POWER SURFACING RE HELP

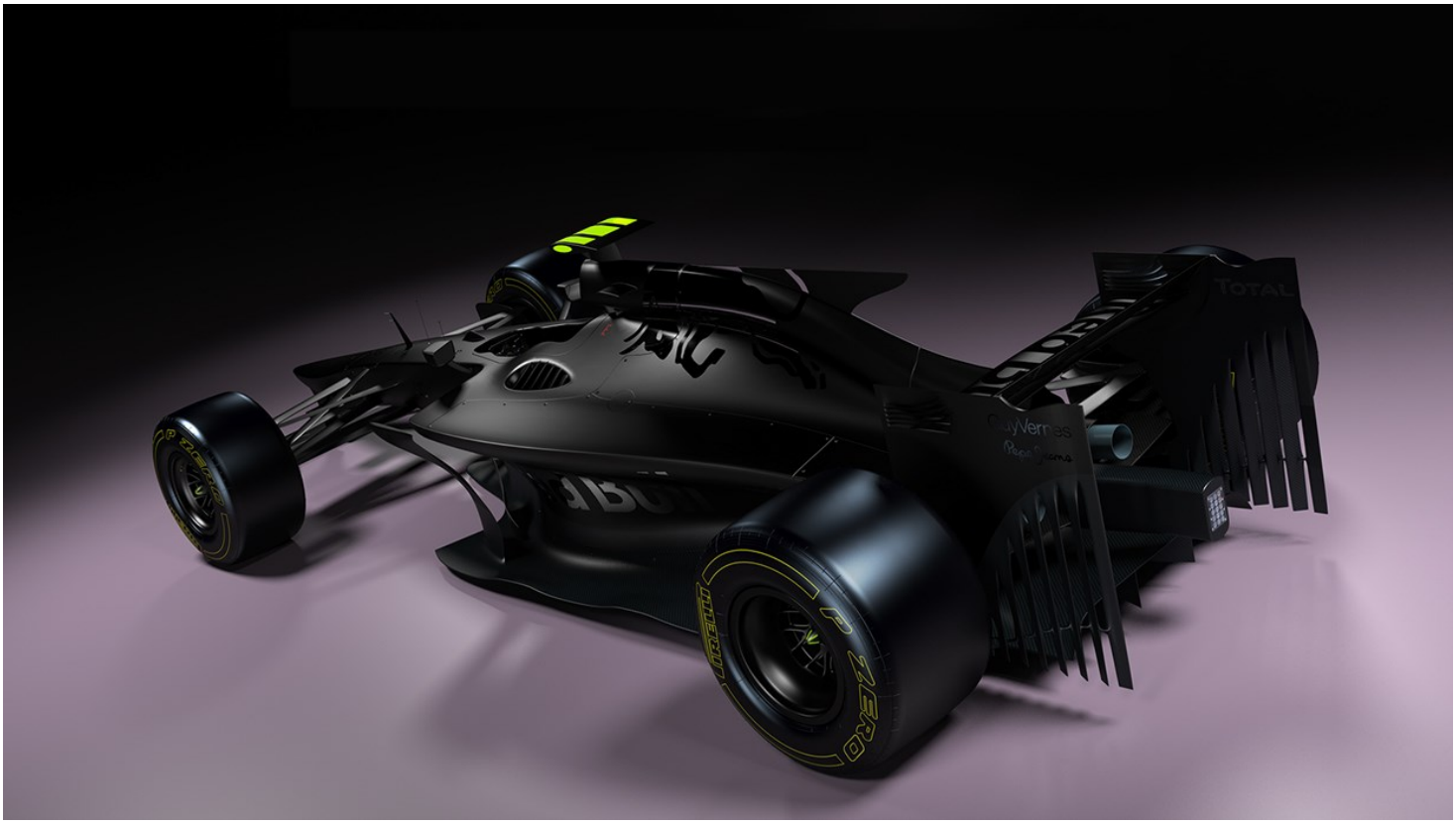


Table Of Contents

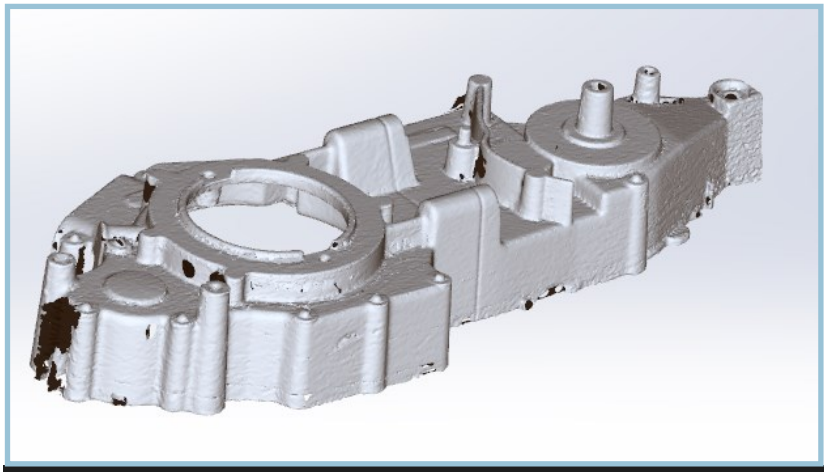
Table Of Contents (continued)

What is Retopologizing?

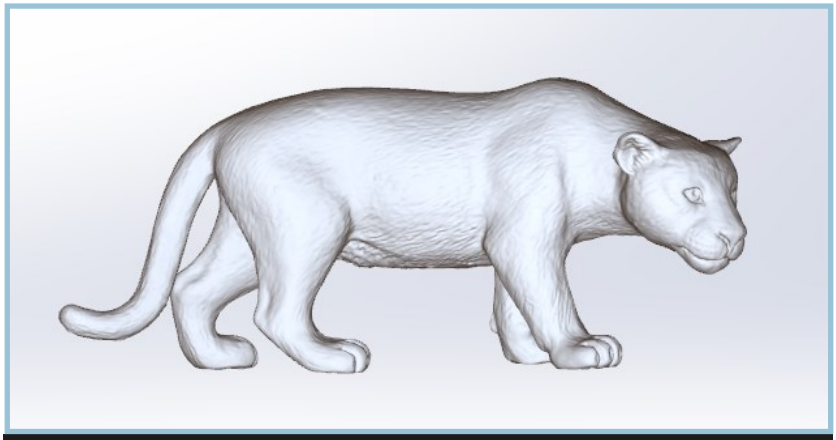
In today's world, few objects are simple enough to be reverse engineered from only a handful of accurate measurements. The complexity and range of manufacturing methods produces objects that rely heavily on organic shapes for both form and function. To reverse engineer these non-mechanical shapes, retopologizing 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.



A) Unprocessed scanned mesh- note holes and rough surface

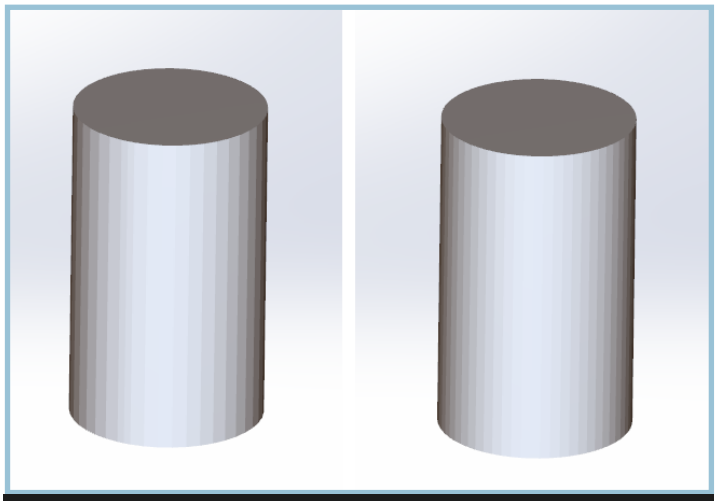


B) Cleaned scanned mesh

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 though 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

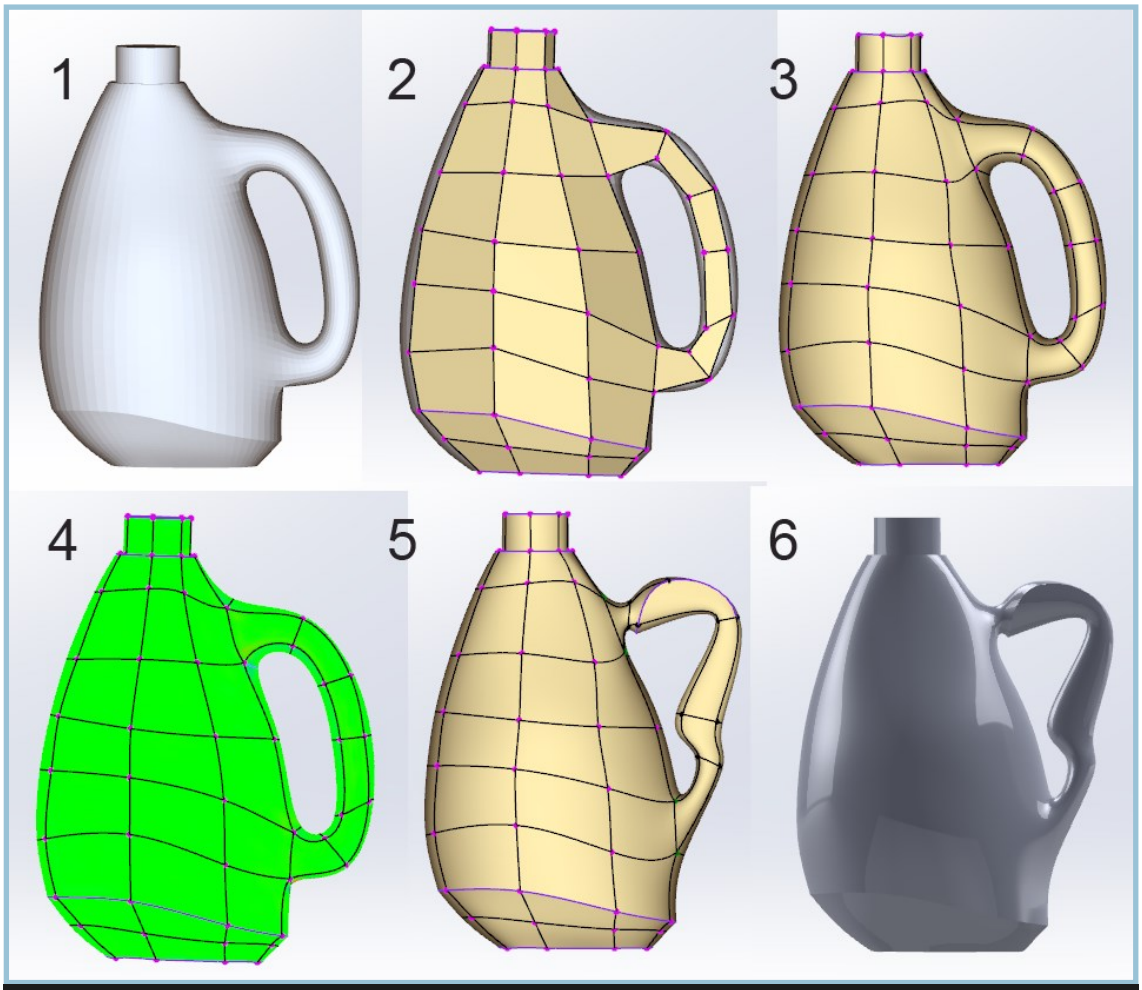
Retopologizing vs. Constraints

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

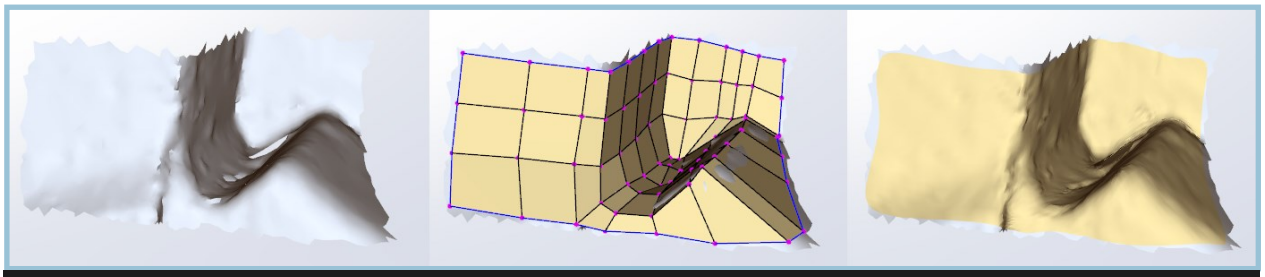
The general retopologizing procedure is fairly straight forward. First obtain or generate a reference mesh, clean it up or prepare it if necessary (Image 1) then proceed with the following: Create the new SubD on top of the reference mesh (Image 2). Interpolate to maintain the reference mesh details (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).



General Workflow, basic steps

Obtaining a Reference Mesh

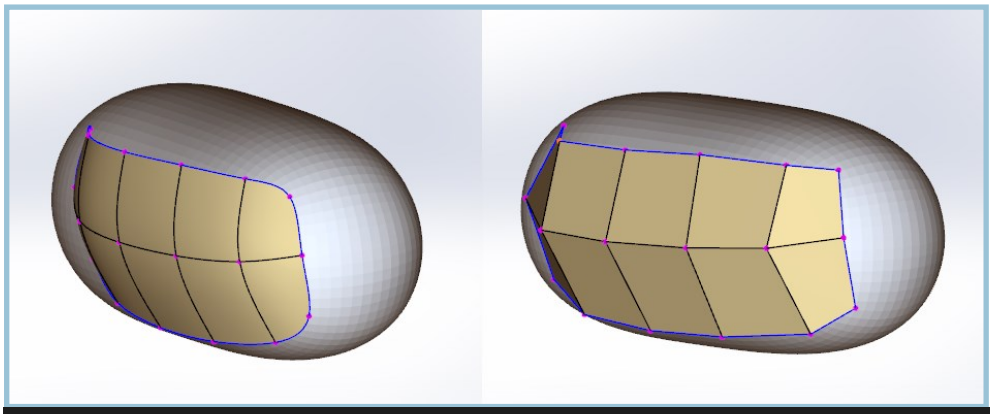
The reference mesh is a polygonal representation of the object's surface. The smaller the faces, the more accurate the reference mesh tends to be. You can start with a scanned mesh in either obj or stl format. Point cloud data is not currently supported. You can also generate a reference mesh from an 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 Interpolation subdivisions- the poor reference mesh surface is reproduced

Retopo Basics

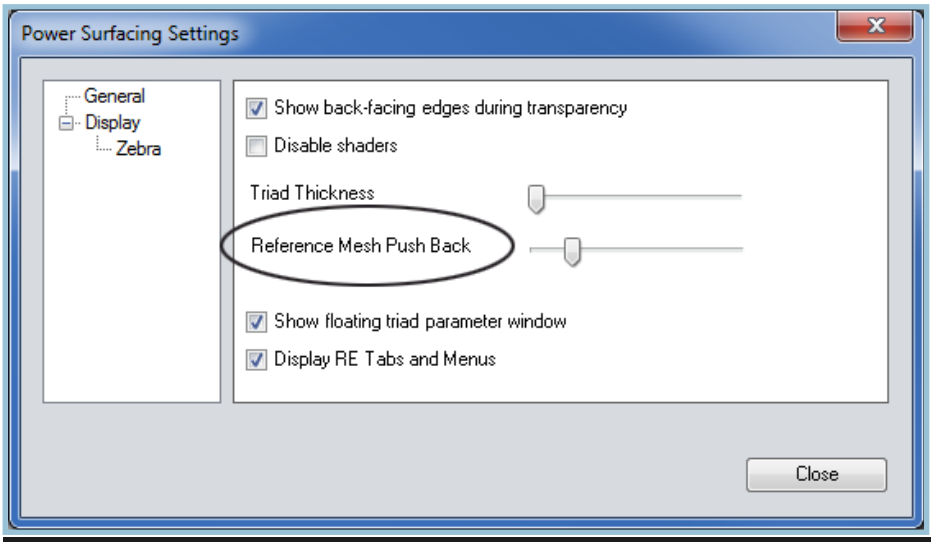
With a suitable reference mesh acquired and prepared, you will create a new SubD surface on top of the reference mesh using the various Power Surfacing RE tools. 'Retopoing' is normally done in Control Mesh display mode so that you can see the true location of the constrained vertices on the reference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

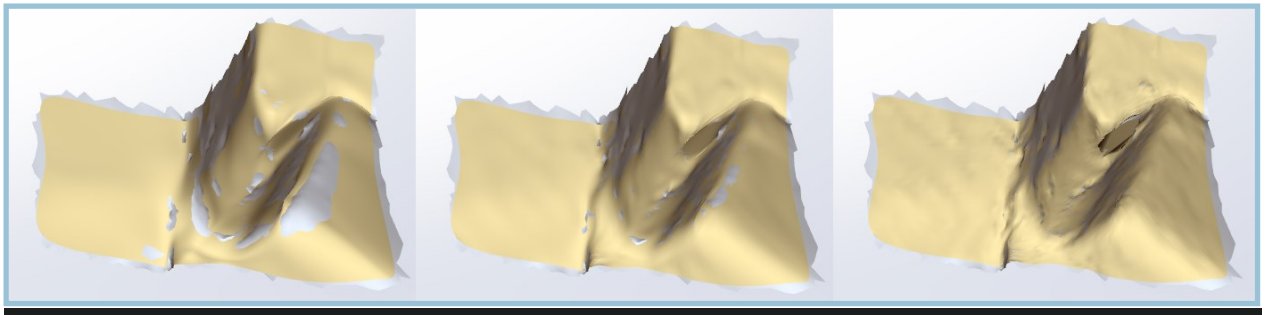
Adjusting the Reference Mesh Display

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

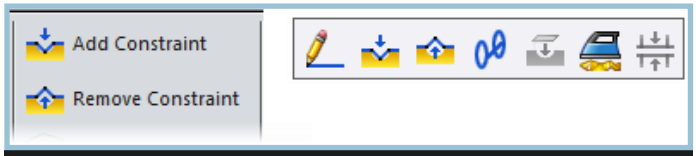
Setting the slider to the far left will show you where the actual SubD mesh faces are in relation to the reference mesh. At the far right, the SubD mesh will be drawn last or on top of the reference mesh. The effect will vary according to the viewing angle as well. Feel free to adjust the amount slider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

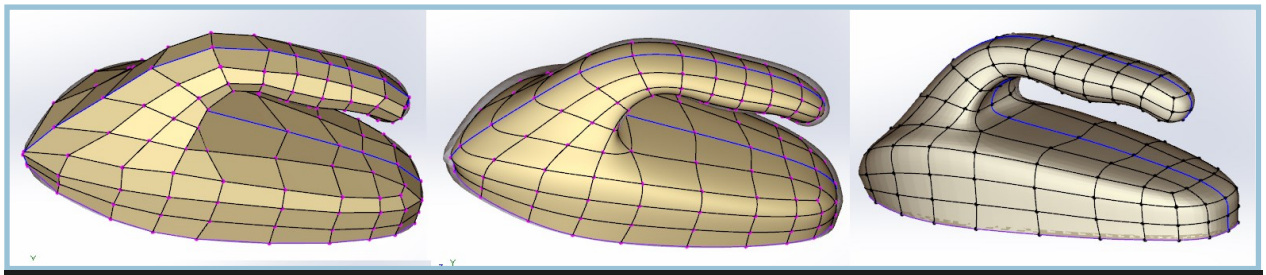


Add and Remove Constraints in the tool bar, left, and the lower section of the right-click menu, right

Using Interpolation

When you've finished the retopologizing process, you can use *Add Interpolation* 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 *Add Interpolation* step is an important one.

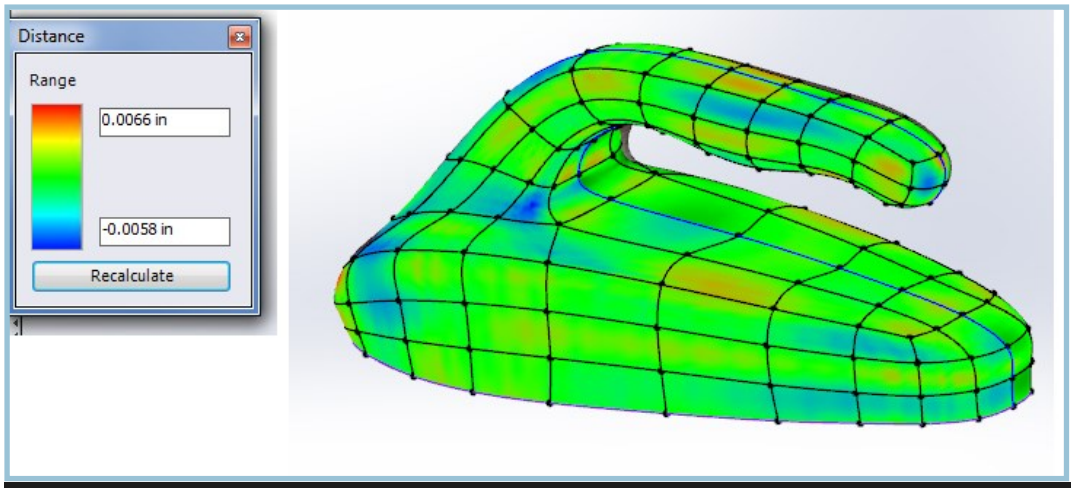
The *Add Interpolation* command brings the subdivided mesh back out to the reference mesh's surface. A single level of internal subdivision may be sufficient depending on the density of the retopo mesh and the amount of detail you want to retain from the reference mesh. A second level of subdivision 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. *Add Interpolation* works only on constrained vertices.



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

Checking Accuracy

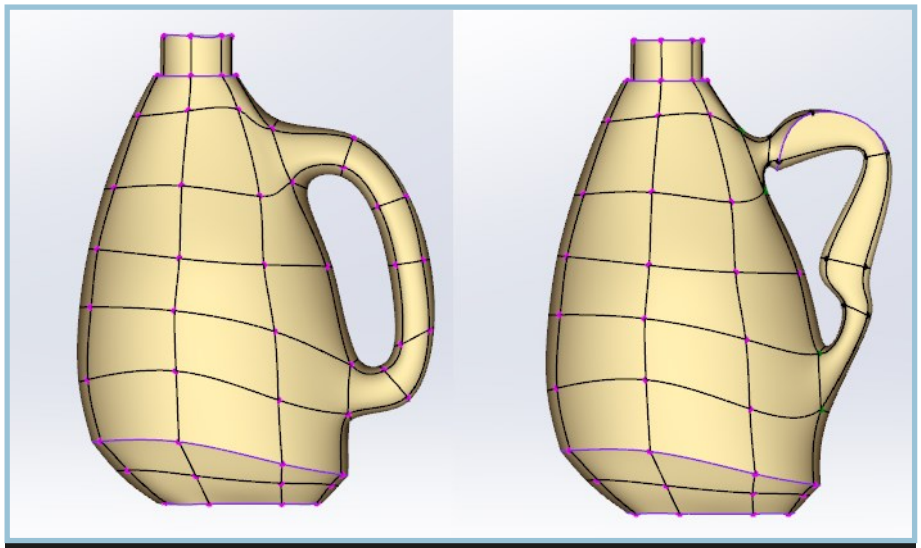
At any time, but especially during the *Add Interpolation* step, you may check the accuracy of the interpolation using the Distance display tool. This tool gives you visual feedback showing where the retopo mesh deviates from the reference mesh and by how much. The range is automatically calculated using 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 retoplogized and then converted to CAD

Preparing the Reference Mesh

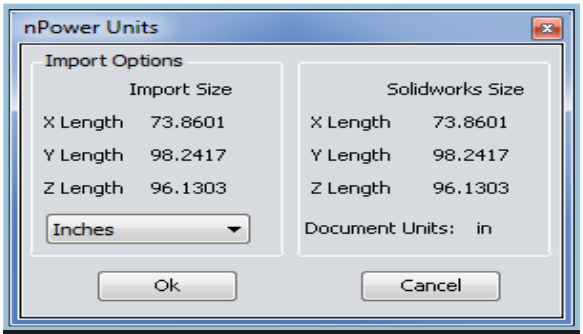
Depending on the reference mesh source, its condition and your goal for the reptopologized version of it, you can streamline the process by preparing the reference mesh beforehand. The goal is to remove unwanted faces, orient the mesh for easier handling, repair faulty or changed geometry and finally, 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.



The Import dialog for setting units

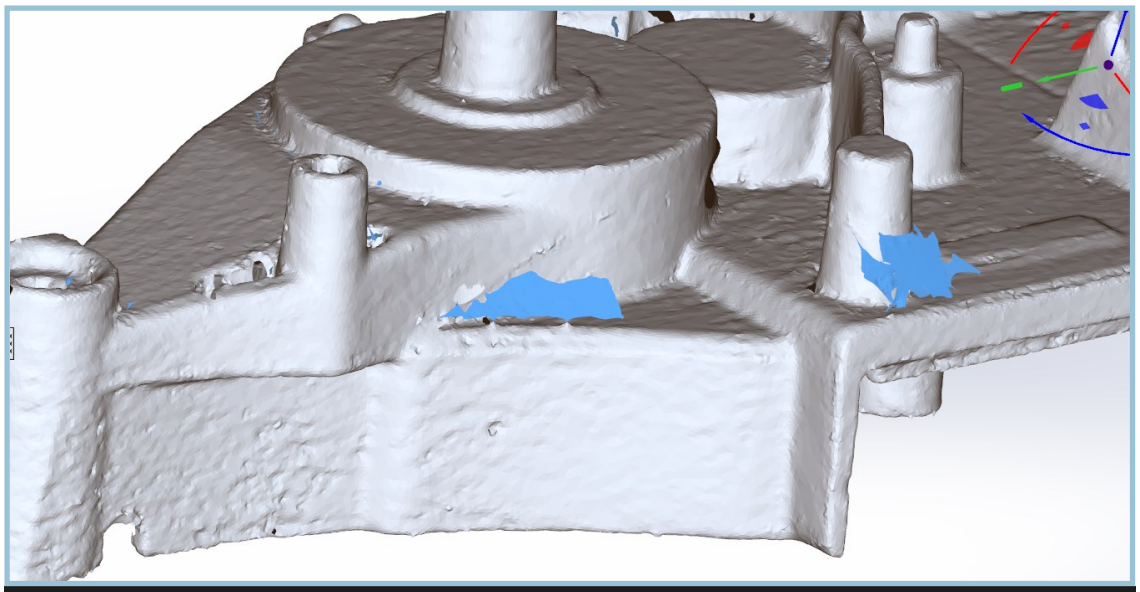
Note: Neither file type has a concept of units so you must specify them on import. If you are unsure, you can check the value as it would be shown in the part file's default units.

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.

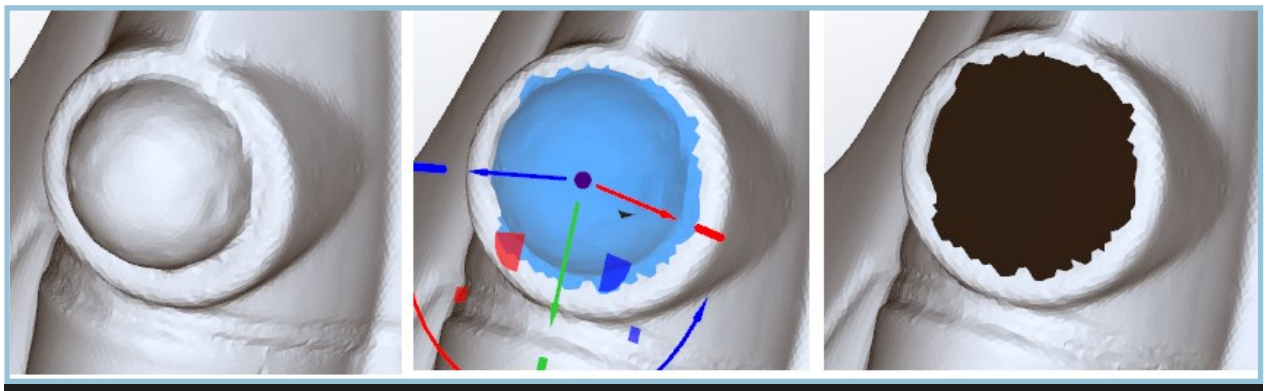
Removing unwanted Geometry

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 places areas.

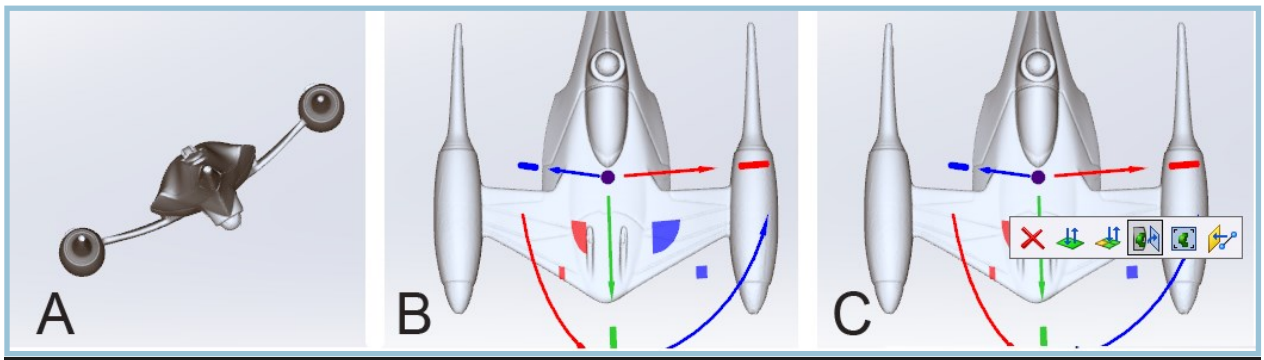
3. Select *Delete* from the Power Surfacing tab or the right-click menu.



Removing unwanted features or features better suited to CAD modeling

Re-orienting the Scanned Mesh

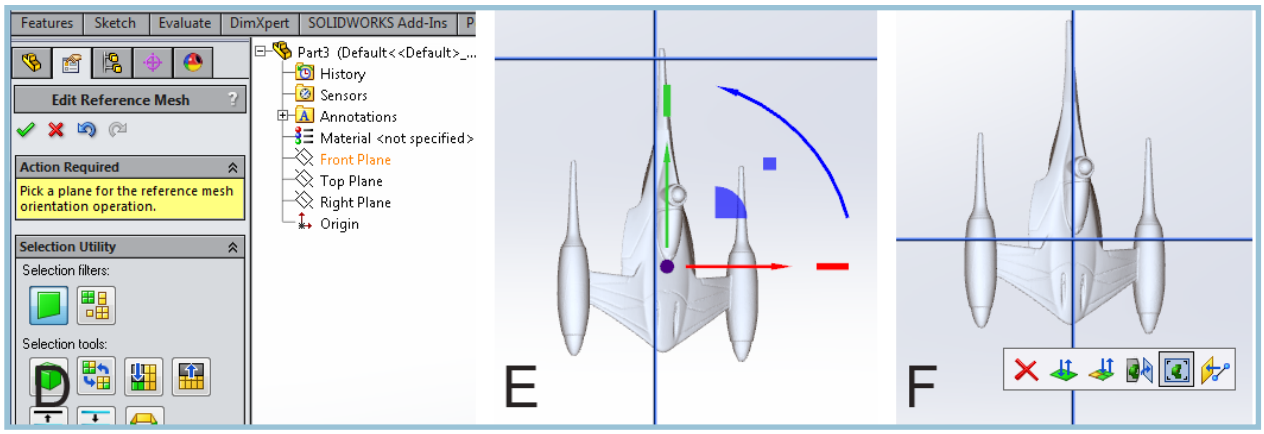
A typical problem with scanned meshes is that they rarely come in nicely aligned in orthographic projection. Good alignment can make modeling exponentially easier, so the next step is to rotate and center the reference mesh before going any further. While many models will not have a logical face to use for setting alignment, even an approximation will help once you have begun the retopo process.



A scanned mesh with poor alignment, Image A. Selecting a face for alignment, Image B. Selecting Orient to Plane from the right-click menu, Image C.

1. Make sure you are in *Edit Reference Mesh* mode.
2. Locate the most logical surface to use to align to one of the default construction planes.
3. Select a single face, or an area of faces that should approximate the plane's orientation, Image B.
4. From the right-click menu, choose *Orient to Plane*, Image C.

Tip: Use the *Back Face Cull Selection* tool to help with multiple selections, or deselect extra faces that don't belong with your 'surface' selection.



Selecting a plane from the viewport list tree, Image D. The model oriented to the Front Plane, Image E. The model aligned to the selected plane using the selection as the reference point, Image F.

Re-orienting the Scanned Mesh (continued)

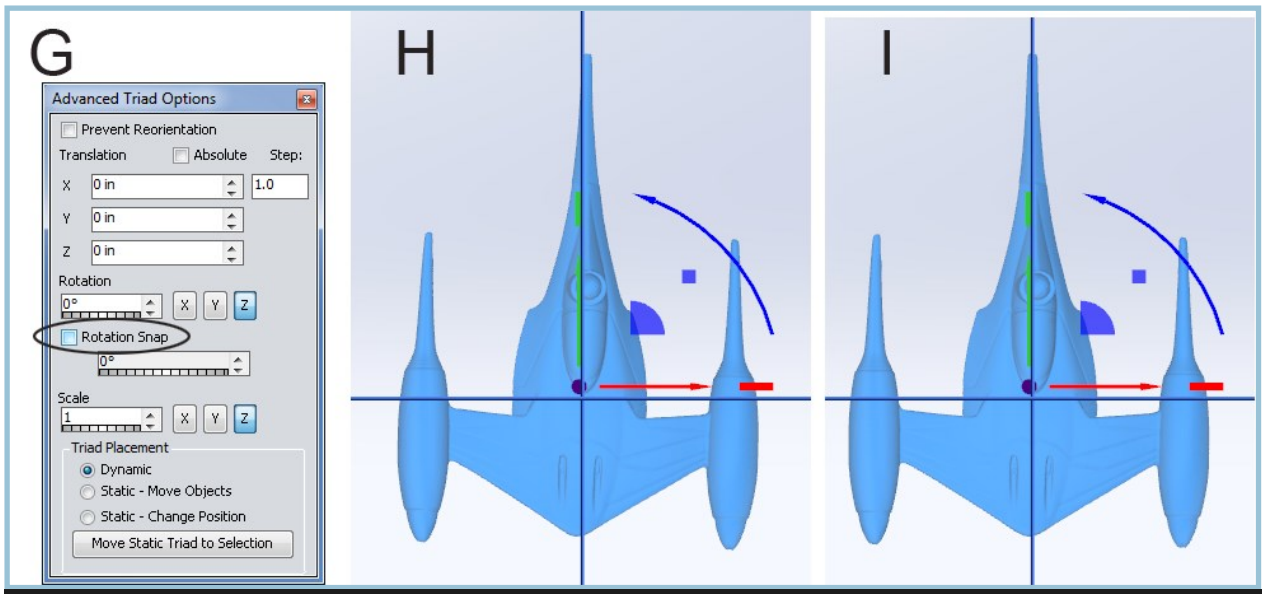
5. From the top left of the viewport, open the SOLIDWORKS feature tree and select the construction plane of your choice (Top, Front, Right Side, etc.), Image D.

6. Right-click again and select Center Selection, Image F.

7. Select the same plane you chose earlier.

8. Set the view to that plane as well.

9. Turn on (show) the plane that should now bisect the object either vertically or horizontally.



Turning off Rotation Snap, Image G. The original Z orientation, Image H. The rotated mesh, Image I.

10. From the Power Surfacing tab, locate the drop down containing Advanced Triad Options and select it.

11. Uncheck Rotation Snap, Image G.

12. Select the entire reference object, Image H.

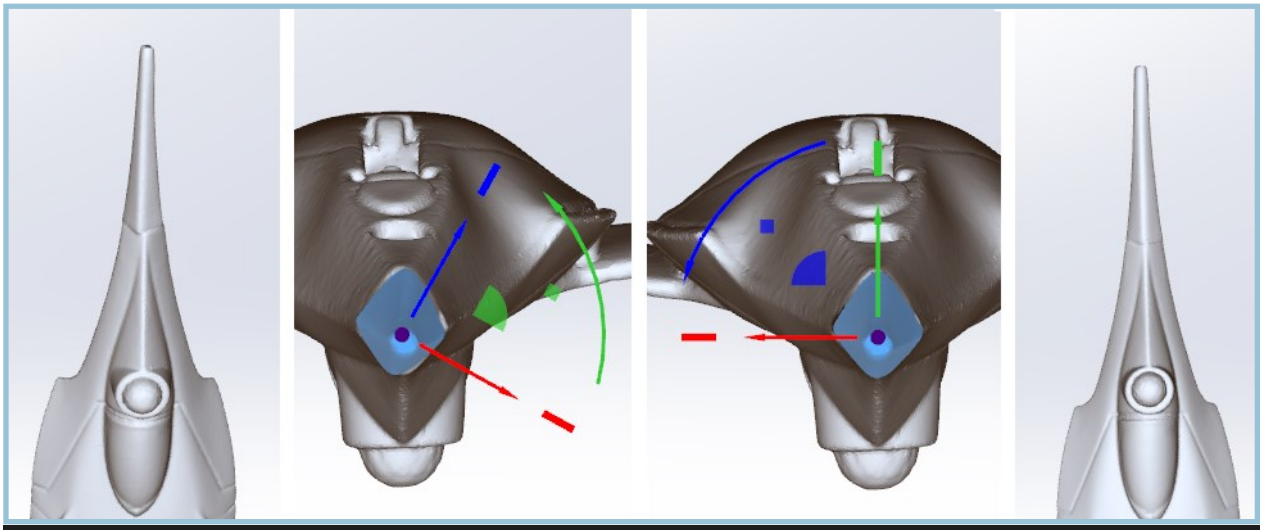
13. Rotate it until you are happy with its alignment to the view, Image I.

14. From the Power Surfacing menu, Save, select Save Control Mesh and append its name with _Aligned.

15. Hide the construction planes.

Re-orienting the Scanned Mesh (continued)

Selected parts of the mesh can be oriented individually with the same technique to improve scan or original model problems.



The nose of the aircraft misaligned, leftmost image. The nose faces selected and viewed from the front of the model, center-left. The faces rotated using viewport coordinates, center-right. The corrected model, rightmost image.

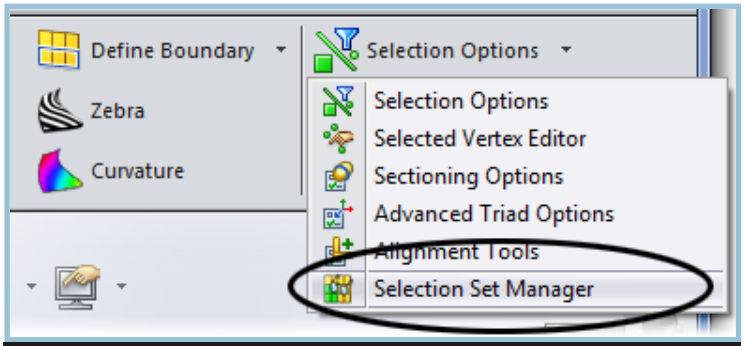
Selection Sets

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 be 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:

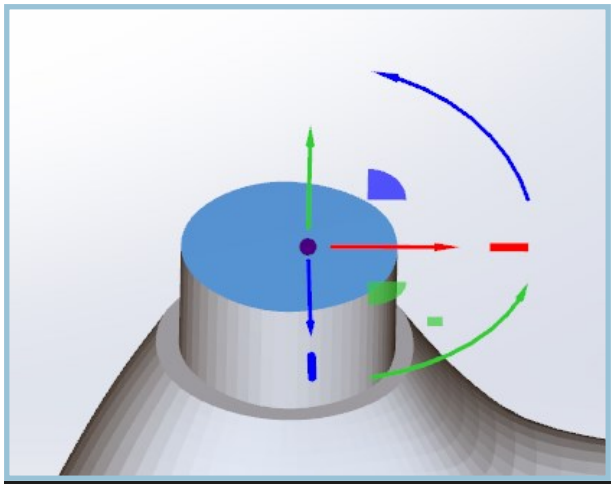
1. From the Power Surfacing RE tool bar, select *Edit Reference Mesh* if you are not already in that mode.
2. From the Power Surfacing tab, click the Advanced tools drop-down and choose *Selection Set Manager.*



Opening the Selection Set Manager

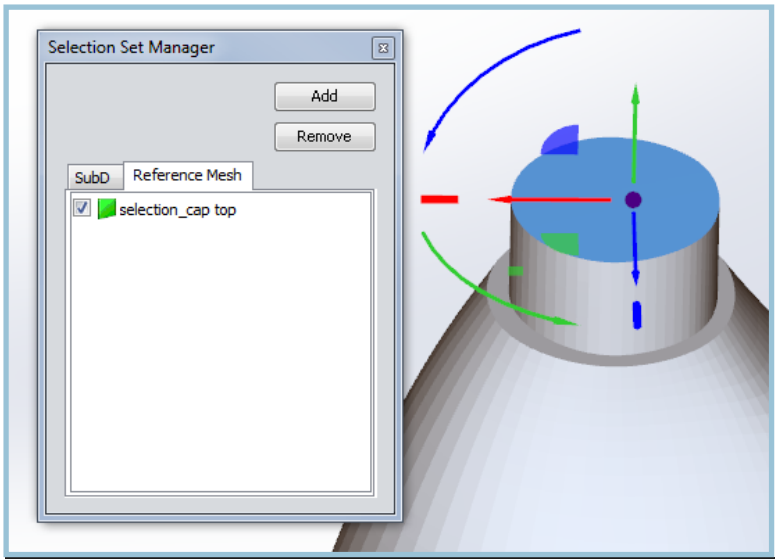
Selection Sets (continued)

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**.

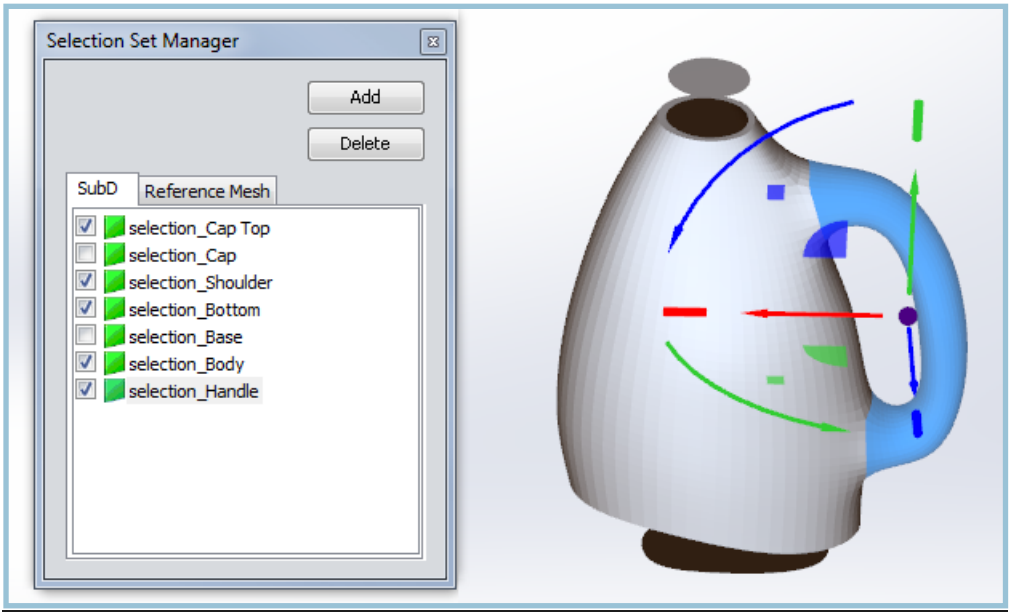


Adding and renaming the new selection set

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.

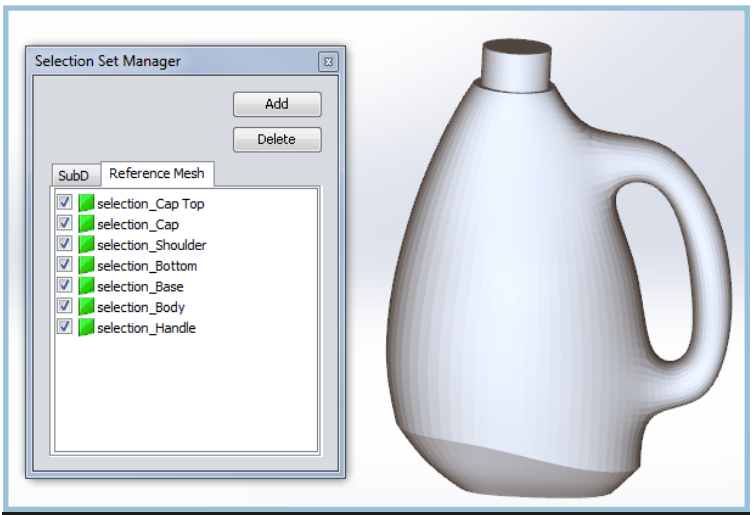


Creating the rest of the selection sets and hiding a few of them

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 sets following

Creating Proxy surfaces

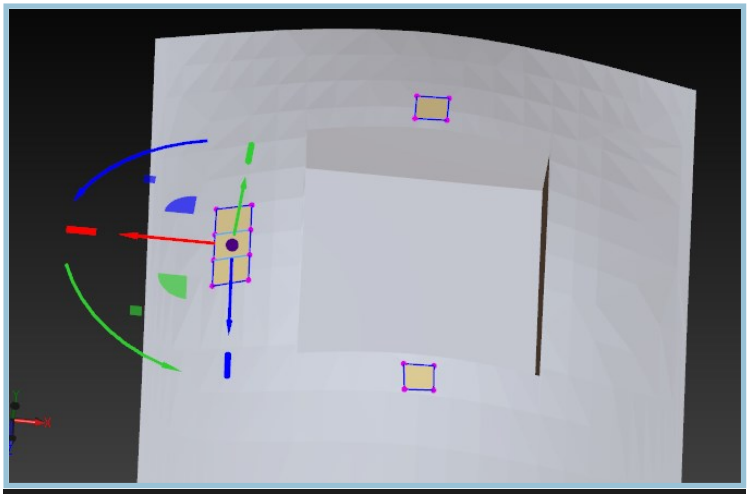
There are several reasons to improve the surface of your reference before going through the retopo process. Some features are better added with the conventional SOLIDWORKS workflow. Some existing features may be unwanted in the new version of the object. In the case of scanned meshes, there may be areas 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

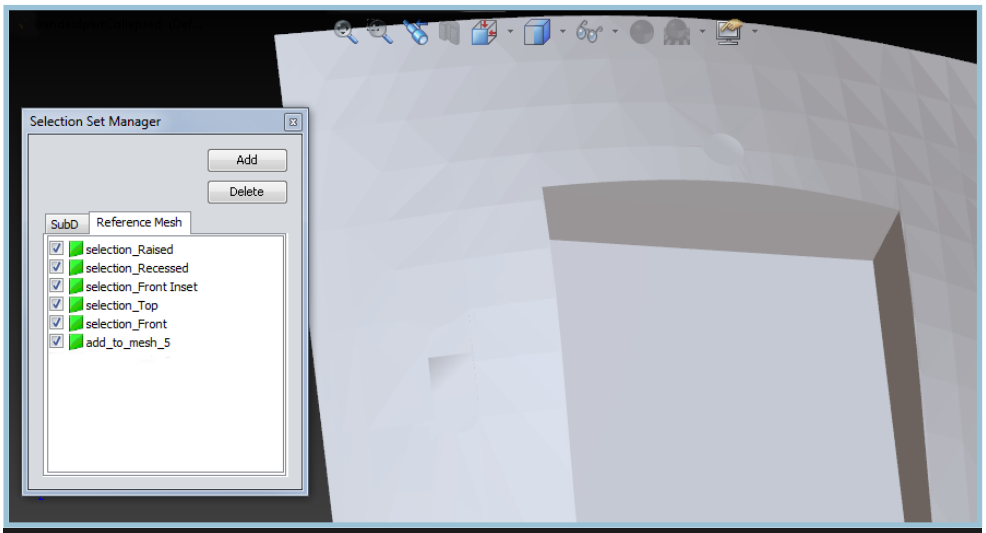
Next you will add the 'bandages' to the reference mesh.

3. In Face sub-object mode, select all of the new faces.

4. n the Power Surfacing Retopo tab, pick the *Add to Mesh* button, .

Creating Proxy surfaces (continued)

The selected quads turn white and are now part of the reference mesh. They are also added as a new selection set.

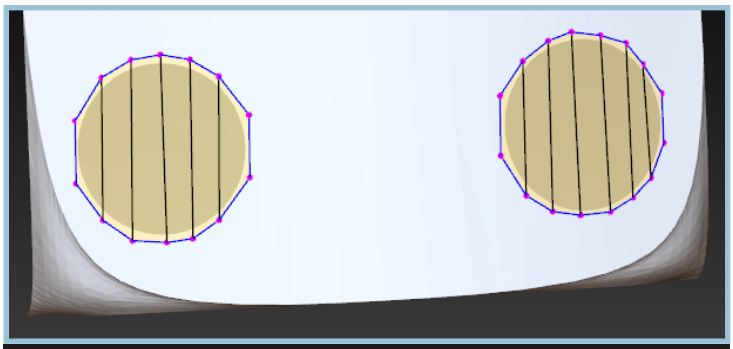


Adding to the reference mesh

The holes left over from hiding the raised section will require a bit more work. Because the surface around the holes is not planar, you will have to create more quads to account for the curvature. In this case, the curvature is lateral across the surface so the task remains reasonably simple.

5. Use *Draw Edges* to cover the holes left by the hidden raised areas.

The faces are arranged to account for the curved surface. *Transparency* has been turned on to be able to see the reference mesh beneath the new quads.

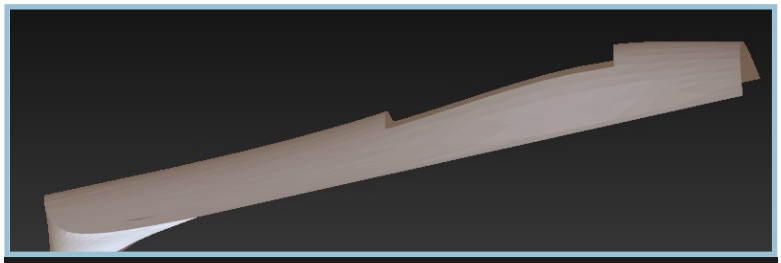


Creating the faces to cover a surface curved in one direction only

6. Once again, select the new faces and use *Add to Mesh* to add them to the reference mesh.

Creating Proxy surfaces (continued)

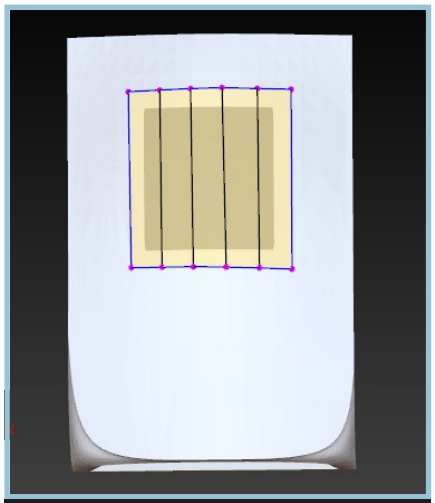
The final area to be prepped is the void left from removing the recessed area. Not only is it a fairly large opening, it also contains compound curvature.



Side view of the reference mesh showing the compound curvature

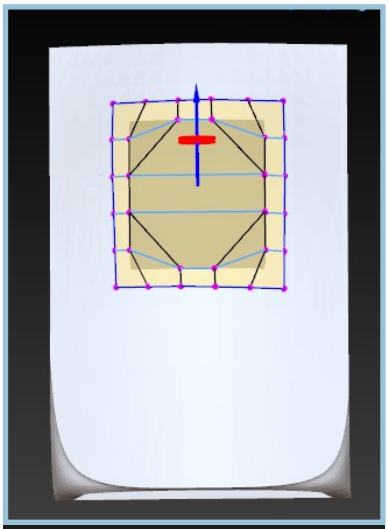
As with the holes left by the raised areas, you will want to choose one direction to layout the first quads. In this example, aligning the first quads vertically gives a better result than horizontally. You can add a subdivision or two after the patch is blocked in so you can keep the quads fairly large at first. The other difference is that you should allow a larger boarder around the hole so you will be able to incorporate the curvature later in the process.

1. Roughly block in the quads to cover the hole.



The blocked-in surface

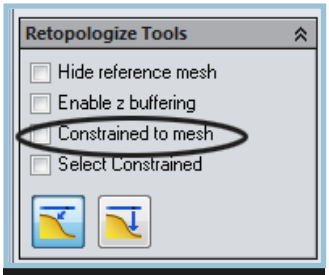
Before inserting the horizontal edges, you will have to turn off *Constraint to mesh* mode to prevent newly created vertices from snapping back to the edges of the hole as in the following image.



Vertices on newly inserted edges snapping to the reference mesh in Constrained to Mesh mode

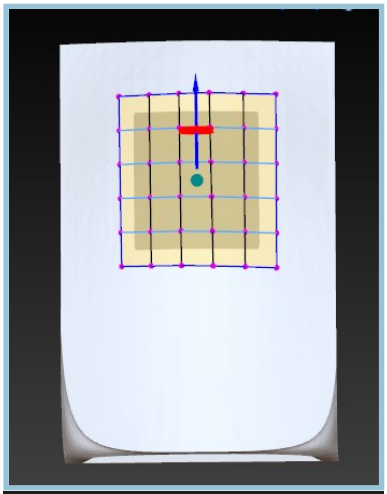
Creating Proxy surfaces (continued)

2. From the Retopologize Tools rollout in the Command panel, uncheck *Constrained to Mesh*.



Turning off Constrained to Mesh mode

3. Now select the vertical edges and use *Insert Loops*, , to add the new edge loops.



The new edges without the vertices snapping to the edge of the reference mesh

The “Constrained” attribute on the vertices remains intact, but the new vertices are not yet being actively constrained. Before turning *Constrained to Mesh* mode back on, you will set the vertices that are over the hole to unconstrained.

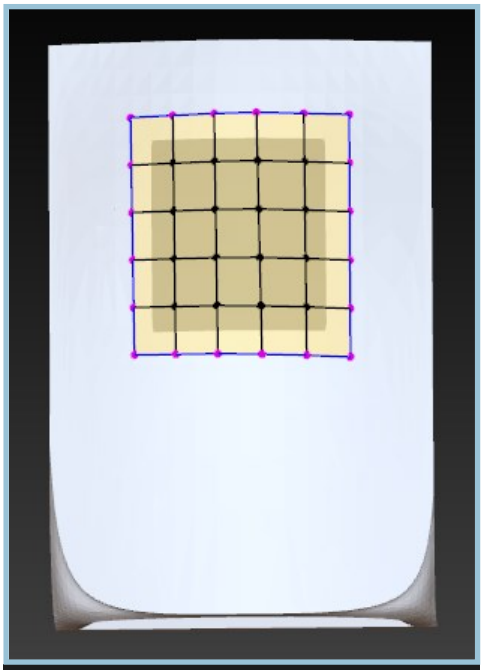
4. Select the vertices that are over the hole.
5. From the right-click menu or the tool bar, select *Remove Constraint*, .

Creating Proxy surfaces (continued)

The unconstrained vertices now show as black instead of magenta colored.

6. Toggle the *Constrained to Mesh* mode back on.

The unconstrained vertices remain in place.



The vertices over the hole set to unconstrained

Next you can start to add more subdivisions so the surface will be smooth enough to work well as a reference mesh. When new geometry is added, the new vertices adopt the 'constrained' attribute depending on the surrounding vertices.

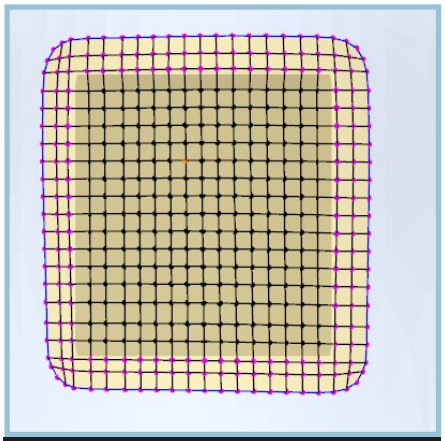
From the right-click menu or the tool bar, use *Sub-Divide All,*  , twice to add more geometry.

The vertices assigned the 'constrained' attribute depending on their neighbors

Creating Proxy surfaces (continued)

In this scenario, several of the rows and columns will have to be moved away from the edge of the hole to avoid being snapped over to the existing surface

8. Move the borderline vertices back away from the edge of the hole, then set all of the vertices that are *not* over the hole to constrained using *Add Constraint*,  .

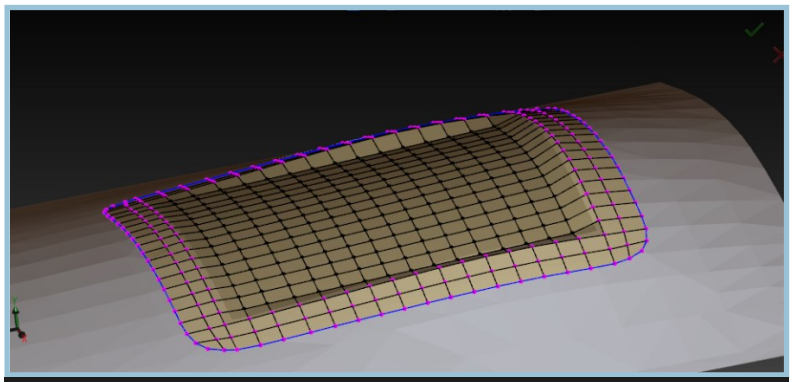


The vertices assigned the 'constrained' attribute depending on their neighbors

9. Turn *Constrained to Mesh* back on in the Control panel.

10. Click the *Snap Vertices to Reference Mesh* command in the tool bar.

Now there's only one major task left to do. If you look at a side view of the mesh, you will notice that the new unconstrained vertices do not yet reflect the shape of the reference mesh's surrounding curvature.

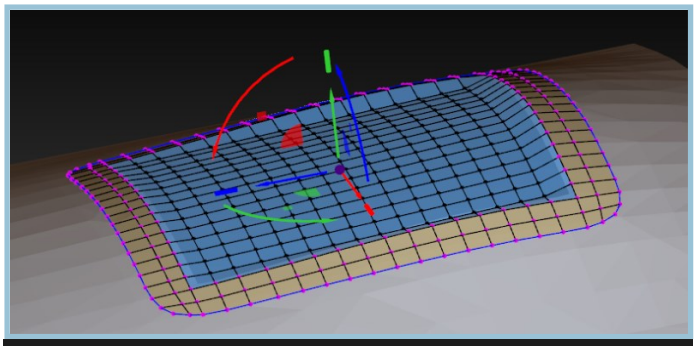


The unconstrained vertices only showing the curvature from the original quads

Creating Proxy surfaces (continued)

To approximate the missing surface, you will use the *Curve Selected* command. It interpolates the curvature from the surrounding geometry and affects the selected faces accordingly. Now is the perfect time to use another of the selection options.

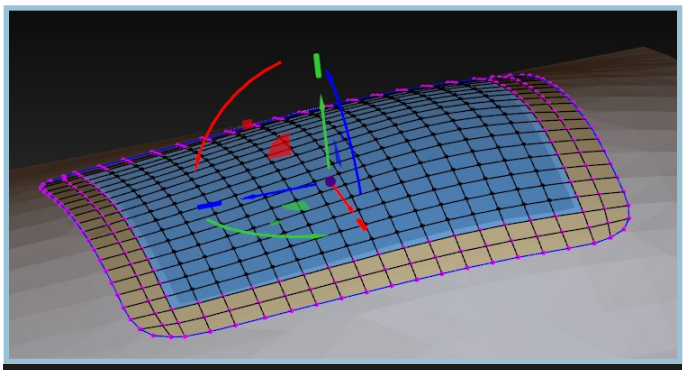
1. In the Command panel, from the Retopologize Tools rollout, check *Select Constrained*.
2. Switch to Face selection.
3. Hover over the unconstrained vertices and select them when they become highlighted.



Selecting the unconstrained faces using the Select Constrained command

4. From the Tools menu, Power Surfacing, Specialized Tools sub-menu, select *Curve Selected*.

The surface curves gracefully to match the surrounding geometry.



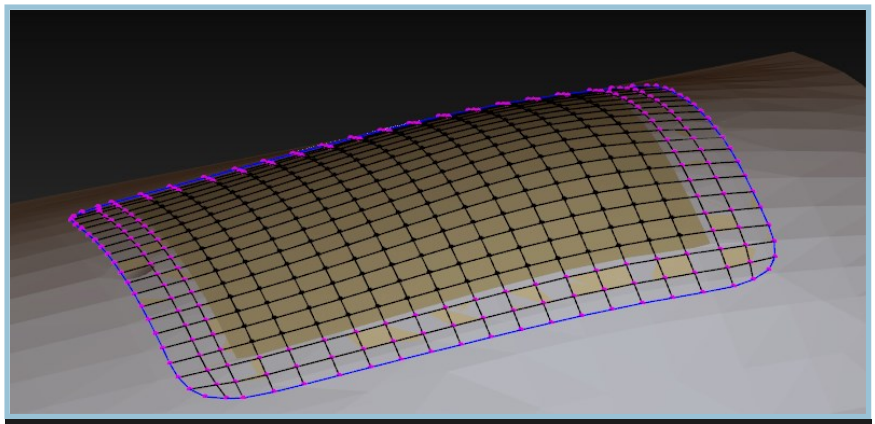
The unconstrained faces interpolating their curvature from the surrounding geometry

Creating Proxy surfaces (continued)

Before adding this surface to the reference mesh, you should check the surface with the *Enable Z buffering* turned on. When this setting is off, it helps ensure that the retopo SubD mesh is drawn after, or on top of, the reference mesh by overriding the Reference Mesh Push Back setting.

5. Turn off *Select Constrained*.
6. From the Retopologizing Tools rollout, check *Enable Z buffering*.

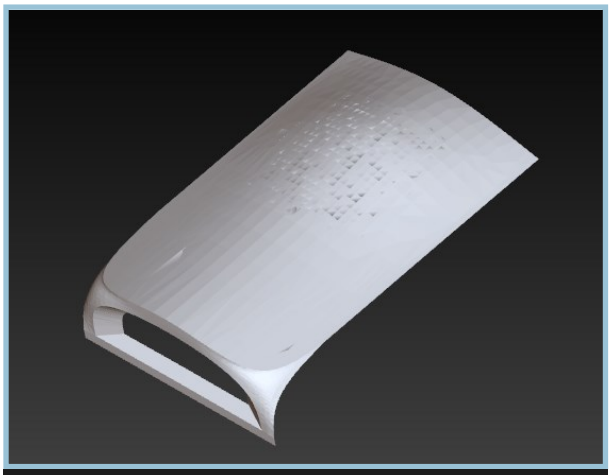
Enable Z buffering shows a good surface match between SubD and reference mesh.



The surface transitioning nicely between reference mesh and hole

7. And finally, select all of the faces and click *Add to Mesh*.

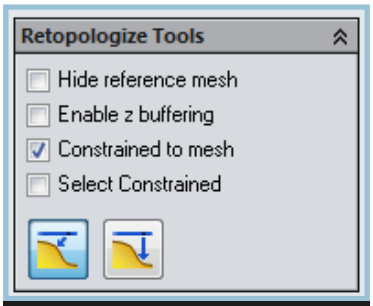
The reference mesh is now cleaned up and ready to be retopologized.



The reference mesh ready for retopologizing

The Retopologize Tools Rollout

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.



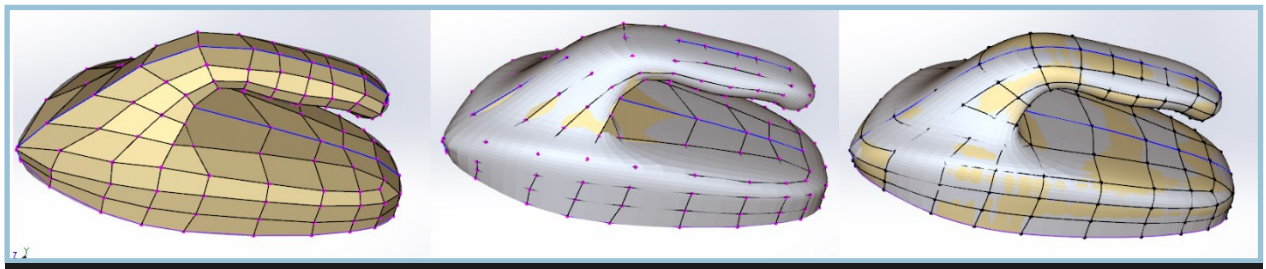
The Reference Mesh Tools rollout in the Command panel

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*.



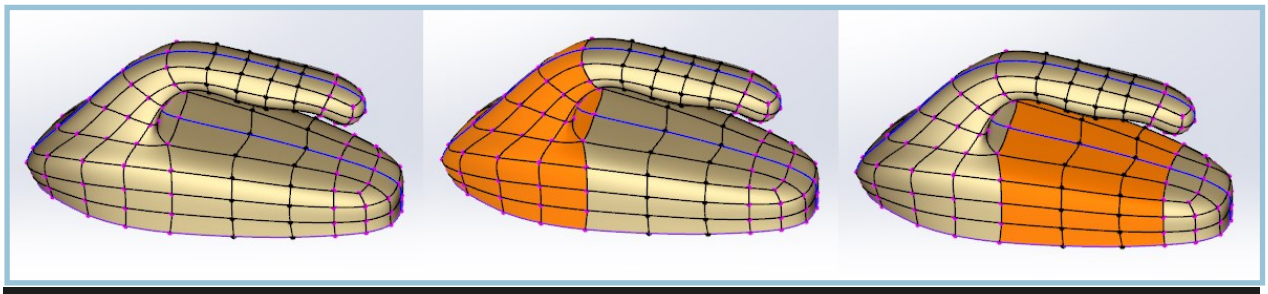
Enable Z Buffer turned off (the default during the retopologizing process) and showing the retopo surface in front of the reference mesh, left image. The center image, shows Z buffering turned on, where only vertices and edges on top of the reference mesh's surface show. Right image shows mesh after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



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

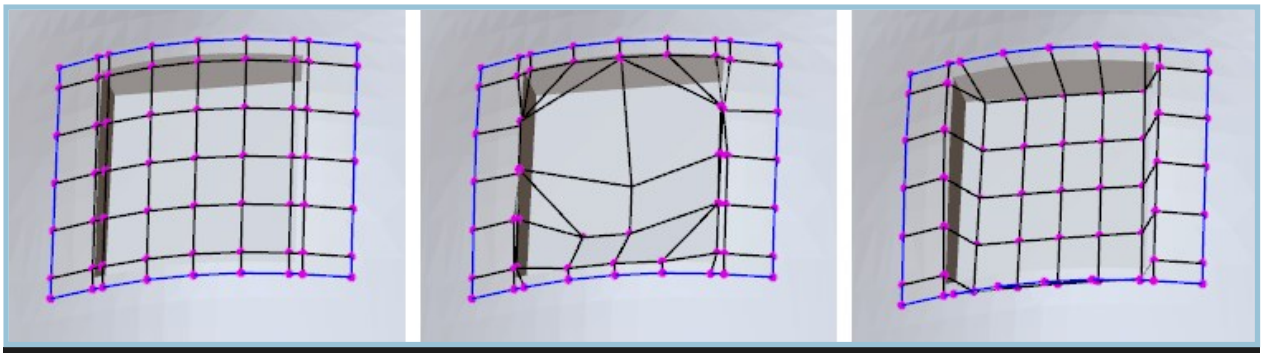
Constraint Snapping Options


Snap Closest Point


Snap Normal

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.



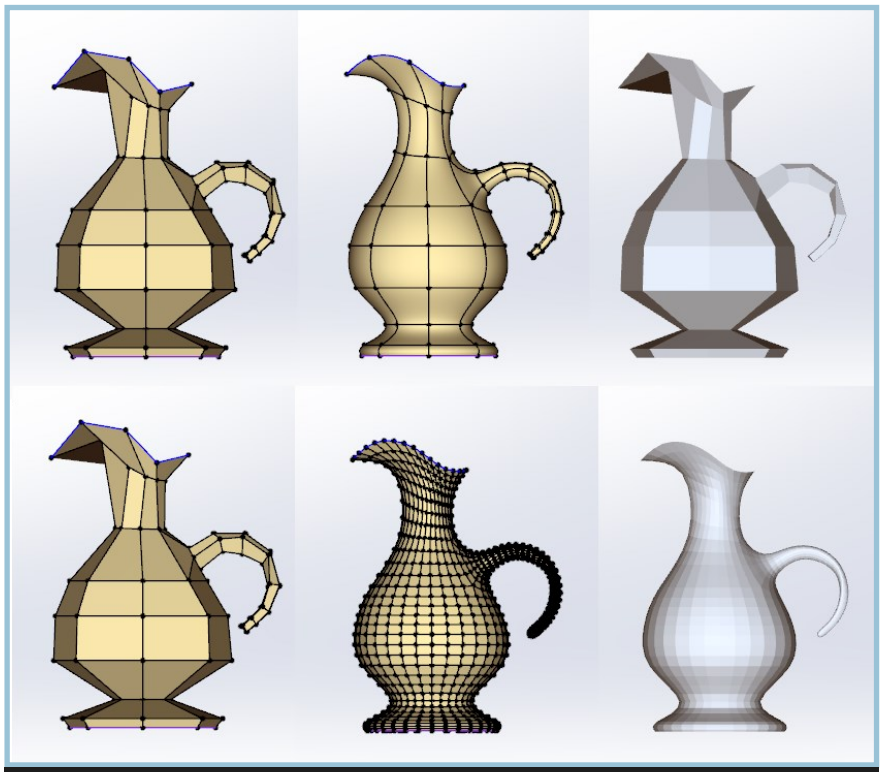
Snapping options: Constrained to Mesh turned off, left, Snap Closest Point, center, Snap Normal, right

The Power Surfacing Retopo 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.

Retopologize Object

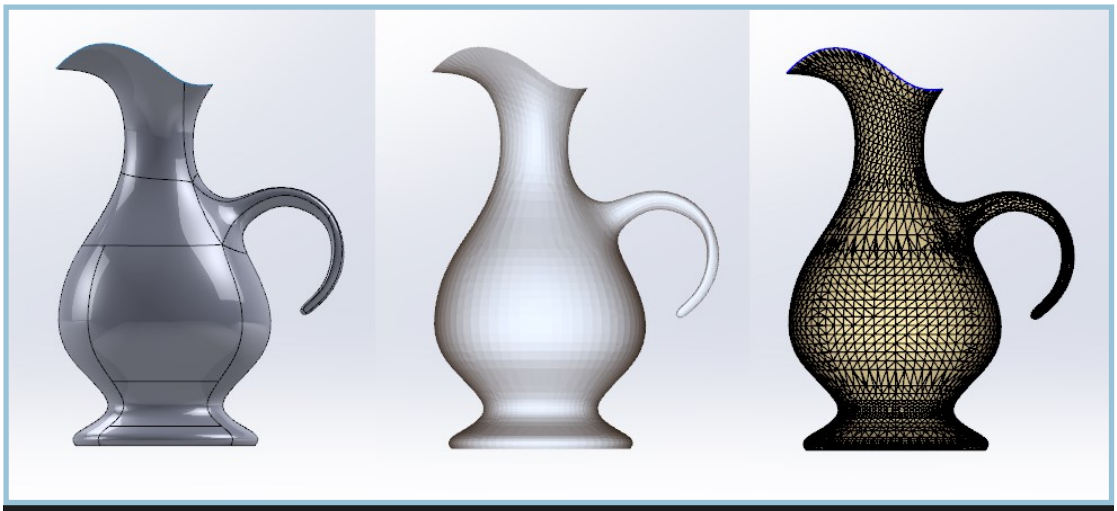
This command turns a polygonal mesh (SubD *or* triangle-based) into a reference mesh. Note that the object's history, if any, will be lost on conversion. Only the Control Mesh is used for conversion so you should use *Subdivide All* beforehand to give your reference mesh enough resolution to work with.



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

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

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.

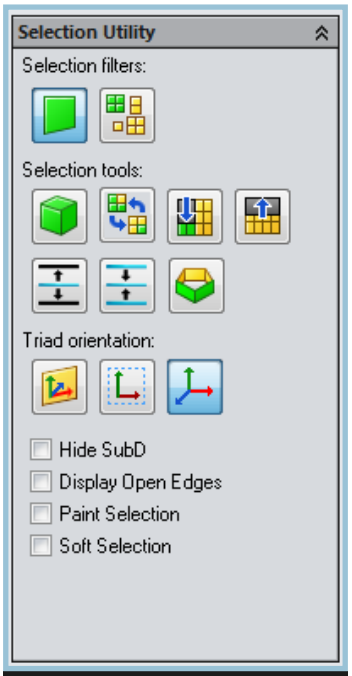
Hide SubD- This option allows you to hide the retopo or regular SubD mesh, allowing you to edit the reference mesh at any time.

Display Open Edges- This option displays the reference mesh's open edges. This can help problem solve with tools that require knit edges to work correctly.

Paint Selection- This tool lets you paint or select faces using a circular range area. It only selects front facing faces.

Soft Selection- This lets you select a face or set of faces and then define the falloff distance around the selection. It only works on face selections.

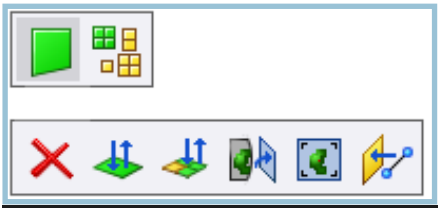
Note: Be aware that when you transform selected faces in Edit Reference Mesh mode, the mesh will not update until you release the left mouse button.



The Edit Reference Mesh control panel tools and options

Edit Reference Mesh (continued)

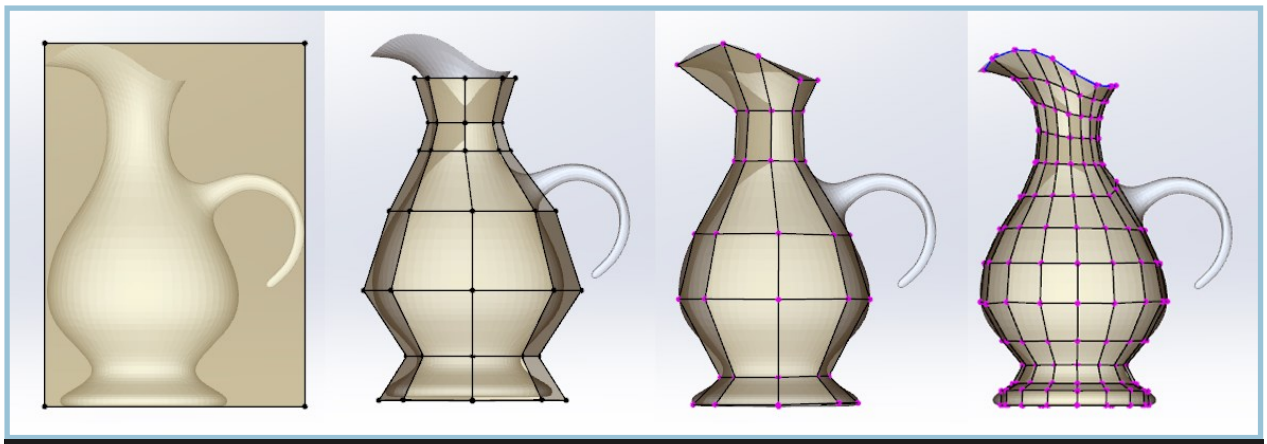
In the right-click menu and tool bar, *Flip Reference Mesh Normals* and *Flip Selected Mesh Normals* are also available. When you are finished editing the reference mesh, you must click on the green check mark to accept the changes and exit the edit mode.



The Edit Reference Mesh right-click menu

Create bounding box

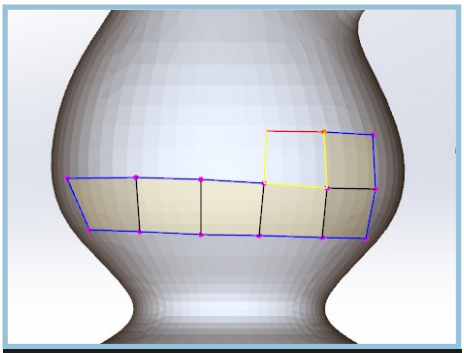
This tool creates a simple bounding box around the reference mesh as the starting point for your SubD. Unlike other retopo tools, the vertices will *not* be set to Constrained. Use this method to block in your retopo mesh where you can block in a simple proxy mesh, constrain the vertices and then subdivide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



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

Draw Edges

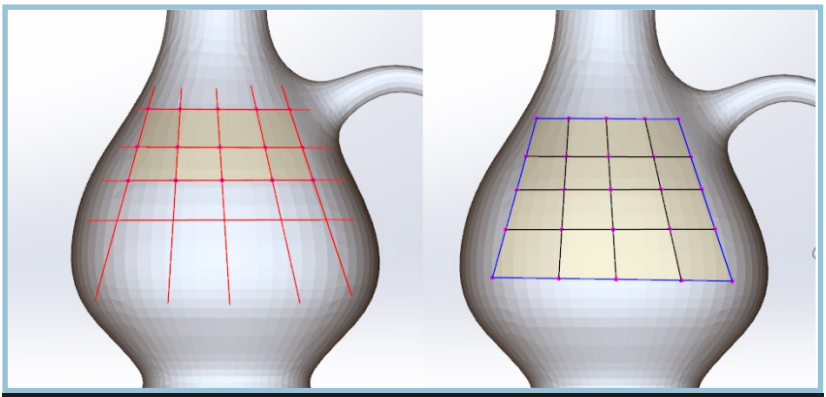
This is the most useful of the retopo tools. It lets you create quads directly on the reference mesh, showing previews and filling in whenever possible. Use <backspace> to undo edges while you are still in Draw Edges mode. Right-click to end Draw Edges mode. Orphaned or red edges will cause any geometry 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 position is clicked

Face Grid

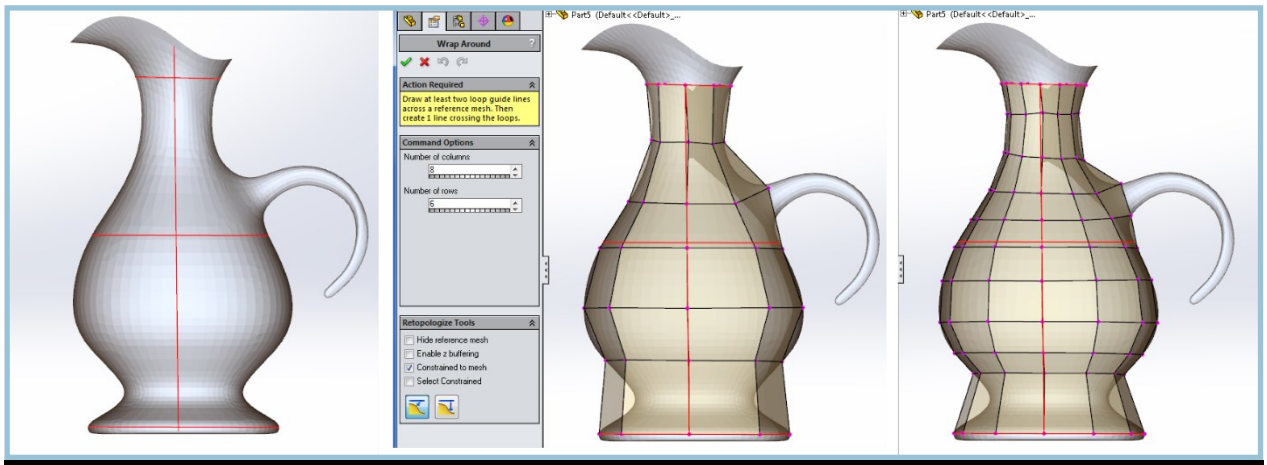
This tool lets you create a quick grid of horizontal and vertical edges. When four edges are closed, a face or quad is created on the reference mesh. You can freehand lines or click to make straight lines.



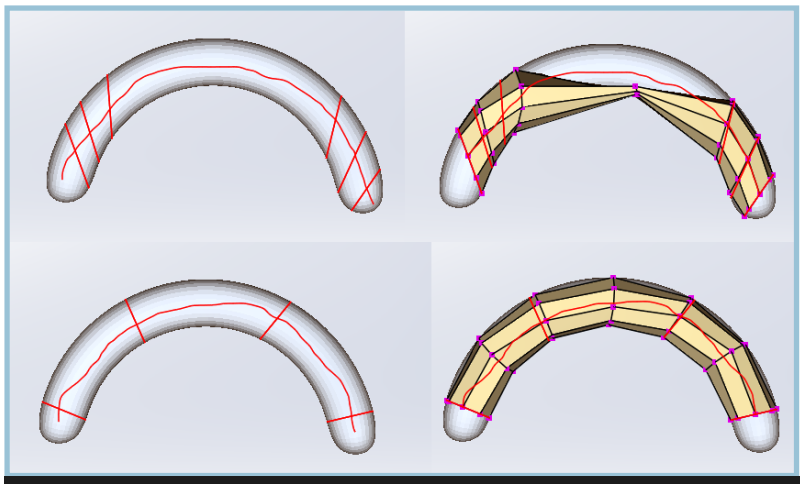
The grid guide lines, left, and the finished faces, right

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.



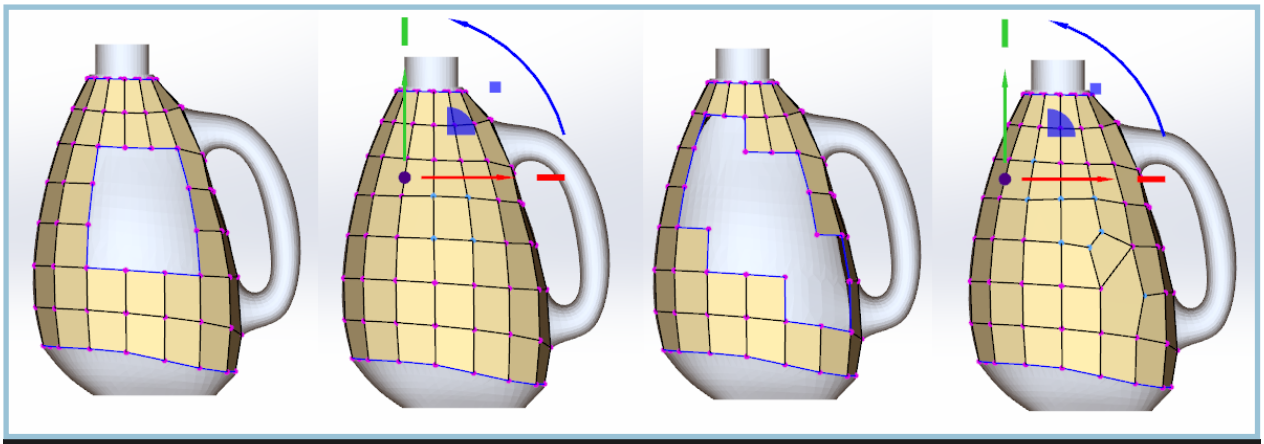
Wrap round: The guide lines, left, changing the rows and columns, center, the new rows and columns



Affecting the wrap around with internal guide lines: poorly placed, top, well placed, bottom

Smart Fill

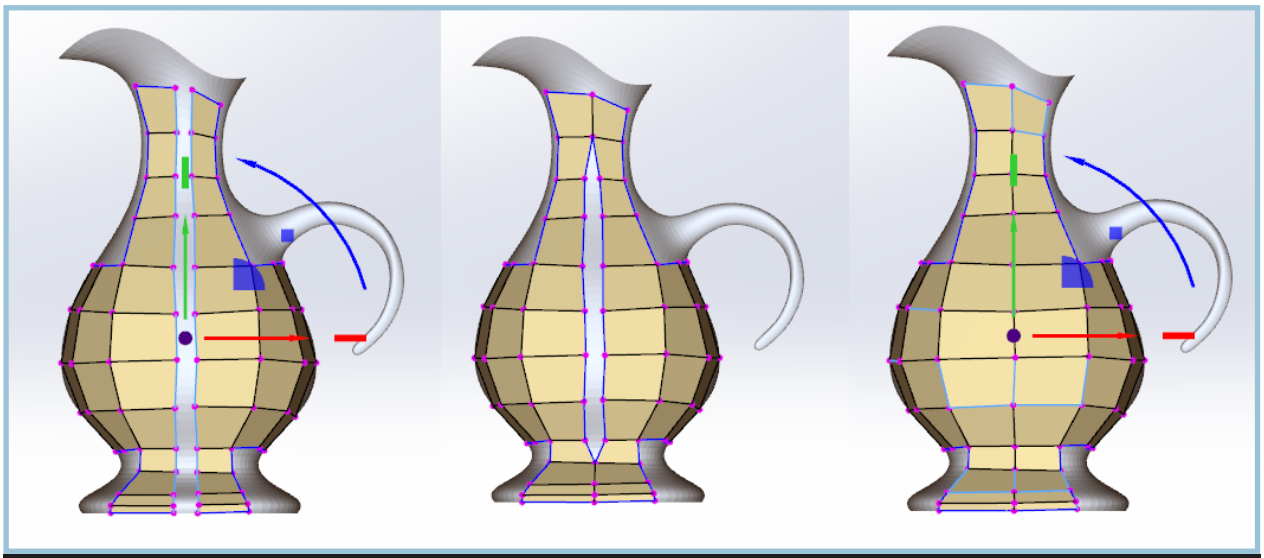
Smart fill will fill and attempt to retopologize a hole in your retopo mesh. The more evenly spaced the surrounding faces, the better the result will be. You can also use Erase and Insert Edges to reconfigure the result if necessary, as it may be quicker to repair the result rather than manually subdivide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

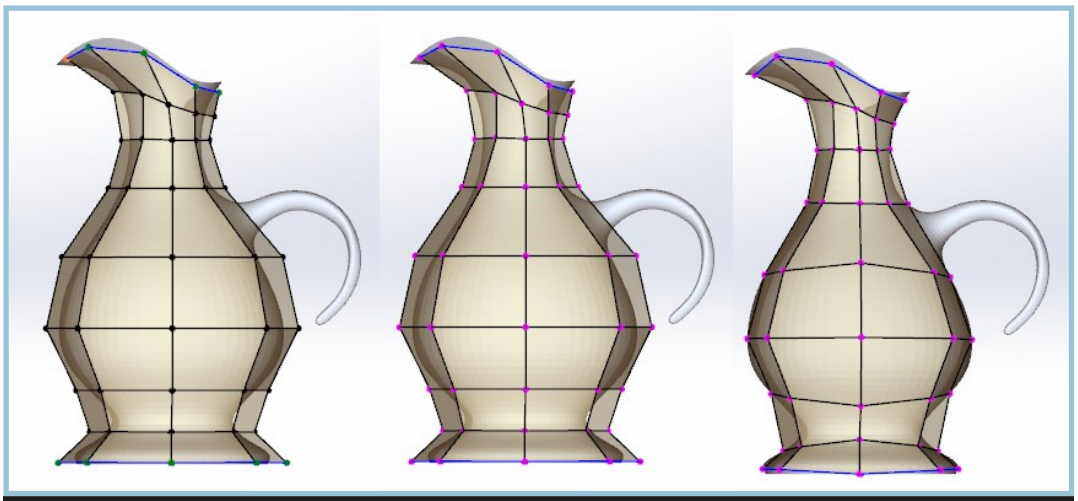
This tool welds lines of edges together as long as both ends are open. You must have the same number of edges selected on both sides.



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

Snap Vertices to Reference Mesh

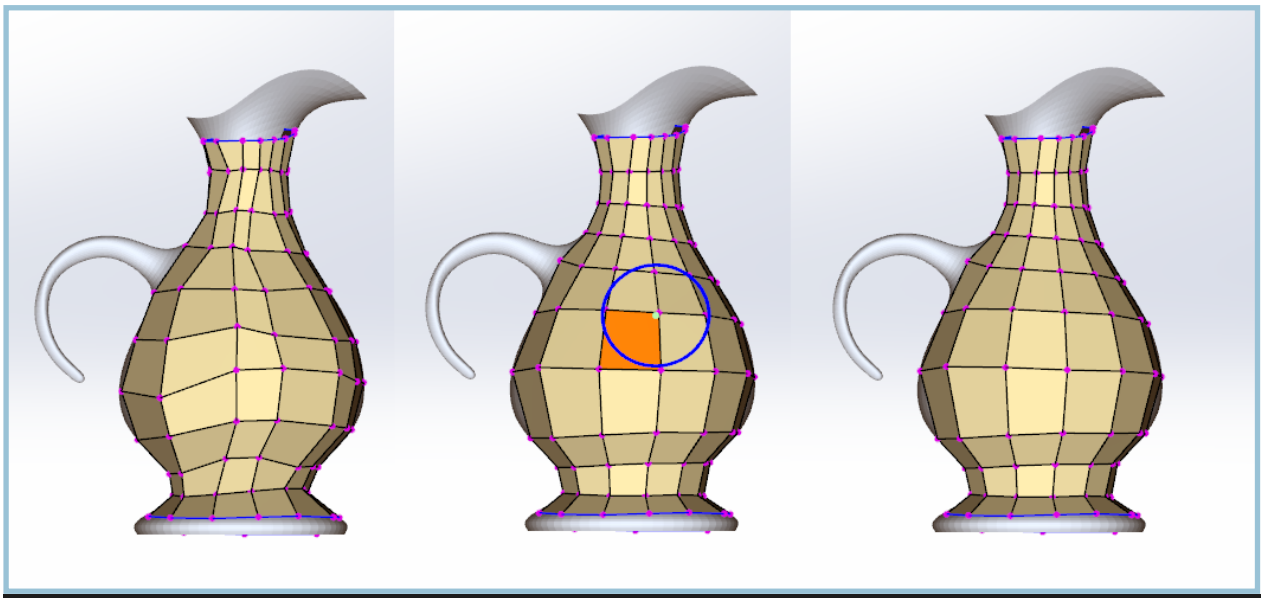
This command snaps all constrained vertices down to the reference mesh. Use this command when you have been working with Constrained to Mesh mode turned off. Turning *Constrained to Mesh* mode on with not automatically snap constrained vertices down to the mesh.



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

Sub-D Smooth

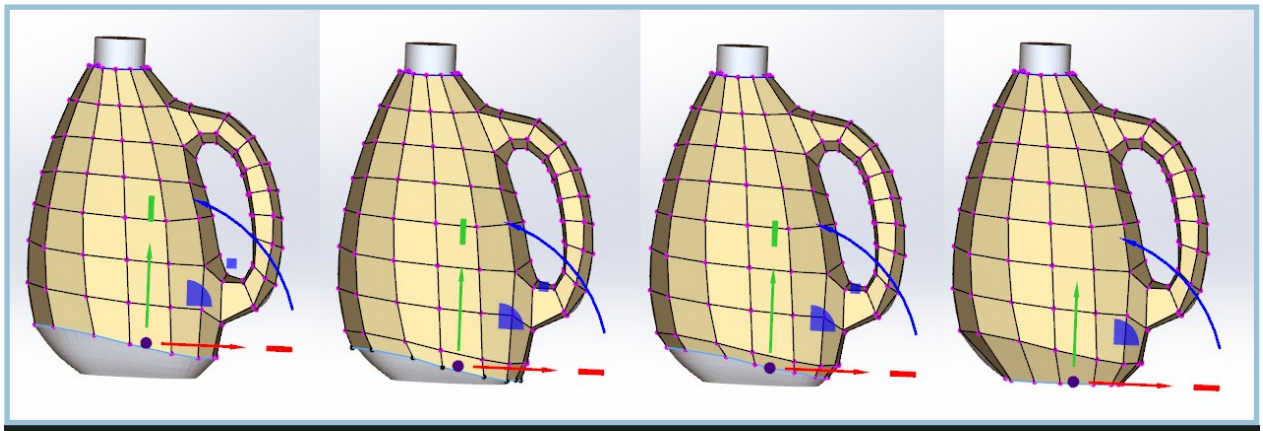
This tool averages the distance between vertices as it is painted over a retopologized surface. It should only be used on constrained vertices in *Constrained to Mesh* mode.



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

Add Constraint

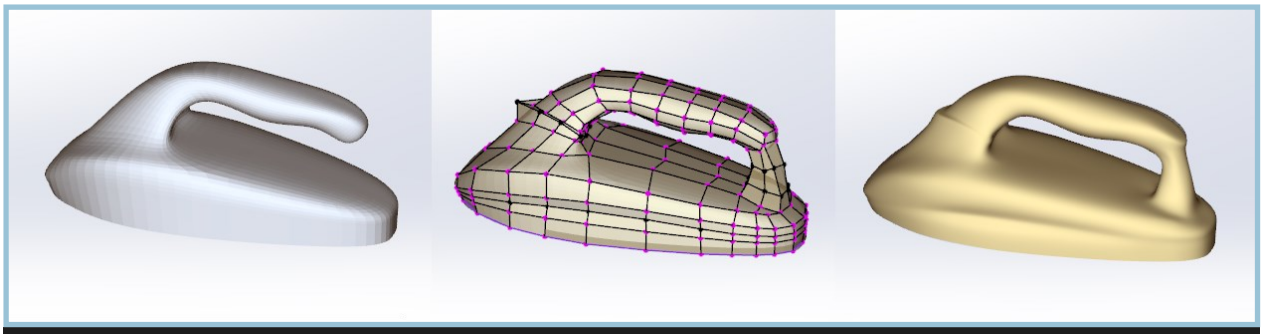
This tool sets the constrained attribute to true on the selected vertices. If edges or faces are selected, the vertices in the selections will be set to Constrained. Constrained vertices will snap to and stay on the surface of reference meshes while in Constrained to Mesh mode.



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

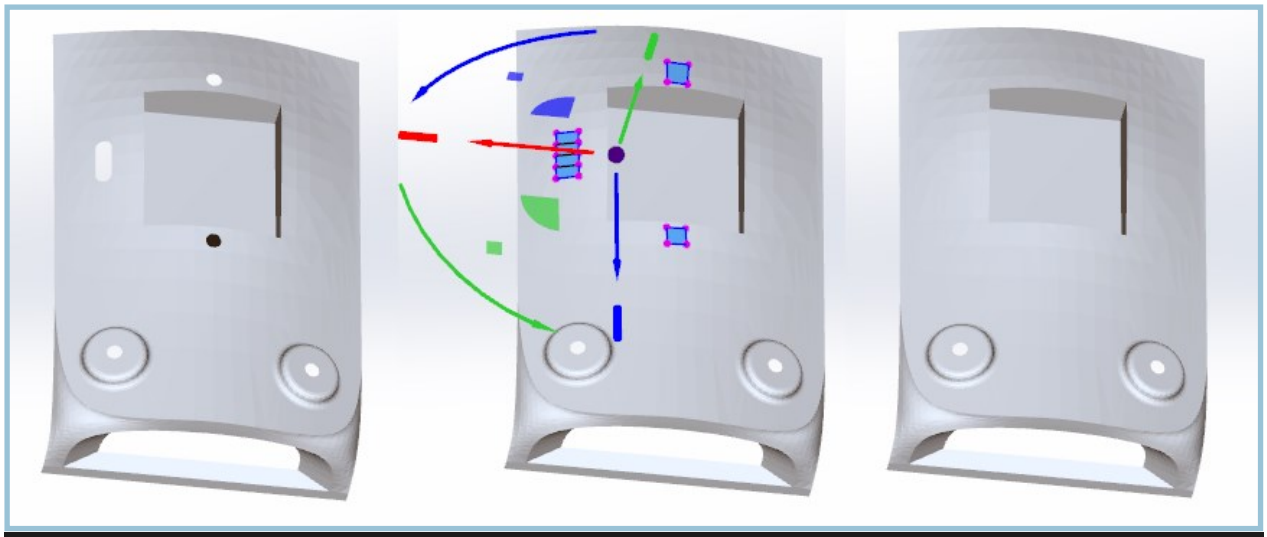
This tool sets the constrained attribute to false on vertices. If edges or faces are selected, the vertices in the selections will be set to unconstrained. Unconstrained vertices will not snap to and stay on the surface of reference meshes regardless of mode.



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

Add 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.



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

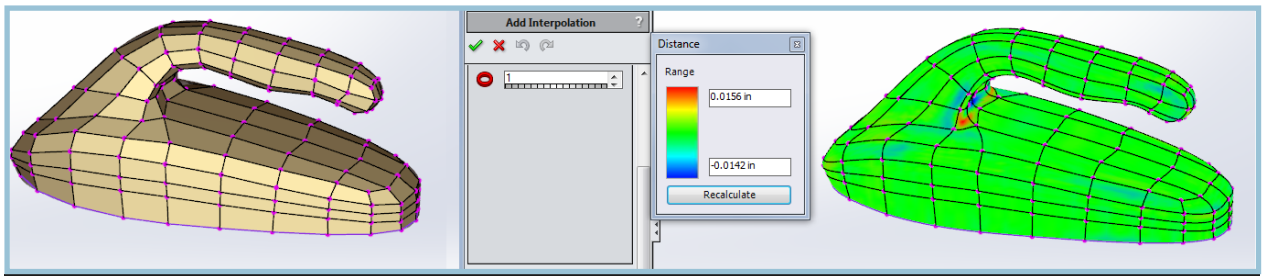
Distance

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 it. Green is considered on the mesh within the Ranges shown.

Distance (continued)

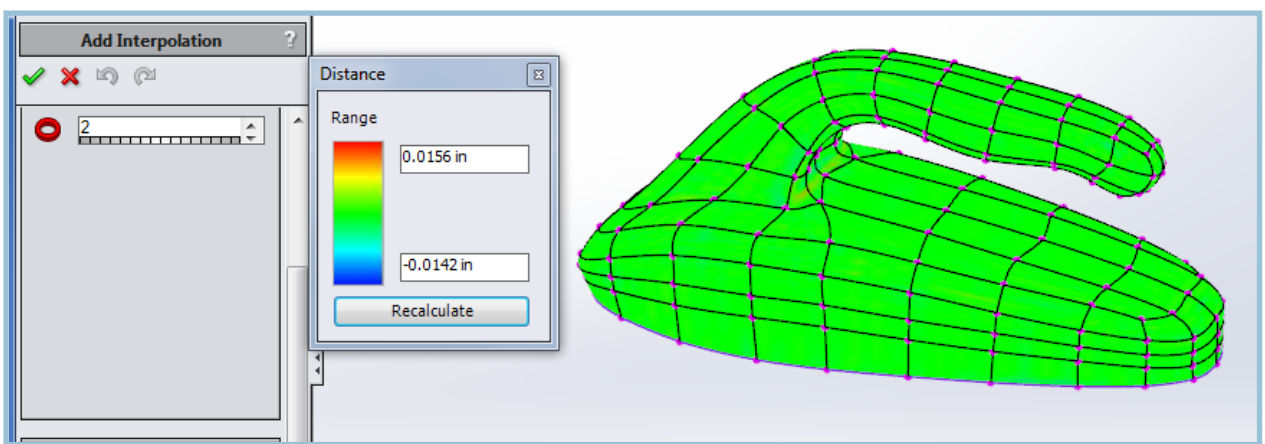


The Control mesh, left, using Add Interpolation, one subdivision, and the range of distance of the interpolated mesh, center, the mesh showing high (red) and low (blue) deviation from the reference mesh, right

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.

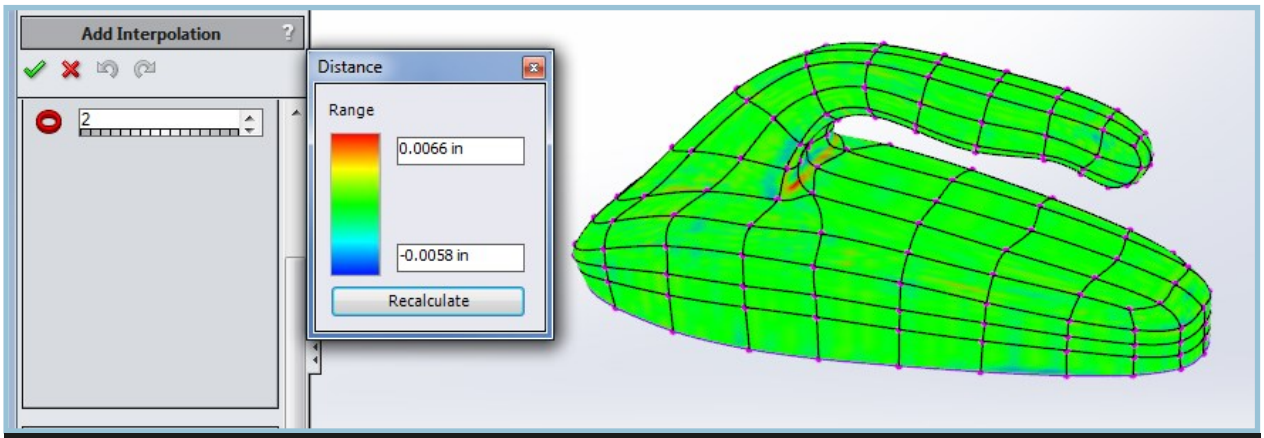


A second subdivision producing a closer fit, with range not yet updated, left, before the updated color feedback, right

Distance (continued)

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

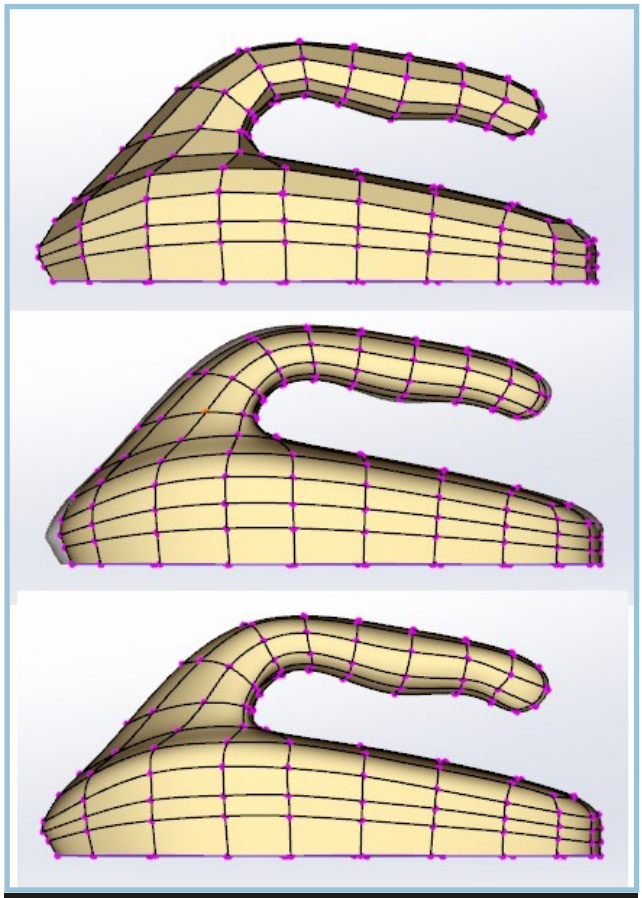
For problem areas, consider canceling the Add Interpolation, adding or adjusting edge loops for better coverage, and using Add Interpolation. If you have already accepted the interpolation, you can use Clear Interpolation before making changes.

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.

Add Interpolation

Before converting the retopo mesh, you must use Add Interpolation. This calculates the offset required to put the subdivided vertices (and addition subdivision levels) back to the same location as they were in the control mesh, e.g., the surface of the reference mesh. This step is the final step before converting the retopo mesh to a SOLIDWORKS solid or surface. Only vertices marked as “Constrained” will be affected. If you wish to adjust the model further, you should use Clear Interpolation before making the changes.



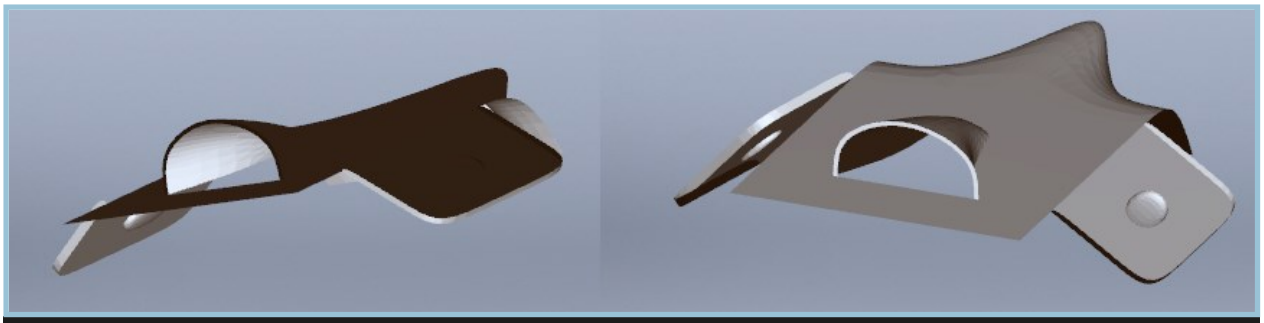
Using Add Interpolation: The SubD in Control Mesh display, top, the SubD in SubD display, center, the SubD with corrected vertex locations after Add Interpolation

Clear Interpolation

In the Add Interpolation drop down, once it has been used, you will have the option to Clear Interpolation. Use Clear Interpolation if you have accepted Add Interpolation and wish to make further changes to the retopo mesh’s constrained vertices before converting. Upon clearing the Interpolation, the mesh will continue to be shown in SubD display mode. Be sure to switch to Control Mesh display mode before making adjustments or additions to the constrained vertices.

Flip Reference Mesh Normals

This tool will flip all of the reference mesh normals in the event that the reference mesh comes in inside out. This is especially useful when dealing with surfaces rather than solids where the logical outer surface cannot be automatically determined. This option is only available in Edit Reference Mesh mode.



An imported surface with normals on the wrong side, left, after Flip Reference Mesh Normals, right

Flip Selected Mesh Normals

This tool will flip the reference mesh's *selected* face normals only. This option is extremely useful when the reference originally was composed of multiple surfaces and especially mirrored parts. This option is only available in Edit Reference Mesh mode. The Select by Element, , selection filter can often make selecting inverted faces as easy as a single click.

A reference mesh with faces inverted for one element on import, left, after selecting by element and using Flip Selected Normals, right

Orient To Plane

This alignment tool will help to re-orient a scanned reference mesh that has come in in an un-orthographic orientation. To use it, select a face or multiple faces, select the tool, then select the plane that will be used as the reference orientation. The entire reference mesh will be rotated to align with the selected plane using the average orientation of the selected face or faces as a temporary pivot point. This option is only available in Edit Reference Mesh mode.

Center To Plane

This alignment tool will center a scanned reference mesh on the selected face or faces. To use it, select a face or multiple faces, select the tool, then select the plane that will be used as the reference center. The entire reference mesh will be moved to align with the selected plane using the location of the selected face or faces as a temporary pivot point. This option is only available in Edit Reference Mesh mode.