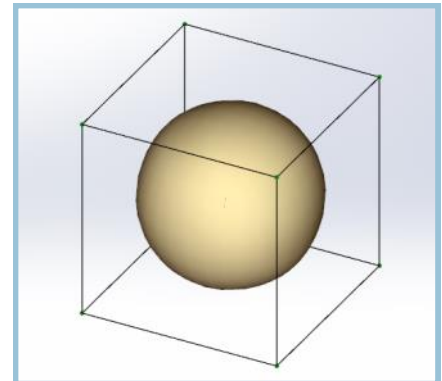


What is a Power Surface?

It is like freeform NURBS control polygon with some of the restrictions typically associated with NURBS removed. In the graphics industry it is also known as a Subdivision Surface object or SubD for short. The guys at Pixar originally developed this technology for modeling organic forms and you can see it used in all of its animated films.

Another way to think about a SubD is that it is like a soap bubble where the edges of the control polygon pull on the soap bubble. If you put two edges close together they exert more of a pulling effect and cause the soap bubble to be more curved. You can also increase the curvature of the soap bubble by increasing the weighting on an edge and pulling the soap bubble closer to that edge. If you increase the weighting on the edge all of the way to 100% then you create a creased edge that is not smooth.

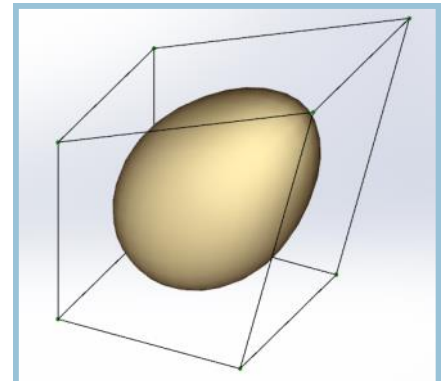


Soap Bubble With Box Control Cage

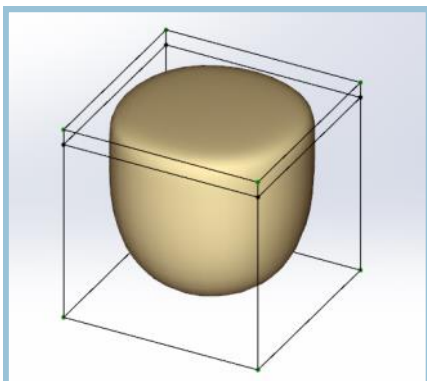
Creating SubDs

There are several ways to create SubD objects for use in SolidWorks. Power Surfacing provides several Primitive objects that can be used as starting points; you can use sketches to generate SubDs, or you can import mesh objects directly into the Power Surfacing editor.

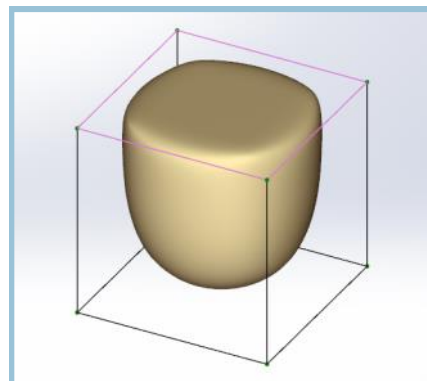
Once created, you can constrain SubD geometry to existing analytic Solidworks geometry. We have the ability to make the connection with Solidworks and maintain Curvature Continuity between the native Solidworks body and the Power Surfacing generated body. This capability makes Power Surfacing a powerful addition to your existing surface modeling workflows. You can construct objects with are a combination of Solidworks native surfaces and Power Surfacing generated surfaces.



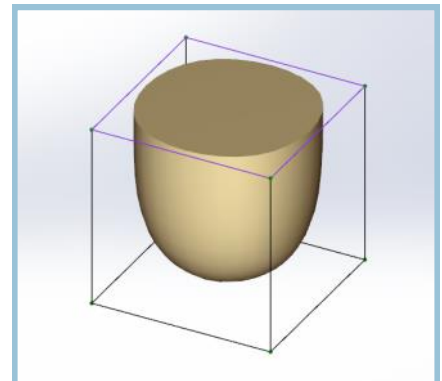
Moved Edge Changes Soap Bubble



Additional Edges More Curved



Weighted Edges Pull Harder



100% Weight Produces Creased Edges

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Power Surfacing RE Help

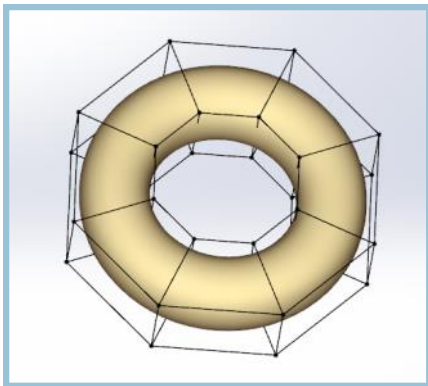
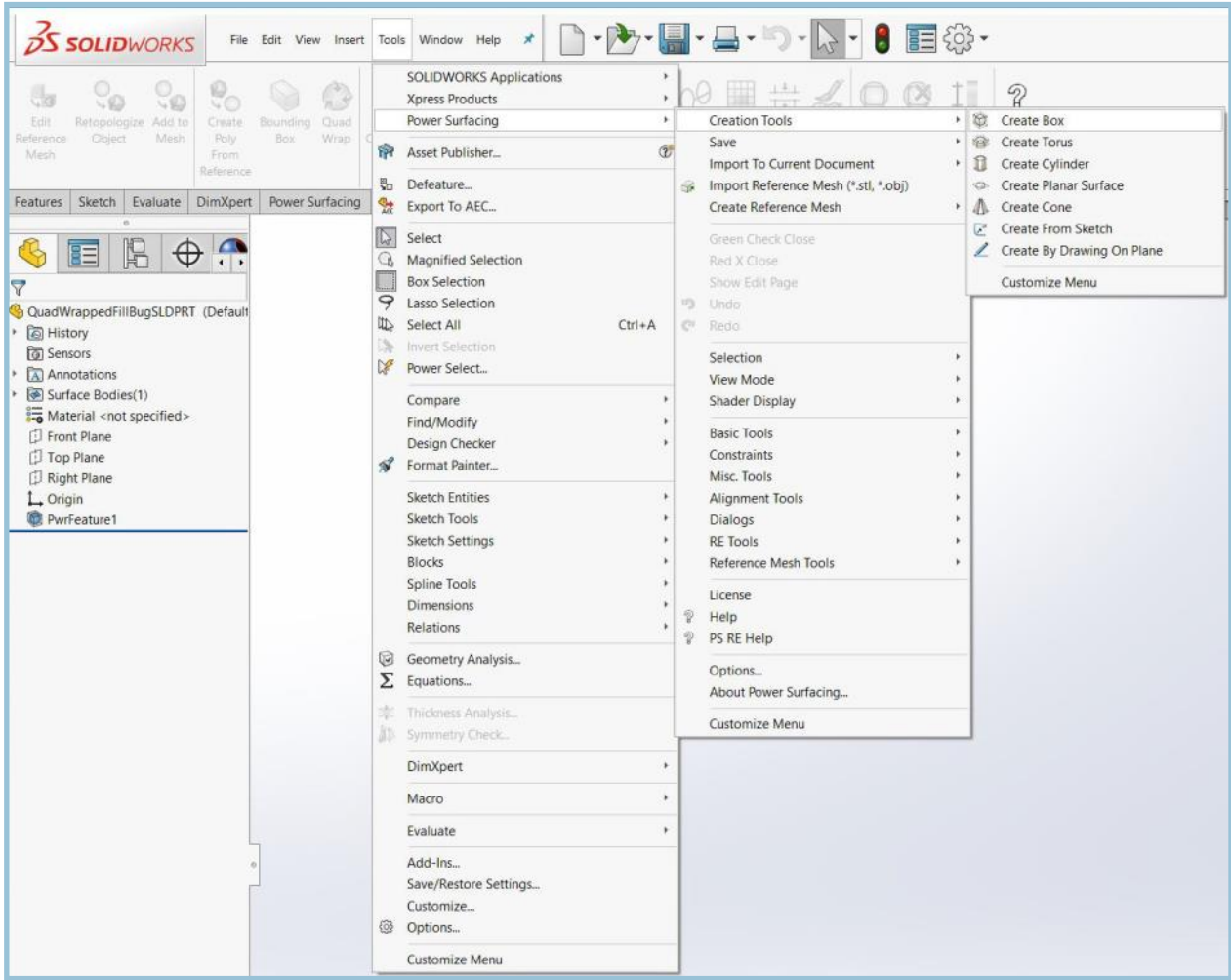
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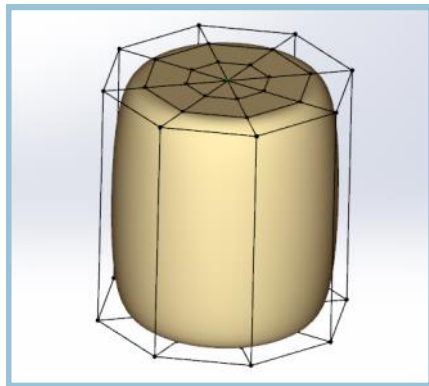
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From the Power Surfacing Menu

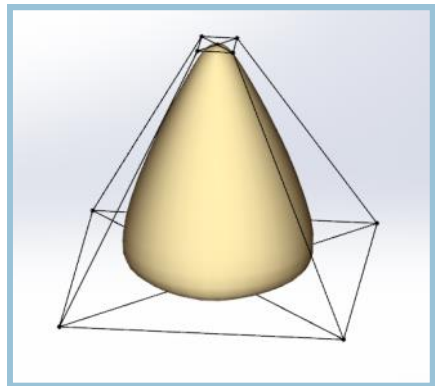
You can create both the Part file and the SubD model directly from the Power Surfacing menu when you select one of the Power Surfacing primitive objects. It is also possible to create most of the primitives while already in editing mode to add a primitive to the existing SubD.



Default Torus Shape



Cylinder Shape

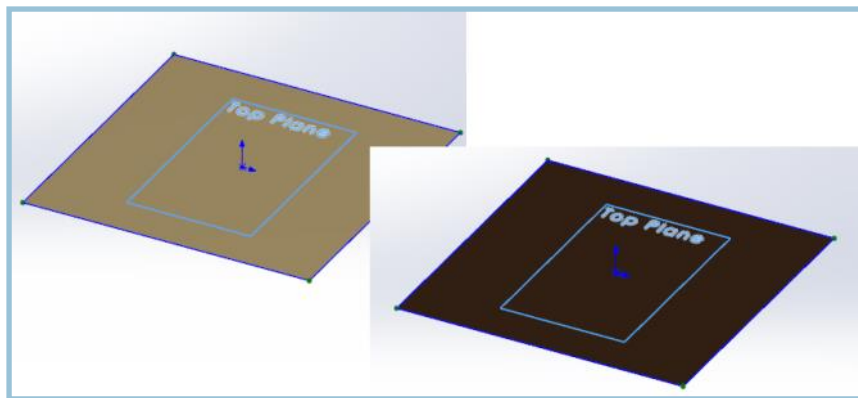
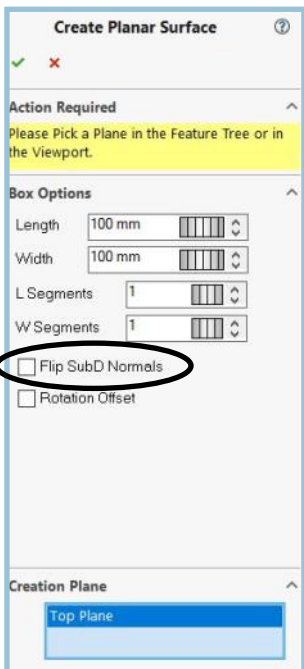
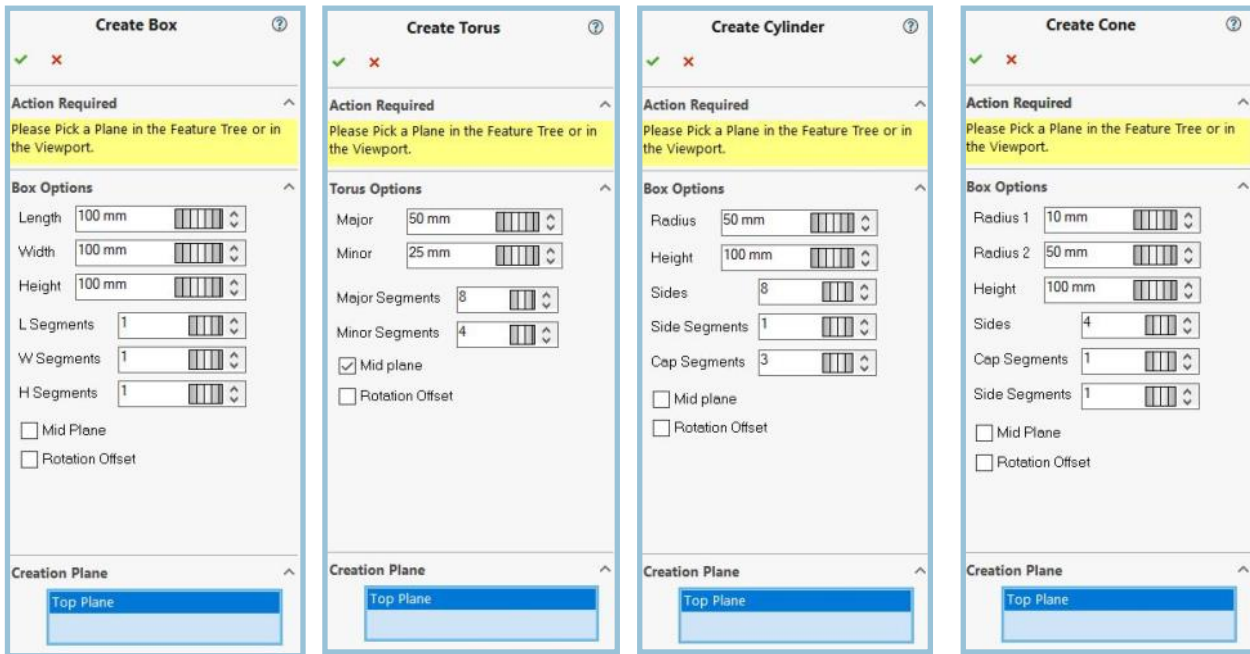


Default Cone Shape

Create Box, Torus, Cylinder, Planar Surface, Cone

Power Surfacing lets you choose between Box, Torus, Cylinder, Plane and Cone primitive objects for use as the starting Control Mesh. Each allows you to set its various parameters. You must click the Accept check mark to start editing the new SubD. You can cancel the creation by clicking the red X.

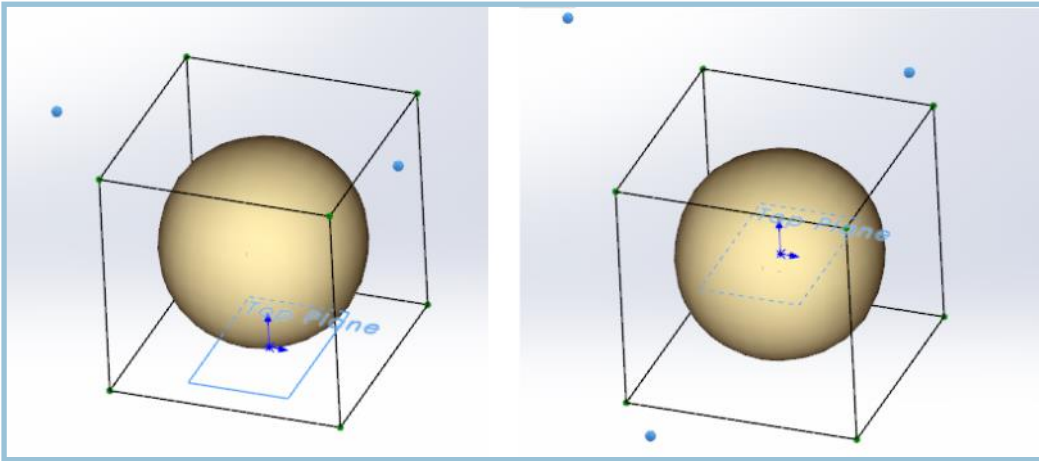
The Planar Surface is an open edged SubD object. If you are used to creating with surfaces rather than volumes you may wish to use this option to start your SubD. If you plan on extending the edges upward, you may want to use the Flip Normals option before creation.



Flip SubD Normals—Normal left image; flipped right image

Mid Plane

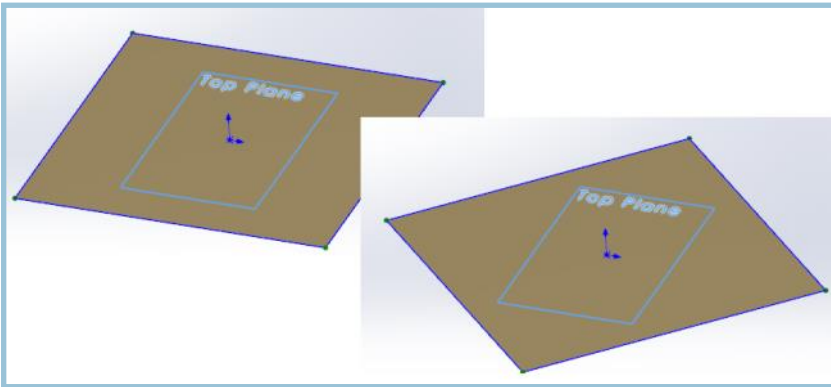
All Primitives have the option to be extruded from the Create Plane upward or, if checked, from Mid Plane.



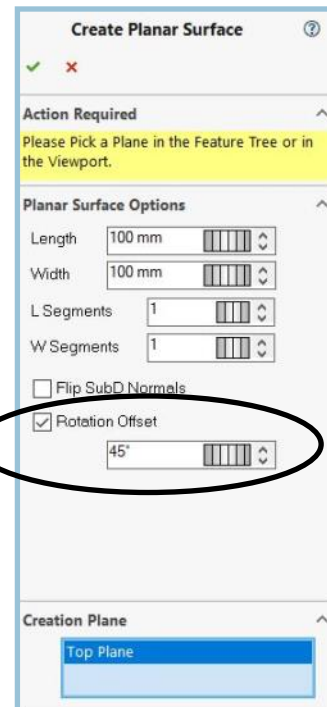
Extruded up from the plane (left image); and extruded mid-plane (right image)

Rotation Offset

Primitive objects can also be rotated to an offset during the create process. Checking Offset Rotation brings up a text box that allows you to type in the rotation in degrees.

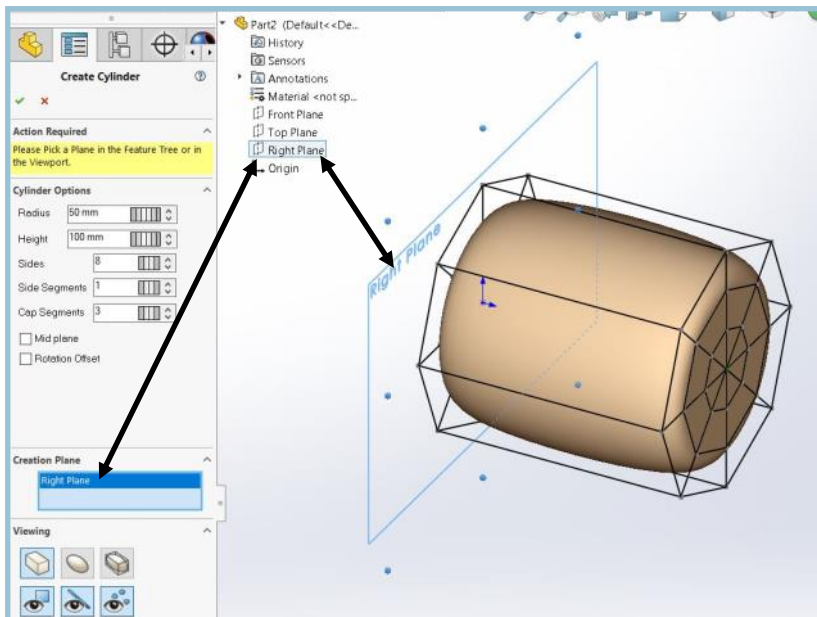


The left plane uses no rotation offset; the right plane is set to 45 degrees



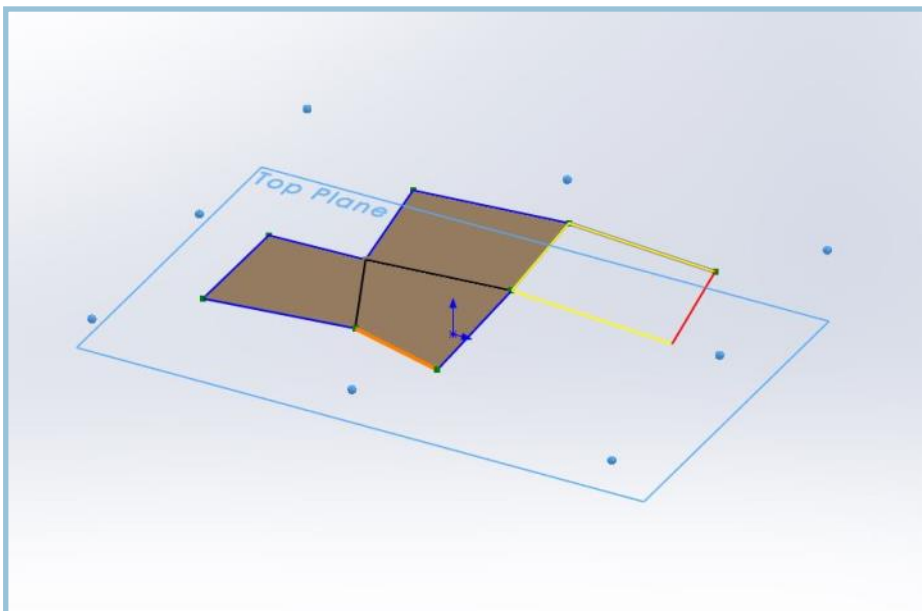
Creation Plane (all primitives)

In the Creation Plane section, you can set the creation plane and change the default orientation for the primitive by selecting it from the Feature Tree in the viewport. The selected plane will appear in both the selection list box and in the viewport.



Create By Drawing On Plane

You can start a model by drawing directly on a datum plane. As you draw, it will fill in quads automatically. This command, like other primitives, can also be used in an existing object to add faces to the model on a given plane. Note that the resulting faces are on the datum plane, but not constrained to the datum plane. Therefore points can be moved off the plane in subsequent editing operations.



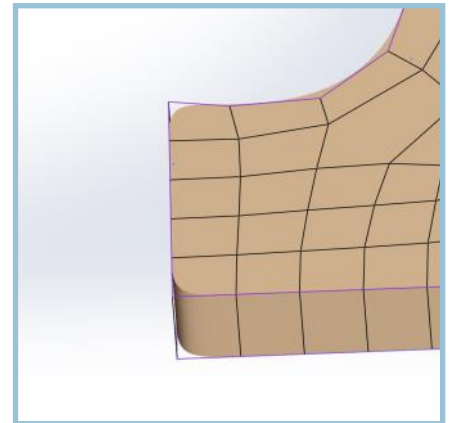
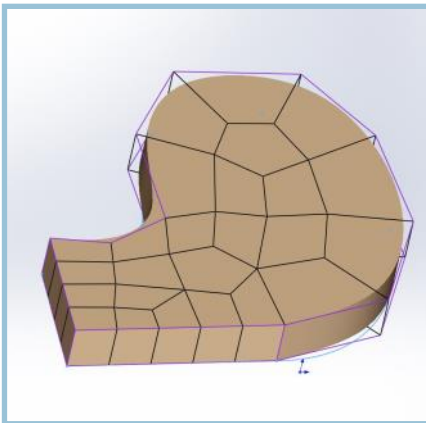
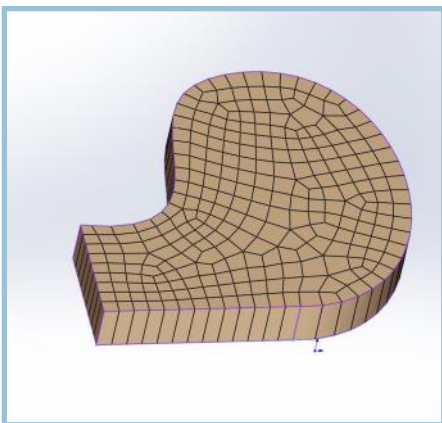
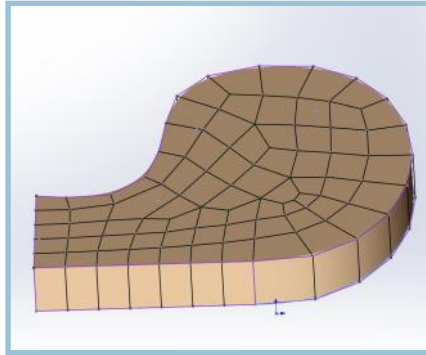
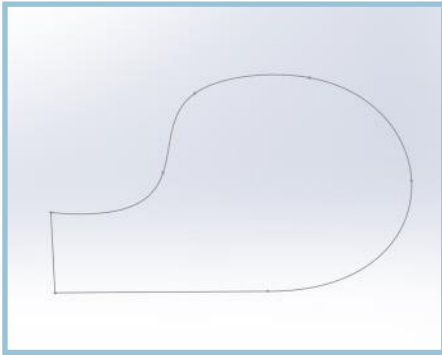
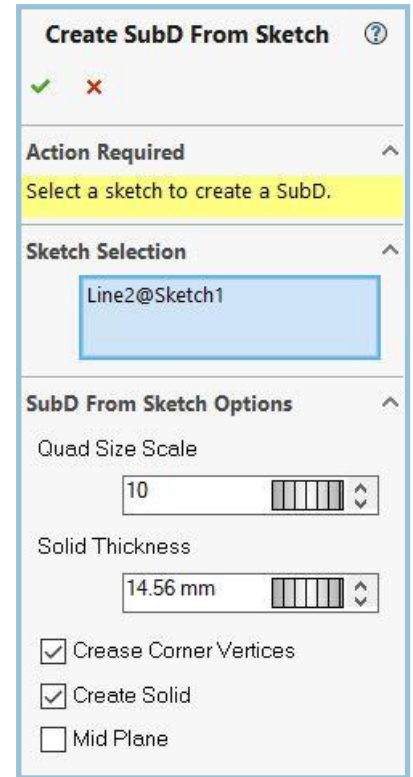
Create From Sketch

You may use sketches to create the SubD solid by selecting the *Create From Sketch* option from the creation dropdown. If the sketch was selected prior to selecting *Create From Sketch*, it will automatically be put into the *Sketch Selection* box. If it was not selected, selecting any part of the sketch is sufficient. If the sketch is not visible, you can select it from the Feature Tree in the viewport.

With a sketch selected, you will need to select either *Create Solid SubD From Sketch*, or *Create Planar SubD From Sketch*. With a Solid, you have the option to extrude the SubD up from the create plane or mid plane, just as with the primitives. Preliminary extrusion height is calculated using the average edge length, but is easy to adjust once in Power Surface edit mode.

The sketch will do its best to approximate the curves with the subdivided mesh.

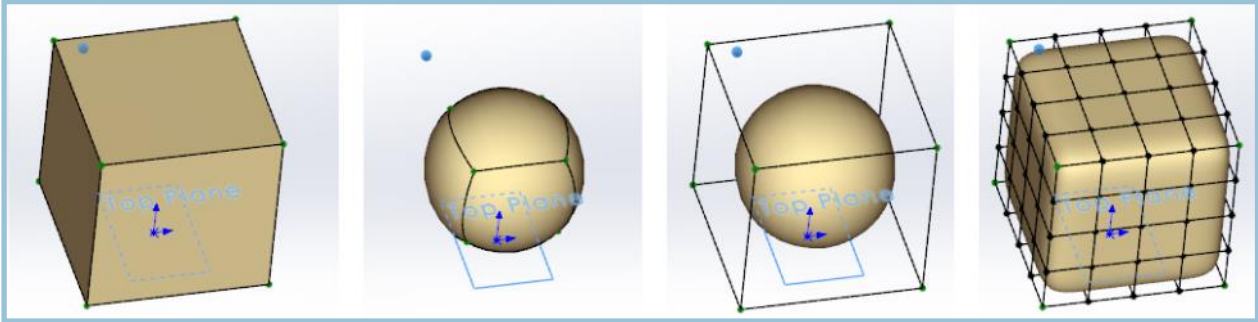
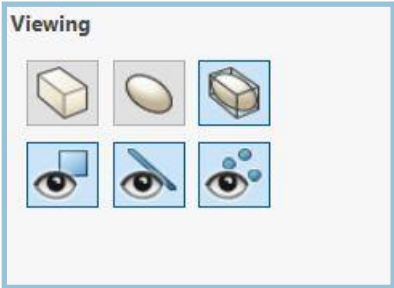
Note that the association with the sketch is not maintained and you are free to edit the resulting SubD model in any way you see fit.



Left Top: original sketch; Middle Top: Default solid with 10% Quad Size; Right Top: uncheck Crease Corner Vertices
Left Bottom: 4% Quad Size Scale; Right Bottom: 12% Quad Size Scale.

Viewing

While setting up the creation method for your SubD, you can also change the display mode. The top three icons change display from Display Control Polygon, Display the SubD, and SubD Mesh with the Control Edges and Vertices. The bottom three icons allow you to change the visibility of the Faces, Edges, and Vertices. Sometimes you may want to see the shape without the edges.



The images are from left to right: Control Mesh, SubD Mesh, SubD Mesh Faces plus Control Cage, Control Cage box with additional subdivisions.

Selection

Region Select

Power Surface follows the same conventions as SolidWorks for region select. Dragging from right to left selects in crossing mode, where sub-objects crossed by the marquee will be selected. In windowed mode, dragging from left to right, only the sub-objects fully within the marquee will be selected.

When using the selection marquee, back-facing objects will be selected along with those on the front.

General

Faces, edges and vertices are previewed in hover mode with orange highlight. When selected, or picked, the sub-object or sub-objects will turn cyan colored.

To add to the selection, you can hold down the ctrl key. To remove sub-objects from the selection, you can hold the shift key down.

Status Bar

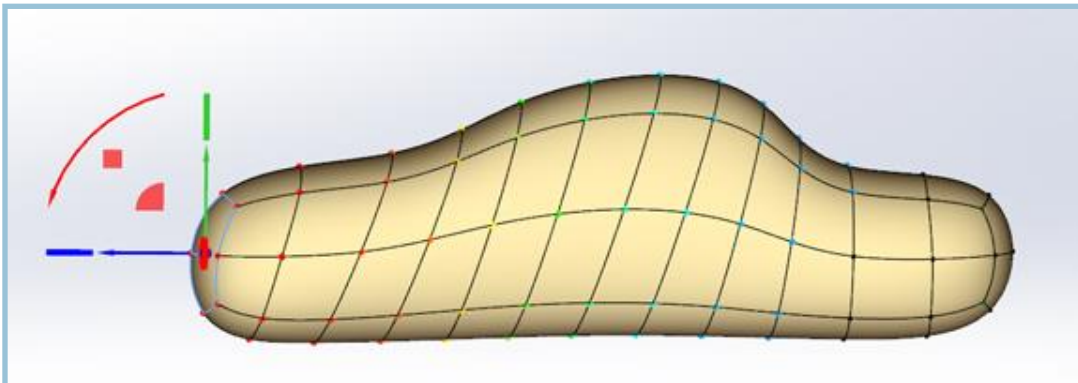
The number and type of currently selected vertices, edges or faces are shown on the status bar, lower left. Depending on the selection, it may also show location, length, weight, or other pertinent information.

Smart Selection (Double Clicking)

In Edge selection or Any selection mode, the position of the cursor over the edge will determine if an edge loop or edge ring will be selected. Double clicking the outer third of the edge will select a loop, while double clicking the center third will select a ring. With faces, the edge nearest the cursor will determine the direction of the face loop. Zoom in closer for more accuracy.

Soft Selection

Selections can also be "soft." The influence of transforms can be set to expand beyond the selection, diminishing with distance.

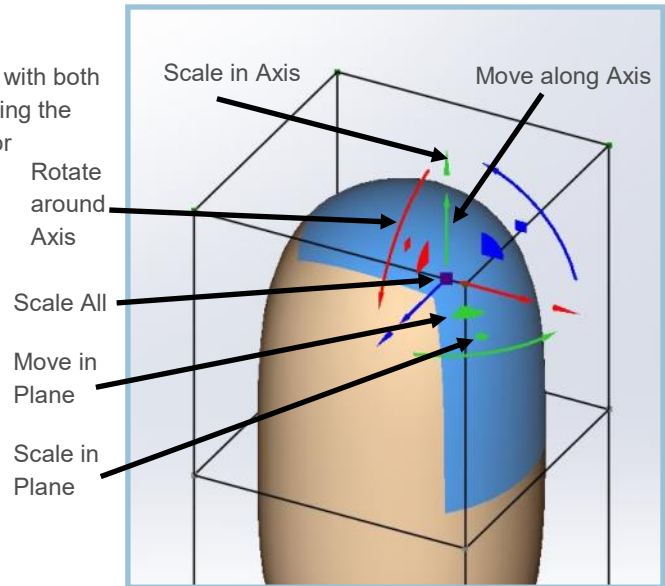


Gizmos/Triad

Power Surfacing has its own unique context aware gizmo. Besides the usual transforms, move, rotate and scale, you will be able to access many of the command's advanced parameters right in the viewport.

The Transform Triad or Gizmo

The **Transform** gizmo provides access to Move, Rotate and Scale with both single and dual axis functionality. Uniform scale is performed by using the center of the gizmo. Use the Wedge shaped portion of the gizmo for dual axis moves. Use the diamond shaped portion to perform dual axis scales.



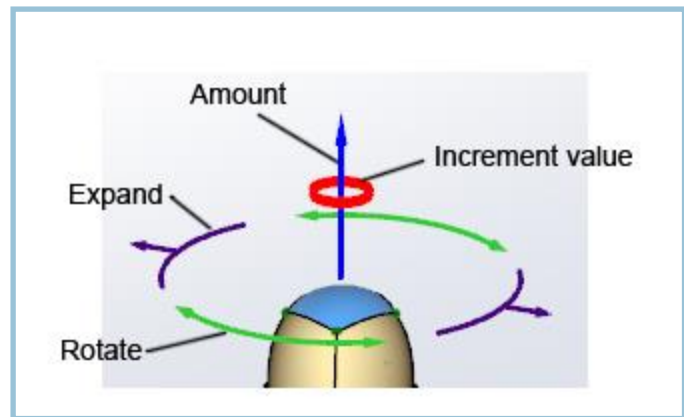
Specialty Gizmos

The **Expand** gizmo allows you to increase or decrease the size of a face or sub-object selection during extrudes or bevels.

The **Rotation** gizmo allows you to rotate the selected sub-objects while the command using it is active.

The **Increment Value** gizmo allows you to increase segments in Extrude, Bridge and Insert Loops commands. With Partial Crease activated, it lets you change the crease amount on selected edges.

The **Value Adjustment** gizmo allows you to adjust float values pertaining to the current command while active. Commands such as Bridge allow the tension at either end to be adjusted. Commands such as Insert Loops allow you to Slide or Pinch the previewed edge loops.



Property Manager

Accept, Cancel, Undo, Redo

Accept and Cancel are context sensitive commands. When you are in the middle of an operation that needs to be ended, such as Extrude or Insert Loops, the Accept command will finish or end the operation.

When no operation is in progress, the commands will act as follows:



The OK ,or Accept button, represented by the green check mark, will convert the Power Surfacing model to a BREP or NURBS model that can then be modified with the usual SolidWorks features.



The Cancel button, represented by the red check mark, will abandon any edits done to the Power Surfacing model since the last conversion.



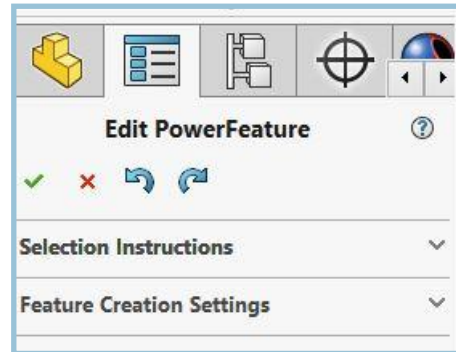
Undo , Ctrl + Z

Undoes the last action performed on the SubD. This includes selections, but stays separate from SolidWorks commands. The SubD's construction stack remains independent of conversions allowing you to be able to undo previous actions after returning to edit the SubD after a conversion.



Redo, Ctrl + X

Reverses the last undo command.



Feature Creation Settings

Post Processing

The post processing option allows additional flexibility when constraining to existing Solidworks surfaces as follows:

Create 3D Sketch: this option creates a 3D sketch along the edges where the Power Surface geometry joins the Solidworks geometry. This sketch can be used in conjunction with the Trim and Knit commands to build up a solid from surfaces.

Delete and Knit: this option is primarily used when attaching a Power Surface to a face or faces of an existing Solidworks body. This operation will automatically delete faces in the original body that are used by the Power Feature. Then it will knit the remaining faces from the delete with the faces produced by the Power Feature and often produce a resulting solid body. It is even possible to use a Power Feature to connect two separate bodies.

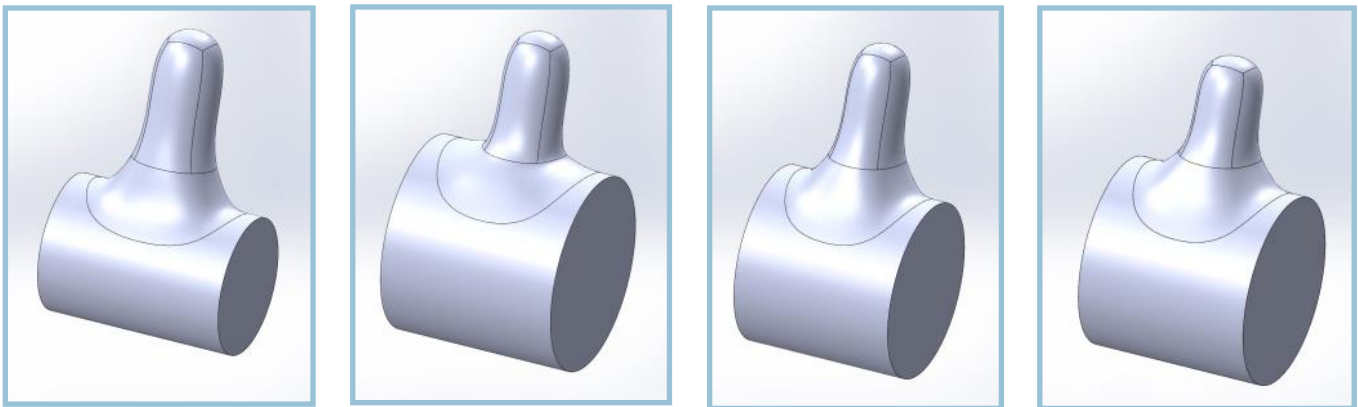
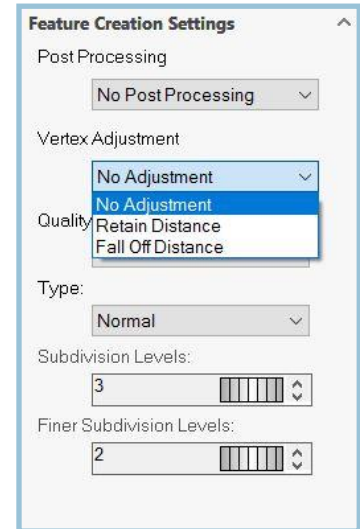
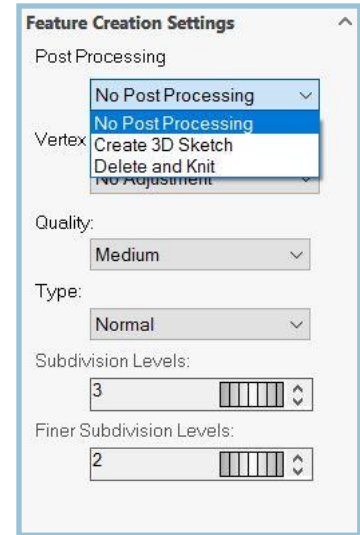
Vertex Adjustment

This option allows the user to override what happens to unconstrained vertices in the Power Feature when an underlying parametric change occurs as follows:

No Adjustment: unconstrained vertices are not moved (default)

Retain Distance: unconstrained vertices maintain a constant distance from the reference geometry they are closest to. For example if you are constraining to a cylinder and the cylinder size grows from 75mm to 100mm, all unconstrained vertices will be moved out by 25mm when the parametric update occurs.

Fall Off Distance: unconstrained vertices will be moved in proportion to their distance from the reference geometry. In the above example, vertices near the cylinder would move close to 25mm and vertices the furthest from the cylinder would move only slightly.



Left to Right: 1) Original SubD constrained to a cylinder which will change radius; 2) No Adjustment option; 3) Retain Distance option; 4) Fall Off Distance option.

Feature Creation Settings (continued)

Quality

This option allows the user to control the quality which the conversion to Solidworks geometry is processed at. Note that the higher settings will yield larger file sizes and slower processing, but will give more precise results. It is also possible to control this setting directly using **Manual Override**. If you have a very heavy quad mesh, for example something from Z-brush, this option will allow you to set the conversion to Zero Subdivisions and Zero Finer Subdivisions.

Quality-> Coarse, Medium, Fine, Very Fine, or Manual Override

Type-> Normal, 1 to 1 Planar, 1 to 1 NURBS

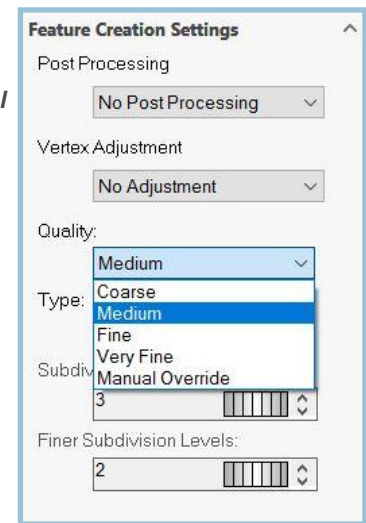
Normal converts the SubD to a minimal number of NURBS surfaces.

1 to 1 Planar converts the control mesh to a NURBS body with only planar surfaces. Use this option for a quick way to save the part file without waiting for a full convert.

1 to 1 NURBS converts the SubD creating one NURBS surface for each face in the control mesh. Use this option when you need to register features from particular SubD edges. Some Features, such as Fillet may need the edges reselected after edits to the SubD.

Manual Override Notes:

- 1) Typically very heavy and smooth quad meshes from programs like Z-Brush will convert with zero in both options. You can convert models with 1 million to 5 million quads this way.
- 2) The total number of subdivision should be no more than 6. If you want 4 *Subdivision Levels*, you should put no more than 2 *Finer Subdivision Levels*.
- 3) The subdivision levels for *Course*, *Medium*, *Fine*, and *Very Fine* will be shown when those options are selected. It takes an educated guess at what these numbers should be based on the angle between the polygons. For example if you create a default box it will typically choose 3 and 2 for *Medium*. If you subdivide that same box two times and convert, it will choose 1 and 2 for *Medium*.



Command Options

This context sensitive panel contains the parameters associated with the currently active command. When no command is being used, the rollout will be empty. See the individual features for information on their parameters and usage. When an operation is in progress, the Accept and Cancel buttons above the rollout will be associated with the current operation.

Selection Utility Panel

The Selection Utility rollout contains options pertaining to SubD selection and manipulation that are available at any time during the edit session.

Selection Filters

Select Any

Allows for selection of any of the three sub-object types: Vertex, Edge, and Face. Once a sub-object is selected, Add (Shift Key) and Remove (Ctrl Key) act only on that selection type. With nothing selected, dragging a region will select the current sub-object type only. An Edge Ring smart selection is available by double clicking in the middle half of an edge. Edge Loop smart selection is available by double clicking nearer the ends of the edge. Face Ring smart selection is available by double clicking the face near the edge in the direction of the desired ring.

Face, Edge, Vertex

Allows the selection of the corresponding face, edge or vertex types only.

Edge Loop

Selects a connected set of continuous edges that may form a closed loop. Interior loops stop and start at vertices that don't have exactly four edges. Exterior loops will continue around the boundary until they reach their starting point regardless of the edge count of their vertices.

Edge Ring

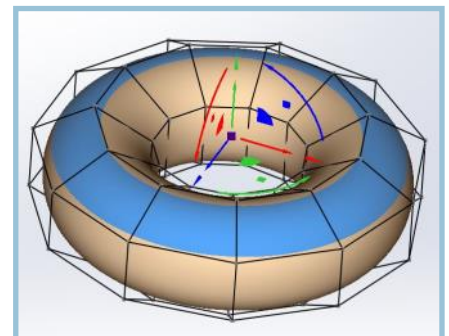
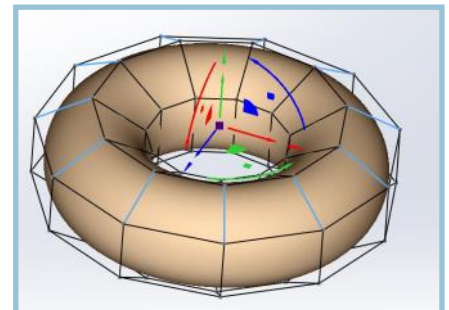
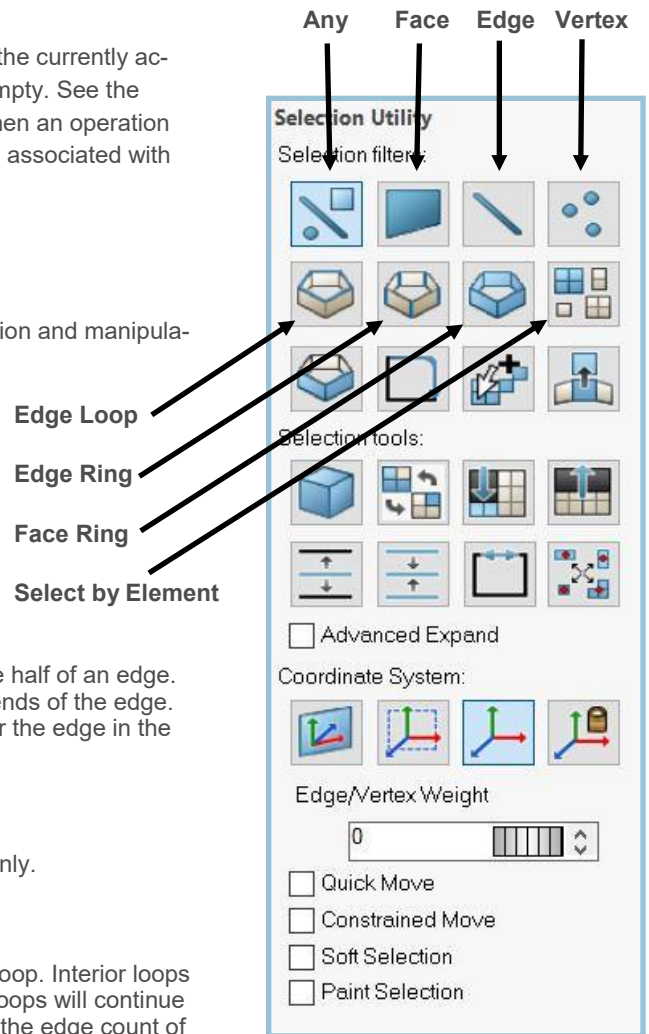
A set of edges between connected four sided faces, where the two edges on a given face do not share vertices. In other words, edges not connected to each other. See the middle image to right.

Face Ring

A connected set of four sided faces where opposite edges of each face form the connections. Face mode direction is determined by the location the face is picked. See the bottom image to the right.

Select by Element

Selects all the faces which are directly or indirectly connected to the given face. This is sometimes a shell or region.



Selection Utility Panel (continued)

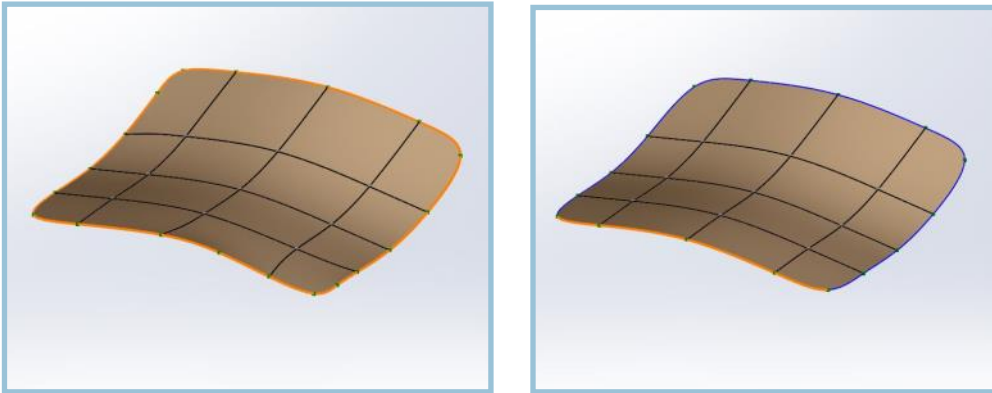
Selection Filters (continued)

Back Face Cull

When doing fence selection this flag will disallow selection of backward facing faces.

Stop At Corners

When selecting a lamina loop this flag will stop the selection at a corner where only 2 edges meet. The left image below is the default. The right image shows selection when *Stop At Corners* is selected.



Continuous Selection

Continuous selection allows you to hold down on the mouse and select any object that the mouse goes over. It is typically used to select a group of faces quickly.

Copy On Move

When selecting this button, any moves of faces will copy the faces and create new faces. Note that this is a one time use button and it will turn off after the move is executed. To do multiple copies you will need to reselect this button each time.

Back Face Cull

Stop At Corners

Continuous Selection

Copy On Move

Selection Utility

Selection filters:

Selection tools:

Advanced Expand

Coordinate System:

Edge/Vertex Weight

0 [Slider]

Quick Move

Constrained Move

Soft Selection

Paint Selection

Selection Utility Panel (continued)

Selection Tools

Select All

Selects the entire object in the current mode. If you are in Any mode, the last used selection type will be used.

Invert

Inverts the selection in whatever sub-object mode you are in.

Hide Selected

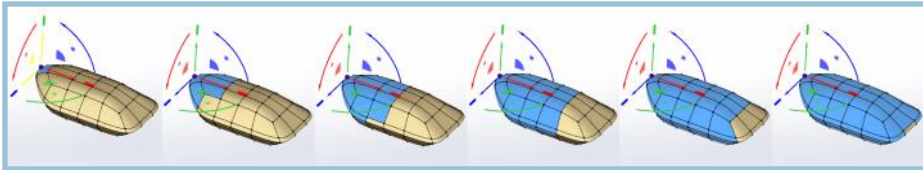
Hides selected faces. When used with vertex or edge mode, all faces associated with the vertex or edge will be hidden as well.

Unhide

Makes all hidden faces and their corresponding edges and vertices visible.

Expand Selection

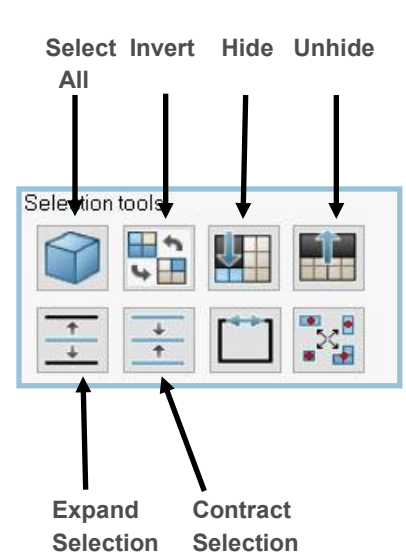
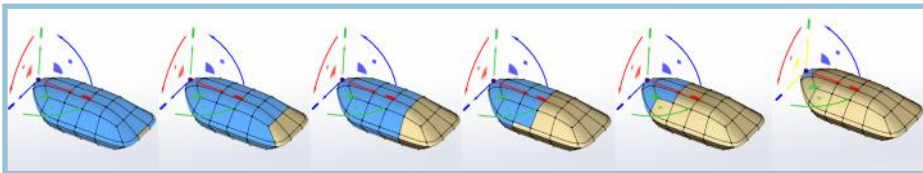
Expands the current sub-object selection one row/column in all directions each click of the button.



The selection expanded to the adjacent sub-object with each click of the button

Contract Selection

Contracts the current sub-object selection one row/column in all directions with each click of the button. Contracting the selection will not necessarily give the same result as undoing an expand selection. Once expanded to include all sub-objects, Contract will not function. At that point you should use the regular undo.

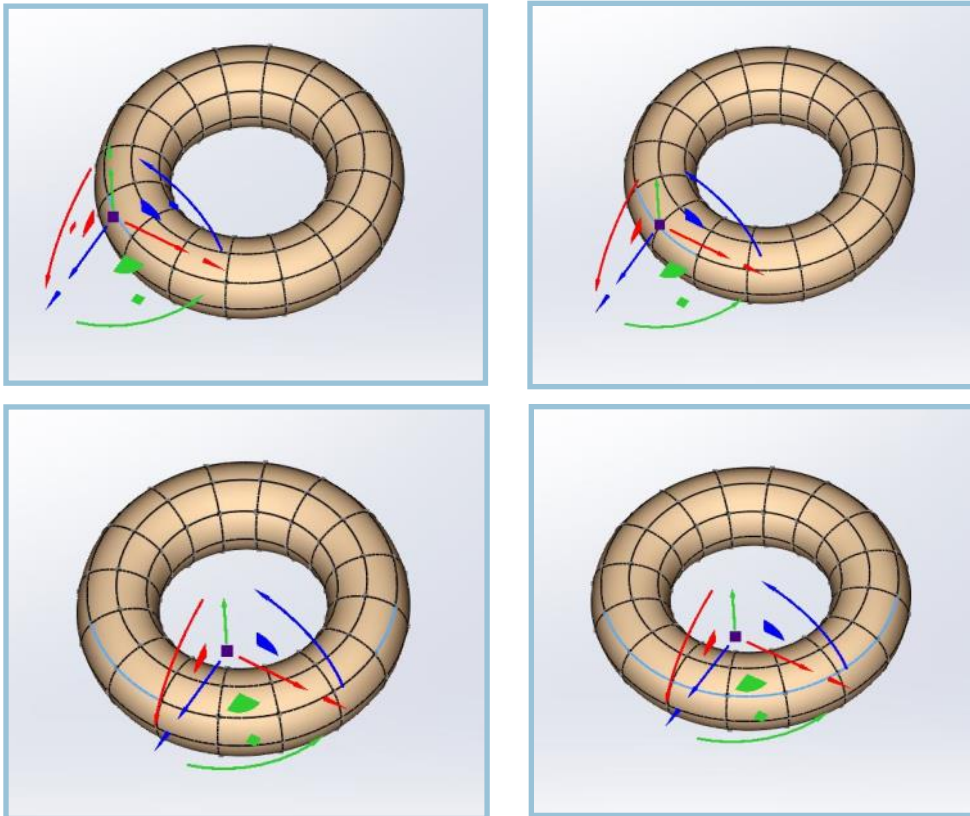


Selection Utility Panel (continued)

Selection Tools (continued)

Partial Loop

This command allows you to select a series of edges in a loop but not the whole loop. This command requires 3 edge selections. The first edge is the start edge (upper left image below). The second edge defines the direction of the pick (upper right image below). The third edge defines the end of the loop (lower left image below). After these three edges are selected you can select the partial loop button and it will select the remaining edges of the partial loop (lower right image below).

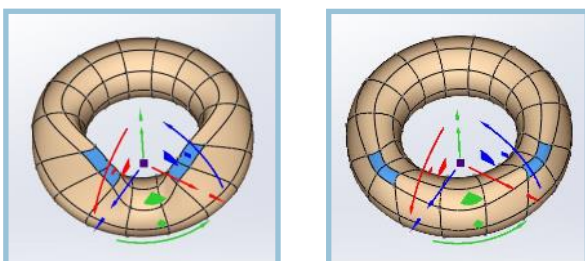


Partial Loop Selection

Scale About Local Origin

Scale About Local Origin

If you select this button, subsequent scaling of objects will be about the local origin of a connected set of objects not the origin of the entire selection set. This is the equivalent of scaling separate selections using the same scale factor. It is often useful for adjusting objects on opposite sides of a symmetric object. The image to the left below shows scaling two groups of two faces without this option and the image on the right shows scaling with this option.

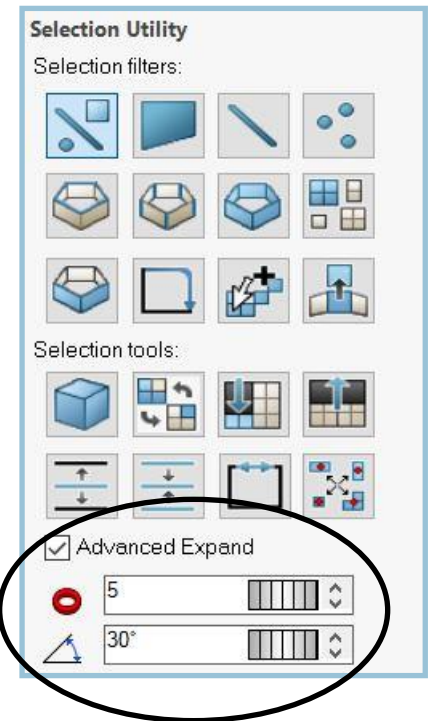
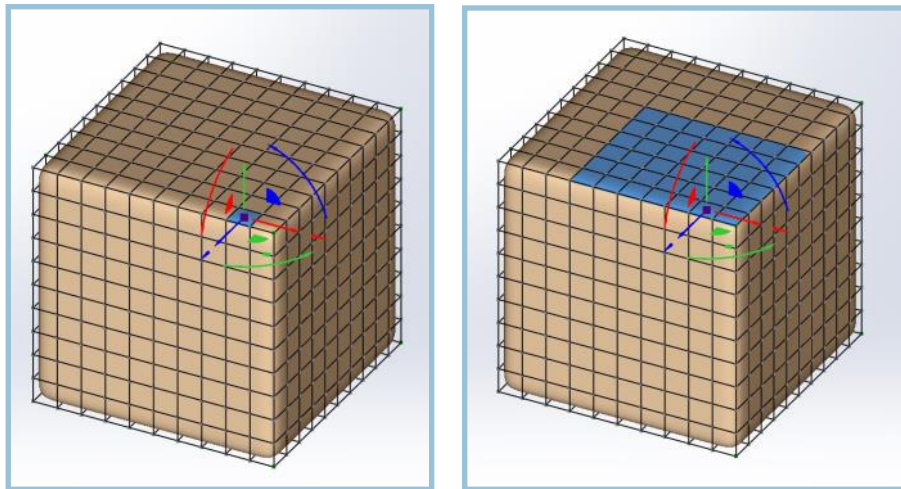


Selection Utility Panel (continued)

Selection Tools (continued)

Advanced Expansion

The advanced expansion check brings up two number boxes that control what happens when the *Advanced Expand* button is selected. The first number is the number of rings around the current selection that advanced expansion selects. The second number controls the maximum angle which the expansion will flow over. This option allows you to select planar faces (angle < 1) or faces in a homogeneous region. The left image below is a face selected. The right image shows advanced expansion using the default settings.



Triad Orientation

These options allow you to change the coordinate system used by the transform gizmo or triad. As a default, the triad uses local or "Geometry" coordinates.

Geometry

With Faces, Z is aligned with the face's normal or perpendicular. With a single edge, an axis is aligned with the edge. With a single vertex, the average of the faces connected to the vertex is used to orient the triad. With two adjacent vertices selected, the triad is oriented to the connecting edge. When the selection no longer has relevance to any of these scenarios, the triad reverts to world coordinates.

Screen

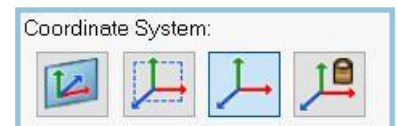
Y is vertical on the screen and X is horizontal.

World

Uses the world coordinate system.

Lock Orientation

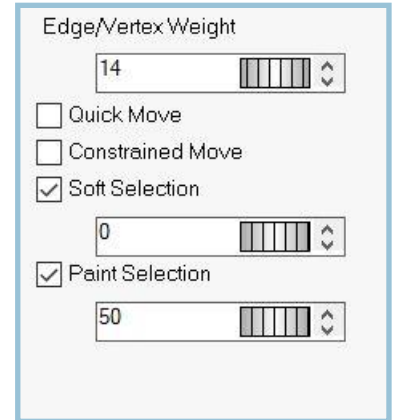
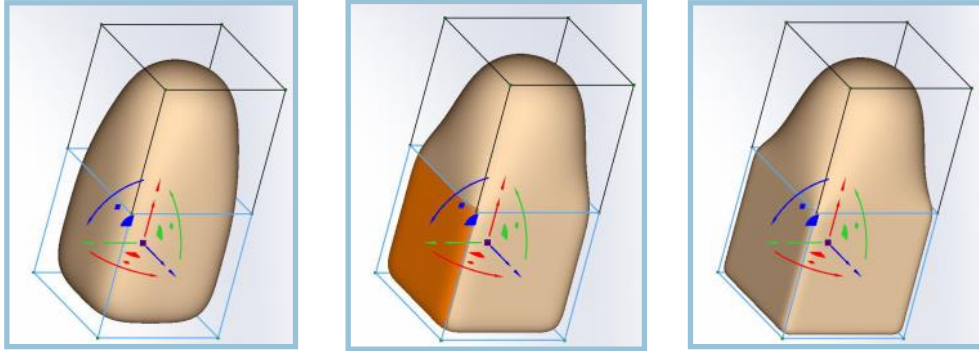
This button allows you to lock the current orientation of the triad at a given angle. Typically you will select an edge or face in *Geometry* mode to pick up its orientation and then *Lock Orientation* and proceed select and modify other objects.



Selection Utility Panel (continued)

Edge/Vertex Weight

This number box will display the weight (creasing strength) of the currently selected edges or vertices. Note that a face selection will be the equivalent of selecting all of the edges of that face for weighting. You can modify the weights of the selected objects using this number box. Below you can see a set of selected edges with weights of 20, 40 and 60.



Quick Move

This option enables you to move a vertex without needing to select it and move the gizmo. It saves the orientation of the last move and utilizes that so you can click-hold-move vertices using the previous orientation. Typically you will be moving vertices in one of the planes.

Constrained Move

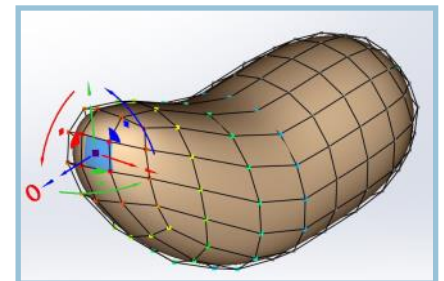
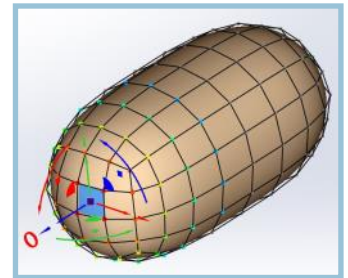
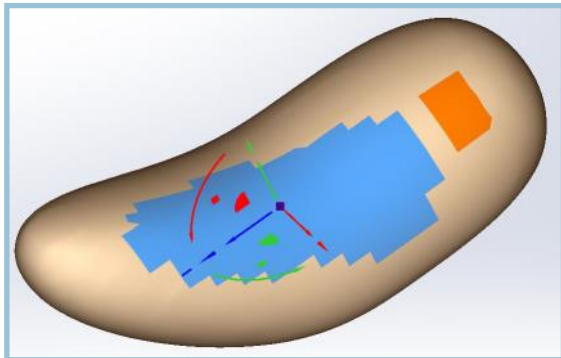
This option constrains the move of a vertex to be along one of the edges which are attached to the vertex. If you grab a vertex on a corner of a box, you can only move it along one of the edges coming into that vertex.

Soft Selection

This option allows you to move vertices near a selection in a falloff manner. Vertices near the selection move some but not as much as their selection. The color coding shows the falloff relative to the move. The number corresponds to a percentage of the overall model size. See the before and after images to the right for a move with a 33% soft selection setting.

Paint Selection

Paint selection allows you to select by painting (holding down on the mouse) over a given area and it picks up all selections within the radius of the paint tool. See the image below for an example.



Viewing Panel

Display Modes

Control

The polygonal mesh the SubD is based upon. Sub-Object selections are generally easier in this mode, but the final SubD representation may look vastly different depending on the mesh. The more faces the control mesh has, the closer the SubD resembles the control mesh, but the harder it is to edit the overall shape of the object.

SubD

Subdivision representation of the control mesh. The levels of subdivision can be adjusted for a smoother visual representation in Edit mode. This will not affect the NURBS conversion in any way.

SubD Cage

Displays both control mesh and subdivision surface. In this mode, you have the benefits of both an accurate representation of the model plus the ease of selection of the control mesh.

Visibility

When checked, the Faces, Edges and Vertices will be shown in the viewport. Choosing a selection mode that is currently hidden or suppressed automatically makes that element type visible again.

Face - Shows faces in the viewport.

Edge - Shows edges in the viewport.

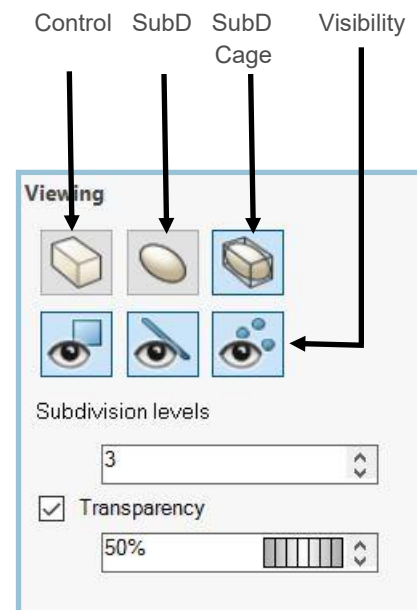
Vertex - Shows vertices in the viewport.

Subdivision Levels

This option lets you set the subdivision levels used on the SubD representation of the model in the viewport. Three or four levels are suggested for simple models. Dense models may be better viewed in two or three levels.

Transparency

This setting allows transparency to be toggled off and on. When active, you may set the amount of transparency. Use this option when working with image planes or sketches as templates.



Hot Keys and Shortcuts

1) **Add To Selection:** hold the **Ctrl** key while clicking edges, faces or vertices. The selected entity will be added if it is not already in the selection set. If it is in the selection set, it will remain there.

2) **Toggle Selection:** hold the **Shift** key while clicking edges, faces or vertices. The selected entity will be added if it is not already in the selection set. If it is in the selection set, it will be removed.

3) **Additive Mode:** there are two modes. The first mode requires select faces or open edges. Then hold down the **A/a** key and start dragging the gizmo. This will create new faces or edges. This is a quick way to do a simple *Extrude* or *Extend*. The second mode inserts loops. Hold the **A/a** key while hovering over an edge. You will see a preview of the loop to be inserted. Click and the loop will be created and inserted into the mesh. This is a quick way to do a simple *Insert Loops* command.

4) **Face Ring:** double click the mouse near an edge of a quad face to get a ring of faces selected. This is a shortcut so you will not need to use the *Face Ring* selection mode.

5) **Edge Ring:** double click near the middle third of the edge and it will select a ring of edges. An edge ring is a set of edges on opposite sides of quad faces from the start edge.

6) **Edge Loop:** double click near the ends of an edge to select all of the faces in the edge loop.

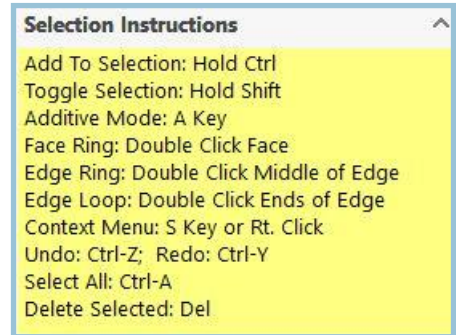
7) **Context Menu:** you can bring up the context menu with the most common commands by either hitting the **S/s** key or doing a **Right Click** on the mouse.

8) **Undo:** hit the *Ctrl-Z* key to undo a single command. You can walk back in the Power Surface command stack as far as you like.

9) **Redo:** hit the *Ctrl-Y* key to redo any prior **Undo** commands. Note that as soon as you execute any commands after **Undo**, the **Redo** commands will be cleared.

10) **Select All:** hit the *Ctrl-A* key to select all of current type entity.

11) **Delete Entity:** hit the *delete* key to delete selected faces, edges, or vertices.



Dialogs Fly-out Group

The dialogs group fly-out gives the user access to some of the advanced features to increase productivity and provide additional functionality as follows:

Selection Options provides advanced selection filters.

Selected Vertex Editor enables direct editing of vertex coordinate values.

Sectioning Options brings up a dialog to control planar sectioning.

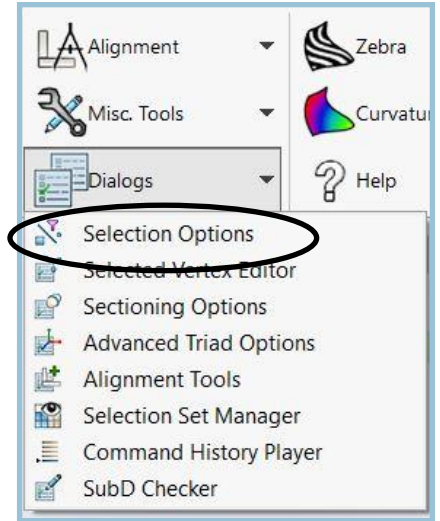
Advanced Triad Options provides some advanced tools for working with the triad in selection mode.

Alignment Tools shows many of the alignment tools in a single dialog.

Selection Set Manager allows users to create and edit named selection sets.

Command History Player allows advanced operations on the Undo/Redo history stack.

SubD Checker allows the user to check the SubD model for possible issues.



Selection Options (Filters)

This option shows the Selection Filters dialog. This is helpful for complicated crease and other edge-based operations where a large number of sub-objects are involved. The choices are as follows:

Keep Co-planar Edges: keep only edges in selection list which are between two control mesh faces which lie on the same plane.

Remove Co-planar Edges: keep only edges in selection list which are NOT between two control mesh faces that lie on the same plane.

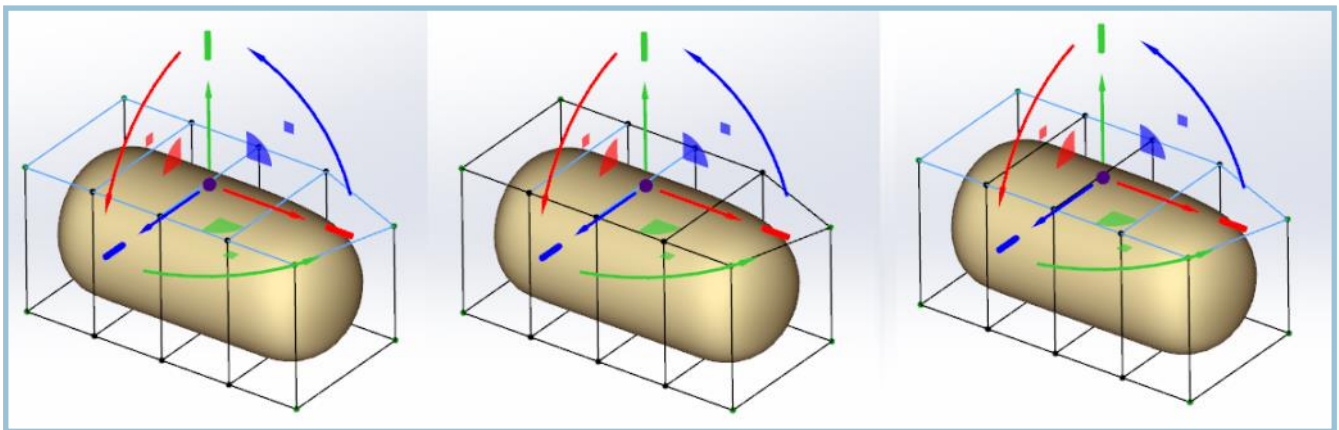
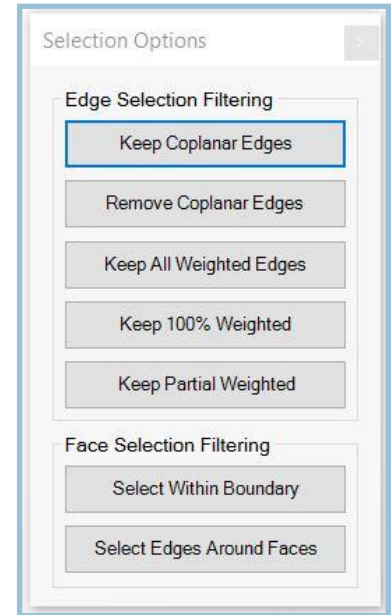
Keep All Weighted Edges: removes all edges from selection set which have edge weight of zero value

Keep 100% Weighted: removes all edges from selection set which have edge weight less than 100%

Keep Partial Weighted: removes all edges from the selection set which have an edge weight of zero or 100%.

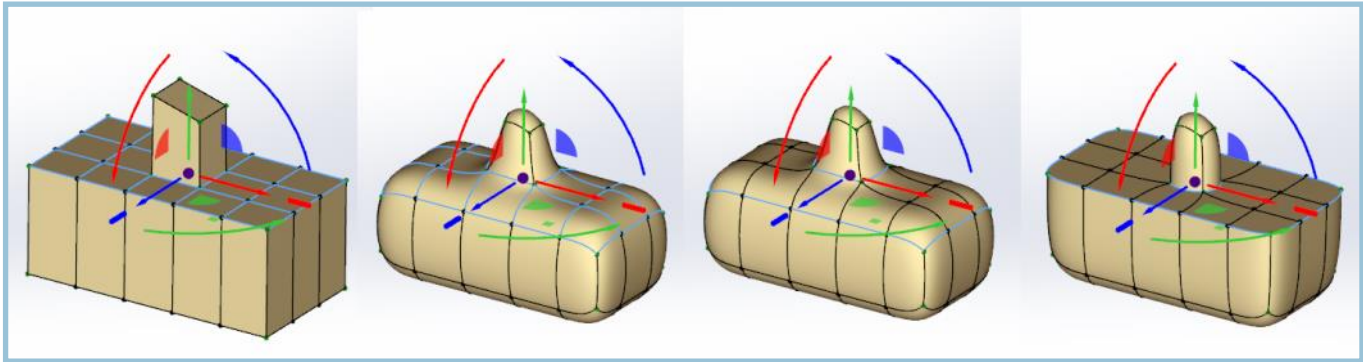
Select Within Boundary: Given a face that is selected within a boundary, this option will select all of the faces within that boundary.

Select Edges Around Faces: selects the set of edges which border a face selection.

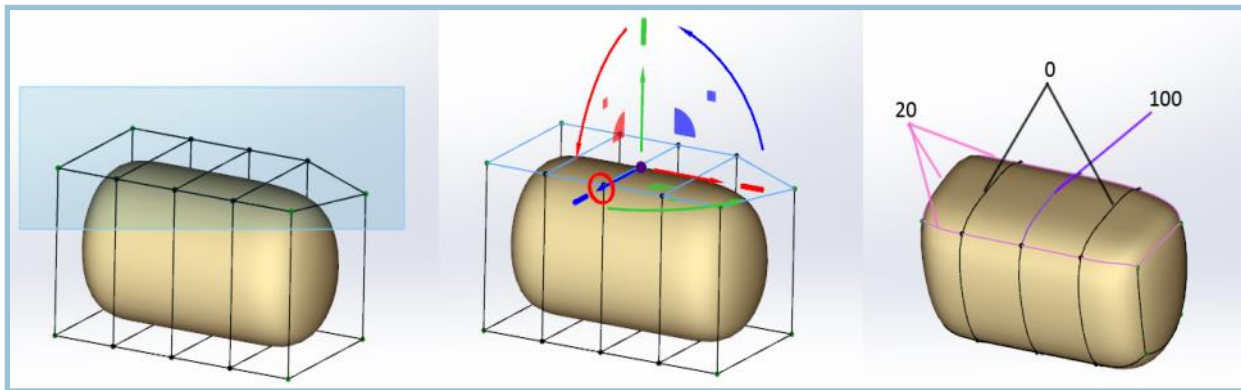


Edges in Selection Set (left image); Filtered to keep co-planar interior Edges (middle image); filtered to keep edges which are not co-planar (right image)

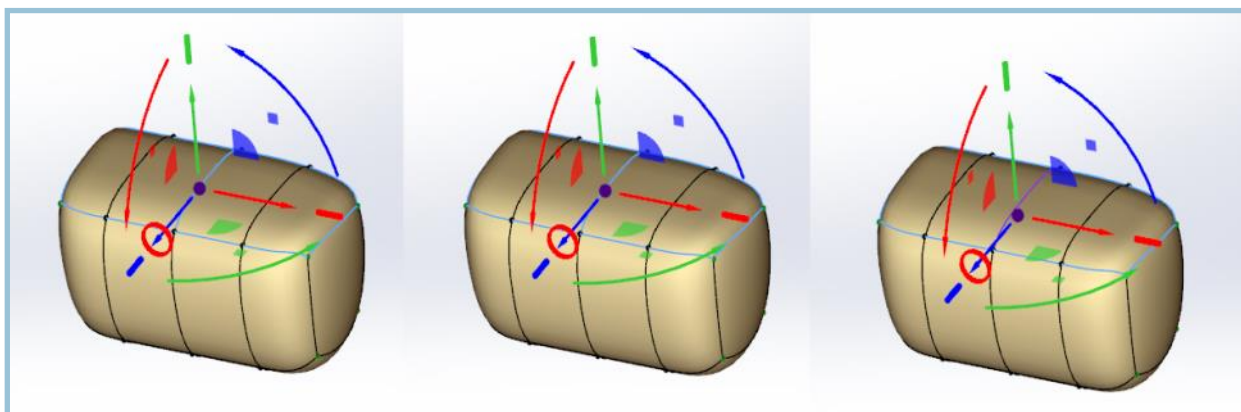
Selection Options (continued)



Original Edge Selection in control mesh display mode (first image), original selection in Sub-D mode (second image); Remove Co-planar Edges filter applied (third image); after applying hard edge command to the filtered selection (fourth image).



Selecting the top edges, (left image), the edge selection used in the following examples, (center image). Note the corner vertex dropped lower. The top outer edge weights set at 20, the top center edge set to 100 and the other two top edges set to 0, (right image).

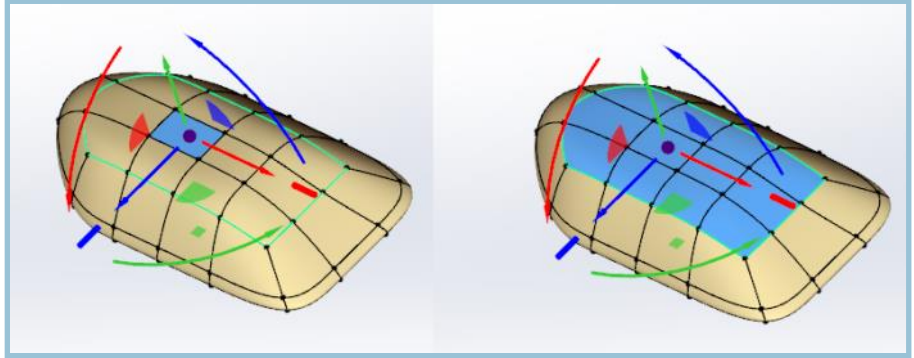


The selection with non-zero weighted edges remaining, (left image), only edges with a weight of 100% remaining, (center image), and only partial weighted edges, e.g., between 0 and 100, (right image).

Selection Options (continued)

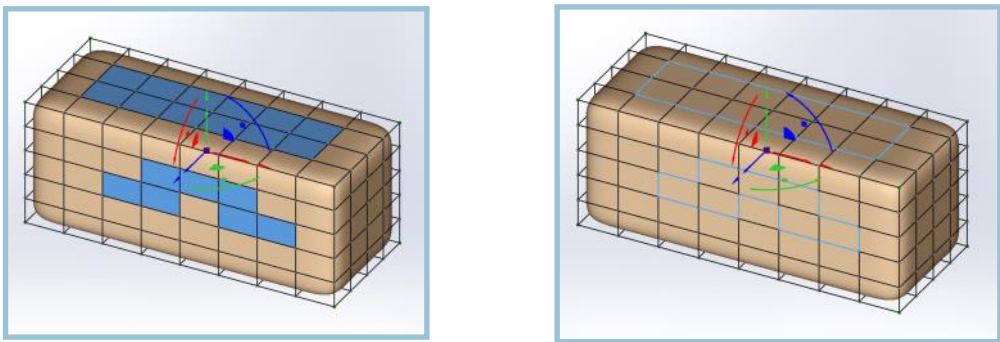
Select Within Boundary

Selects all faces within the boundary of the selected face. You will need to utilize the Define Boundary command to create boundaries from groups of faces.



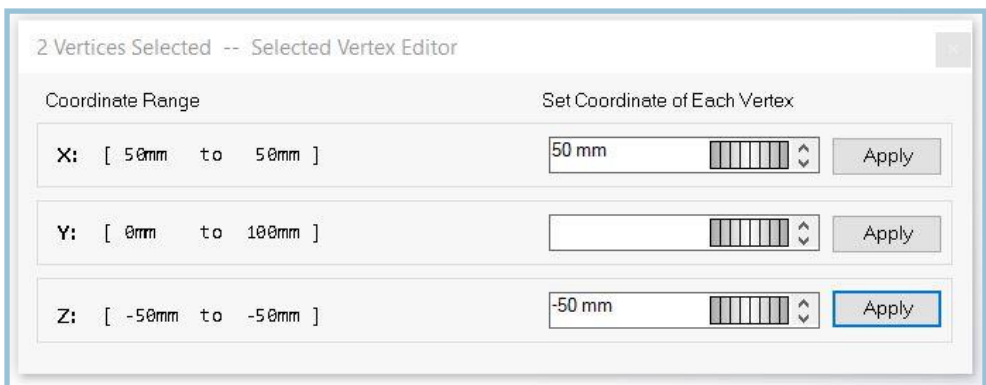
Select Edges Around Faces

Selects all edges bordering one selected face and one non selected face. The left image below shows the initial face selection and the second shows the result of *Select Edges Around Faces*.



Selected Vertex Editor

This editor allows you to manually set the location in the world coordinate system on a per vertex basis. Note that you may choose one or more vertices. If you type a value into the dialog box, you need to hit apply or return to get the points to update.



Sectioning Options

You can use Section View to help you get a feel for your SubD's shape regardless of topology. Note that sections are display mode helpers only and do not refine or change your mesh in any way.

Display

Sections- Checking Sections shows the section or sections in the viewport. See top image on right.

X-Ray- Shows the full section through the model even when the SubD is fully opaque. See second image on right. See left image below.

Cutaway- This option hides the faces, but leaves the edges showing on the mesh on one side of the section through the SubD model. See middle image below.

Options

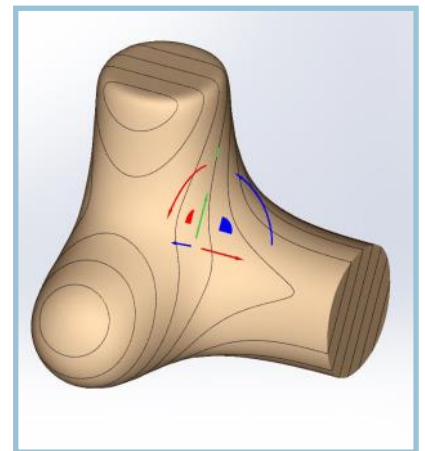
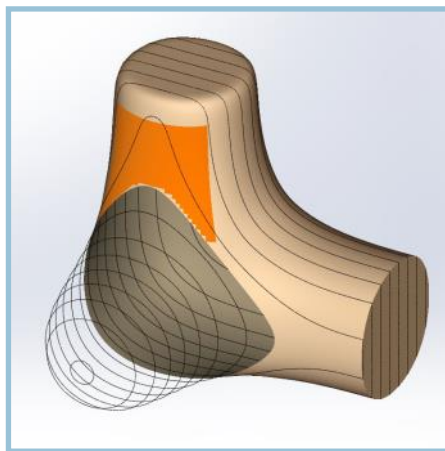
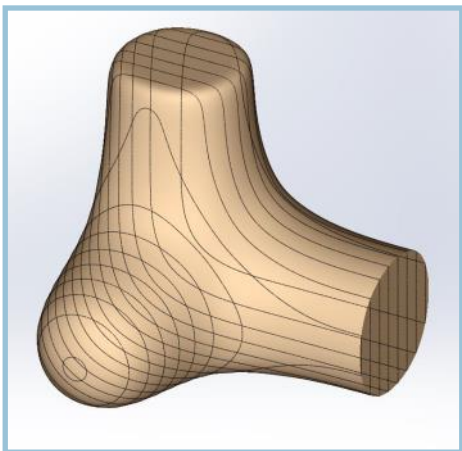
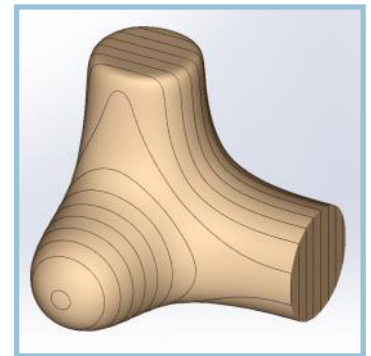
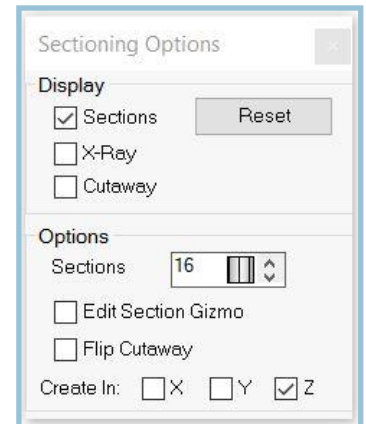
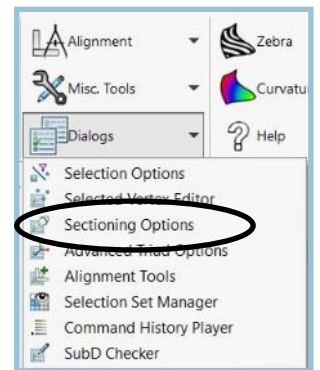
In this area, you can adjust the number of sections and how and where they are drawn through the model.

Sections- You may adjust the number of sections drawn on the model. If you set this value to more than one, Cutaway is automatically disabled.

Edit Section Gizmo- When checked, a triad/gizmo appears to allow you to manually position and orient the section plane. If you are using more than one section, all sections will be parallel to the transformed section. See right image below.

Flip Cutaway- This check box toggles the side of the model that is cutaway.

Create-In: - This option lets you turn on sections for any or all of the section plane's X, Y and Z directions.



Advanced Triad Options

These settings give you control over where and how transforms are performed on the SubD. You can specify specific transformations and specify how the triad acts under different conditions.

Prevent Reorientation: As a default, the triad reorients itself to give clear access to its individual components. When checked, Prevent Reorientation will prevent the triad from changing orientation so that it always represents the currently selected coordinate system. It is recommended that you turn on this option while typing in transform values in this section.

Translation: As a default, translation is relative. When checked, **Absolute** will position the selected sub-object or sub-objects' center point to that location in world space.

Rotation: With Rotation, you may choose the axis you wish to affect with the X,Y, and Z buttons. Rotation is always relative.

Rotation Snap: Rotation Snap can be toggled off and on here. The snap value will show as 0 when the option is off, but the previous snap value will be retained for the next time Rotation Snap is turned on in this session.

Scale: For Scale, you may choose the axis or axes you wish to affect with the X,Y, and Z buttons.

Rotation Snap: Rotation Snap allows you to choose from several preset snap degrees. As a default, the snap is set to 90 degrees. This is a proximity snap, not a forced snap.

Triad Placement: The options in this section allow you to transform selections using different pivot points. Power Surfacing will keep a static transform location that you can adjust and use any time you need it.

Dynamic: With Dynamic, the default setting, the triad is on the selected sub-object or at the center of multiple sub-objects. It changes as your selections change.

Static- Move Objects: With this setting chosen, you can transform the selection relative to the static triad or gizmo.

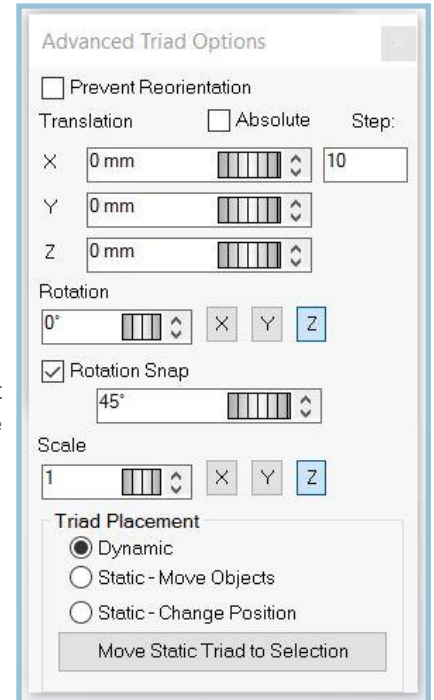
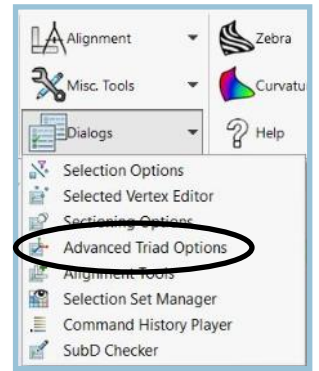
Static- Change Position: With this setting checked, you can manually position the triad to a custom location.

Move Static to Current Selection: This option provides a quick and accurate way to position and align the static triad or gizmo. It may be used in any mode. Select the desired sub-object or sub-objects and click the button to move the static gizmo to the desired location.

General: Units are derived from the Document Settings, but can be overridden during type-in.

If you are using the Geometry, or local, coordinate system, multiple selections work as per the viewport triad. The World coordinate system will be used if a meaningful average cannot be found. If you are not in Absolute mode, the local coordinate system should be respected for all three transform types. Absolute translation will always use World coordinates.

In Relative mode, until you select a different *axis*, *transform* or *sub-object*, the changes will be relative to the starting value since the last selection change was made.



Advanced Triad Options (continued)

Triad Placement

The options in this section allow you to transform selections using different pivot points. Power Surfacing will keep a static transform location that you can adjust and use any time you need it.

Dynamic

With Dynamic, the default setting, the triad is on the selected sub-object or at the center of multiple sub-objects. It changes as your selections change.

Static- Move Objects

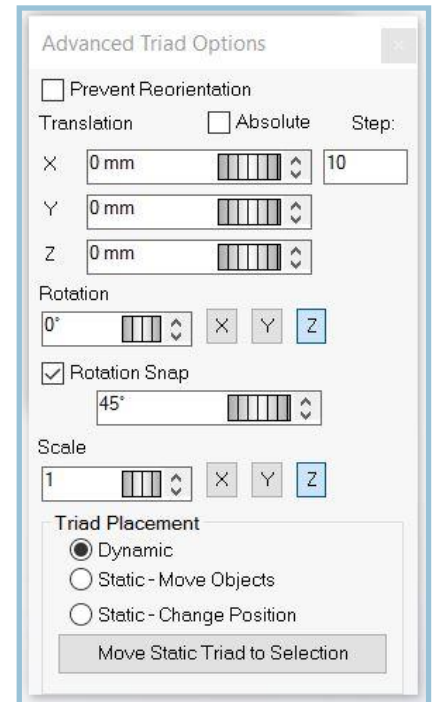
With this setting chosen, you can transform the selection relative to the static triad or gizmo.

Static- Change Position

With this setting checked, you can manually position the triad to a custom location.

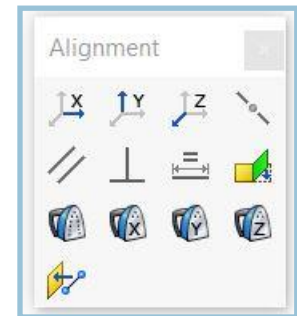
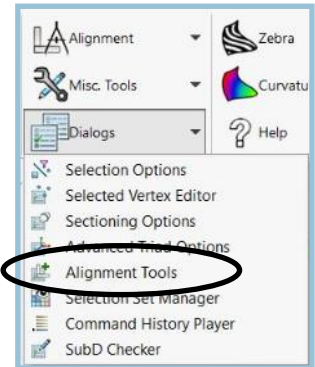
Move Static to Current Selection

This option provides a quick and accurate way to position and align the static triad or gizmo. It may be used in any mode. Select the desired sub-object or sub-objects and click the button to move the static gizmo to the desired location.



Alignment Tools

The alignment tools dialog is a convenience dialog for showing the various alignment tools all together in a dialog that stays active. All of these tools are also available in the Alignment drop down just above the Dialogs drop down. With the alignment tools, the first sub-object selected becomes the target for the alignment.



Creates a linear alignment along the world coordinate system X axis.



Creates a linear alignment along the world coordinate system Y axis.



Creates a linear alignment along the world coordinate system Z axis.



Makes the selected vertices or edges collinear. If vertices are used, the first two selected define the line. If edges are used, the first edge defines the line.



Makes the selected edges or faces parallel to the first one selected.



Makes the selected edges or faces perpendicular to the first one selected.



Makes the selected edges the same length as the first one selected.



Make faces coplanar to the first face selected.



Makes the selected sub-objects planar on the control mesh by averaging the positions of the vertices involved. The SubD representation will not appear flat unless the faces are also bounded by a creased edge.



Creates a planar alignment on the world coordinate system perpendicular to the X axis. The SubD representation will not appear flat unless the faces are also bounded by a creased edge.



Creates a planar alignment on the world coordinate system perpendicular to the Y axis. The SubD representation will not appear flat unless the faces are also bounded by a creased edge.



Creates a planar alignment on the world coordinate system perpendicular to the Z axis. The SubD representation will not appear flat unless the faces are also bounded by a creased edge.



Snaps the selected sub-objects to the specified plane.

Selection Set Manager

Power Surfacing's Selection Set Manager provides a means to retain and name selection sets of vertices, edges and faces. Additionally, face visibility can be managed in the floating window as well.

The Selection Set Manager opens in a floating window (see image on left below)

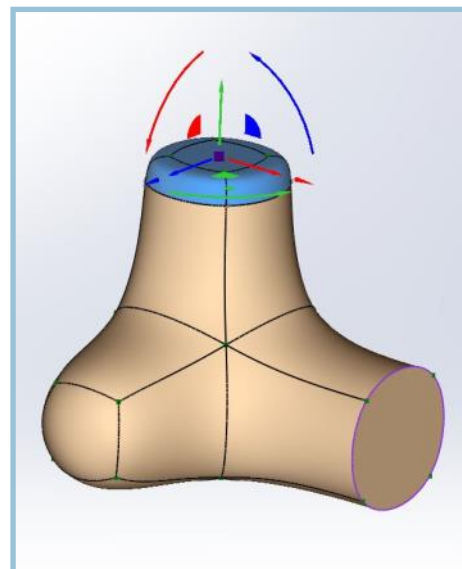
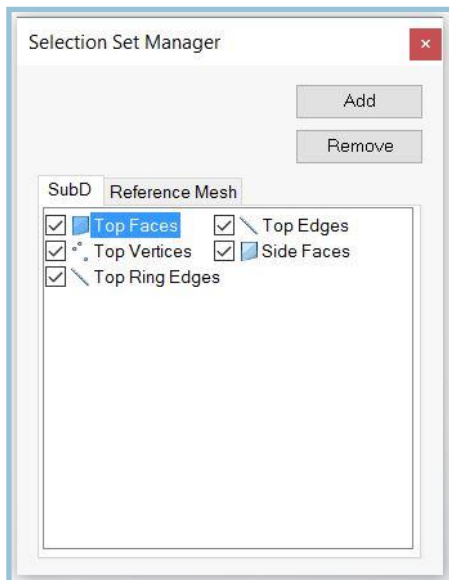
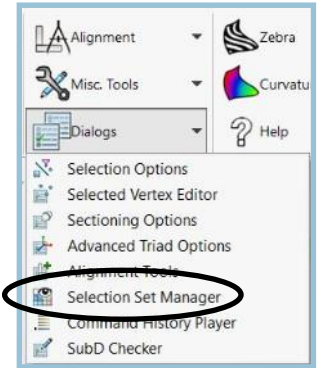
In the floating window, you will see two tabs and an Add and Delete button. The SubD tab is where you can create, name and manage sub-object selections on the SubD. The Reference Mesh tab comes into play when you are preparing reference meshes for retopologizing with Power Surfacing RE. All selection sets start out in the SubD tab. When converting a SubD to a Reference mesh for the retopologizing process, SubD selection sets are transferred to the Reference Mesh tab.

Typically, selection sets are created when the sub-objects in question are difficult to access and/or are frequently used during the modelling process. An example is a handle that is shaped with the Bend tool. As the Bend must be undone before it can be re-calculated, it is handy having the selection available at the click of the mouse. To create a selection set, the process is as follows:

1. Open the Selection Manager through the Dialogs fly-out.
2. Select the vertices, edges or faces you wish to have quick access to.
3. Click Add.
4. Click on the name just added to modify it to a useful name.

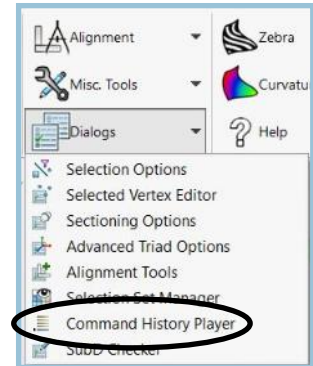
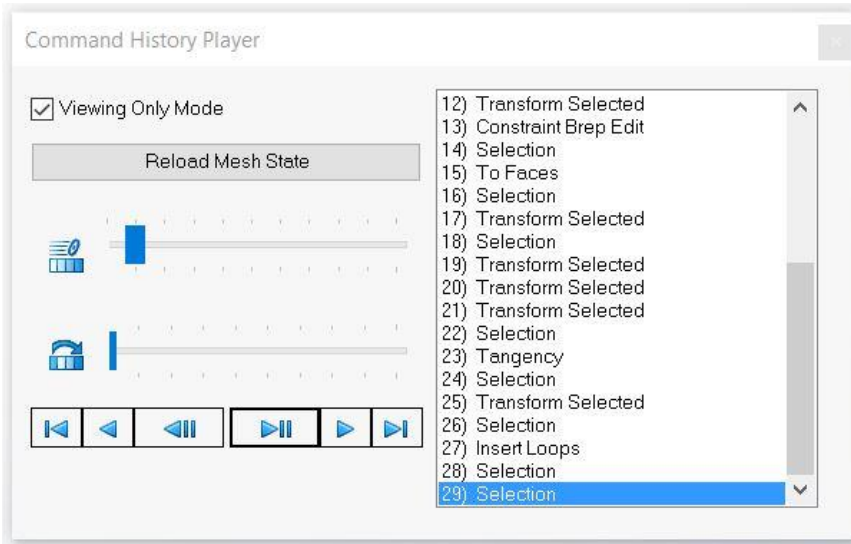
To select the items in the named selection set, click the item in the list. See the two images below.

To hide the items click off the check box. Note that hiding edges and vertices will also hide any adjacent faces.



Command History Player

The Command History Player gives access to the Power Surfacing undo stack where you can use several different options to review the command history of a subD model. You can play through the command stack manually or automatically, forward or backward, from any position. You can select any command in the list to see what the model looks like at that particular point in its creation process.



Viewing Only Mode allows you to investigate creation workflows of existing models without altering the mesh. When unchecked, you can continue editing the mesh from any point in the stack.

Reload Mesh State will reload the mesh into the state it was in when you last opened the player. If you are experimenting with different solutions to your model, you may save them through the Power Surfacing menu > Save > Save Scene. This will save the model in the native .pModel format.

The **Speed** slider lets you adjust playback speed when using Play Forward/Pause or Play Backward/Pause.

The **Step Count** slider allows you to specify how many steps to skip when using step forwards or backwards by step count and also when using the regular Play/Pause or Play Backward/Pause. A maximum of nine steps can be skipped at a time.

Go to start sets the stack to the Initial state.

Step backward by step count allows you to step backward through the stack manually one command at a time or by multiples if Step Count is more than 1.

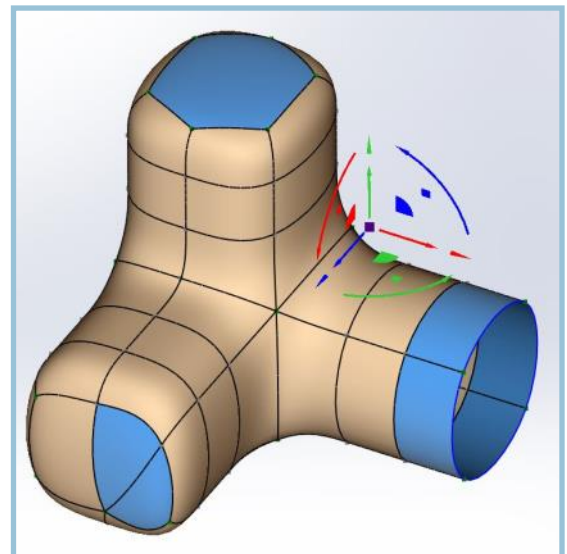
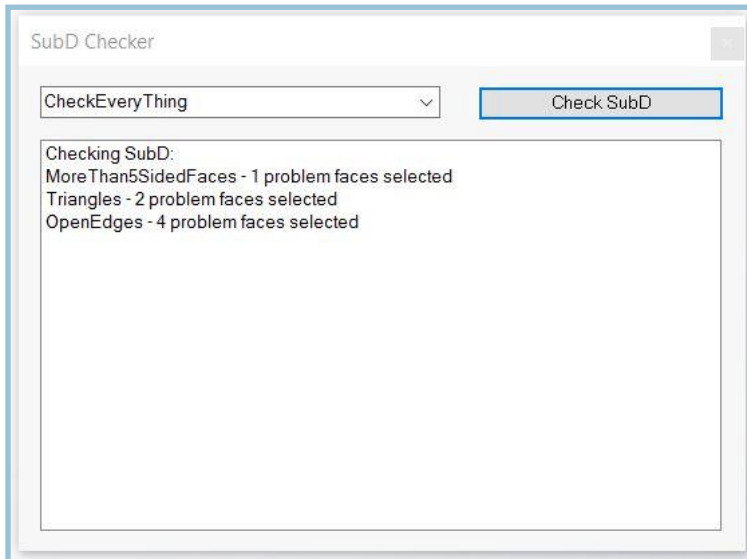
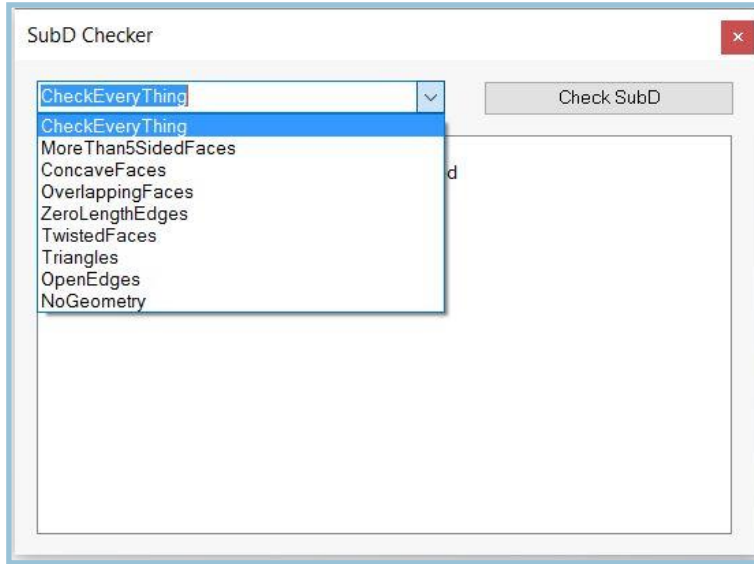
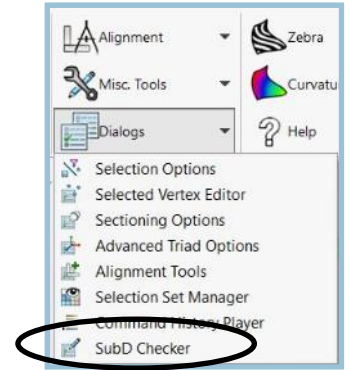
Play Backward/Pause will automatically play back through the command stack one command at a time or by multiples if Step Count is more than 1. You may pause playback and continue by pressing the same button.

Step forward by step count allows you to step forward through the stack manually one command at a time or by multiples if Step Count is more than 1.

Go to End sets the current command to the final command.

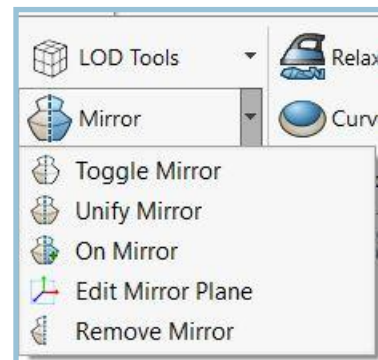
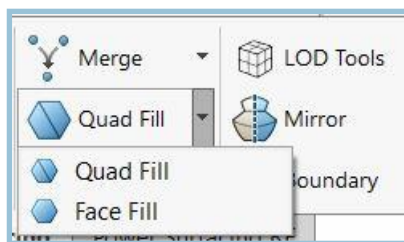
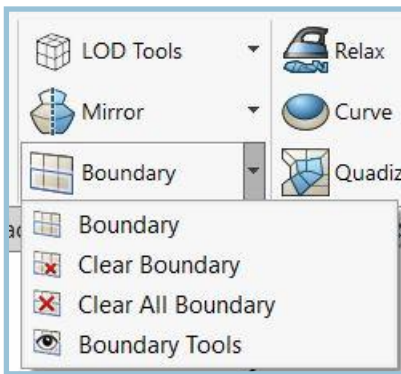
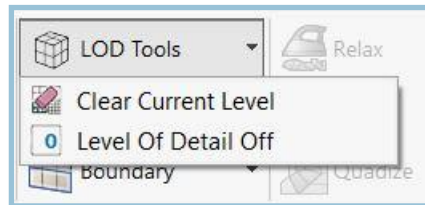
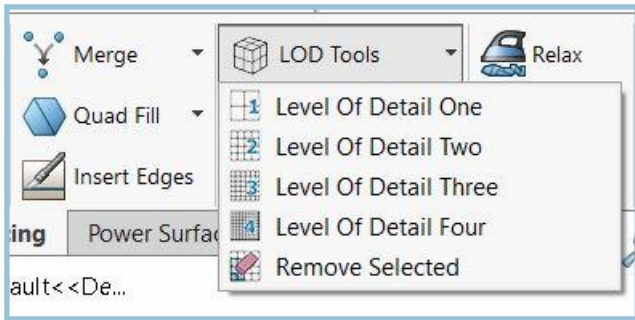
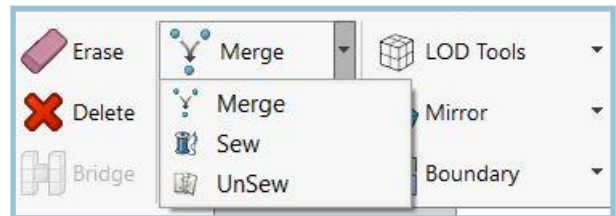
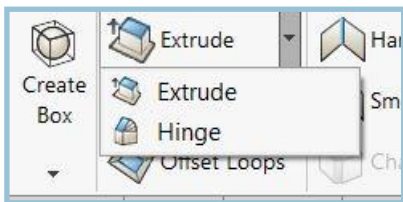
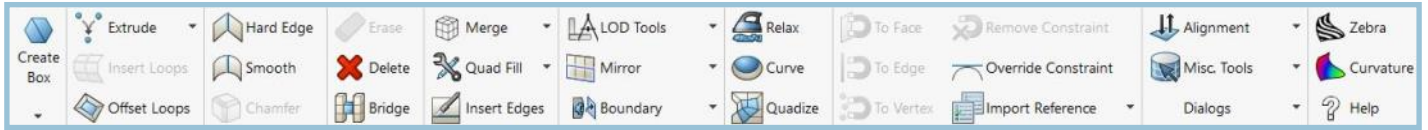
SubD Checker

The Checker looks for potential problems in the SubD mesh. Some of them are serious and some of them will not cause major problems. It is important to be in face selection mode prior to running the checker because it will highlight faces which are in areas that have issues. See the example below with one six sided face, three open edges, and two triangles.



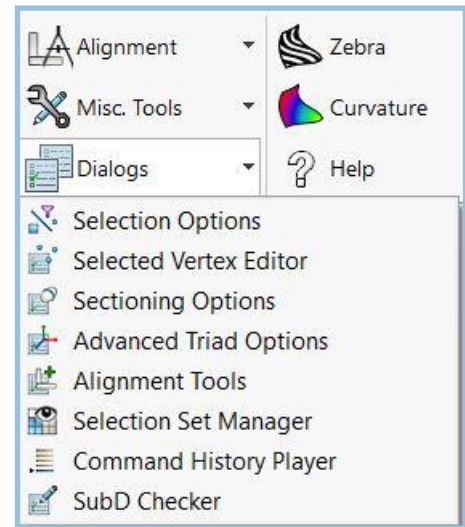
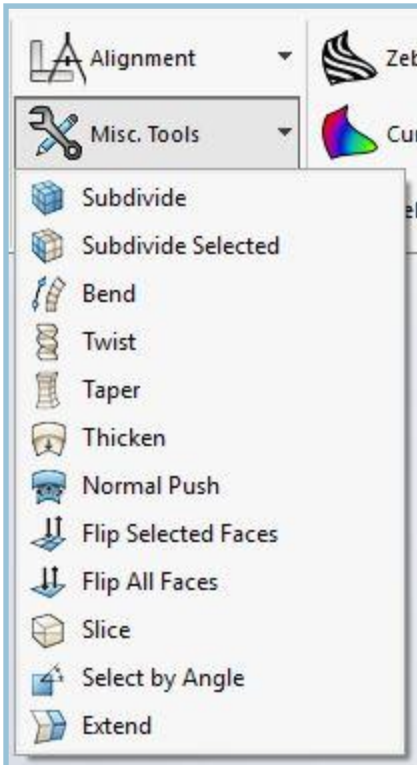
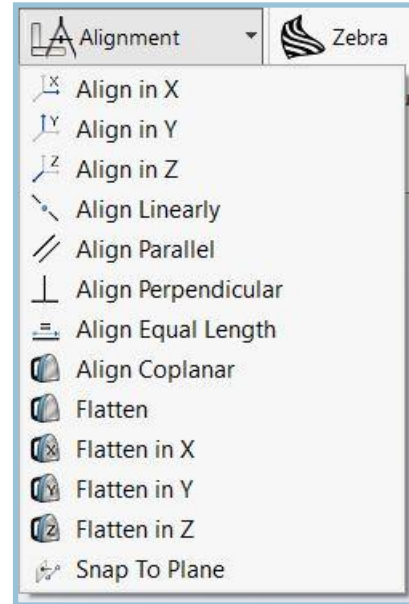
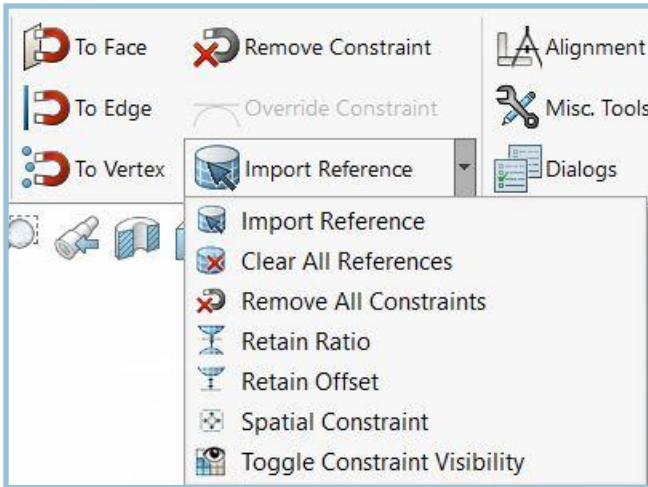
Tools and Flyouts

Tools and commands can be accessed either from the tool bar or from the right-click menu. Once in a command, you will be able to use the specialized Gizmos to take advantage of the command's advanced features. The Power Surfacing toolbar can be customized just like any other SolidWorks toolbar.



Tools and Flyouts (continued)

Note that disabled tools are not shown in the flyouts. Misc. Tools has a different configuration depending on what is selected. If an open edge is selected, it has the *Extend* command visible.



Extrude





The most common use is to extrude a selected face or faces. It can also be used to extrude edges.

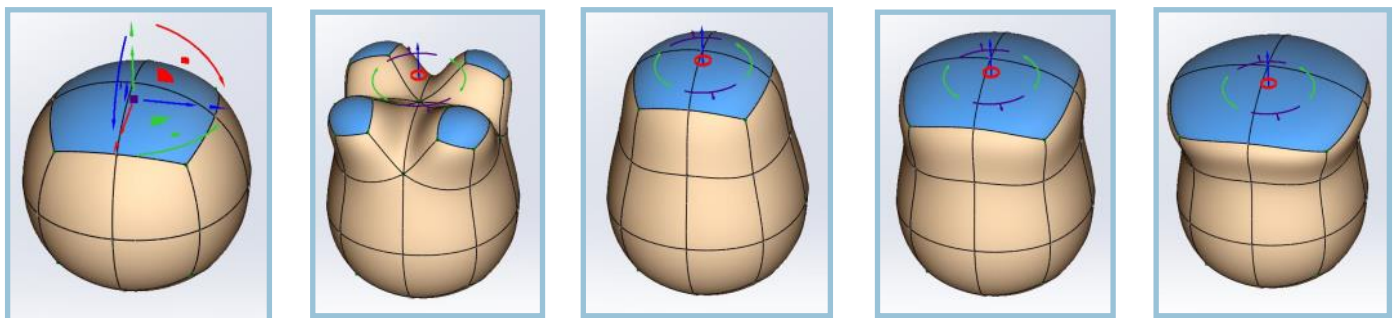
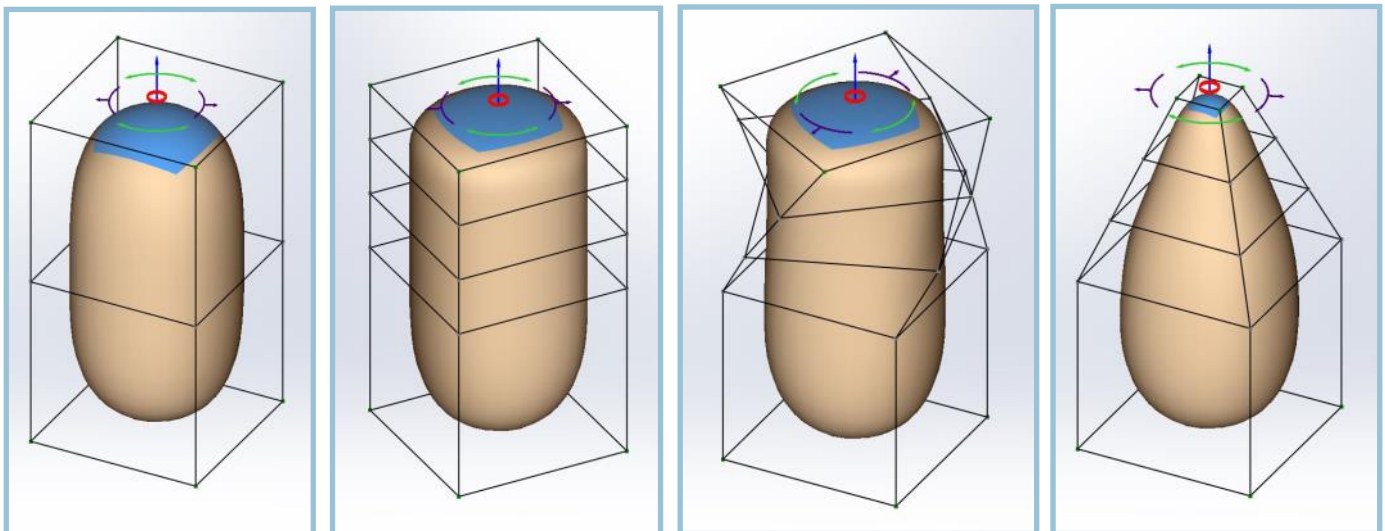
The options are extrusion distance, number of segments (second image below), rotation angle (third image below) and scale (right image below).

If Grouping Type is set to *No Grouping*, a face will be extruded from each of the selected faces see the second image in the bottom row which is an extrude of the first image in the bottom row.

The third image in the bottom row shows an extrude of 4 faces with the *Area Weighted Normal* option. This option creates a single normal by area weighting all normals of the selected faces. The fourth image shows the *Each Vertex Selected Faces* which computes a different normal for each vertex based on the selected faces. The fifth image shows *Each Vertex All Faces*, which computes a different normal for each vertex using all faces adjacent to that vertex not just the selected ones.

Available in Face and Edge mode.

Command Options
 100 mm
 1
 0°
 1
Grouping:
Edge Grouping
Normal:
Area Weighted

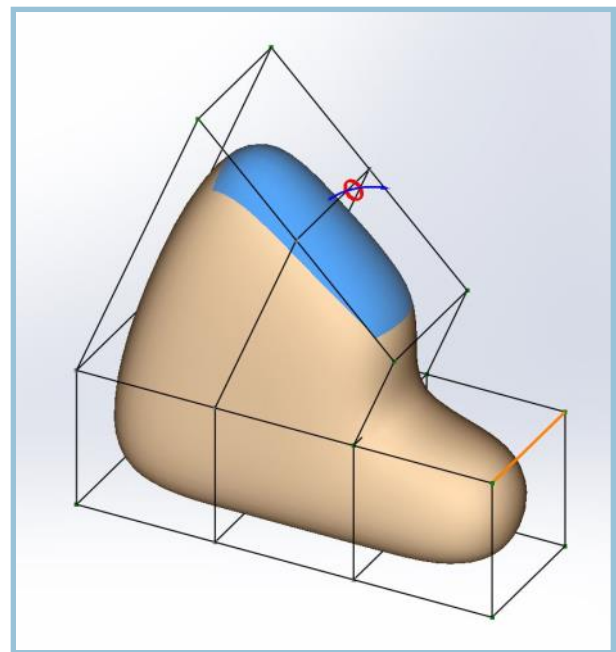
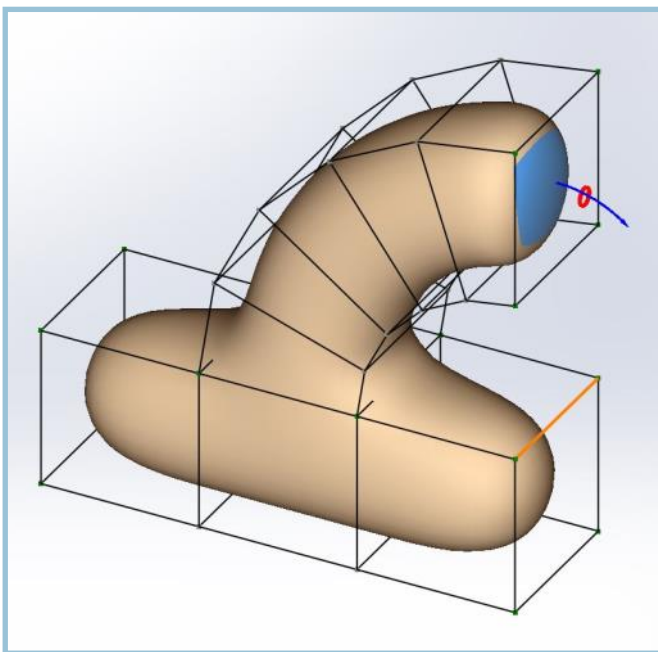
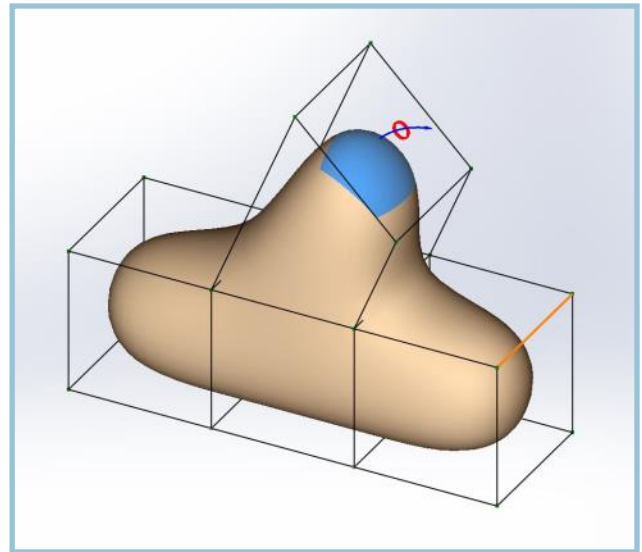
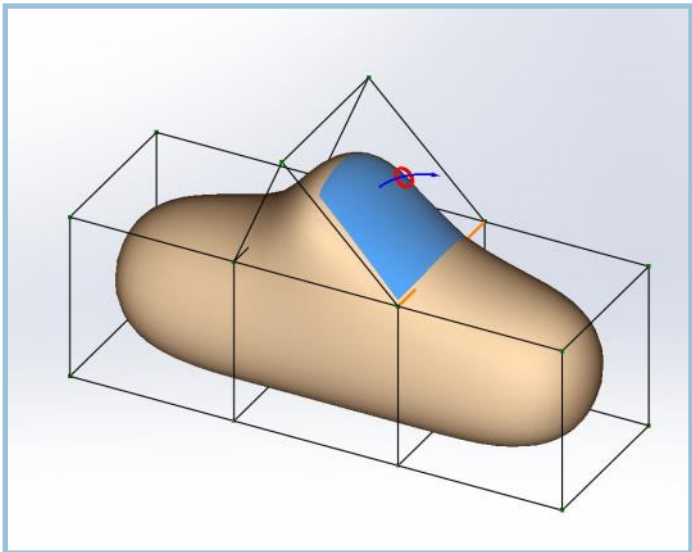
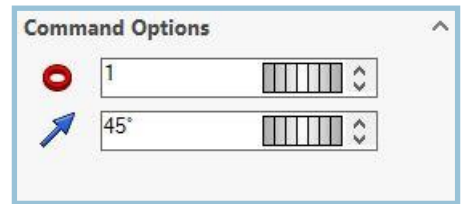




Hinge

Extrudes a face or faces hinging from the specified edge. After initializing the tool, you can set the number of segments and affect the angle of the final face. You must select the hinge edge to initialize the tool.

The top left image shows a hinge with the selected edge on the face. The top right image below shows a hinge with an edge not on the face. The bottom left image shows a 90 degree hinge with 5 segments. The bottom right image shows a hinge with multiple faces.





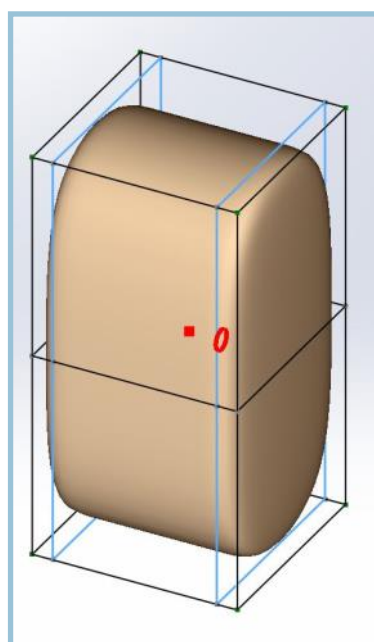
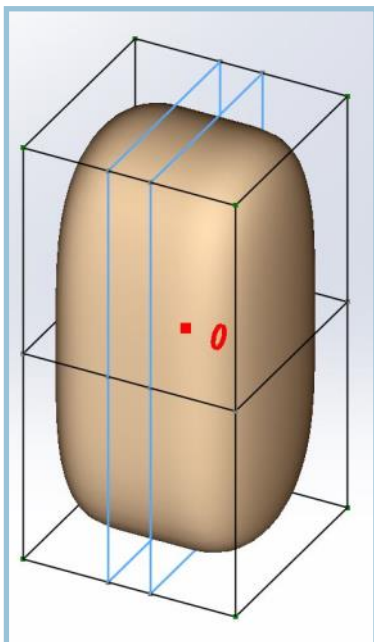
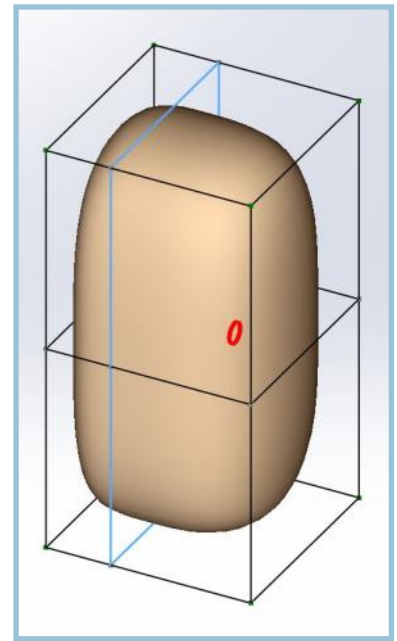
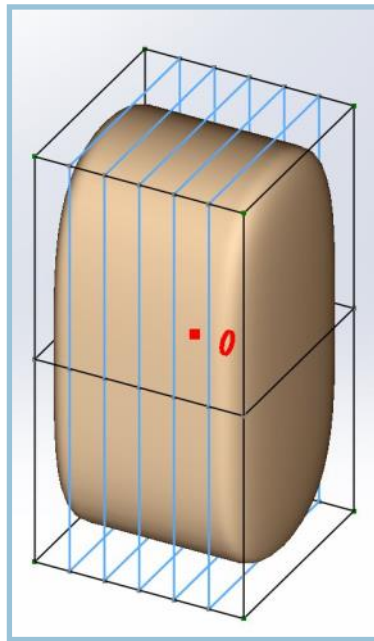
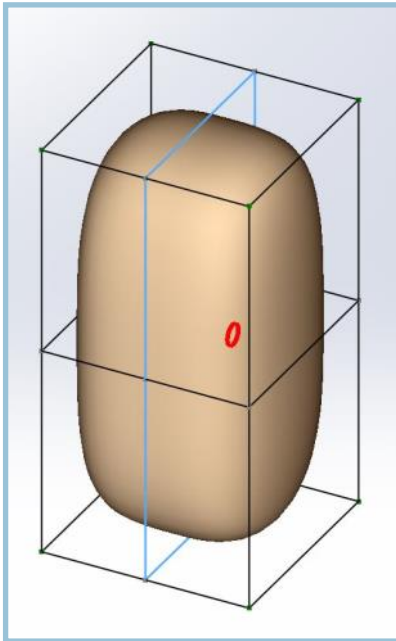
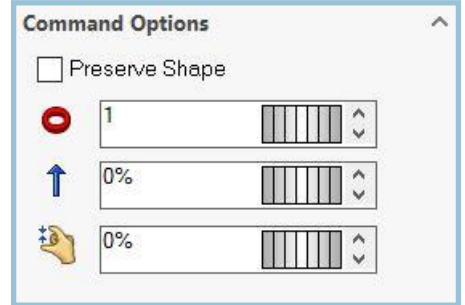
Insert Loops

Insert Loops adds one or more edge loops to the selected edge ring. After initializing the tool, you can set the number of segments or loops, slide them to one side or the other, or pinch them closer or farther apart. Right clicking or picking the Accept check mark ends the command.

Requires at least two edges selected, but works best when selecting with Edge Ring.

Available from Edge or Edge Loop mode.

The upper left image shows default values. The upper middle shows 5 segments. The upper right shows a slide value of 50%. The bottom two images show a pinch value of -20 and 70 respectively.











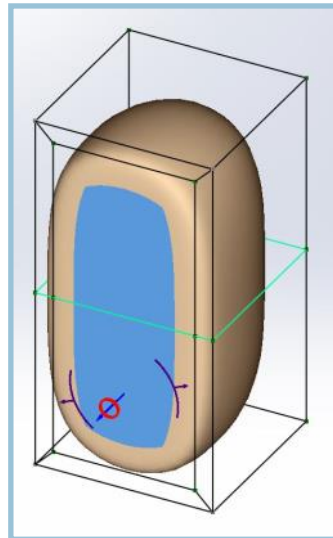
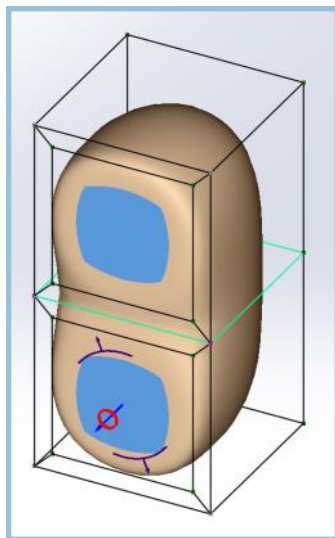
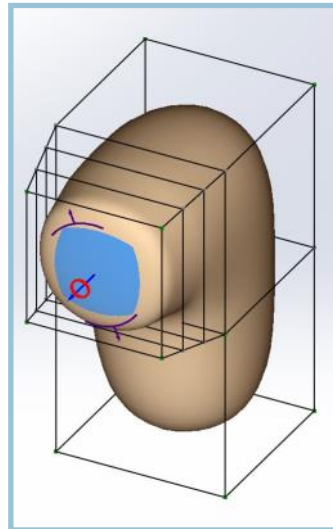
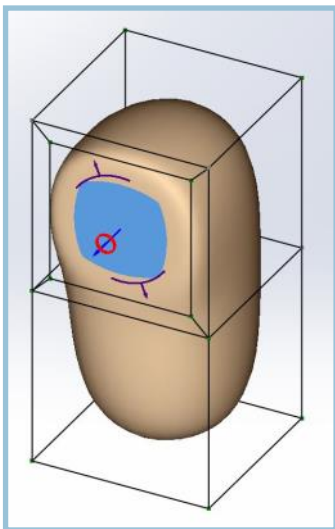
Offset Loops

Creates an offset loop from a selected face or faces. If Grouping Type is set to No Grouping in the Command Parameters window, a loop will be offset on each of the selected faces. Edges can have Offset Loop applied to them to create new faces on either side of the selected edge with additional faces on each end of the edge.

Available from Face and Edge mode.

Upper left image is the default offset parameters. The upper right image shows a distance applied and 3 segments making it behave much like an extrude. The lower left image shows standard *Edge Grouping* with a mirror. The lower right shows *Mirror Crossing* grouping.

Command Options
 0 mm 
 10% 
 1 
Grouping:
Edge Grouping 
Normal:
Area Weighted 



Hard Edge

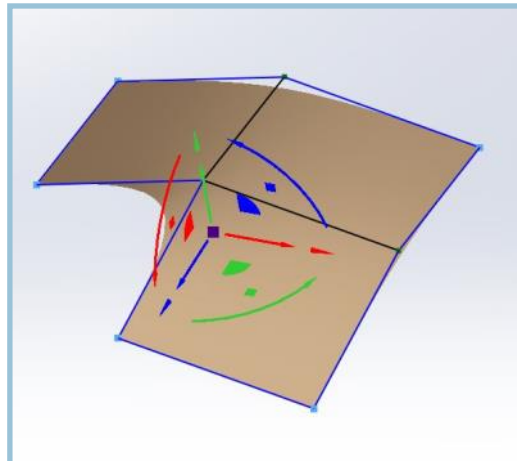
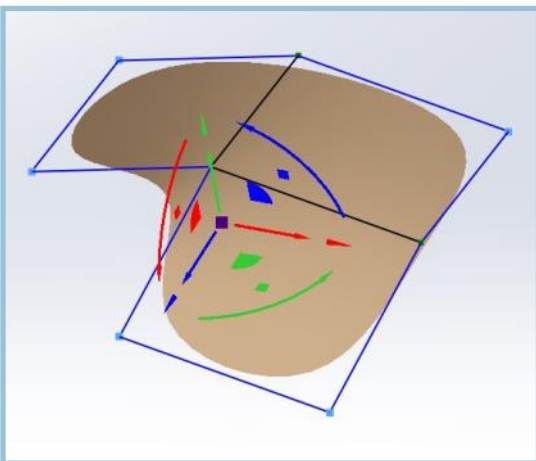
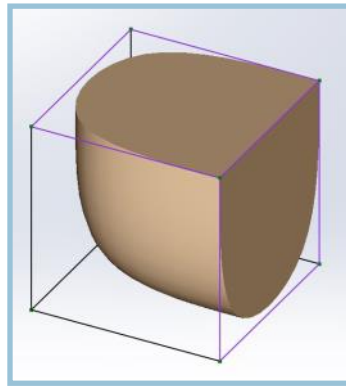
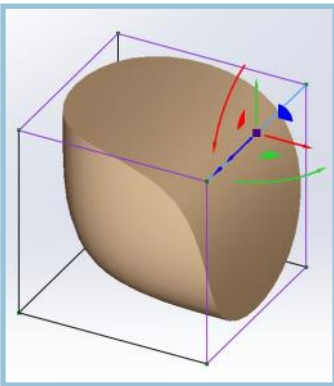
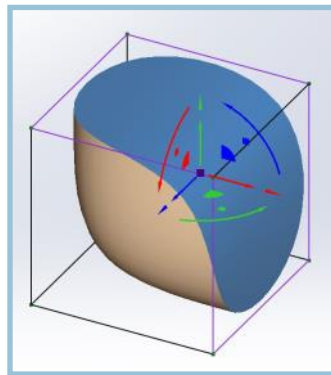
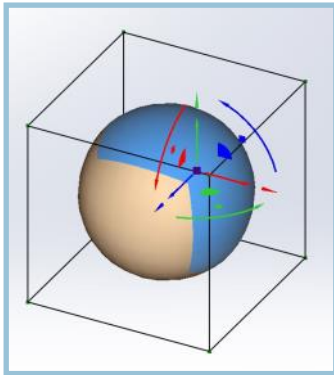
Sets the selected edges or vertices to an weight value of 100 for a fully hard edge or a sharp vertex. Use a hard edge before conversion to be able to use the standard SolidWork's Fillet Feature. Partial weighting can be set through the Property Manager. When faces are selected, only the boundary edges between the selected faces and unselected faces are affected.

Available from Vertex, Edge or Face mode.

The first row below shows Hard Edge applied to two adjacent faces. Note that the interior edge remains smooth.

The second row below shows the Hard Edge applied to a single edge.

The third row below shows Hard Edge applied to vertices. This is most useful when doing surface modeling.



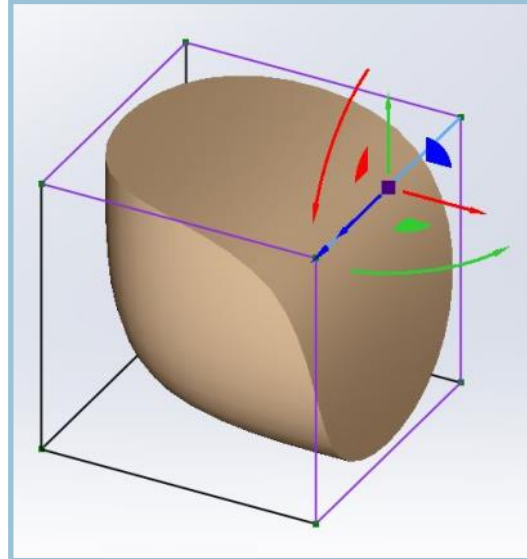
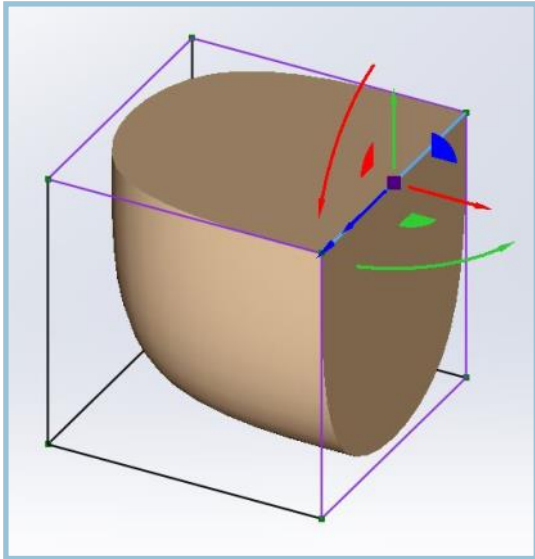


Smooth Edge

Sets the selected edges or vertices to a crease value of 0. Partial creasing can be set through the Property Manager. When faces are selected, only the boundary edges between the selected faces and unselected faces are affected. When faces are selected, only the boundary edges between the selected faces and unselected faces are affected.

Available from Vertex, Edge or Face mode.

The two images below show applying a Smooth Edge to a fully creased edge.

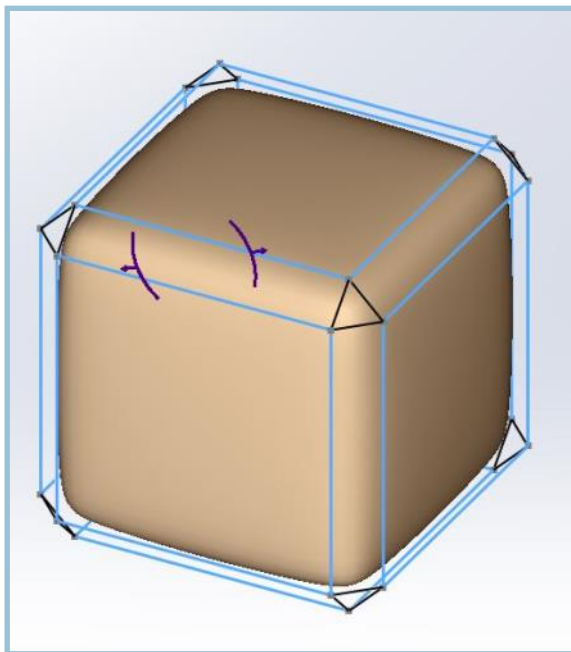
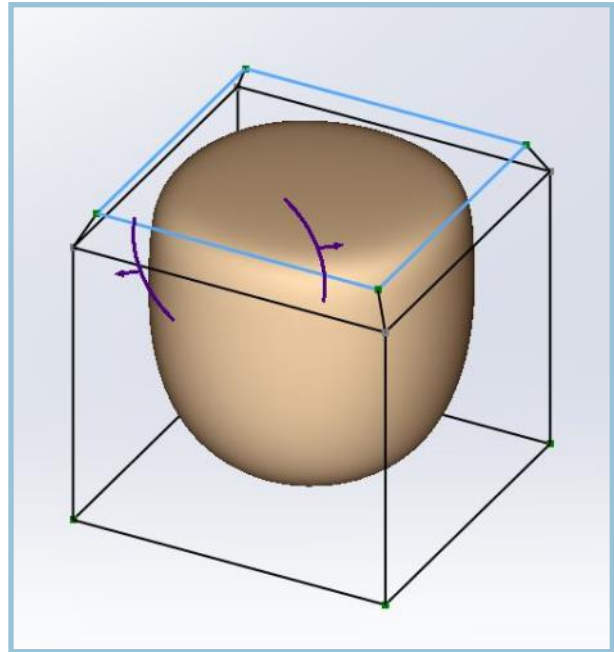
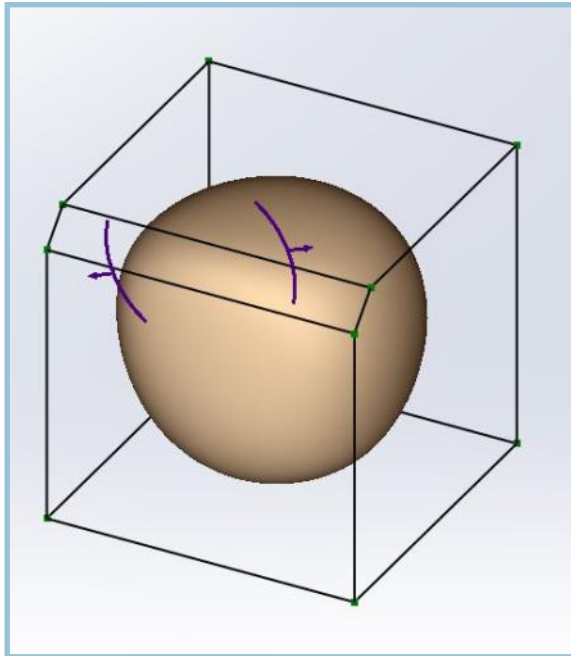
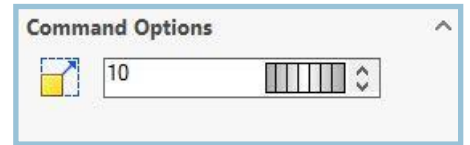




Chamfer

Chamfer is used to bevel edges. It is one way to make smooth corners sharper.

The upper left image shows a single edge chamfer on the standard box. The upper right image shows a chamfer of a loop of edges on the standard box. The lower image shows a chamfer of all of the edges of a standard box.

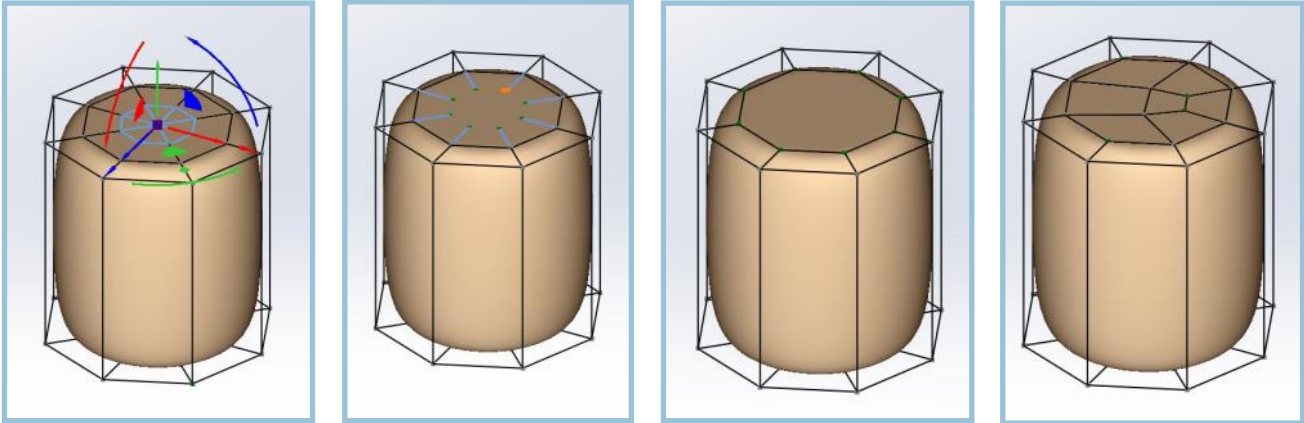


Erase

The Erase tool removes selected edges and any isolated vertices. Note that it does not delete faces adjacent to the edge or vertex. Typically you will use the Erase tool and either the *Quadize* or *Insert Edges* tool to change the flow of the edges in a particular area of the mesh.

Available for Edges.

The following sequence of edge erasing and *Quadize* demonstrates a typical workflow for using the *Erase* tool



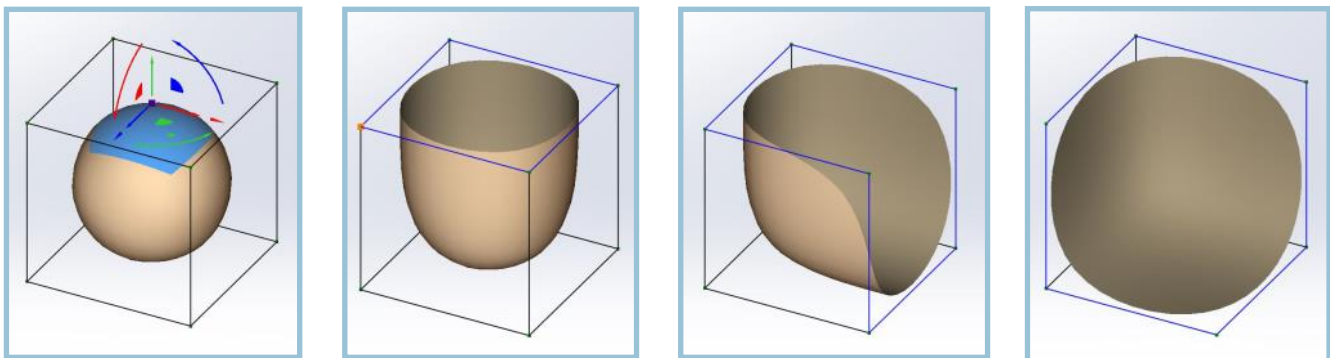
Delete

Deletes the selected sub-object. If the selection is a face, the face will be deleted. If the selection is an edge, the two faces sharing the edge will be deleted. If the selection is a vertex, all faces sharing the vertex will be deleted.

The resulting geometry will be a hole with open edges. These edges can be extruded as is, or the hole can be refilled by first selecting the edges, then using the *Fill Face* tool.

Available for Edges, Vertices and Faces

The following images show deletion of a single face, single edge and a single vertex on a standard box.



Bridge

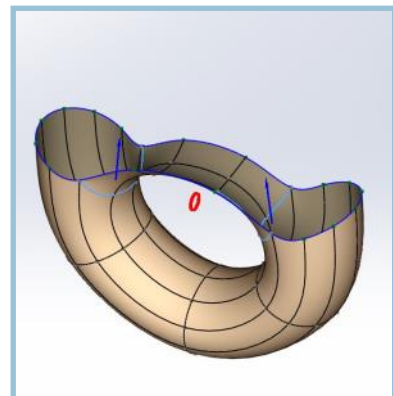
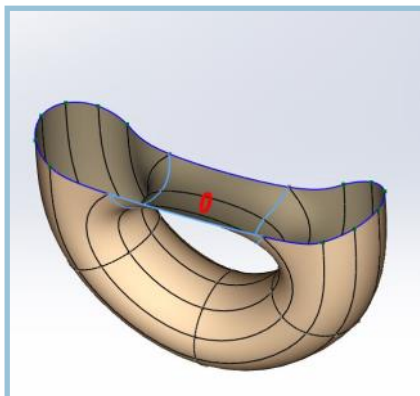
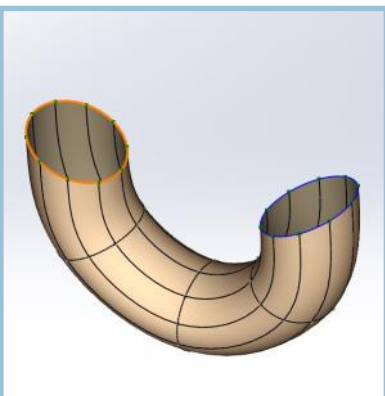
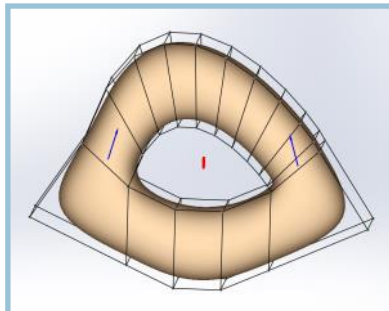
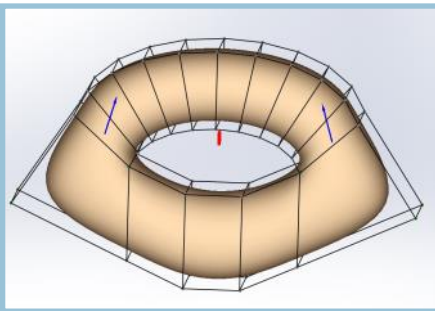
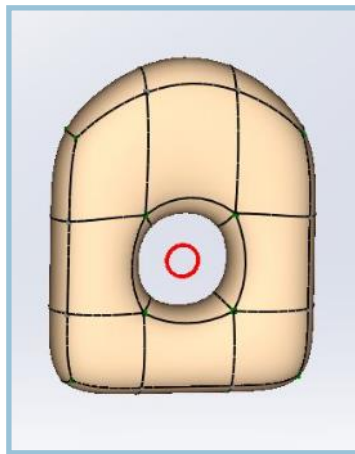
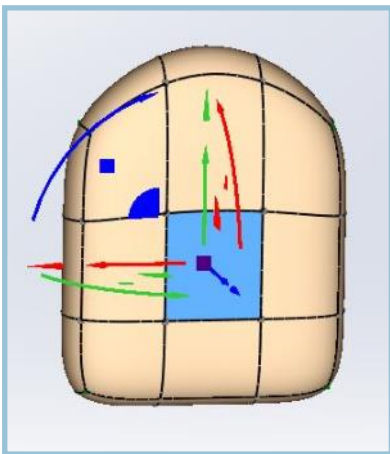
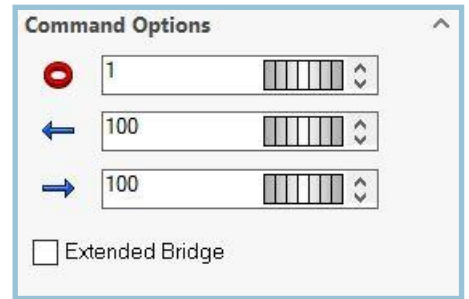
Connects two faces or two groups of faces in face mode, or connects two open edges. After the initial Bridge, you can adjust the number of segments and adjust the tension on either end to affect the bridge's curvature. Right clicking or picking the Accept check mark ends the command.

Available from Face or Edge Mode.

The top sequence of images below show: 1) select two faces one on front and one on back side of the mesh; 2) Bridge the selected faces to make a hole;

The second row of images shows an exterior bridge of two faces then the changing the strength of the left side from 100 to 300 and the right side to 20.

The third row of images demonstrates bridge of 3 edges on either side. The right image on the bottom shows increasing the number of segments to 3.





Merge Vertices

Collapses two or more vertices into one. If Edge mode is used, the vertices that define the selected edge or edges are collapsed or merged. In Face mode, the vertices defining the face or faces are merged into one.

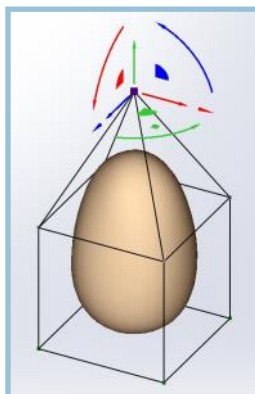
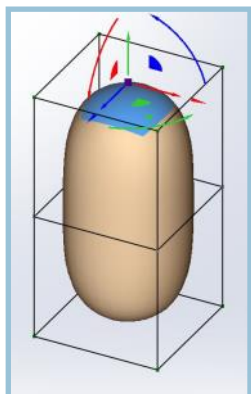
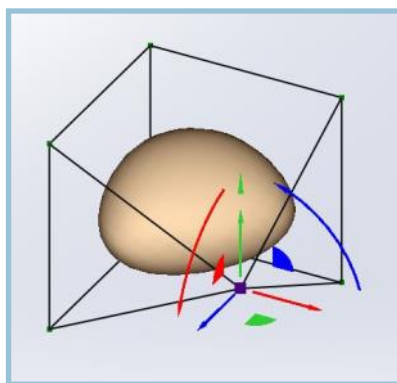
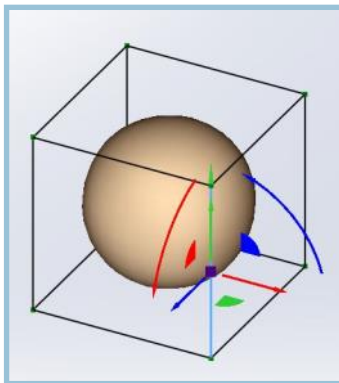
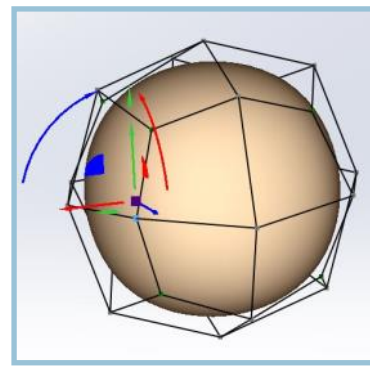
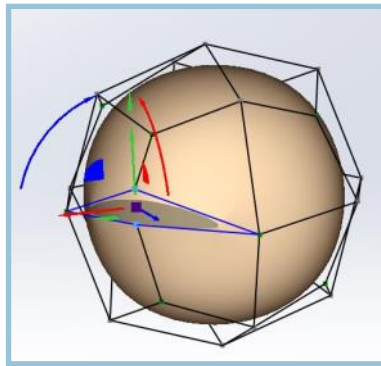
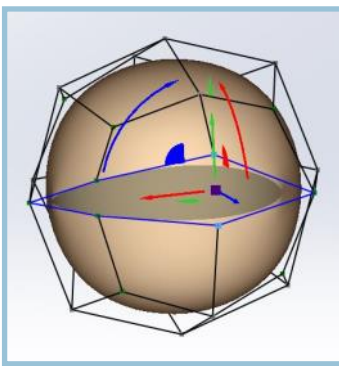
When sub-objects are region selected all at once, their average position is used as the merge point. If one vertex is selected first, then the others added to the selection, the first selection's location is used as the merge point.

Available in Face, Edge and Vertex mode.

The first row of images shows using *Merge Vertices* to close up a hole in a mesh.

The second row shows *Merge Vertices* applied to an edge.

The third row shows *Merge Vertices* used to collapse a face.



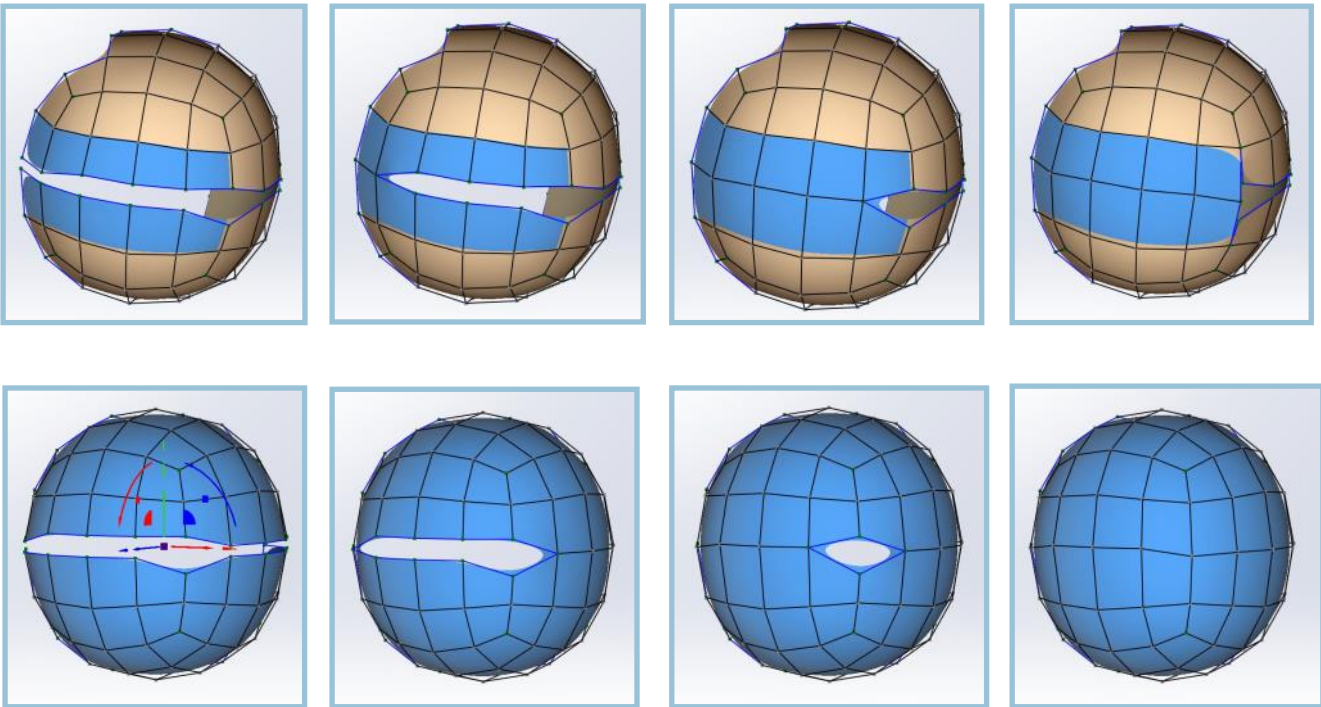
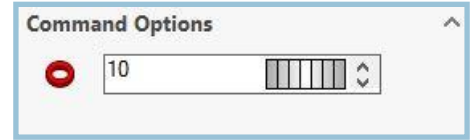
Sew

Sew (or Knit) is an interactive command that allows the user to bring together edges and merge vertices which are close to each other but not necessarily exactly on each other. It is like a *Merge Vertices* command that accepts a bunch of vertices and utilizes a sewing tolerance.

Available in Face model.

The top row of images demonstrates sewing at a value of 10, 12, 18 and 23 respectively. The higher the value the larger the sewing tolerance.

The bottom row of images demonstrates a common workflow where all of the faces are selected and the tolerance value is increased.





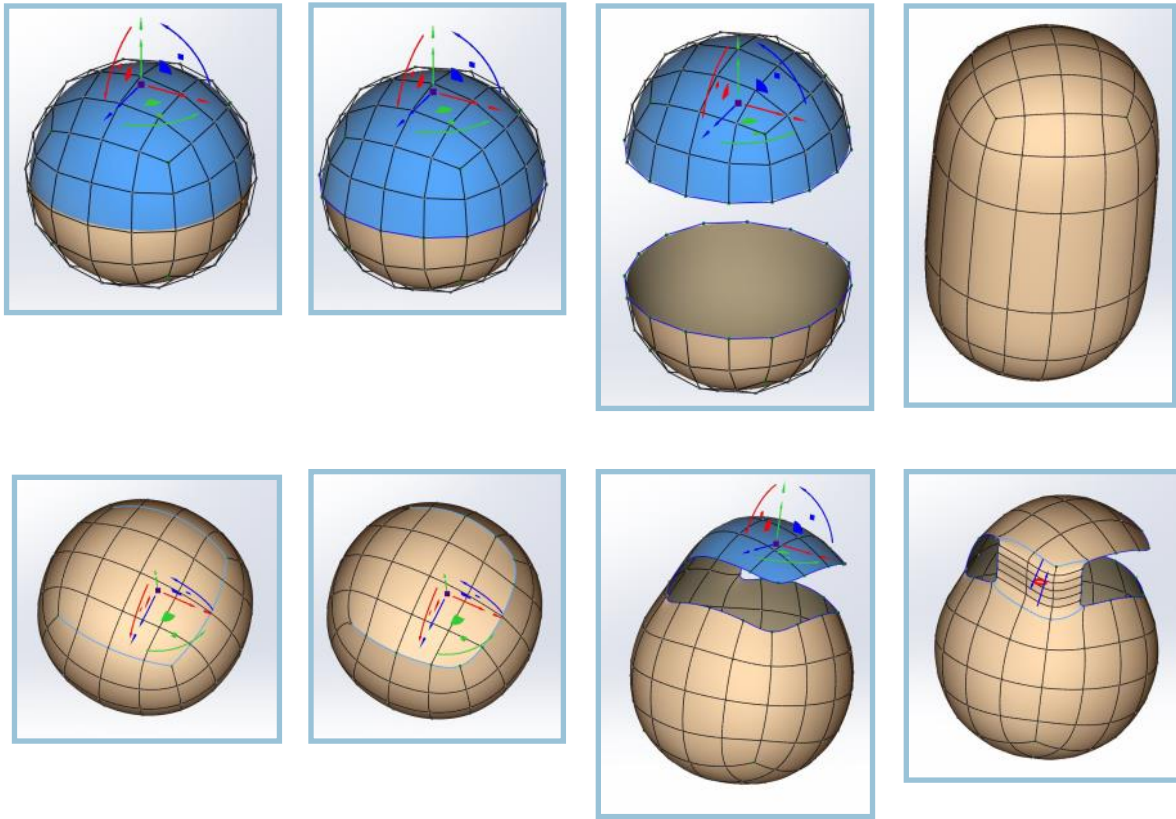
Unsew

Unsew takes selected edges or faces and disconnects them. If edges are selected, the selected edges become open edges. If faces are selected, the boundary around the faces becomes open edges.

Available in Face and Edge mode.

The top row of images demonstrates a workflow of selecting faces, unsewing them, moving them and then bridging between them to create a modified shape.

The bottom row of images demonstrates an edge selection workflow where the faces are never completely detached.

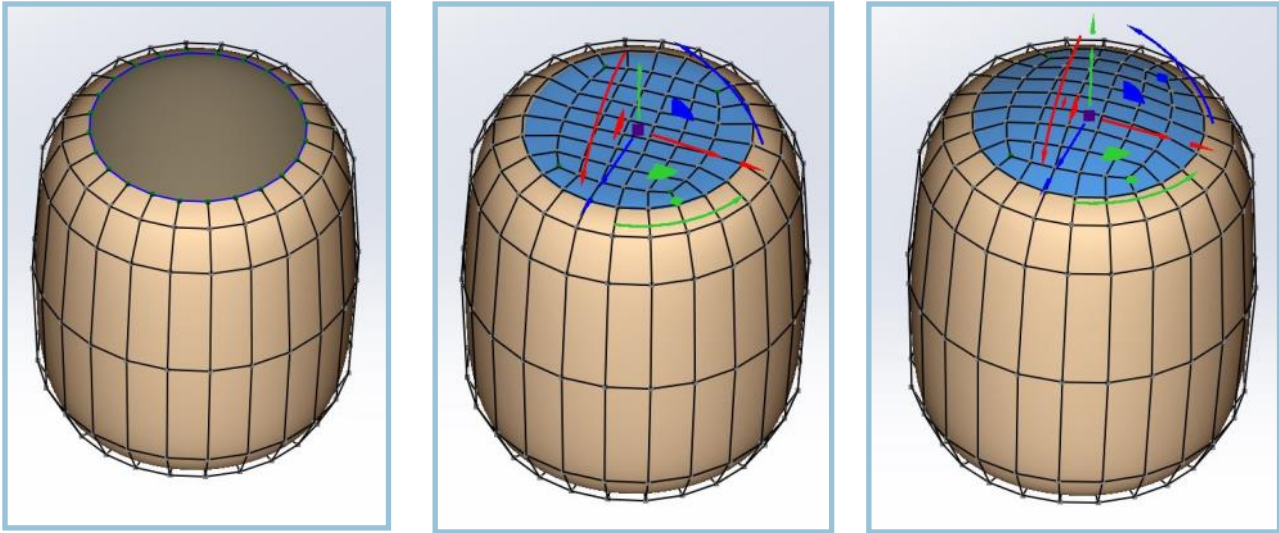


Quad Fill

Creates a set of mostly quad faces to fill an opening. Only one of the open edges need be selected to fill the hole.

Available from Edge or Face mode.

The row of images below show a *Quad Fill* to close an opening and then a *Curve* command to curve the faces created by the *Quad Fill*.



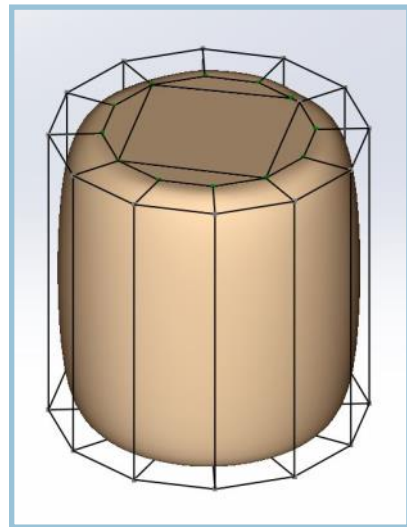
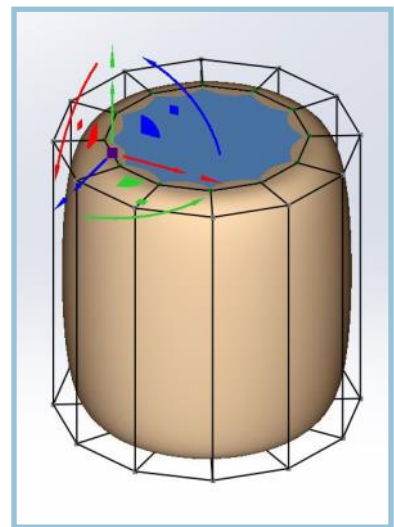
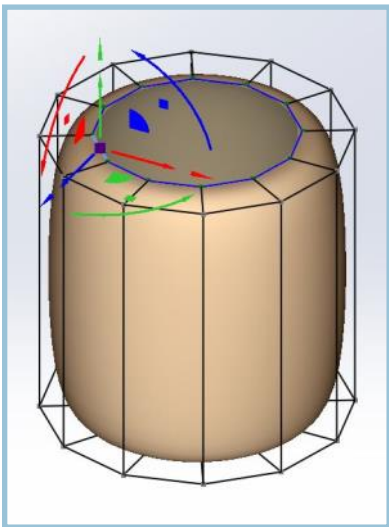
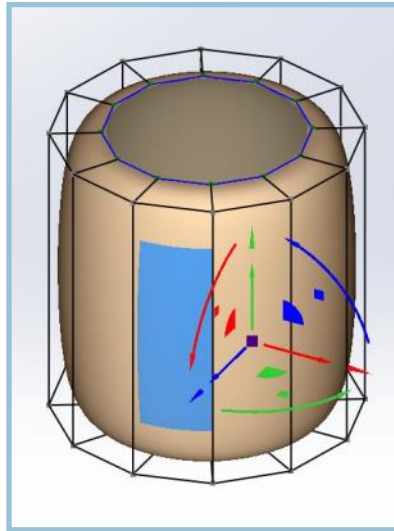
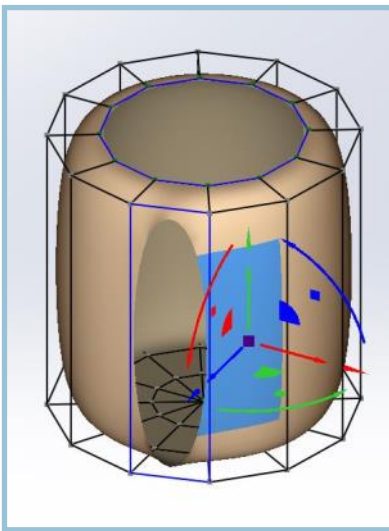
Face Fill

Creates a face to fill an opening. Only one of the open edges or a face on an open edge needs to be selected to fill the hole. The resulting multi-sided face may need further editing to produce quads.

Available from Edge or Face mode.

The first row of images demonstrates filling in a 4 sided hole with quad face by selecting an adjacent face. Edge selection would have done the same thing.

The second row of images demonstrates filling a many sided hole and then using the Insert Edges command to create quads on the 12 sided face.





Insert Edges

Insert Edges allows you to create new edges by clicking on existing edges, faces and vertices. It is a continuous mode allowing you to string edges together. The first time you hit right click, it will allow you to start at a new position. The second right click will exit the mode. It will create vertices anywhere it crosses an existing edge and will automatically snap to vertices and edge centers. Zoom in if the snapping is too strong.

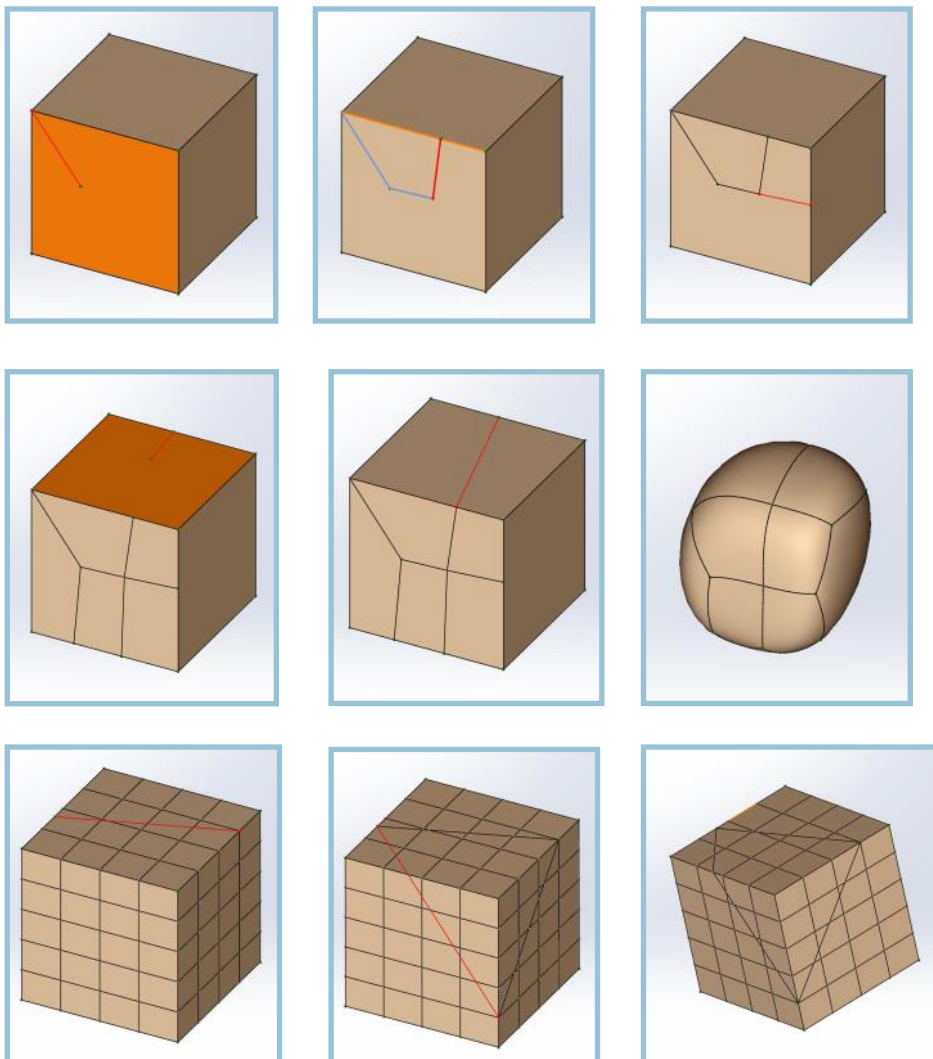
Note that Insert Edges always puts the display into Control Polygon mode. This enables snapping to work properly.

The Face, Edge or Vertex onto which the vertex will be inserted will be displayed in orange.

The first row shows a vertex going into a face; a vertex going into an edge and a vertex going into another vertex.

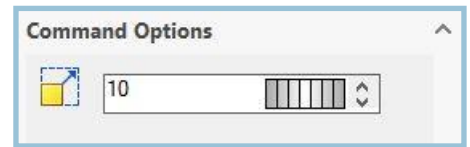
The second row shows right click and start a new edge string on the other side of the top face and connect it back to a vertex on the front face.

The third row demonstrates how Insert Edges works over the top of multiple faces even if there is a sharp corner by projecting down using the viewport orientation.



Relax

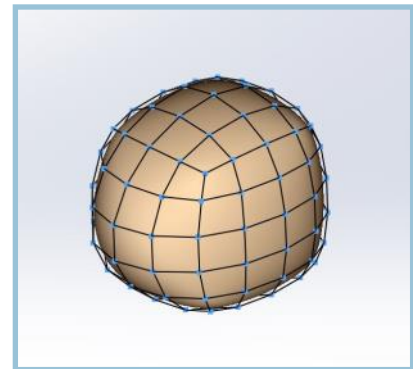
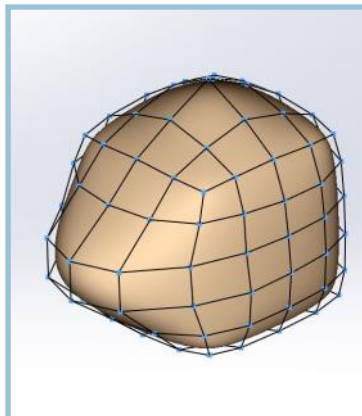
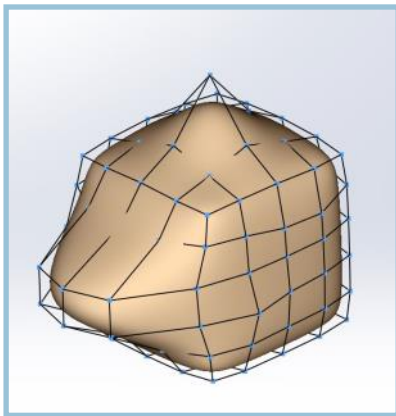
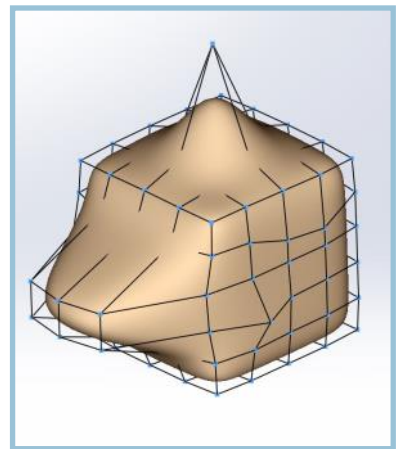
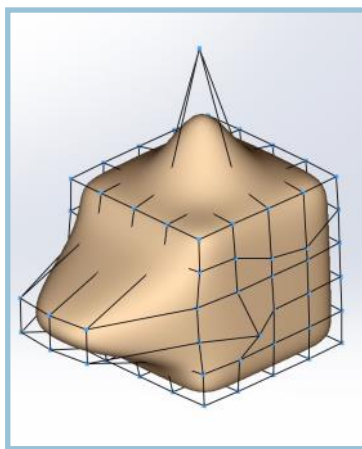
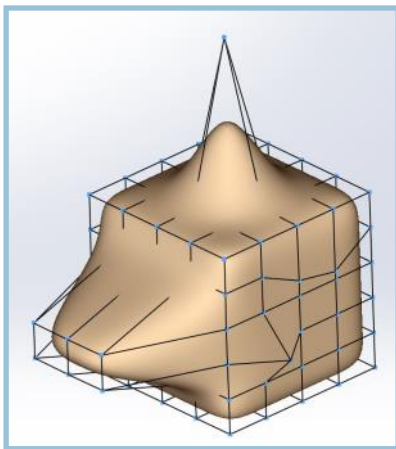
Relax is an iterative mesh smoothing tool. Each iteration of the smoothing command moves the selected vertices closer and closer to the average of its surrounding vertices. The effect is that the mesh relaxes into a smoother and smoother state. Relax is an excellent tool for evening out wild things in the mesh. This tool often is used in the context of large meshes to make them smoother. This tool can be used on all of the vertices or just a selected subset of them.



Note 300 is the maximum number of smoothing iterations that will be performed at one time. If additional smoothing is required, you may restart the command.

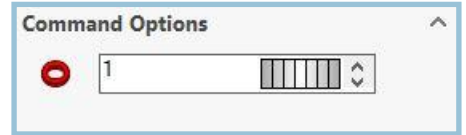
The first row of images show the mesh prior to smoothing; 10 iterations of smoothing (the default); and 20 iterations of smoothing.

The third row of images show 50, 100 and 300 iterations of smoothing.



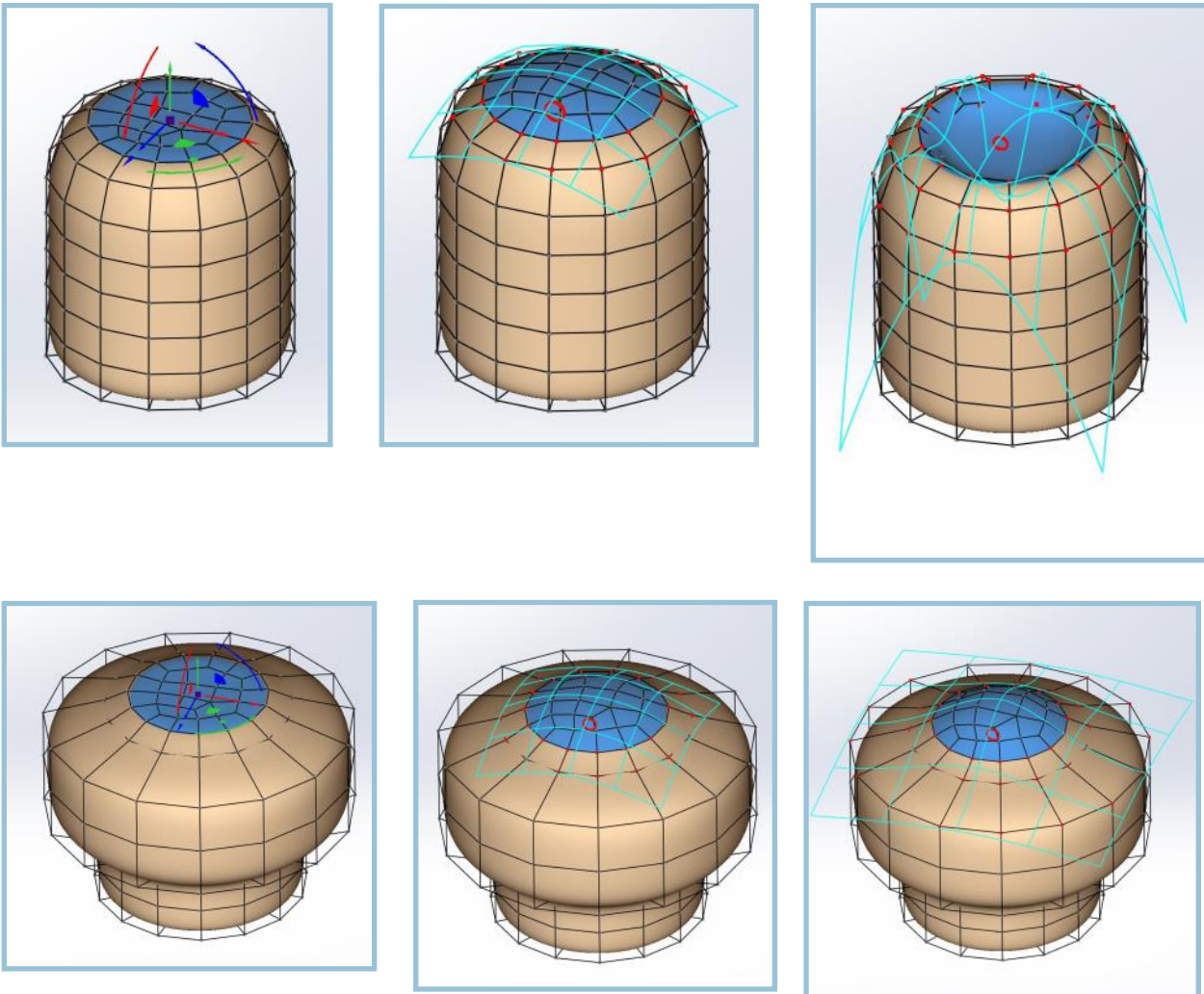
Curve

Curve allows you to match a set of selected faces with the curvature of the surrounding faces. Curve fits a surface to vertices of one or more rings of faces around the selected faces. Typically this will be one or two. If you use a larger number make sure that the points don't wrap around and produce too large of an angle, or the surface will become unpredictable.



The first row shows a Curve example before curving; curving with 1 ring; and curving with two rings. Note that two rings produce a curvature that goes inside.

The second row shows the same progression with a different location for the second row of faces. Note this causes the Curved faces to have additional curvature.

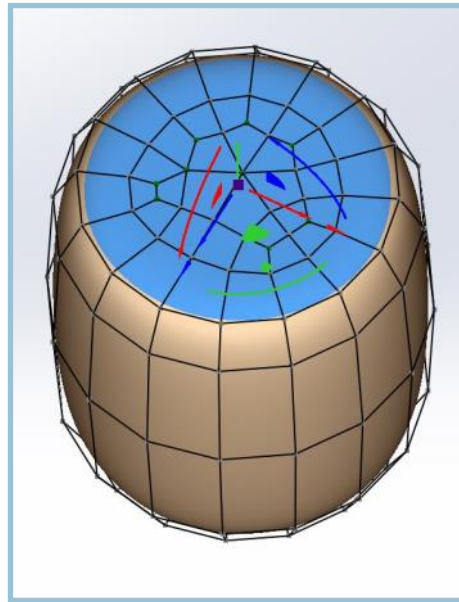
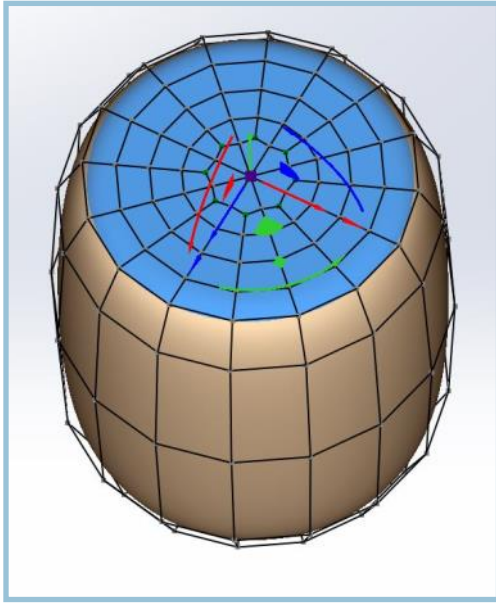




Quadize

Quadize takes groups of selected faces and tries to make a new set of quads on those faces that has a more optimal shape and has mostly quads. This tool is good for cleaning up poor topology.

The following images show a before and after for the Quadize tool.



Level of Detail

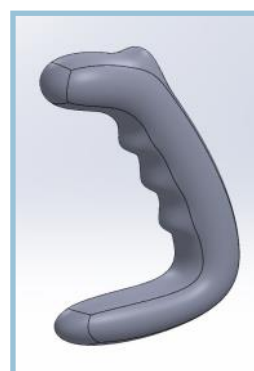
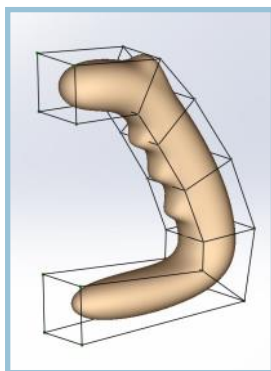
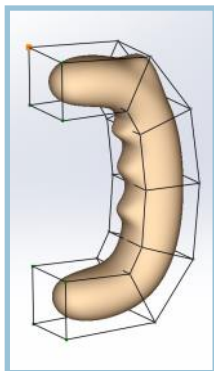
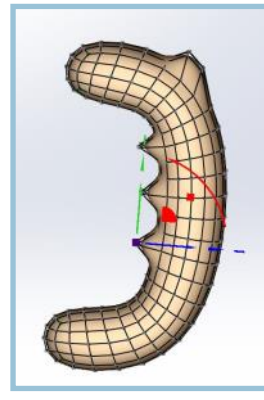
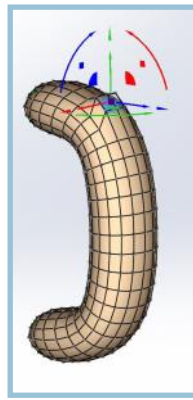
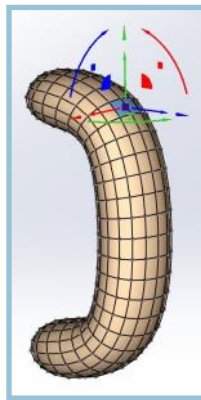
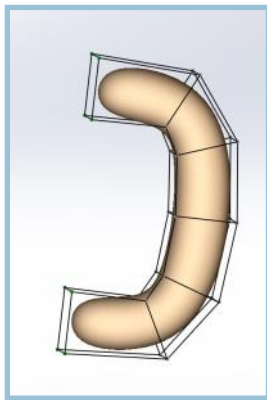
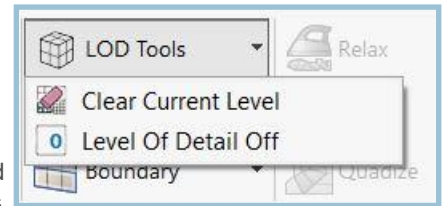
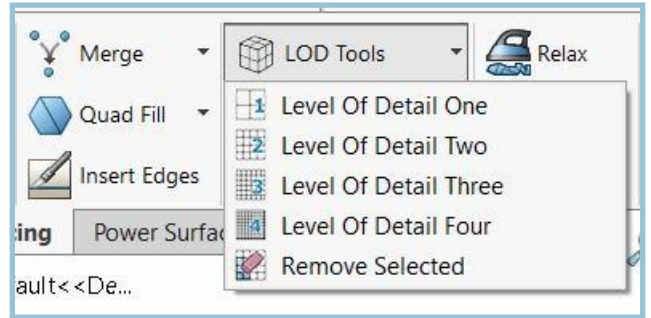
Level of Detail (LOD) provides you with a means to add more detail to the SubD without losing the ability to make broader changes to original version. It essentially adds up to four Subdivide Alls to the SubD while allowing editing at any level individually. The base level is considered to be level 0.

Be aware that aside from the base level, the level of detail geometry (vertices, edges and faces) can *only* be transformed (move, rotate, scale). You may *not* use any tools that change the vertex count of the SubD (Weld, Insert Edges, Extrude, Offset Loop, etc.).

When you first use Level of Detail, the fly-out will give you a choice of level to work in, (see figure to right top). Once you are in LOD mode you can clear the current level or turn off the level of detail and return to level 0.

The images below depict a typical LOD workflow. The first image in the top row is the original handle shape. The second image is after *Level Of Detail Two* command. The third image shows putting a thumb support on the handle by dragging two faces up and over. The fourth image shows putting finger grips on the model by selecting two edges and pulling them toward the center.

The first image in the second row shows the model in the original editing mode after *Level Of Detail Off*. The second image shows an edit of the top level. Note how the LOD remains constant relative to the local face normal. The third image shows the converted Solidworks body.



Level of Detail (continued)

An important feature of the LOD is the fact that the encoding of the LOD modifications are encoded in the local frame of each Face. That means that the LOD modifications will maintain their relative shape even with the top level faces are rotated.

The description of each of the tools is as follows:

Level Of Detail One/Two/Three/Four

These commands choose which level to edit. Note that very fine edits can be made at levels Three and Four. See row 1 below for our model shown at each LOD starting at Level Zero.

Remove Selected

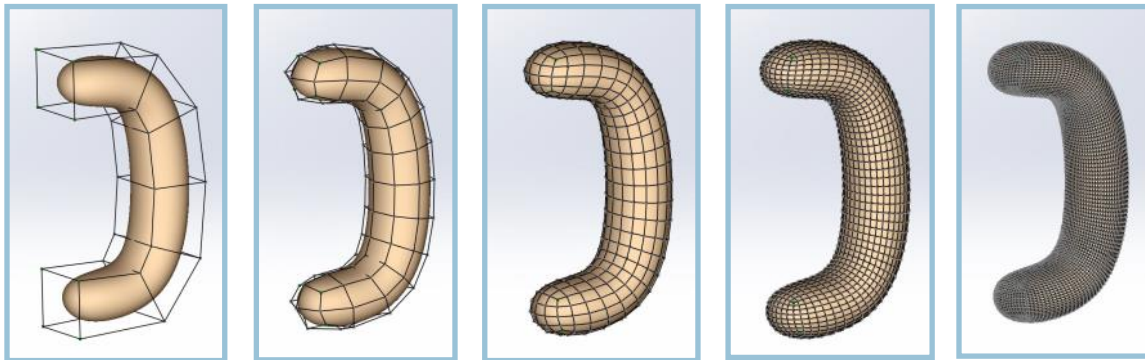
Remove the LOD from the selected Faces, Edges and Vertices.

Clear Current Level

Removes all LOD in the current editing level. You must be in one of the levels to enable this command.

Level Of Detail Off

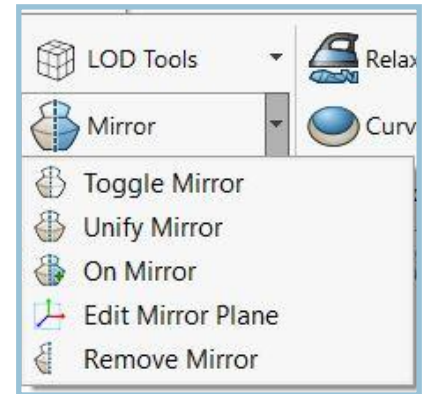
Saves the current LOD edits and returns to the original editing level or Level 0.



Mirror

Creates a mirror image of the model using the selected edges or faces as a guide. In general it will snap the mirror plane to the closest orthographic plane (x, y, or z) for its orientation. Note that the green edges are edges which are constrained to the mirror plane. After you do a mirror, only one side of the mirror is editable. The other side simply reflects the original side.

Available from Edge and Face modes.



Toggle Mirror

Toggles the visibility of the mirrored side of the model on and off. Editing done while the mirrored side is hidden will be seen once the hidden side is toggled back to visible.

Unify Mirror

Remove the mirror and create faces in the SubD where the mirror used to exist.

On Mirror

Takes selected edges or faces and puts them onto the mirror plane. Note that if you add a face, you may need to delete the face to get it to connect to the other side. See illustrations below.

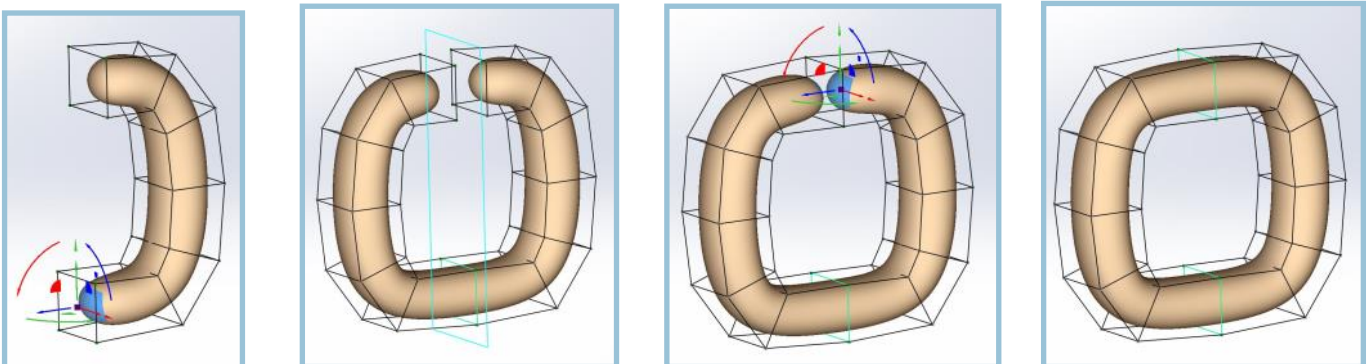
Edit Mirror Plane

This allows you to modify the mirror plane either through a dialog or gizmo manipulation.

Remove Mirror

Removes the mirrored side when Mirror is active.

In the row of images below, the first image shows the original part with the face selected that is to be mirrored. The second image shows the result of the *Mirror*. The third image shows the *On Mirror* with the selection of the top face. Note that all of the edges of the face are now on the mirror plane in the third image. Deleting the face produces the result in the fourth image. It is also possible to delete the face first and then select the loop for the *On Mirror* command and end up with exactly the same result.



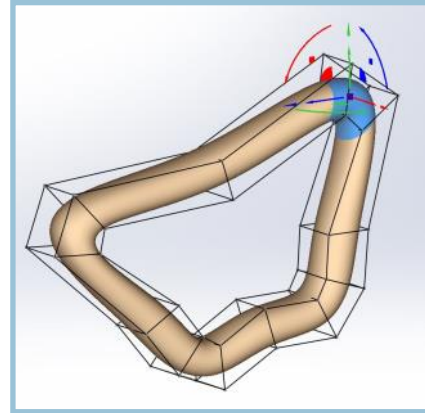
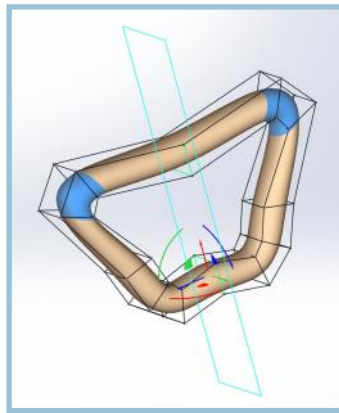
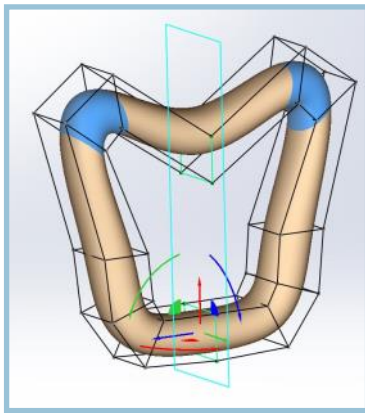
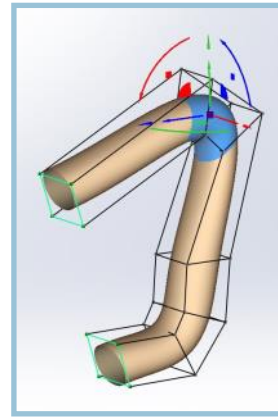
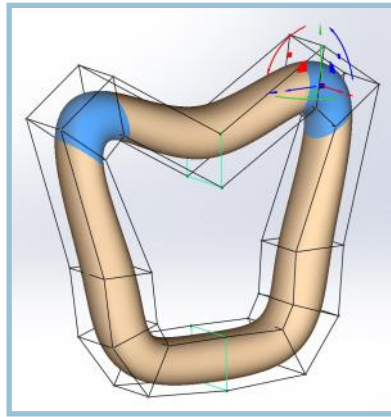
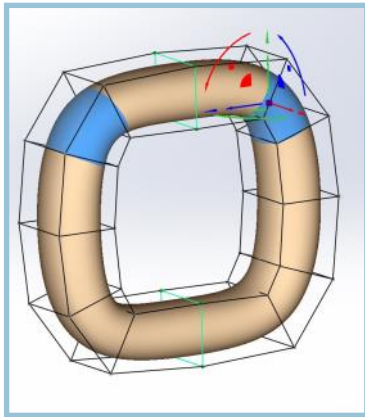
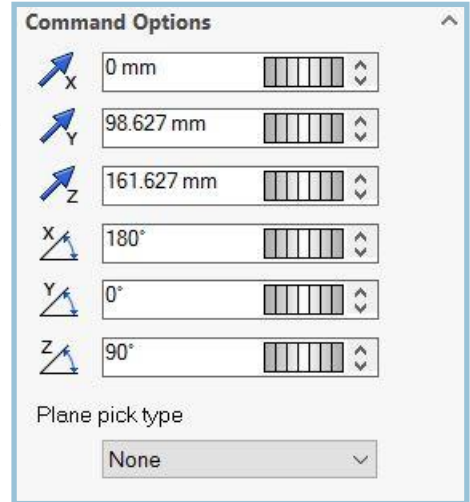


Mirror (continued)

In the top row of images below, the first image demonstrates the highlighting of both the ring of faces on the mirror side and the original side. The second image shows an edit applied to that selection which is reflected on both sides of the mirror. The third image shows the result of *Toggle Mirror* where the mirror side is hidden temporarily until the next *Toggle Mirror* occurs.

In the bottom row of images, the first image shows the result after the *Edit Mirror Plane* tool is selected. The mirror plane is shown in a Cyan color with a transformation gizmo showing to allow you to modify the mirror plane. The dialog on the right allows you to make direct edits to the mirror plane without using the Gizmo. The second image on the bottom row shows a 15 degree Y rotation applied to the mirror plane. The last image in the bottom row shows the result after *Unify Mirror*. Note that only one side is selected.

Note that Mirror with constrained to Solidworks geometry requires you to first Bake . You should bake the to LOD found in the constraints menu.



Boundary

Defines an edge boundary that will be retained in the SolidWorks BREP. This can be helpful in retaining SolidWorks features during repeated editing of the SubD. If you have two face groups selected, they will be part of the same boundary group. Note that green edges will surround the boundary.

Available from face selection mode.

Clear Boundary

Clears the boundary associated with the current faces.

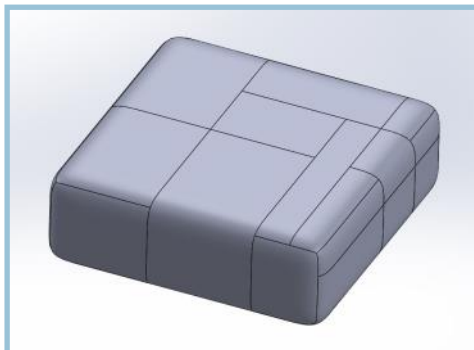
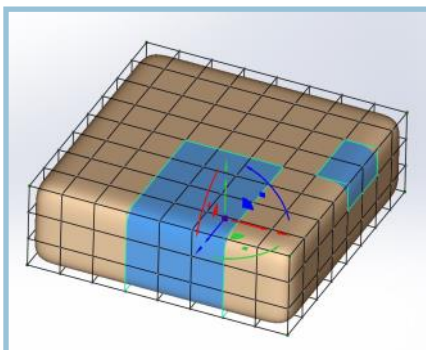
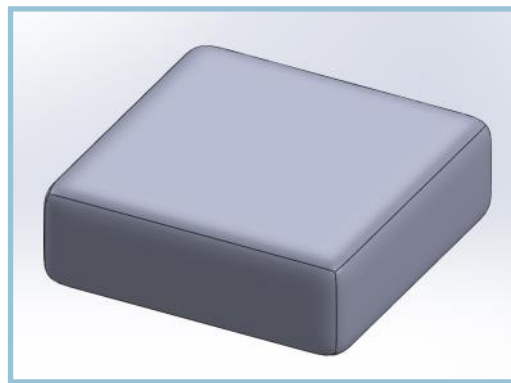
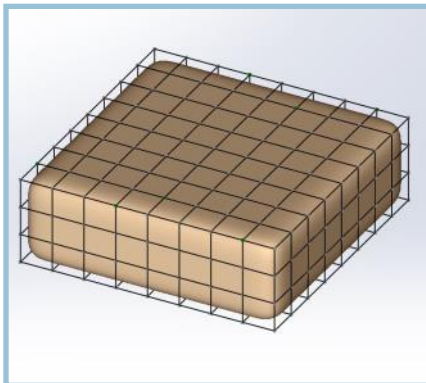
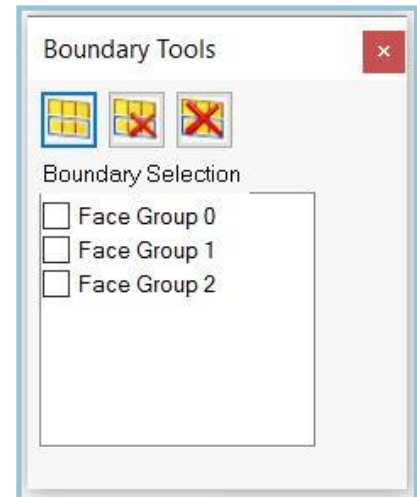
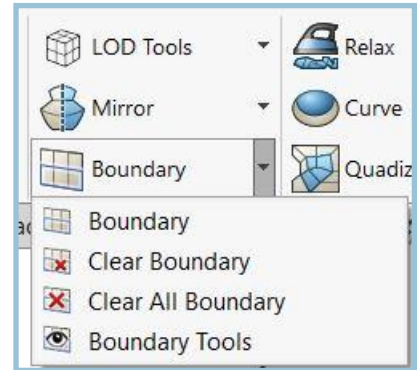
Clear All Boundary

Removes all boundaries from the model.

Boundary Tools

Brings up a dialog (shown to the right) that shows all of the groups of faces for each boundary. This dialog lists the current boundaries. Note that the Face Group 0 is all of the faces. You can select the group and it will select the corresponding faces. If you click the check box it will hide that group of faces. There are also shortcuts to *Boundary*, *Clear Boundary* and *Clear All Boundary* tools in the top of this dialog.

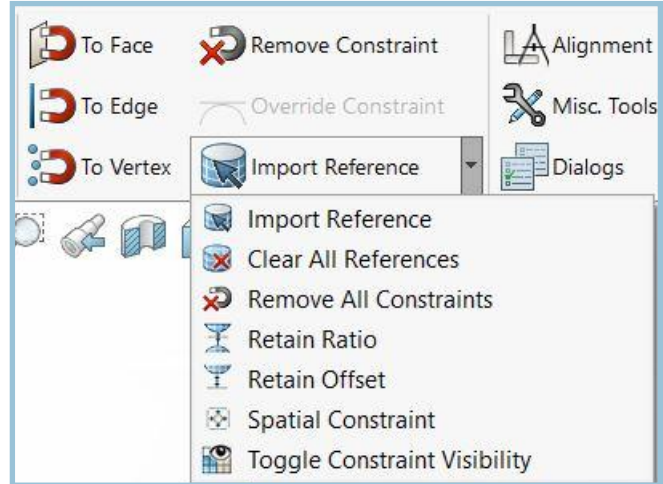
The first row of images below shows the original SubD and corresponding converted Solidworks body. The second row of images shows two groups of faces used to make a boundary and the resulting converted Solidworks body. Note that the green edges in the SubD have corresponding edges in the converted body.



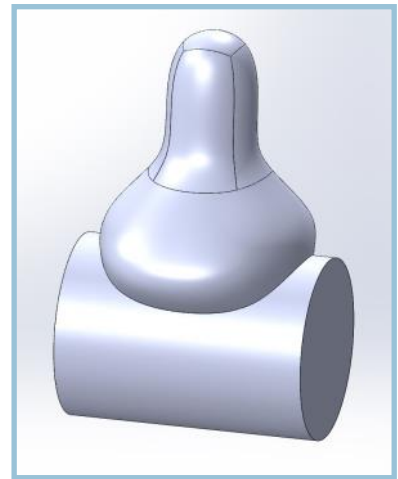
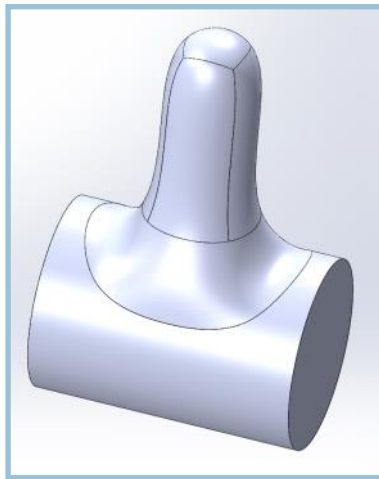
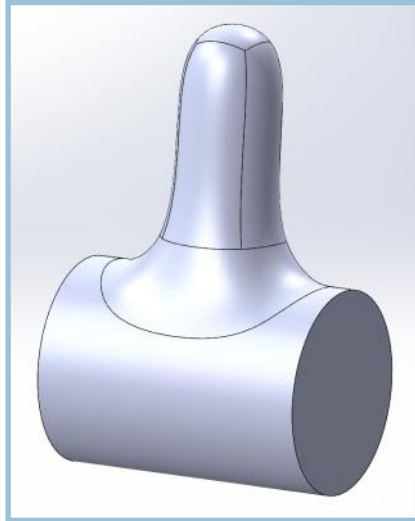
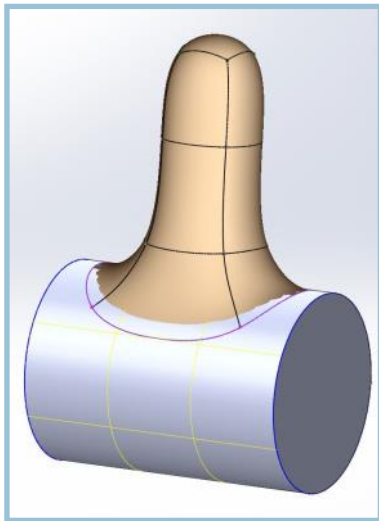
Reference Tools

Power Surfacing provides a way connect and constrain the SubD geometry to existing Solidworks bodies and surfaces. The SubDs can retain a relationship to analytically defined shapes such as circular and ellipse-based cylinders or complex multi-surface elements. These connections may be tangent, perpendicular, at a defined angle or even curvature continuous.

The first image in top row of images below shows the original Power Surface constrained to an extruded circle body. The second image shows a converted curvature continuous body created by the Power Feature using the *Delete and Knit* conversion option. The second image shows the Zebra display and illustrates the curvature continuity achieved.

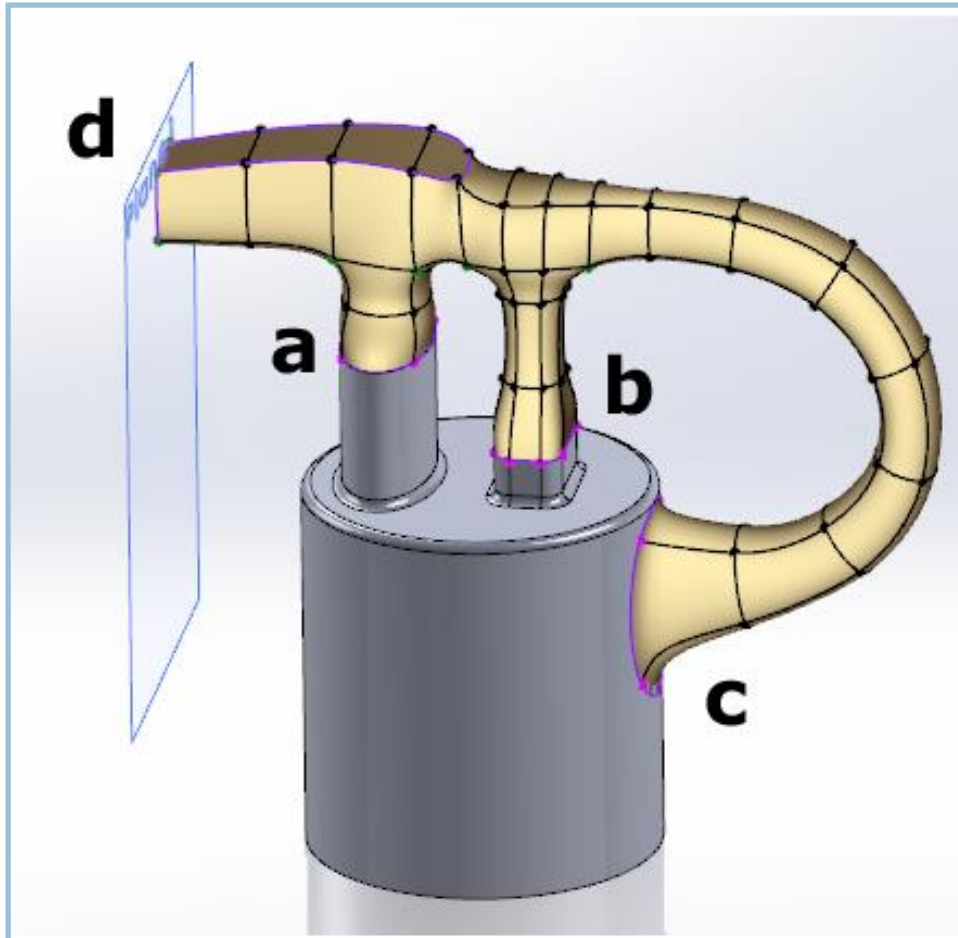


The first image on the bottom shows a Zebra display of tangent continuity. The second image shows no continuity. The third image shows a perpendicular connection to the original body.



Reference Tools (continued)

The image below shows the ability of a Power Surface SubD to connect in multiple different ways to Solidworks. A is a edge constraint on a single edge. B is an edge constraint on 8 different tangent edges. C is a face constraint with tangency. D is a constraint to a datum plane.



Bake to LOD

Occasionally you may wish to bake the modifications made by the constraints into the SubD and remove any reference surfaces. Bake to LOD pushes the modifications imposed by the reference surfaces into the LOD tables. This command is only found in the Tools/Power Surfacing/Constraints menu.

Reference Tools (continued)

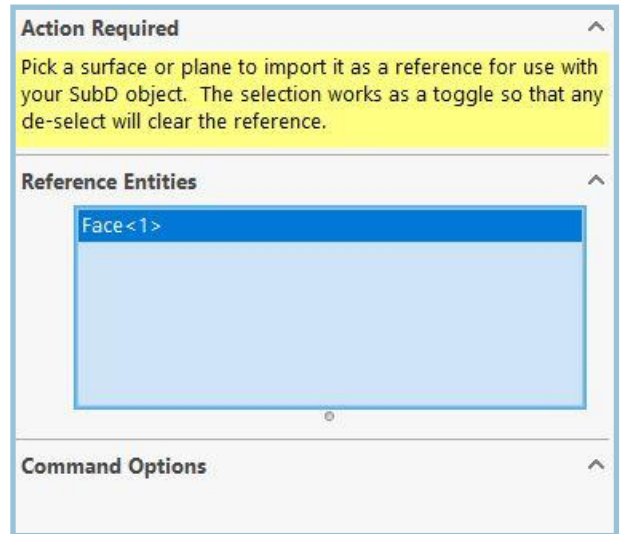
The first tool and only tool initially active is the Import Reference tool. Once references have been imported, the other tools can become active.



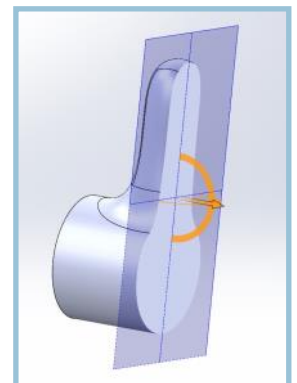
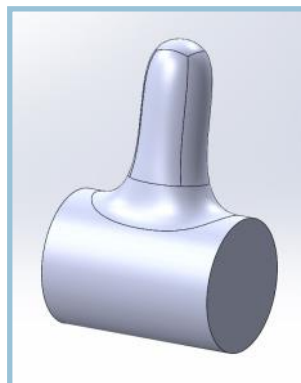
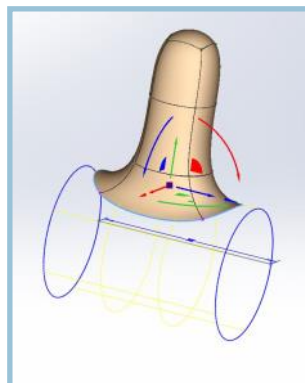
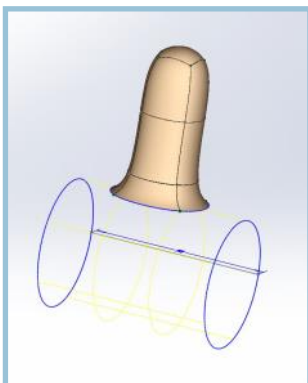
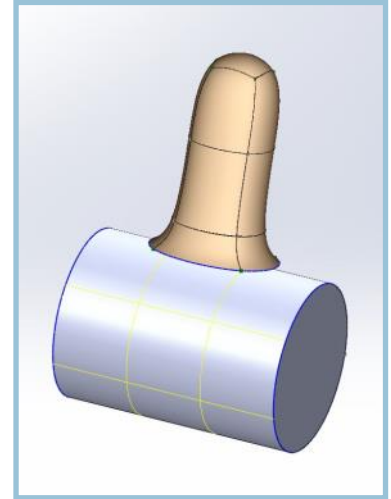
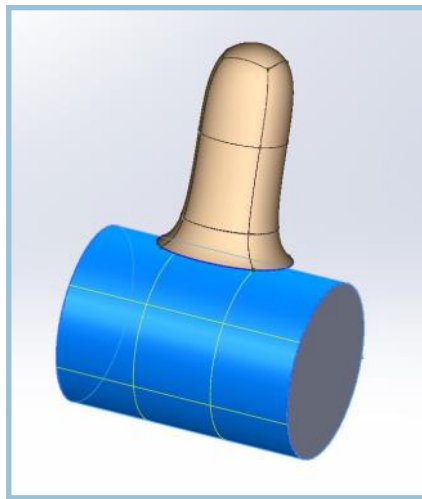
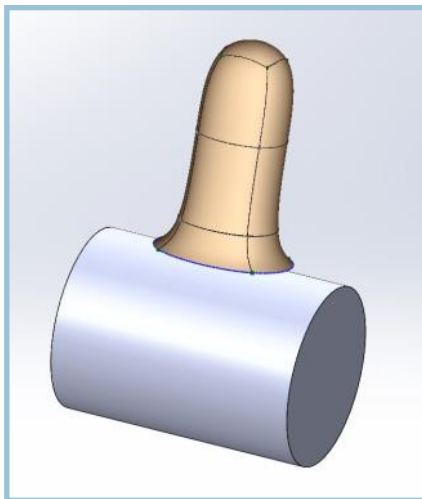
Import Reference

Import reference allows the selection of one or more faces from one or more Solidworks bodies. It is possible to create a SubD which connects two different bodies. The command panel shows the imported reference entities. If you select an already selected face it will deselect it.

The first image in the top row of images below shows what the model looks like prior to Import Reference. The second image shows the selection of the cylindrical face. The image to the right shows the command panel after this selection is made. The third image shows what the model looks like after the accept of the Import Reference. Note that we can now see the imported cylinder with yellow crosshatching and blue edges.



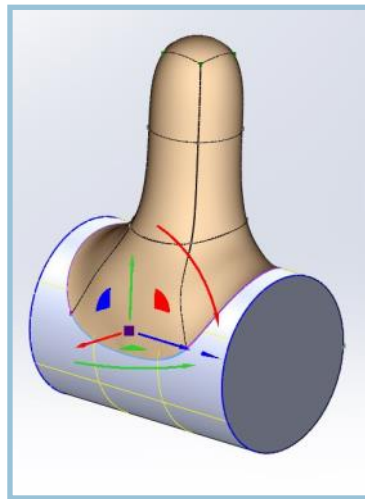
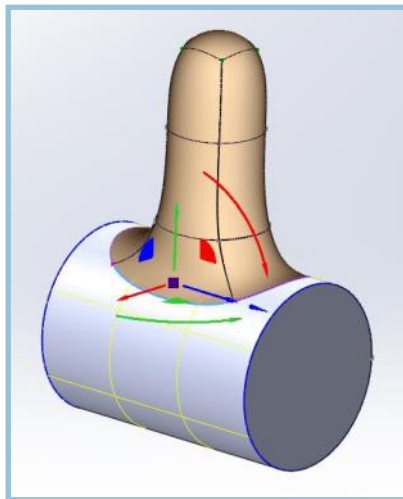
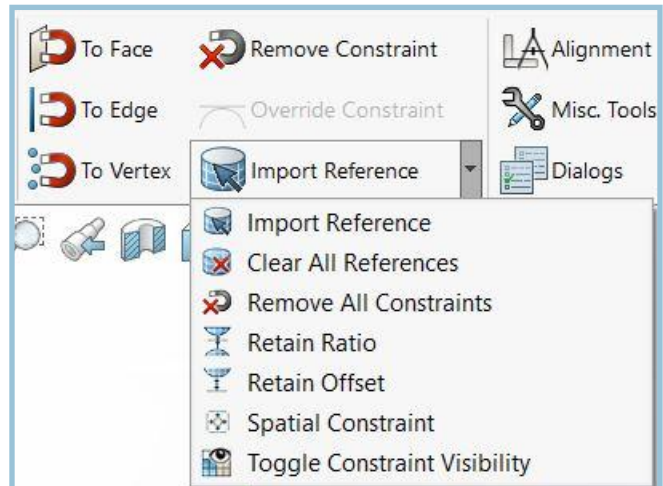
If you use the *Disable Solidworks Blocking* in the *Power Surfacing Options* dialog, you can hide the original body using the history tree on the left hand side of the display. This is shown in the first image in the second row. The second image shows the result after selecting the bottom loop of edges (note that these are open edges) and constraining them to the face using *To Face* tool. The third image shows conversion with the *Delete and Knit* conversion option. This option will be able to create a new solid as shown in the final image in the bottom row.



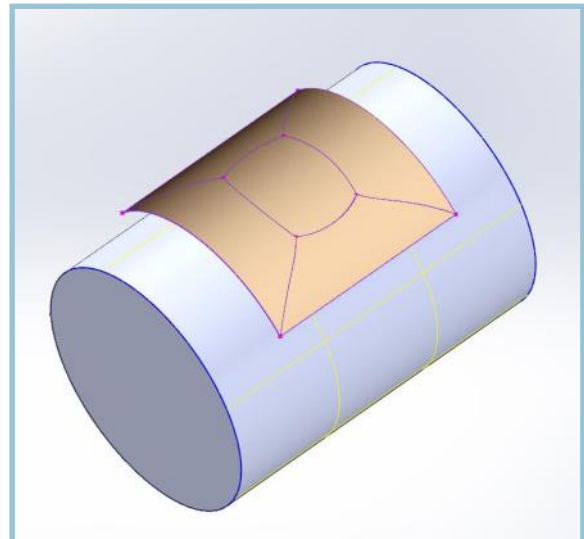
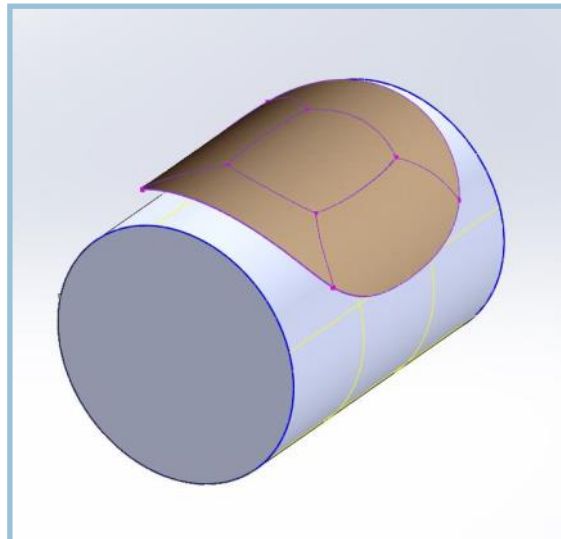
Reference Tools (continued)

To Face

This tool constrains an edge/face or vertex to lie on the closest reference face. Once constrained, the object can be moved around on the face. The two images below show moving an edge constrained to a reference face.



The following two images demonstrate constraining SubD faces to a reference face. Note that we utilized the offset distance in the Override Constraint to move it off the surface a little. The second image below shows the result after making the vertices crease vertices.



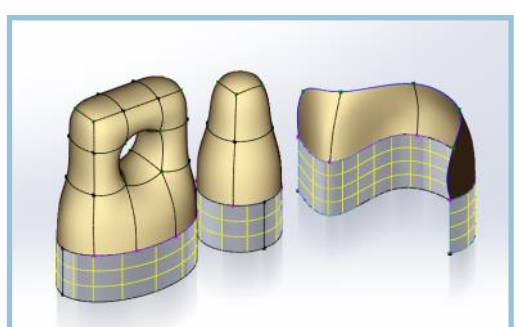
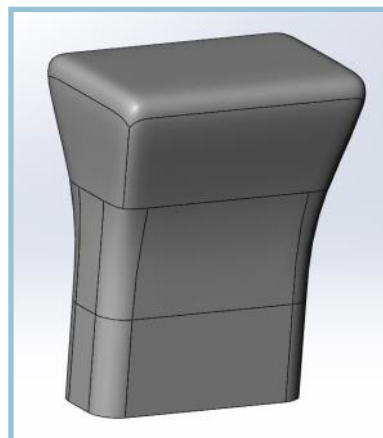
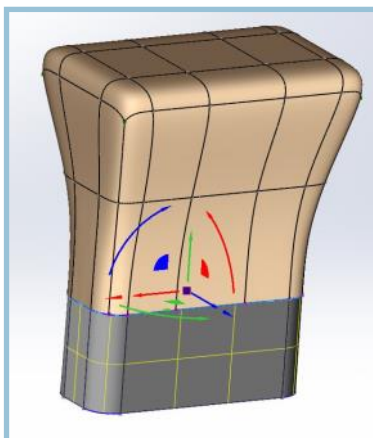
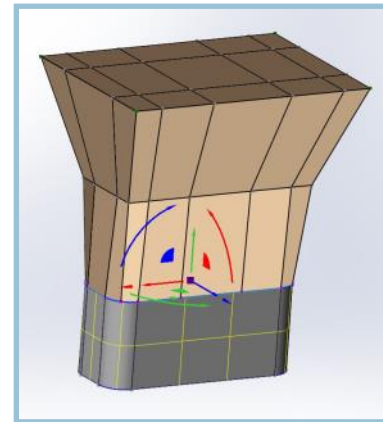
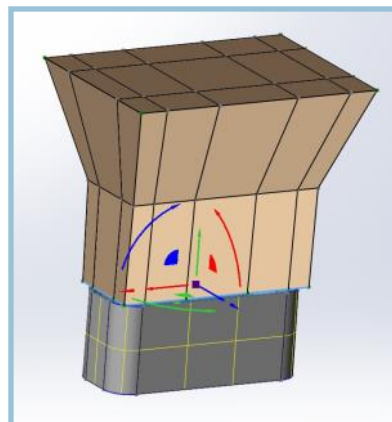
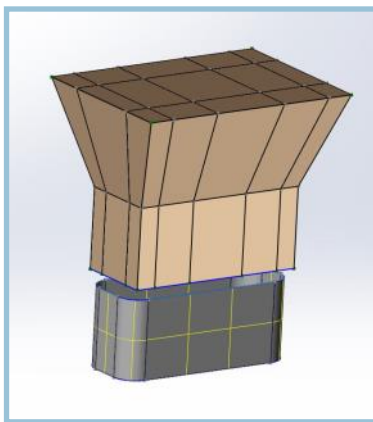
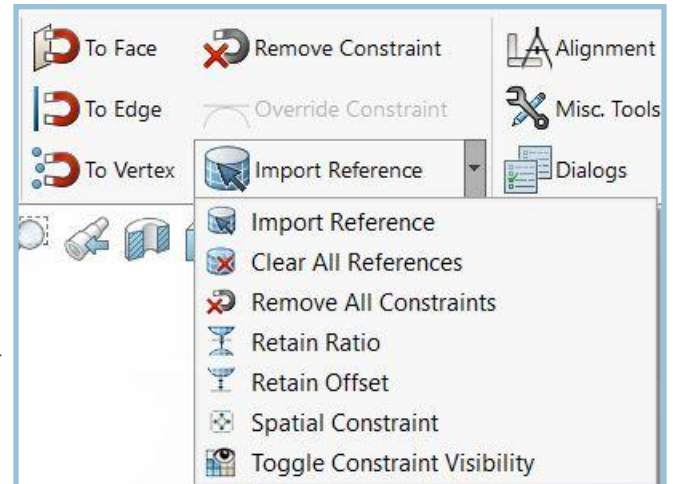
Reference Tools (continued)

To Edge

It is also possible to constrain a SubD edge to edges in the reference geometry. If there is a single periodic edge, such as would be produced by a cylinder, it is not important where the vertices of the SubD fall. However, if there are multiple edges, such as in the example below, it is important to align SubD vertices with reference surface vertices. It is a good workflow to utilize either Cage mode or Control mode while getting the model ready for edge constraints. The control edges need to align properly with the constraint geometry. If this doesn't happen, you may need to remove the constraints and try again after moving things closer to the corresponding reference geometry.

The first image below shows a Power Surface after edge loops have been inserted to align the vertices on the bottom open edges with existing reference vertices. The second image shows moving the open edges into position close to their intended mates. The third image shows the result of the *To Edge* command applied to the entire loop of edges.

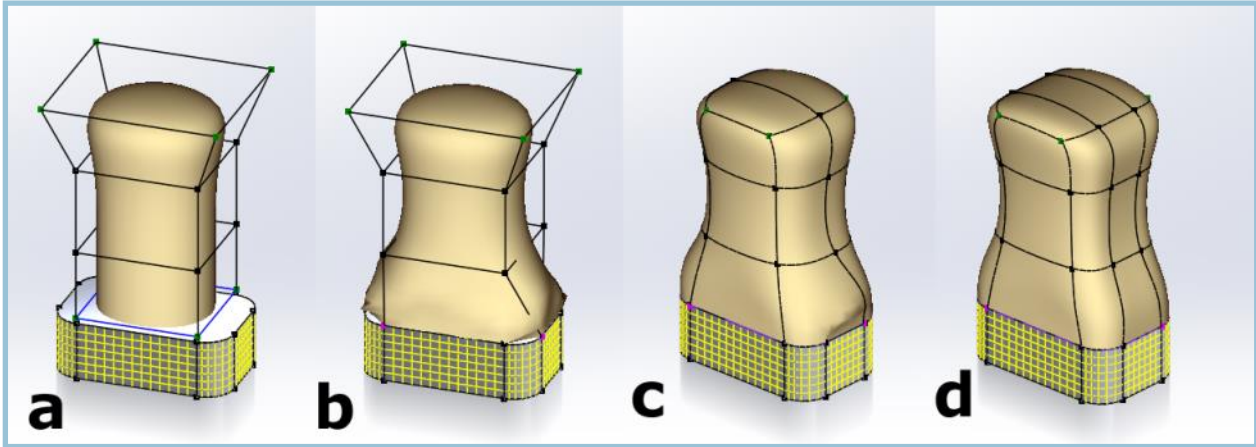
The bottom row of images shows the resulting SubD display and the resulting Solidworks body. Note that *To Edges* is often used when there is a surface with open edges in the Solidworks body or surface. The last image shows some examples of edge constraints.



Reference Tools (continued)

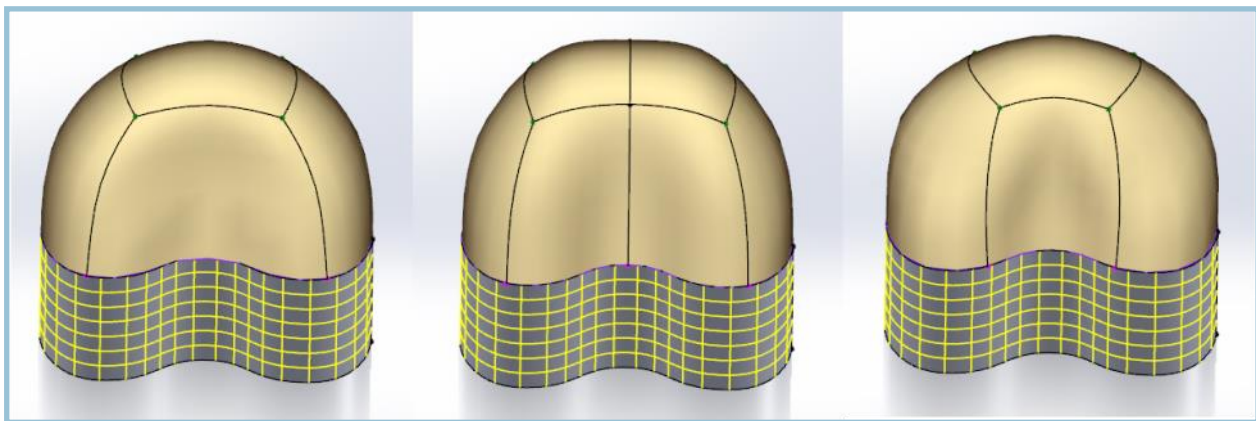
To Edge (continued)

The following sequence of images shows what happens if the SubD vertices don't match with the reference geometry vertices and the process used to fix the issue. The *Insert Loop* and



Draw Edges commands come in handy in cases like this.

The image below shows a case where one subD edge covers too much shape change on the reference geometry. You can see the folding in the middle of the first image below. There are two ways to adjust the model to solve this problem. The first way is to insert an additional loop, as shown in the second image. The other way is to adjust the existing loops by moving them closer to the problem area. The advantage of the second technique is that it doesn't require additional geometry in the SubD. A general rule of thumb is that one edge of a SubD should not



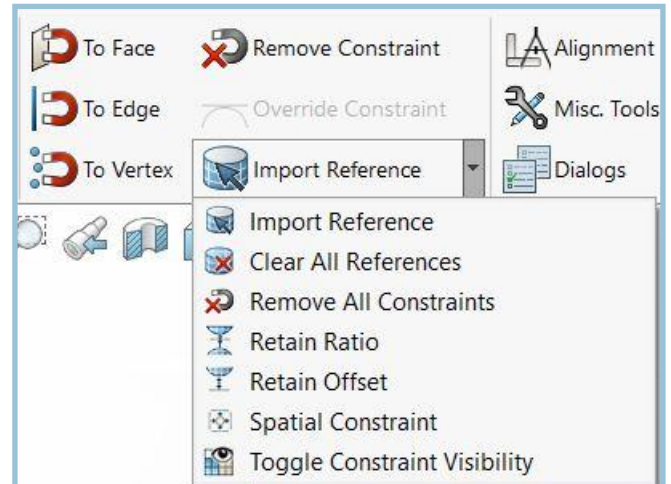
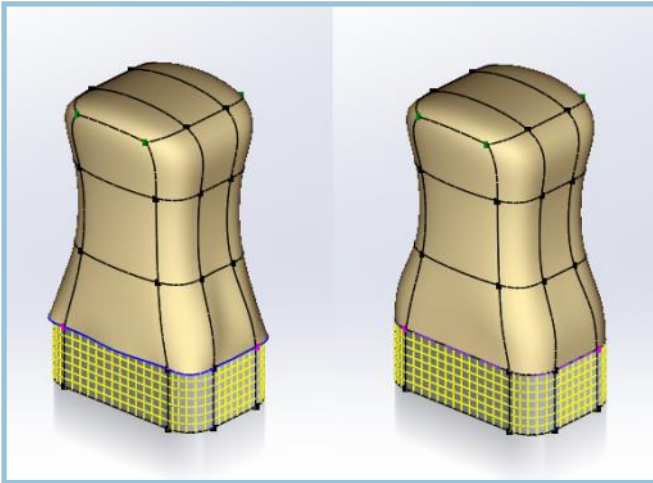
cover more than 90 degrees of curvature in the reference edge.

Reference Tools (continued)

To Vertex

The *To Vertex* tool is often used to either constrain vertices to a surface to keep the resulting geometry in the proximity of the surface, or to start the process of constraining edges.

The image below demonstrates the process of constraining the vertices first then after that constraining the edges. It is a good workflow to make sure all your constraints are constrained correctly.



Remove Constraint

Sometimes you may need to change a constraint or fix an issue if the underlying geometry of the reference surfaces changes substantially. The Remove Constraint tool enables you to remove constraints from selected SubD faces, edges and vertices. If you select a face, Remove Constraint may also remove constraints on the edges and vertices if the constraints are of the same order (i.e. face constraints). Removing edge constraints will conversely remove vertex constraints.

Remove All Constraints

Remove All Constraints removes all constraints and override constraints from the entire SubD mesh. The reference geometry will still remain in place.

Clear All References

Clear All References removes all reference surfaces and removes all constraints from the entire SubD mesh. After this command you will typically need to *Import Reference* again and reapply the constraints to the SubD mesh.

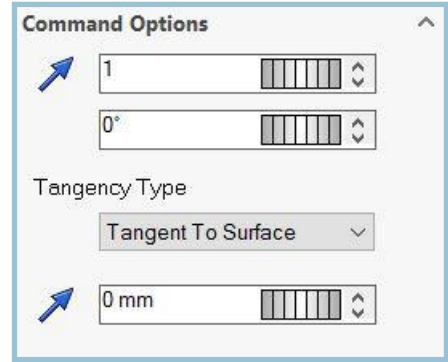
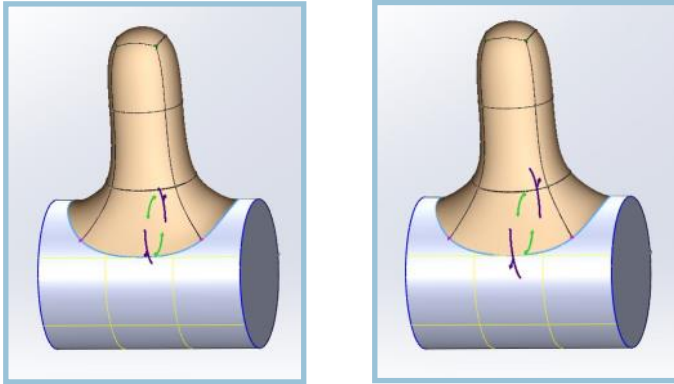
Toggle Constraint Visibility

This tool changes the visibility of the yellow crosshatch and blue edges used to show the reference surfaces in SubD editing mode.

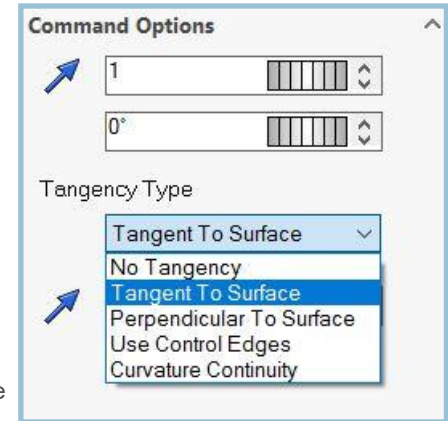
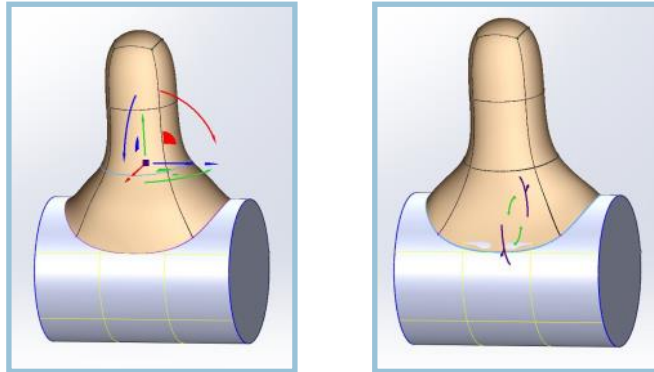
Reference Tools (continued)

Override Constraint

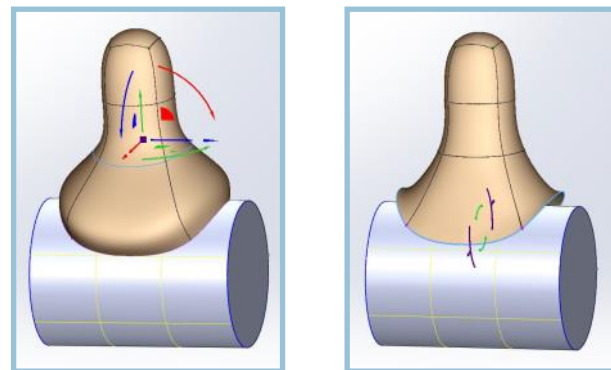
Override Constraint lets you override the default behavior for constraints. The top edit box shows the strength of the tangency. The first images below shows the default tangency which has a strength of 1.0. The second image shows a strength of 0.2. Note how the surface is shallower on the blend.



The first image in the second row shows a rotation angle of 30 degrees applied to the tangency. The rotation angle is the second number box in the dialog. The second image shows Use Control Edges for the tangency control. This is available in the dropdown as shown on the right.



The first image in the third row shows Perpendicular To Surface applied to the edge constraint. The second image shows an offset distance which is controlled by the last number box in the dialog.

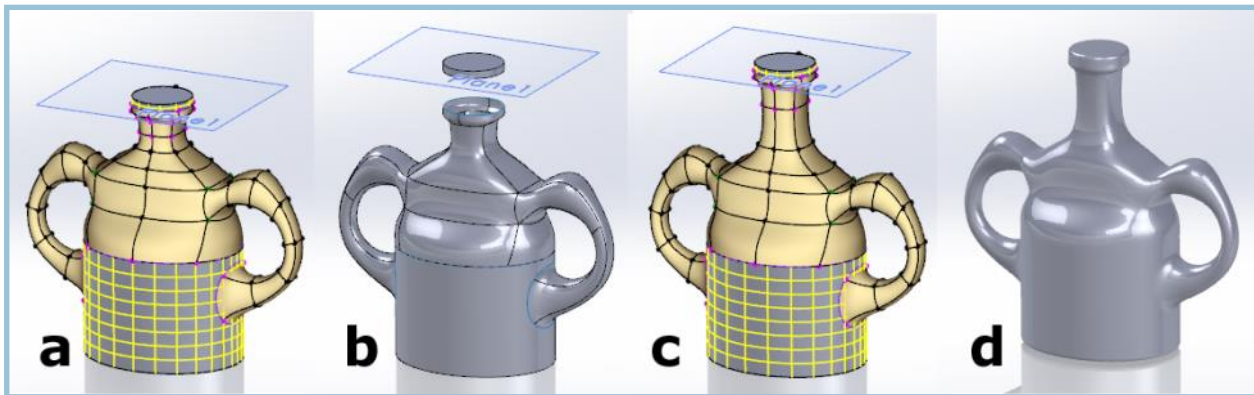
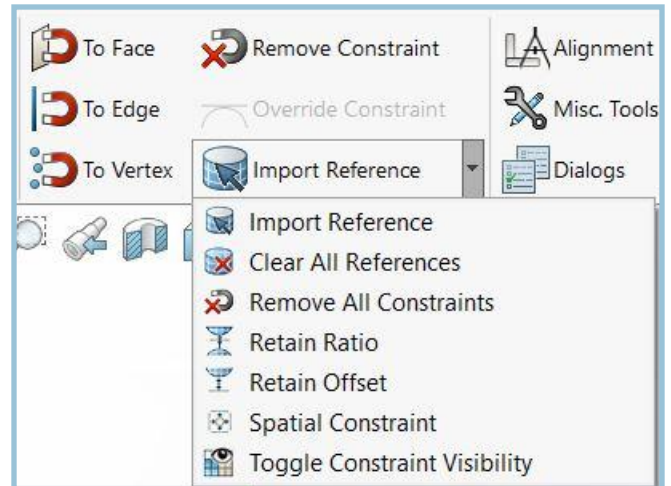


Reference Tools (continued)

Retain Offset

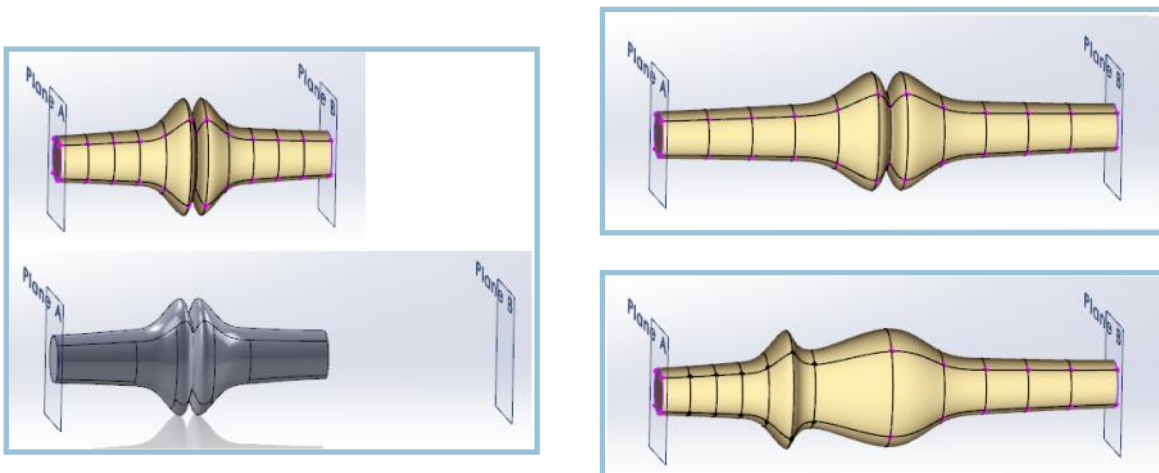
When you want a portion of the SubD to remain intact when a constraint surface is moved or changed, you can specify which parts of the SubD will be affected. Keeping in mind that SubD elements affect their neighbors, you will want to select one edge loop's worth of the SubD beyond your target area.

In the image sequence below, the three loops of vertices right under the plane are set to Retain Offset in image A. As the plane moves all of these vertices will move with the plane as seen in figure C and D.



Retain Ratio

When there are two, roughly parallel, constraining entities, you can choose to have the SubD stretched evenly between the two. As with the Retain Offset option, you must also mark the part of the SubD you wish to be affected. In the image below on the left we set up the Retain Ratio and move the plane. The top image on the right shows the result. The bottom image on the right shows what happens if we only set half of the Retain Offset and the other half to Retain Ratio.

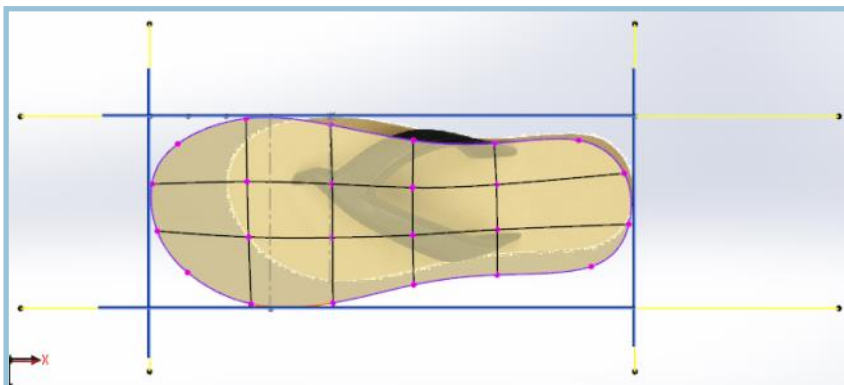
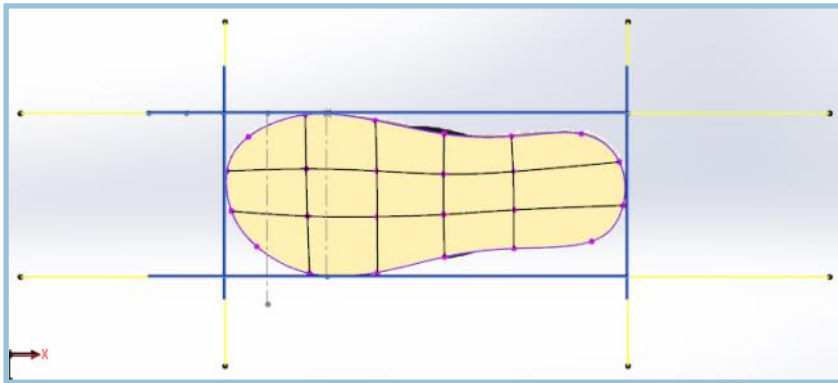
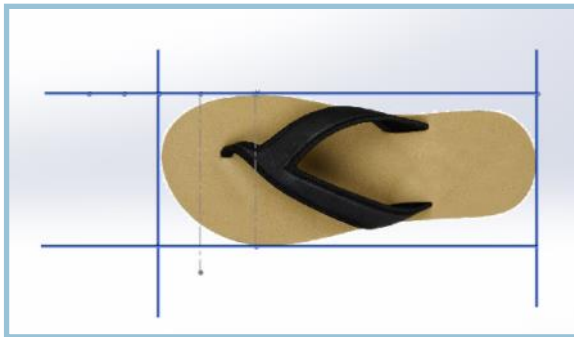
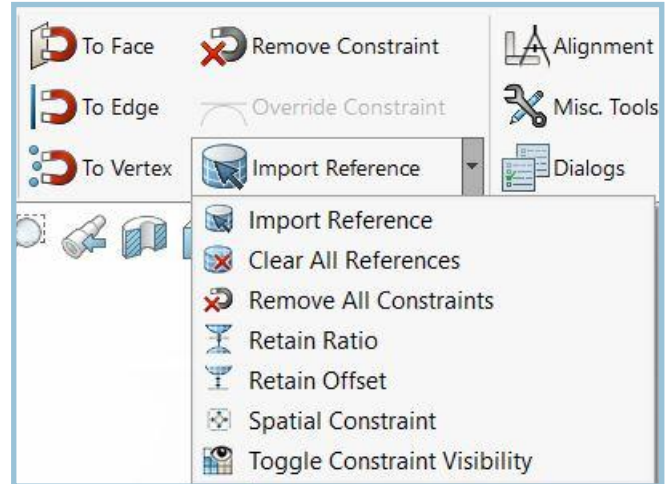


Reference Tools (continued)

Spatial Constraint

The Spatial Constraint is similar to the previous two constraint tools in that the parts of the SubD to which it is assigned are influenced by proximity. With this tool, however, you may *only* use orthogonal planes for the constraint entities. Because it works spatially, it is particularly useful for objects that will have variations that are not necessarily uniformly scaled. With multiple planes, you can easily achieve parametric adjustments in volume. In fact, it is even possible to control the X, Y, and Z coordinates of the points by different sets of planes.

The images below show setting up a default 8 size flip flop and modifying the SubD to a size 12 by changing the planes with spatial constraint set on all vertices.



Alignment Tools

Many of the alignment tools are very much dependent on the order of selection. The second, third, etc. picked objects will typically be aligned to something on the first picked object.

Align in X/Y/Z

Align edges such that they are parallel to the X, Y or Z axis of the world coordinate system.

Align Linearly

This command will make the second, third, etc. selected edges lie on the same line as the first selected edges.

Align Parallel

This command will make picked edges or faces parallel to the first selected edge or face.

Align Perpendicular

This command will make picked edges or faces perpendicular to the first selected edge or face.

Align Equal Length

This command will make picked edges the same length as the first picked edge.

Align Coplanar

This command will make picked faces coplanar with the first picked face.

Flatten

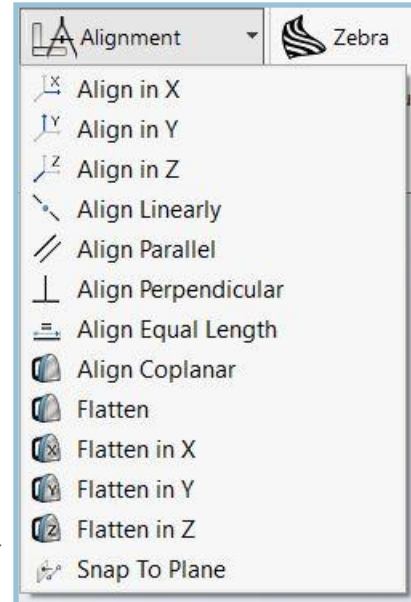
This will compute an average plane and then drop all vertices of the selection to that plane.

Flatten in X/Y/Z

This will flatten all picked vertices to one of the orthogonal planes. The plane selected will be at the average point of the selected objects vertices.

Snap To Plane

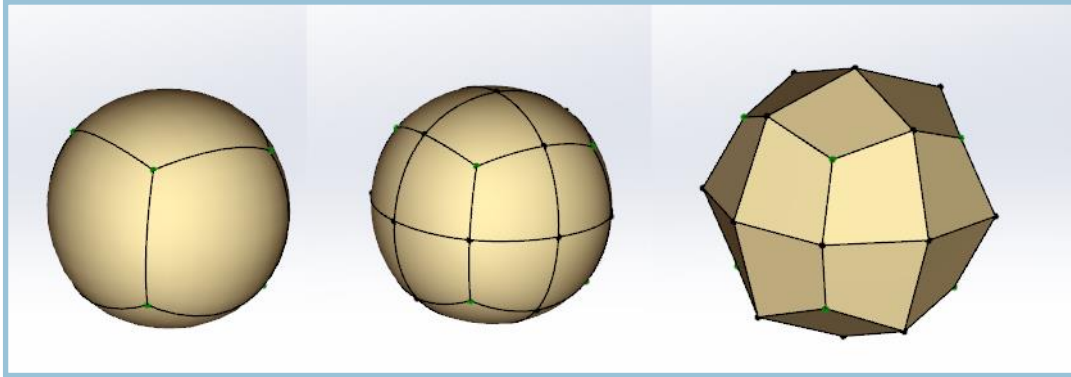
Snap the given objects to a selected Solidworks datum plane.





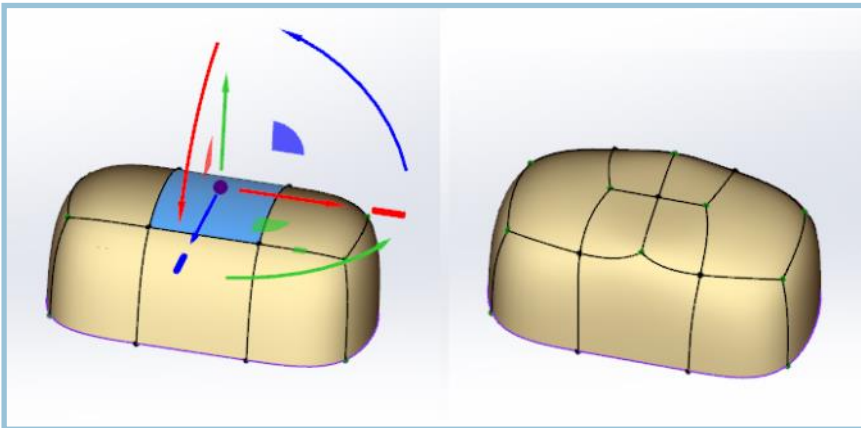
Subdivide All

Subdivides the entire object regardless of current sub-object mode. See the image below for a simple example. Typically you will do this to add additional detail to your model.



Subdivide Selected

Subdivides only the selected faces. See the example below where we have subdivided a single face. You may or may not want to fix resulting 5 sided polygons.



Bend

The bend tool enables you to bend an entire object or just a part of it. Typically you bend from an orthographic view. You start by selecting a line that defines the start of the bend and the direction of the bend. Then you move the cursor to the desired bend angle. The bend angle is shown in the command panel. The Snap Angle allows you to adjust where snapping occurs so you can bend to exact angles more easily. Once the initial bend is done, you can adjust the bend further using the on screen gizmos. Once you are finished, you can click the green check to save the result of the bend.

The first image below shows the bend from an orthogonal view. The first pick point is near the left lower corner of the object and the second pick is near the right lower of the object. As you move the cursor as shown in the second image below, the object bends. The lower image shows bending just a subset of the model.

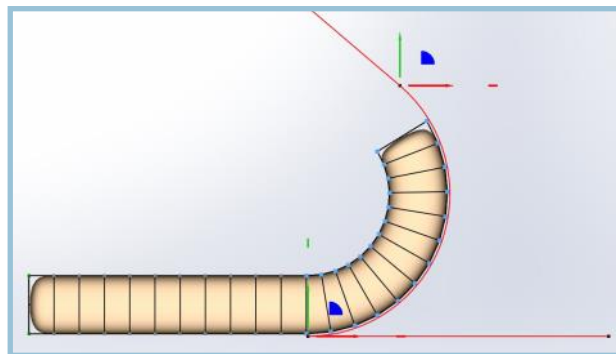
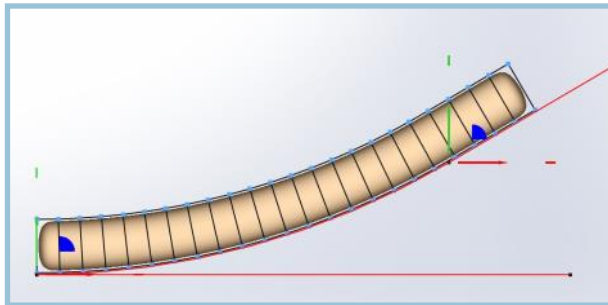
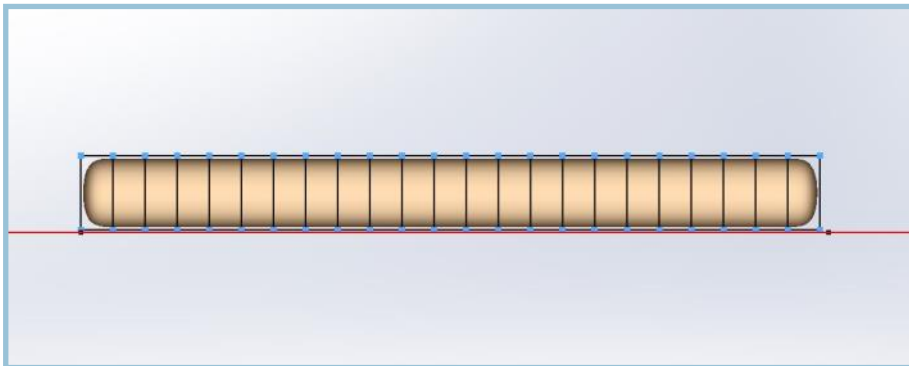
Action Required

Click twice to draw and define the Bend axis. Then, move the mouse to set the desired arc and click when set. Both the first and third points are adjustable. Green check exits mode.

Command Options

Angle 51.252°

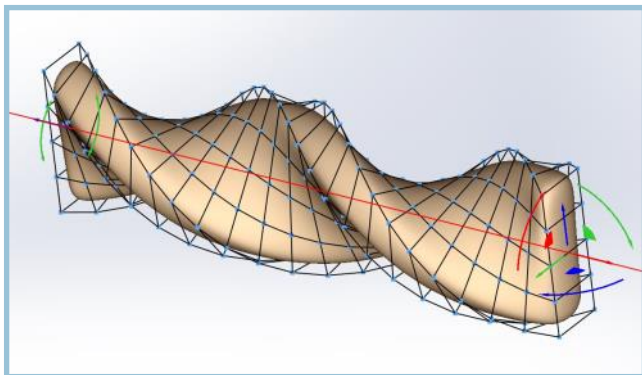
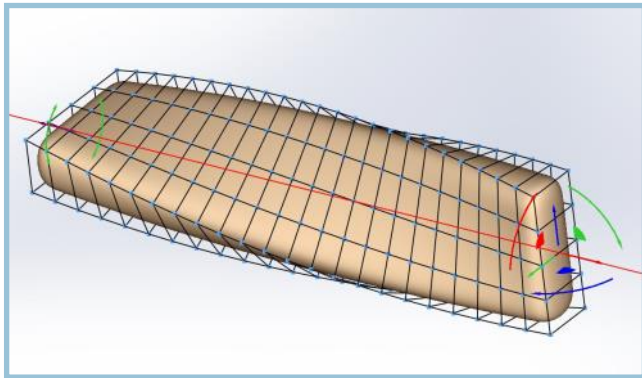
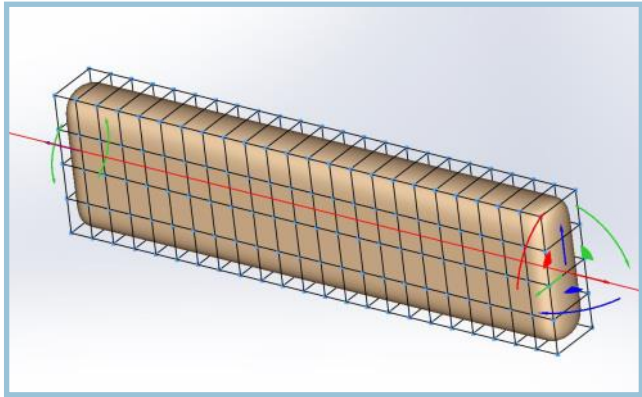
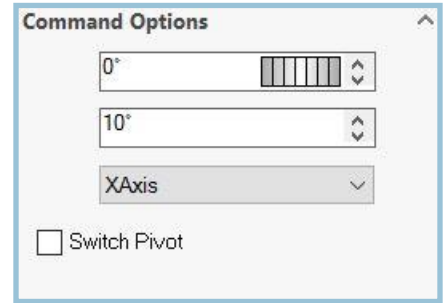
Snap Angle 45°



Twist

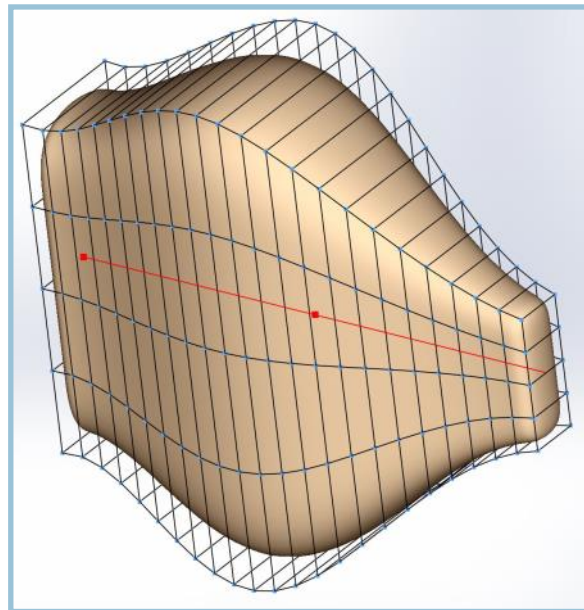
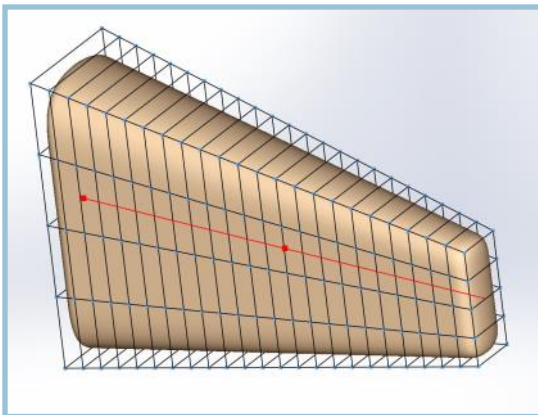
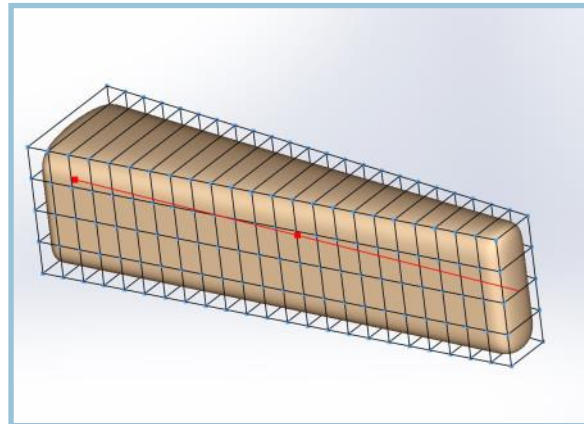
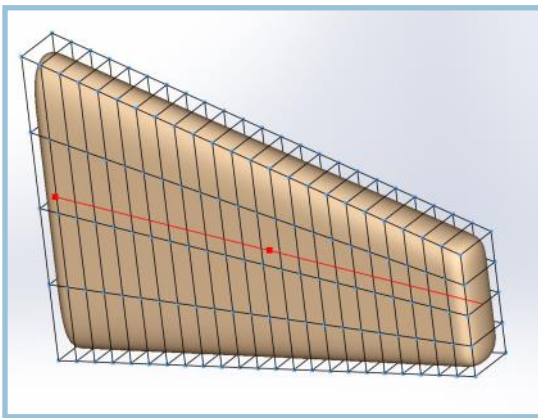
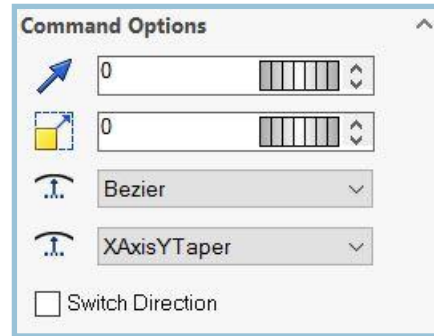
The Twist tool allows you to apply a twist to all or part of a SubD mesh. The first parameter in the dialog shows the twist angle. You can change the twist axis and the end of the twist that stays fixed. Note that if you twist with a large angle you should have sufficient number of subdivisions in the mesh to pick up the twist. For example if you twist a 6 sided box 360 degrees, it will look like it collapses in the middle.

The first image below shows the initial object once the twist command has been started. The second image shows a 90 degree twist. The final image below shows a 360 degree twist.



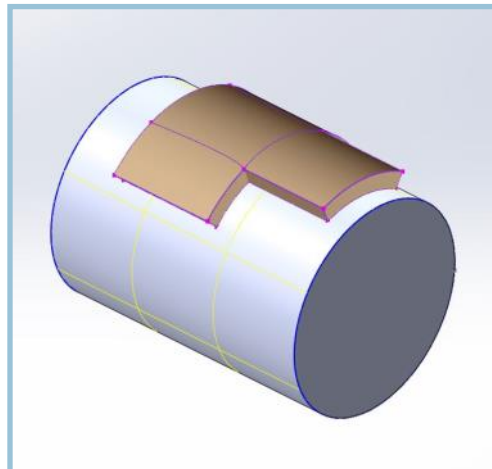
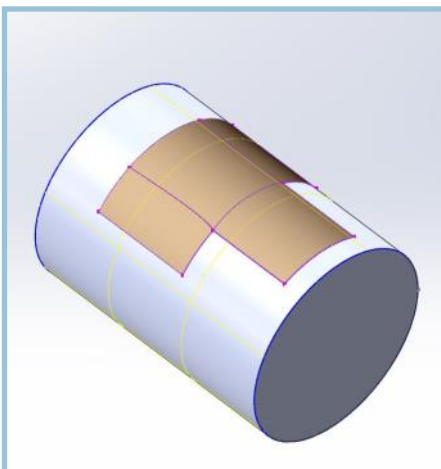
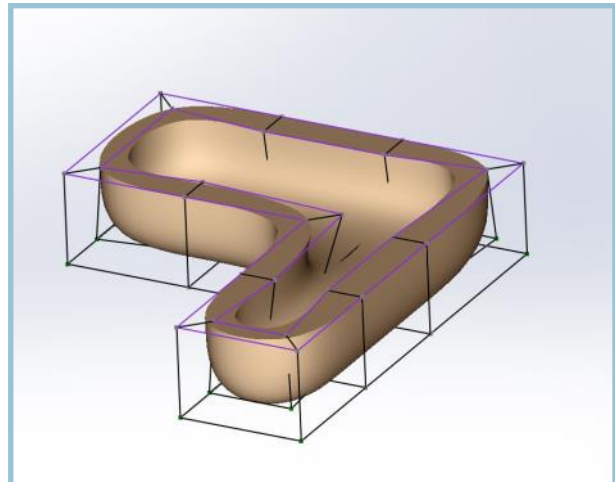
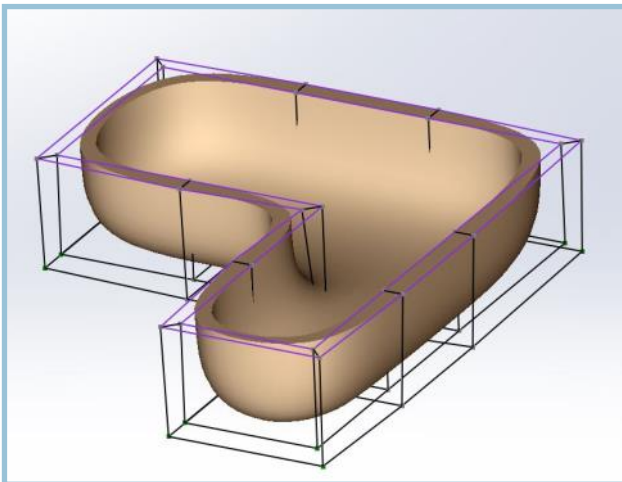
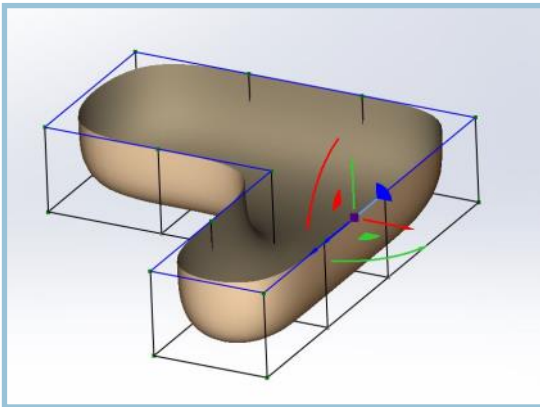
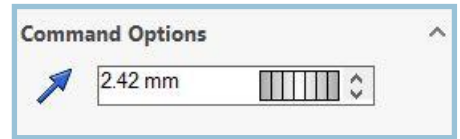
Taper

The taper command grows or shrinks the selected vertices of a mesh along a given axis. The first number box controls the strength of the overall taper. Below we used 1.5 in each of the cases. The upper left image shows a taper along the X axis using Y. The upper right image shows a taper along the X axis using Z and the lower left image shows a taper along both the Y and Z directions. The second dialog box shows a the bowing effect. The lower right image has a bow of 1.0, which effectively doubles the offset at the middle of the taper.



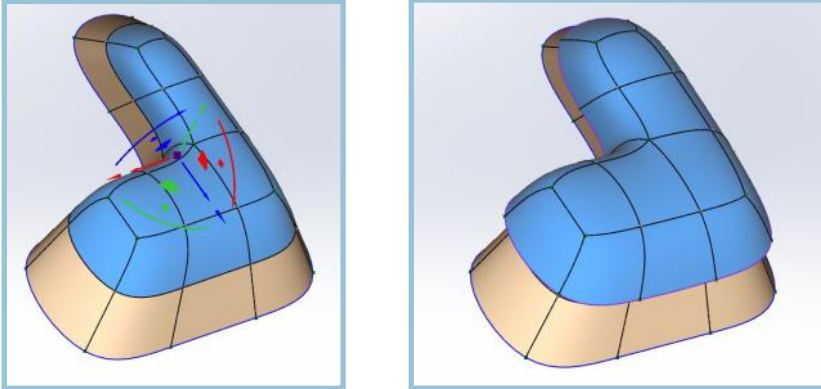
Thicken

Thicken works on all faces of the model and does an offset at each vertex and connects the resulting solid. It does the equivalent of an offset or inset of the body. The first image below shows the original body, the second image shows an offset and the third image shows an inset. If you thicken an object with a constrained face, the constraint will be maintained with an offset on the resulting thickened faces. See the bottom two images for an example of thicken with constraints.



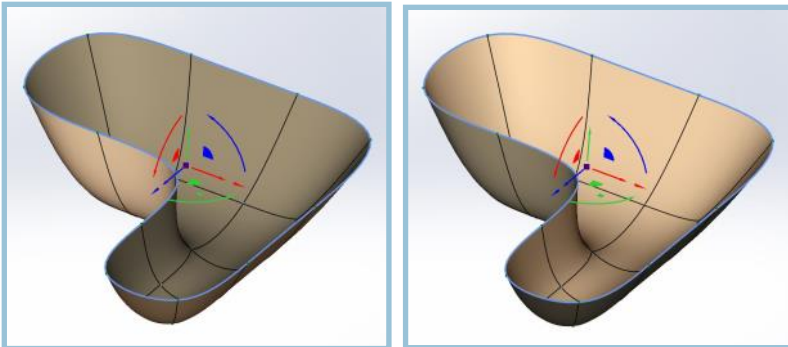
Normal Push

Normal Push copies and then moves the selected faces out along the normal defined at each vertex by the average face normal at that vertex. The first image below shows the selected faces and the second images shows a normal push.



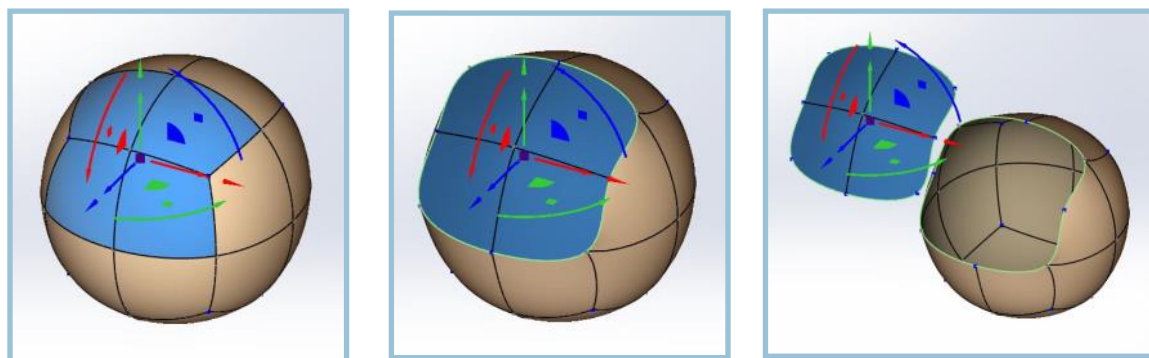
Flip All Faces

Flip Faces flips the normal of all of the faces of the mesh or just the selected faces. Below shows an example of Flip Faces.



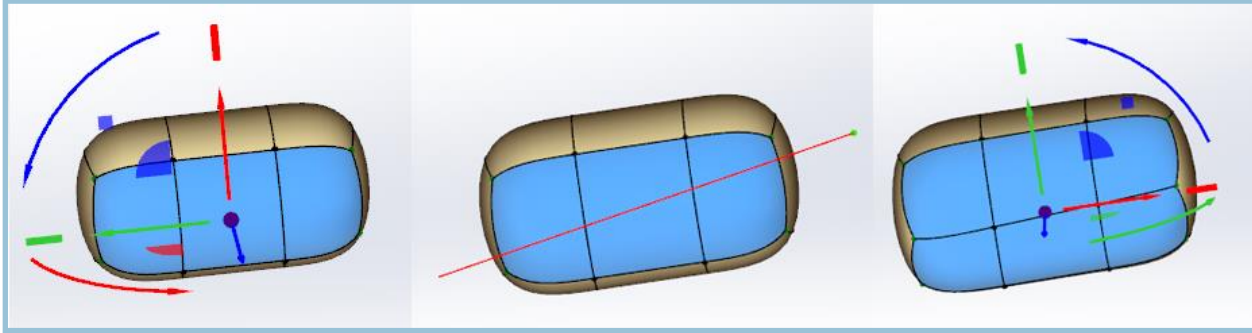
Flip Selected Faces

Flip Faces flips the normal and orientation of just the selected faces. Below shows an example of Flip Selected Faces. Note that this will split the mesh between selected and non-selected faces.



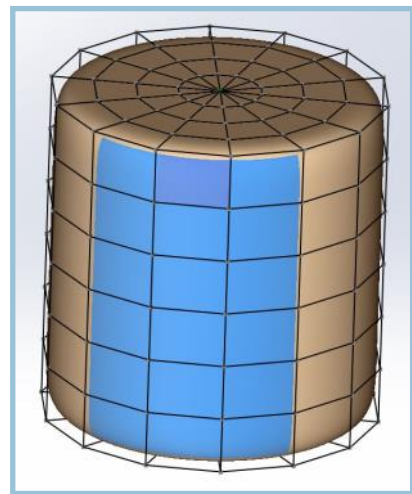
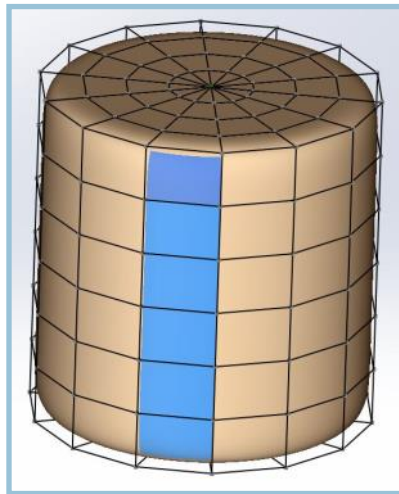
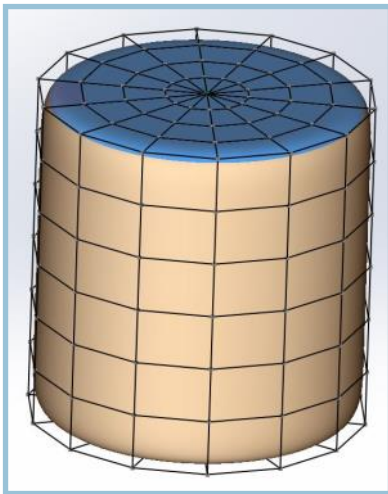
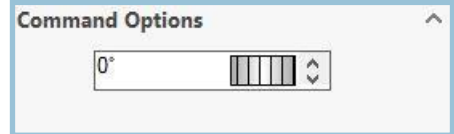
Slice

Slices through the mesh or selected faces creating new edges using a plane defined by two points in screen space, perpendicular to the current view. Slice typically picks points on either side of the mesh to define the slice.



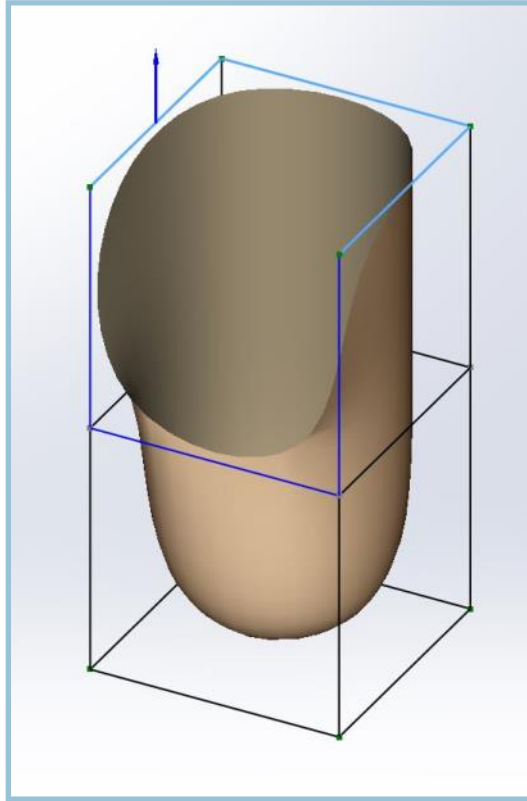
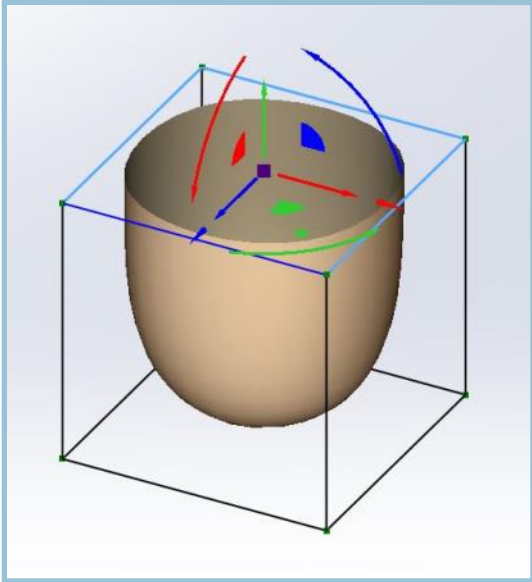
Select by Angle

Select by angle selects all faces adjacent to a selected face within a given angle of the original face. The first two images below show selection with an angle of zero. The third image shows an angle of 40 degrees.



Extend

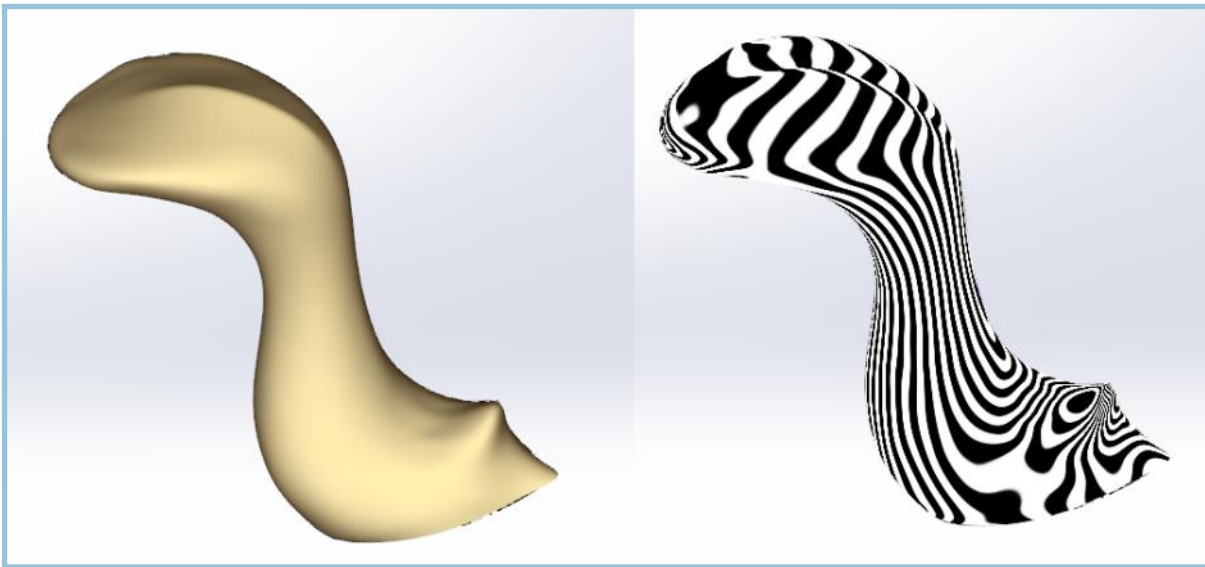
Extend takes selected open edges and extends them along the edges of the adjacent faces.



Analysis Display Tools

Zebra

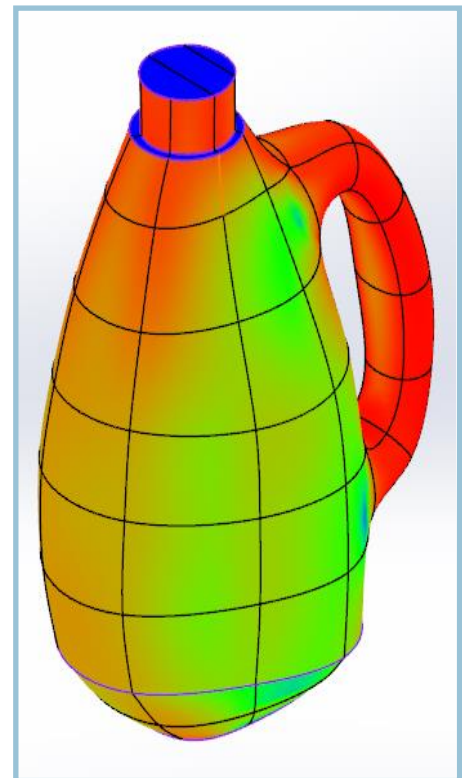
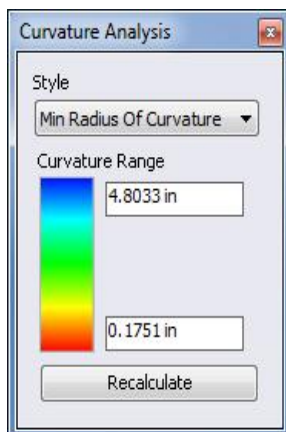
The zebra tool displays an analysis view that allows you to see changes in continuity and curvature. Additional options to control the zebra display settings are available in the Options Panel [shown here](#).



Zebra Analysis of a Joy Stick

Curvature

The curvature analysis tool displays curvature in a color coded manner so you can visualize areas of high and low curvature. This tool is highly interactive so you can perform modeling updates such as moving vertices and see the curvature update dynamically. You may change the curvature range values in the curvature dialog to detect different radius of curvature values, or update them automatically using the Recalculate button..



Curvature Analysis Example

Export Options

Save Control Mesh as *.obj or *.fbx

Exports the control mesh only from the Power Surface model.

Formats: Autodesk .fbx, or Wavefront .obj

Available during editing only.

Save Selected Subset of Control Mesh as *.obj or *.fbx

Exports the selected faces control mesh only from the Power Surface model.

Formats: Autodesk .fbx, or Wavefront .obj

Available during editing only.

Save SubD Mesh as *.obj or *.fbx

Exports the Power Surface model with the current SubD level.

Formats: Autodesk .fbx, or Wavefront .obj

Available during editing only.

Save Current as *.Cyborg3d

Exports the Power Surface model and all of its attributes.

Format: nPower's Internal Power Model Format .cyborg3d

Available during editing only.

Save Reference Mesh as *.obj or *.stl

Exports the the reference mesh as an *.obj or *.stl file.

Note that this is only available in Power Surfacing RE.

Format: Wavefront *.obj or stereolithography file *.stl.

Available during editing only.

Options

From the Power Surfacing menu, Options, you can change several of the default settings.

General

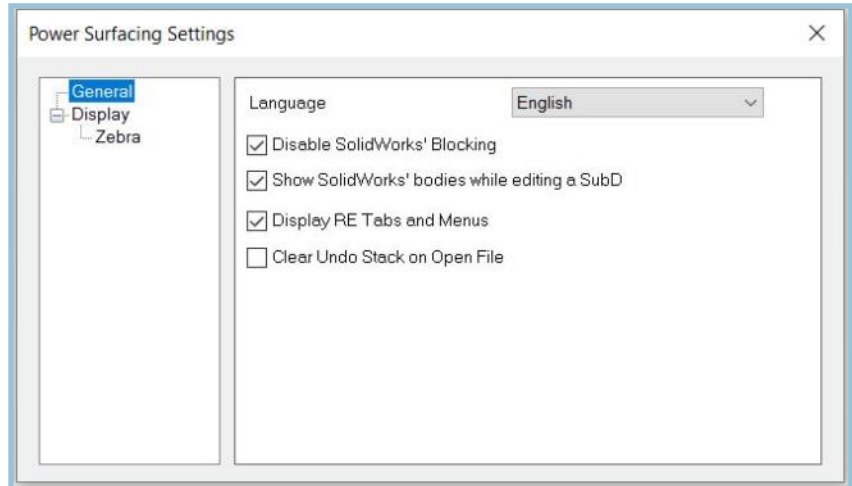
Power Surfacing lets you select from three language options. English, Spanish and XXX.

Disable SolidWorks' Blocking will allow you access to many options generally blocked when you are actively setting up or editing a feature. This is often used to change visibility of SolidWorks objects dynamically within SubD editing mode. **Never save your part file while SolidWorks Blocking is disabled, as you risk corrupting your file and/or losing changes you have made.**

Show SolidWorks' bodies while editing a SubD allows you to keep other SolidWorks bodies visible while creating and editing your SubD model.

Display RE Tabs and Menus this will turn off the RE Tabs and menus.

Clear Undo Stack on Open File this option will clear the undo stack when the file opens and keep the file size small.



Display

From Display, you have an option to show or hide back-facing edges when using transparency.

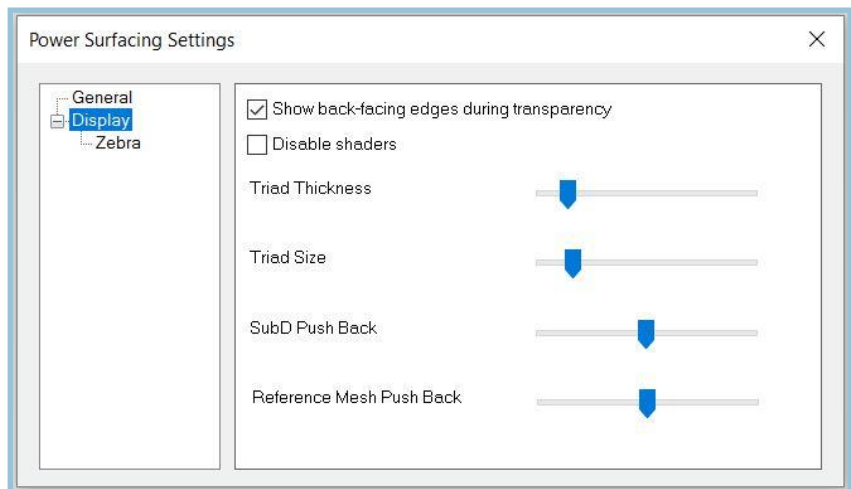
Disable Shaders will disable shaders in SubD edit mode only. This is useful for investigating graphics card compatibility issues. With shaders disabled, you will no longer have access to transparency options.

Triad Thickness allows you to adjust the thickness of the triad gizmo. Note that changing the thickness will affect the pick zone.

Triad Size allows you to change the length of the arrows and size of the gizmo.

SubD Push Back controls how much the SubD mesh faces move back from the edges. If your edges are kind of jaggy, you might want to increase this value.

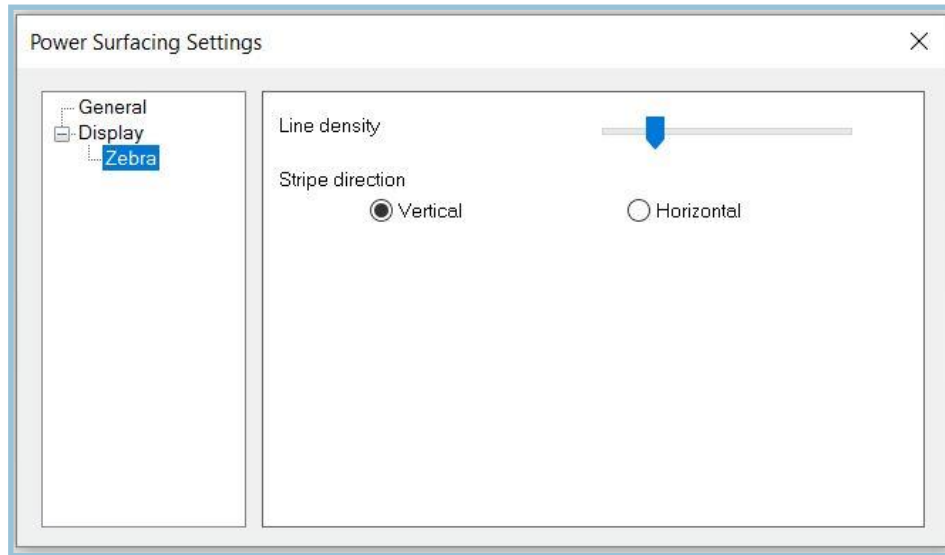
Reference Mesh Push Back controls how much the reference mesh is pushed back from the SubD during retopologizing.



Options, (continued)

Zebra

The Zebra shader option in the Viewing tools section allows you to edit your SubD while observing the reflection pattern as you work.

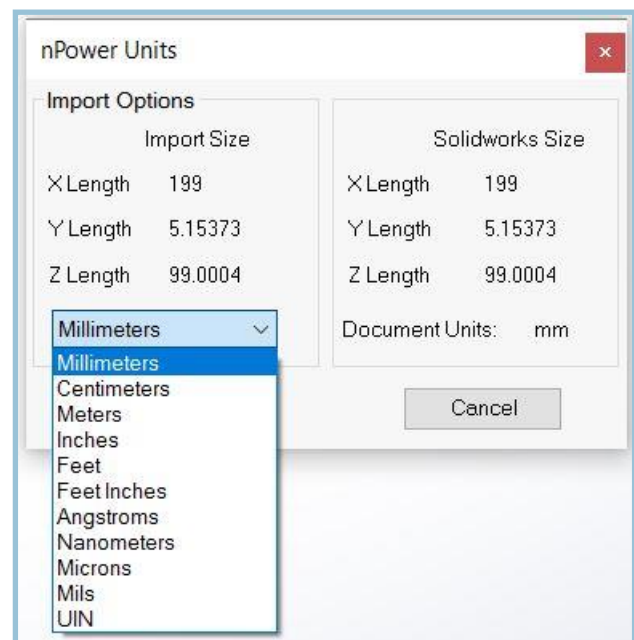
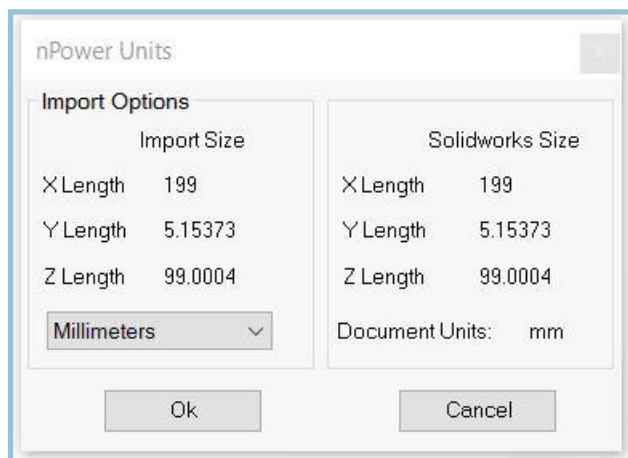


Line Density allows you to control the overall scale of the stripes.

Stripe direction you can select either Vertical or Horizontal for the stripe orientation.

Import

Import/Export options allow you to choose the units associated with .Obj and ,Fbx file types. If an object comes in too small or too large it is likely an issue with the units. Delete the object and import the object again with different units.



Power Surfacing RE

Power Surfacing RE contains all Power Surfacing tools plus additional tools for Reverse Engineering. It is available as a separate product.

The Power Surfacing RE Tools



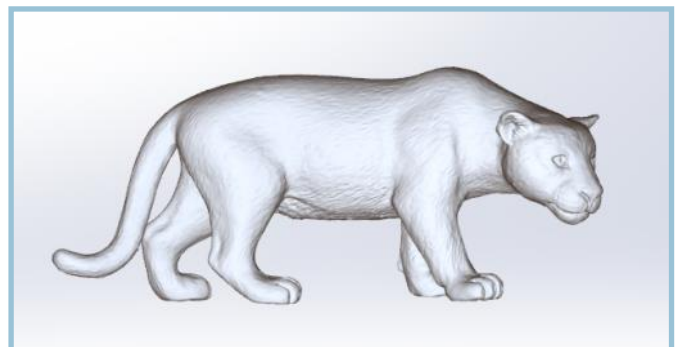
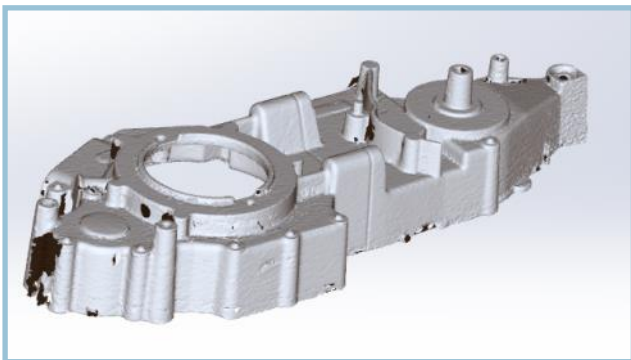
What is Retopologizing?

In today's world, few objects are simple enough to be reverse engineered from only a handful of accurate measurements. The complexity and range of manufacturing methods produces objects that rely heavily on organic shapes for both form and function. To reverse engineer these non-mechanical shapes, retopologizing is the method of choice. Retopologizing is the process of resurfacing an existing model using quad-based SubDs. You may want to modify an existing CAD model that no longer has a creation history, clean up a scanned mesh so it can be converted to a CAD model for manufacturing, or simplify an existing SubD mesh for real-time simulations. Modifications on the original can also be easier when working with a SubD mesh rather than its CAD counterpart.

What can be Retopologized?

Power Surfacing RE is full of tools and features that will allow you to retopologize scanned meshes. These same tools can help you to retopologize existing SOLIDWORKS features and other CAD objects. Additionally, you can 'retopo' imported .obj and .stl files for both backwards engineering and for making modifications.

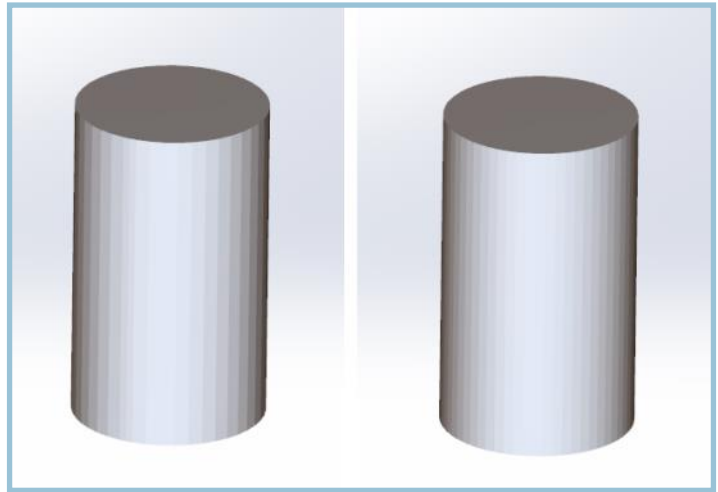
The left image below shows an unprocessed scanned mesh. The image on the right shows a cleaned up scanned mesh.



Workflow Scenarios

With Power Surfacing RE, you have several workflow options from which to choose. Your decision will be made depending on the quality of the reference mesh and the accuracy required from the retopo mesh.

The reference mesh can be a scanned mesh, where it may be straight from the scanner, (Image A), or it may have already been through a clean-up process, (Image B). When the reference mesh has been created from an existing CAD model or part, the resulting reference mesh may be clean, but not provide enough detail for accuracy, (reference Image C). A SubD or polygonal model, (Image D), may be used as a reference mesh to create a simpler mesh for more efficient modification or use in real-time simulation where dimensions rather than detail must be retained.



C) CAD object used as a reference mesh, left cylinder uses medium Image Quality resolution used, right image uses high Image Quality resolution.

The accuracy you require from the retopo process will vary. If the reference mesh is from a very messy scanned mesh or a good scan of an older and distressed object, less accuracy will give a smoother result. If the scan is very clean, or has been processed into a cleaner state, you may opt for a higher degree of accuracy. More detail or accuracy can be achieved by more and smaller faces, or, through the use of the *Add Interpolation* option. When used, the Interpolation step should be the final step before conversion unless modifications are planned.

Clean-up can be performed directly onto the reference mesh to remove unwanted features or improve surfaces prior to the retopologizing process.



D) High poly SubD mesh as reference mesh

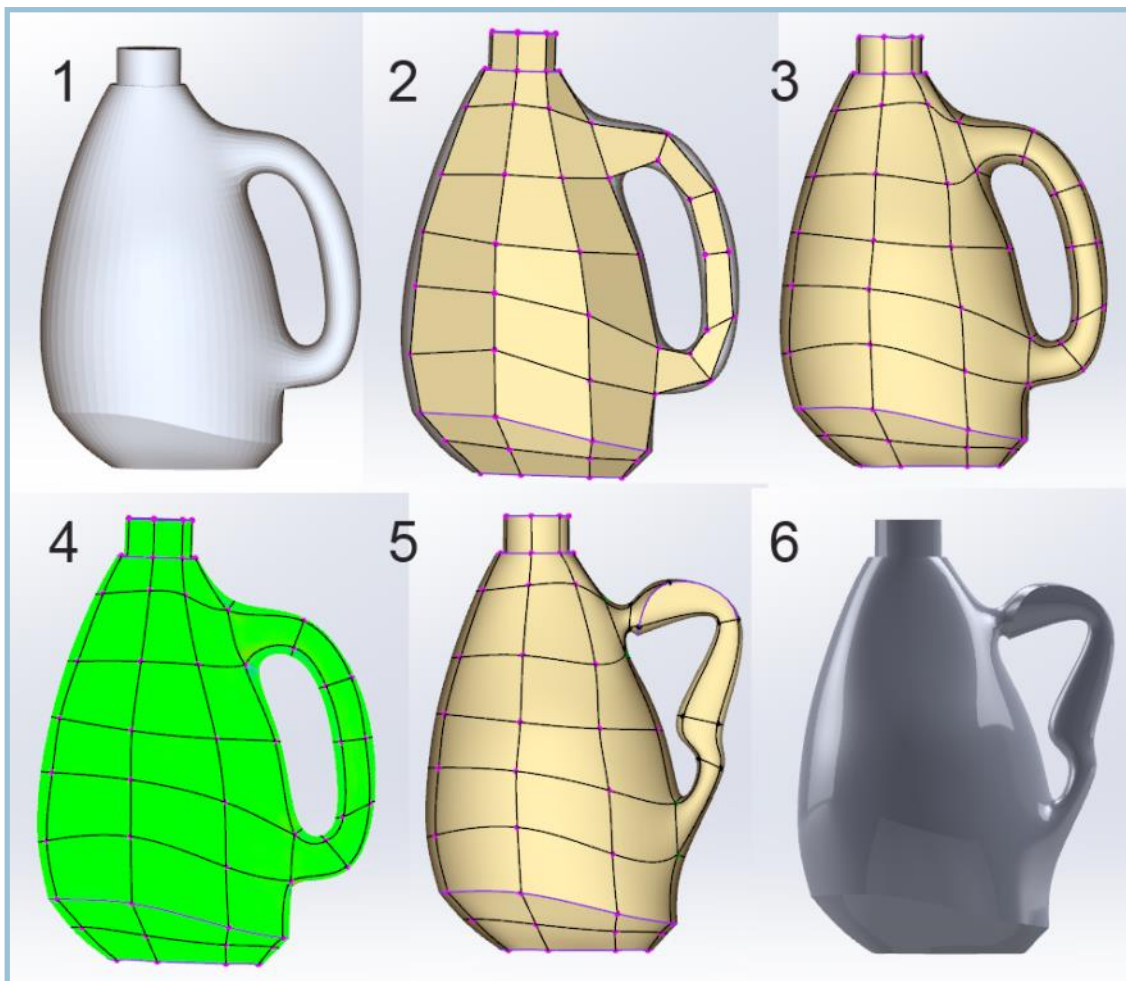
Retopologizing vs. Constraints

Although you *can* use an existing CAD model as your reference mesh, you may be better off using Power Surfacing's constraint system. If the CAD model is an analytic, such as a cylinder, the constraint system will provide more accuracy because the SubD faces and edges can be constrained to an extracted CAD surface or edge. When using a reference mesh for retopo, there is no concept of defining an edge and staying constrained to it. The exception is reference meshes that have open edges.

With the retopo workflow on CAD objects, the SubD can be used to create a solid, but at the expense of accuracy. If accuracy is not crucial and the Interpolation option is not used, the SubD will be much easier to modify or alter. The constraint system is a good way to ensure an accurate connection to an analytic shape, but may require replacing of surfaces, defining the connection shape, trimming and knitting to bring the altered object back into solid form. See Constraints in the Power Surfacing documentation for more information on their use in the modeling workflow.

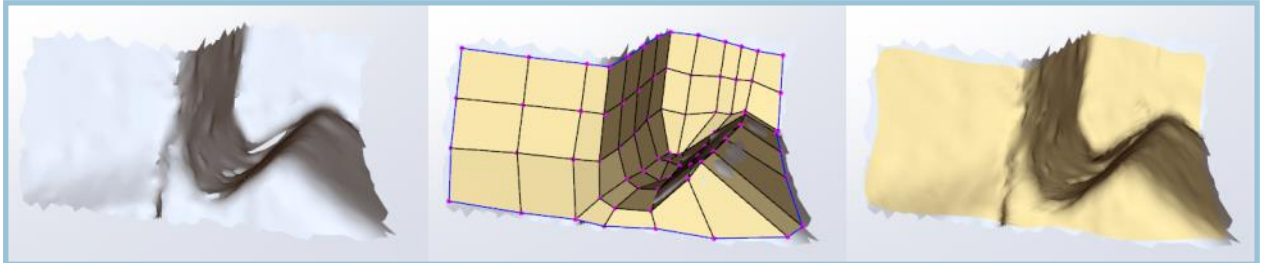
General Workflow

The general retopologizing procedure is fairly straight forward. First obtain or generate a reference mesh, clean it up or prepare it if necessary (Image 1), then proceed with the following: Create the new SubD on top of the reference mesh (Image 2). Interpolate to maintain the reference mesh details (Image 3). Check the result for accuracy (Image 4). Unconstrain and modify target areas (optional, Image 5). The final step is to convert the result to CAD (Image 6).



Obtaining a Reference Mesh

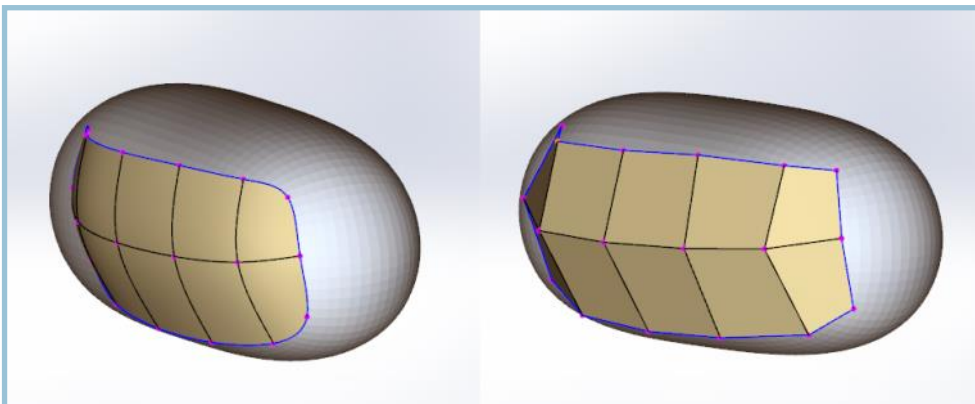
The reference mesh is a polygonal representation of the object's surface. The smaller the faces, the more accurate the reference mesh tends to be. You can start with a scanned mesh in either obj or stl format. Point cloud data is not currently supported, but point clouds can be converted to mesh format by most scanning software (and other tools). You can also generate a reference mesh from an existing SOLIDWORKS feature or even a suitably subdivided SubD mesh. Be aware that the retopologizing workflow snaps the new geometry to the reference mesh; if the retopo mesh (the new SubD mesh) is to be highly subdivided, it can reflect inaccuracies in the reference mesh.



Too many *Shrink Wrap* subdivisions- the poor reference mesh surface is reproduced

Retopo Basics

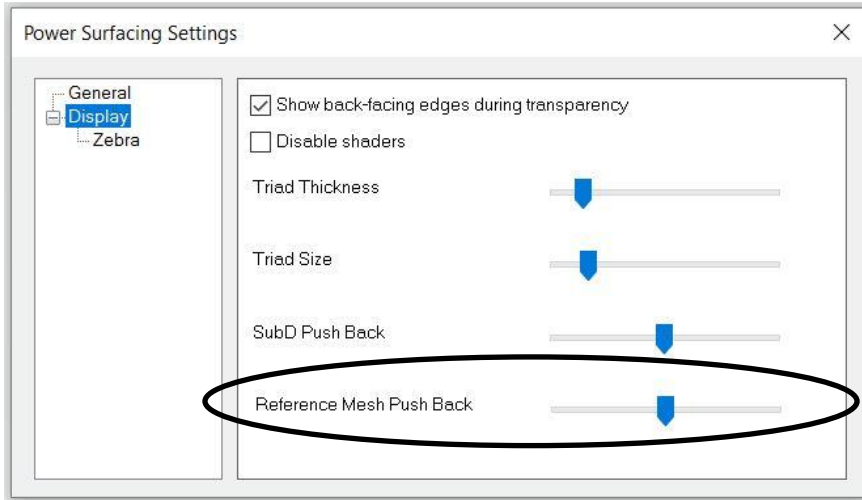
With a suitable reference mesh acquired and prepared, you will create a new SubD surface on top of the reference mesh using the various Power Surfacing RE tools. 'Retopology' is normally done in Control Mesh display mode so that you can see the true location of the constrained vertices on the reference mesh's surface.



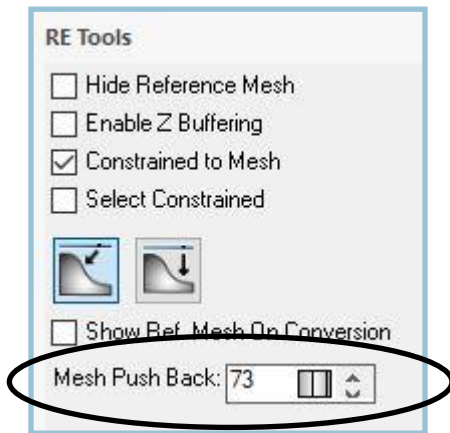
Constrained vertices (magenta colored) pulled back from the reference mesh's surface in SubD display mode, left, and showing their surface constraint in Control Mesh display, right

Adjusting the Reference Mesh Display

You may have noticed that the SubD's [flat] faces are fully visible 'behind' the reference mesh's surface. This draw order effect was implemented to make modeling easier. If you chose to use the reference mesh as a 3D template and do not want to constrain the SubD vertices to it at some point, you may want to adjust the draw order *Reference Mesh Push Back* amount in the Options dialog . It can be found under General in the Power Surfacing Options.

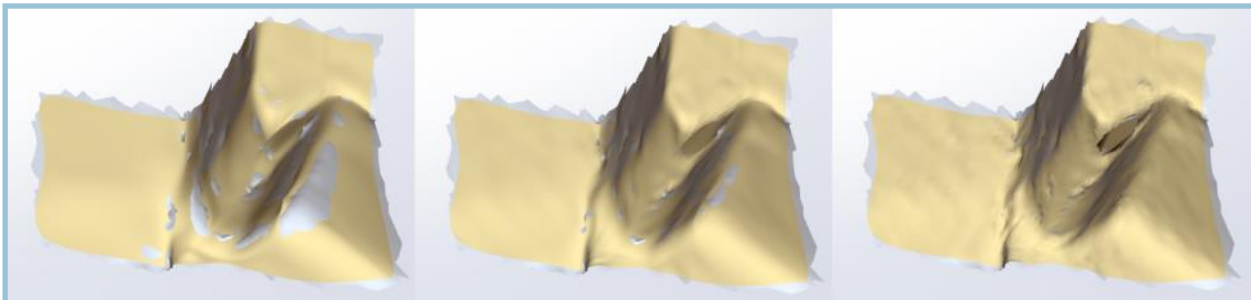


The Power Surfacing Options dialog where the Reference Mesh Push Back is adjusted



Quick Access to Reference Mesh Push Back is provided in RE Tools panel.

Setting the slider to the far left will show you where the actual SubD mesh faces are in relation to the reference mesh. At the far right, the SubD mesh will be drawn last or on top of the reference mesh. The effect will vary according to the viewing angle as well. Feel free to adjust the amount slider as needed.

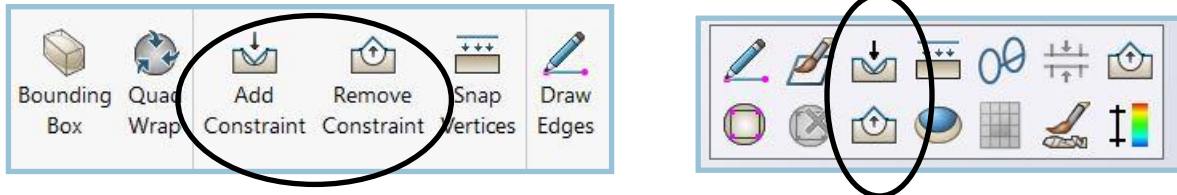


Push Back at minimum with 1, 2 and 3 subdivision levels of interpolation

Vertex Attributes

As the new faces are drawn on the existing surface, the vertices are automatically marked as 'constrained' and are constrained to the reference mesh's surface. Constrained vertices are magenta colored rather than black. You may un-constrain and re-constrain vertices as needed through the tool bar or right-click menu. A few of the Power Surfacing tools, such as *Extrude* and *Extend*, do *not* automatically constrain new vertices.

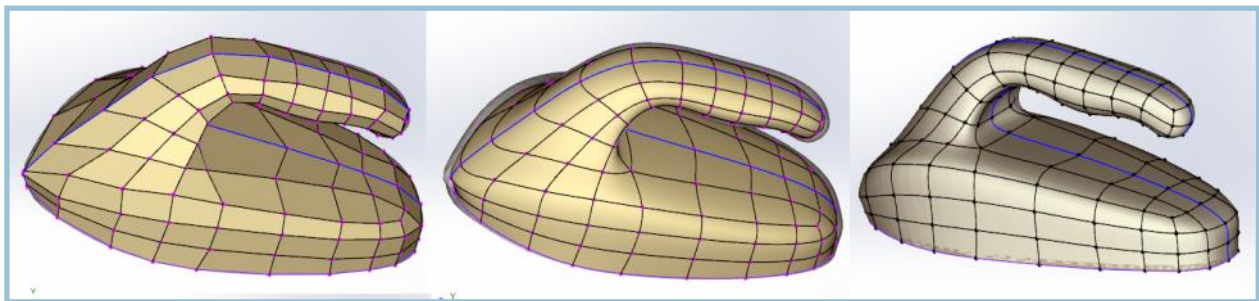
Add Constraint and *Remove Constraint* are available in the Power Surfacing RE toolbar and RE right click menu as shown below



Using Shrink Wrap

When you've finished the retopologizing process, you can use *Shrink Wrap* to increase the accuracy of the mesh with respect to the reference mesh. You've been working in Control Mesh display as you created the retopo mesh, but switching to SubD display, you will see that the SubD version, by its very nature, shrinks inward to produce its curvature continuous surface. Because the subdivided version is what is used to generate the CAD model, the *Shrink Wrap* step is an important one.

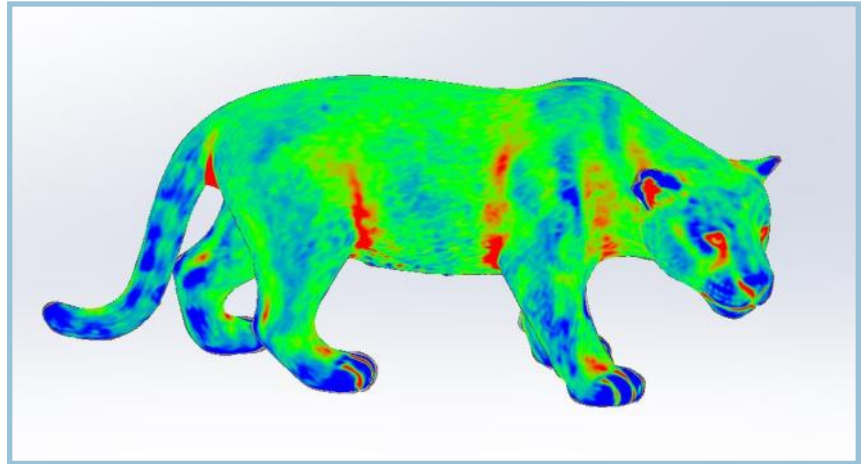
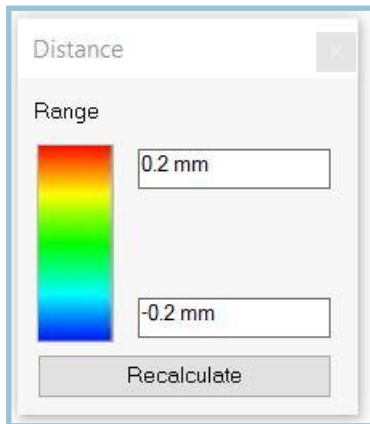
The *Shrink Wrap* command brings the subdivided mesh back out to the reference mesh's surface. A single level of internal subdivision may be sufficient depending on the density of the retopo mesh and the amount of detail you want to retain from the reference mesh. A second level of subdivision will be more accurate but may reveal unwanted detail. A third level of subdivision is rarely required and can take a long amount of time to calculate. *Shrink Wrap* works only on constrained vertices.



The retopo'd mesh in Control Mesh display, upper left, sucked inward in the SubD display mode, upper middle, pushed out to the reference mesh surface after shrink wrap, upper right

Checking Accuracy

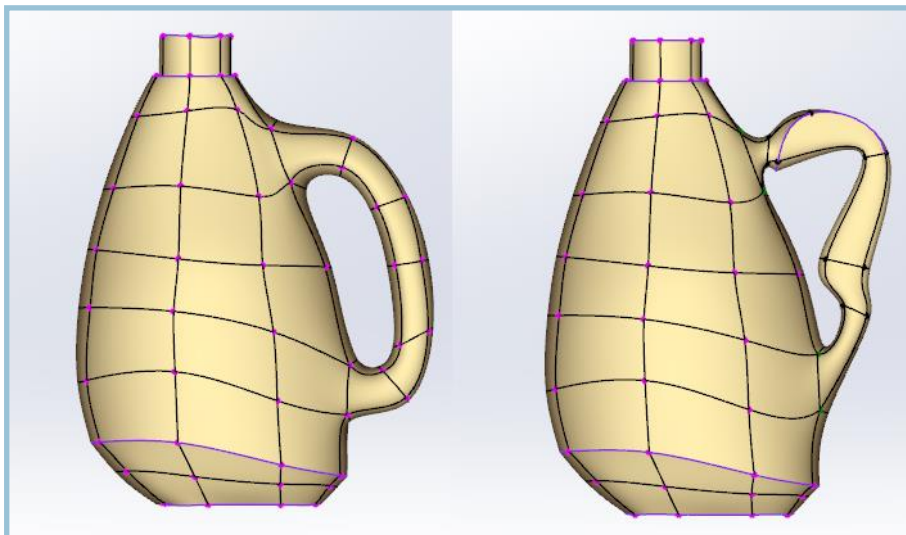
At any time, but especially during the *Add Interpolation* step, you may check the accuracy of the interpolation using the Distance display tool. This tool gives you visual feedback showing where the retopo mesh deviates from the reference mesh and by how much. The range is automatically calculated using the maximum deviations and is displayed on the mesh where green is on, blue is under and red is over the reference mesh. When using *Add Interpolation*, you should use the Recalculate option to update the range after adding additional divisions.



Checking the distance deviation in the dialog, left, and visually, on the SubD, right

Modifying the Interpolated Mesh

If you plan on making alterations to your retopo mesh you should make them *after* using *Add Interpolation* and only after un-constraining the vertices that will be involved with the manipulation. Several of the retopo tools, such as *Offset* and *Extrude* will automatically un-constrain vertices as part of their functionality. Tweaking unconstrained vertices allows you to alter the mesh quickly to arrive at your desired result. Be aware that moving constrained vertices may produce undesired results as their offset and influence is recalculated.



A retopologized mesh, left, modified, right

Converting the Retopo Mesh to CAD

As with regular SubDs, when you are happy with your mesh, you will want to convert it to a SOLIDWORKS solid or surface. The default setting for the conversion quality is Medium. Occasionally, you may need to use the Fine setting to ensure that all edges are properly knit.



The eagle reference mesh retopologized and then converted to CAD

Preparing the Reference Mesh

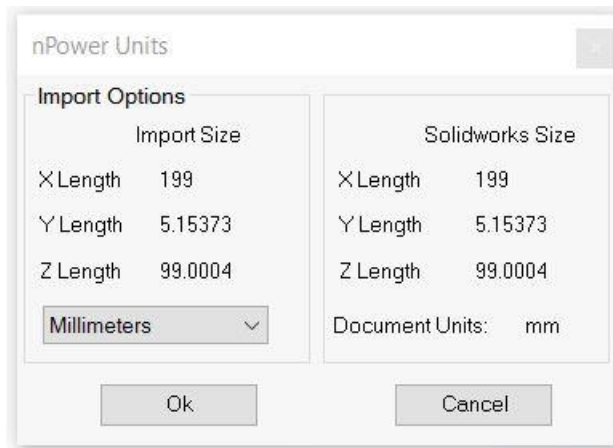
Depending on the reference mesh source, its condition and your goal for the retopologized version of it, you can streamline the process by preparing the reference mesh beforehand. The goal is to remove unwanted faces, orient the mesh for easier handling, repair faulty or changed geometry and finally, set up selection sets to make the retopo process go much smoother.

Importing the Reference Mesh

Power Surfacing RE can use either .OBJ or .STL files as reference meshes.

To import an STL or OBJ file:

1. Create a new Part file.
2. From the Power Surfacing menu (under Tools in SOLIDWORKS 2015), select Import Scanned/Reference Mesh.
3. Select either an OBJ or an STL file.
4. Select the units for the imported file.



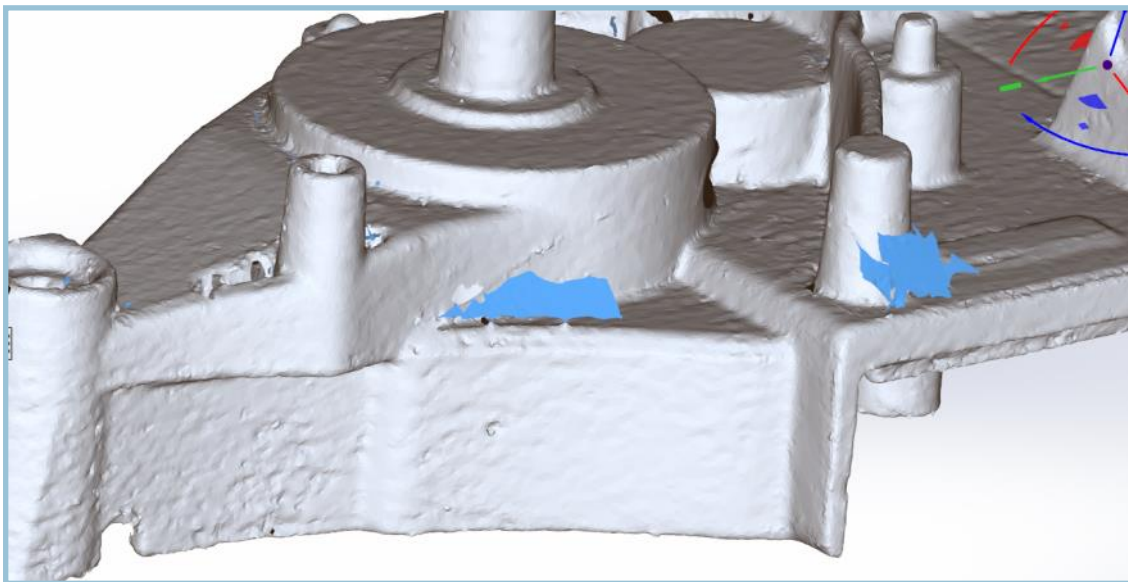
Note: Neither file type has a concept of units, so you must specify them on import. If you are unsure, you can check the value as it would be shown in the part file's default units.

Obj files can also be imported through SOLIDWORKS File/Open but you must select PS OBJ Reference Mesh (*.OBJ) as the file type *before* selecting the OBJ file.

Removing Isolated Geometry

The first step in preparing a scanned mesh for retopologizing is to remove any stray faces left behind from the scanning process. The Select Connected option in the selection section of the Command panel makes this task fairly simple. You must, however, be in *Edit Reference Mesh* mode.

1. From the Power Surfacing RE tab's Tool bar, select Edit Reference Mesh.
2. In the Command Panel's Selection Utility, in the Selection Filters, choose *Select by Element*.
3. Click on the main part of the reference mesh.



Using Select Connected and Invert Selection to find and delete stray polygons

4. Hold the Shift key down and select anything else that should be included in the reference mesh.
5. From the Selection Tools section, select *Invert Selection*.
6. When you are sure the remaining faces are all strays, use the Delete key on the keyboard or use the *Delete* tool from the Power Surfacing tool bar or the right-click menu.
7. You can exit Edit Reference Mesh mode by selecting the green check mark in the Control panel.

Tip: If selecting the main part of the mesh only selects a small part of the object, turn on *Display Open Edges* to see if the model is too fragmented for this technique.

Removing unwanted Geometry

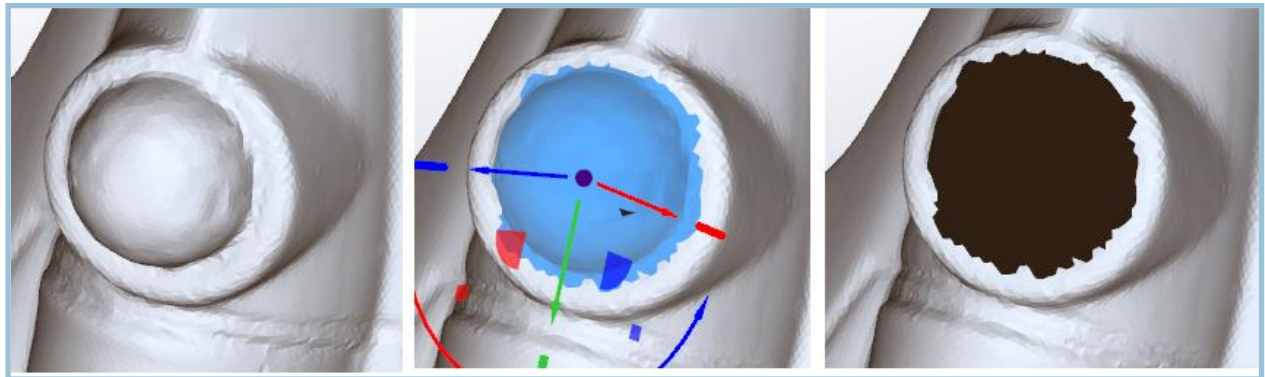
Because a scanned mesh can be overly large and unwieldy, the next step should be to delete any *features* that could interfere with the retopo process. This can be anything from bumps and warts left by the scanning process, to 'features' that can be added easier and more accurately as SOLIDWORKS features. As part of the 'prepping' process, you will be adding patches to cover holes left by both the scanner and removal of unwanted features. Surfaces that will be hidden by this type of proxy geometry should also be removed.

1. Make sure you are in *Edit Reference Mesh* mode.

2. Select the geometry you wish to remove.

Tip: Use *Paint Selection* for awkwardly placed areas.

3. Select *Delete* from the Power Surfacing tab or the right-click menu.



Removing unwanted features or features better suited to CAD modeling

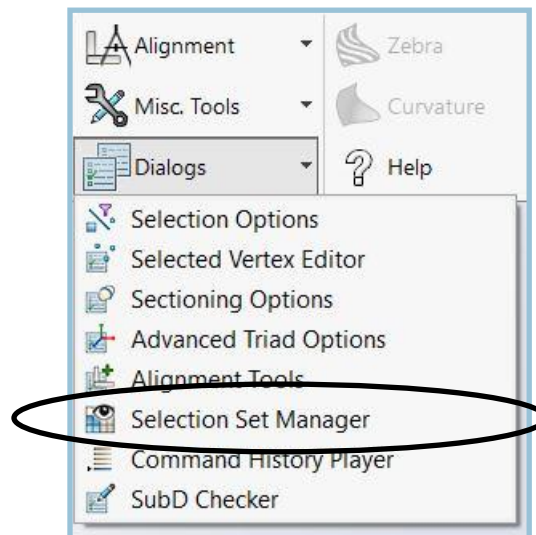
Selection Sets

Selection sets provide multiple functions. They can be used to hide or show parts of the reference mesh to provide easier access to various parts of the reference mesh. They can also be used to position retopo mesh edges with reference mesh features.

With constrained vertices, if the reference mesh has open edges, the vertices will automatically stop at the open edge when pulled toward it. When a portion of the reference mesh is hidden, the visible edge is treated as if it was an open edge. If the reference mesh can easily be divided into sections along its 'hard' edges, the retopo process can be streamlined to a great extent.

The procedure is as follows:

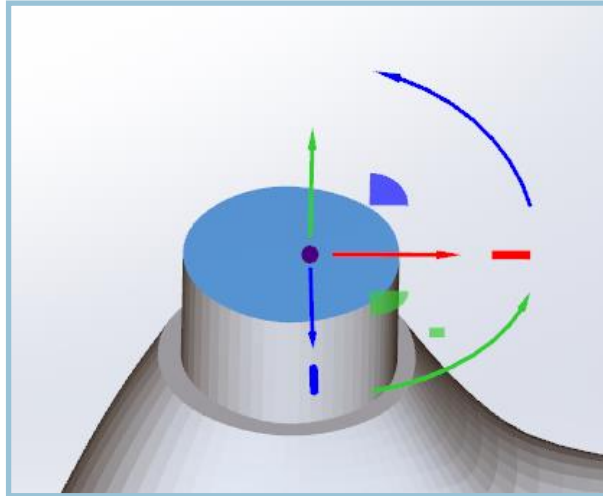
1. From the Power Surfacing RE tool bar, select *Edit Reference Mesh* if you are not already in that mode.
2. From the Power Surfacing tab, click the Advanced tools drop-down and choose *Selection Set Manager*.



Opening the Selection Set Manager

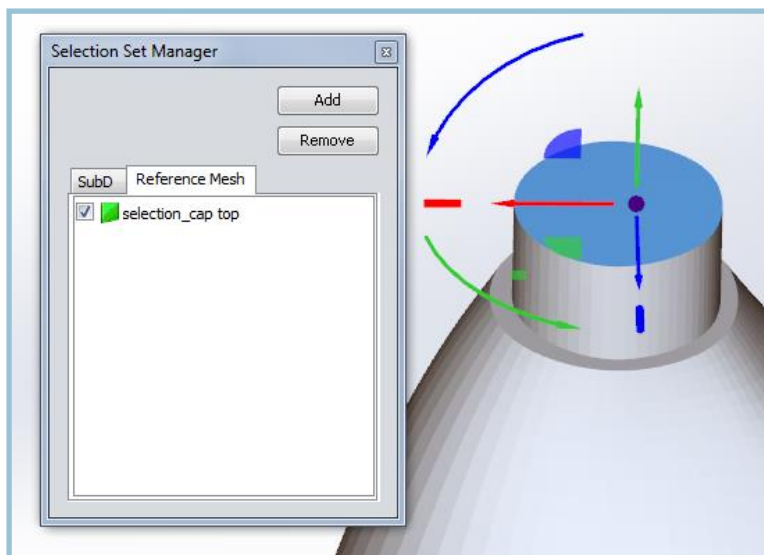
Selection Sets (continued)

3. Select the faces on the mesh that define the hard edge (fully weighted) section.



Selecting an area bounded by hard edges

4. In the *Selection Set Manager*, click the Add button and then click on the new selection set, selection_0 and rename it **selection_cap top**.

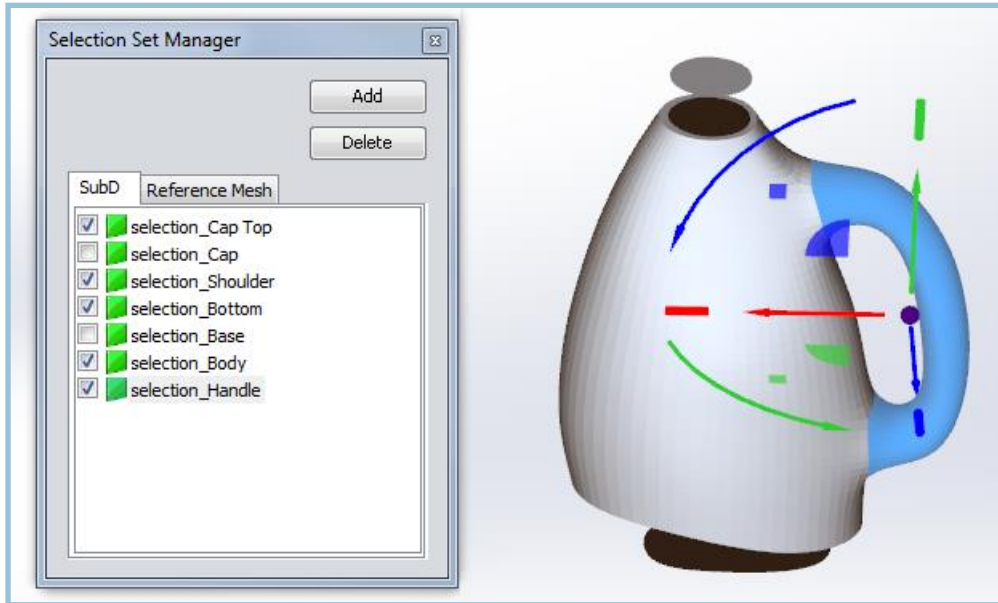


Adding and renaming the new selection set

The new selection set shows the icon for a *face* selection. The check box allows you to turn the selection off and on (hide/show). Clicking on the selection set name will select it in the viewport.

Selection Sets (continued)

5. Create selection sets for the rest of the mesh, naming as you go.

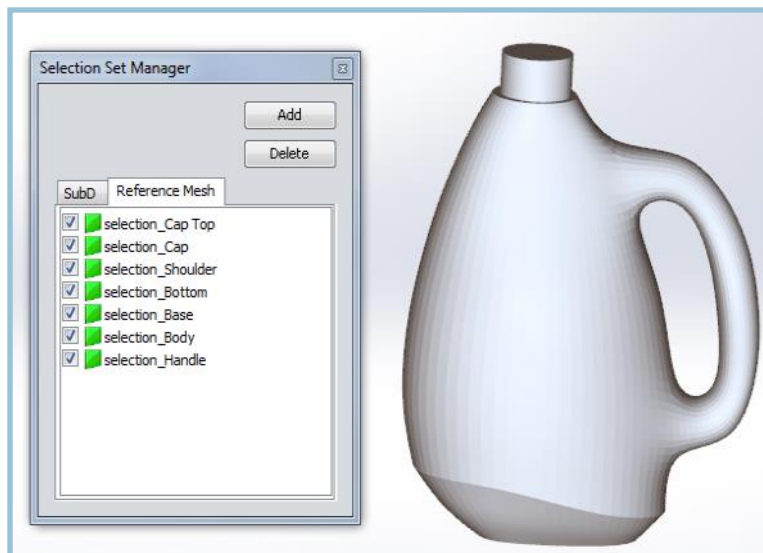


Creating the rest of the selection sets and hiding a few of them

Once you have every other set created, you can turn them off and select the remaining sections using *Select by Element*.

With the selection sets defined, now is a good time to save the scene. Previously, you saved the oriented reference mesh as an obj. The obj file type does not support selection sets, so you will be saving this one as a .pmodel, Power Surfacing's native format.

6. From the Tools menu, Power Surfacing, Save, *Save Scene*, appending the name with SS for selection sets, if you wish.



The polygonal mesh converted to a reference mesh and its selection

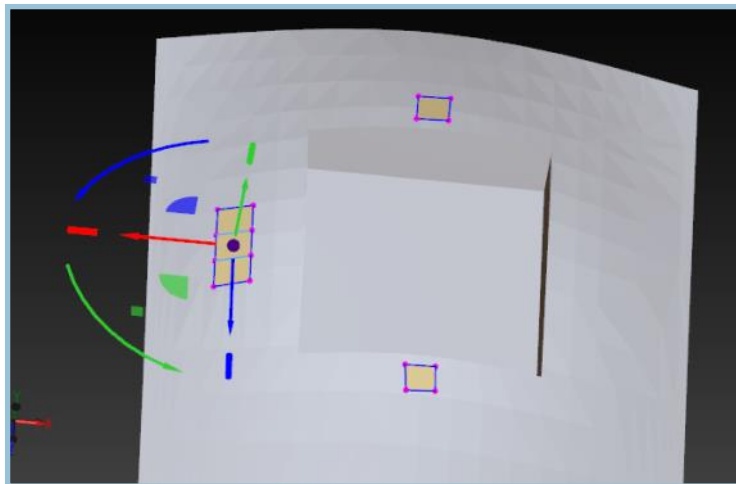
Creating Proxy surfaces

There are several reasons to improve the surface of your reference mesh before going through the retopo process. Some features are better added with the conventional SOLIDWORKS workflow. Some existing features may be unwanted in the new version of the object. In the case of scanned meshes, there may be areas that didn't scan well, or areas where the original object was worn, injured or otherwise undesirable. Whatever the reason, providing a new proxy surface for a constrained vertex to snap to will save you time in the long run.

In this example, the raised and recessed areas are better done as SOLIDWORKS features. With the faces turned off, the retopo geometry won't snap to the hidden faces, so the next step is to create the proxy surfaces. This will give your retopo geometry something to snap to that lets you retain a continuous surface.

Small areas and flat areas are fairly simple to fill. The small bolt holes in this example are a good place to begin.

1. Using the *Draw Edges* tool, , create a small quad to go over each of the two small holes and the slot.
2. You may want to use *Insert Edges*, , to make sure the slot's quad conforms to the surface.



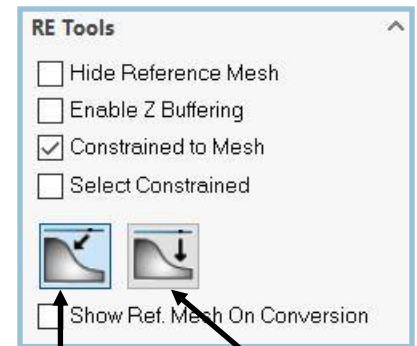
Using small quads to cover holes in the reference mesh

Next you will add the 'bandages' to the reference mesh.

3. In Face sub-object mode, select all of the new faces.
4. In the Power Surfacing Retopo tab, pick the *Add to Mesh* button, .

The Retopologize Tools Panel

When you are in the Power Surfacing RE tab, you will see the Retopologize Tools rollout in the Command panel. These options can be very useful during the retopo process.



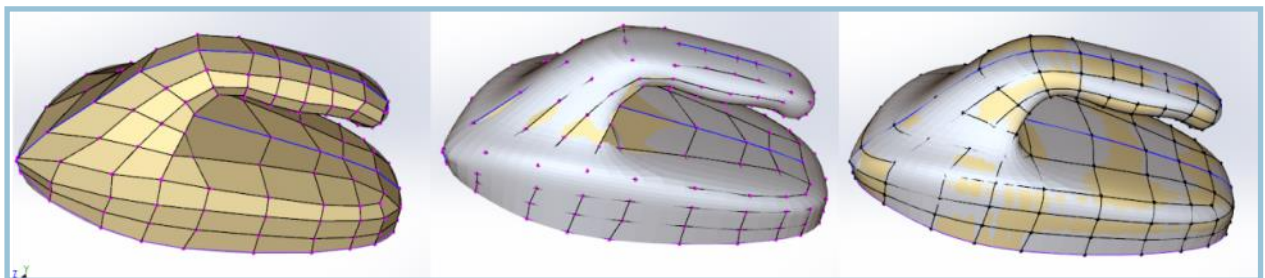
Snap Closest Point Snap Normal

Hide Reference Mesh

When checked, this hides the reference mesh, allowing you a full view of the SubD retopo mesh.

Enable Z buffering

In order to see the retopo mesh's faces, which will normally fall behind the reference mesh's faces, Z buffering is generally turned off. It may be useful to turn it on when checking the Interpolative result. Depending on the curvature of the model, you may also want to adjust the amount of offset used *when Enable Z buffering* is turned off. That is managed through the Power Surfacing Options dialog under "Display." It is called *Reference Mesh Push Back*.



Enable Z Buffer turned off (the default during the retopologizing process) and showing the retopo surface in front of the reference mesh, left image. The center image, shows Z buffering turned on, where only vertices and edges on top of the reference mesh's surface show. Right image shows mesh after interpolation where the two surfaces should occupy the same space.

Constrained to Mesh

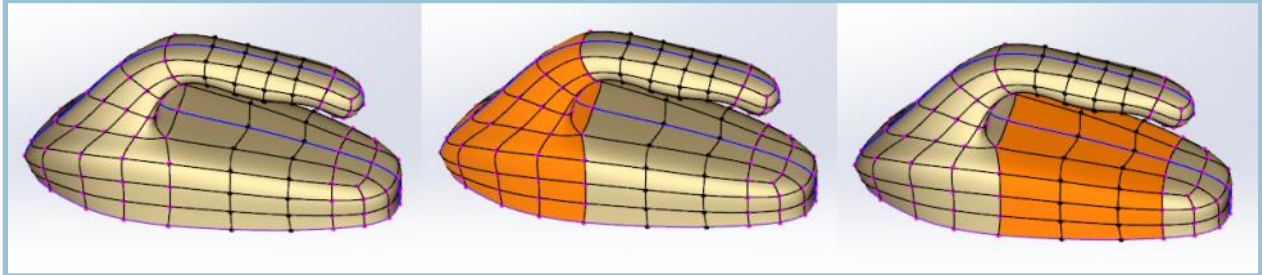
In this mode, which is the default during the retopo process, any vertex with its constraint attribute set to *true* is constrained to the reference mesh. If *Constrained to Mesh* is turned off, vertices will not be constrained to the mesh, regardless of the state of their constraint attribute. This provides you with a way to affect the vertices without changing their constraint attribute. When re-enabled, you may need to use *Snap Vertices to Reference Geometry* to update the locations.

Select Constrained

This retopo-specific selection filter allows you to select all adjacent constrained faces or vertices, *or*, all un-constrained adjacent faces or vertices. The possible selections will highlight during hover. The selection filter works with vertex, edge and face selection.

Select Constrained

This retopo-specific selection filter allows you to select all adjacent constrained faces or vertices, *or*, all un-constrained adjacent faces or vertices. The possible selections will highlight during hover. The selection filter works with vertex, edge and face selection.

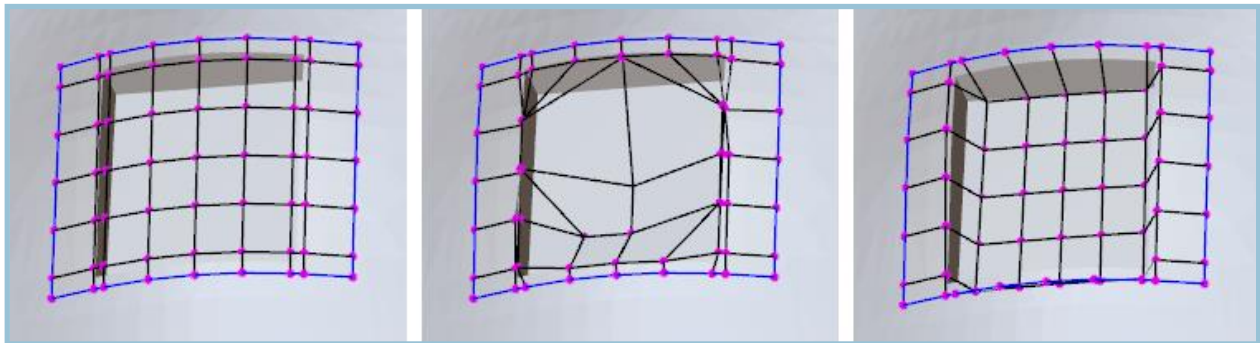


The retopo mesh showing constrained (magenta) and un-constrained vertices (black), left, adjacent constrained vertices, center, and adjacent unconstrained vertices, right.

Constraint Snapping Options

When snapping newly constrained vertices to the reference mesh, you have two options: *Snap Closest Point*, where a vertex will snap to the closest point on the reference mesh, or *Snap Normal*, where the vertex will move along its vertex normal (the average of its surrounding faces' normals) until it reaches the reference mesh's surface. A face normal is an imaginary line drawn perpendicular to the face.

In the following example, Constrained mode was turned off while the horizontal subdivisions were created over the recessed part of the model. In the center image, *Snap to Closest Point* was set before turning on Constrained mode and using *Snap Vertices to Reference Geometry*. In the far right image, Snap Normal was used. The snap to Normal option may give better results when creating the retopo mesh outside of *Constrained to Mesh* mode and snapping onto the reference mesh afterwards.



Snapping options: Constrained to Mesh turned off, left, Snap Closest Point, center, Snap Normal, right

Show Ref. Mesh On Conversion

Sometimes it is useful to see the reference mesh as you are working with Solidworks tools outside of the Power Surfacing editing mode. This check box will keep the reference mesh visible and allow you to create Solidworks geometry (sketches, holes, bosses, etc.) while using the reference mesh as a visual reference.

The Power Surfacing Reverse Engineering Tools

The retopologizing tools are found in the Power Surfacing RE tab and the Command panel. Most of the tools and features are available only during the retopo process, e.g., when there is a reference mesh available. The RE Tools allow you to create or modify geometry that is based on a reference mesh. During the retopo process, many of the most useful tools can also be found in the lower addition to the right-click menu.

Edit Reference Mesh

In this mode, you are able to make changes to the reference mesh. Along with some of the familiar selection tools and filters, it has some additional options as shown in the Command panel.

Selection Utility Panel

There are two selection options in this mode: Select Faces and Select Element. Select Element is useful for removing stray small pieces that often come with scanned meshes. See the workflow example on the next page.

Select All and **Select Invert** work the same way as they do in SubD editing mode.

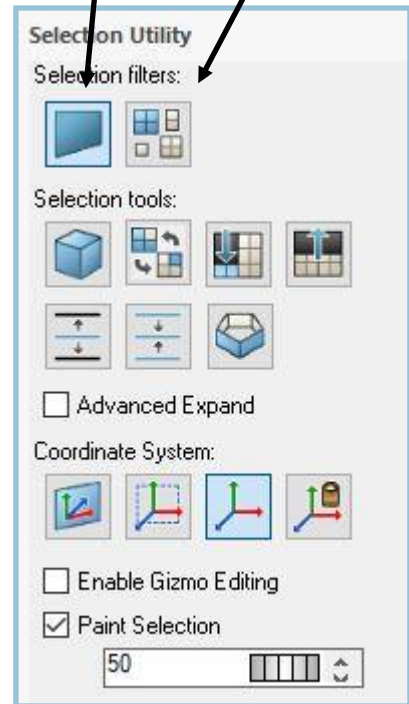
Hide Selected Faces and **Unhide All** - This option allows you to hide the selected faces, allowing you to edit the reference mesh at any time.

Expand Selection and **Contract Selection** and **Advanced Expand** work the same way the do in SubD editing.

Enable Gizmo Editing is off by default. This allows paint selection to work without moving anything by mistake. To move and rotate the entire mesh or selected parts of the mesh using the Gizmo, you need to enable this.

Paint Selection lets you paint or select faces using a circular range area. It only selects front facing faces. It is on by default when entering this mode

Select Faces Select By Element



Right Click Menu

In the right-click menu and tool bar,

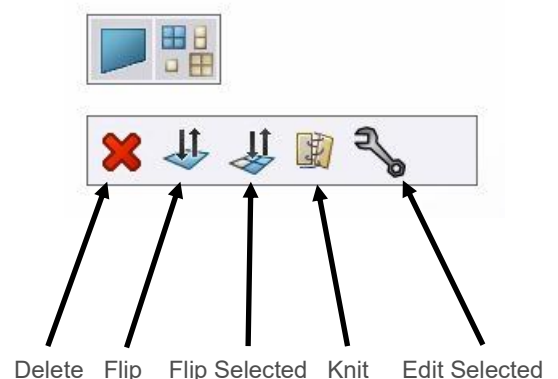
Delete: *deletes selected faces.*

Flip Reference flips orientation of all faces in reference mesh.

Flip Selected flips orientation of only selected reference mesh faces.

Knit allows you to reconnect edges of a reference mesh.

Edit Selected is a powerful function that allows you to edit a subset of the reference mesh using the standard SubD modeling tools. Typically you will fill holes and smooth areas using this tool. See the Edit Selected workflow example in the following pages.

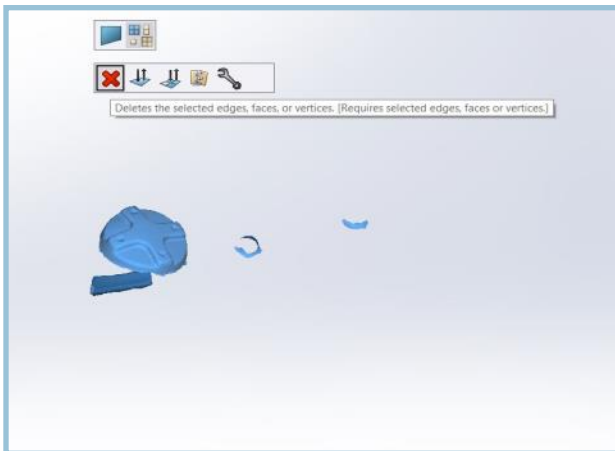
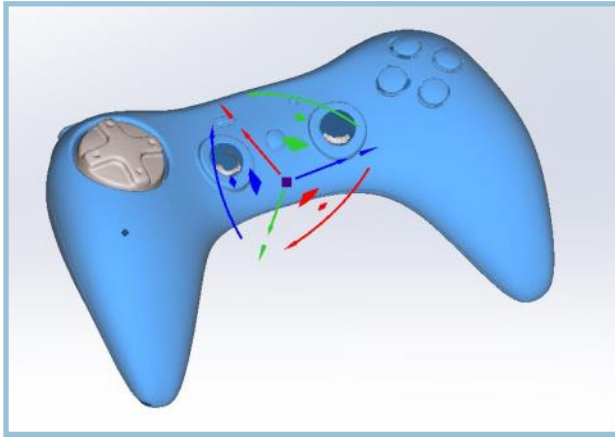


Select By Element Workflow

One of the first things you should do with a scanned mesh is to remove any small disconnected pieces from the main body that you don't plan to use in your reverse engineering process. Often scans have small pieces that should be removed. Also there may be pieces of the scan that you don't want to use and you can use the paint tool with Delete to remove them from the mesh.

The following example shows this workflow:

- 1) Turn off *Paint Selection* and Turn on *Select By Element*. Note that *Select By Element* doesn't work when *Paint Selection* is on.
- 2) Select the parts of the mesh you wish to keep as in the upper left image.
- 3) Use *Hide Selected Faces* to temporarily hide the main body of the mesh. (Another option would be to use *Invert Selection*.)
- 4) Now use fence selection to select all of the remaining pieces of the mesh and delete them using the right click *Delete* button as shown in the lower left image.
- 5) *Unhide All* to show the main body of the mesh and begin working on it as shown in the lower right image.



Mesh Orientation and Alignment Tools

Often an imported mesh needs to be moved and oriented prior to being cleaned up and retopologized. The mesh orientation tools provided in the *Tools / Power Surfacing / Reference Mesh Tools* menu will enable you to do this. All of the orientation tools require you to select one or more faces. For orientation it will use the average normal of the selected faces as the reference. For movements it will use the center of the selection which is at the average of all of the vertices of the selected faces.

Orient to X/Y/Z-Axis

These tools will allow you to orient the entire mesh. The average normal of the selected faces on the mesh will be oriented to the X, Y or Z axis. For example you may select a face on the top and orient it with the Z axis.

Orient to Negative X/Y/Z-Axis

These tools will allow you to orient the entire mesh. The average normal of the selected faces on the mesh will be oriented with the negative X, Y or Z axis. For example, you might select a face that should be on the bottom and orient it with the negative Z axis.

Move to Origin

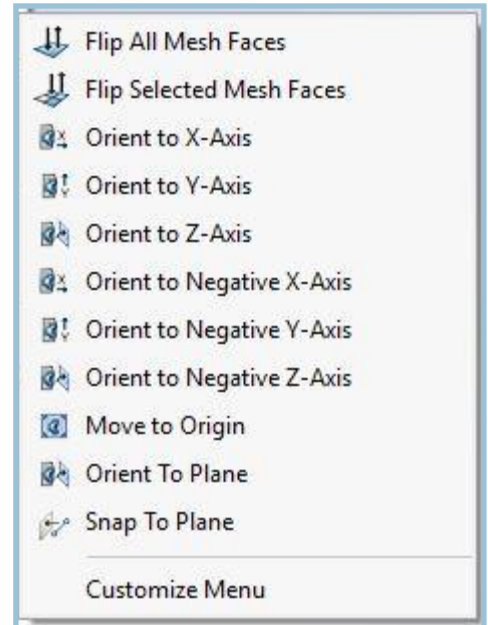
This command moves the entire mesh by moving the center of the given selection to the origin of the world coordinate system.

Orient To Plane

This command orients the entire mesh to the the normal of the selected faces to the normal of a selected construction plane.

Snap To Plane

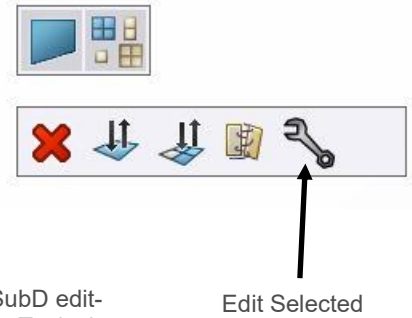
This command moves the entire mesh to by moving the center of the given selection to the origin of the world coordinate system.



Edit Selected Workflow

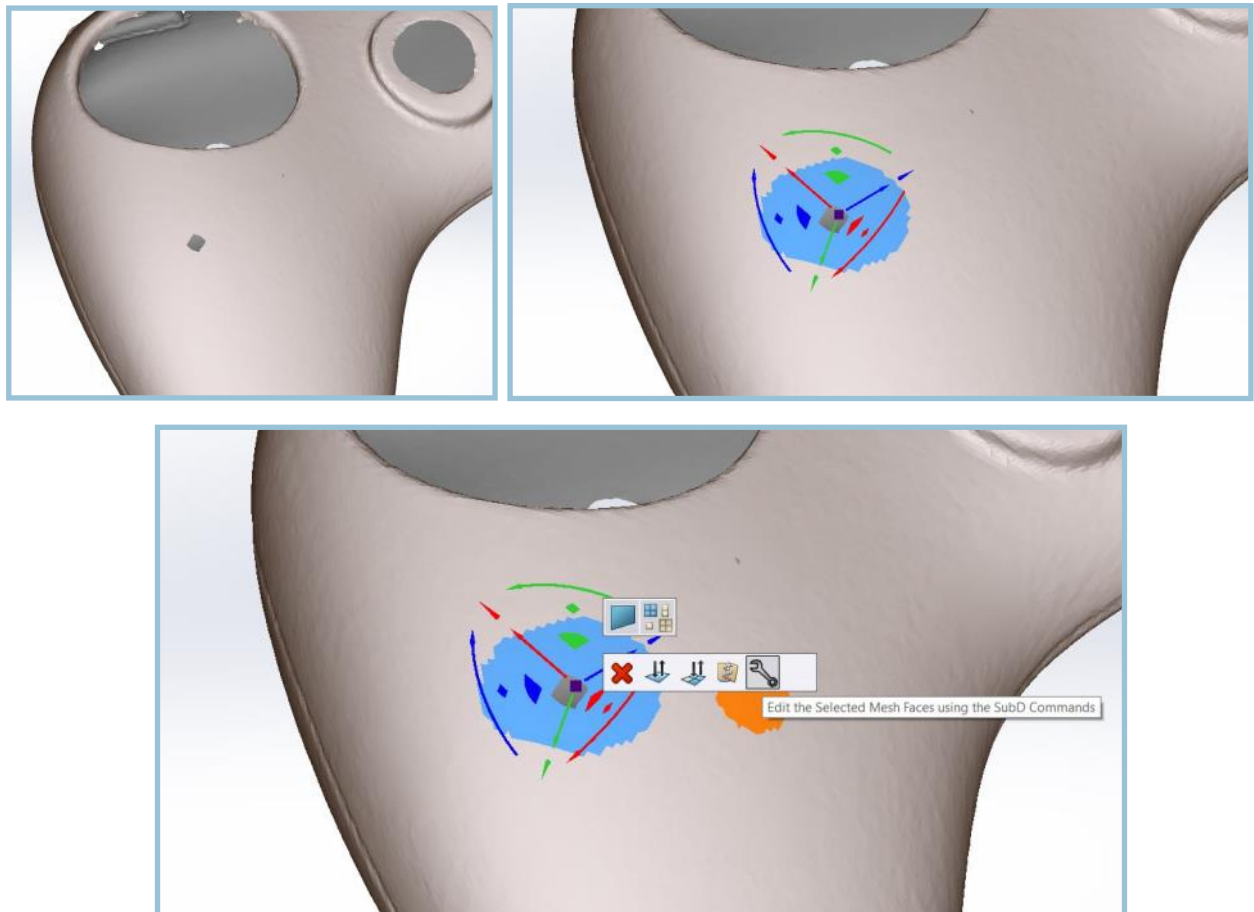
Using Paint Selection along with the Edit Selected workflow allows you to extract a small subset of the often very large reference mesh and edit it using the SubD modeling editing tools. We have found this to be a huge advantage both in terms of performance and functionality over previous workflows possible in earlier versions of Power Surfacing RE. The basic procedure is as follows:

- 1) Use Paint Selection to select a subset of the reference mesh to work on.
- 2) Use the right click menu to invoke *Edit Selected*, which will put you into standard SubD editing mode with the selected subset where all of the SubD editing tools are available. Typically you will want to turn the visibility of the edges to on.
- 3) Use the SubD tools to fill the holes, smooth an area of the mesh, patch an area of the mesh, fill in missing geometry, etc. Note that you should be somewhat careful not to modify the outer boundary where the edited part of the mesh shares edges with the original reference mesh. Otherwise you might create an unexpected tear in the mesh.
- 4) Finish editing using the Green Check to push the edited subset back into the reference mesh.
- 5) Repeat this process for different parts of the mesh that need to be cleaned up.



Lets start with a simple example of filling a small hole in the mesh.

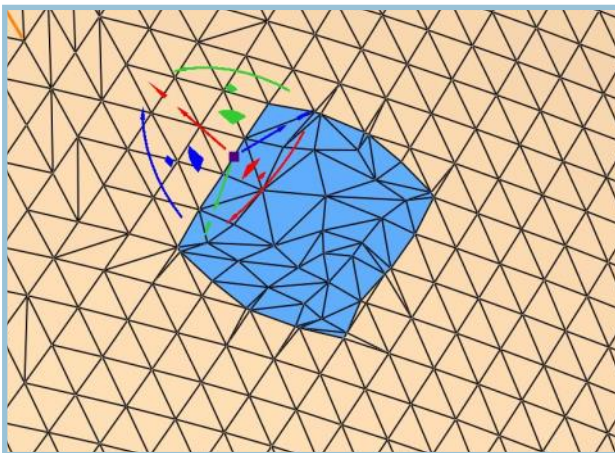
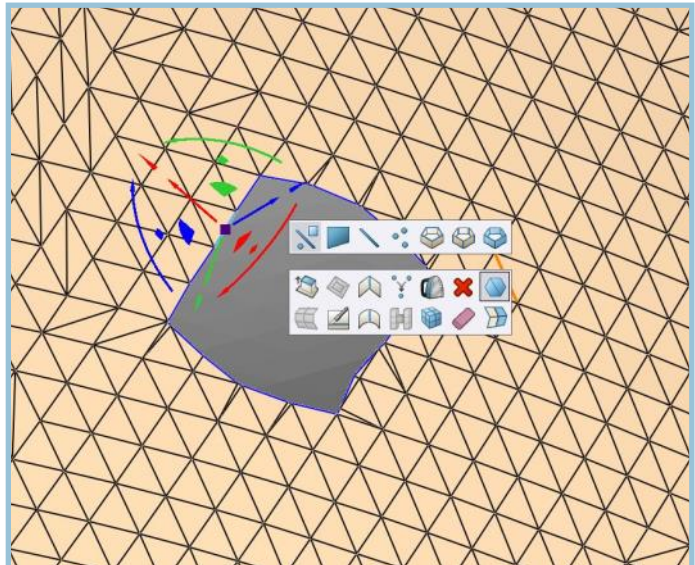
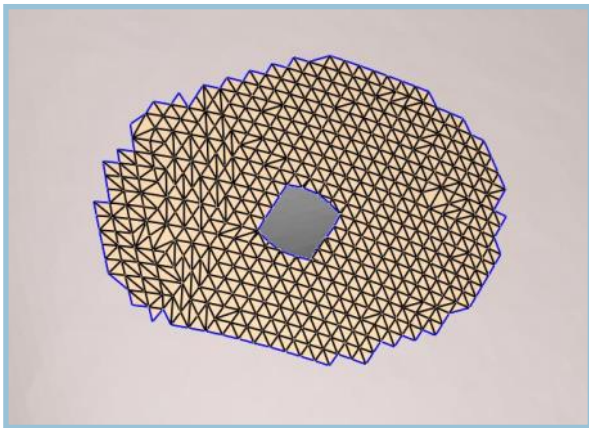
The upper left image below shows the small hole in the original mesh. The upper right image below shows the result after paint selection. The lower image below shows *Edit Selected* being picked in the right click menu.



Edit Selected Workflow (continued)

The upper left hand image shows the subset of the mesh we are editing after we have turned on edge display in the SubD editing mode. The upper right hand image shows selection of an edge on the hole and right click to select *Quad Fill*.

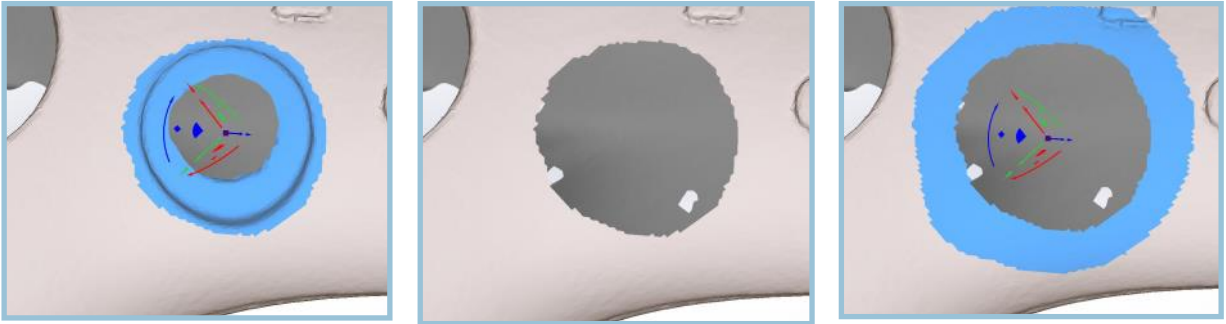
The lower left image shows the result of the *Quad Fill* operation. Note that *Quad Fill* will internally use the *Smooth* command with one ring of faces to match the curvature of the surrounding mesh. In cases with larger holes you may want to invoke a separate *Smooth* with a higher number of face rings. The lower right image is what is shown after we use the Green Check to save the result of our edit.



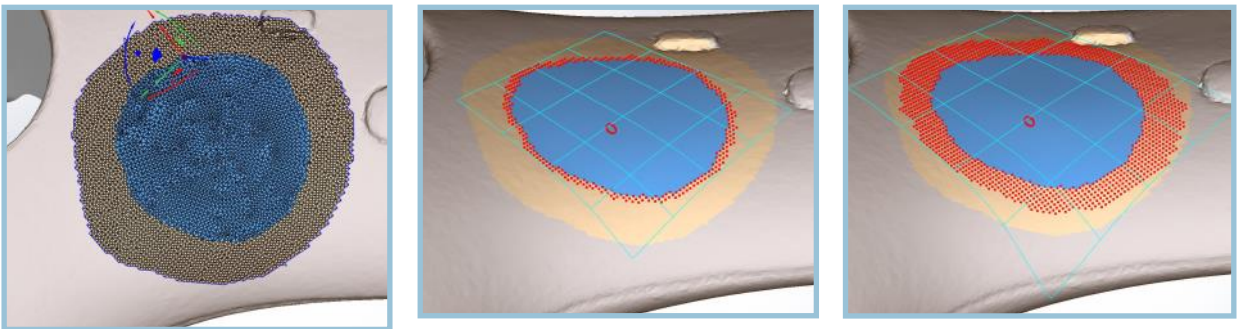
Edit Selected Workflow (continued)

Now let's take a look at a more complex example where we have to remove a feature, and then fill a larger hole in the mesh where it is important to match the existing curvature.

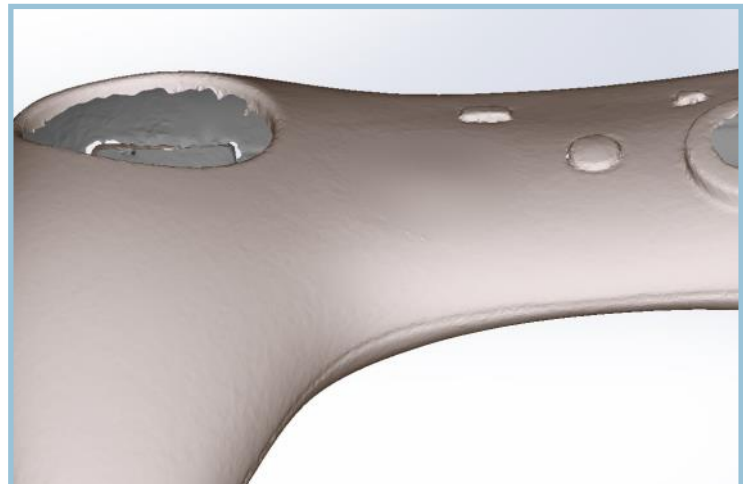
- 1) Use Paint Selection and right click Delete to remove the ridge on the mesh, as shown in the first two images below.
- 2) Now select around the hole to fill and Edit Selected that area, as shown in the right image below.



- 3) Now use the *Quad Fill* command to fill the hole in SubD editing mode, as in left image below.
- 4) Use the *Smooth* command, as shown in the middle image below.
- 5) Change the number of rings in the *Smooth* command to 6, as shown in the right image below.

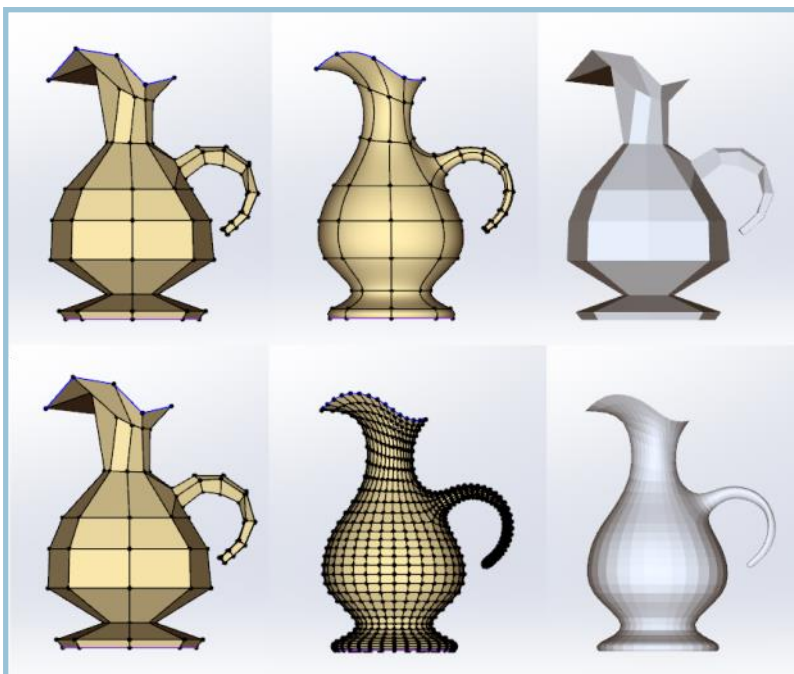


- 7) The image on the right shows the final result in the reference mesh.



Retopologize Object

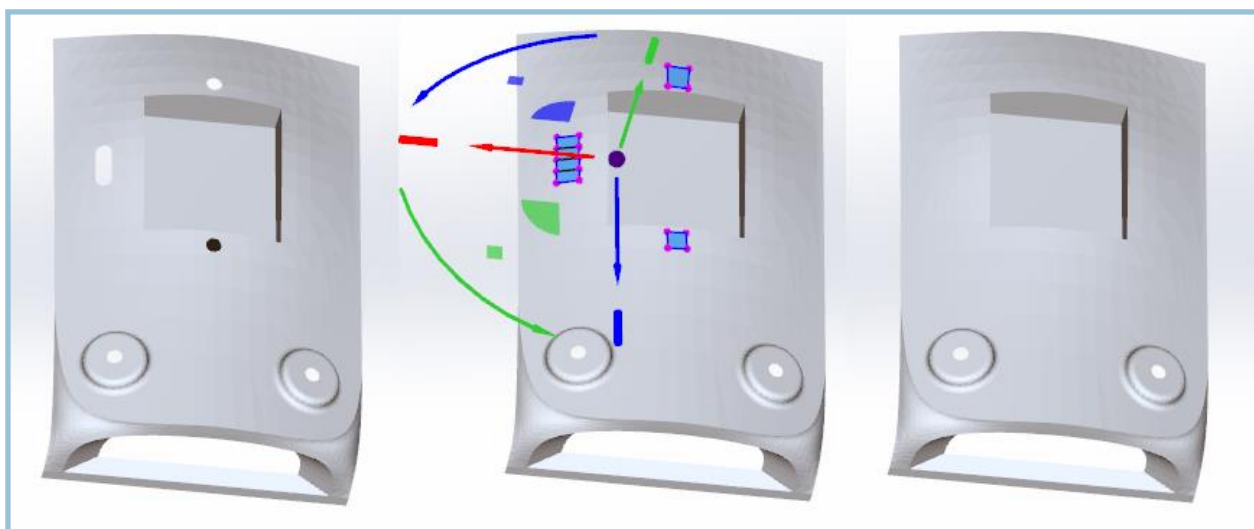
This command turns a polygonal mesh (SubD or triangle-based) into a reference mesh. Note that the object's history, if any, will be lost on conversion. Only the Control Mesh is used for conversion so you should use *Subdivide All* beforehand to give your reference mesh enough resolution to work with.



A SubD converted directly to a reference mesh, top row, the same SubD with two Subdivide All operations before converting

Add Selected to Mesh

This command will add the selected SubD faces to the reference mesh. A selection set will be created and added to the reference mesh's selection sets. Note this is a somewhat obsolete method of fixing holes. See the Edit Selected method in editing a reference mesh. It is much easier to fix things that way.

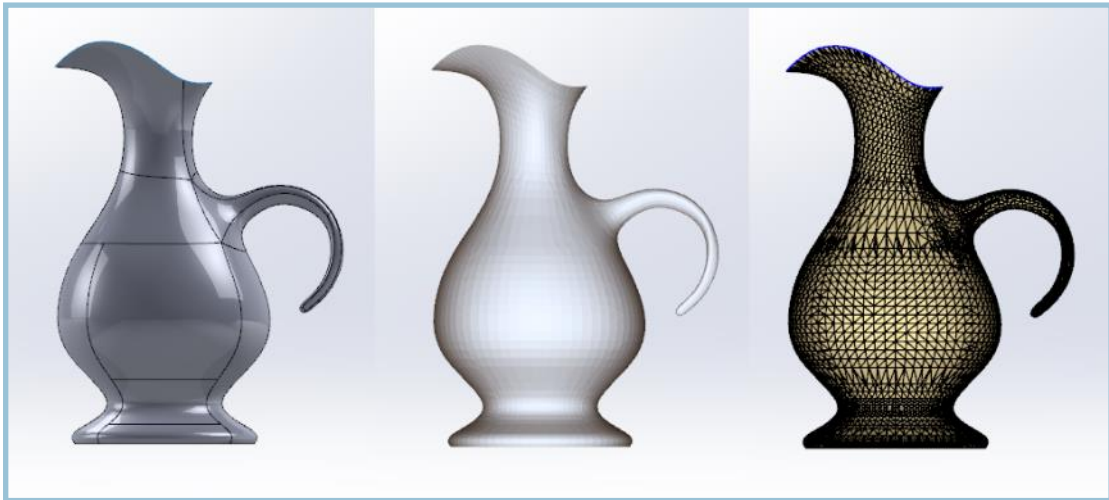


A reference mesh with undesirable holes, left, creating SubD patches, center, and added in as part of the reference mesh, right



Create Poly from Reference

This command will create a polygonal mesh from a reference mesh. It will probably not produce a quad-based mesh suitable for SubD modeling, but will provide an alternate means of repairing the mesh or setting up selection sets as it has access to all of the regular Power Surfacing selection and selection filtering tools.

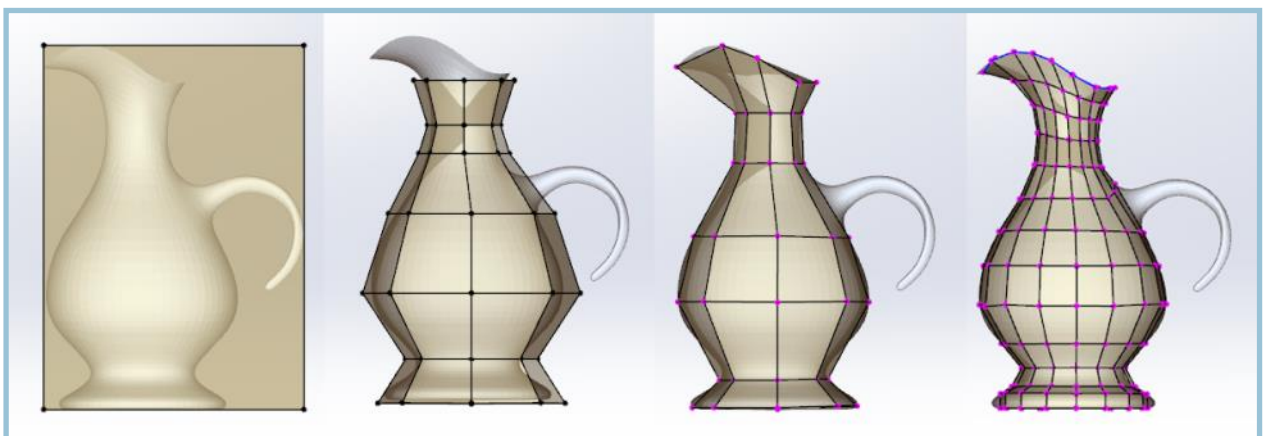


Using a solid body as a reference mesh and converting it to a polygonal mesh



Bounding box

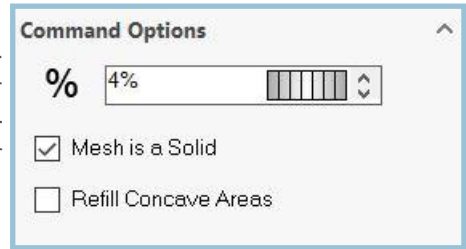
This tool creates a simple bounding box around the reference mesh as the starting point for your SubD. Unlike other retopo tools, the vertices will *not* be set to Constrained. Use this method to block in your retopo mesh where you can block in a simple proxy mesh, constrain the vertices and then subdivide it.



The bounding box, far left, blocked in, center left, vertices set to constrained, center right, one more subdivision, far right

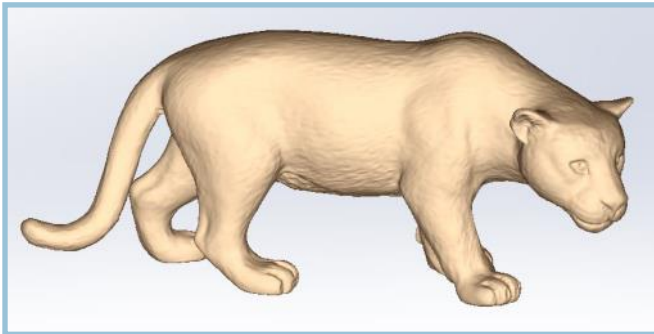
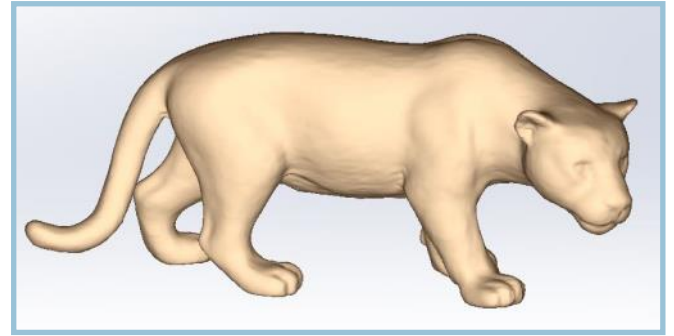
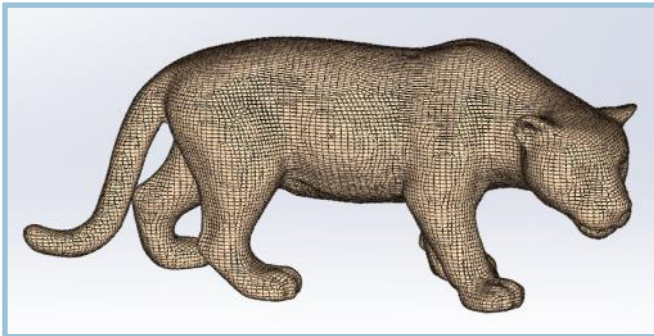
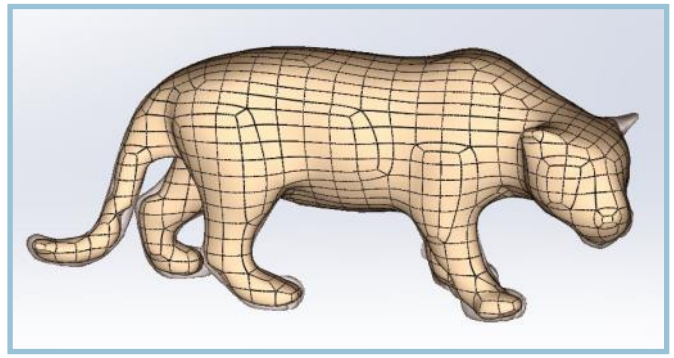
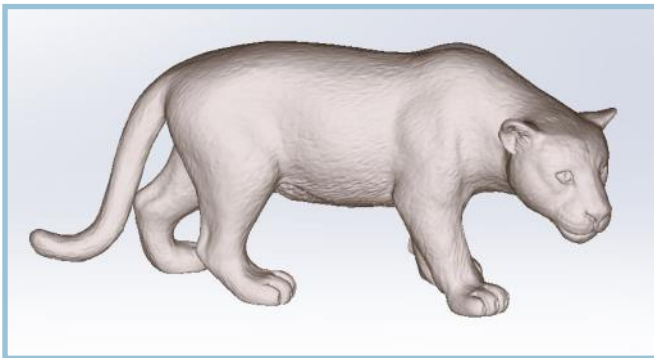
Quad Wrap

Quad Wrap is a powerful tool to automatically retopologize objects. You can control the size of the quad relative to the model size in the command options. If your mesh is not a closed solid, you should uncheck *Mesh is a Solid* to get correct results. *Refill Concave Areas* can be used to fill in concave areas that are smaller than the quad size. It can also be done manually using Smart Fill.



The following sequence of images shows a typical workflow:

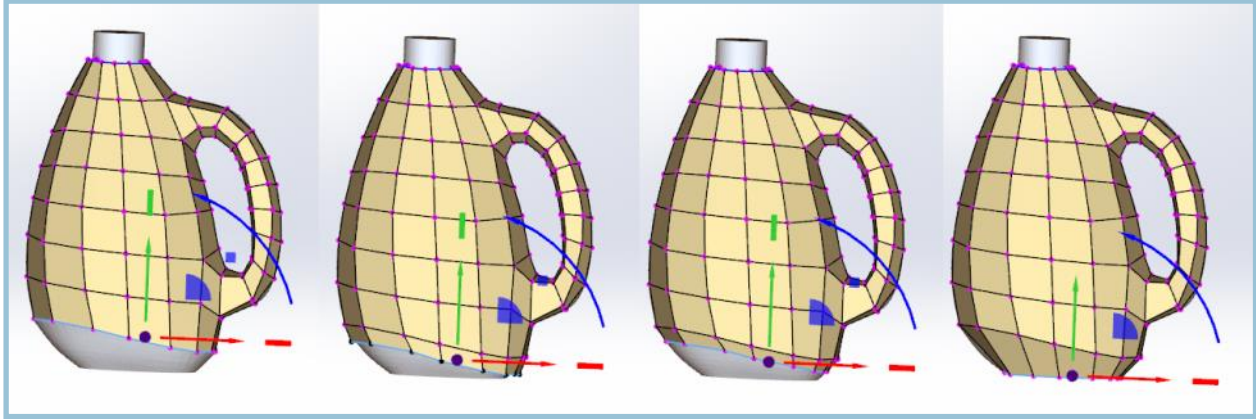
- 1) Top Left is the original scanned mesh
- 2) Top right is the default Quad Wrap with 4% quad size.
- 3) Middle Left shows a 1% quad size. Note that going lower than 0.5% is not recommended.
- 4) Middle Right shows 1% quad size without the edges.
- 5) Bottom Left shows the result of Shrink Wrap with 1 subdivision level.
- 6) Bottom Right shows the resulting Solidworks body which comes in as a valid solid body.





Add Constraint

This tool sets the constrained attribute to true on the selected vertices. If edges or faces are selected, the vertices in the selections will be set to Constrained. Constrained vertices will snap to and stay on the surface of reference meshes while in Constrained to Mesh mode.

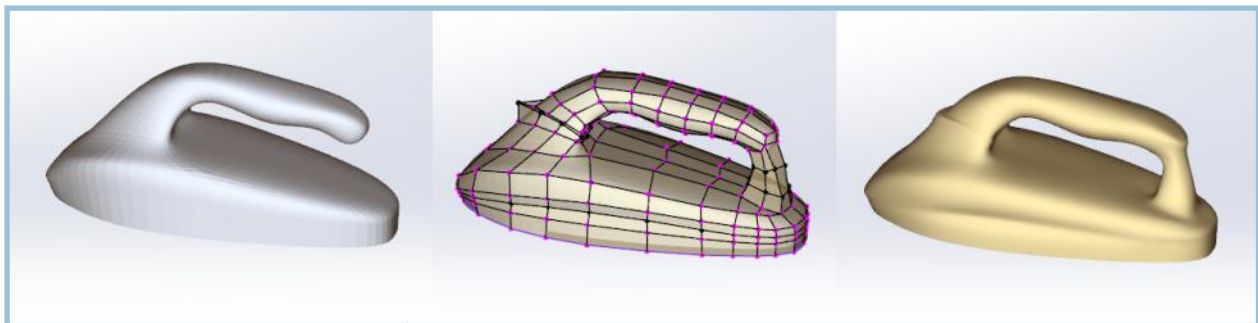


Constrained vertices selected, left, Extend generating unconstrained vertices, left-center, new vertices set to constrained, center-right, newly constrained vertices pulled down to base, right



Remove Constraint

This tool sets the constrained attribute to false on vertices. If edges or faces are selected, the vertices in the selections will be set to unconstrained. Unconstrained vertices will not snap to and stay on the surface of reference meshes regardless of mode.

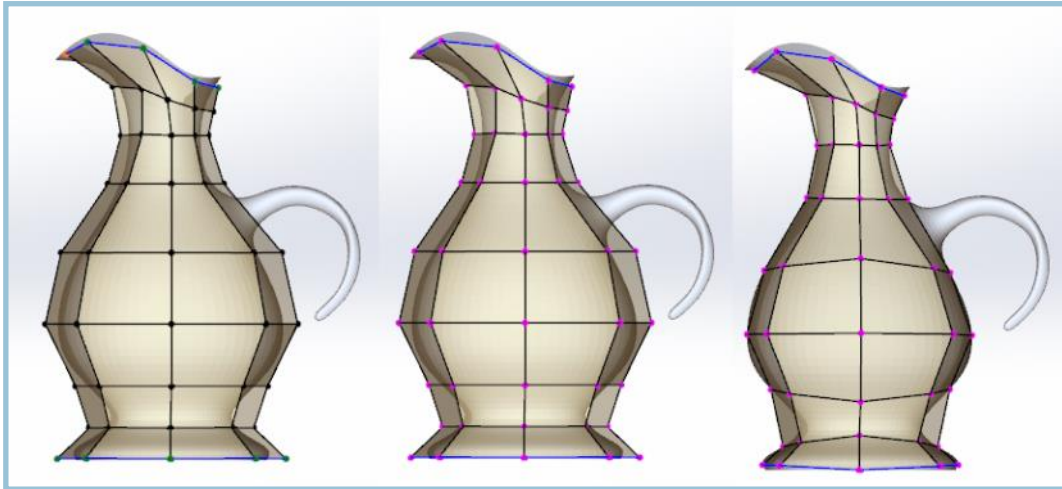


The original reference mesh, left, the retopo'd version with unconstrained vertices used to modify the original, center, and the altered version, right



Snap Vertices to Reference Mesh

This command snaps all constrained vertices down to the reference mesh. Use this command when you have been working with Constrained to Mesh mode turned off. Turning *Constrained to Mesh* mode on will not automatically snap constrained vertices down to the mesh.

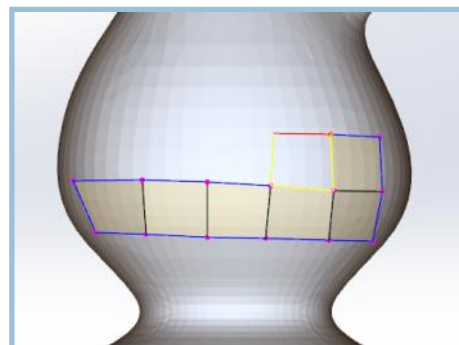


A proxy mesh created without constraints, left, setting the vertices set to Constrained, center, the constraints updated using Snap Vertices to Reference Mesh, right



Draw Edges

This is the most useful of the retopo tools. It lets you create quads directly on the reference mesh, showing previews and filling in whenever possible. Use <backspace> to undo edges while you are still in Draw Edges mode. Right-click to end Draw Edges mode. Orphaned, or red edges, will cause any geometry created after them to be deleted upon Accept, so the Draw Edges command should be restarted as soon as they are detected.



Drawing edges where the proposed face is shown in preview before the last vertex



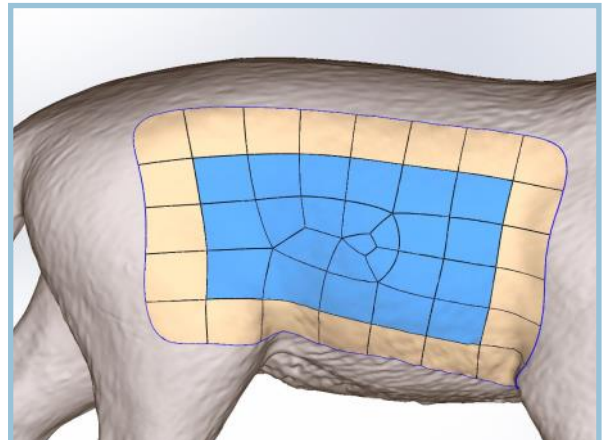
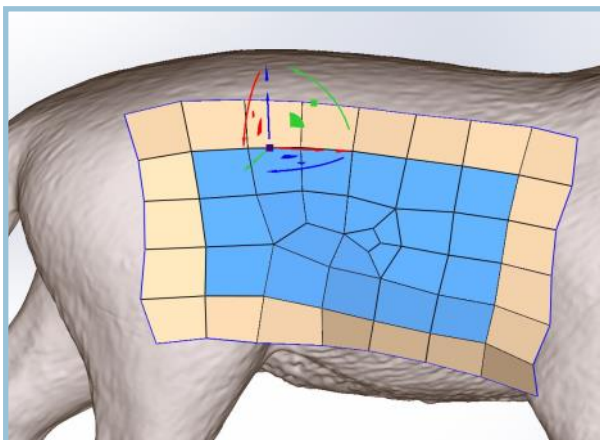
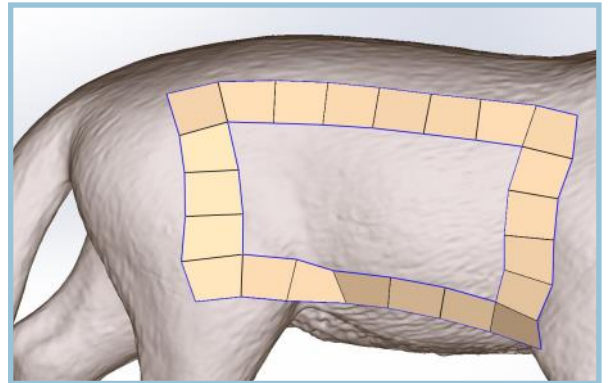
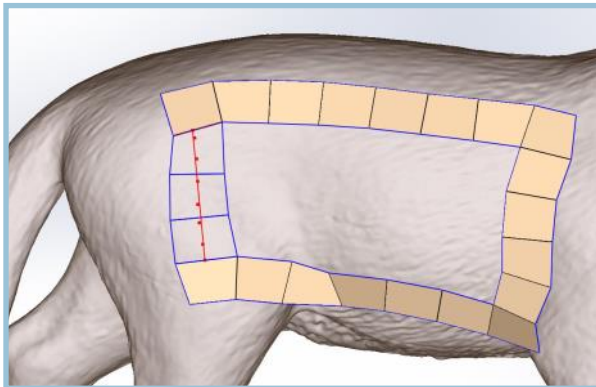
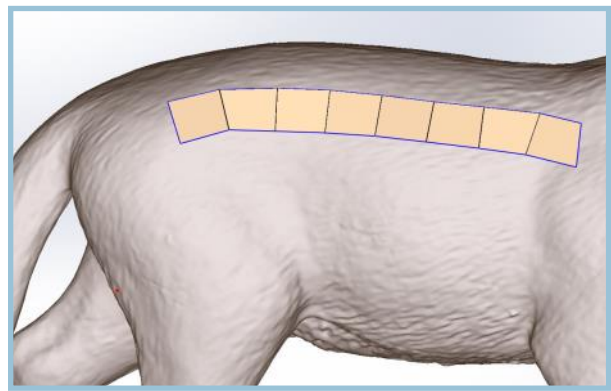
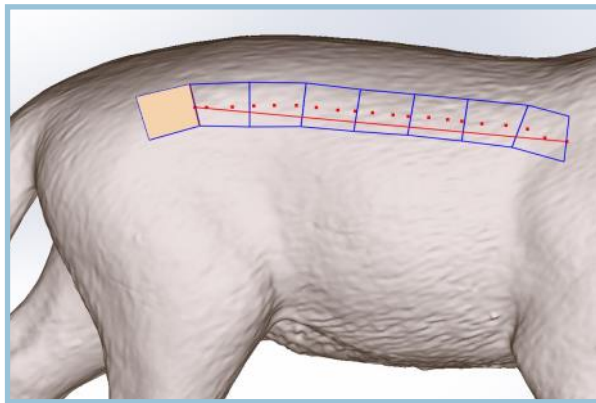
Paint Faces

This tool lets you create a row of similarly shaped and sized faces with one stroke of the brush. After starting the tool hold down the mouse either on a face or on an edge and drag the mouse along the path to create a string of faces. You will see a preview of the faces as you draw. The tool is very smart and will allow you to connect the end into faces or drag along beside an existing row of faces.

First we create an initial quad using Draw Edges of the appropriate size. Then we use Paint Faces making a square area with only 4 painting strokes. Note that the final stroke automatically connects back when we move the cursor over the starting face. Typically you might finish this sequence by using Smart Fill to fill in the interior and Shrink Wrap to snap it down to the mesh.

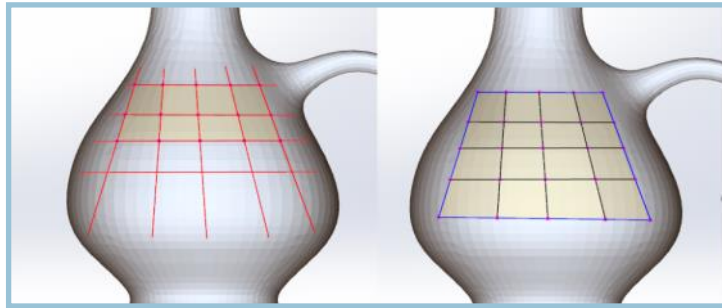
Action Required

Left-click and hold a starting edge or face and drag the mouse to create faces. Release the mouse button to finish. Hold Shift makes linear layout.



Face Grid

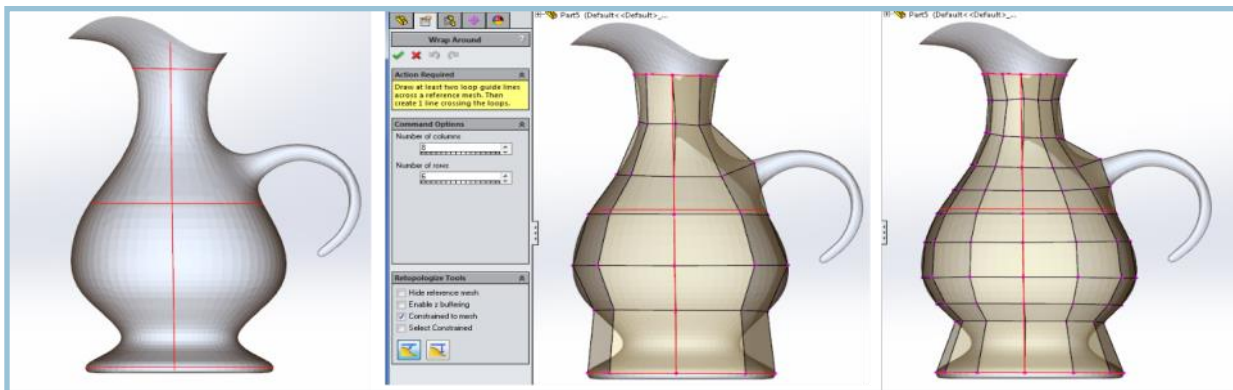
This tool lets you create a quick grid of horizontal and vertical edges. When four edges are closed, a face or quad is created on the reference mesh. You can freehand lines or click to make straight lines.



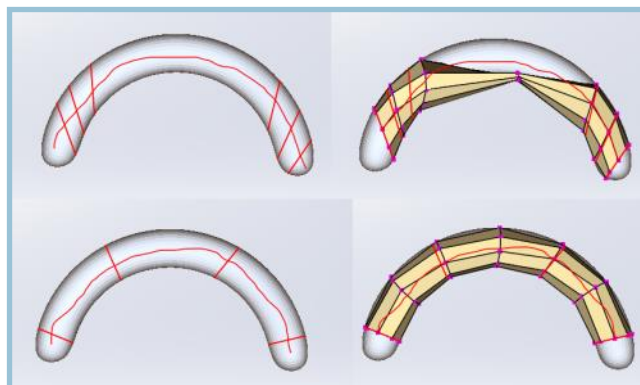
The grid guide lines, left, and the finished faces, right

Wrap Around

The Wrap Around tool lets you Retopologize cylindrical areas of your reference mesh by creating a few guidelines and then crossing them with a final line. You can then use the controls in the Command Panel to specify the number of rows and columns you want. The position and orientation of internal guide lines is used to generate the mesh on more complex reference meshes.



Wrap Around: The guide lines, left, changing the rows and columns, center, the new rows and columns

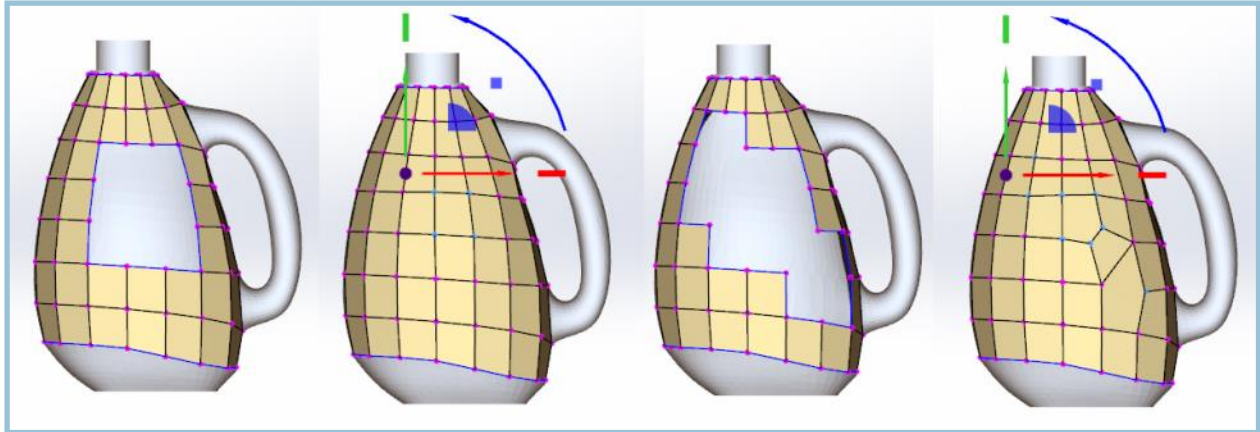


Affecting the Wrap Around with internal guide lines: poorly placed, top, well placed, bottom



Smart Fill

Smart Fill will fill and attempt to retopologize a hole in your retopo mesh. The more evenly spaced the surrounding faces, the better the result will be. You can also use Erase and Insert Edges to reconfigure the result if necessary, as it may be quicker to repair the result rather than manually subdivide the entire hole after a *Fill Face*. To use Smart Fill, select at least one of the open edges around the hole.

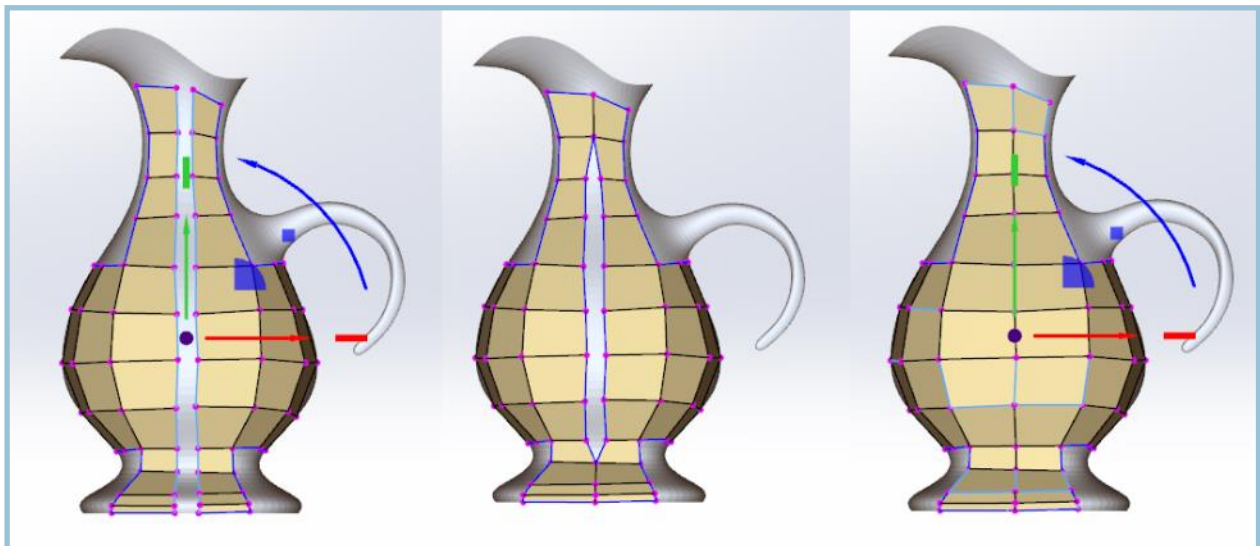


Smart Fill used on a simple opening, left, and a more complex opening, right



Glue Edge Group

This tool welds lines of edges together as long as both ends are open. You must have the same number of edges selected on both sides.

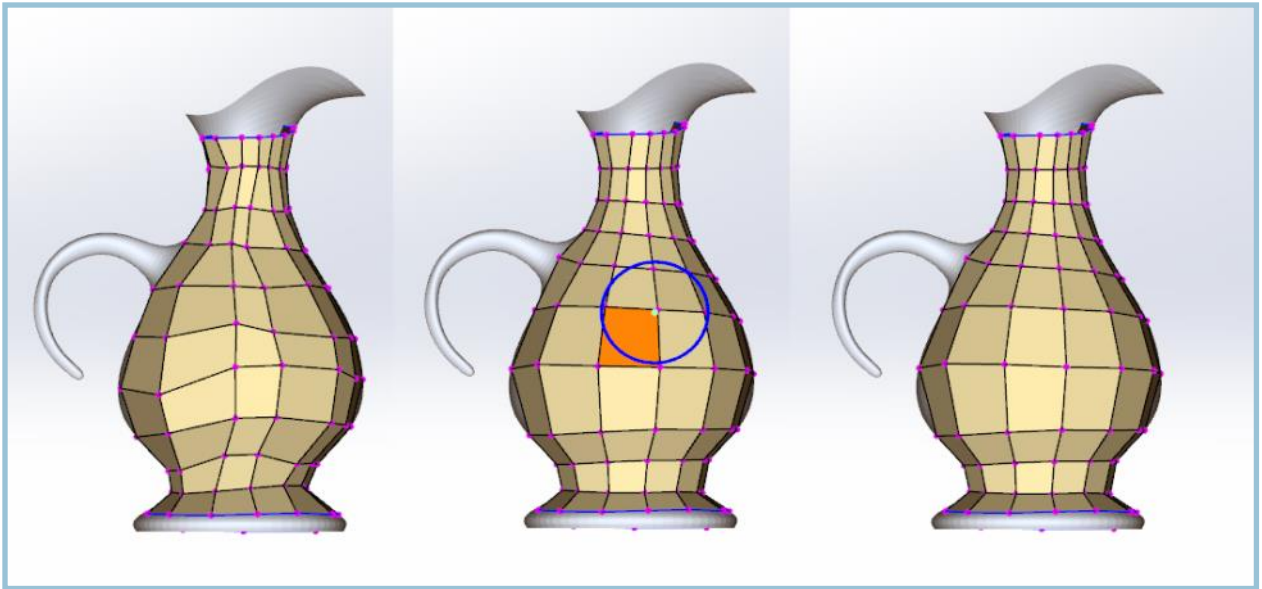


Glue Edge Group used on two open ended edges, left, a close-ended set of edges where the end edges are not part of the selections, center, the end result for both scenarios, right



Paint Relax

This tool averages the distance between vertices as it is painted over a retopologized surface. It should only be used on constrained vertices in *Constrained to Mesh* mode.



Unevenly spaced vertices, left, using Smooth to even out the spacing by 'paint' the area, center, the end result, right- all vertices remain constrained to the reference mesh surface



Shrink Wrap (formerly Interpolation)

Shrink Wrap pulls the SubD down closer to the mesh by embedding information into the mesh to make the subdivision converge to the mesh. The number box controls the number of subdivisions to use in the shrink wrap process.

Default Shrink Wrap keeps the shrink wrap in a table which often gets destroyed during editing operations. This was the previous default.

Bake into Offset will subdivide the mesh and perform an offset such that the subdivided mesh of the offset converges down to the reference mesh. This operation removes all of the constraints and you are free to edit the resulting mesh in any way.

Bake into Level of Detail will move the shrink wrap table into the LOD table. Most operations such as Extrude support the LOD table, thus it is much more editable than the default option.

The top left image shows the default of 0 levels in Shrink Wrap. The top right image shows shrink wrapping using 2 levels. Note the much higher accuracy.

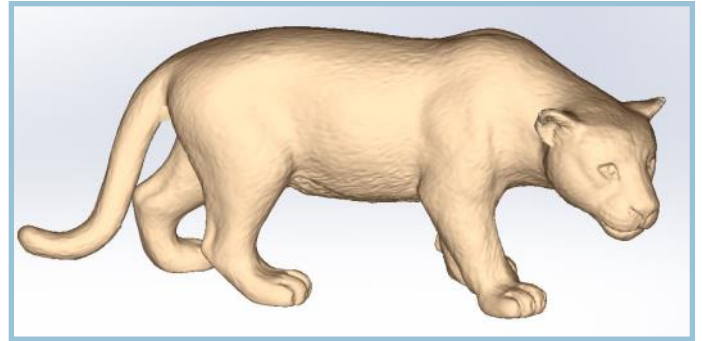
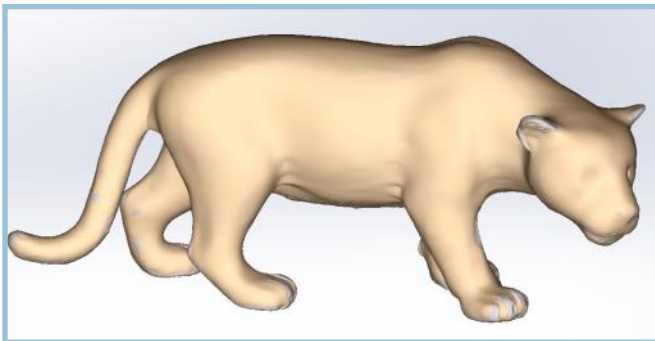
Action Required ^

Please accept the interpolation mesh as your edit mesh if you are happy with the results. Otherwise, cancel to return to your original mesh. Use the interpolation subdivision levels to adjust accuracy but note that it will increase surface complexity.

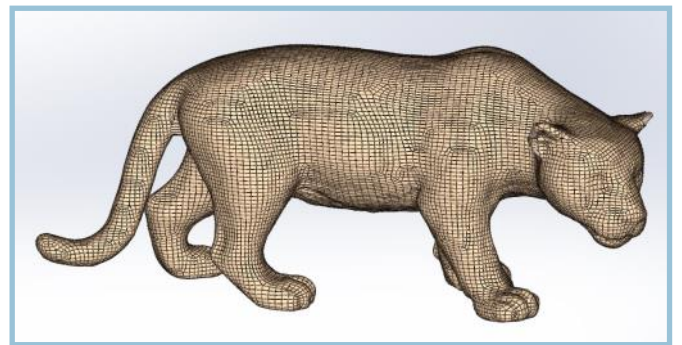
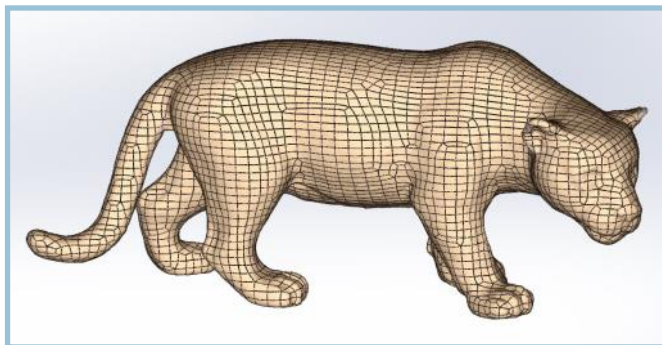
Command Options ^

Shrink Wrap Output Type:

- Default Shrink Wrap
- Bake into Offset
- Bake into Level of Detail



The left image below shows Bake into Level of Details. Note that the number of polygons has not changed with this operation. The right image shows Bake into Offset with 1 level of subdivision. Note that there are now 4 times as many faces.







Clear Shrink Wrap

Clear Shrink Wrap will reset the *Shrink Wrap* table and clear it out. This only works if you are using the Default *Shrink Wrap*. A typical workflow would be to do a *Shrink Wrap*, test the quality using the Distance command and then use *Clear Shrink Wrap* and address any problem areas.

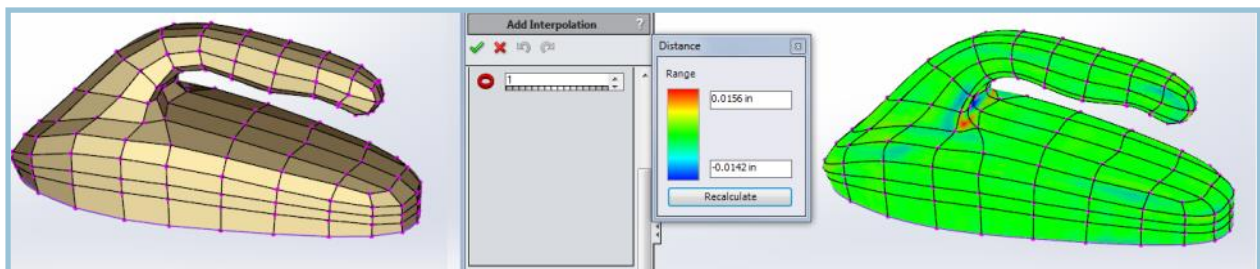


Distance

The Distance display toggle will color the retopo mesh according to how far away the vertices, edges and faces are to the corresponding reference mesh. When it is first turned on for a mesh, the maximum distances for above and below the mesh's surface are set as the Range. Red indicates that the retopo mesh is above the reference mesh while blue indicates that it is below the reference mesh. The range can be recalculated at any time and can also be set manually. The colors update as the vertices, faces and edges are repositioned. The tool can be used to visually check the accuracy of the retopo mesh with the reference mesh. The work flow is as follows:

1. In Control Mesh display mode, retopologize the model.
2. Click Add Interpolation, , and wait for procedure to be completed.
3. Activate the Distance display dialog, .

The Range of distance the retopo mesh is from the reference mesh is color-coded with red being above the ref mesh and blue being below the mesh. Green is considered on the mesh within the Ranges shown.



The Control mesh, left, using Add Interpolation, one subdivision, and the range of distance of the interpolated mesh, center, the mesh showing high (red) and low (blue) deviation from the reference mesh, right

Distance (continued)

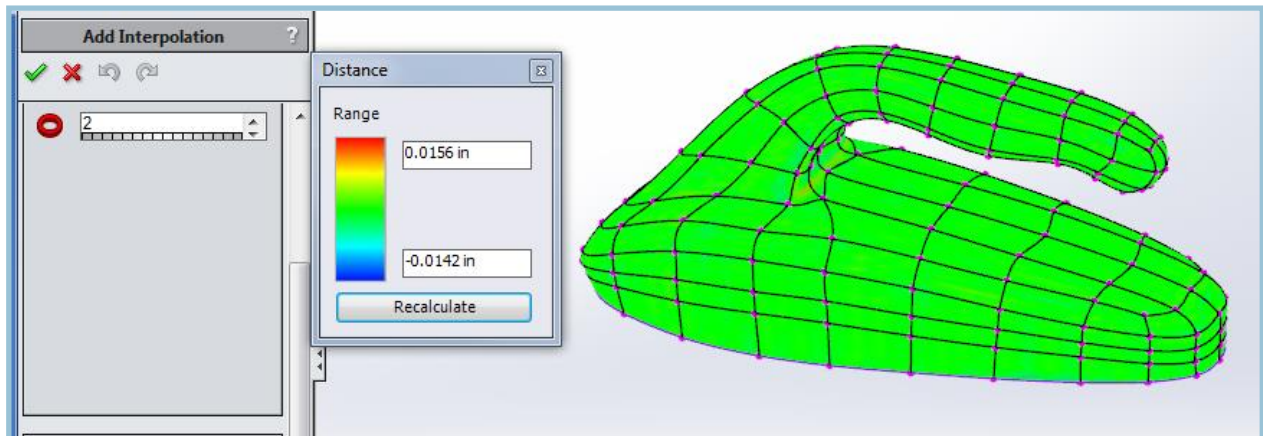
4. Observe the Range to determine the accuracy of the retopo SubD.

5. If it is not close enough, increase the number of subdivisions.

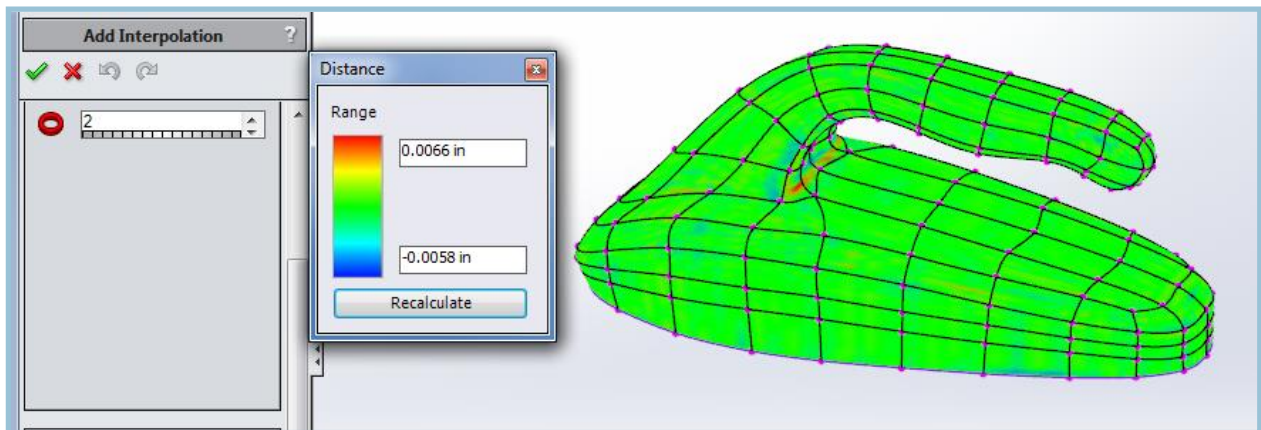
Two subdivisions will generally be enough. Be aware that more subdivisions will take more time to convert to SOLIDWORKS solids or surfaces.

Tip: For scanned meshes where the surface did not scan well, you may prefer to keep the default single subdivision to avoid unwanted detail.


6. When the model has been updated, press Recalculate to see the updated Range. Ideally, you will want to see several pale splotchy areas showing a fairly even distance variation.



A second subdivision producing a closer fit, with range not yet updated, left, before the updated color feedback, right



A second subdivision producing a closer fit, with range not yet updated, left, before the updated color feedback, right



Distance (continued)

For problem areas, consider canceling the Add Interpolation, adding or adjusting edge loops for better coverage, and using Add Interpolation. If you have already accepted the interpolation, you can use Clear Interpolation before making changes.

Tip: The only time the Range is automatically updated is with the first use of the Distance tool in a session. A good practice is to click the Update button after Add Interpolation, unless, of course, you would prefer to use the previous values. An example would be after adding unconstrained modifications to the retopo mesh.

If a specific Range is required, you can set the minimum and maximum ranges manually.