximum Shear Stress	Maximum shear stress (also known as the Tresca stress) is the difference between the maximum and minimum principal stresses.

Strain (elastic)	A general three-dimensional strain state is calculated in terms of three normal (X, Y, Z) and three shear (XY, YZ, XZ) strain components aligned to the specified coordinate system.
Principal Strain (elastic)	From elasticity theory, an infinitesimal volume of material at an arbitrary point on or inside the solid body can be rotated such that only normal strains remain and all shear strains are zero. The three normal strains that remain are called the principal strains. The principal strains are always ordered such that ε1>ε2> ε3. The principal strains and maximum shear strain are called invariants, meaning their value does not depend on the orientation of the part or assembly with respect to its specified coordinate system.
Force Reaction	The force reaction is the reaction load over a constraint, so the resultant forces balance out the forces applied. For example, if a cantilever beam has a force of 1000N, the resultant magnitude will be 1000N distributed between the constraint faces. **Note:** When applied to hinged, ball and displaced supports and remote displacements, the reaction resultant monitor is automatically applied to the entire constraint location. 
Moment Reaction	The moment reaction is the local moment reaction that a support exerts on a structure to maintain equilibrium when an external moment or load is applied. The reaction moment is calculated about the support centroid (or remote point location). Reaction moments are more "elemental" moments of how the surface behaves.
Contact Reaction	The reaction forces on the contact.
Contact Stress	Defined in terms of either pressure measured normal to the surfaces in contact, or frictional stress measured parallel to the surfaces in contact. For Bonded contacts, the frictional stress is shear stress at the contact location. Frictional stress is only available in the **Refine** stage.

Connection Factor of Safety	For Contacts and Contact Groups: Factor of safety (FOS) is defined by a shear or tensile stress service limit of the bond on a contact or contact group. The shear or tensile limit value is divided by the average contact pressure stress or average contact frictional stress to obtain the factor of safety. Peeling, twisting, or other combined effects are ignored, so this should be used only as guidance. For Bolts (**Refine** stage only): FOS is based on the AISC WSD approach for bolt bearing (no slip), Section J3: • Shear FOS = F_{nv} / f_{rv} Where F_{nv} is the shear stress limit for the bolt equal to $F_{nt} / 1.66$ and F_{nt} is the yield stress limit for the bolt. **Note:** 1.66 comes from the ratio used in table J3.2 of AISC, of bolt minimum yield stress for tension against yield stress for shear when threads are present (worst case). • Tensile FOS = $\min(F'_{nt}, F_{nt}) / f_t$ Where F'_{nt} is the modified yield stress limit for bolts in tension and shear Defined in AISC (eqn J3-3B) for ASD computations as the minimum of F_{nt} or: $1.3 F_{nt} - 2 F_{nt} f_{rv} / F_{nv}$ Replacing F_{nv} with $F_{nt} / 1.66$: $1.3 F_{nt} - 3.33 f_{rv}$ f_t is the maximum tensile stress in this bolt. This is an indication for the bolt strength only and should be used as a guide. Bolt pull through, plate failure, bolt grouping effects, etc. are all ignored, and will need to be assessed separately.
Joint Total Force	The action of the reference body on the moving body.
Joint Total Moment	The action of the reference body on the moving body.
Bolt Axial Stress	The stress component due to the axial load encountered in a bolt.
Bolt Shear Force	The force perpendicular to the bolt axis.
Bolt Torsional Moment	The moment about the bolt axis.
Modal Frequency	The undamped natural frequency response of a structure. Mode monitors are created automatically based on the number of modes being simulated.
Factor of Safety	The ratio of the material's tensile yield strength to the calculated von Mises stress. A factor of safety less than 1.0 indicates that the material will yield under the applied loading.

Fluid Variables

Static Pressure	The fluid gauge pressure.
Total Pressure	The gauge pressure obtained by bringing the fluid isentropically to rest, sometimes referred to as stagnation pressure.
Velocity	The fluid velocity field.
Vortices Lambda 2	A vortex core line detection method used to identify vortices from a three-dimensional fluid velocity field. A point in the velocity field is part of a vortex core only if at least two of its three eigenvalues (ordered as $\lambda_1 \boxtimes \lambda_2 \boxtimes \lambda_3$) are negative, that is, if $\lambda_2 < 0$.
Fan pressure rise	The fan pressure rise is calculated from the fan curve based on the computed volumetric flow. Combined with the volumetric flow, it represents the fan operating point where the fan is in stable equilibrium with the system.
Total Force	The static pressure is used to determine the force of the fluid on the surface (including viscous contributions). Results for fluid-solid interfaces are available only on the fluid side. If specifying the monitor location per condition, any face on the solid or fluid side can be selected, returning results for the entire fluid side. For individual faces, you must select a face on the fluid side.
Pressure Force	The pressure force of the fluid on the surface. Results for fluid-solid interfaces are available only on the fluid side. If specifying the monitor location per condition, any face on the solid or fluid side can be selected, returning results for the entire fluid side. For individual faces, you must select a face on the fluid side.
Viscous Force	The viscous force of the fluid on the surface. Results for fluid-solid interfaces are available only on the fluid side. If specifying the monitor location per condition, any face on the solid or fluid side can be selected, returning results for the entire fluid side. For individual faces, you must select a face on the fluid side.
Mach Number	The ratio of velocity and speed of sound. Available for compressible flows on selected bodies.
Density	The mass per unit volume of the fluid.
Volume Flow	The rate at which fluid flows through the specified flow inlet or outlet, measured by volume.
Mass Flow	The rate at which fluid flows through the specified flow inlet or outlet, measured by mass.
Pressure Drop	The difference in average total pressure measured at the inlets compared with the outlets.
Flow Uniformity	The uniformity index represents how a specified field variable varies over a surface, where a value of 1 indicates the highest uniformity. You can select Velocity or Temperature.
Radiative Heat Flux	The radiative heat flux is the flow of energy due to radiation per unit time per unit area.

Wall Yplus	The dimensionless distance from the wall to the first mesh node. It indicates where in the boundary layer that node lies when wall functions are used to model the near-wall profile. The value should lie between 30 - 300. Details about the equations used for the correlation model of the near wall treatment can be found in the **Fluent Theory Guide**.

Note: If a surface body extends outside the flow region, monitors can report values slightly different from those on a surface body restricted to the flow region.

Thermal Variables

Temperature	The temperature distribution throughout the structure or fluid. This result is available when applied to geometry or loads and constraints.
Heat Flux	The flow of energy per unit time per unit area. **Note:** • On solid thermal bodies in **Explore** and **Refine**, you can monitor the heat flux magnitude or vector components. • On solid thermal bodies in **Explore** and **Refine**, the heat flux is displayed as a vector result for contours and as well as other result displays. • On faces, the monitor is for heat flux monitor normal to the surface. ° In **Refine**, you can apply monitors to faces of fluid thermal walls, solid thermal walls, and fluid-solid interfaces. ° In **Explore**, you can apply monitors to faces of solid thermal walls and fluid-solid interfaces. ° A negative heat flux on a face represents outflow on external faces, or transfer of heat from solid to fluid on fluid-solid interfaces. ° Heat flux is zero on insulated fluid thermal and solid thermal walls, and on excluded fluid-solid interfaces.
Heat Flow	The flow of heat into the fluid region. Heat Flow can be evaluated on walls, inlets and outlets, and fluid-solid interfaces. The value is relative to a reference temperature of 25C, therefore, it is possible to see negative values on inlets and positive values on outlets. For fluid-solid interfaces, the heat flow is computed on the solid side. A positive heat flow value means heat is flowing from fluid to solid. Heat flow monitors are not available for fluid-fluid interfaces.

Electromagnetics Variables

Peak gain	Gain is four pi times the ratio of an antenna's radiation intensity in a given direction to the total power accepted by the antenna. It is the maximum value of gain and is unitless.

Peak directivity	Directivity is defined as the ratio of an antenna's radiation intensity in a given direction to the radiation intensity averaged over all directions. It is the maximum value of directivity and is unitless.
Peak radiant intensity	Peak radiant intensity is defined as the maximum radiant intensity. It is measured in Watts/Steradian.
Front back ratio	Front back ratio is defined as the ratio of radiant intensity to antipode radiant intensity in the direction of peak radiant intensity. It is unitless. The direction of peak radiant intensity is the center of the main lobe.
Efficiency total	Efficiency total is defined as the ratio of radiated power to incident power. It is unitless.
Efficiency radiated	Efficiency radiated is defined as the ratio of radiated power to accepted power. It is unitless.

Electric Variables

Voltage	Voltage simulates the application of an electric potential. Represents contours of constant electric potential (voltage) in conductor bodies. This is a scalar quantity.
Current Density	Current simulates the application of an electric current. Can be computed for any solid conductor body. It is displayed as a vector.
Joule Heat	Joule heat is the heat generated by the flow of current through the body. This occurs in a conductor carrying an electric current. Joule heat is proportional to the square of the current, and is independent of the current direction.

22.4.2. Setting Up Monitors

After applying some physics conditions to your model, a list of physics based predefined or custom monitors is available for you to use in your simulation.

To create a monitor:

1. Select a location on the geometry where you would like to apply a monitor. Depending on the defined physics, current stage, and variable to monitor, the location may be a point, line, 2D flat surface, or plane within the geometry, or faces, bodies or edges of the geometry.
2. From the **Monitors** menu in the ribbon of the **Simulation** tab, choose a monitor from the list or select **Custom** to define your own.

> **Note:** If you select a predefined monitor without specifying the location first, the monitor will be applied to the entire body. To change the location of a monitor, double-click the monitor in the list, then select the **Replace location** tool guide and specify the new location.

3. If you are creating a custom monitor, select a variable from the list in the **Monitors** HUD. Optionally, include any additional details to monitor, such as Minimum or Maximum values, and in some cases additional components, such as X, Y, Z directions. If you've defined a local coordinate system for a structural simulation, you can also

select the coordinate system for your monitor. When finished, click the green check mark to accept the custom monitor you created.

Open charts by clicking the monitor you created in the monitor list.

4. Solve your simulation and view the calculated numerical results of each monitor by clicking the **Monitors** list in the top right corner of the graphics window. Learn more about what you can do with monitors in this list.

> **Note:** In a modal analysis, when modeling natural frequencies, the monitored frequency mode is applied to all bodies.

To compare differences in results between two locations, follow these steps.

22.4.3. Computing Differences in Monitored Locations

Computing differences in results is useful when, for example, you want to determine the pressure drop between two locations. To calculate the difference in results between two locations in your geometry:

1. From the **Monitors** menu in the ribbon of the **Simulation** tab, choose **Custom**.
2. With the **Select** tool active, select the primary location of the monitor. The selected geometry appears orange.
3. Use the **Select second location to compute difference** tool to select the secondary location of the monitor. The selected geometry appears blue.
4. Select a variable from the list in the **Monitors** HUD. Optionally, include any additional details to monitor, such as Minimum or Maximum values, and in some cases additional components, such as X, Y, Z directions. If you've defined a local coordinate system for a structural simulation, you can also select the coordinate system for your monitor. When finished, click the green check mark to accept the custom monitor you created.
5. Solve and view the differences in results in the monitors list by clicking **Monitors** in the top right corner of the graphics window. Learn more about what you can do with monitors in this list.

22.4.4. Modifying and Plotting Monitors

After creating monitors and solving the simulation, you can view the numerical outputs of your monitors by clicking the **Monitors** list in the top right corner of the graphics window. This list consists of default monitors that the Discovery application includes based on the physics provided in your current simulation. It also includes any monitors that you create. Hover over any of the monitors in the list to highlight their location in the model. Click any monitors to show plots of the calculated results. You can click and drag the monitor charts and place them anywhere in the graphics window.

You can edit and update most monitors in the list, but there are some preset monitors that cannot be modified, such as Factor of Safety for structures and pressure drop for fluids.

To edit a monitor, you can double-click the monitor in the list or right-click and select **Edit**. Right-clicking also provides you with additional options such as hiding plots, deleting, renaming, duplicating, and adding new monitors.

Anytime you make updates to your physics setup and solution, the numerical results of the monitors will be updated, and arrows appear next to each of the variables in the list of monitors. The directions of the arrows (up or down) indicate an increase or decrease in the results since the last solution. The colors of the arrows indicate an improvement (green) or deterioration (red) in results since the last solution. White arrows (or black arrows if using a light UI theme) represent results that did not impact the solution.

With each update to your physics setup and solution, the monitor charts are updated with the new values plotted. You can select **Set target values** for your monitor.

- Specify the maximum and minimum limits. Red highlighted sections represent the out-of-bounds values that are not desirable. Error notifications are issued if the monitor falls outside the set range.
- Specify the target.

For simulations where you have solutions in **Explore** and **Refine**, monitor results are displayed for both on the chart. An orange circle represents one data point in Explore, while a purple triangle represents a data point in Refine. But you can click **Show results for current stage only** to view the monitor results only fo the current stage.

If you have defined variations in your simulation, you can display the variations as separate data series in the monitor chart. Use **Choose which variations to show** to select the variations you want to view. You can choose to show results for select variations, all the variations, none, starred, or variations with test cases. Additionally, select **Open variations chart for this monitored value** to view the variations chart for the selected variable.

For time-dependent solutions (available in the **Explore** stage only), steady-state fluid flow simulations, and topology optimization simulations, the monitor chart may have a **Details** button depending on the location and variable settings, which will open another chart showing plots of the variables as a function of time or iteration.

Right-click the charts to access more options. You can clear the current or previous design changes plotted in the chart. Additionally, you can export each chart's data or data for all monitored variables to a comma-separated values (CSV) file that can be used by other software tools for further analysis. Data for detailed charts, such as time-dependent monitor data, can also be exported.

You can close and then re-open all monitors using the **Show/Hide chart** button in the **Results Arc**, at the bottom right corner of the graphics window.

22.4.5. Charts

After applying some physics conditions to your model, you can set up charts for use in your simulation.

Available charts, and how you access them, depend on your simulation type.

For structural simulations, you can select available variables and specify Minimum or Maximum values, and in some cases the XYZ directional components.

For random vibration analysis, you can set up PSD charts.

For electromagnetic simulations, you can select from a number of default charts, as well as selecting custom chart types.

This section contains the following topics:

22.4.5.1. Setting Up Structural Charts

After you apply structural physics conditions to your model, you can define charts for viewing results.

1. Select the **Charts** icon in the ribbon of the **Simulation** tab. The Charts HUD appears.
2. Select a variable from the tool guides list.
3. Under **Options**, include any additional details to chart, such as Minimum or Maximum values, and in some cases directional components, X, Y, Z.
4. Solve your simulation and view the calculated numerical results of each chart by clicking the **Charts** list in the top right corner of the graphics window.

22.4.5.2. Setting Up PSD Charts for Random Vibration Analysis

After you set up your random vibration analysis, you can define PSD charts for viewing the results of your analysis.

1. Select the **Charts** icon in the ribbon of the **Simulation** tab. The PSD Charts HUD appears.

2. Select a PSD variable from the tool guides list.
3. Under **Options**, define your directional components. You can also select a coordinate system, either the global coordinate system or a local one.
4. Solve your simulation and view the calculated numerical results of each chart by clicking the **Charts** list in the top right corner of the graphics window.

22.4.5.3. Setting Up Electromagnetic Charts

After you apply physics conditions to your model, charts are available for your use in viewing results.

1. Select the **Charts** icon in the ribbon of the **Simulation** tab and choose a pre-defined chart from the list, or select **Custom** to define your own.

   See Electromagnetic Charts on page 926 for a list of available charts.
2. If you are creating a custom chart, select the editor in each chart panel.
3. Solve your simulation and view the calculated numerical results of each chart by clicking the **Charts** list in the top right corner of the graphics window.

22.4.5.4. Modifying and Plotting Charts

You can edit and update your charts.

Charts can be viewed by clicking the **Charts** list in the top right corner of the graphics window. To edit a chart, double-click the chart in the list or right-click and select additional options such as hiding, duplicating, renaming, and deleting. You can also add and edit notes.

Whenever you make updates to your physics setup and solution, the numerical results of the charts are also updated.

With each update to your physics setup and solution, the charts are updated with the new values plotted. You can click and drag the charts and place them anywhere in the graphics window.

You can close and then re-open all charts using the **Show/Hide chart** button in the **Results Arc**, at the bottom right corner of the graphics window.

If you have defined variations in your simulation, you can display the variations as separate data series in the chart. Use **Choose which variations to show** to select the variations you want to view. You can choose to show results for select variations, all the variations, none, starred, or variations with test cases. Additionally, select **Open variations chart for this monitored value** to view the variations chart for the selected variable.

If you have multi-port simulations in an electromagnetics simulation, the S-Parameter charts (dB and Phase) map and display a single source-sink port. You can **Add New Series** to add other source-sink combinations. If the number of series exceed three rows, use the scroll wheel to view the combinations.

22.4.5.5. Available Charts

The Discovery applications includes the following default charts.

Structural Charts

The following default charts are available for structural simulations:

Response PSD Chart	The PSD charts (or PSD Response Charts) describe how the structure behaves locally (at each vertex) in terms of spectral values due to the base excitation. From the PSD charts, you can compute the sigma values, available in the Results ARC.

Electromagnetic Charts

The following default charts are available for electromagnetic simulations:

Smith Chart	The Smith chart simultaneously displays Smith, it is a useful graphical aid for elec the radio frequency disciplines.
Gain	Gain is the ratio of power emitted in that
S-Parameters (dB)	S- parameters are Scattering parameters plots show the magnitude on a logarithn
S-Parameters (Phase)	S- parameters are Scattering parameters Phase plots show the complex phase.

In addition, you can create **Custom** charts, including polar plots. A polar plot is a 2D circular chart divided by the spherical coordinates *R* (radius) and *theta* (angle from the X-axis).

22.4.6. Tabular Results

After you apply some physics conditions to your model, tabular results are available for you to use in your simulation. Use the **Tables** drop-down list from the **Simulation** ribbon to view a tabular summary of monitor results for each defined physics condition. You can choose from common general result types. In addition, for some conditions location tables are created by default.

With each update to your physics setup and solution, the tables are updated with the new values plotted. The results are specific to each stage. You can click and drag the tables and place them anywhere in the graphics window.

This section contains the following topics:

22.4.6.1. Available Tabular Results

The following tabular results tables are available by default:

Constraint Force Reaction	This table displays tabular results for the reaction force, defined as the reaction load over a constraint, where the resultant forces balance out the forces applied.
Constraint Moment Reaction	This table displays tabular results for the reaction moment, defined as the local moment reaction that a support exerts on a structure to maintain equilibrium when an external moment or load is applied.
Maximum Body Temperature	This table displays the maximum temperature for each solid body in a simulation.

See Results Variables for more information.

22.5. Saved Scenes

A scene is the capture of a model view and can be automatically created when a physics condition is added. A scene can include the model orientation and visibility with material coloring, results display definition including notations and legend, and results presentation options.

- A scene can be exported to 2D or 3D image format.
- Saved scene snapshots are scenes with specific results applied and can be exported to a ZIP archive.
- Scene animations can be saved to a *.mp4 file.

The **Results Comparison Dashboard** is of particular interest when working with saved scenes, allowing you to visually compare up to two results.

> **Note:** **Explore** only supports up-to-date results in one simulation at a time, so if you have multiple simulations, only scenes in one simulation will contain results. If you want results from multiple simulations, switch to a simulation that you want results from, solve, and then create the report for that simulation. You must do this for each simulation that you want results from.

This section contains the following topics:

22.5.1. Physics Conditions Available as Default Saved Scenes

Default scenes can be added or deleted automatically as some physics conditions are added or deleted. Use the **Saved Scene Results** setting in the **Results** settings panel to control this behavior or override the behavior for a given document with the **Scene** dropdown list. An individual default scene cannot be renamed or deleted. See Results Variables on page 917 for a full description of the physics conditions.

> **Note:** Electromagnetic physics conditions do not generate automatic default scenes.

Fluid Flow Variables

Condition	Result Type
Total Pressure	Exterior contours
Velocity	Curved vectors, colored by magnitude

Fluid Thermal Variables

Condition	Result Type
Temperature	Exterior contours
Total Pressure	Exterior contours
Velocity	Curved vectors, colored by magnitude or temperature

Solid Thermal Variables

Condition	Result Type
Displacement, magnitude	Exterior contours
Temperature	Exterior contours
von Mises Stress	Exterior contours

Structural Variables

Condition	Result Type
Displacement, magnitude	Exterior contours
von Mises Stress	Exterior contours

Thermal Variables

Condition	Result Type
Temperature	Exterior contours

22.5.2. Saving or Exporting Saved Scenes, Snapshots and Animations

There are several ways to save or export scenes, snapshots and animations:

- A scene can be exported to 2D or 3D image format.
- Saved scene snapshots are scenes with specific results applied and can be exported to a ZIP archive.
- Scene animations can be saved to a *.mp4 file.
- Saved scene snapshots can be compared in the **Results Comparison Dashboard**.

This section contains the following topics:

22.5.2.1. Exporting Scenes to 2D or 3D Image Files

Use the export function to capture model views as 2D or 3D images. Images captured with **Scene** are of better quality than typical screen captures and can be inserted into documents or presentations for use outside of the Discovery application.

To export a scene to a 2D or 3D image file:

1. From the **Scene** dropdown in the **Simulation** tab, select your image type:
   - Select **Export to 2D Image** to export the current scene as a *.png file.
   - Select **Export to 3D Image** to export the current scene as a *.glb file.

   The *.glb file format is a single-file binary GLTF format for 3D scenes and models. The *.glb format is supported only if **Outer contours** on **All faces**, **Streamlines**, or curved **Vectors** are selected.

   Streamlines and curved vectors are only available for fluids physics types.
2. A save file dialog prompts for a file name and location to save.

22.5.2.2. Creating and Exporting Saved Scenes Snapshots to ZIP

Use the export function to save snapshots to a ZIP archive:

- Export 2D and 3D snapshots.
- Export the snapshots for the current simulation, stage, and run.
- Generate missing snapshots for the current variation before export.

To create and export a snapshot, you will need at least one saved scene in your gallery.

1. Select **Snapshot Saved Scenes** from the **Scene** dropdown in the **Simulation** tab to manually generate saved scene snapshots. Alternatively, configure the Discovery application for automatic snapshot generation.

   The application animates the "snapshooting" so you can see the progress.
2. To export, select **Export Snapshots to ZIP** from the dropdown.
3. A save file dialog prompts for a file name and location to save.

Limitation: Scene snapshots must be generated within the current session and stage if they are to be exported to ZIP.

22.5.2.3. Saving Scene Animations to MP4

Use the save function to save a scene animation to a *.mp4 file:

- Animation of deformation and modal deformation is supported in **Explore**.
- Animation of deformation, modal deformation, streamlines, and curved vectors is supported in **Refine**.

To save an animation, you need to have valid results and be currently animating the scene.

1. Click **Solve** to solve the simulation in **Explore** or **Refine**.
2. Click a supported animation display type in the Results arc; for example, **Deformation** for a structural simulation in **Explore** or **Refine**, or **Streamlines** for a fluid simulation in **Refine**.
3. Click **Show animation**.
4. To save the animation, select **Save Animation to File** from the dropdown.
5. A save file dialog prompts for a file name and location to save.

22.5.3. Automatic Capture of Scenes for Viewing

Whether working in **Refine** or in **Explore** with defined variations, the Discovery application captures your saved scene settings with the results, and creates a new 2D snapshot. For cases in which you cannot capture a scene without forcing a re-solve, click **Snapshot Saved Scenes** from the **Scene** dropdown menu in the ribbon to take a snapshot of all defined scenes with the current results and store it with the project.

> **Note:** When new snapshots are taken, the matrix and viewer must be closed and re-opened for the new snapshots to become available.

22.5.3.1. Capturing Scenes Manually

There may be cases in which you want to capture scenes manually.

1. Navigate to **Settings** > **Results** > **Saved Scene Results** to select **Save snapshots to file** and set **Exclude scenes older than *n* runs**. This value determines how many snapshots from the **Saved scene snapshot matrix** are included in a project when you save it. The value is set to 3 by default, but you can specify a value from 0 to 10.
2. Click **Save Current Scene** from the **Scene** dropdown in the Simulation tab or in the top right of the graphics window.
3. Click **Solve** to solve the simulation in **Refine** or **Update** in the defined **Variations** table.
4. Select **Snapshot Saved Scenes** from the **Scene** dropdown in the Simulation tab to generate saved scene snapshots. The application animates the "snapshooting" so you can see the progress.
5. To export, select **Export Snapshots to ZIP** from the dropdown list.
6. A save file dialog prompts for a file name and location to save.

22.5.4. Results Comparison Dashboard

When capturing and working with scenes, the Discovery application automatically updates your snapshots with results. Work locally with defined variations (**Refine** or **Explore**) or remotely in the cloud (**Explore**) to capture and download your saved scenes with results and visually compare snapshots in the **Results comparison dashboard**.

The dashboard is comprised of:

- a matrix displaying snapshot scenes for each defined variation which updates dynamically as you solve. Thumbnails displayed in the table are low resolution, but high-resolution images are available for download.
- a detailed view of one or two scenes for comparison, and
- defined variations charts.

The layout of these components within the dashboard is flexible. Use the layout control icons in the top right to change and prioritize what is most important to you.

This section contains the following topics:

22.5.4.1. Comparing Snapshots Results

To compare snapshot results, select **Results comparison dashboard** from the **Variations** tab toolbar.

1. Save one or more scenes by selecting **Save Current Scene** (top right) or from the **Scene** ribbon dropdown list.
2. Expand the scene gallery and right-click a scene, select **Show Results Comparison Dashboard** to launch the dashboard in a separate window.
3. From the **Variations** tab toolbar, select **Update defined variations**, choose **Local** or **Remote** and select the appropriate update option from the dropdown list.
4. The **Results comparison dashboard** updates as each variation solves and displays thumbnail images. Hover over thumbnail images in the viewer for informative tooltips.
5. Select up to two snapshot thumbnails to view the associated details.

Lock the views to synchronize both images to ease comparison studies.

22.5.4.2. Results Comparison Dashboard Options

Several filters are available to manage the snapshot matrix. The filter dropdown options update dynamically based on the work done in the defined variations table.

- **Simulation run**
- **Design change**
- **Test case**

Right-click an image for additional options:

- **Show Snapshot** opens a higher resolution image of the snapshot
- **Save Image As...** .png file
- **Download High-quality Image...** is available for remotely solved results from the cloud

The detailed view of the scene image supports:

- zoom and pan of the 2D scene image,
- display of default monitors, and
- linking selected scenes to keep them in synch.

Important: Limitations when using the Results comparison dashboard include:

- Remote solving in the cloud is limited to **Explore**.
- Local solving in **Explore** will not generate snapshots on a standard solve but will generate when solving one or more rows in the **Variations** table.
- Vary the snapshot orientation and magnification to aid in visual comparison.

Version 2026 R1 - © Ansys, Inc. All rights reserved.
Contains proprietary and confidential information of Synopsys, Inc., ANSYS, Inc., subsidiaries and affiliates.

23.1.1. Copyright and Trademark Information

© 2026 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

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.

Protected by US Patents 7,639,267, 7,733,340, 7,830,377, 7,969,435, 8,207,990, 8,244,508, 8,253,726, 8,330,775, 10,650,172, 10,706,623, 10,769,850, D916,099, D916,100, 11,269,478, 11,475,184, 2023/0004695, 11,640,485, and 12,056,420.

Copyright © 2003-2025 ANSYS, Inc. All Rights Reserved. SpaceClaim is a registered trademark of ANSYS, Inc.

Portions of this software Copyright © 2010 Acresso Software Inc. FlexLM and FLEXNET are trademarks of Acresso Software Inc.

Portions of this software Copyright © 2008 Adobe Systems Incorporated. All Rights Reserved. Adobe and Acrobat are either registered trademarks or trademarks of Adobe Systems Incorporated in the United States and/or other countries

Ansys Workbench and GAMBIT and all other ANSYS, Inc. product names are trademarks or registered trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries.

Contains BCLS (Bound-Constrained Least Squares) Copyright (C) 2006 Michael P. Friedlander, Department of Computer Science, University of British Columbia, Canada, provided under a LGPL 3 license which is included in the SpaceClaim installation directory (lgpl-3.0.txt). Derivative BCLS source code available upon request.

Contains SharpZipLib Copyright © 2009 C#Code

Anti-Grain Geometry Version 2.4 Copyright © 2002-2005 Maxim Shemanarev (McSeem).

Some SpaceClaim products may contain Autodesk® RealDWG by Autodesk, Inc., Copyright © 1998-2010 Autodesk, Inc. All rights reserved. Autodesk, AutoCAD, and Autodesk Inventor are registered trademarks and RealDWG is a trademark of Autodesk, Inc.

CATIA is a registered trademark of Dassault Systèmes.

Portions of this software Copyright © 2010 Google. SketchUp is a trademark of Google.

Portions of this software Copyright © 1999-2006 Intel Corporation. Licensed under the Apache License, Version 2.0. You may obtain a copy of the License at http://www.apache.org/licenses/LICENSE-2.0.

Contains DotNetBar licensed from devcomponents.com.

KeyShot is a trademark of Luxion ApS.

MatWeb is a trademark of Automation Creations, Inc.

2008 Microsoft ® Office System User Interface is licensed from Microsoft Corporation. Direct3D, DirectX, Microsoft PowerPoint, Excel, Windows, Windows Vista and the Windows Vista Start button are trademarks or registered trademarks of Microsoft Corporation in the United States and/or other countries.

Portions of this software Copyright © 2005 Novell, Inc. (http://www.novell.com)

Creo Parametric and PTC are registered trademarks of Parametric Technology Corporation.

Persistence of Vision Raytracer and POV-Ray are trademarks of Persistence of Vision Raytracer Pty. Ltd.

Portions of this software Copyright © 1993-2009 Robert McNeel & Associates. All Rights Reserved. openNURBS is a trademark of Robert McNeel & Associates. Rhinoceros is a registered trademark of Robert McNeel & Associates.

Portions of this software Copyright © 2005-2007, Sergey Bochkanov (ALGLIB project). *

Portions of this software are owned by Siemens PLM © 1986-2011. All Rights Reserved. Parasolid and Unigraphics are registered trademarks and JT is a trademark of Siemens Product Lifecycle Management Software, Inc.

This work contains the following software owned by Siemens Industry Software Limited: D-CubedTM 2D DCM © 2021. Siemens. All Rights Reserved.

SOLIDWORKS is a registered trademark of SOLIDWORKS Corporation.

Portions of this software are owned by Spatial Corp. © 1986-2011. All Rights Reserved. ACIS and SAT are registered trademarks of Spatial Corp.

Contains Teigha for .dwg files licensed from the Open Design Alliance. Teigha is a trademark of the Open Design Alliance.

Development tools and related technology provided under license from 3Dconnexion. © 1992 – 2008 3Dconnexion. All rights reserved.

TraceParts is owned by TraceParts S.A. TraceParts is a registered trademark of TraceParts S.A.

Contains a modified version of source available from Unicode, Inc., copyright © 1991-2008 Unicode, Inc. All rights reserved. Distributed under the Terms of Use in http://www.unicode.org/copyright.html.

Portions of this software Copyright © 1992-2008 The University of Tennessee. All rights reserved. [1]

Portions of this software Copyright © XHEO INC. All Rights Reserved. DeployLX is a trademark of XHEO INC.

This software incorporates information provided by American Institute of Steel Construction (AISC) for shape data available at http://www.aisc.org/shapesdatabase.

This software incorporates information provided by ArcelorMittal® for shape data available at http://www.sections.arcelormittal.com/products-services/products-ranges.html.

All other trademarks, trade names or company names referenced in SpaceClaim software, documentation and promotional materials are used for identification only and are the property of their respective owners.

*Additional notice for LAPACK and ALGLIB Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:-Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.-Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer listed in this license in the documentation and/or other materials provided with the distribution.-Neither the name of the copyright holders nor the names of its contributors may be used to endorse promote products derived from this software without specific prior written permission.

BCLS is licensed under the GNU Lesser General Public License (GPL) Version 3, Copyright (C) 2006 Michael P. Friedlander, Department of Computer Science, University of British Columbia, Canada. A copy of the LGPL license is included in the installation directory (lgpl-3.0.txt).

Please contact open.source@ansys.com for a copy of the source code for BCLS.

Eigen is licensed under the Mozilla Public License (MPL) Version 2.0, the text of which can be found at: https://www.mozilla.org/media/MPL/2.0/index.815ca599c9df.txt. Please contact open.source@ansys.com for a copy of the Eigen source code.

HDF5 (Hierarchical Data Format 5) Software Library and Utilities

Copyright (c) 2006, The HDF Group.

NCSA HDF5 (Hierarchical Data Format 5) Software Library and Utilities

Copyright (c) 1998-2006, The Board of Trustees of the University of Illinois.

All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted for any purpose (including commercial purposes) provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions, and the following disclaimer.

2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions, and the following disclaimer in the documentation and/or materials provided with the distribution.

3. In addition, redistributions of modified forms of the source or binary code must carry prominent notices stating that the original code was changed and the date of the change.

4. All publications or advertising materials mentioning features or use of this software are asked, but not required, to acknowledge that it was developed by The HDF Group and by the National Center for Supercomputing Applications at the University of Illinois at Urbana-Champaign and credit the contributors.

5. Neither the name of The HDF Group, the name of the University, nor the name of any Contributor may be used to endorse or promote products derived

from this software without specific prior written permission from The HDF Group, the University, or the Contributor, respectively.

DISCLAIMER:

THIS SOFTWARE IS PROVIDED BY THE HDF GROUP AND THE CONTRIBUTORS "AS IS" WITH NO WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED. In no

event shall The HDF Group or the Contributors be liable for any damages suffered by the users arising out of the use of this software, even if advised of the possibility of such damage. Anti-Grain Geometry - Version 2.4 Copyright (C) 2002-2004 Maxim Shemanarev (McSeem)

Permission to copy, use, modify, sell and distribute this software is granted provided this copyright notice appears in all copies. This software is provided "as is" without express or implied warranty, and with no claim as to its suitability for any purpose.

Some ANSYS-SpaceClaim products may contain Autodesk® RealDWG by Autodesk, Inc., Copyright © 1998-2010 Autodesk, Inc. All rights reserved. Autodesk, AutoCAD, and Autodesk Inventor are registered trademarks and RealDWG is a trademark of Autodesk, Inc.

CATIA is a registered trademark of Dassault Systèmes.

Portions of this software Copyright © 2013 Trimble. SketchUp is a trademark of Trimble Navigation Limited.

This software is based in part on the work of the Independent JPEG Group.

Portions of this software Copyright © 1999-2006 Intel Corporation. Licensed under the Apache License, Version 2.0. You may obtain a copy of the License at http://www.apache.org/licenses/LICENSE-2.0

Contains DotNetBar licensed from devcomponents.com.

Portions of this software Copyright © 1988-1997 Sam Leffler and Copyright (c) 1991-1997 Silicon Graphics, Inc.

KeyShot is a trademark of Luxion ApS.

MatWeb is a trademark of Automation Creations, Inc.

2010 Microsoft ® Office System User Interface is licensed from Microsoft Corporation. Direct3D, DirectX, Microsoft PowerPoint, Excel, Windows/Vista/Windows 7/Windows 8/Windows 10 and their respective Start Button designs are trademarks or registered trademarks of Microsoft Corporation in the United States and/or other countries.

Portions of this software Copyright © 2005 Novell, Inc. (Licensed at http://stuff.mit.edu/afs/athena/software/mono_v3.0/arch/i386_linux26/mono/mcs/class/Managed.Windows.Forms/System.Windows.Forms.RTF/)

Pro/ENGINEER and PTC are registered trademarks of Parametric Technology Corporation.

POV-Ray is available without charge from http://www.pov-ray.org. No charge is being made for a grant of the license to POV-Ray.

POV-Ray License Agreement

DISTRIBUTOR'S LICENCE AGREEMENT

Persistence of Vision Raytracer(tm) (POV-Ray(tm))

13 August 2004

Licensed Versions: Versions 3.5 and 3.6

Please read through the terms and conditions of this license carefully. This is a binding legal agreement between you, the "Distributor" and Persistence of Vision Raytracer Pty. Ltd. ACN 105 891 870 ("POV"), a company incorporated in the state of Victoria, Australia, for the product known as the "Persistence of Vision Raytracer(tm)", also referred to herein as "POV-Ray(tm)". The terms of this agreement are set out at http://www.povray.org/distribution-license.html ("Official Terms"). The Official Terms take precedence over this document to the extent of any inconsistency.

1. INTRODUCTION

1.1. In this agreement, except to the extent the context requires otherwise, the following capitalized terms have the following meanings:

(a) Distribution means:

(i) a single item of a distribution medium, including a CD Rom or DVD Rom, containing software programs and/or data;

(ii) a set of such items;

(iii) a data file in a generally accepted data format from which such an item can be created using generally available standard tools;

(iv) a number of such data files from which a set of such items can be created; or

(v) a data file in a generally accepted data storage format which is an archive of software programs and/or data;

(b) Derived Code means all software which is derived from or is an adaptation of any part of the Software other than a scene file;

(c) Intellectual Rights means:

(i) all copyright, patent, trade mark, trade secret, design, and circuit layout rights;

(ii) all rights to the registration of such rights; and

(iii) all rights of a similar nature which exist anywhere in the world;

(d) Licensed Version means the version set out at the top of this agreement against the heading "Licensed Version" and all minor releases of this version (ie releases of the form x.y.z);

(e) POV Associate means any person associated directly or indirectly with POV whether as a director, officer, employee, subcontractor, agent, representative, consultant, licensee or otherwise;

(f) Modification Terms means the most recent version from time to time of the document of that name made available from the Site (g) Revocation List means the list of that name linked to from the Official Terms;

(h) Site means www.povray.org;

(i) Software means the Licensed Version of the Persistence of Vision Raytracer(tm) (also known as POV-Ray(tm)) (including all POV-Ray program source files, executable (binary) files, scene files, documentation files, help files, bitmaps and other POV-Ray files associated with the Licensed Version) in a form made available by

POV on the Site;

(j) User Licence means the most recent version from time to time of the document of that name made available from the Site.

2. OPEN SOURCE DISTRIBUTIONS

2.1. In return for the Distributor agreeing to be bound by the terms of this agreement, POV grants the Distributor permission to make a copy of the Software by including the Software in a generally recognised Distribution of a recognised operating system where the kernel of that operating system is made available under licensing terms:

(a) which are approved by the Open Source Initiative (www.opensource.org) as complying with the "Open Source Definition" put forward by the Open Source Initiative; or

(b) which comply with the "free software definition" of the Free Software Foundation (www.fsf.org). 2.2. As at June 2004, and without limiting the generality of the term, each of the following is a "generally recognised Distribution" for the purposes of clause 2.1: Debian, Red Hat (Enterprise and Fedora), SuSE, Mandrake, Xandros, Gentoo and Knoppix Linux distributions, and officially authorized distributions of the FreeBSD, OpenBSD, and NetBSD projects.

2.3. Clause 2.1 also applies to the Software being included in the above distributions 'package' and 'ports' systems, where such exist;

2.4. Where the Distributor reproduces the Software in accordance with clause 2.1:

(a) the Distributor may rename, reorganise or repackage (without omission) the files comprising the Software where such renaming, reorganisation or repackaging is necessary to conform to the naming or organisation scheme of the target operating environment of the Distribution or of an established package management system of the target operating environment of the Distribution; and (b) the Distributor must not otherwise rename, reorganise or repackage the Software.

3. DISTRIBUTION LICENCE

3.1. Subject to the terms and conditions of this agreement, and in return for Distributor agreeing to be bound by the terms of this agreement, POV grants the Distributor permission to make a copy of the Software in any of the following circumstances:(a) in the course of providing a mirror of the POV-Ray Site (or part of it), which is made available generally over the internet to each person without requiring that person to identify themselves and without any other restriction other than restrictions designed to manage traffic flows;

(b) by placing it on a local area network accessible only by persons authorized by the Distributor whilst on the Distributor's premises;

(c) where that copy is provided to a staff member or student enrolled at a recognised educational institution;

(d) by including the Software as part of a Distribution where:

(i) neither the primary nor a substantial purpose of the distribution of the Distribution is the distribution of the Software. That is, the distribution of the Software

is merely incidental to the distribution of the Distribution; and

(ii) if the Software was not included in the Distribution, the remaining software and data included within the Distribution would continue to function effectively and

according to its advertised or intended purpose;

(e) by including the Software as part of a Distribution where:

(i) there is no data, program or other files apart from the Software on the Distribution;

(ii) the Distribution is distributed by a person to another person known to that person; or

(iii) the Distributor has obtained explicit written authority from POV to perform the distribution, citing this clause number, prior to the reproduction being

made.

3.2. In each case where the Distributor makes a copy of the Software in accordance with clause 3.1, the Distributor must, unless no payment or other consideration of any type is received by Distributor in relation to the Distribution:

(a) ensure that each person who receives a copy of the Software from the Distributor is aware prior to acquiring that copy:

(i) of the full name and contact details of the Distributor, including the Distributor's web site, street address, mail address, and working email address;

(ii) that the Software is available without charge from the Site;

(iii) that no charge is being made for the granting of a licence over the Software.

(b) include a copy of the User Licence and this Distribution License with the copy of the Software. These licences must be stored in the same subdirectory on the distribution medium as the Software and named in such a way as to prominently identify their purpose;

3.3. The Distributor must not rename, reorganise or repackage any of the files comprising the Software without the prior written authority of POV.

3.4. Except as explicitly set out in this agreement, nothing in this agreement permits Distributor to make any modification to any part of the Software.

4. RESTRICTIONS ON DISTRIBUTION

4.1. Nothing in this agreement gives the Distributor: (a) any ability to grant any licence in respect of the use of the Software or any part of it to any person;

(b) any rights or permissions in respect of, including rights or permissions to distribute or permit the use of, any Derived Code;

(c) any right to bundle a copy of the Software (or part thereof), whether or not as part of a Distribution, with any other items, including books and magazines. POV may, in response to a request, by notice in writing and in its absolute discretion, permit such bundling on a case by case basis. This clause 4.1(c) does not apply to Distributions permitted under clause 2;

(d) any right, permission or authorisation to infringe any Intellectual Right held by any third party.

4.2. Distributor may charge a fee for the making or the provision of a copy of the Software.

4.3. Where the making, or the provision, of a copy of the Software is authorised under the terms of clause 3 but not under those of clause 2 of this agreement, the total of all fees charged in relation to such making or provision and including all fees (including shipping and handling fees) which are charged in respect

of any software, hardware or other material provided in conjunction with or in any manner which is reasonably connected with the making, or the provision, of a copy of the Software must not exceed the reasonable costs incurred by the Distributor in making the reproduction, or in the provision, of that copy for which the fee

is charged.

4.4. Notwithstanding anything else in this agreement, nothing in this agreement permits the reproduction of any part of the Software by, or on behalf of:

(a) Any person currently listed on the Revocation List from time to time;

(b) Any related body corporate (as that term is defined in section 50 of the Corporations Law 2001 (Cth)) of any person referred to in clause 4.4(a);

(c) Any person in the course of preparing any publication in any format (including books, magazines, CD Roms or on the internet) for any of the persons identified in paragraph (a);

(d) Any person who is, or has been, in breach of this Agreement and that breach has not been waived in writing signed by POV; or

(e) Any person to whom POV has sent a notice in writing or by email stating that that person may not distribute the Software.

4.5. From the day two years after a version of the Software more recent than the Licensed Version is made available by POV on the Site clause 3 only permits reproduction of the Software where the Distributor ensures that each recipient of such a reproduction is aware, prior to obtaining that reproduction, that that reproduction of the Software is an old version of the Software and that a more recent version of the Software is available from the Site.

5. COPYRIGHT AND NO LITIGATION

5.1. Copyright subsists in the Software and is protected by Australian and international copyright laws.

5.2. Nothing in this agreement gives Distributor any rights in respect of any Intellectual Rights in respect of the Software or which are held by or on behalf of POV. Distributor acknowledges that it does not acquire any rights in respect of such Intellectual Rights.

5.3. Distributor acknowledges that if it performs out any act in respect of the Software without the permission of POV it will be liable to POV for all damages POV may suffer (and which Distributor acknowledges it may suffer) as well as statutory damages to the maximum extent permitted by law and that it may also be liable to

criminal prosecution.

5.4. Distributor must not commence any action against any person alleging that the Software or the use or distribution of the Software infringes any rights, including Intellectual Rights of the Distributor or of any other person. If Distributor provides one or more copies of the Software to any other person in accordance with the agreement, Distributor waives all rights it has, or may have in the future, to bring any action, directly or indirectly, against any person to the extent that such an action relates to an infringement of any rights, including Intellectual Rights of any person in any way arising from, or in relation to, the use, or distribution, (including through the authorisation of such use or distribution) of:(a) the Software;

(b) any earlier or later version of the Software; or

(c) any other software to the extent it incorporates elements of the software referred to in paragraphs (a) or (b) of this clause

5.4.

6. DISCLAIMER OF WARRANTY

6.1. To the extent permitted by law, all implied terms and conditions are excluded from this agreement. Where a term or condition is implied into this agreement and that term cannot be legally excluded, that term has effect as a term or condition of this agreement. However, to the extent permitted by law, the liability

of POV for a breach of such an implied term or condition is limited to the fullest extent permitted by law.

6.2. To the extent permitted by law, this Software is provided on an "AS IS" basis, without warranty of any kind, express or implied, including without limitation, any implied warranties of merchantability, fitness for a particular purpose and non-infringement of intellectual property of any third party. The Software has inherent limitations including design faults and programming bugs.

6.3. The entire risk as to the quality and performance of the Software is borne by Distributor, and it is Distributor's responsibility to ensure that the Software fulfils Distributor's requirements prior to using it in any manner (other than testing it for the purposes of this paragraph in a non-critical and non-production environment), and prior to distributing it in any fashion.

6.4. This clause 6 is an essential and material term of, and cannot be severed from, this agreement. If Distributor does not or cannot agree to be bound by this clause, or if it is unenforceable, then Distributor must not, at any time, make any reproductions of the Software under this agreement and this agreement gives the

Distributor no rights to make any reproductions of any part of the Software.

7. NO LIABILITY

7.1. When you distribute or use the Software you acknowledge and accept that you do so at your sole risk. Distributor agrees that under no circumstances will it have any claim against POV or any POV Associate for any loss, damages, harm, injury, expense, work stoppage, loss of business information, business interruption,

computer failure or malfunction which may be suffered by you or by any third party from any cause whatsoever, howsoever arising, in connection with your use or distribution of the Software even where POV was aware, or ought to have been aware, of the potential of such loss.

7.2. Neither POV nor any POV Associate has any liability to Distributor for any indirect, general, special, incidental, punitive and/or consequential damages arising as a result of a breach of this agreement by POV or which arises in any way related to the Software or the exercise of a licence granted to Distributor under this

agreement.

7.3. POV's total aggregate liability to the Distributor for all loss or damage arising in any way related to this agreement is limited to the lesser of: (a) AU$100, and (b) the amount received by POV from Distributor as payment for the grant of a licence under this agreement.

7.4. Distributor must bring any action against POV in any way related to this agreement or the Software within 3 months of the cause of action first arising. Distributor waives any right it has to bring any action against POV and releases POV from all liability in respect of a cause of action if initiating process in relation to that action is not served on POV within 3 months of the cause of action arising. Where a particular set of facts give rise to more than one cause of action this clause 7.4 applies as if all such causes of action arise at the time the first such cause of action arises.

7.5. This clause 7 is an essential and material term of, and cannot be severed from, this agreement. If Distributor does not or cannot agree to be bound by this clause, or if it is unenforceable, then Distributor must not, at any time, make any reproductions of the Software under this agreement and this agreement gives the Distributor no rights to make any reproductions of any part of the Software.

8. INDEMNITY

8.1. Distributor indemnifies POV and each POV Associate and holds each of them harmless against all claims which arise from any loss, damages, harm, injury, expense, work stoppage, loss of business information, business interruption, computer failure or malfunction, which may be suffered by Distributor or any other

party whatsoever as a consequence of:

(a) any act or omission of POV and/or any POV Associate, whether negligent or not;

(b) Distributor's use and/or distribution of the Software; or

(c) any other cause whatsoever, howsoever arising, in connection with the Software. This clause 8 is binding on Distributor's estate, heirs, executors, legal successors, administrators, parents and/or guardians.

8.2. Distributor indemnifies POV, each POV Associate and each of the authors of any part of the Software against all loss and damage and for every other consequence flowing from any breach by Distributor of any Intellectual Right held by POV.

8.3. This clause 8 constitutes an essential and material term of, and cannot be severed from, this agreement. If Distributor does not or cannot agree to be bound by this clause, or if it is unenforceable, then Distributor must not, at any time, make any reproductions of the Software under this agreement and this agreement gives the Distributor no rights to make any reproductions of any part of the Software.

9. HIGH RISK ACTIVITIES

9.1. This Software and the output produced by this Software is not fault-tolerant and is not designed, manufactured or intended for use as on-line control equipment in hazardous environments requiring fail-safe performance, in which the failure of the Software could lead or directly or indirectly to death, personal injury, or severe physical or environmental damage ("High Risk Activities"). POV specifically disclaims all express or implied warranty of fitness for High Risk Activities and, notwithstanding any other term of this agreement, explicitly prohibits the use or distribution of the Software for such purposes.

10. ENDORSEMENT PROHIBITION

10.1. Distributor must not, without explicit written permission from POV, claim or imply in any way that:

(a) POV or any POV Associate officially endorses or supports the Distributor or any product (such as CD, book, or magazine) associated with the Distributor or any reproduction of the Software made in accordance with this agreement; or(b) POV derives any benefit from any reproduction made in accordance with this agreement.

11. TRADEMARKS

11.1. "POV-Ray(tm)", "Persistence of Vision Raytracer(tm)" and "POV-Team(tm)" are trademarks of Persistence of Vision Raytracer Pty. Ltd. Any other trademarks referred to in this agreement are the property of their respective holders. Distributor must not use, apply for, or register anywhere in the world, any word, name

(including domain names), trade mark or device which is substantially identical or deceptively or confusingly similar to any of Persistence of Vision Raytracer Pty. Ltd's trade marks.

12. MISCELLANEOUS

12.1. The Official Terms, including those documents incorporated by reference into the Official Terms, and the Modification Terms constitute the entire agreement between the parties relating to the distribution of the Software and, except where stated to the contrary in writing signed by POV, supersedes all previous

negotiations and correspondence in relation to it.

12.2. POV may modify this agreement at any time by making a revised licence available from the Site at http://www.povray.org/distribution-license.html.

This agreement is modified by replacing the terms in this agreement with those of the revised licence from the time that the revised licence is so made available. It is your responsibility to ensure that you have read and agreed to the current version of this agreement prior to distributing the Software.

12.3. Except where explicitly stated otherwise herein, if any provision of this Agreement is found to be invalid or unenforceable, the invalidity or unenforceability of such provision shall not affect the other provisions of this agreement, and all provisions not affected by such invalidity or unenforceability shall remain in

full force and effect. In such cases Distributor agrees to attempt to substitute for each invalid or unenforceable provision a valid or enforceable provision which achieves to the greatest extent possible, the objectives and intention of the invalid or unenforceable provision.

12.4. A waiver of a right under this agreement is not effective unless given in writing signed by the party granting that waiver. Unless otherwise stipulated in the waiver, a waiver is only effective in respect of the circumstances in which it is given and is not a waiver in respect of any other rights or a waiver in respect of

future rights or actions.

12.5. The validity and interpretation of this agreement is governed by the laws in force in the State of Victoria, Australia. Distributor submits to the exclusive jurisdiction of the courts of that State and courts located within that State exercising federal jurisdiction.

12.6. References in this agreement to "written" and "writing" mean on paper or by fax and expressly exclude email and other forms of electronic communication.

13. CONTACT INFORMATION

13.1. This clause 13 does not form part of the agreement. License inquiries can be made via email; please use the following address (but see 13.2 below prior to emailing) : team-coord-[three-letter month]-[four-digit year]@povray org. for example, team-coord-jun-2004@povray.org should be used if at the time you send the email it is the month of June 2004. The changing email addresses are necessary to combat spam. Old email addresses may be deleted at POV's discretion.

13.2. Note that the address referred to in 13.1 may change for reasons other than those referred to in that clause; please check the current version of this document at http://www.povray.org/distribution-license.html. for the current address. Your inability or failure to contact us is no excuse for violating the licence.

13.3. Do NOT send any email attachments of any sort other than by prior arrangement. Do not send email in HTML format. EMAIL MESSAGES INCLUDING ATTACHMENTS WILL BE DELETED UNREAD.

13.4. The following postal address is only for official license business. Please note that it is preferred that initial queries about licensing be made via email; postal mail should only be used when email is not possible, or when written documents are being exchanged by prior arrangement. While it is unlikely this address will change in the short term it would be advisable to check http://www.povray.org/distribution-license.html for the current one prior to sending postal mail.

Persistence of Vision Raytracer Pty. Ltd.

PO Box 407

Williamstown,

Victoria 3016

Australia

POV-Ray Licence Agreement

GENERAL LICENSE AGREEMENT

FOR PERSONAL USE

Persistence of Vision Ray Tracer (POV-Ray)

Version 3.6 License and Terms & Conditions of Use

version of 1 February 2005

(also known as POVLEGAL.DOC)

Please read through the terms and conditions of this license carefully. This license is a binding legal agreement between you, the 'User' (an individual or single entity) and Persistence of Vision Raytracer Pty. Ltd. ACN 105 891 870 (herein also referred to as the "Company"), a company incorporated in the state of Victoria, Australia, for the product known as the "Persistence of Vision Ray Tracer", also referred to herein as 'POV-Ray'.

YOUR ATTENTION IS PARTICULARLY DRAWN TO THE DISCLAIMER OF WARRANTY AND NO LIABILITY AND INDEMNITY PROVISIONS. TO USE THE PERSISTENCE OF VISION RAY TRACER ("POV-RAY") YOU MUST AGREE TO BE BOUND BY THE TERMS AND CONDITIONS SET OUT IN THIS DOCUMENT. IF YOU DO NOT AGREE TO ALL THE TERMS AND CONDITIONS OF USE OF POV-RAY SET OUT IN THIS LICENSE AGREEMENT, OR IF SUCH TERMS AND CONDITIONS ARE NOT BINDING ON YOU IN YOUR JURISDICTION, THEN YOU MAY NOT USE POV-RAY IN ANY MANNER. THIS GENERAL

LICENSE AGREEMENT MUST ACCOMPANY ALL POV-RAY FILES WHETHER IN THEIR OFFICIAL OR CUSTOM VERSION FORM. IT MAY NOT BE REMOVED OR MODIFIED. THIS GENERAL LICENSE AGREEMENT GOVERNS THE USE OF

POV-RAY WORLDWIDE. THIS DOCUMENT SUPERSEDES AND REPLACES ALL PREVIOUS GENERAL LICENSES.

INTRODUCTION

This document pertains to the use of the Persistence of Vision Ray Tracer (also known as POV-Ray). It applies to all POV-Ray program source files, executable (binary) files, scene files, documentation files, help files, bitmaps and other POV-Ray files contained in official Company archives, whether in full or any part thereof, and are herein referred to as the "Software". The Company reserves the right to revise these rules in future versions and to make additional rules to address new circumstances at any time. Such rules, when made, will be posted in a revised license file, the latest version of which is available from the Company website at

http://www.povray.org/povlegal.html.

USAGE PROVISIONS

Subject to the terms and conditions of this agreement, permission is granted to the User to use the Software and its associated files to create and render images. The creator of a scene file retains all rights to any scene files they create, and any images generated by the Software from them. Subject to the other terms of this license, the User is permitted to use the Software in a profit-making enterprise, provided such profit arises primarily from use of the Software and not from distribution of the Software or a work including the Software in whole or part.

Please refer to http://www.povray.org/povlegal.html for licenses covering distribution of the Software and works including the Software. The User is also granted the right to use the scene files, fonts, bitmaps, and include files distributed in the INCLUDE and SCENES\INCDEMO sub-directories of the Software in their own scenes. Such permission does not extend to any other files in the SCENES directory or its sub-directories. The SCENES files are for the User's enjoyment and education but may not be the basis of any derivative works unless the file in question explicitly grants permission to do such.

This licence does not grant any right of re-distribution or use in any manner other than the above. The Company has separate license documents that apply to other uses (such as re-distribution via the internet or on CD) ; please visit http://www.povray.org/povlegal.html for links to these. In particular you are advised that the sale, lease, or rental of the Software in any form without written authority from the Company is explicitly prohibited. Notwithstanding anything in the balance of this licence agreement, nothing in this licence agreement permits the installation or use of the Software in conjunction with any product (including software) produced or distributed by any party who is, or has been, in violation of this licence agreement or of the distribution licence (http://www.povray.org/distribution-license.html)

(or any earlier or later versions of those documents) unless:

a. the Company has explicitly released that party in writing from the consequences of their non compliance; or

b. both of the following are true:

i. the installation or use of the Software is without the User being aware of the abovementioned violation; and

ii. the installation or use of the Software is not a result (whether direct or indirect) of any request or action of the abovementioned party (or any of its products), any agent of that party (or any of their products), or any person(s) involved in supplying any such product to the User.

COPYRIGHT

Copyright © 1991-2003, Persistence of Vision Team.

Copyright © 2003-2004, Persistence of Vision Raytracer Pty. Ltd.

Windows version Copyright © 1996-2003, Christopher Cason.

Copyright subsists in this Software which is protected by Australian and international copyright laws. The Software is NOT PUBLIC DOMAIN. Nothing in this agreement shall give you any rights in respect of the intellectual property of the Company and you acknowledge that you do not acquire any rights in respect of such intellectual property rights. You acknowledge that the Software is the valuable intellectual property of the Company and that if you use,

modify or distribute the Software for unauthorized purposes or in an unauthorized manner (or cause or allow the forgoing to occur), you will be liable to the Company for any damages it may suffer (and which you acknowledge it may suffer) as well as statutory damages to the maximum extent permitted by law and also that you may be liable to

criminal prosecution. You indemnify the Company and the authors of the Software for every single consequence flowing from the aforementioned events.

DISCLAIMER OF WARRANTY

express or implied, including without limitation, any implied warranties of merchantability, fitness for a particular purpose and non-infringement of intellectual property of any third party. This Software has inherent limitations including design faults and programming bugs. The entire risk as to the quality and performance of the Software is borne by you, and it is your responsibility to ensure that it does what you require it to do prior to using it for any purpose (other than testing it), and prior to distributing it in any fashion. Should the Software prove defective, you agree that you alone assume the entire cost resulting in any way from such defect.

This disclaimer of warranty constitutes an essential and material term of this agreement. If you do not or cannot accept this, or if it is unenforceable in your jurisdiction, then you may not use the Software in any manner.

NO LIABILITY

When you use the Software you acknowledge and accept that you do so at your sole risk. You agree that under no circumstances shall you have any claim against the Company or anyone associated directly or indirectly with the Company whether as employee, subcontractor, agent, representative, consultant, licensee or otherwise ("Company Associates") for any loss, damages, harm, injury, expense, work stoppage, loss of business information, business interruption, computer failure or malfunction which may be suffered by you or by any third party from any cause whatsoever, howsoever arising, in connection with your use or distribution of the Software even where the Company were aware, or ought to have been aware, of the potential of such loss. Damages referred to above shall include direct, indirect, general, special, incidental, punitive and/or consequential. This disclaimer of liability constitutes an essential and material term of this agreement. If you do not or cannot accept this, or if it is unenforceable in your jurisdiction, then you may not use the Software.

INDEMNITY

You indemnify the Company and Company Associates and hold them harmless against any claims which may arise from any loss, damages, harm, injury, expense, work stoppage, loss of business information, business interruption, computer failure or malfunction, which may be suffered by you or any other party whatsoever as a consequence of any act or omission of the Company and/or Company Associates, whether negligent or not, arising out of your use and/or distribution of the Software, or from any other cause whatsoever, howsoever arising, in connection with the Software. These provisions are binding on your estate, heirs, executors, legal successors, administrators, parents and/or guardians.

This indemnification constitutes an essential and material term of this agreement. If you do not or cannot accept this, or if it is unenforceable in your jurisdiction, then you may not use the Software.

HIGH RISK ACTIVITIES

This Software and the output produced by this Software is not fault-tolerant and is not designed, manufactured or intended for use as on-line control equipment in hazardous environments requiring fail-safe performance, in which the failure of the Software could lead or directly or indirectly to death, personal injury, or severe physical or environmental damage ("High Risk Activities"). The Company specifically disclaims any express or implied warranty of fitness for High Risk Activities and explicitly prohibits the use of the Software for such purposes.

CRYPTOGRAPHIC SIGNING OF DOCUMENTS

Changes to this Agreement and documents issued under its authority may be cryptographically signed by the POV-Ray Team Co-ordinator's private PGP key.

In the absence of evidence to the contrary, such documents shall be considered, under the terms of this Agreement, to be authentic provided the signature is

valid. The master copy of this Agreement at http://www.povray.org/povlegal.html will also be signed by the current version of the team-coordinator's key.

The public key for the POV-Ray Team-coordinator can be retrieved from the location https://secure.povray.org/keys/. The current fingerprint for it is

B4DD 932A C080 C3A3 6EA2 9952 DB04 4A74 9901 4518.

MISCELLANEOUS

This Agreement constitutes the complete agreement concerning this license. Any changes to this agreement must be in writing and may take the form of

notifications by the Company to you, or through posting notifications on the Company website. THE USE OF THIS SOFTWARE BY ANY PERSON OR ENTITY IS

EXPRESSLY MADE CONDITIONAL ON THEIR ACCEPTANCE OF THE TERMS SET FORTH HEREIN. Except where explicitly stated otherwise herein, if any provision of this

Agreement is found to be invalid or unenforceable, the invalidity or unenforceability of such provision shall not affect the other provisions of this agreement, and all provisions not affected by such invalidity or unenforceability shall remain in full force and effect. In such cases you agree to attempt to substitute for each invalid or unenforceable provision a valid or enforceable provision which achieves to the greatest extent possible, the objectives and intention of the invalid or unenforceable

provision. The validity and interpretation of this agreement will be governed by the laws of Australia in the state of Victoria (except for conflict of law provisions).

CONTACT INFORMATION

License inquiries can be made via email; please use the following address (but see below prior to emailing) : team-coord-[three-letter month]-[four-digit year]@povray.org for example, team-coord-jun-2004@povray.org should be used if at the time you send the email it is the month of June 2004. The changing email addresses are necessary to combat spam and email viruses. Old email addresses may be deleted at our discretion.

Note that the above address may change for reasons other than that given above; please check the version of this document at http://www.povray.org/povlegal.html for the current address. Note that your inability or failure to contact us for any reason is not an excuse for violating this licence.

Do NOT send any attachments of any sort other than by prior arrangement.

EMAIL MESSAGES INCLUDING ATTACHMENTS WILL BE DELETED UNREAD.

The following postal address is only for official license business. Please note that it is preferred that initial queries about licensing be made via email ; postal mail should only be used when email is not possible, or when written documents are being exchanged by prior arrangement.

Persistence of Vision Raytracer Pty. Ltd.

PO Box 407

Williamstown,

Victoria 3016

Australia

Portions of this software are owned by Siemens PLM © 1986-2013. All Rights Reserved. Parasolid, Unigraphics, and SolidEdge are registered trademarks and JT is a trademark of Siemens Product Lifecycle Management Software, Inc.

SolidWorks is a registered trademark of SolidWorks Corporation.

Portions of this software are owned by Spatial Corp. © 1986-2013. All Rights Reserved. ACIS, SAT and SAB are registered trademarks of Spatial Corp.

Contains Teigha for .dwg files licensed from the Open Design Alliance. Teigha is a trademark of the Open Design Alliance.

Development tools and related technology provided under license from 3Dconnexion. © 1992 – 2008 3Dconnexion. All rights reserved.

•TraceParts is owned by TraceParts S.A. TraceParts is a registered trademark of TraceParts S.A.

Copyright © 1991-2017 Unicode, Inc. All rights reserved.

Distributed under the Terms of Use in http://www.unicode.org/copyright.html. Permission is hereby granted, free of charge, to any person obtaining a copy of the Unicode data files and any associated documentation (the "Data Files") or Unicode software and any associated documentation (the "Software") to deal in the Data Files or Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, and/or sell copies of the Data Files or Software, and to permit persons to whom the Data Files or Software are furnished to do so, provided that either (a) this copyright and permission notice appear with all copies of the Data Files or Software, or

(b) this copyright and permission notice appear in associated Documentation.

THE DATA FILES AND SOFTWARE ARE PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE

WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT OF THIRD PARTY RIGHTS. IN NO EVENT SHALL THE COPYRIGHT HOLDER OR HOLDERS INCLUDED IN THIS NOTICE BE LIABLE FOR ANY CLAIM, OR ANY SPECIAL INDIRECT OR CONSEQUENTIAL DAMAGES, OR ANY DAMAGES WHATSOEVER RESULTING FROM LOSS OF USE, DATA OR PROFITS, WHETHER IN AN ACTION OF CONTRACT, NEGLIGENCE OR OTHER TORTIOUS ACTION, ARISING OUT OF OR IN CONNECTION WITH THE USE OR PERFORMANCE OF THE DATA FILES OR SOFTWARE.

Except as contained in this notice, the name of a copyright holder shall not be used in advertising or otherwise to promote the sale, use or other dealings in these Data Files or Software without prior written authorization of the copyright holder.

Portions of this software Copyright © 1992-2008 The University of Tennessee. All rights reserved.

This product includes software developed by XHEO INC (http://xheo.com).

Portions of this software are owned by Tech Soft 3D, Inc. Copyright © 1996-2013. All rights reserved. HOOPS is a registered trademark of Tech Soft 3D, Inc.

Portions of this software are owned by MachineWorks Limited. Copyright ©2013. All rights reserved. Polygonica is a registered trademark of MachineWorks Limited.

Apache License

Version 2.0, January 2004 http://www.apache.org/licenses/

TERMS AND CONDITIONS FOR USE, REPRODUCTION, AND DISTRIBUTION

1. Definitions.

"License" shall mean the terms and conditions for use, reproduction, and distribution as defined by Sections 1 through 9 of this document.

"Licensor" shall mean the copyright owner or entity authorized by the copyright owner that is granting the License.

"Legal Entity" shall mean the union of the acting entity and all other entities that control, are controlled by, or are under common control with that entity. For the purposes of this definition,

"Control" means (i) the power, direct or indirect, to cause the direction or management of such entity, whether by contract or otherwise, or (ii) ownership of fifty percent (50%) or more of the outstanding shares, or (iii) beneficial ownership of such entity.

"You" (or "Your") shall mean an individual or Legal Entity exercising permissions granted by this License.

"Source" form shall mean the preferred form for making modifications, including but not limited to software source code, documentation source, and configuration files.

"Object" form shall mean any form resulting from mechanical transformation or translation of a Source form, including but not limited to compiled object code, generated documentation, and conversions to other media types.

"Work" shall mean the work of authorship, whether in Source or Object form, made available under the License, as indicated by a copyright notice that is included in or attached to the work (an example is provided in the Appendix below).

"Derivative Works" shall mean any work, whether in Source or Object form, that is based on (or derived from) the Work and for which the editorial revisions, annotations, elaborations, or other modifications represent, as a whole, an original work of authorship. For the purposes of this License, Derivative Works shall not include works that remain separable from, or merely link (or bind by name) to the interfaces of, the Work and Derivative Works thereof.

"Contribution" shall mean any work of authorship, including the original version of the Work and any modifications or additions to that Work or Derivative Works thereof, that is intentionally submitted to Licensor for inclusion in the Work by the copyright owner or by an individual or Legal Entity authorized to submit on behalf of the copyright owner. For the purposes of this definition,

"Submitted" means any form of electronic, verbal, or written communication sent to the Licensor or its representatives, including but not limited to communication on electronic mailing lists, source code control systems, and issue tracking systems that are managed by, or on behalf of, the Licensor for the purpose of discussing and improving the Work, but excluding communication that is conspicuously marked or otherwise designated in writing by the copyright owner as "Not a Contribution."

"Contributor" shall mean Licensor and any individual or Legal Entity on behalf of whom a Contribution has been received by Licensor and subsequently incorporated within the Work.

2. Grant of Copyright License. Subject to the terms and conditions of this License, each Contributor hereby grants to You a perpetual, worldwide, non-exclusive, no-charge, royalty-free, irrevocable copyright license to reproduce, prepare Derivative Works of, publicly display, publicly perform, sublicense, and distribute the Work and such Derivative Works in Source or Object form.

3. Grant of Patent License. Subject to the terms and conditions of this License, each Contributor hereby grants to You a perpetual, worldwide, non-exclusive, no-charge, royalty-free, irrevocable (except as stated in this section) patent license to make, have made, use, offer to sell, sell, import, and otherwise transfer the Work, where such license applies only to those patent claims licensable by such Contributor that are necessarily infringed by their Contribution(s) alone or by combination of their Contribution(s) with the Work to which such Contribution(s) was submitted. If You institute patent litigation against any entity (including a cross-claim or counterclaim in a lawsuit) alleging that the Work or a Contribution incorporated within the Work constitutes direct or contributory patent infringement, then any patent licenses granted to You under this License for that Work shall terminate as of the date such litigation is filed.

4. Redistribution. You may reproduce and distribute copies of the Work or Derivative Works thereof in any medium, with or without modifications, and in Source or Object form, provided that You meet the following conditions:

(a) You must give any other recipients of the Work or Derivative Works a copy of this License; and

(b) You must cause any modified files to carry prominent notices stating that You changed the files; and

(c) You must retain, in the Source form of any Derivative Works that You distribute, all copyright, patent, trademark, and attribution notices from the Source form of the Work, excluding those notices that do not pertain to any part of the Derivative Works; and

(d) If the Work includes a "NOTICE" text file as part of its distribution, then any Derivative Works that You distribute must include a readable copy of the attribution notices contained within such NOTICE file, excluding those notices that do not pertain to any part of the Derivative Works, in at least one of the following places: within a NOTICE text file distributed as part of the Derivative Works; within the Source form or documentation, if provided along with the Derivative Works; or, within a display generated by the Derivative Works, if and wherever such third-party notices normally appear. The contents of the NOTICE file are for informational purposes only and do not modify the License.

You may add Your own attribution notices within Derivative Works that You distribute, alongside or as an addendum to the NOTICE text from the Work, provided that such additional attribution notices cannot be construed as modifying the License.

You may add Your own copyright statement to Your modifications and may provide additional or different license terms and conditions for use, reproduction, or distribution of Your modifications, or for any such Derivative Works as a whole, provided Your use, reproduction, and distribution of the Work otherwise complies with the conditions stated in this License.

5. Submission of Contributions. Unless You explicitly state otherwise, any Contribution

intentionally submitted for inclusion in the Work by You to the Licensor shall be under the terms and conditions of this License, without any additional terms or conditions. Notwithstanding the above, nothing herein shall supersede or modify the terms of any separate license agreement you may have executed with Licensor regarding such Contributions.

6. Trademarks. This License does not grant permission to use the trade names, trademarks, service marks, or product names of the Licensor, except as required for reasonable and customary use in describing the origin of the Work and reproducing the content of the NOTICE file.

7. Disclaimer of Warranty. Unless required by applicable law or agreed to in writing, Licensor provides the Work (and each Contributor provides its Contributions) on an "AS IS" BASIS, WITHOUT WARRANTIES OR CONDITIONS OF ANY KIND, either express or implied, including, without limitation, any warranties or conditions of TITLE, NON-INFRINGEMENT, MERCHANTABILITY, or FITNESS FOR A PARTICULAR PURPOSE. You are solely

responsible for determining the appropriateness of using or redistributing the Work and assume any risks associated with Your exercise of permissions under this License.

8. Limitation of Liability. In no event and under no legal theory, whether in tort (including negligence), contract, or otherwise, unless required by applicable law (such as deliberate and grossly negligent acts) or agreed to in writing, shall any Contributor be liable to You for damages, including any direct, indirect, special, incidental, or consequential damages of any character arising as a result of this License or out of the use or inability to use the Work (including but not limited to damages for loss of goodwill, work stoppage, computer failure or malfunction, or any and all other commercial damages or losses), even if such Contributor has been advised of the possibility of such damages.

9. Accepting Warranty or Additional Liability. While redistributing the Work or Derivative Works thereof, You may choose to offer, and charge a fee for, acceptance of support, warranty, indemnity, or other liability obligations and/or rights consistent with this License. However, in accepting such obligations, You may act only on Your own behalf and on Your sole responsibility, not on behalf of any other Contributor, and only if You agree to indemnify, defend, and hold each Contributor harmless for any liability incurred by, or claims asserted against, such Contributor by reason of your accepting any such warranty or additional liability.

END OF TERMS AND CONDITIONS