PowerPack

User Guide

V11

POWERPACK USER GUIDE

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First Edition

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PowerPack extends the feature set of ViaCAD to add tools specific for repairing meshes and parts for 3D Printing, as well as add tools for Entity Management, and Advanced Modeling tools.

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- Close Simple Mesh Holes
- Remove Collapsed, Overlaps, Intersections Facets
- Flip and Rebuild Facet Normals
- Remove Collapsed, Overlaps, Intersections
- Flip and Rebuild Facet Normals
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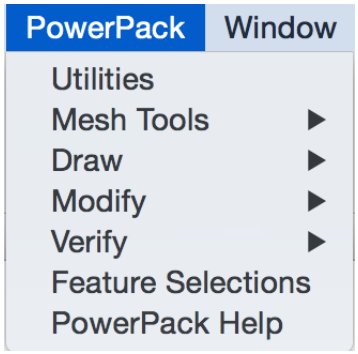
-

Installing

PowerPack is activated by either the serial number. Once activated, PowerPack is accessed from the menu bar after the Verify and before the Window commands.

Menus

Selecting PowerPack from the menu bar displays the popup below.



Utilities
The Utilities Tool palette provides commands for a variety of entity management operations such as Assorted Colors, Preview Layers, AutoLayer, Isolate Layer, Change Scale, Spell Check, Rename, and Show By Name.

Mesh Tools
The Mesh Tools menu provides commands for verifying and repairing common issues with meshes such as welding vertices, locating free or manifold edges, repairing holes, and add/delete/combine/split facets.

Draw
The Draw menu provides additional construction methods for creating new geometry such as lines, circles, splines, and surfaces.

Modify
The Modify Tools menu provides commands for editing or repairing geometry such as curves, surfaces, or solids.

Verify
The Verify menu provides additional tools for validating and isolating common geometry issues for curves, surfaces, and solids.

Feature Selections
The Feature Selections tool palette provides commands for selecting objects by more specific attributes. For meshes, this includes selecting by vertex, edge or face and for solids features such as blend, protrusion, or face networks.

PowerPack Help
Provides access to Internet based help content for PowerPack.

Utilities

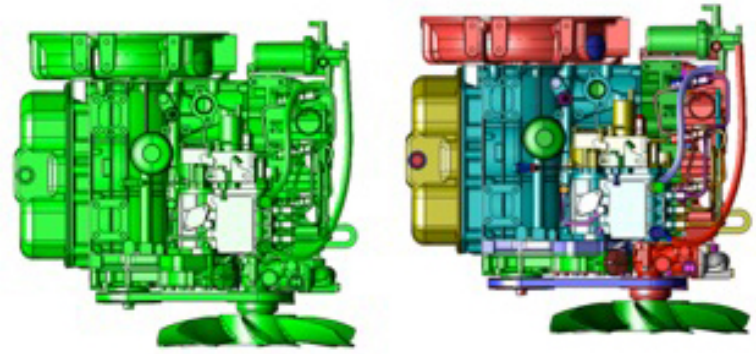
The Utilities menu provides tools for a variety of entity management operations such as Assorted Colors, Preview Layers, AutoLayer, Isolate Layer, Change Scale, Spell Check, Rename, and Show By Name.

Assorted Colors

The Assorted Colors tool assigns up to 20 colors per group of entities selected. The tool is useful for visually identifying individual parts such as a file import.

Example 1: Assorted Colors

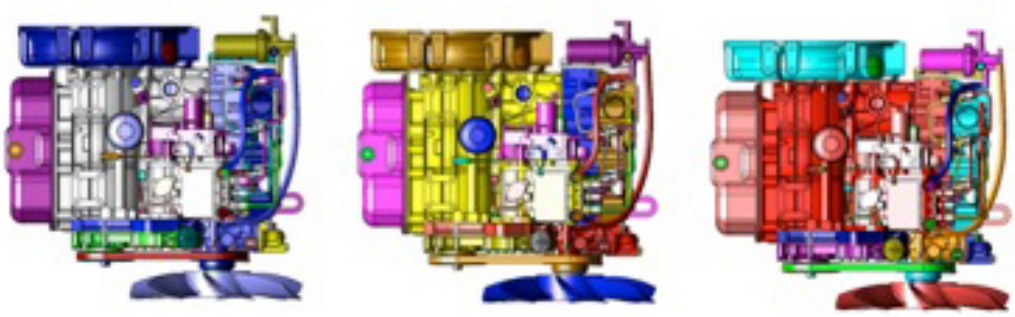
1. Select Assorted Colors tool from the PowerPack menu.
2. Select entities to apply Assorted colors.
3. Entities colors updated.

- 

Example 2: Assorted Colors

1. Select Assorted Colors tool from the PowerPack menu.
2. Select entities to apply Assorted colors.
3. Select entities to apply Assorted colors.
4. Select entities to apply Assorted colors.

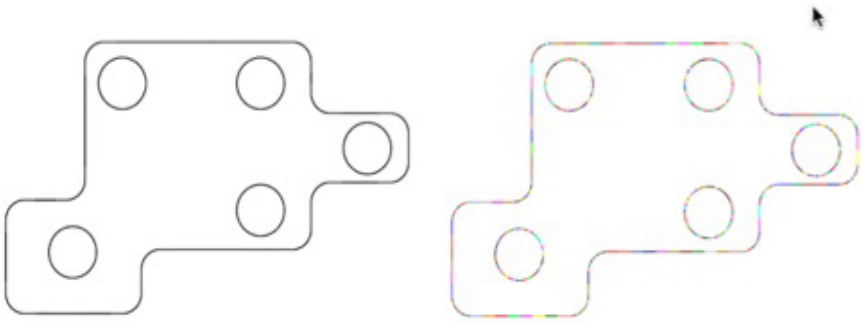
Repeatedly using on a set of entities will offset the start color for each selection, providing alternative color arrangements.



Example 3: Assorted Colors

1. Select Assorted Colors tool from the PowerPack menu.
2. Select curves to apply Assorted colors.

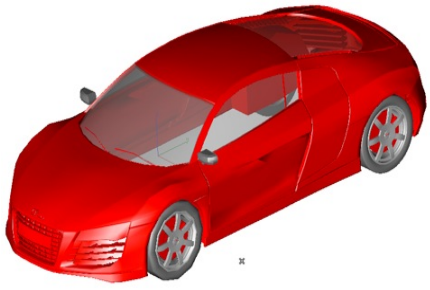
The tool is also useful for visually identifying individual curves in a profile.

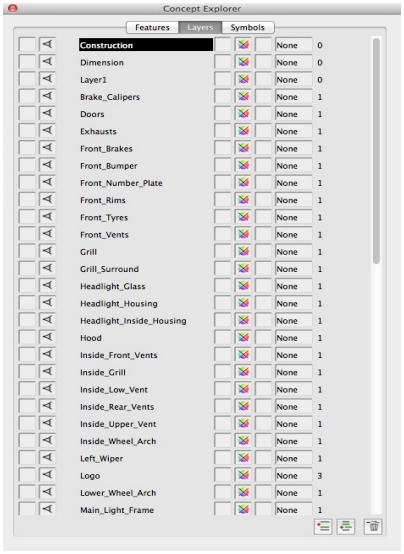


AutoLayer

AutoLayer is a tool that will automatically assign an entity to a layer based on same shape or by entity name. Use AutoLayer to help organize files imported from other applications that do not support layering.

By Name Creates a new layer for each of the selected entities and uses the entity name for the layer. If the name is shared, each entity is put into the same layer.





By Same Shape Creates a new layer for entities that have the same shape. Entities with the same shape are placed in the common layer. Shape comparisons data

include volume, area, length, width, height.

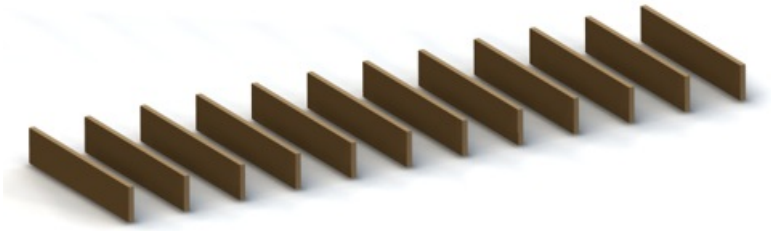
Tip: Use the Separate all Parts if the imported file contains one mesh.

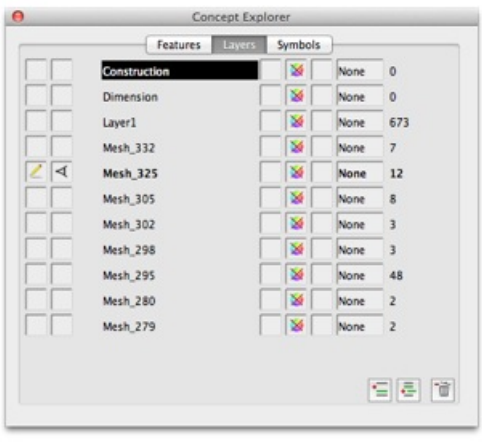
Example 1: By Name

1. Import an obj file that contains named elements.
2. Select Auto Layer from PowerPack.
3. Select all entities in the drawing.
4. Each entity is assigned layer where the layer name is the entity name.

Example 2: By Same Shape

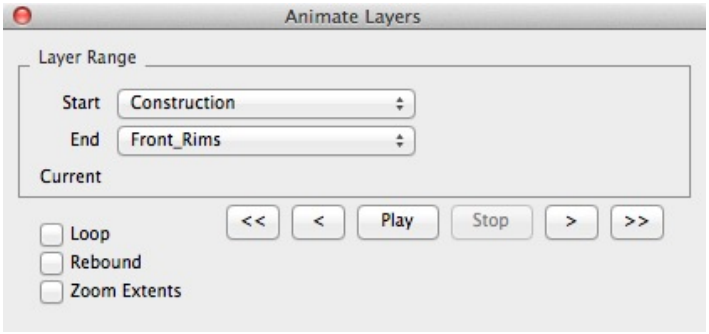
1. Import a STL file that contains mesh elements with same shapes.
2. Select Auto Layer from PowerPack.
3. Select all entities in the drawing.
4. New layers are created and populated with same shapes.





Preview Layers

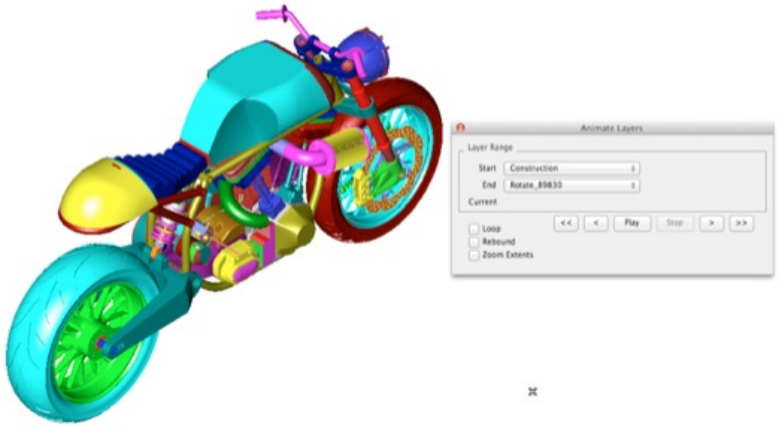
The Preview Layers utility provides a user interface to rapidly step through layers. The play button will animate through the layer start and end region. The step forward and backward buttons manually provide control through the layers.



Start The starting layer to animate
End The ending layer to animate
Loop Continue to animate once the end layer is reached with the Start layer.
Rebound Continue to animate once the end layer is reached with the End layer.
Zoom
Extents For each layer, zoom extents before the repaint.
Play Start the animation display.

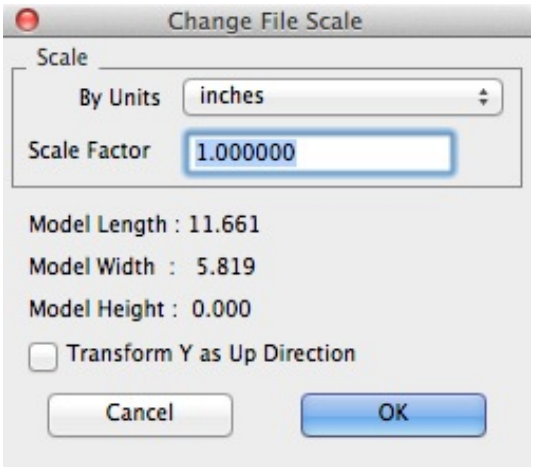
Example: Preview Layers

1. Select Preview Layers tool from the PowerPack menu.



Change File Units

There are many file formats that do not contain unit settings that include STL, OBJ, PLY, 3DS, and older DXF files. The Change File Units command allows you to quickly change a file scale based on Units or Scale Factor.

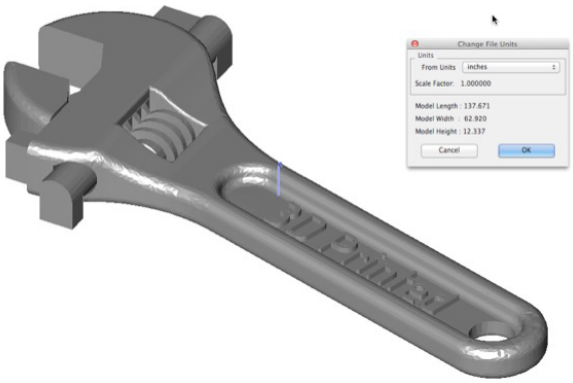


By Units Determines the scale factor by converting between units.
Scale Factor Specify the scale factor by typing in a value in the edit control.
Model Length
Model Width
Model Height Shows the model size if the Scale Factor is applied to the current drawing.
Transform Y as Up Direction Rotates the model about the x-axis 90 degrees.

Example 1: Change File Scale

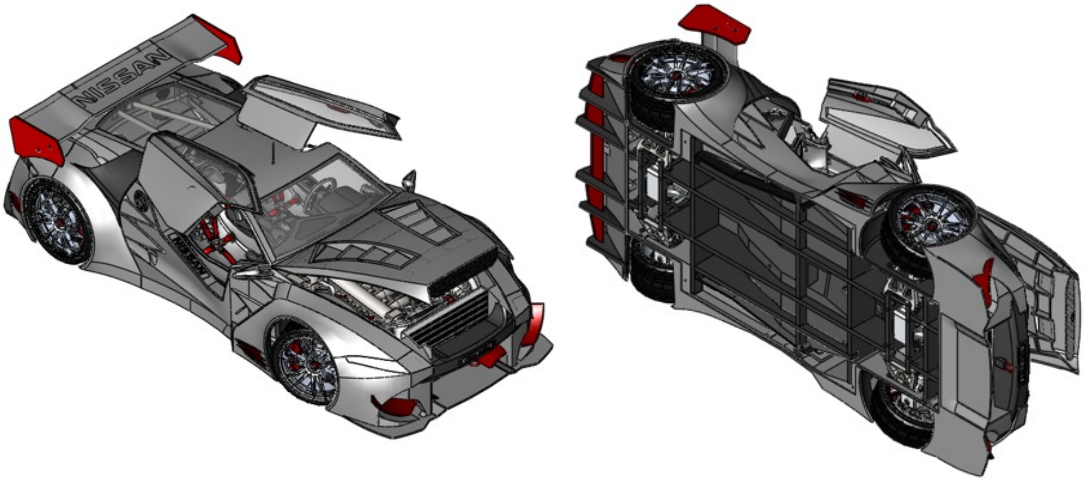
In the example below, a STL file is imported into the application where inches are the default units but the file was created in millimeters. The Change File Units dialog box show the model extents to be 137x63x12 inches, which is not consistent with an actual wrench. By setting the From Units to millimeters, the wrench is scaled to the correct size.

1. Select the Change File Scale option from the PowerPack set of tools.
2. Select Millimeters from the Units pull down.
3. Select OK.
4. Each entity in the file is scaled by the Scale Factor.



Example 2: Change File Scale
In Another feature of the Change File Scale command is the ability to swap y and z. For example, in the file below the model is transformed using the Transform Y as Up Direction option.

1. Select the Change File Scale option from the PowerPack set of tools.
2. Select the Transform Y as Up Direction option.
3. Select OK.
4. Each entity in the file is rotated such that y is now z.

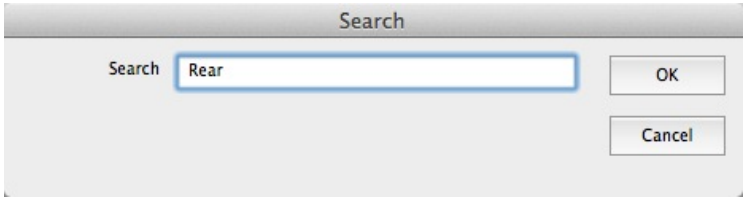


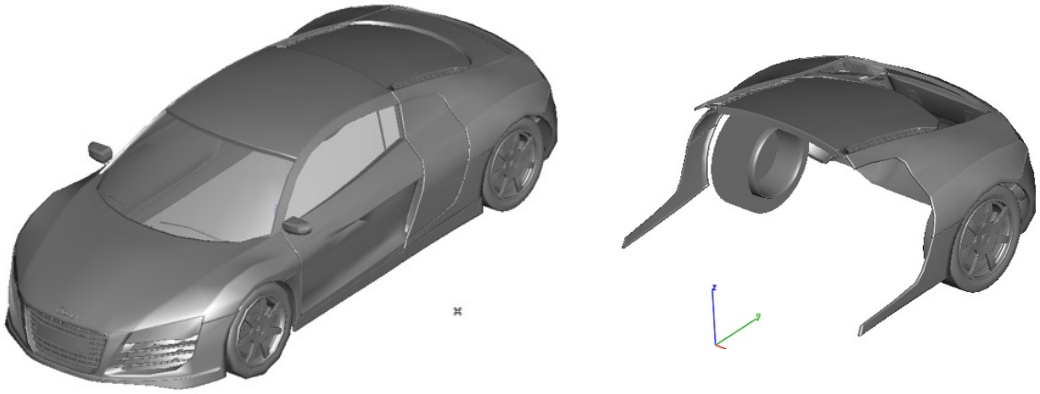
Show Only By Name

The Show Only by Name tool displays just the entity with the provided name. This tool changes the Show/Hide flags of the entity.

Example: Show Only By Name

1. Select Show Only By Name tool from the PowerPack menu.
2. In the Search dialog box, type in the string to search.
3. All entities with the string specified are displayed only In the example below all entries that include the string "Rear" are displayed. In the example illustrated below, fourteen objects are displayed.

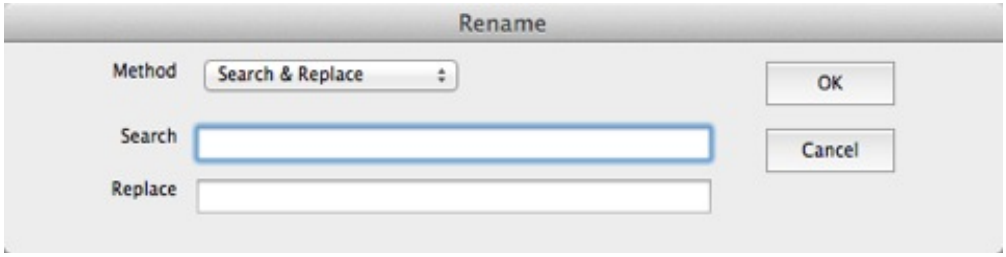




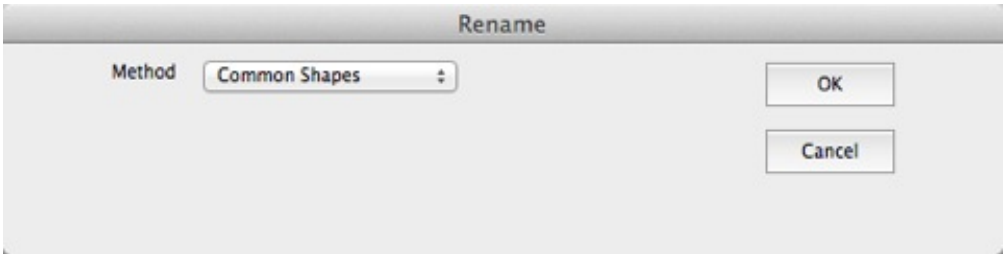
Rename

The Rename tool provides a way to rename large number of entities. This tool is especially useful for renaming entities imported from external applications. For example, Inventor attaches the full assembly representation to the part name. Renaming supports two methods:

Search and Replace Entity names containing the Search string are replaced with the Replace string.

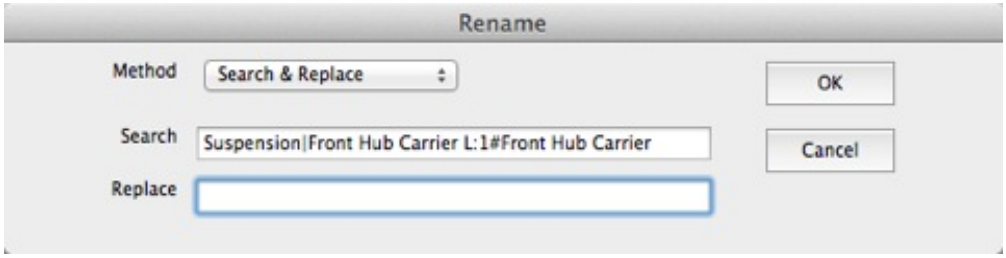


Common Shapes All entities sharing the same geometric shape are renamed to the same name. This is especially useful for creating Bill of Materials.

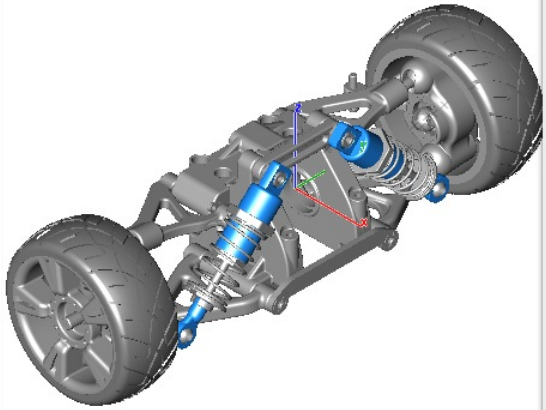


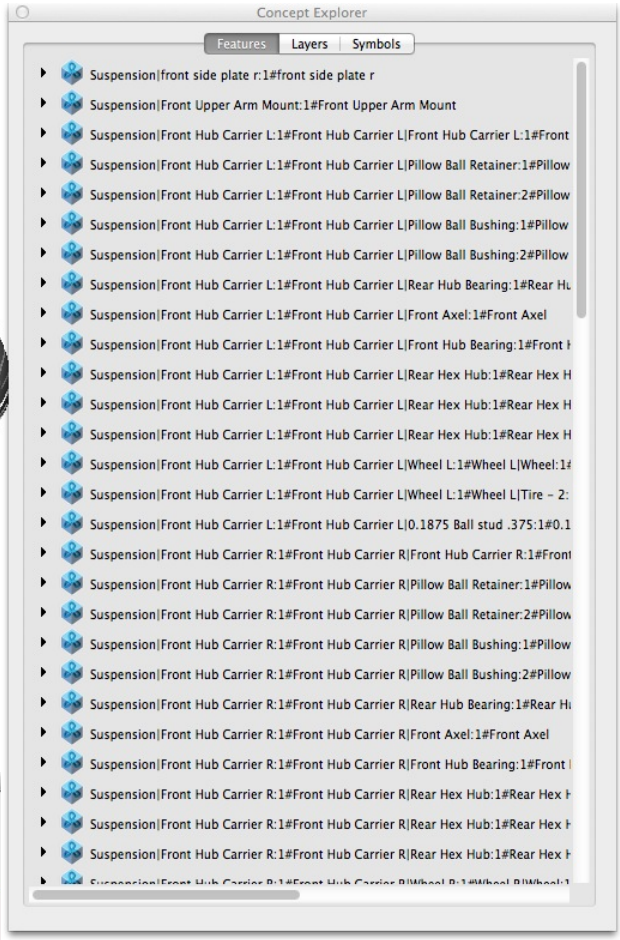
Example: Rename

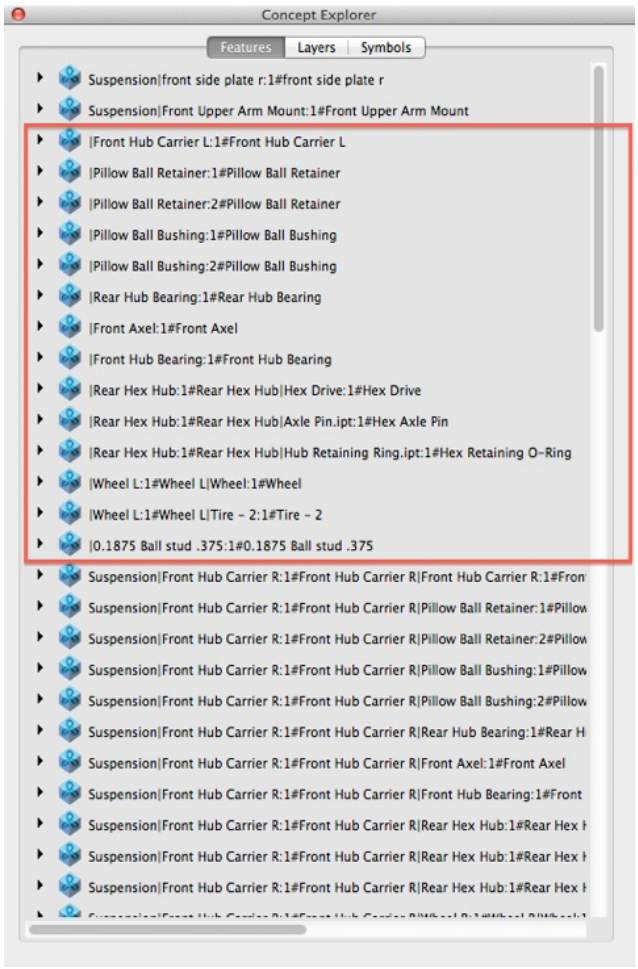
1. Select Rename tool from the PowerPack menu.
2. In the Rename dialog box, type in the string to rename.



3. Click Ok to rename all entities.



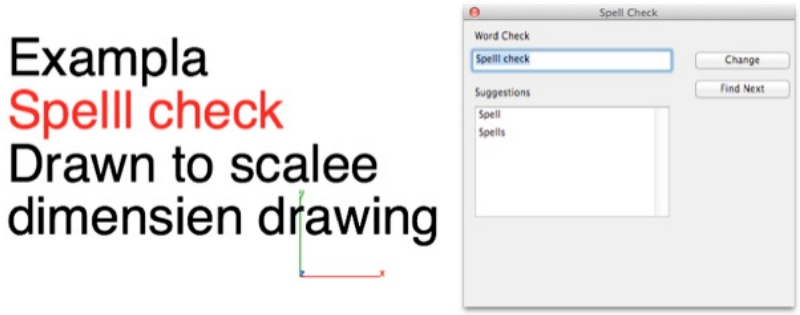




Spell Check
The Spell Check examines all visible text for possible misspellings and provides suggestions.

Example: Spell Check

1. Select the Spell Check option from the PowerPack set of tools.
2. Each entity with potential spelling errors is displayed in the Word Check field.
3. Select Change to update to the Suggested value.
4. Select Find Next to proceed to the next entity.



Isolate Layer From Selection

Sets the display and work layer from the selected objects.

Example: Isolate Layer

1. Select the Isolate Layer option from the PowerPack set of tools.
2. Select an object.
3. The display and contraction layers are updated to the selected object.

Mesh Tools

The Mesh Tools menu provides commands for verifying and repairing common issues with meshes such as welding vertices, locating free or manifold edges, repairing holes, and add/delete/combine/split facets.

- Mesh Analysis
- Auto Repair
- Show Free Edges
- Show Non-Manifold Edges
- Remove Collapsed Facets
- Identify Overlaps
- Check Intersections
- Remove Duplicate Facets
- Fix Flipped Normals
- Weld Vertices
- Remove Unused Vertices
- Close Simple Holes
- Rebuild Normals
- Flip One Normal
- Close Seam
- Make Vertices Planar
- Segment Mesh
- Separate All Parts
- Facet From Lines
- Add Facet
- Delete Facets
- Split Facet Edge
- Convert To Quads
- Convert To Triangles
- Reduce Triangles
- Combine
- Split By Select
- Coordinate Compare
- Segment Analytic
- Align Mesh
- Mesh to Analytic
-

Mesh Analysis
The Mesh Analysis command examines a collection of facets and displays the following information.

- Number of Facets
- Number of Triangles
- Number of Quads
- Surface Area
- Volume (if closed)
- Open Edges
- Collapsed or degenerative faces
- Overlaps
- Double faces
- Number of parts
- Minimum edge length
- Maximum edge length
- Average edge length
- Average aspect ratio
- Highest valence count

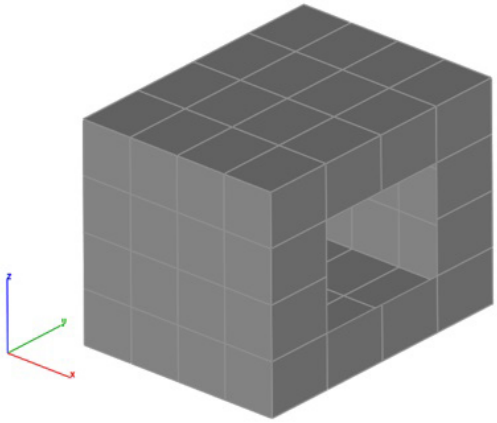
Auto Repair
The Auto Repair command inspects and repairs a mesh for the following instances:

- Dangling Vertices
- Stitch Triangles
- Remove Double Faces
- Remove Collapsed Faces

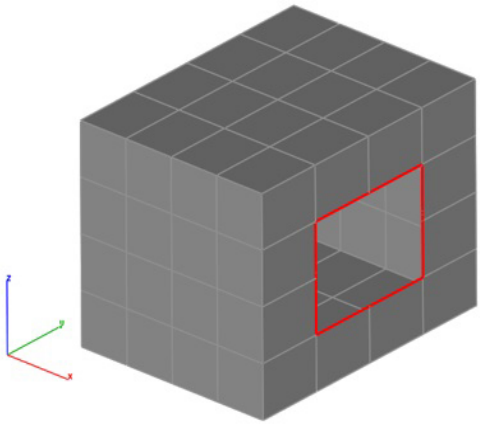
Show Free Edges
The Show Free Edges command inspects a mesh to see if there are any free edges. If there are free edges, the user is presented with an option to make permanent lines out of the edges illustrated below in red.

Example:

1. Select a mesh object.



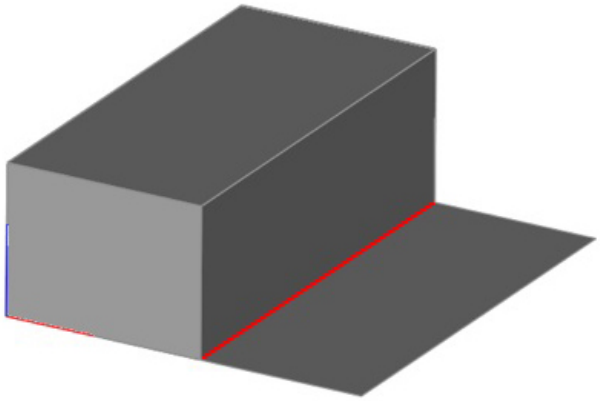
2. A dialog box is displayed with the number of free edges found. Free edges are displayed as red lines in the model.



3. Press Undo to remove the red lines located at the mesh free edges.

Show Non-Manifold Edges

The Show Non-Manifold Edges examines all edges of a mesh to determine if any one edge has more than two facets sharing the edge. Edges that are determined to be non-manifold are displayed with a red line along the edge. Select Undo: Non-Manifold Edges to remove the line from the file.

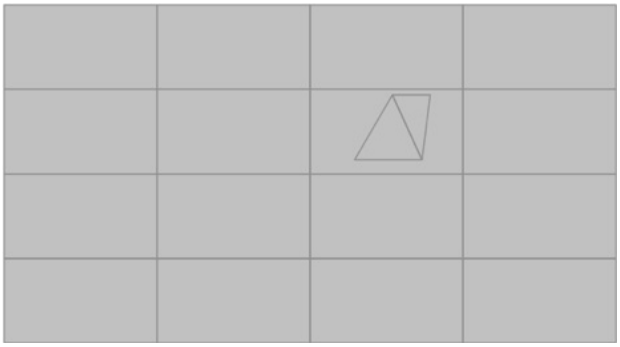


Remove Collapsed Facets
A collapsed facet contains zero area and can cause issues in other operations. Use the Remove Collapsed Facets command to remove zero area facets from the mesh.

Identify Overlaps
The Identify Overlaps tool examines a mesh for overlapping planar facets. If Overlaps are detected, an option is displayed to remove the facets.

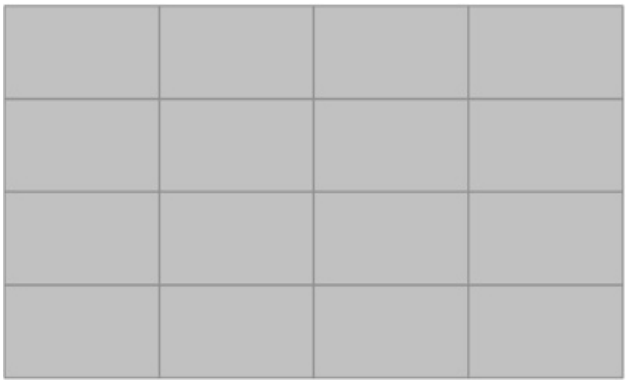
Example with Two Overlapping Facets:

1. Select the Mesh with possible overlaps.



2. A dialog box is displayed reporting that two overlaps were found.

3. Pressing Yes will delete the overlaps.



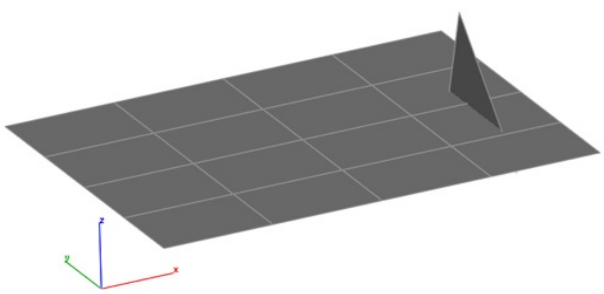
4. Pressing No will show the overlaps as Line entities. Select Undo to remove the lines.

Check Intersections

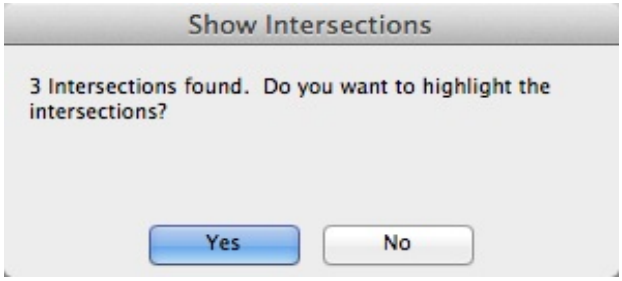
The Check Intersections tool examines a mesh for non-planar intersecting facets. If intersections are detected, an option is displayed to remove the facets or display the intersection.

Example:

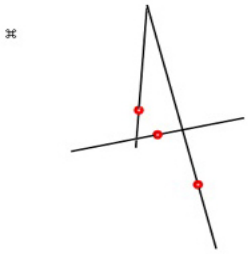
1. Select the Mesh with possible intersections.



2. A dialog box is displayed with the number of intersections found.

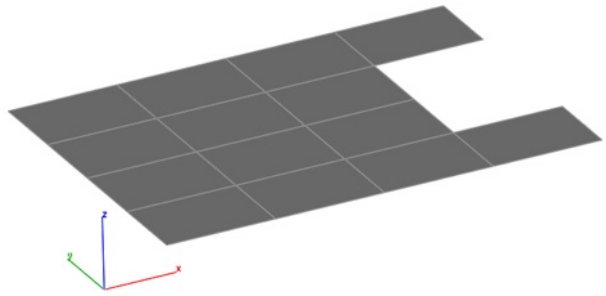


3. Pressing Yes will create lines and points at the intersections.



4. A dialog box is displayed asking if you want to delete all facets involve with intersections.

5. Pressing Yes will remove the intersecting facets.

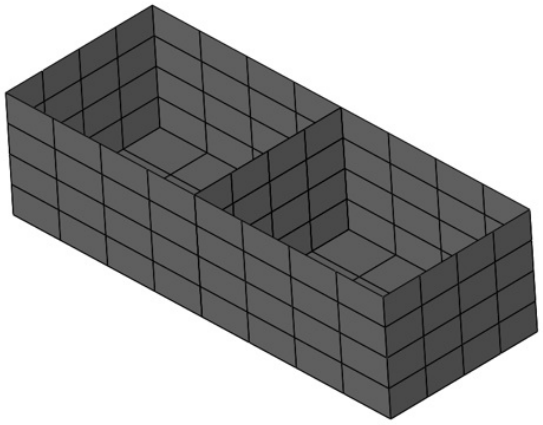


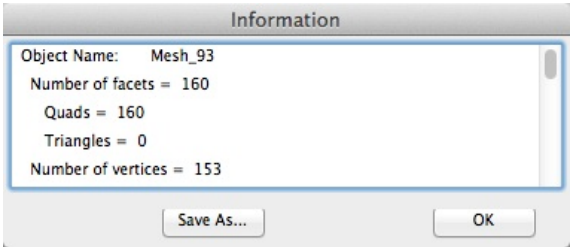
Remove Duplicated Facets

This tool removes all facets that are duplicated.

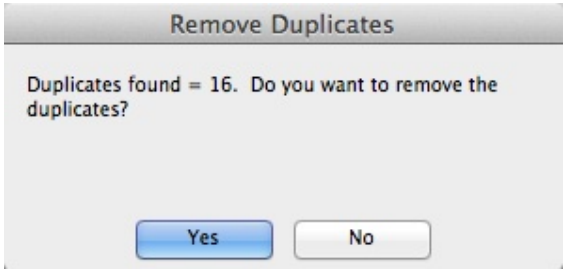
Example:

1. Select a mesh with possible duplicated facets. Original mesh has 160 facets.

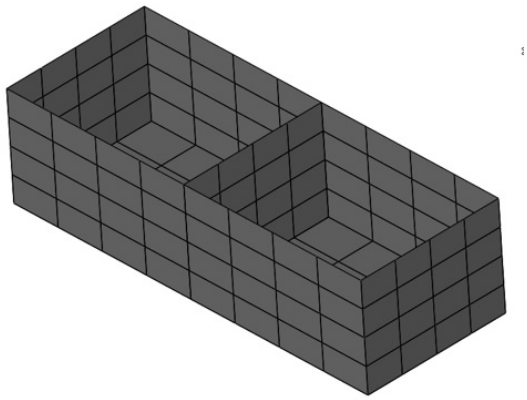


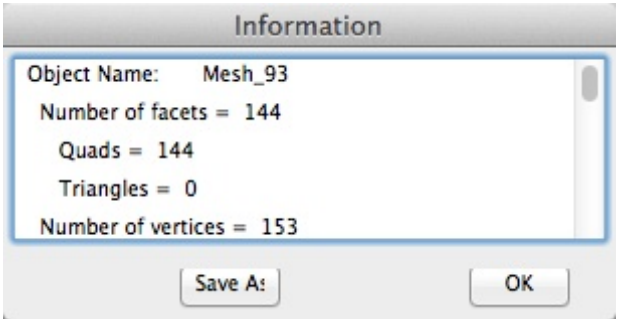


2. Select Yes to remove the duplicated facets.



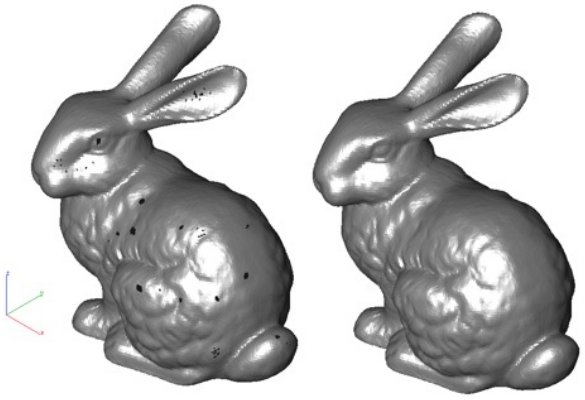
Mesh is reduced by 16 facets to 144.





Fix Flipped Normals

This tool corrects surface normals that are not properly oriented.



Weld Vertices

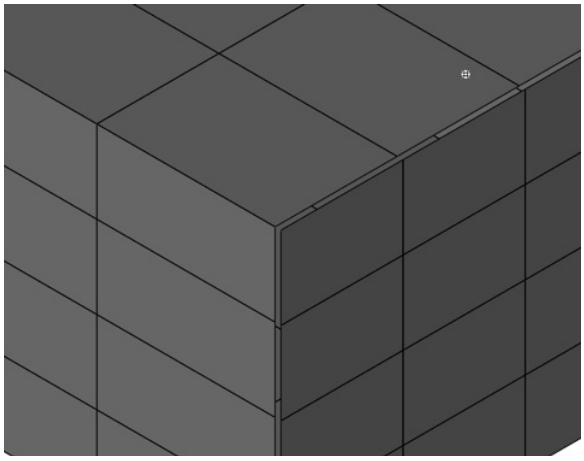
The Weld Vertices tool joins vertices of a mesh. The tool has two options:

1. Entire Mesh All vertices within the mesh within the specified tolerance are examined for joining.

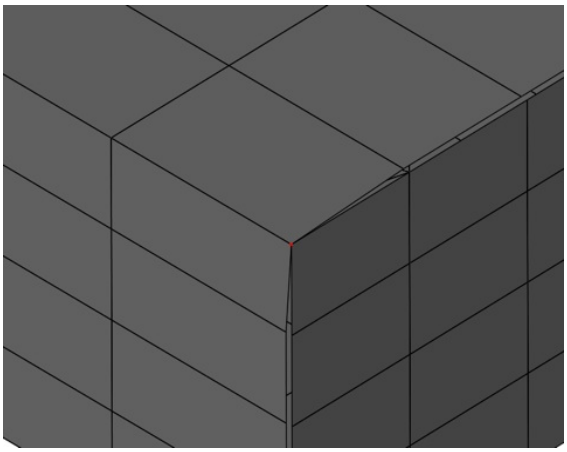
2. Selected Only the selected vertices are joined. The selected vertices are joined independent of a tolerance.

Example:

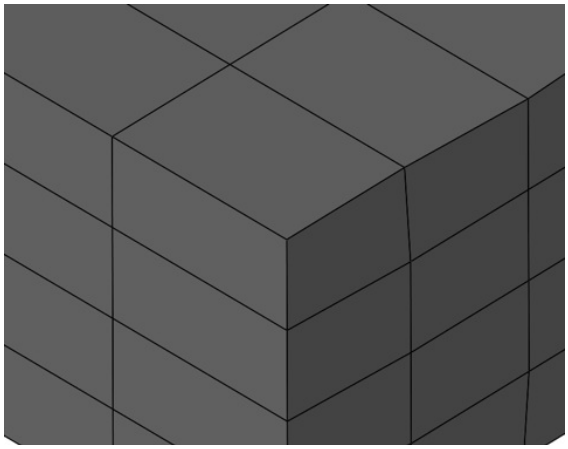
1. Pick "Selected" from the pull down menu.



2. Pick two vertices that are to be joined together.



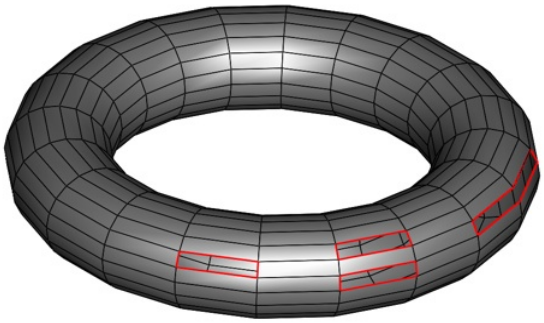
3. Pick "Entire Mesh" from the pull down menu.
4. Specify a tolerance from within the data entry window.
5. Vertices within the tolerance are joined together.



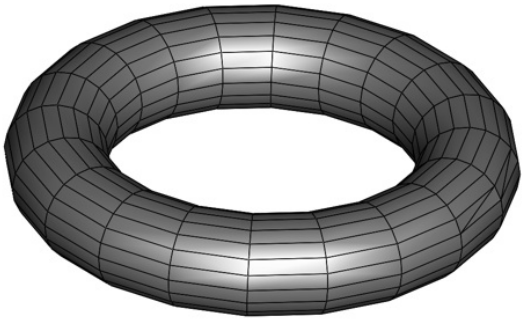
Remove Unused Vertices
Compacts the vertices associated with a mesh to be the minimum necessary for the definition.

Close Simple Holes
The Close Simple Holes tool attempts to fill with quads or triangles holes identified in a mesh. The tool has two options allowing for Closing All Holes or Close specific edges.

Example 1: Close All Holes
1. Select a mesh to close all holes.

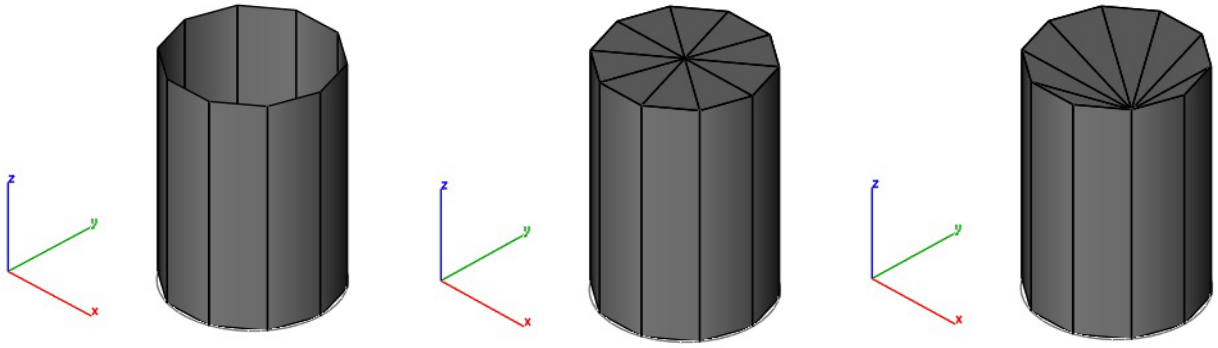


2. Mesh is filled with triangles or quads.



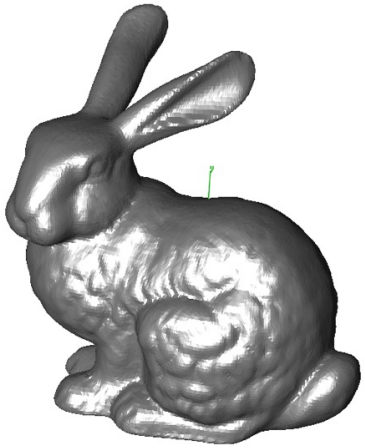
Example 2: Close Edge

1. Select an edge of a mesh to close.
2. Edge is triangulated and filled with facets.
3. From the Option menu, select "Add Center Point" to close a mesh about an approximated hole center.

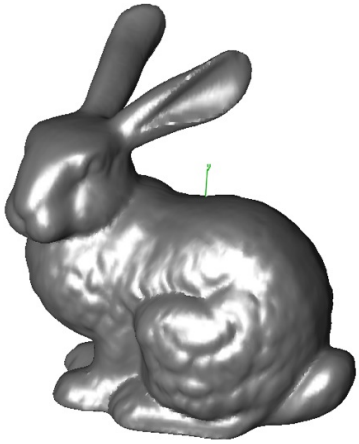


Rebuild Normals
The Rebuild Normal tool recalculates all normals. The normal at a vertex is the average of the neighbors.

Example:
1. Select mesh model to rebuild normals.



3. Mesh normals are updated.

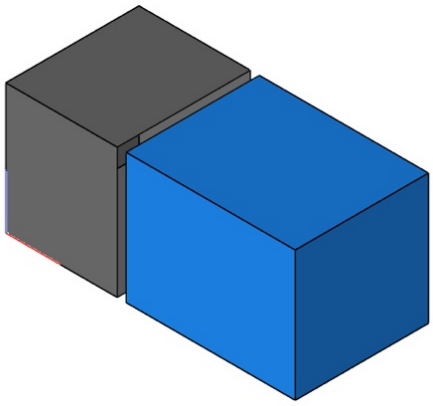


Flip One Normal
The Flip One Normal command prompts the user to select a specific facet to flip the normal.

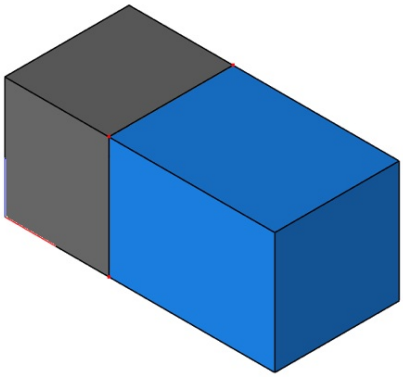
Close Seam
The Close Seam tool will move vertices within a specified tolerance between to mesh objects.

Example:

1. Select the Close Seam tool.
2. Box select a region that captures the two mesh vertices to check.

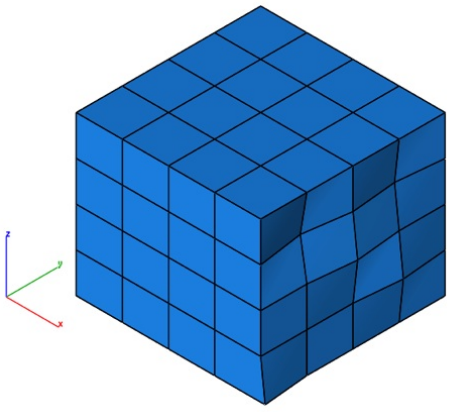


3. Vertices of Mesh 2 that are within the specified distance are moved to Mesh.

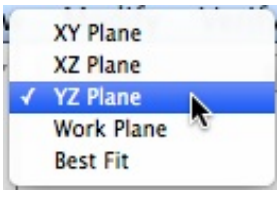


Make Vertices Planar

The Make Vertices Planar tool takes the selected vertices and projects the vertices into a specified plane.

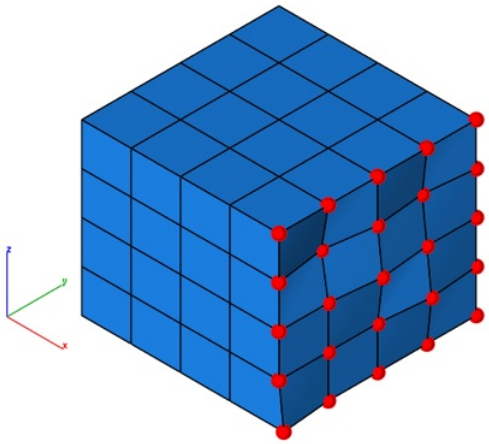


Example:

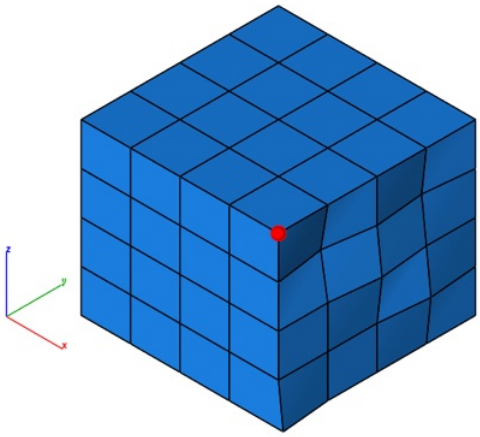


1. Select the Make Vertices Planar command
2. Select the YZ Projection Plane from the data entry window pull down menu.

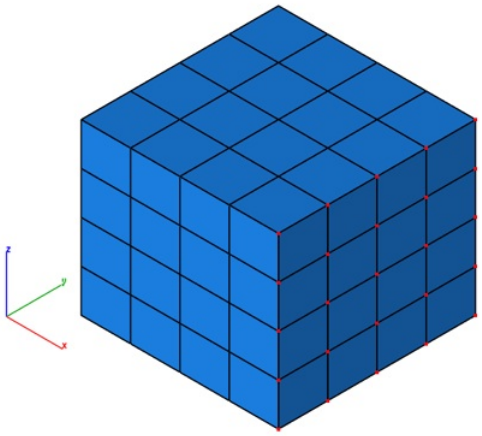
4. Select the vertices to project.



5. Select a point in the projection plane.



6. The mesh vertices are projected into the plane.

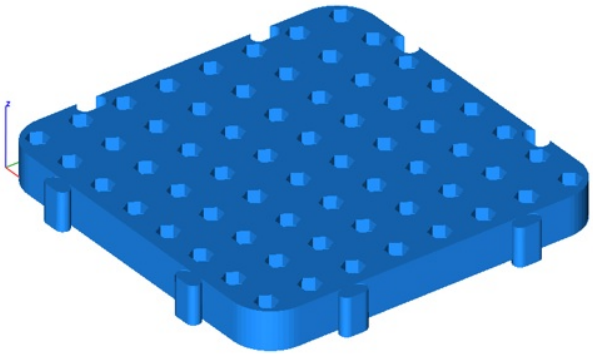


Segment Mesh

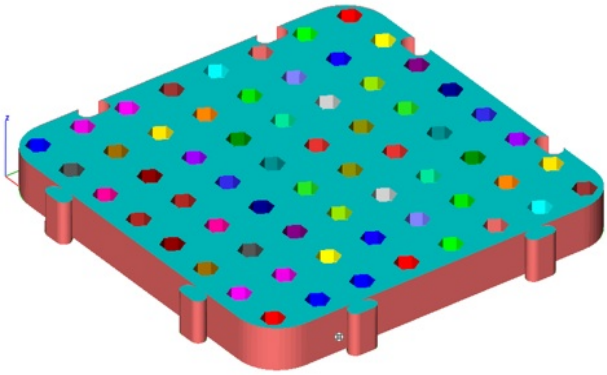
The Segment Mesh tool decimates a mesh into planar and connected components.

Example:

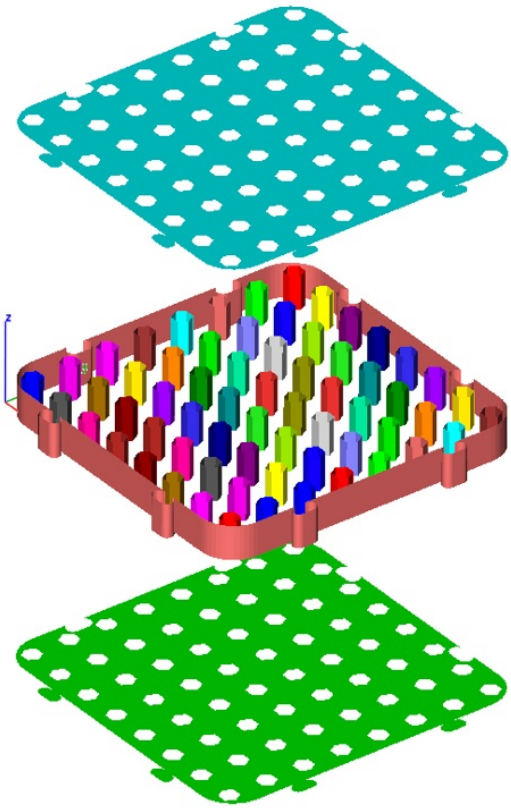
1. Select the Segment Mesh tool.
2. Select the single mesh object to segment.



3. The selected mesh is converted into a collection of meshes separated by planar and connected elements. In this example 67 additional mesh objects created.
4. Meshes colored by other tools to better show individual meshes.

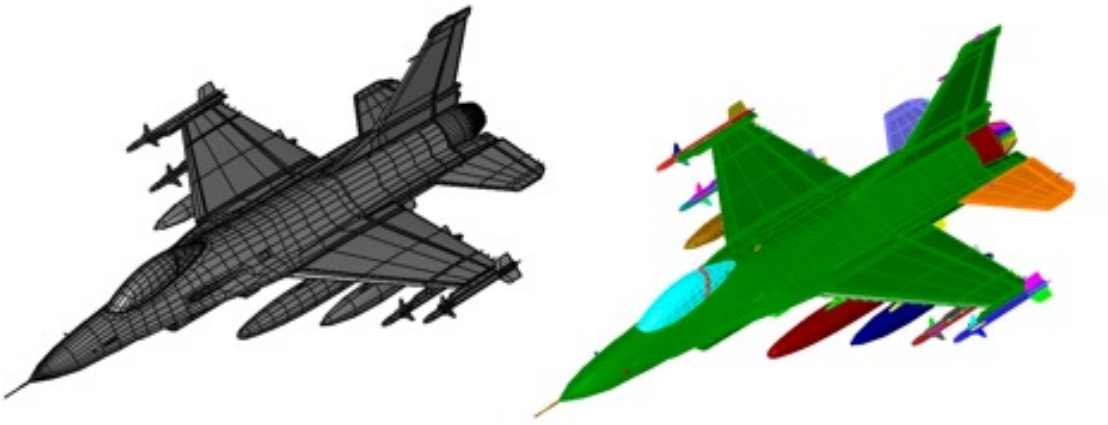


5. Meshes exploded by additional tools to better show individual meshes.



Separate All Parts

The Separate All Parts command examines the connectivity between facets to determine individual mesh parts. This is useful for separating individual meshes that come in as one mesh as a result of a file import such as STL. In the example below, an OBJ file representing a F-16 was imported as one mesh. Using the Separate All Parts command, 166 individual components were extracted.

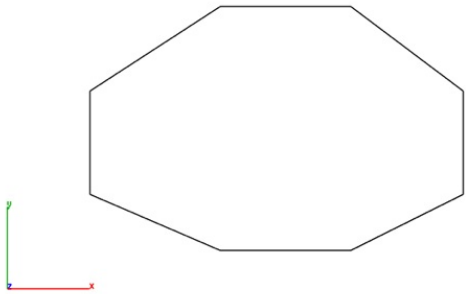


Facet From Lines

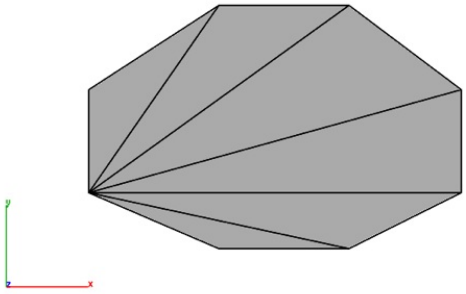
The Facet From Lines tools create triangles from a collection of connected line objects.

Example 1:

1. Select the Facet From Lines tool.
2. Select the lines to facet.

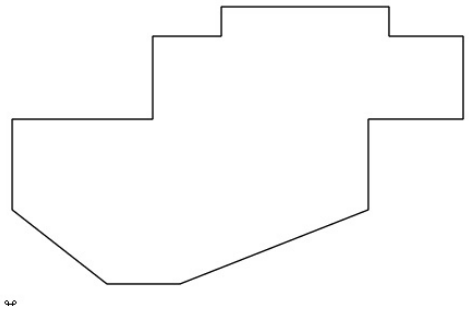


3. Resultant triangles from lines.

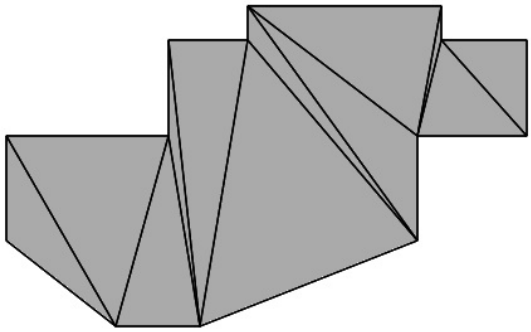


Example 2:

1. Select the Facet From Lines tool.
2. Select the lines to facet.



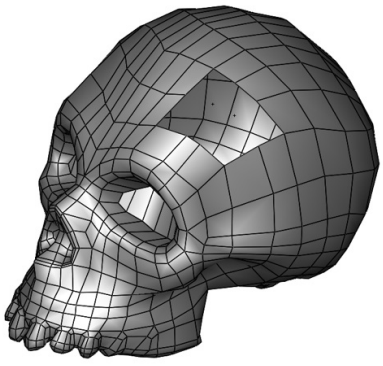
3. Resultant triangles from lines



Add Facet

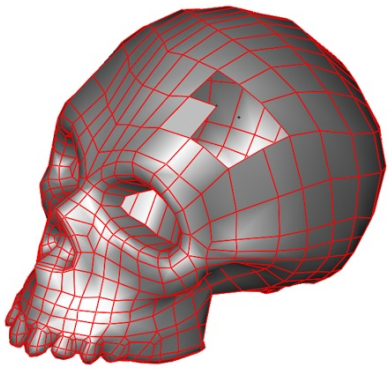
The Add Facet command will insert 3 or 4 sided facets into an existing mesh by the user specifying vertex locations.

Example:

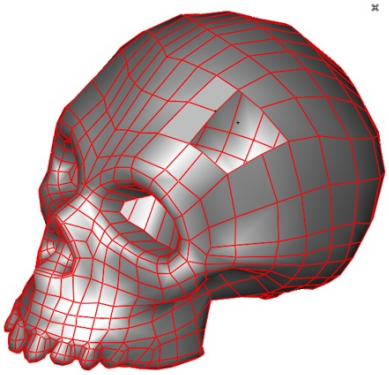


1. Select the Add Facet command.

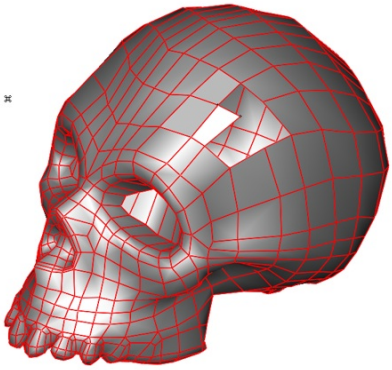
2. Select the mesh.
3. Specify four vertex locations for first facet.



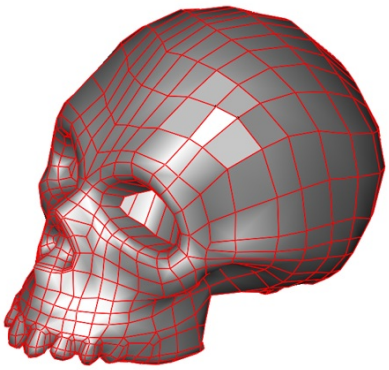
4. Specify four vertex locations for second facet.



5. Specify four vertex locations for third facet.



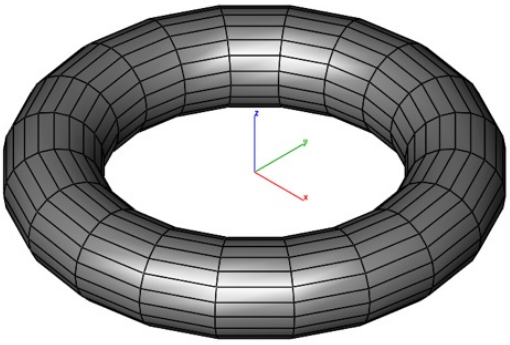
6. Continue until all desired facets are added.



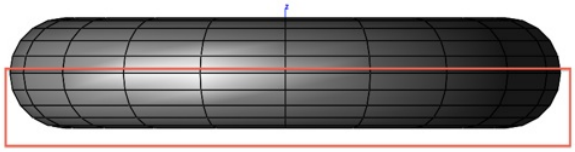
Delete Facet

The Delete Facets command removes facets using the box selection interface. Use Deep Select to delete one facet at a time from a mesh.

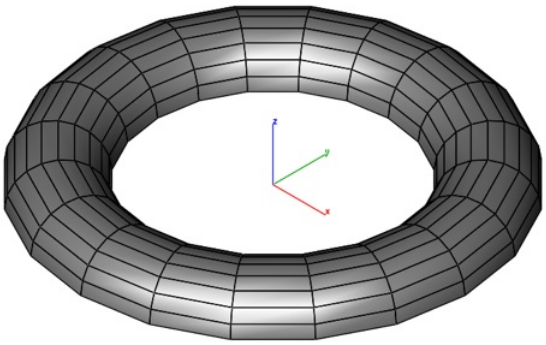
Example:



1. Select the Delete Facets command.
2. View the modeling in an orientation that allows you to select the facets to delete. Box select the region to delete.



3. Selected facets are removed from the mesh.

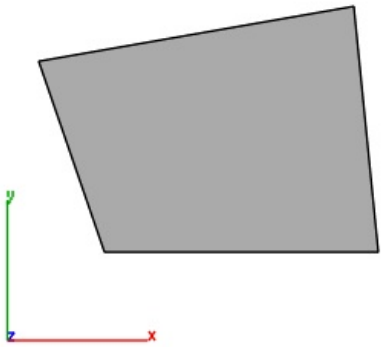


Split Facet Edge

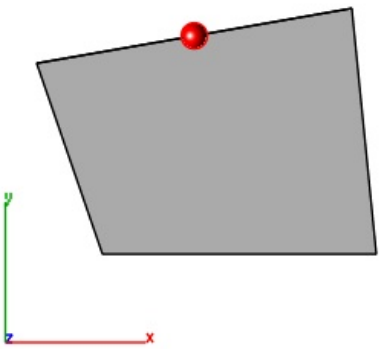
The Split Facet Edge command takes a triangle or quad and splits an edge introducing two additional facets.

Example 1:

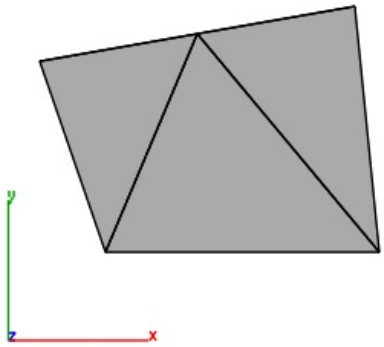
1. Select the Split Facet Edge command.



2. Select the mesh to split an edge.
3. Specify location to split edge.

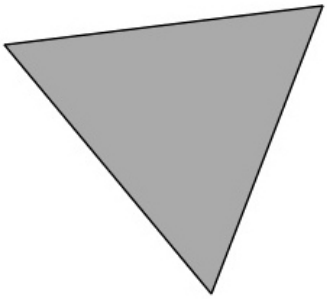


4. Selected mesh is split into three triangles.

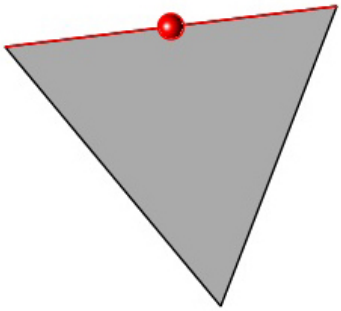


Example 2:

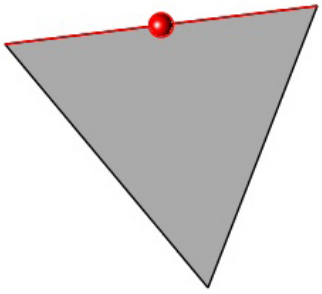
1. Select the Split Facet Edge command.
2. Select the mesh to split an edge.



3. Specify location to split edge.

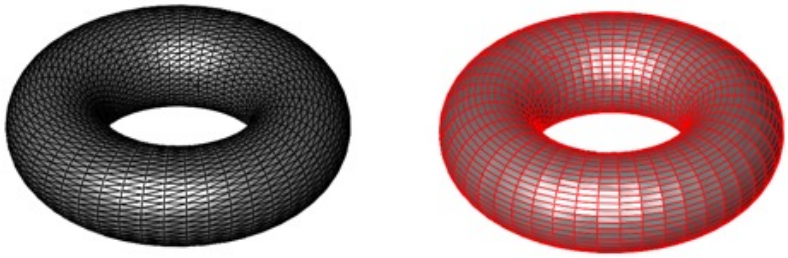


4. Selected mesh is split into two triangles.



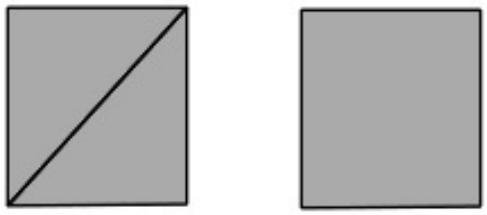
Convert To Quads

The Convert To Quads command examines a triangle mesh and recreates quads were possible.



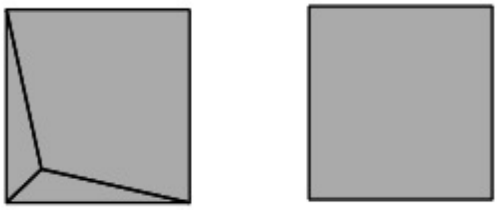
The following cases are supported:

1. Shared Edge (Two Triangles -> One Quad)

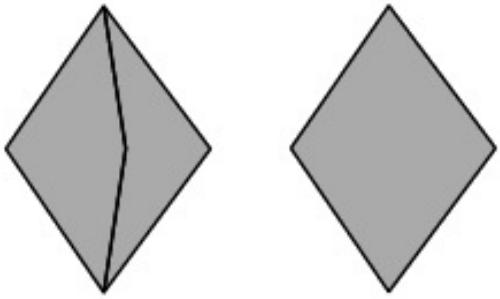


2. Shared Center Vertex (Four Triangles -> One Quad)

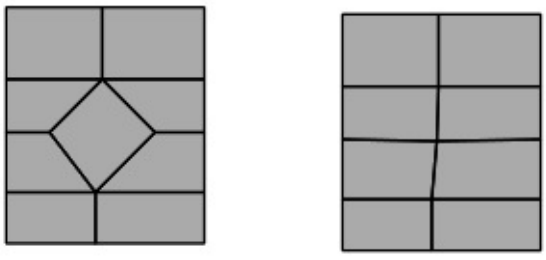
3. Shared Center Vertex (Three Triangles -> One Quad)



4. Doublet (Two Quads -> One Quad)



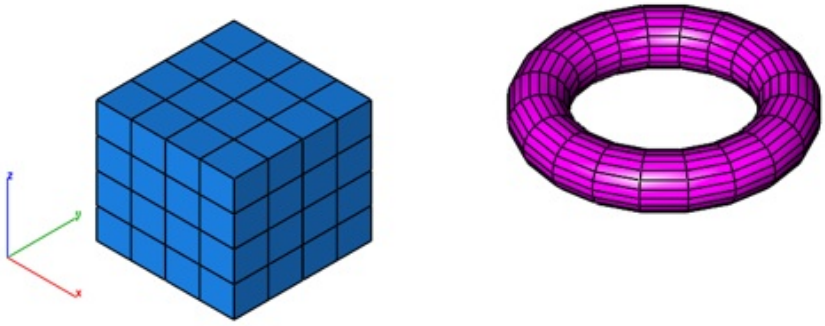
5. Inner Quad Diamonds (Five Quads -> Four Quads)



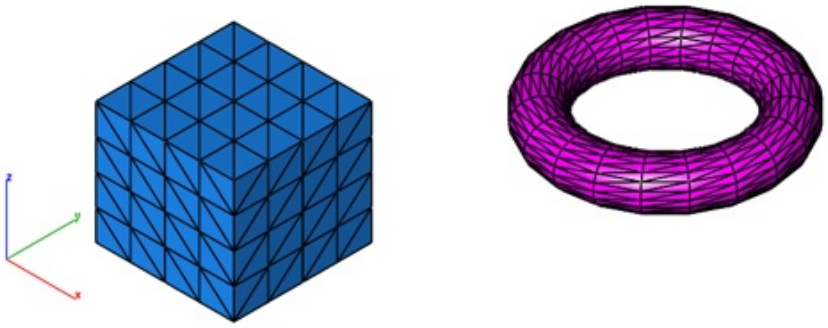
Convert to Quad Demonstration Video
Converts a 47k STL file (all triangles) into a quad dominate mesh suitable for conversion to a NURB solid.

Convert To Triangles
The Convert To Triangles command changes all quads into triangles.

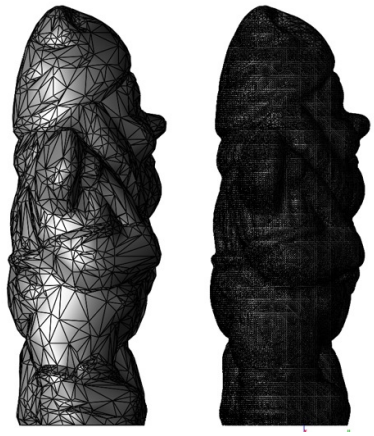
Example:
1. Select the Convert to Triangles command.
2. Select the mesh objects to convert to triangles.



3. Selected mesh is converted into triangles. A dialog box displays how many quads were converted into triangles.

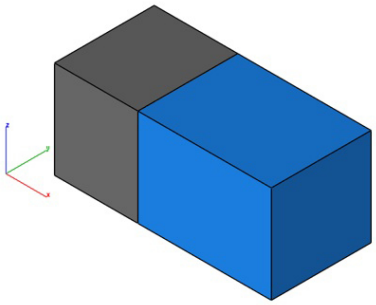


Reduce Triangles
The Reduce Triangles command removes facets based on edge length size and curvature. The model below was reduced from 139,422 facets to 4,352.

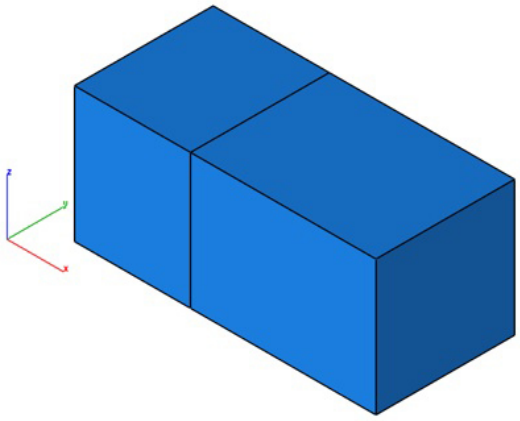


Combine
The Combine tool merges two meshes into one. Vertices that are shared are merged together.

Example:



1. Select the Combine tool.
2. Select the two mesh objects to join as one mesh.

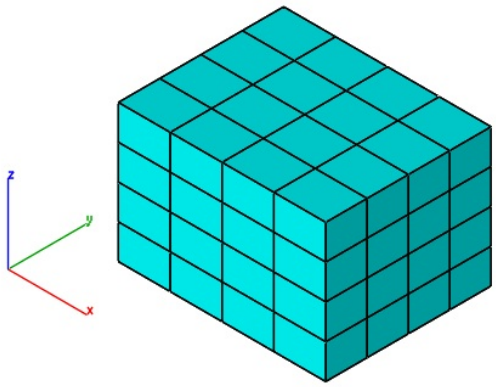


Note: The combine tool does not perform a boolean operation.

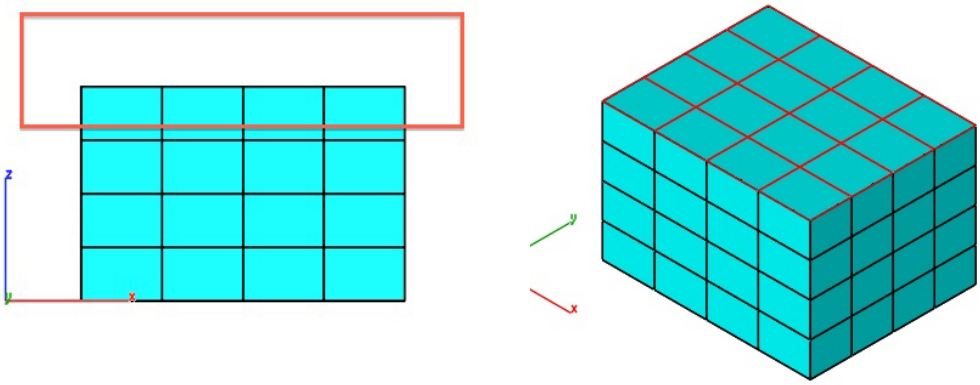
Split By Select

The Split by Select tool separates a collection of selected facets into a new mesh.

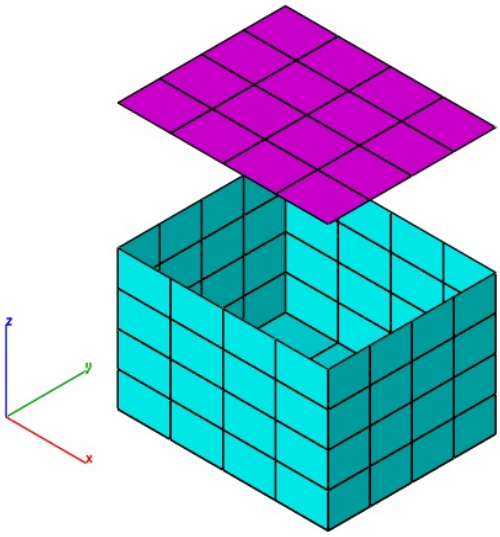
Example:



1. Select the Split by Select tool.
2. Box select the facets to split into a new mesh object.



3. Facets are split and two new mesh objects are created. Image below moves and colors the objects using separate tools for visualization.

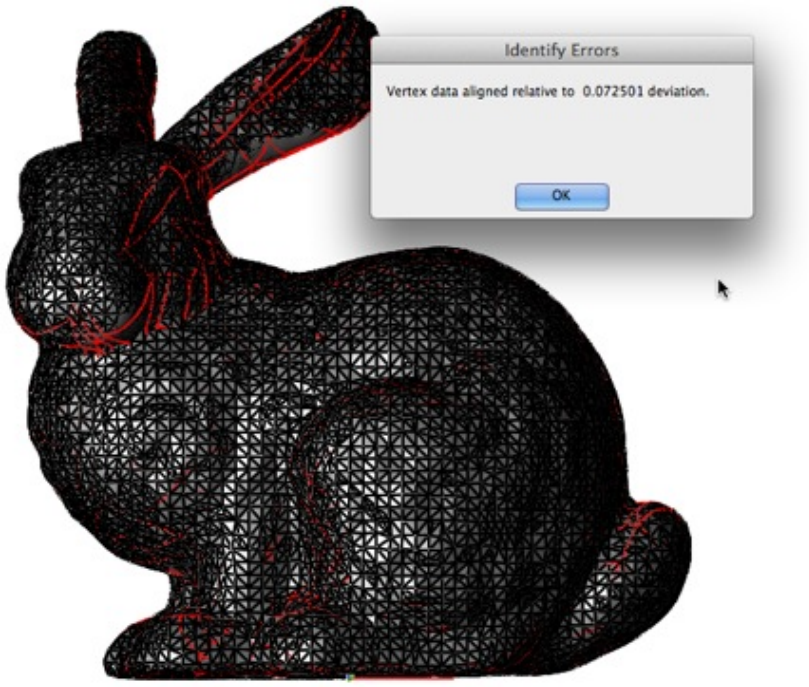


Coordinate Compare
The Coordinate Compare tool will precisely compare a set of points to a NURB model. For data points, you can select a point cloud or a mesh object to compare with the NURB model. A NURB model is either a surface or solid.

Depending on the point data set size, this tool is computationally heavy. However, this tool is multi-processor aware and will use additional processers if they are available.

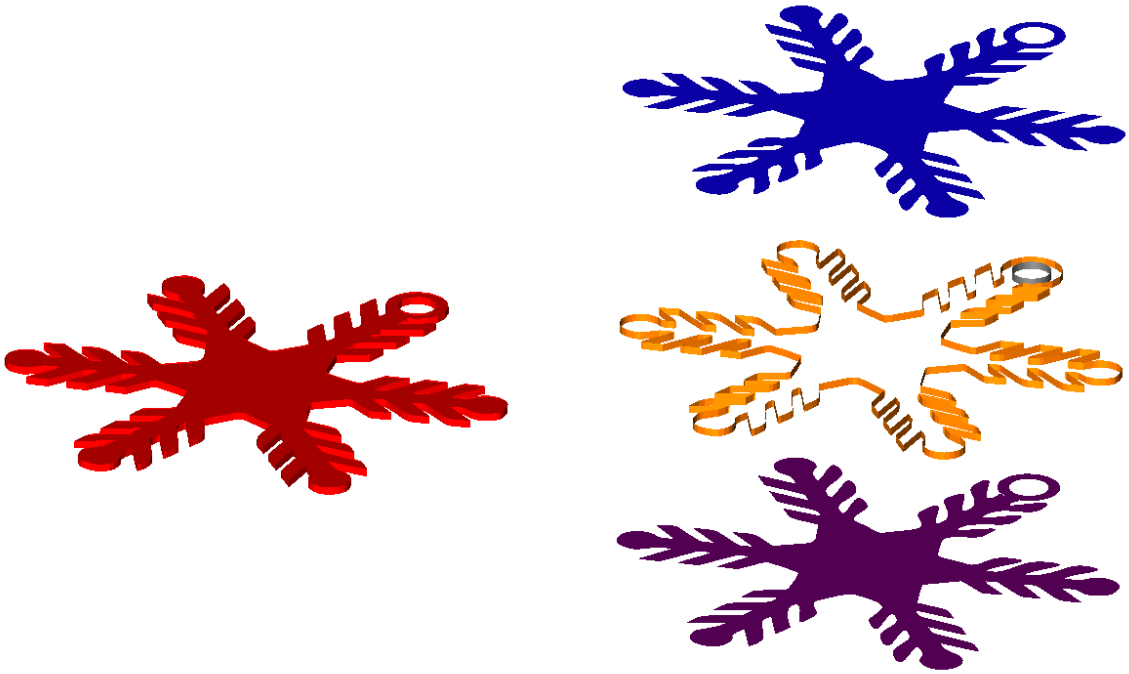
In addition to displaying a dialog box indicating the maximum deviation between two parts, line objects are created representing the minimum distance vector between the two parts.

Note: A Point cloud object is created when reading in PLY files.

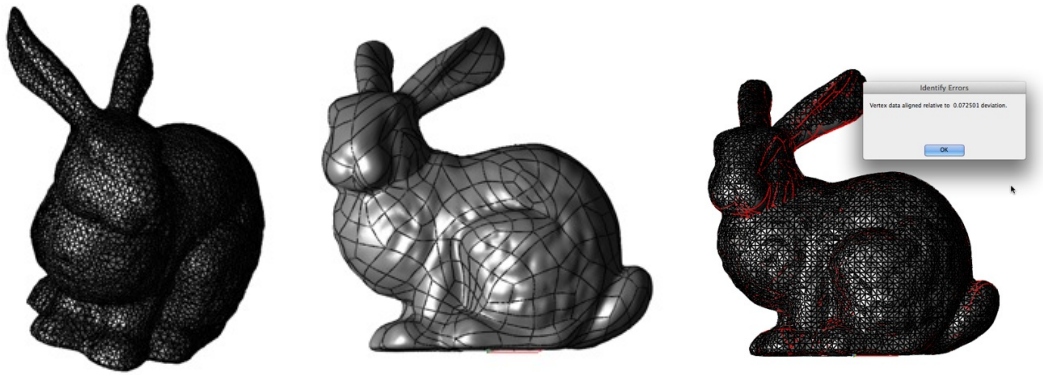


Segment Analytic
The Segment Analytic tool breaks down a complex mesh into a collection of simple analytic meshes such as planar, ruled, and revolved. The Segment Analytic tool is useful to manually convert a mesh into a solid allowing.

In the example below, the red mesh on the left is segmented and converting into three analytical surfaces, which can be manually stitched into a solid.

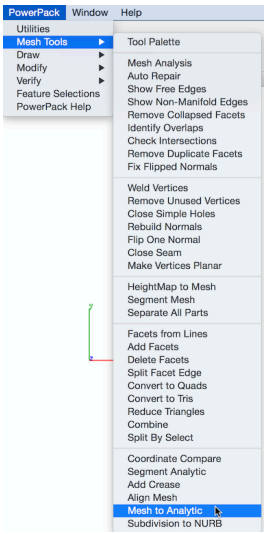


Align Mesh
The Align Mesh to Part tool will transform a mesh to lie precisely on a surface or solid. The tool will additionally display a dialog box indicating how closely the mesh represents the surface or solid.

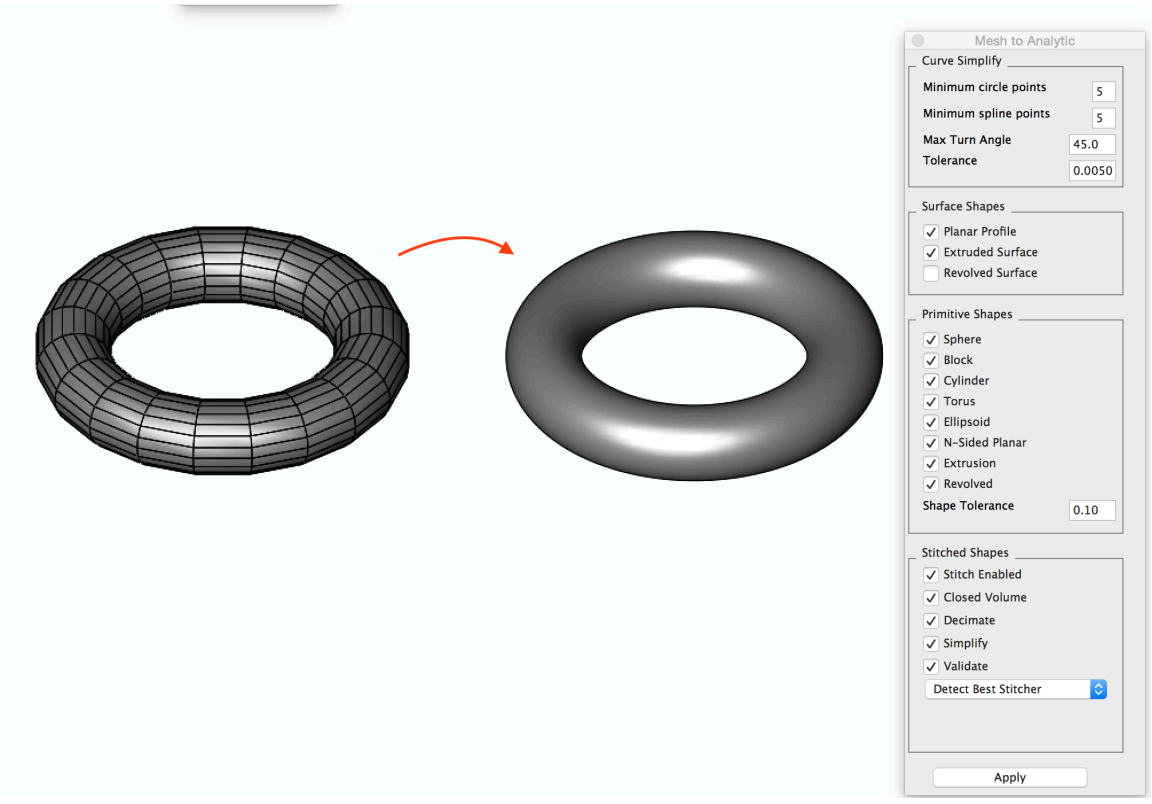


Mesh To Analytic
This tool converts a collection of triangle and/or quad based facets into an analytical solid created from primitives, profiles, or stitched facets. An analytical solid is a final shape consisting of planar, cylindrical, spherical, torus, and extruded faces.

Example:
1. Select the Mesh to Analytic tool from the PowerPack menu.

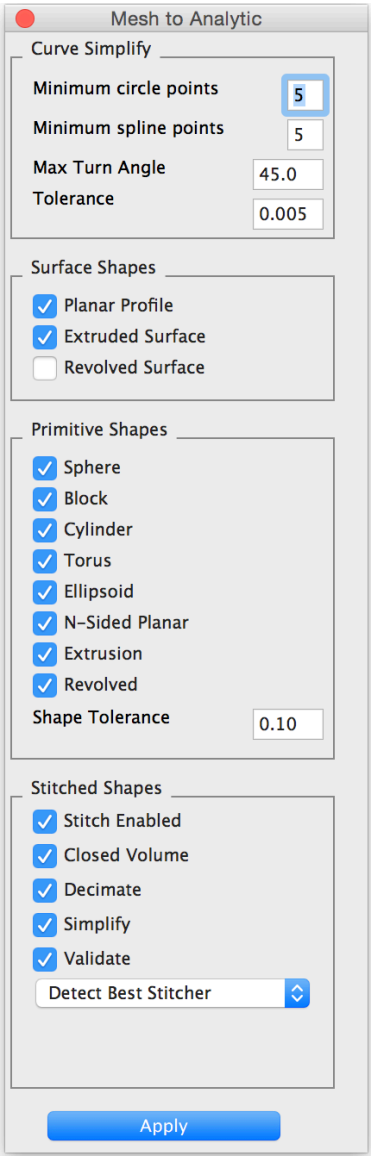


2. The Mesh to Analytic dialog box is displayed, use the default settings.
3. Select a mesh to convert.



Mesh To Analytic Settings

The Mesh to Analytic dialog box provide access to all the settings for converting a mesh into a solid. The tool provides a variety of options that set attributes for tolerances, simplification types, closed/open, and validation.



Curve Simplify
- Minimum circle points
  - # Points needed to represent circle
- Minimum spline points
  - # Points needed to represent spline
- Max Turn Angle
  - # Angle between segments of circle/spline
- Tolerance
  - Deviation between polyline and curve

Surface Shapes
- Planar Profile
  - Planar face with holes
- Extruded Surface
  - Open shape extruded
- Revolved Surface
  - Open shape revolved

Primitive Shapes
- Sphere
  - Center point sphere solid
- Block
  - Center point block solid
- Cylinder
  - Two point cylinder solid
- Torus
  - One point torus solid
- Ellipsoid
  - One point ellipsoid solid

Extrusion
- Extruded closed profile

Revolved
- Close profile 360 revolution

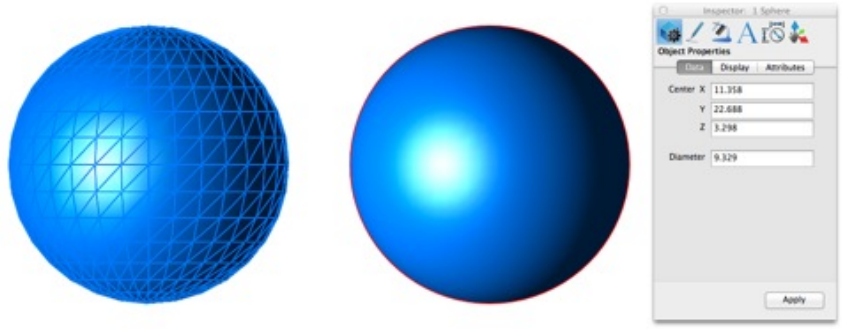
Shape Tolerance
- Deviation between solid and mesh

Stitched Shapes
- Stitch Enabled
- Closed Volume
- Decimate
- Simplify
- Validate
- Detect Best Stitcher

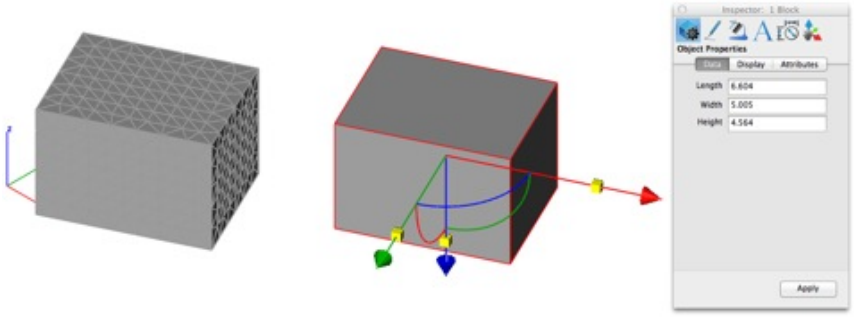
Mesh To Analytic Examples

Mesh to Primitive

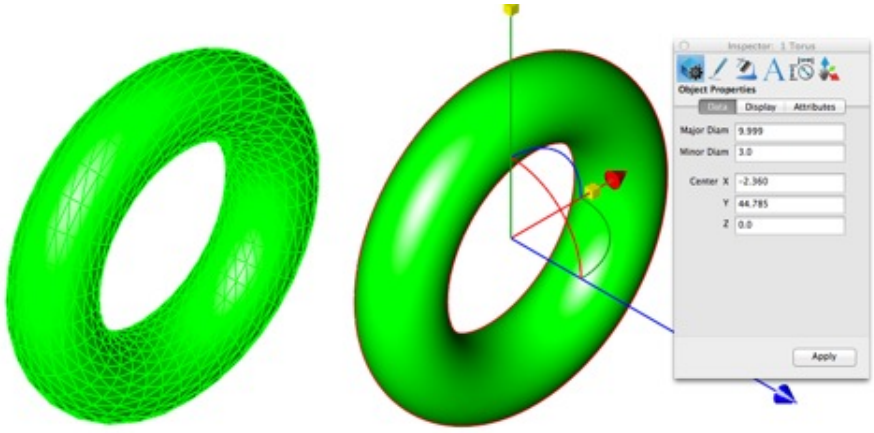
Sphere : Converts a collection of triangle or quad facets into a solid sphere defined by a center point and diameter.



Block ： Converts a collection of triangle or quad facets into a solid block defined by length, width, and height. An axis is defined to the part extracted from the orientations of the planes.

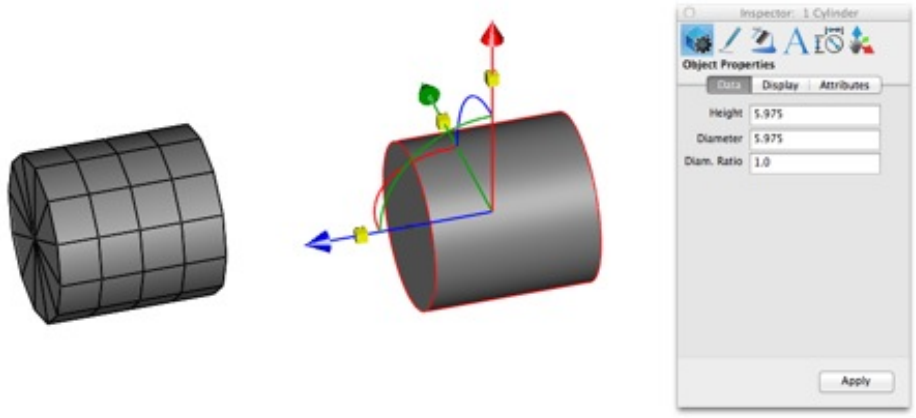


Torus ： Converts a collection of triangle or quad facets into a solid torus defined by a major and minor diameter and center point.

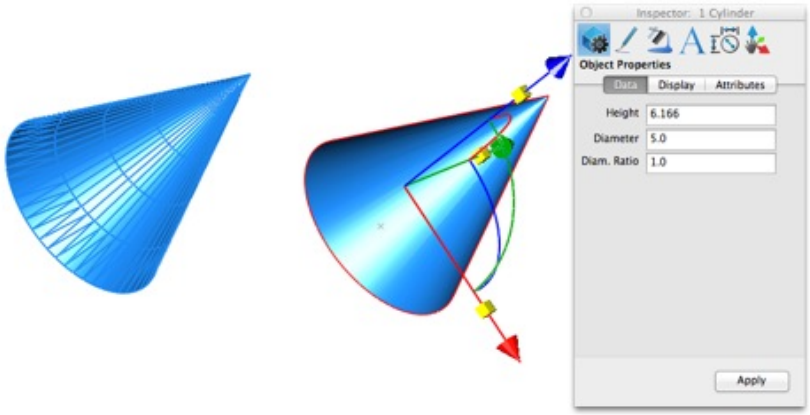


Cylinder ：Converts a collection of triangle or quad facets into a solid

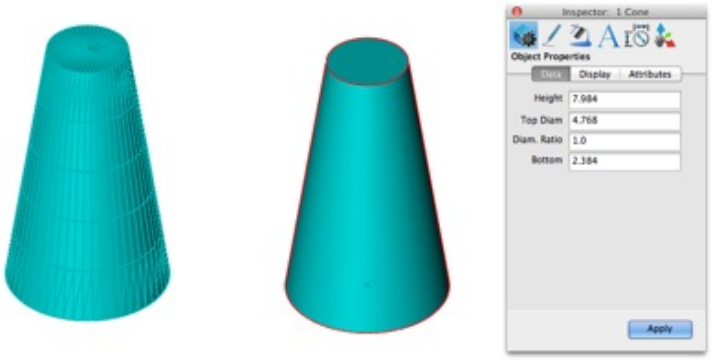
cylinder defined by height, diameter, point at base center, and point at top center.



Cone ： Converts a collection of triangle or quad facets into a solid cone defined by height, diameter, point at base center.

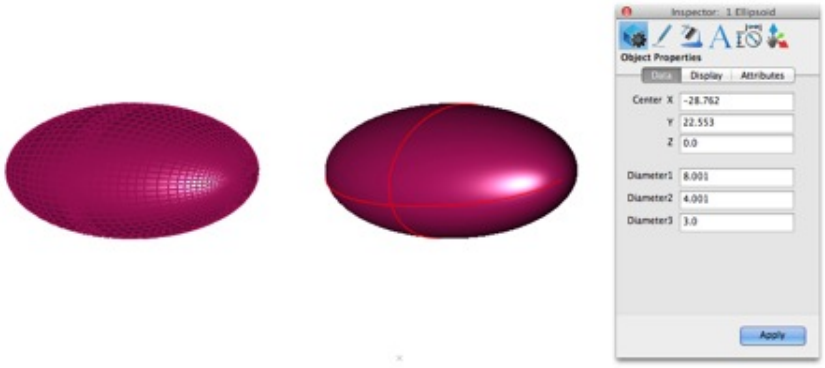


Truncated Cone ： Converts a collection of triangle or quad facets into a solid truncated cone defined by height, base & top diameter, and point at base center.

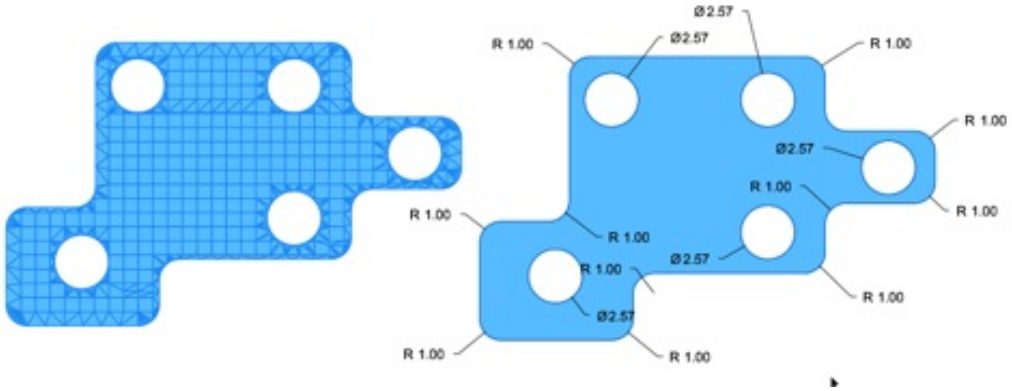


Ellipsoid ： Converts a collection of triangle or quad facets into a solid ellipsoid defined by center, and three diameters. An axis is defined for the

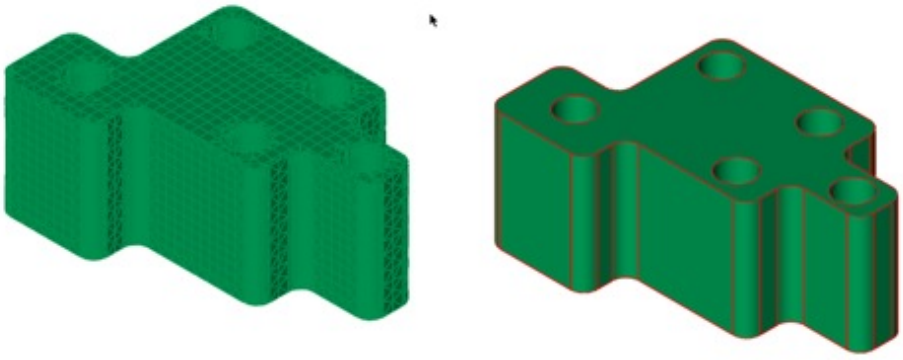
part based on the orientation.



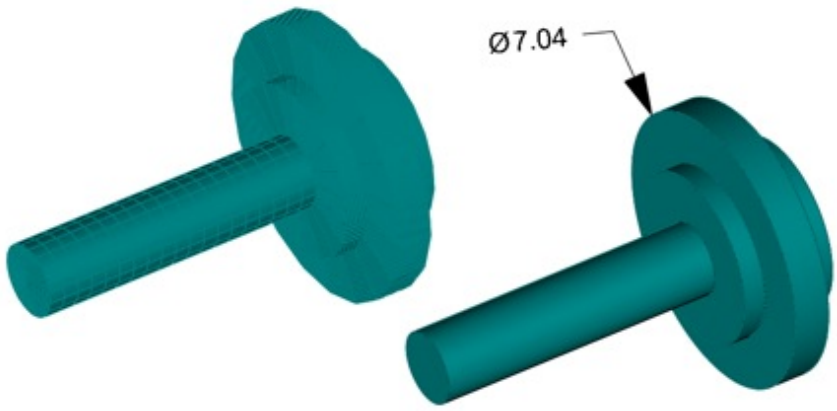
Mesh from Profile ： Converts a planar collection of triangle or quad facets into a planar surface. Mesh segments corresponding to arcs and circles are converted.



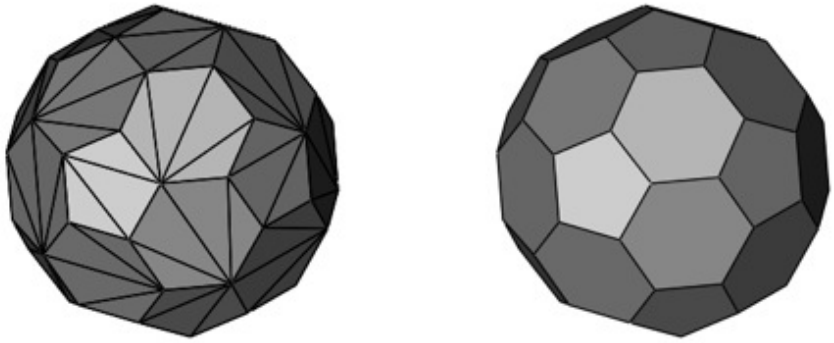
Extrusion ： Converts a collection of triangle or quad facets into an extruded solid. Includes conversion of precise cylindrical holes and fillets.



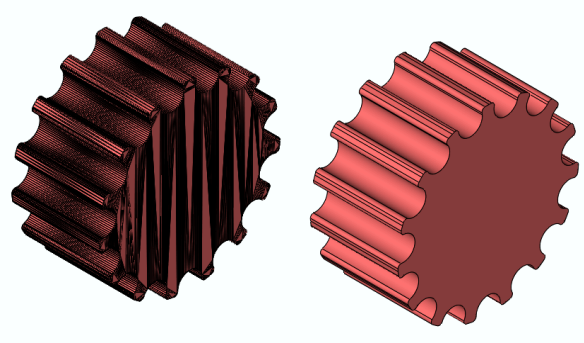
Revolved and Lathed ： Converts a collection of triangle or quad facets into a revolved solid. Includes conversion of precise cylindrical holes and fillets. Only full 360-degree lathe and revolved shapes are recognized.



Mesh from Stitched Faces ： N-Sided Converts a collection of triangle or quad facets into a N-Sided solid up to 20 sides.



Arbitrary Stitched ： Converts a collection of triangle or quad facets into planar facetted solid.



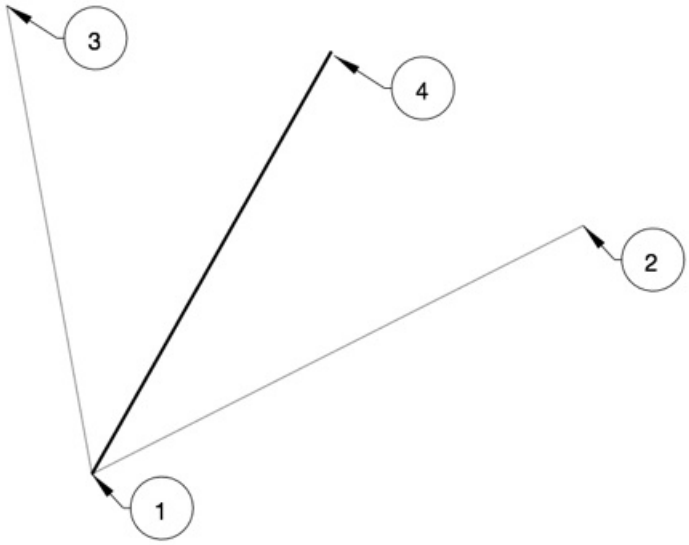
Draw

The Draw Menu provides additional construction methods for creating new geometry such as lines, circles, splines, and surfaces.

- Line Bisector
- Line Best Fit
- Line Normal To Surface
- Line Perpendicular To Two Curves
- Circle Best Fit
- Circle Normal To Curve
- Average Two Curves
- Curve From Two Views
- Best Fit Plane
- Surface Grid
- Shadow Projections

Line Bisector

The Line Bisector command creates a line that bisects at specified angle using four point locations.



Example:

1. Select the Line Bisector command.
2. Pick the start of the bisector line, shown by label 1 in the above figure.
3. Pick the start of the angle to bisect, as shown by label 2 in the above figure.
4. Pick the end of the angle to bisect, as shown by label 3 in the above figure.

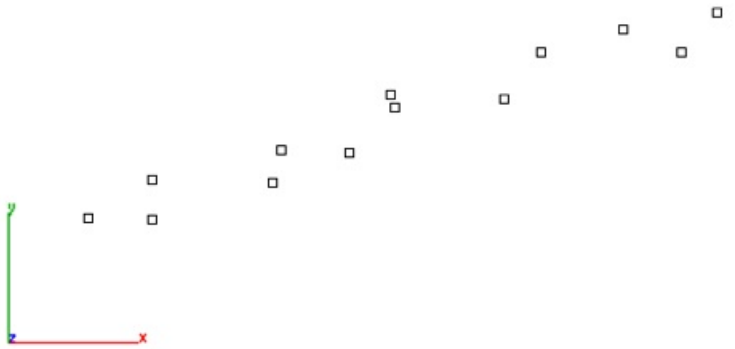
5. Pick the end of the end of the line, as shown by label 4 in the above figure. This location defines the length of the line.

Line Best Fit

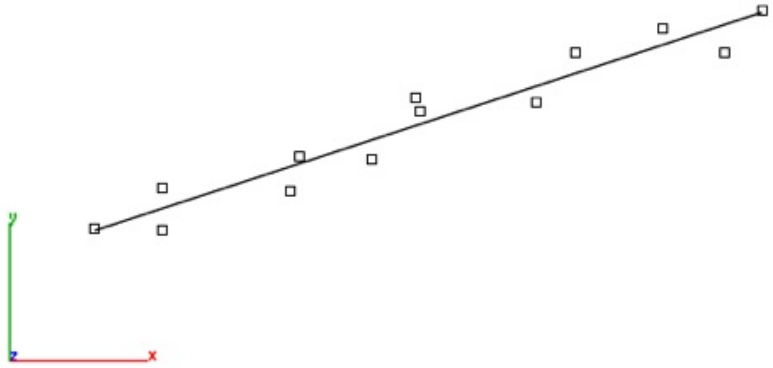
The Best Fit Line calculates a line that best represents the data. This tool works only with 2D XY data.

Example:

1. Select the Best Fit Line Tool.
2. Pick or digitize a location for the points to best fit.

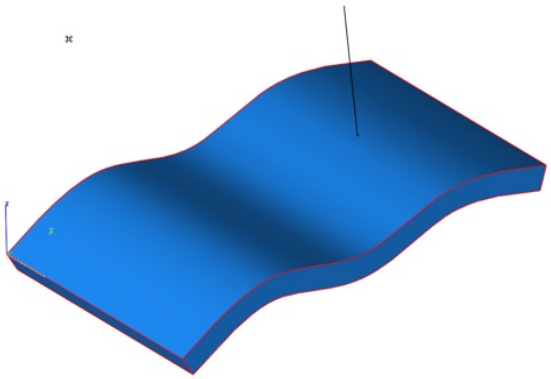


3. A line entity is created that best fits the data.

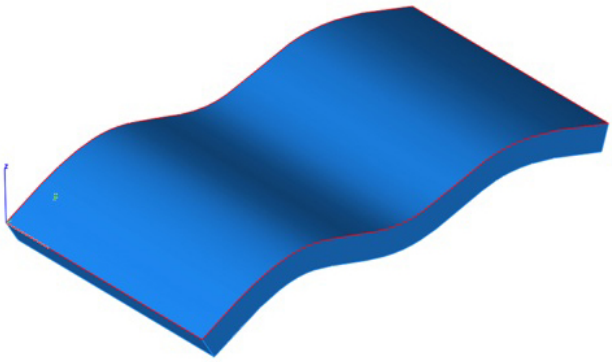


Line Normal To Surface

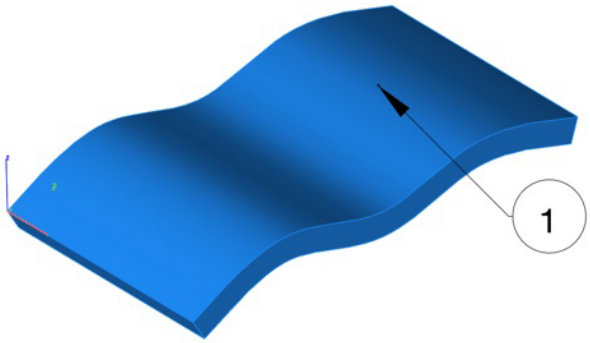
Constructs a line normal to a surface or face of solid.



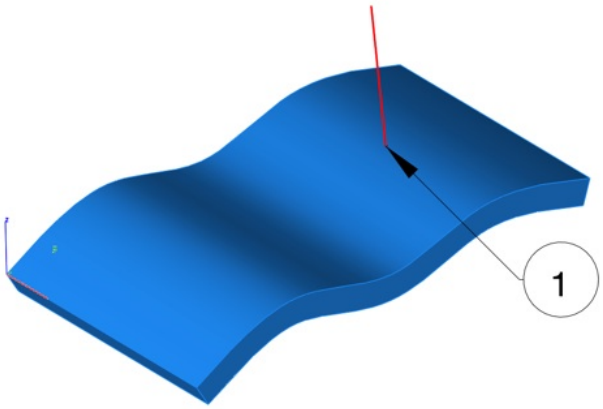
Example:
1. Select the Line Normal to Surface/Face 2. Select the face line is normal to.



2. Pick a location on the surface where line is normal

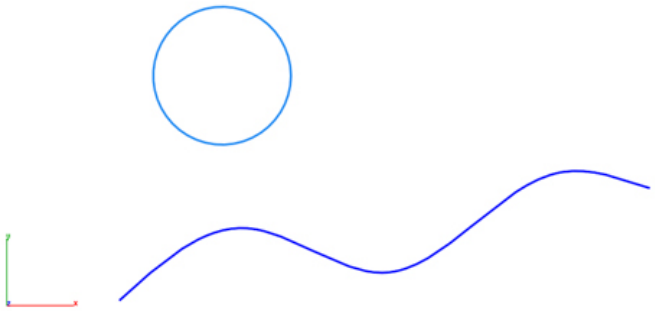


3. Line is created normal to surface.

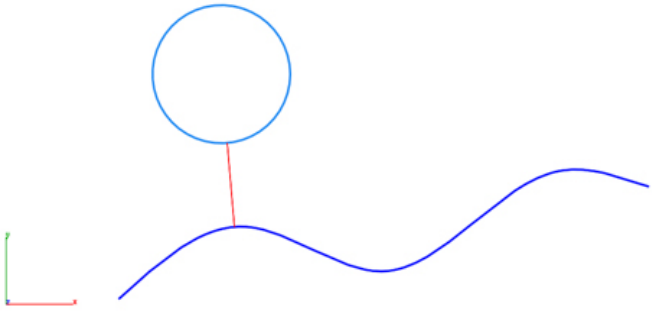


Line Perpendicular To Two Curves
This tool will calculate a line that is perpendicular between two curves.

Example:
1. Select the Line Perpendicular from Two Curves tool.
2. Select two curves that are in the same plane.



3. A line is created between the two curves.

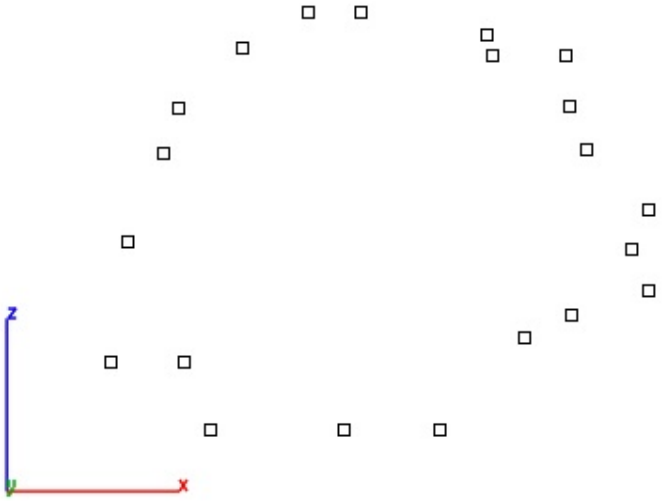


Best Fit Circle

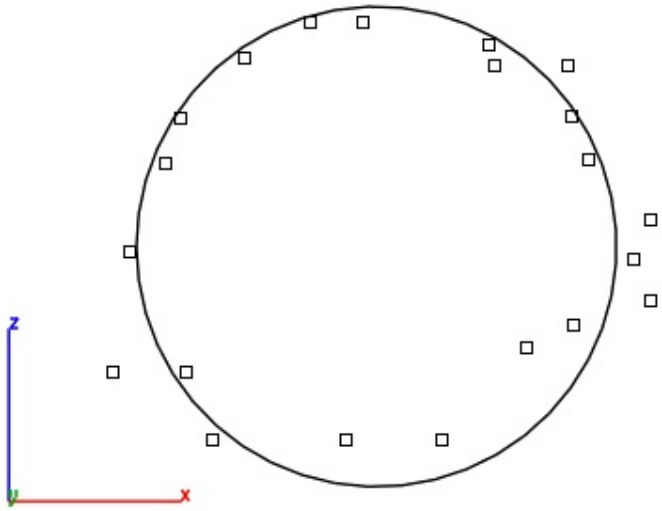
The Best Fit Circle command will calculate a 3D circle for the supplied points.

Example:

1. Select the Best Fit Circle command.
2. Specify locations for the Best Fit Circle.



3. A circle object is created for the given data.

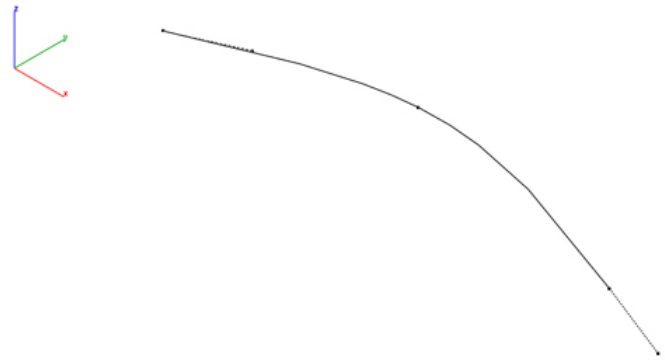


Circle Normal To Curve

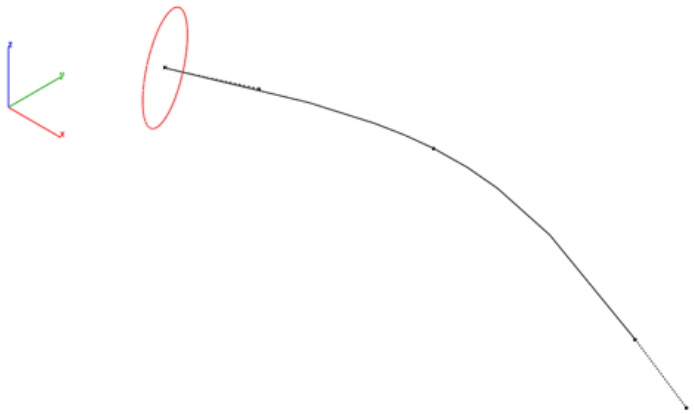
This tool constructs a circle normal to a curve at a specified location.

Example:

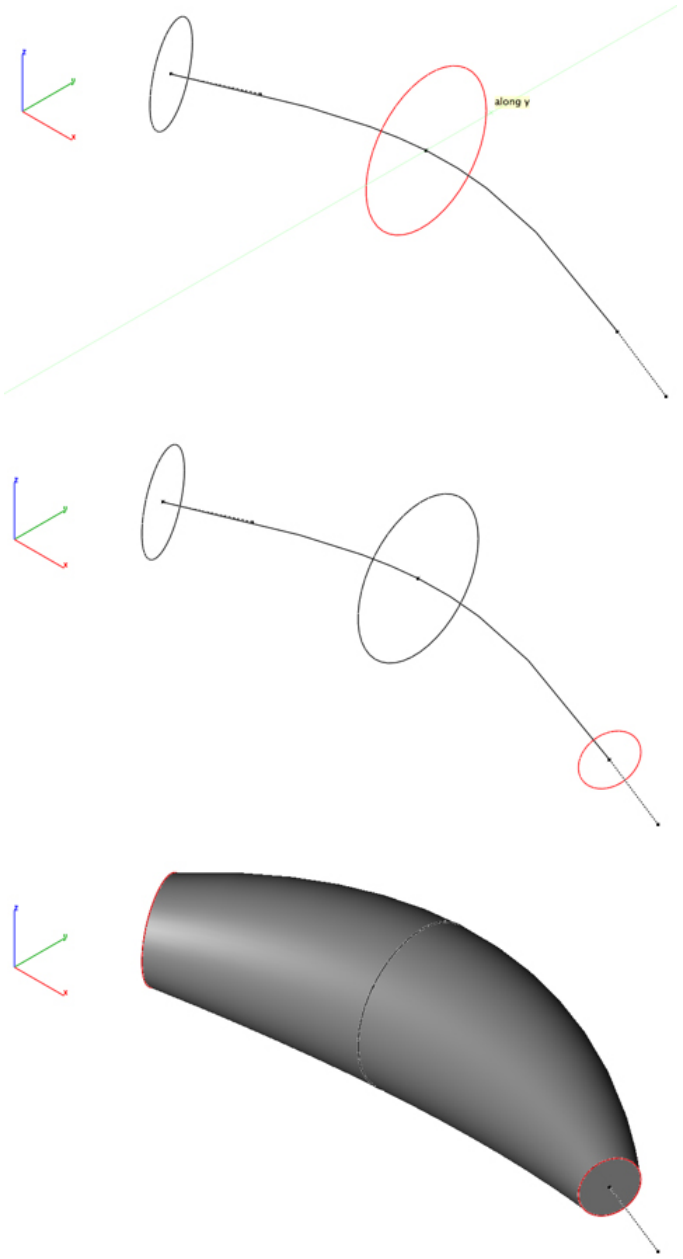
1. Select the Circle Normal to Curve tool.
2. Specify a location for circle center along the curve.



3. Move cursor to set the circle diameter. Use data entry window to set precise diameter.

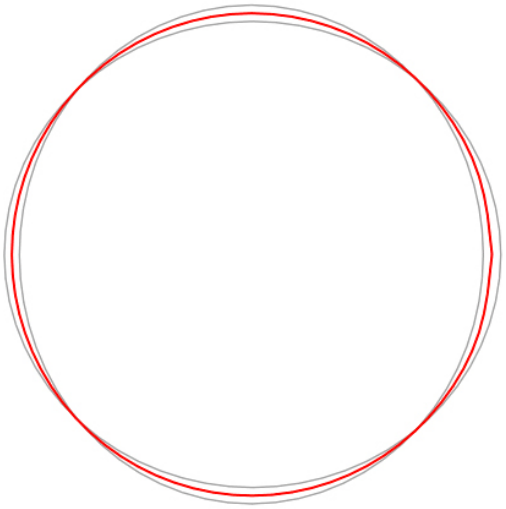


4. As an example use, the circles can be used as a profile of skinned solid.



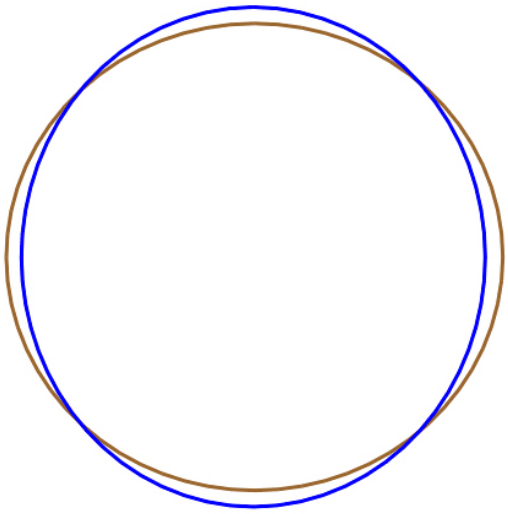
Average Two Curves

The Average Two Curves command takes as input two curves and averages the closest between to create a third curve.

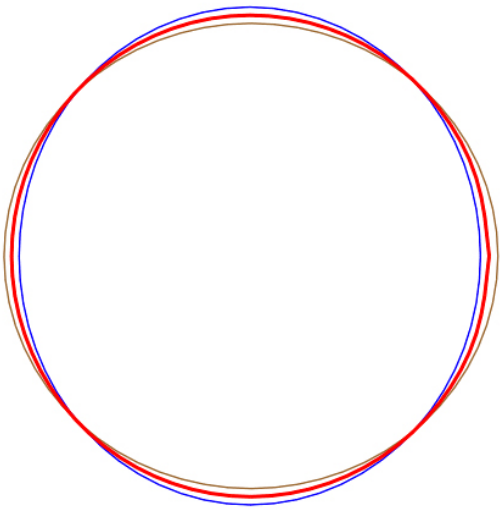


Example:

1. Select the Average Two Curves command.
2. Select two curves.

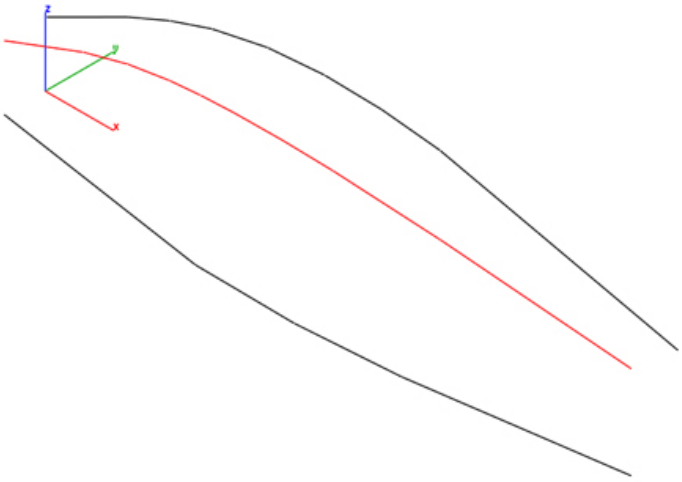


3. A curve is created that is the average to the two selected curves. In the image below, the red curve is the average of the blue and brown input curves.



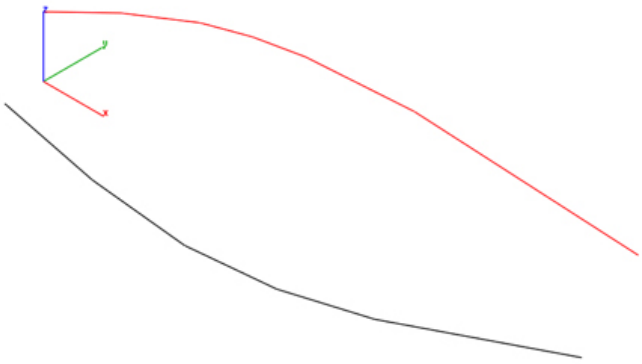
Curve From Two Views

The Curve Normal to Two Views tool projects creates a curve by the intersection of two projected curves.

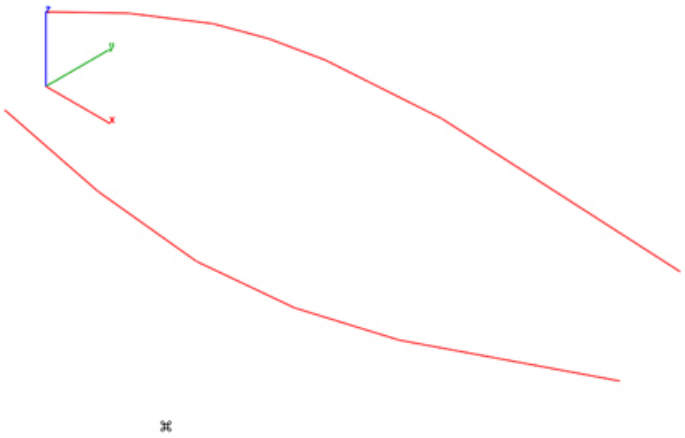


Example:

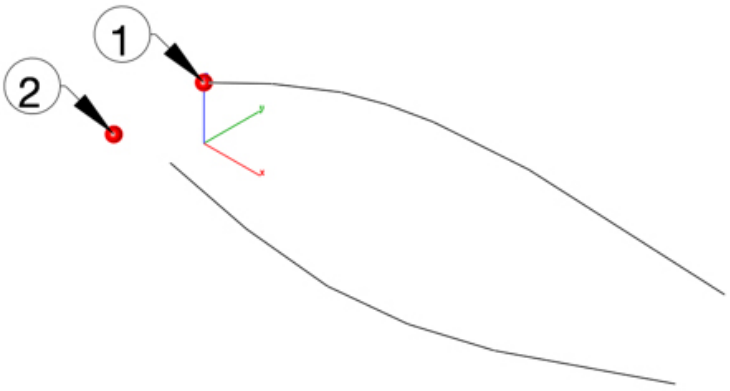
1. Select the Curve Normal to Two Views.
2. Select the first curve for first projection.

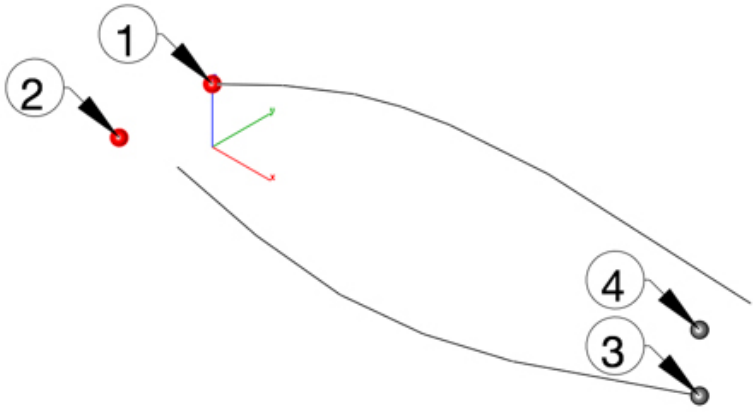


3. Select the second curve for second projection.

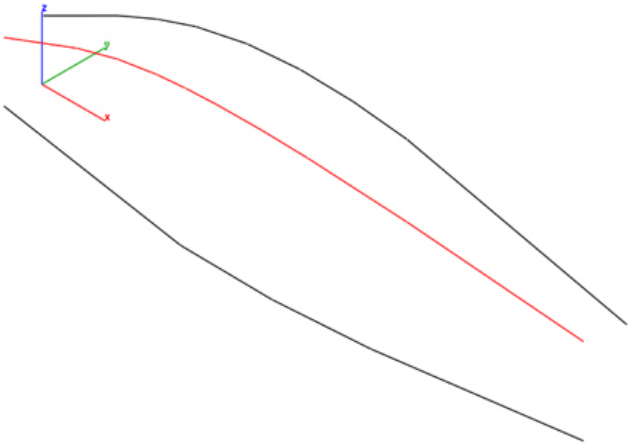


4. Specify start point for first projection direction.
5. Specify end point for first projection direction.





6. Specify start point for second projection direction.
7. Specify end point for second projection direction.
8. A spline is created at the intersection of the two projected curves.

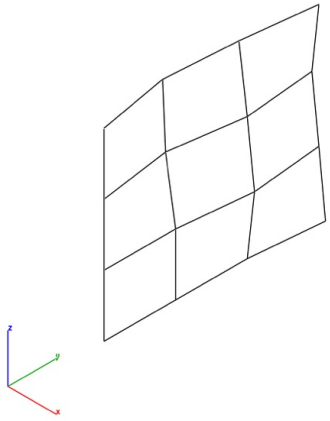


Best Fit Plane

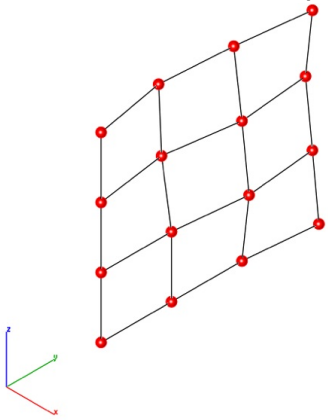
The Best Fit Plane tool calculates an infinite plane object from a collection of supplied 3D points.

Example:

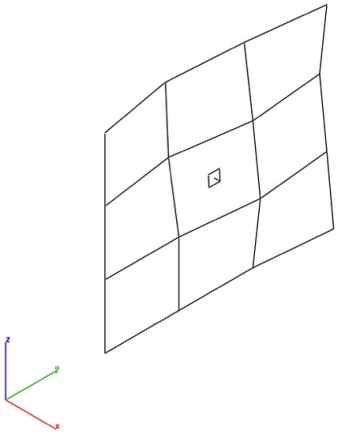
1. Select the Best Fit Plane tool.



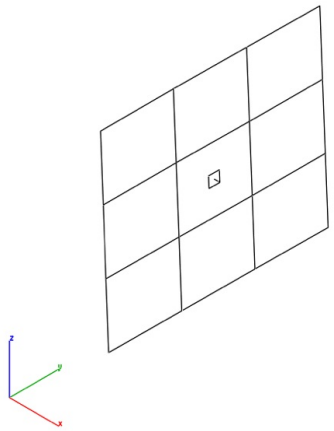
2. Pick 3D points for best plane fit. Seen as red spheres for this example.



3. Infinite plane object is created from the data.



4. An example use is shown below where the plane is used to project the curves into a common plane.



Surface Grid

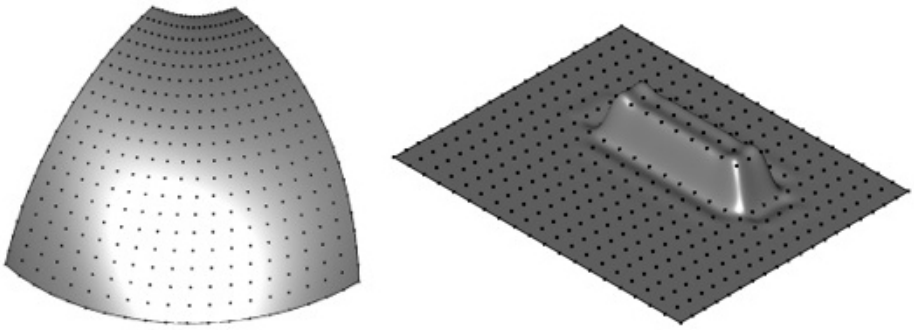
The Surface Grid creates a NURB surface from either a reference surface or by specifying two diagonals.

Data Entry Values:

#U Number of grid points along the u direction

#V Number of grid points along the v direction

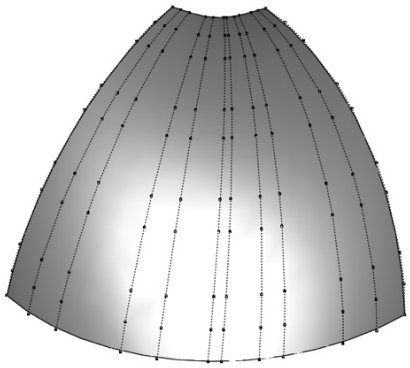
Tolerance The tolerance between the grid points and resulting surface.



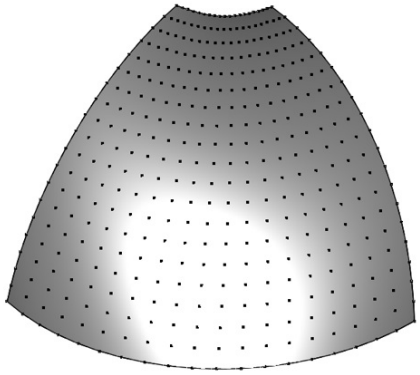
Example 1: Grid By Reference Surface

1. Select the Surface Grid command.
2. Select Reference Surface from the pull down menu.
3. Specify the grid settings.

4. Select a reference face or surface.



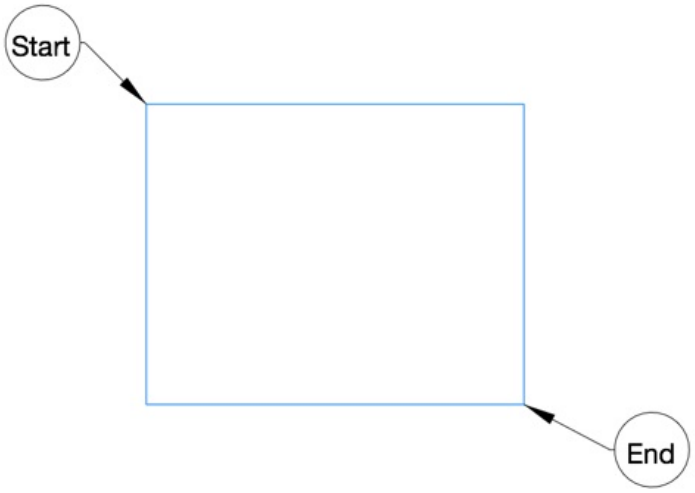
5. Select a reference face or surface. A new surface is created with the specified grid settings.



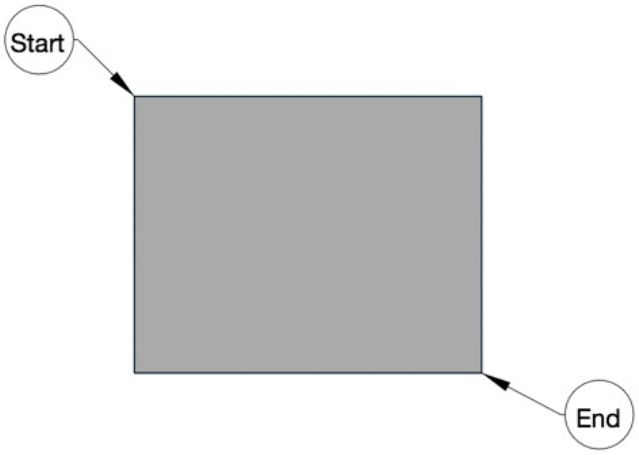
Example 2: Grid By Diagonals

1. Select the Surface Grid command.
2. Select Reference Surface from the pull down menu.
3. Specify the grid settings.

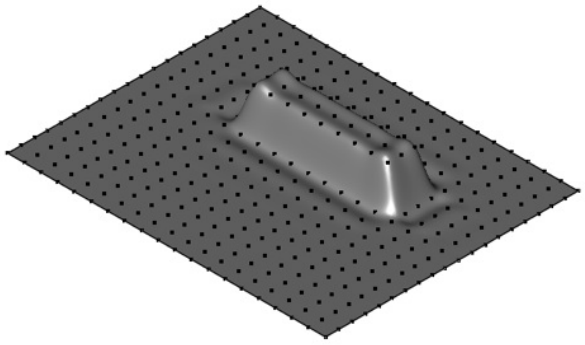
4. Select start diagonal.



5. Select end diagonal.
6. A surface grid is created between the diagonals.



7. Display and edit the grid points by turning on Edit: Show Points.



Shadow Projections

The Shadow Projection command projects a solid outline into a plane. Select xy, yz, xz, or WorkPlane as the plane to project the outline into.

Modify

The Modify Tools menu provides commands for editing or repairing geometry such as curves, surfaces, or solids.

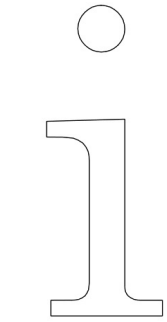
- Repair Profile
- Simplify Profile
- Create Outline
- Remove Duplicate Entities
- Remove Zero Length Curves
- Convert To Spline Surface
- Simplify Solid
- Part Repair

Repair Profile
The Repair Profile command examines and repairs the following common issues with a closed collection of curves.

Small Overlaps
Small Gaps
Non-Planar curves

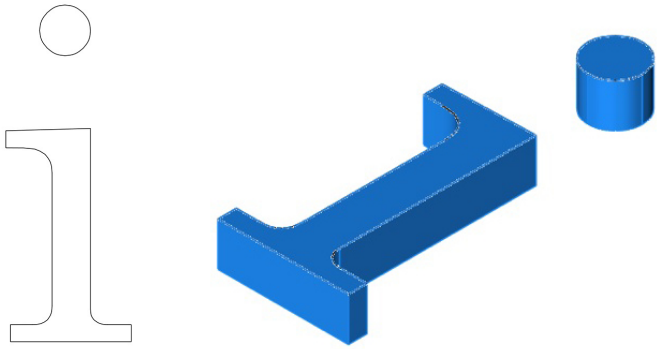
Example 1:

1. Select the Repair Profile command.
2. Select the curves in a profile to repair.



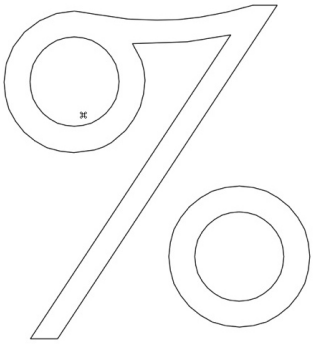
overlap

3. Overlap issue detected and repaired. Profile is then suitable for profile based operations such as extrude into a solid.



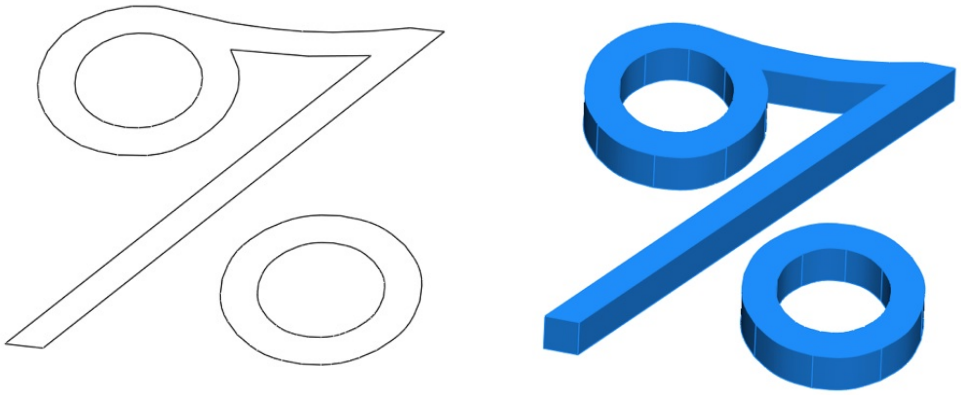
Example 2:

1. Select the Repair Profile command
2. Select the curves in a profile to repair



gap

3. Gap issue detected and repaired. Profile is then suitable for profile based operations such as extrude into a solid.

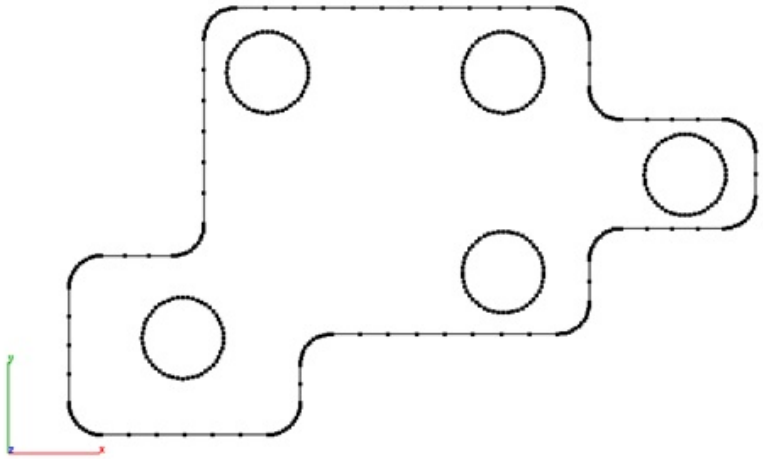


Simplify Profile

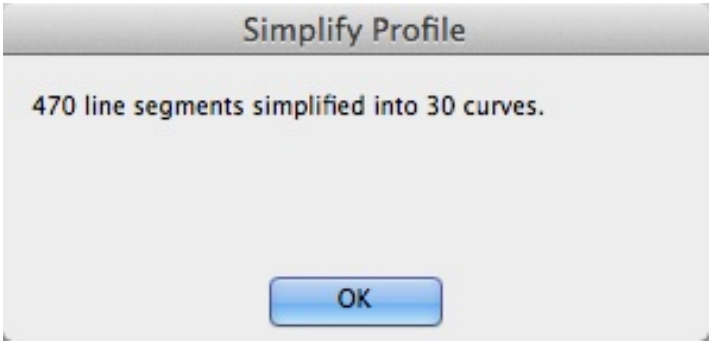
The Simplify Profile command converts lines/polylines into simplified line, arcs, circles, and splines.

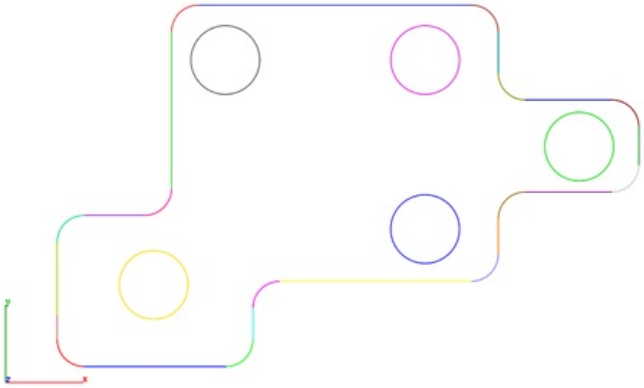
Example 1:

1. Select the lines in the profile to simplify.



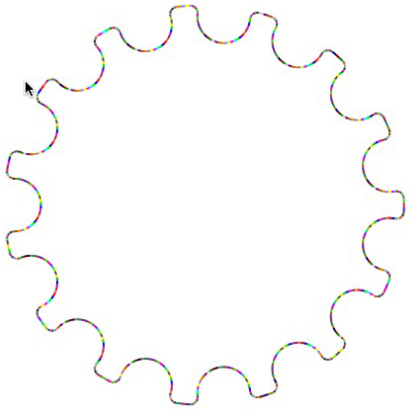
2. Dialog is displayed showing results. In this example, 470 lines are converted into 17 arcs and 13 lines.



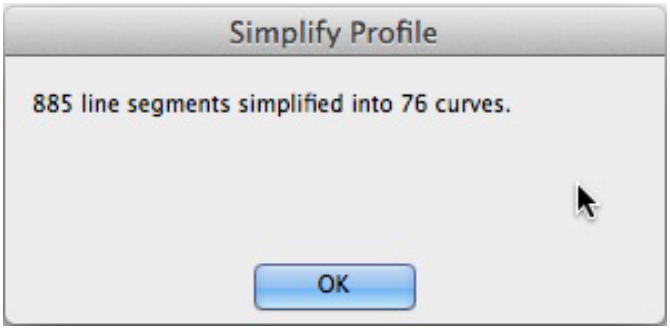


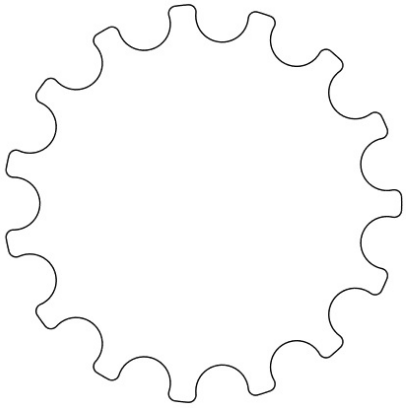
Example 2:

1. Select the lines in the profile to simplify.



2. Dialog is displayed showing results. In this example, 885 lines are converted into 45 arcs and 31 lines.



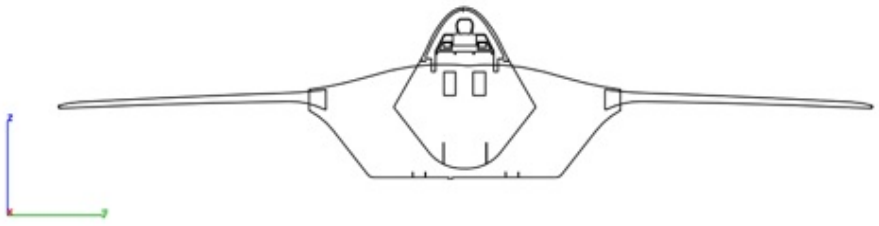


Create Outline

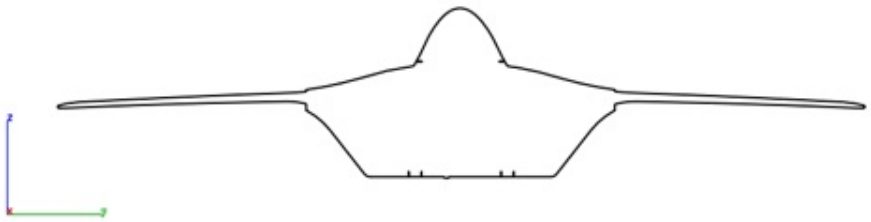
The Create Outlines tool takes a group of selected curves and finds the outline relative to the current WorkPlane.

Example:

1. Select the Create Outlines command.
2. Set the WorkPlane for the outline.
3. Select the curves for the outline command.



4. Curves are trimmed or removed from the original selection set to correspond to the profile outline.

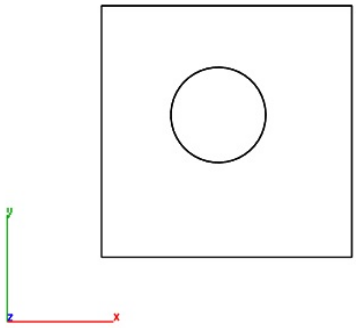


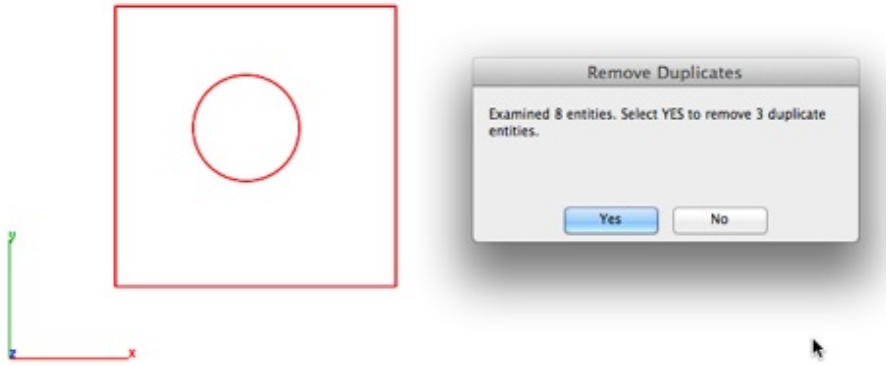
Remove Duplicates

The Remove Duplicates command examines curves and solids for duplicate representations.

Example:

1. Select the Remove Duplicates command.





2. Select the objects to check for duplicates.

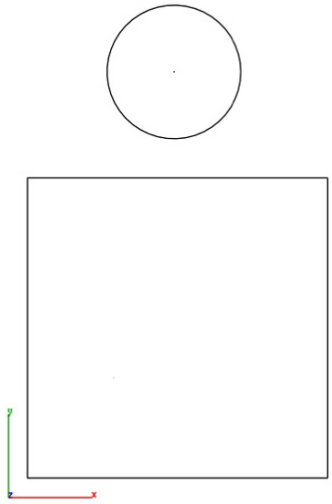
3. Select Yes to remove the duplicates. Selecting No leaves the duplicates within the drawing file.

Remove Zero Length Curves

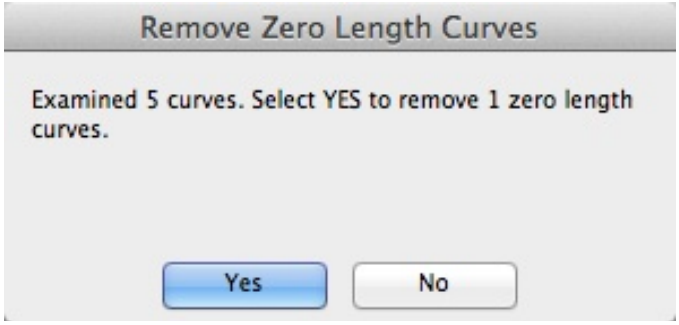
The Remove Zero Length Curves commands deletes curves below a specified length and radius.

Example:

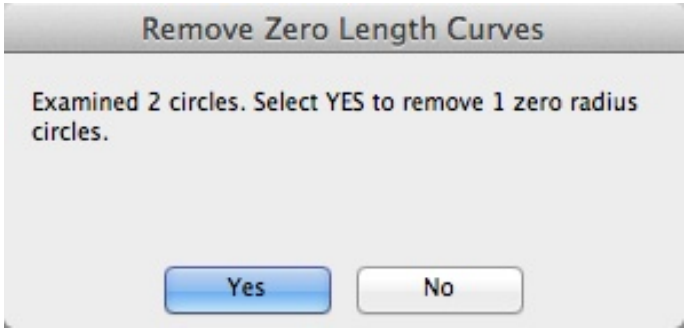
1. Select the Remove Zero Length Curves tool.
2. Set the tolerance value for smallest length and radius.
3. Select the curves to check for lengths and radius.



4. Select Yes to remove small length curves.



5. Select Yes to remove small radius curves.

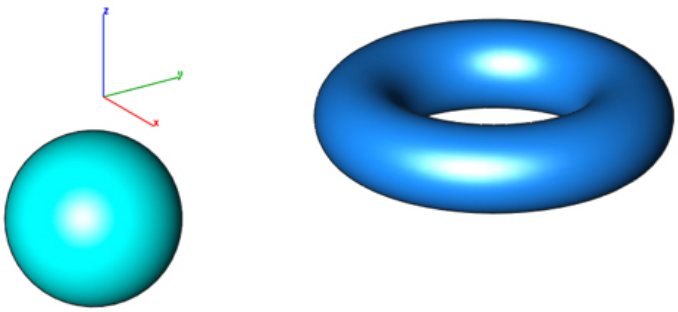


Convert to Spline Surface

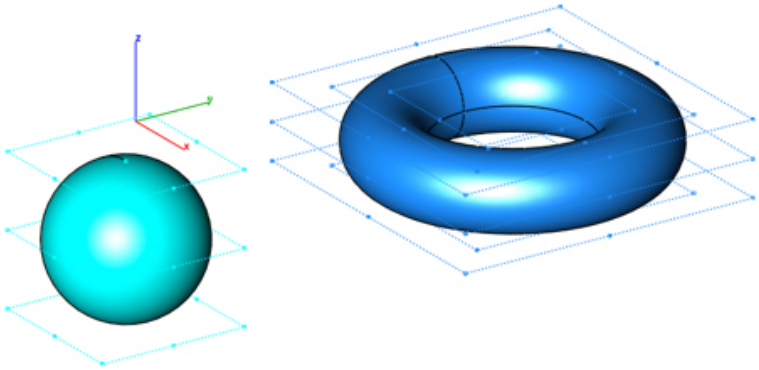
The Convert to Spline Surface tool changes analytic faces to NURBs providing access to control vertices.

Example:

1. Select the Convert To Spline Surface.
2. Select the objects to convert.



3. Illustration with converted spline surfaces and Show Points enabled.



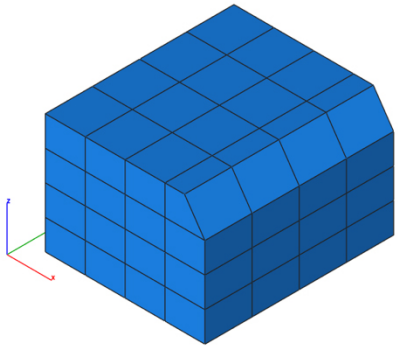
Simplify Solid

The Simplify Solid tool converts NURB into analytic faces. The following operations are performed as part of simplification.

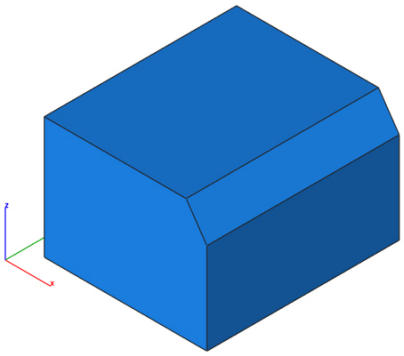
Replacement of small sliver faces
Merge similar neighboring faces
Convert NURB faces into planar, cylinder, and cone.
Clean the body of unnecessary edges

Example:

1. Select the Simplify Solid tool.



2. Select the part to simplify.



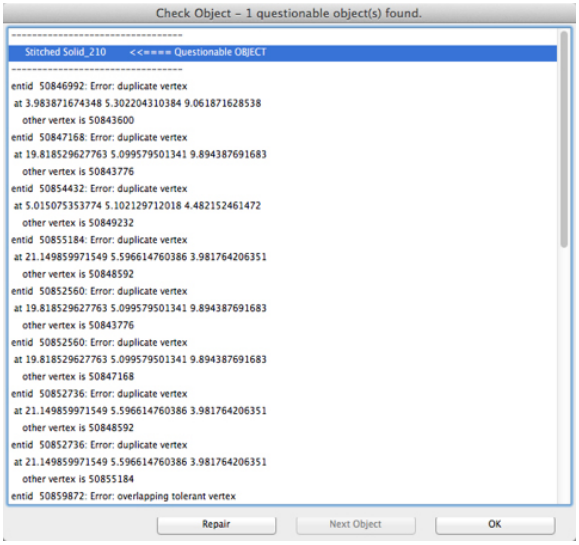
3. Results from simplify.

Part Repair

Attempts to repair a part using alternate stitching, repair, and translator technology.

Example:

1. Select the surface or solid to repair.



2. Select the Part Repair tool. Part is repaired if possible.
3. Select the surface or solid to repair. Repaired part is created as separate object from the original. Verify Check Object confirms part is now repaired.

Verify

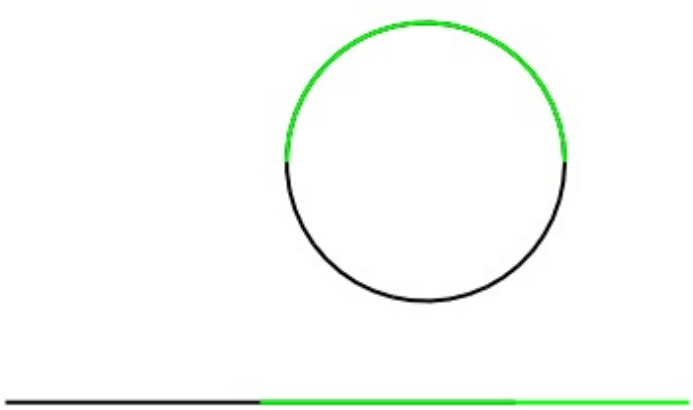
The Verify menu provides additional tools for validating and isolating common geometry issues for curves, surfaces, and solids.

Verify

- Show Overlaps
- Show Gaps
- Locate Inflections
- Check Curve Tangency
- Locate Minimum Curvature
- Show Surface Free Edges
- Show Non-Manifold Edges
- Identify Suspect Edges

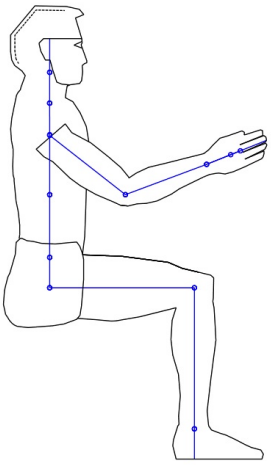
Show Overlaps

The Show Overlaps command examines selected curves for overlaps. Options are provided to remove the overlaps and to show the overlaps in a thickened red color. Illustration below shows an arc overlapping a circle and a line overlapping another line.

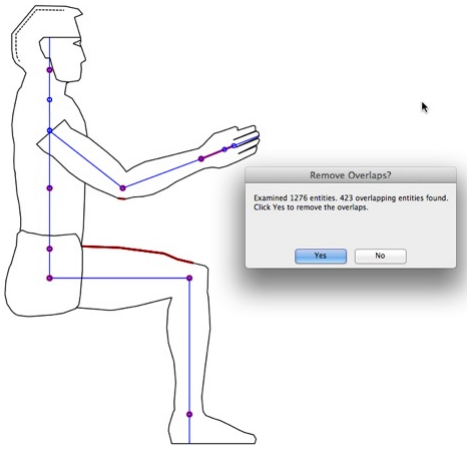


Example:

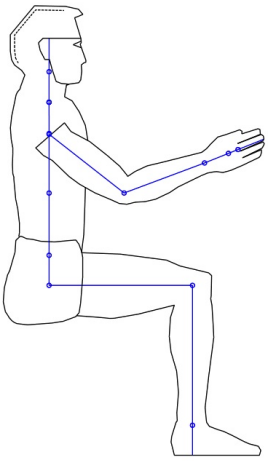
1. Select the Show Overlaps tool.



2. Select the curves to be examined. A dialog box reports the number of overlaps identified. Clicking YES will delete the overlaps. Clicking No will highlight overlaps in a thickened red color.



3. Click Yes to remove the overlaps.

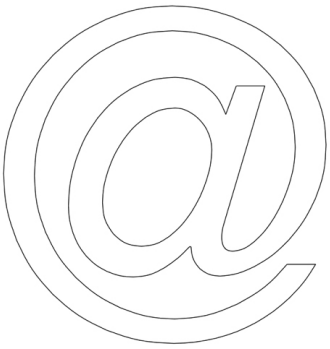


Show Gaps

The Show Gaps tool identifies open regions in a curve profile.

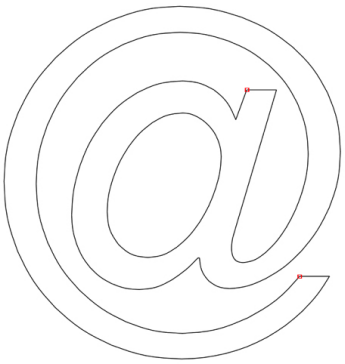
Example:

1. Select the Show Gaps command.
2. Select the curves to examine.



3. A dialog box is displayed reporting the results.

4. Gaps locations are marked with point objects.



Locate Inflections

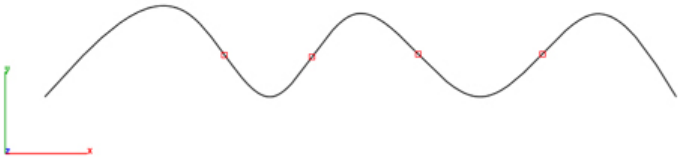
The Locate Inflection tool identifies positions on curves where the second derivative equals zero. This is the location where the sign of the curvature changes. Often inflections points are used as indicators of curve smoothness.

Note: Lines, arcs, ellipses, and conics do not have inflection points.

Example:

1. Select a spline to calculate inflection points.

2. Inflections are calculated and added as point objects to the file.

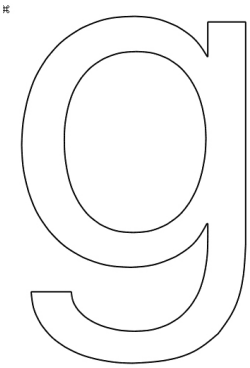


Check Curve Tangency

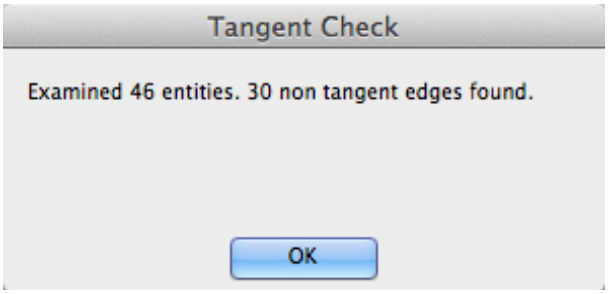
The Check Curve Tangency examines a collection of curves checking on the tangency at the connection location.

Example:

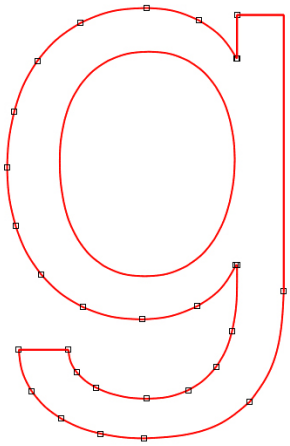
1. Select the Check Curve Tangency command.
2. Select two or more curves to examine tangency.



3. A dialog box is displayed reporting possible issues.



4. Point objects are created at those locations where tangency issues exist.



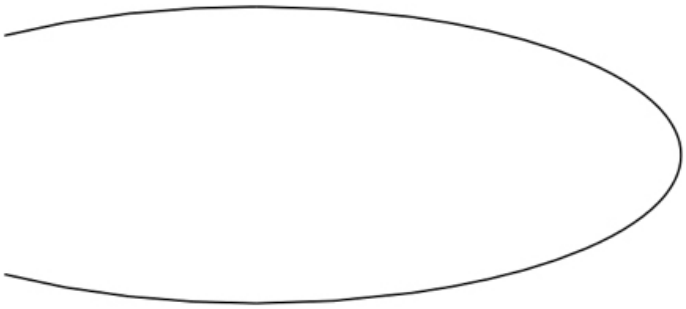
5. Use Edit: Show Points to show tangency handles to manually correction issues.

Locate Minimum Curvature

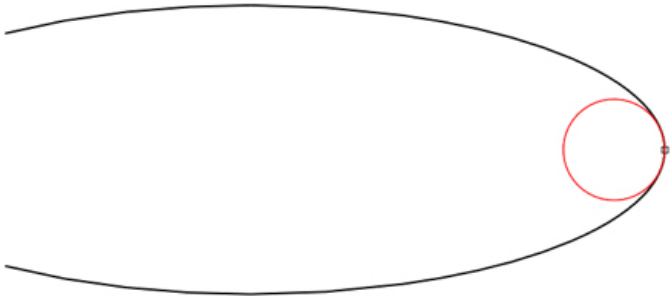
The Locate Minimum Curvature tool identifies the minimum curvature location on a curve. The location is marked with the creation of a circle.

Example:

1. Select a curve to calculate the minimum curvature location.



2. The minimum curvature location is marked with a circle.

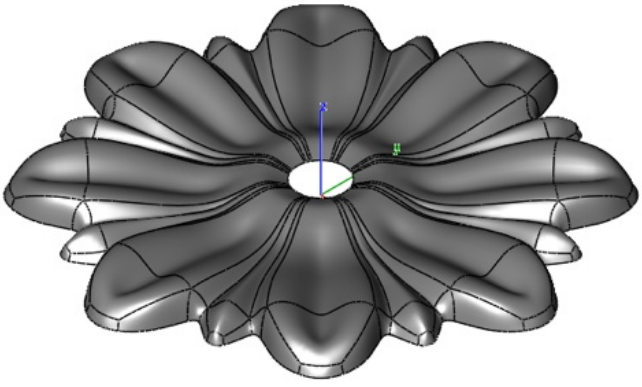


Show Free Edges

The Show Free Edges command creates new curves at surface edges that have only one face attached. The new curves are highlighted in red. Use Undo to remove the curves.

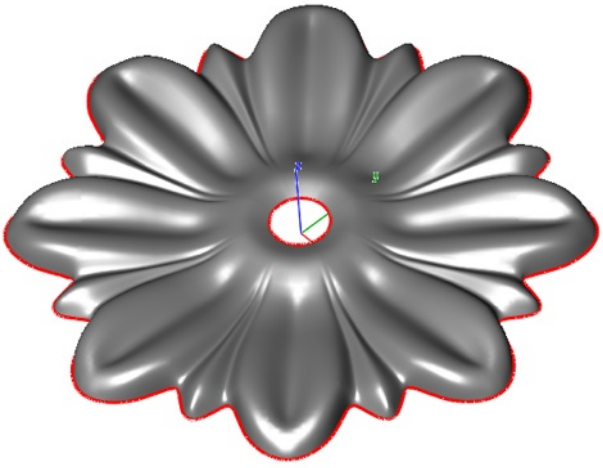
Example 1:

1. Select the Show Free Edges command.



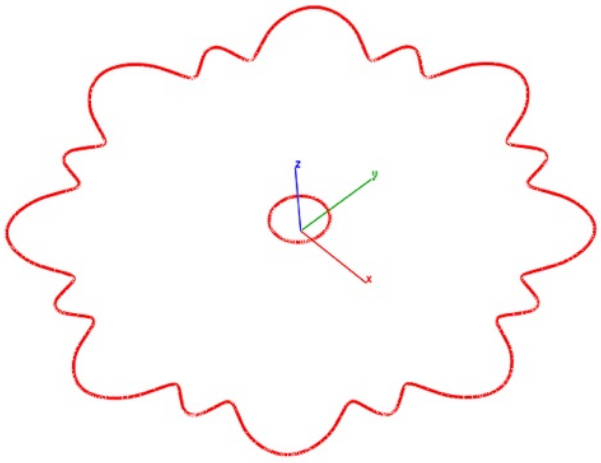
2. Edges are recreated as curves highlighted in red. A dialog box

indicates how many free edges were found.

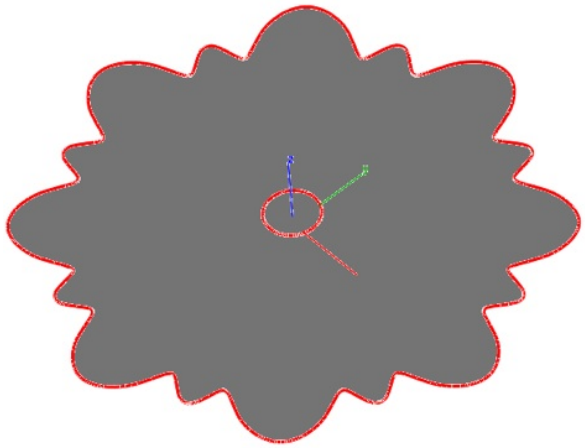


Example 2: Using Free Edges to Convert to Solid

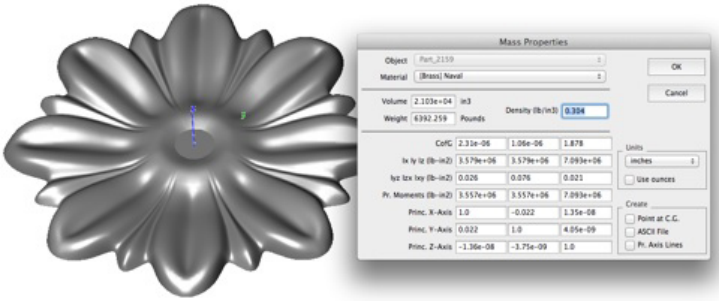
1. Hide the surface from the previous command, leaving just the free edges.

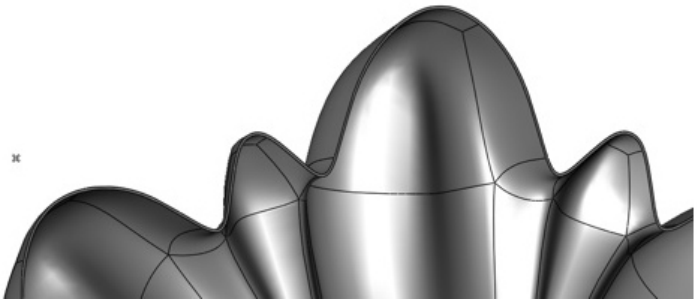


2. Select the Cover tool from the surface palette and select all the curves to create a planar surface.



3. Select the Stitch tool to combine the surfaces into a water tight volume suitable for mass property calculations or shelling modeling operations.



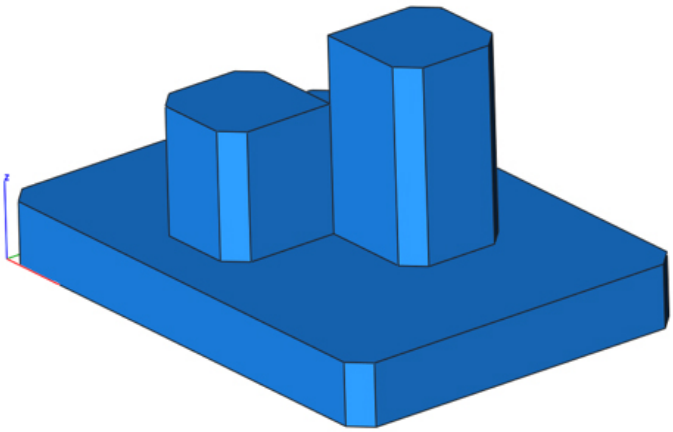


Show Non-Manifold Edge

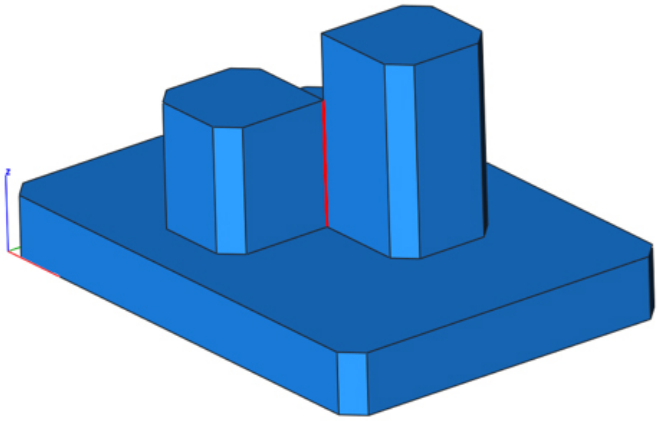
The Show Non-Manifold Edge tool identifiers edges in a solid part that have more than 2 shared faces.

Example:

1. Select the Show Non-Manifold Edge tool.
2. Select the solid part to examine.



3. Non-Manifold edges are displayed as red lines added to the drawing file.

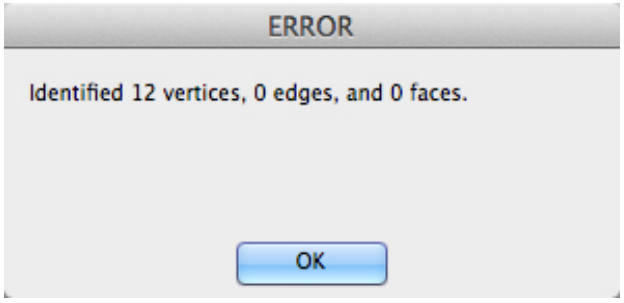


Identify Suspect Edges

Creates a point, edge, or face along a feature that appears to have either topology or geometry issues.

Example:

1. Select the Identify Suspect Edges tool.
2. Select the solid part.



Feature Selections

The Feature Selections tool palette provides commands for selecting objects by more specific attributes. For meshes, this includes selecting by vertex, edge or face and for solids features such as blend, protrusion, or face networks.

Feature Selections

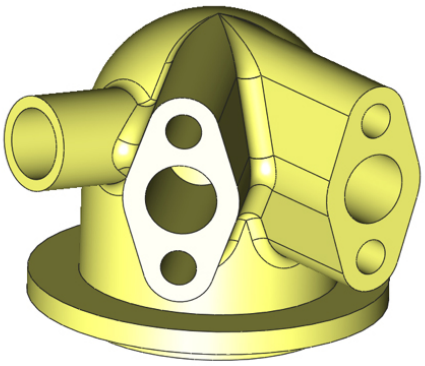
- Select Blend Network
- Select Protrusion Network
- Select Depression Network

Select Blend Network

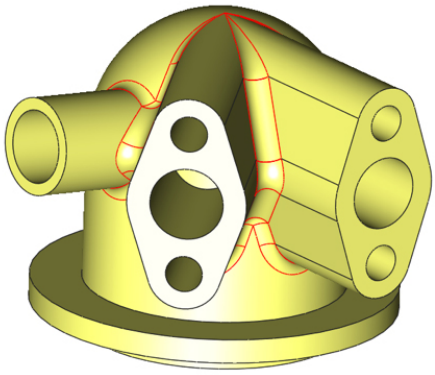
The Select Blend Network tool is used to remove blends from a part which is especially useful when the part does not have history.

Example:

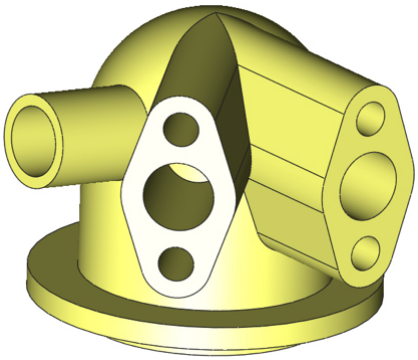
1. Select one face that is part of a blend network of faces.



2. All blends connected to the selected blend are highlighted as selected.



3. To remove all selected blend faces, select the Edit: Cut tool.

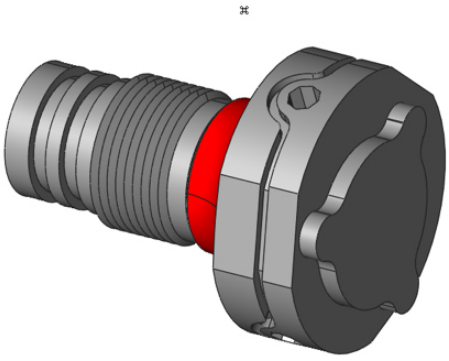


Select Protrusion Network

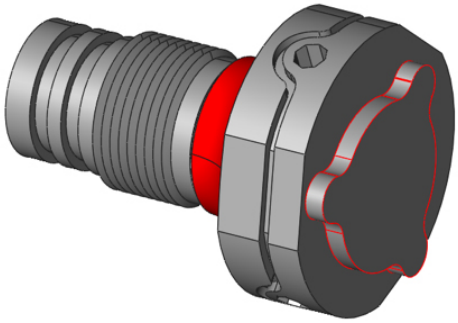
The Select Protrusion Network helps you quickly select all faces associated with a protrusion within a solid part.

Example:

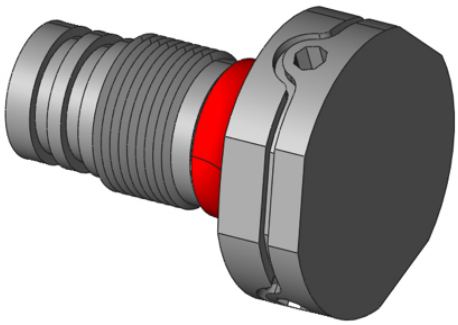
1. Pick the Select Protrusion Network command.



2. Select one face of the protrusion face network. All associated faces will be added to the selection.



3. Use Edit: Cut to delete the selected faces.

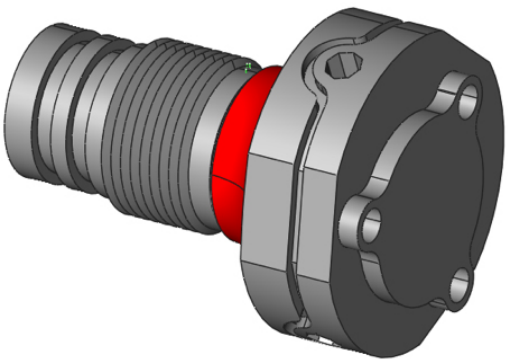


Select Depression Feature

The Select Depression Feature helps you quickly select all faces associated with a depression within a solid part.

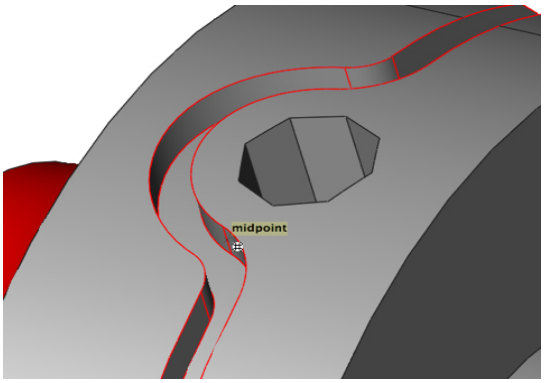
Example:

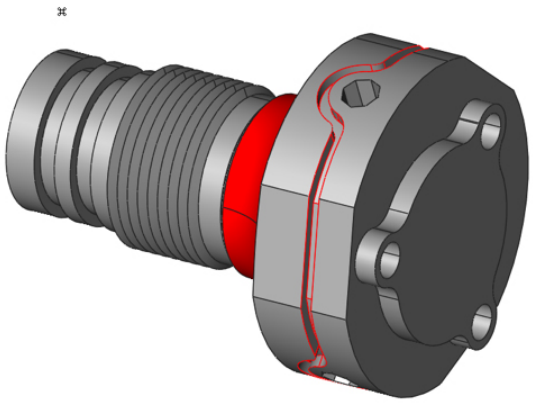
1. Pick the Select Depression Feature tool.



2. Select any face that is part of the depression network of faces. All

associated faces will be added to the selection.





3. Select the Edit: Cut command to remove all selected faces.