Geomagic

Design™ X

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Geomagic®

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Contents

Introduction

Overview

This guide covers the fundamental commands and workflows utilizing the **Geomagic Design X** software.

- The graphical user interface and navigation
- Data types, processing and usage
- Standard **Geomagic Design X** workflows
- Various modeling workflows

Intended Audience

This guide is targeted to users of **Geomagic Solutions** products who require an overall understanding of the **Geomagic Design X** software.

Prerequisites

None

Guide Length

3-days

Guide Conventions

Bold Text	Indicates mouse button clicks to locate a command in the user interface in the format of **Tab > Group > Command**, example: **Model> Create Solid > Extrude** **Model***(Tab)* **> Create Solid** *(Group)* **> Extrude** *(Command)* Bold text also may be used to draw attention to or create emphasis for key concepts in activity steps.
•	A dotted bullet indicates an actionable item in the training activities.
☞	This symbol is used to identify additional information within the context of an activity step. Text following this symbol is not actionable within an activity step.
◉	A circled bullet is an information item found outside of an activity.
	This symbol is used to strongly advise or convey important information when used.

Graphical User Interface

Introduction to the Graphical User Interface (GUI).

ACTIVITY: Getting Started

Objective

Locate the primary components of the application user interface.

Training File

01 – Mouse Control.xrl

1. From the Windows Start button, select **Start > All Programs > 3D Systems > Geomagic Design X** to start the application.

or

2. From the **Desktop**, double-click the **Geomagic Design X** icon to start the application. The application will start as shown in **Figure 1**.

☞ *Clicking a **Tab** will change the current ribbon to display different command **Tab Groups**.*

☞ *The **Tree** displays the commands used and objects that are opened, imported or created.*

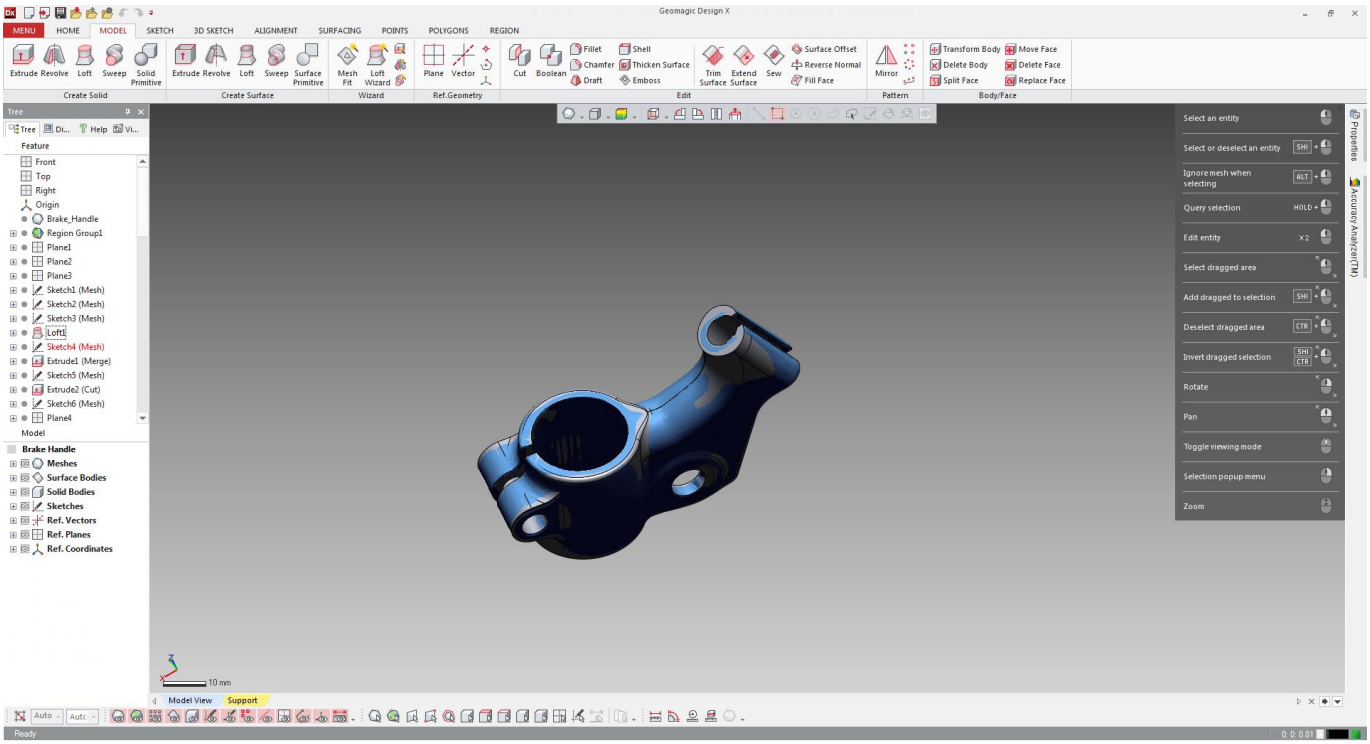


Figure 1 – Graphical User Interface

File Open

1. Click the **Open** button or select **Open** from the menu tab.
   - Use the **Open Files** dialog to navigate to the training files folder. Choose **01 – Mouse Control.xrl** from the **Open Files** dialog.

☞ *Files in **Geomagic Design X** format have an .XRL extension.*

   - Click **Open**. The file is loaded and displayed in the **Model View**, see **Figure 2**.

☞ *The Open command is for opening native Geomagic Design X files. Other file types need to be Imported.*

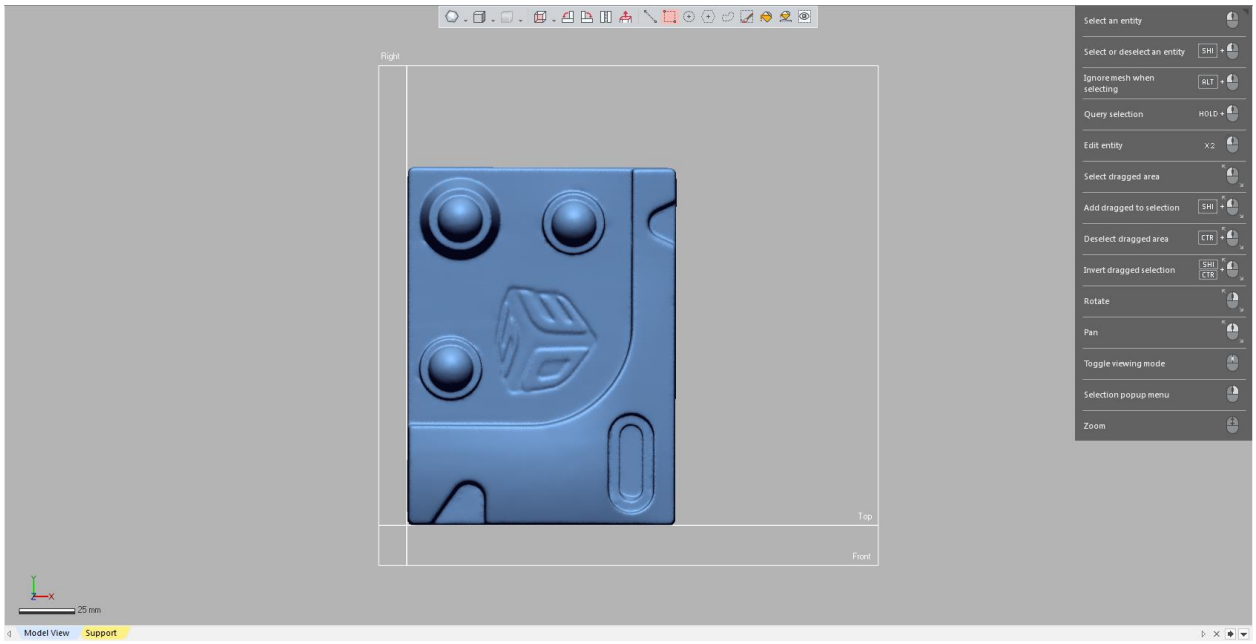


Figure 2

End of Activity

Ribbon

All commands used to create models are stored in the **tabs** at the top of the interface. Each tab has a different category of commands that are classified into groups. The Model Tab is the default tab, but all tabs can be selected with the left-mouse button.

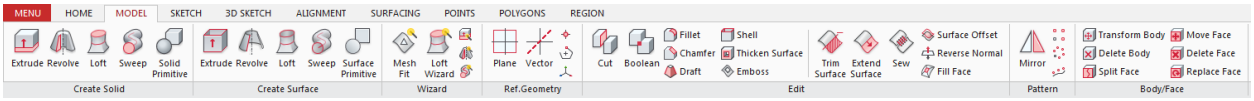
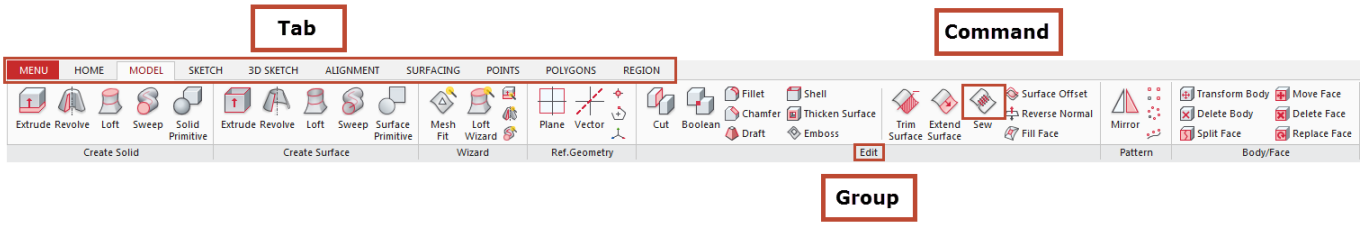


Figure 3

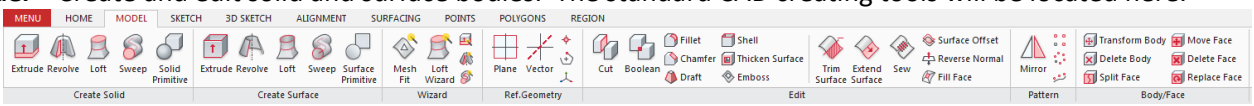
Tabs

The layout of the new ribbon user interface is shown in **Figure 3.** Each Tab is broken down into **Groups**, and the **Commands** are located within each Group.

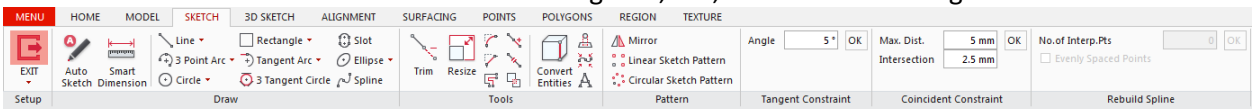


The three tabs where the majority of training tools will be found are the **Model, Sketch,** and **Polygons** tab.

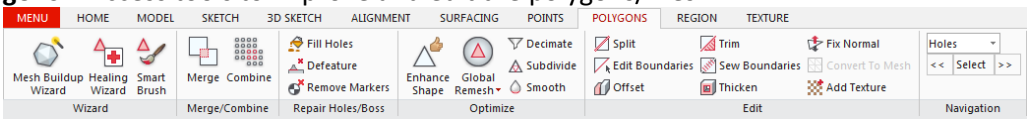
Model – Create and edit solid and surface bodies. The standard CAD creating tools will be located here.



Sketch – Create and edit a 2D or 2D Mesh Sketch using Line, Arc, and other sketching tools.



Polygons – Access tools to improve and edit the polygons/mesh.



Groups

Each **Group** within a tab will hold a set of commands that work similarly or have similar objectives.

Commands

Each **Command** will have its own properties and dialogs. Commands that are similar will have a drop down menu to toggle between the options and save space on the toolbar.

All commands not found directly on the Ribbon will be located in the **MENU** tab. Commands are categorized and stored here. Selecting a command from the **MENU** tab is the same as selecting directly from the Ribbon.

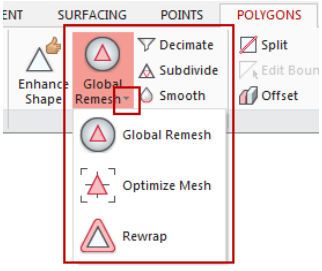


Figure 4

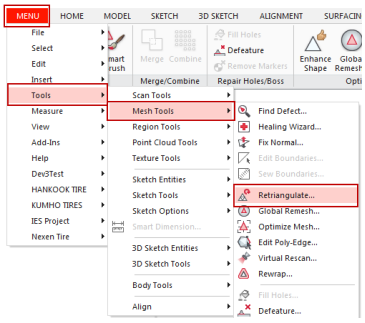


Figure 5

Upper-Side Toolbar

This toolbar contains data display modes, view options, and selection tools.

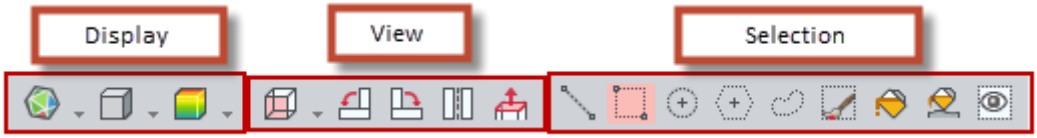


Figure 6

Mesh Display

	Point Set - Display the mesh as only the poly-vertices.
	Wireframe - Display the mesh as only the poly-edges.
	Shaded - Display the mesh as shaded poly-faces.
	Shaded with Edges - Display the mesh as shaded poly-faces with poly-edges visible.
	Curvature Display Mode - Turn on or off the visibility of the curvature map of the mesh.
	Region Display Mode - Turn on or off the visibility of the Regions.
	Geometry Type Display Mode - Change the region display to classify all region types as individual colors.

Body Display

	Wireframe - Display the bodies as only the edges.
	Hidden Line – Display the body as edges with the edges in the background displayed as dashed lines.
	Shaded - Display the bodies as faces with no drawn edges.
	Shaded with Edges - Display the bodies as faces with edges visible.

Accuracy Analyzer

	Deviation for Body – Compare the solid or surface body with the used scan data.
	Deviation for Mesh – Compare the mesh to its previous state.
	Curvature – Analyze solid or surface for areas of high curvature.
	Continuity – Display quality of edge continuity.

	**Iso Line** – Display the iso lines that define the surface.
	**Environment Mapping** – Display zebra stripes on the surface to show continuity.

Views

	**Viewpoint** - fly out will display the standard views of the Model View. All standard views are listed: Front, Back, Left, Right, Top, Bottom, and Isometric.
	**Rotate View Counter-Clockwise –** Rotate the Model View 90° to the left.
	**Rotate View Clockwise –** Rotate the Model View 90° to the right.
	**Flip Viewpoint** – Change viewing direction 180°.
	**Normal To (Ctrl+Shift+A)** – Orient the view perpendicular to a clicked surface.

Selection Tools

Toggle between the different selection shape options.

	**Line -** Select entities on a user drawn line on the screen.
	**Rectangle -** Select entities within a user drawn rectangle on the screen.
	**Circle -** Select entities within a user drawn circle on the screen.
	**Polyline -** Select entities within a user drawn polyline on the screen.
	**Freehand -** Select entities within a freehand drawn curve on the screen.
	**Paint Brush -** Select entities on a freehand drawn path on the screen.
	**Flood Selection** - Select all polygons connected to a selected polygon.
	**Extend to Similar -** Select an area of mesh with polygons connected by similar curvature.
	**Visible Only -** Select only visible objects in the current view. Deselect to select through the object.

Navigation & Selection

This section uses a polygon object as an example; all commands in this section work the same for point and CAD objects except where noted.

Mouse Commands

ACTIVITY: Using the Mouse

Objective

Learn the mouse operations in the **Model View**.

In the upper right corner of the screen, an arrow can be open or closed to display all the functions available with the mouse. This display is dynamic and will only show what is available at the moment. **Figure** 7 displays the Default mouse options.

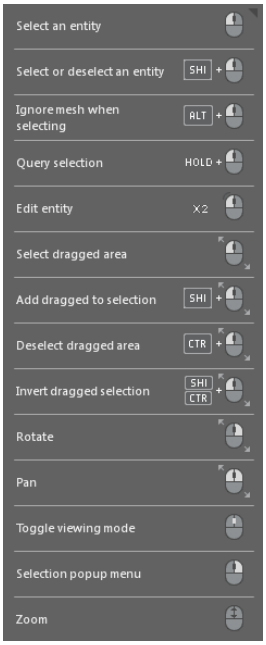


Figure 7

Rotate, Pan, and Zoom

1. **Rotate** the view by first placing the cursor in the **Model View**; press and hold the **right-mouse button (RMB)**.
   - **Moving** the mouse in all directions performs a spherical (3D) rotation pivoting around the center of the screen.
   - **Moving** the mouse vertically performs a planar rotation, parallel to the current view.

2. **Pan** the part. Press and hold down the **right-mouse button (RMB)** and then the **left-mouse button (LMB)**.
   - Position the current view by using the **Pan** command
   - **Move** the mouse right or left to pan from one side to the other. **Move** the mouse away or towards to pan from top to bottom.
   - Release to stop the pan operation.

3. Change the magnification of the object in the **Model View** by using the **Zoom** command.

- Place your cursor in the **Model View** at a location of interest and use the **scroll-wheel** to zoom in by rolling the **scroll-wheel** up. To zoom out, roll the **scroll-wheel** down.

☞ *The **Zoom** defaults to zooming to the center of the screen. This can be changed in Preferences.*

Using Selection Tools

1. Select poly-faces of the mesh.

- Click the **Rectangle Selection Mode** icon.
- Place the cursor on the object, click and hold the **left-mouse button (LMB)**.
- **Drag** your cursor to define the second corner of selection zone. As you move the cursor, a rectangle is visible in the area to be selected.
- **Release** the **LMB**. The poly-faces in the selection zone are highlighted.

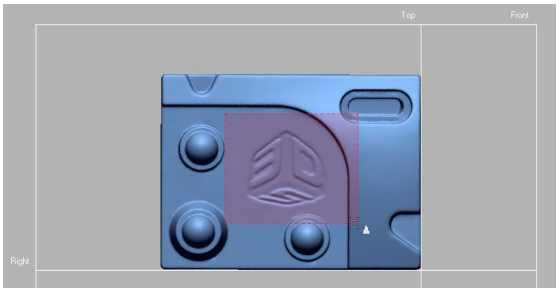


Figure 8

2. Deselect portions of the previously selected poly-faces.

- Hold down the **CTRL** key while selecting with the **LMB** in the same area highlighted area.
- Release the mouse button before releasing **CTRL**.

☞ *Clicking the **LMB** in empty space in the **Model View** deselect everything.*

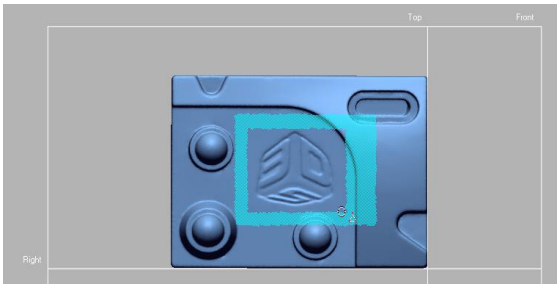


Figure 9

3. Add more poly-faces to the previous selections.

- Hold down the **SHIFT** key while selecting other areas of the mesh.

4. Delete selected areas by pressing the **DEL** key

5. Press **CTRL+Z** or the undo button on the keyboard to undo the last command.

Figure 10

☞ *The brown surface seen is the back face of the mesh. Blue is the default color of the normal side.*

6. Select and delete additional areas using the **Line, Circle**, **Polygon**, **Freehand,** and **Paint Brush Selection Modes.**

- Select the tools from the **Upper-Side Toolbar** and make selections on the object in the **Model View**.

- Press the **DEL** key to delete selected areas.

☞ *The **Line Selection Tool** is active only on polygon objects.*

Remember to hold Shift or Ctrl to add and removed portions of selected data.

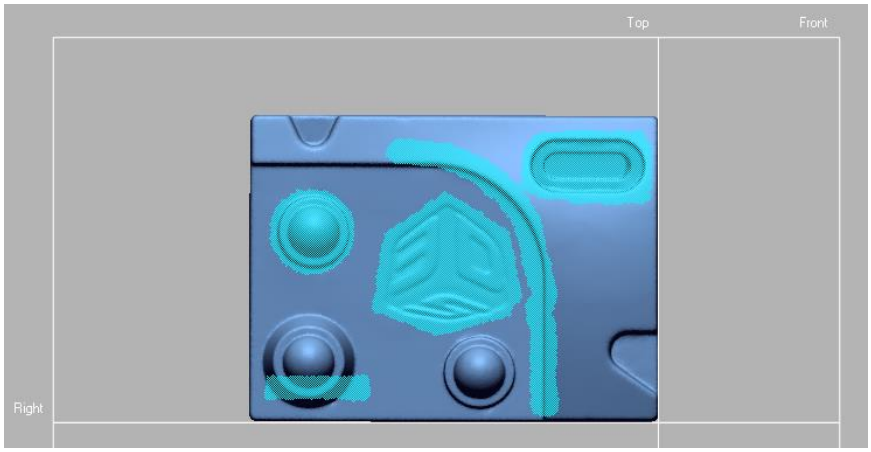


Figure 11 - Example of selected areas

7. Press **CTRL+Z** to recover (undo) the most recently deleted areas of the object.

☞ *There is one level of **undo** in the application.*

8. Deselect the **Visible Only** icon on the **Upper-Side Toolbar** and select an area on the object using the **Rectangle Selection Tool**.

- Rotate the object around to view the other side of your selection. Notice that turning off **Visible Only** causes the current selection tool to affect all areas within the selection boundary, not just those visible on screen, see **Figure 12** and **Figure 13**.
- Click in empty space to clear all selections.

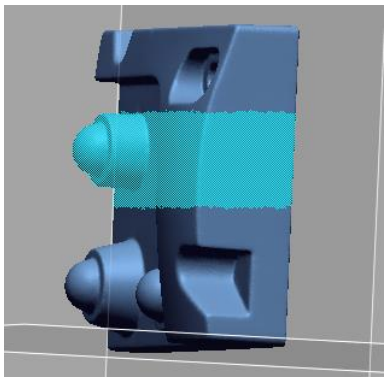


Figure 12 - Initial Selection

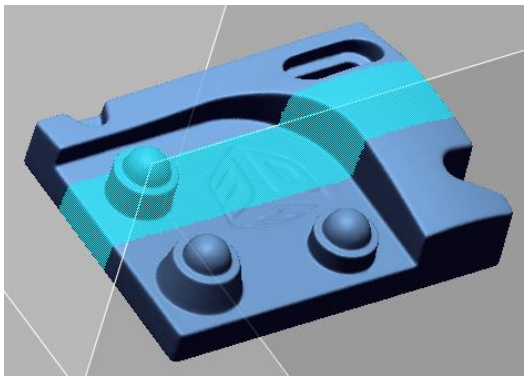


Figure 13 - Result of Selection

9. From the **Upper-Side Toolbar**, select the **Polyline Selection Tool** to create a polygonal selection.

- Click a set of points as shown in **Figure 14**. As you click, a boundary line will appear from click to click.
- To define the last point in your **Polyline Selection**, move your cursor to the last location and press the **right-mouse button (RMB)** or double click the **LMB** to end the process and make your selection.

- **Rotate** the object slightly to review the selected data.
- **Double click** in space to clear the selection.

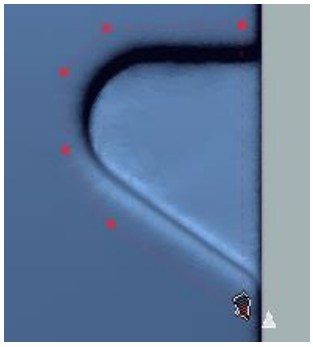


Figure 14

10. Choose the **Extend to Similar Selection Tool** to select adjacent triangles that meet have a similar curvature.

- Click on the area as shown in **Figure 15**.

☞ *Adjacent triangles in the polygon mesh that have a small degree difference between them will be selected as shown in **Figure** 16.*

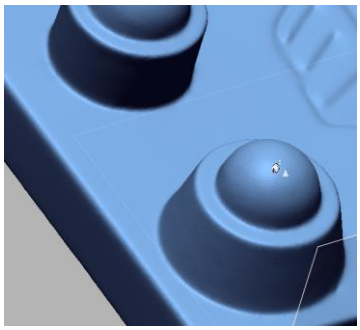


Figure 15

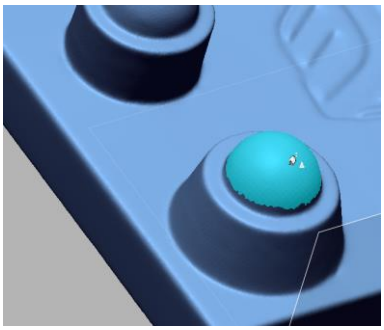


Figure 16

11. Press **Esc** to remove the selection.

☞ *Certain tools will not clear selections when clicking the mouse in space.*

- *The majority of commands in the application operate using the following selection syntax:*
  - If nothing is selected when a command is activated; the command will affect the entire active object or all active objects.
  - If a selection is present when a command is activated; the command will affect the selected area only.

End of Activity

Docks

Docks contain the properties for many different application needs. Important Design X options are contained within these docks. Docks can be positioned anywhere on the screen. They can also be pinned, hidden, and closed completely.

Tree Dock Group

ACTIVITY: Tree Management

Objective

Explore the different available docks.

- *If a dock is closed, make it visible again with a **RMB** click on the **bottom toolbar** and select the dock from the list that appears.*

Training File

02 – User Interface.xrl

Tree

The **Tree** dock on the left side is essential to the modeling process. The top half is the **Feature Tree**, and the bottom half is the **Model Tree**. The **Feature Tree** can be compared to a history tree since it lists the steps taken to create a model. The **Model Tree** lists entities that are still present in the model. The visibility of these objects is controlled here as well.

1. Click the **Open** icon on the **Quick Access Toolbar**.
   - Use the **Open Files** dialog to navigate to the folder where the training files reside. Choose **02 – User Interface.xrl** from the **Open Files** dialog and press **Open**.

☞ *This XRL file is a partially modeled scan file.*

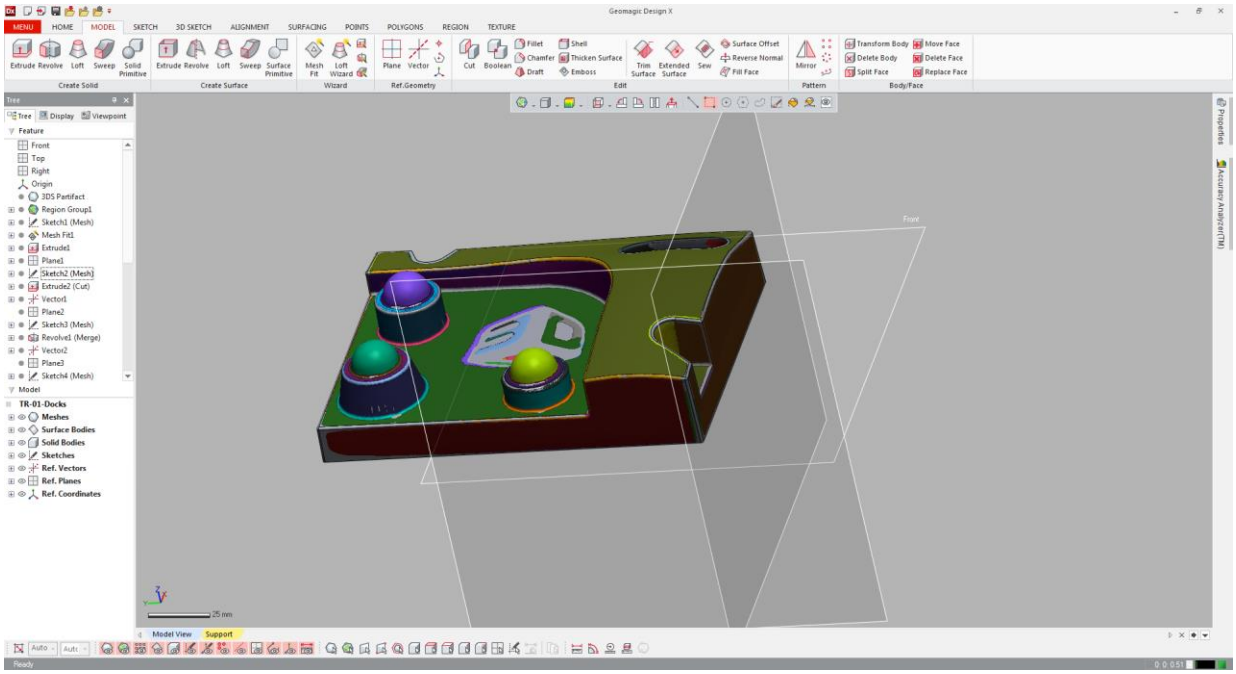


Figure 17

2. The **Feature Tree** tracks the parametric history modeling progress. Each step to create an entity is tracked in chronological order, and can be edited.

- Find **Extrude 1** in the **Feature Tree**. **Click** on the **+** sign next to it to view the associated entities.
- **RMB** click on **Extrude 1** to pull up the context menu.

☞ *A right-click context menu is available on entities in the tree. The commands available from the context menu change depending on the type of object selected.*

- Select **Edit** from the menu, as seen in **Figure 19**.
- Press the **ESC** key to cancel the edit.

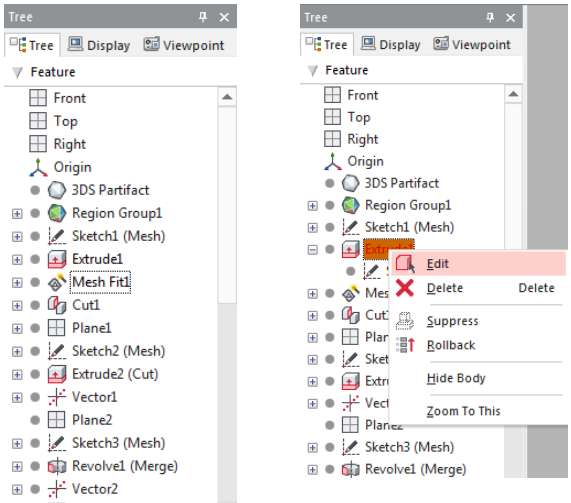


Figure 19

Figure 18 –
Feature and Model Tree

3. Geomagic Design X is a parametric history based modeler, meaning steps can be edited. Certain points in the modeling history can be accessed, and the steps in the **Feature Tree** can be reordered.

- **LMB click** one time on the **Mesh Fit 1** entity. Then **click and hold** the **LMB** on **Mesh Fit 1** and drag it to be above **Extrude 1**.

☞ *The Mesh Fit surface can now be used in the Extrude step since it comes first in the tree.*

- **RMB** on **Cut 1** in the tree and select **Rollback**.

☞ *The model can now be viewed as if the Cut and all steps after have never been performed.*

- **RMB** on an entity in the tree and select **Roll to End** to go back to last step.

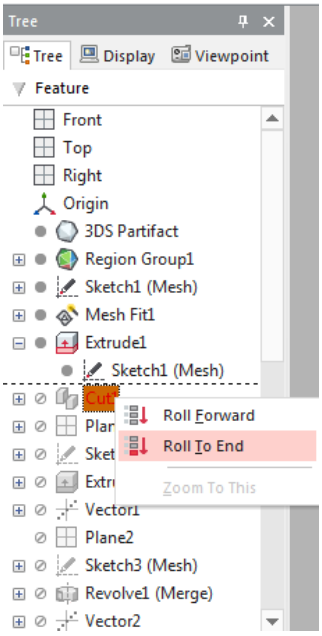
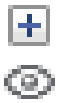


Figure 20

4. The **Model Tree** lists all entities existing in the model. **Practice** controlling the visibility in the part.



- **Click** on the **+** next to **Ref. Planes** to view all Planes in the Model.
- **Click** on the **Eye Icon** next to the + to turn off visibility for all Planes.
- **Click** on the **Closed Eye** to turn the planes back on.
- **Click** on and off the **Eyes** next to the **Front, Top,** and **Right Planes** to hide the planes individually.

☞ *Turn on and off visibility for other entities in the model.*

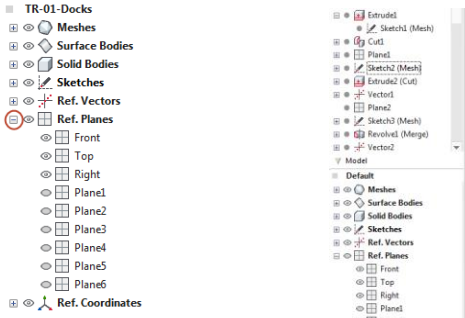


Figure 21

Figure 22

Display

The **Display dock** is defaulted as a tab next to the **Tree**. **Display** contains display options for the scan data and the bodies. It also contains additional ways to view and display data in the Model View.

1. **Click** on the **Display Tab** next to **Tree.**
2. **Place/remove** a **c**heckmark next to **World Axis & Scale, Scene Grid, Gradient Background** and **Label** to toggle visibility of each item in the Model View.

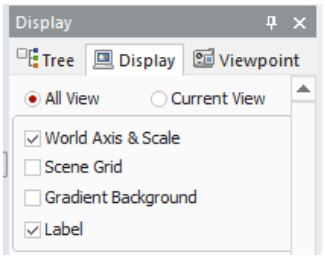


Figure 23

3. The **General** group has options for controlling the way the overall data is displayed, including Transparency, Lighting, and View settings.
4. The **Mesh/Point Cloud** group lists all the different ways to view the scan data.
5. The **Region** group gives the option to view the geometry of the selected or all regions.
6. The **Body** group lists the ways to view a surface or solid body and it allows for control of the resolution of the bodies.
7. The **Sketch & 3D Sketch** group has specific visibility options for components of sketches.

☞ *Be cautious changing options without changing them back where they started. The General>Lock option will disable all mouse rotation ability.*

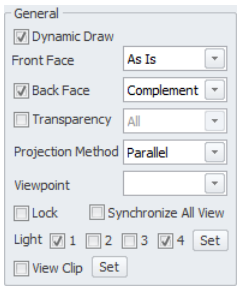


Figure 24

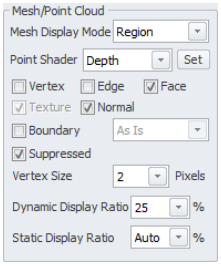


Figure 25

Figure 26

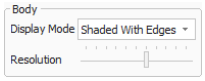


Figure 27

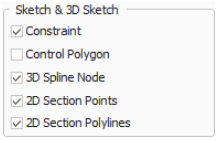


Figure 28

Help

The **Help Tab** contains a list of contents and commands to find additional information about each topic. The **Index** tab is searchable. The content of each help will explain what the tool is, the benefits of using it, how to use the tool, and then all of the options of the tool in detail. **Figure 29** shows an example of the layout for the Help tab.

☞ ***F1*** *will access the* ***Content Help*** *menu for the command that is currently being used.*

Figure 29 – Help

Viewpoint

The **Viewpoint Tab** creates and edits capture states of the current view of the model. The views can be accessed later to remember set, non-standard, views of the part.

1. Create a **Viewpoint** of the model.

- **Rotate**, **Zoom** and **Turn on/off Visibility** of the model to get a view approximately like **Figure 30**.

- **Click** on the **Add Viewpoint** button.
- **Rotate** and **Zoom** the model to a different orientation.
- Select the **Apply Viewpoint** button to return to the state of **Viewpoint1.**

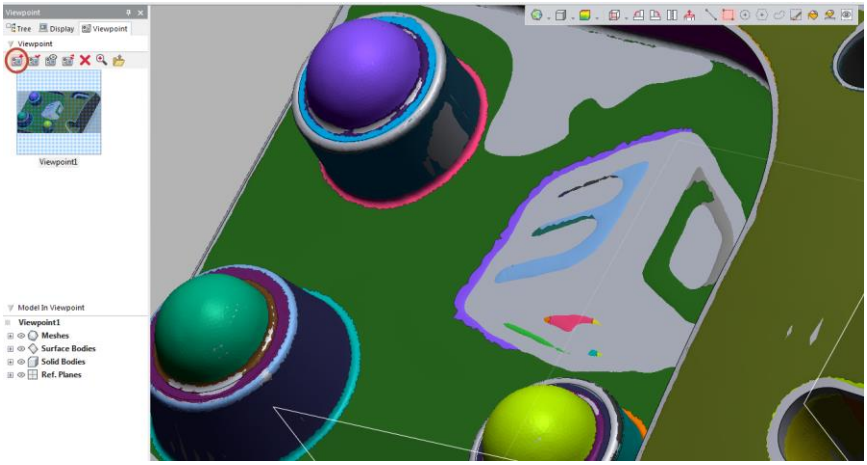


Figure 30

2. Use the **Reassign Viewpoint** command to change the Viewpoint1.

- In the **Model Tree**, turn off visibility of the Solid Bodies.
- Select **Reassign Viewpoint.**

☞ *Viewpoint 1 is now reassigned to the same view but with the Solid Bodies turned off.*

End of Activity

Accuracy Analyzer™ and Properties

ACTIVITY: Check accuracy and properties of a model

Objective

Explore the different deviation displays for checking accuracy of a model. Also, explore the different information that can be displayed in the Properties dock.

☞ *Docks can be pinned to the screen using the push pin located in the corner.*

Accuracy Analyzer™

The **Accuracy Analyzer™** allows the user to check the accuracy of their part throughout the entire process. The CAD body can be compared to the base scan data with its deviations displayed in a color map. Different ways to display the quality and continuity of the surfaces are also found here. Deviations between meshes can be analyzed. The dock is where calculation options can be set. Once deviations are applied, a **Color Bar** will appear to control the tolerance values.

1. View the deviations between the solid body and the mesh.
   - Select the radio button next to **Deviation for Body.**
2. Adjust the tolerance range of the part
   - **Double click** on the **0.1** in green, as shown in **Figure 31**, to alter the value.
   - Change the value to 0.15.

☞ *Changing the upper tolerance will automatically change the lower value.*

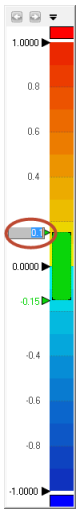


Figure 31

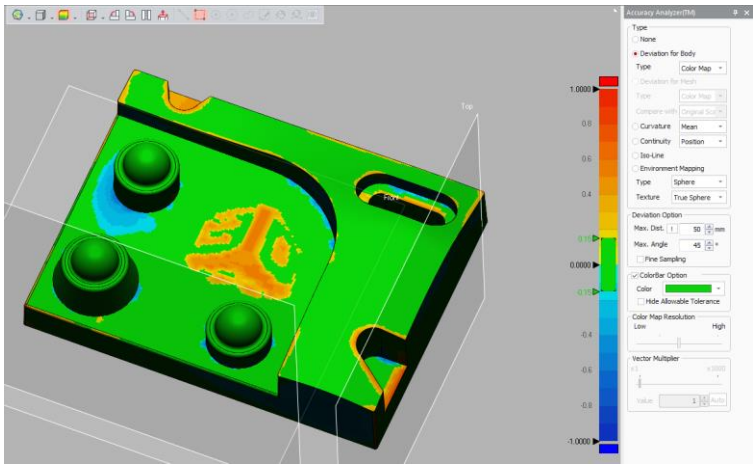


Figure 32

3. Turn the Accuracy Analyzer off by selecting **None** at the top of the dock.

Properties

The **Properties** dock displays information about any selected entities. Some properties can be changed, visibility can be turned on or off, and more information can be calculated inside Properties.

1. Calculate the Volume of the mesh.
   - Select the **3DS Partifact** mesh from the Feature Tree or Model Tree.
   - In the **Properties** dock, click **Calc.** next to Volume.

☞ *If the mesh is not fully closed, Design X will calculate the volume of the assumed closed shape.*

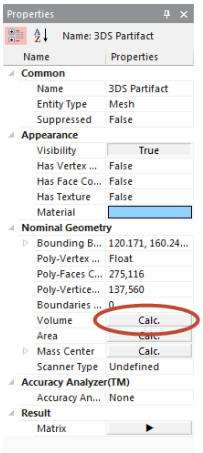


Figure 33

2. Change the color of the mesh.
   - In the Upper-Side Toolbar, change the Mesh Display Mode to **Shaded.**
   - In the Model Tree, turn the eye off next to **Solid Bodies.** The part should look like **Figure 34**.
   - Select the **blue** bar next to **Material** in the **Properties** dock. Then select the three dots on the right (**Figure 35**).
   - Change the mesh color by selecting one of the colored circles.
   - Press **OK** to accept.

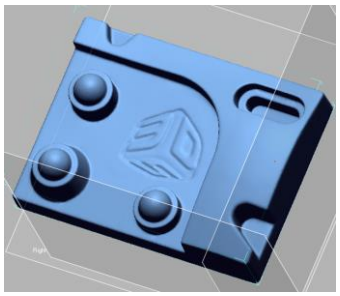


Figure 34

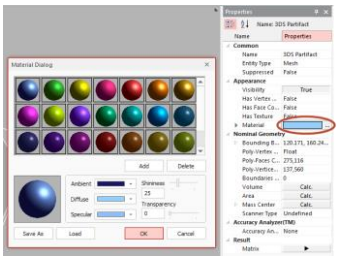


Figure 35

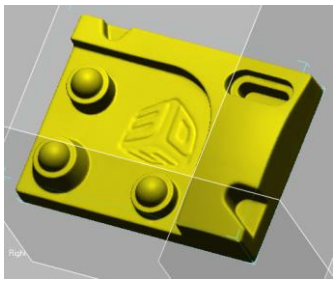


Figure 36

End of Activity

Bottom Toolbar

This toolbar contains a second location to hide and show entities, filter tools, and measuring tools.

Visibility

Turn on and off the visibility of groups of entities.

	Meshes (Ctrl+1)
	Regions (Ctrl+2)
	Point Clouds (Ctrl+3)
	Surface Bodies (Ctrl+4)
	Solid Bodies (Ctrl+5)
	Sketches (Ctrl+6)
	3D Sketches (Ctrl+7)
	Ref. Points (Ctrl+8)
	Ref. Vectors (Ctrl+9)
	Ref. Planes (Ctrl+0)
	Ref. Polylines
	Ref. Coordinate
	Measurements

Filter

Activate selection filters. User can only select the allowed entity types.

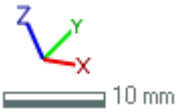
	Meshes/Point Clouds
	Regions
	Poly-Faces
	Poly-Vertices
	Mesh Boundaries
	Bodies
	Faces
	Loops
	Edges
	Vertices
	Ref. Geometries
	Sketches
	Constraints Dimensions
	Clear All Filters

Measure

Find dimensions between any entities in the Model View.

	Distance – Measure a linear distance between two entities.
	Angle – Measure an angular dimension between two entities.
	Radius – Find a radius using 3 points or a circular entity.
	Section – Create a dimension on a 2D section of an object.
	Mesh Deviation – Measure deviations between separate scan data.

World Axis and Scale



The **World Axis** displays the orientation of the global coordinate system. The Scale below shows the size of the part.

Dialogs

Geomagic dialogs have several controls in common with each other; this section describes the most commonly used controls and conventions. The majority of **Geomagic Control** commands are dialog based.

OK button – A checkmark that accepts any changes made while in the dialog and exits the dialog. A subset **check** can also mean **Apply**.

Cancel button - Discards any changes made while in the dialog and exits the dialog.

Don't Quit Command with OK button – When the option is selected, the user will remain in the command after clicking **OK** to be able to use the command again.

Preview button – Allows the user to see what changes will be made after the command is run.

Next Stage button – Only available in some commands, when another stage of options is needed. Generally the next stages have calculation options.

Previous Stage button – Go back a stage to make changes.

User Break – Cancel a command before it finishes.

Estimate – Estimate various quantities in options by reading the surrounding mesh.

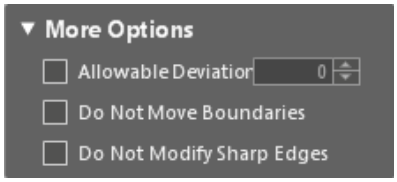
Entity Selection – The yellow box frame around an entity type means the user can select objects, such as scan data, ref. geometry, and CAD faces, to complete the command. Multiple selections can be made in one command.

☞ *If there is a vertical red stripe on the left side, an entity is required for completion.*

Select None – Remove all entities listed.

Deselect Last – Remove the last selected entity.

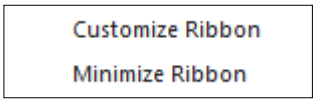
Roll-Up Groups - when rolled up, an arrow in the group title points downward. Clicking anywhere on a rolled up group title will expand the group and reveal additional dialog.



Context Menus

Geomagic Design X has several context menus activated by right-clicking in one of three locations; at the top of the screen on a user interface element, on an object in the **Model View** tab and in the **Tree Dock**.

User Interface Context Menu



- Quickly add and reorder any command to the **Ribbon**.
- Hide the ribbon; reappears when a tab is clicked.

Model View Context Menu

Right click on any object or blank space in the **Model View** will activate an action menu. Different selected entities produce different menus.

- Immediately enter a **Sketch** mode.
- Quickly start a **Wizard** command.
- Toggle through common sketching tools.
- **Accept** and **Cancel** commands with ease.

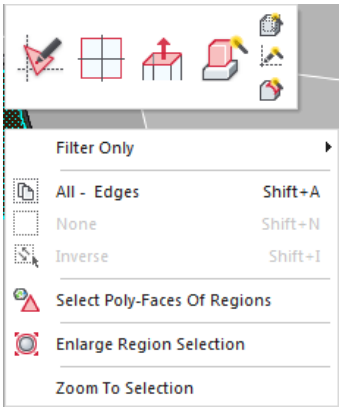


Figure 37

Dynamically changing menu

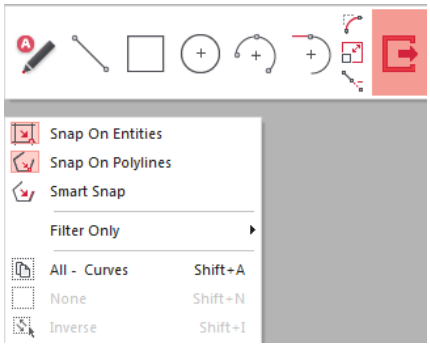


Figure 38

Context Menu – Sketch Mode

Tree Context Menu

Right Click on any entity in the **Feature** or **Model Tree** to open a context menu about the entities. Generally, this is the easiest way to find a step to edit or change the visibility state.

- **Edit** a command or sketch.
- **Suppress** an entity in the modeling history.
- **Rollback** and **Roll Forward** to a certain point of time in the model.
- **Hide** and **Show** object visibility.
- **Export** individual entities as a more generic file format.

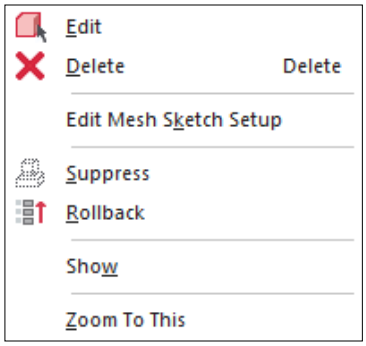


Figure 39

Context Menu – Feature Tree

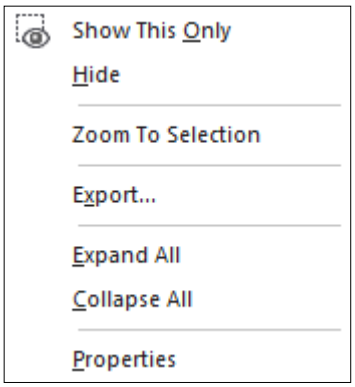


Figure 40

Context Menu – Model Tree

Data Types

Meshes, or Polygons, are common data types which consist of thousands of triangles derived from a large point cloud. The application can import several hardware device native file formats. Plug-ins are provided for several hardware devices, enabling direct input of scan data into the application.

Point Data

- Raw point data is a collection of points, each having an X, Y, Z location. When viewed up close, raw point data is displayed as simple colored points, as shown in **Figure 41**.
- Shading point data enables a better view of the shape. Raw data is unshaded due to its lack of Normal Information.
  - *Data collected using a scanner in LiveScan is automatically contains the Normal Information.*
- For best use with Design X, Points can be triangulated and merged to generate a Mesh.

Figure 41 - Raw Point Data

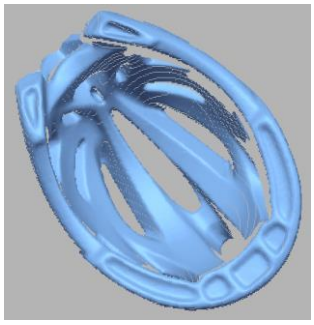


Figure 42 - Shaded Point Data with Normal Information

Mesh Data

A Mesh, or polygon, is a group of triangles, formed by triangulating point data to create the appearance of a surface for the data. Mesh data can be generated automatically when the option is selected inside of LiveScan in Design X. When a mesh object is generated the normal faces are colored blue by default, the back face or opposite side of the normal face is colored brown. The back faces of all meshes are the complementary color of the front face. The front and back face are determined by the direction of the normal of the points.

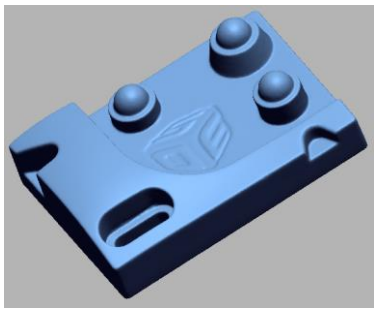


Figure 43 – Default mesh appearance

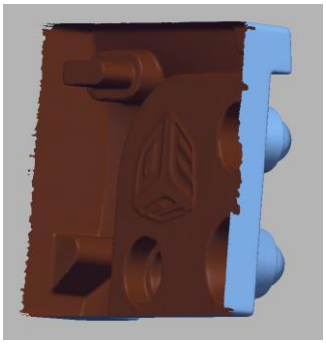


Figure 44 – Back faces of mesh

Clicking the **Display Panel** tab and toggling on **Edges** in the **Geometry Display** roll-up group will reveal the mesh construct of any mesh object.

CAD Data

CAD data is a collection of faces or bounded surfaces created in Design X, or imported in from other software.

Design X uses two different types of CAD bodies. Design X designates if a body is a **Solid** or **Surface** body. Solid bodies are grey, and surface bodies are yellow.

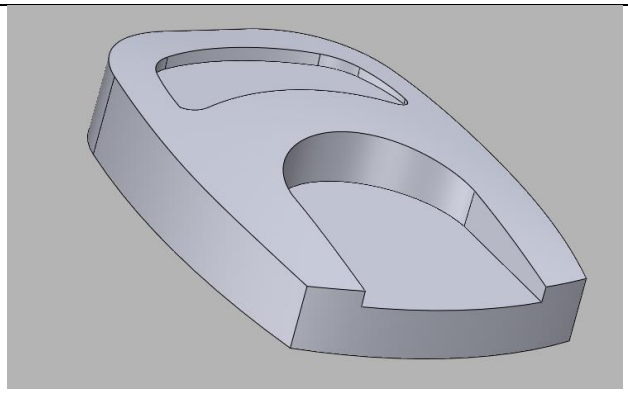


Figure 45

Solid Bodies – A collection of connected surface elements resulting in a manifold body.

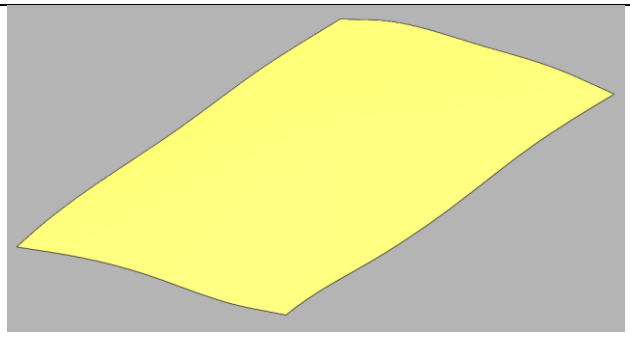


Figure 46

Surface Bodies – A non-manifold area in 2D or 3D space with defined boundaries.

CAD data is typically the primary result of using the Geomagic Design X software. It occurs by:

- Generating sketches, extrudes, revolves, lofts, etc. inside of Design X.
- Importing external CAD files, **IGES**, **STEP** or CAD neutral file exchange formats
- Directly opening or importing supported native CAD file formats

Scan Preparation

Overview

On many occasions, the scan data will require post scanning processing. Design X has the capability to align multiple scans, merge scans into one cohesive mesh, and then edit and enhance the mesh. After a mesh is generated, there are many options for output. A clean mesh can be exported to be 3D Printed, or a CAD body can created and sent for other forms of manufacturing.

Mesh Buildup Wizard

The **Mesh Buildup Wizard (MBW)** is a collection of tools to quickly create a single mesh. The scan data can be aligned and cleaned up in the MBW.

ACTIVITY: Mesh Buildup Wizard

Objective

Create a single mesh from multiple scans of points.

Highlighted Commands

- **Mesh Buildup Wizard**
  - **Align Between Scan Data**
  - **Merge**

Training File

03a – Mesh Buildup Wizard.xrl

1. Open training file **03a – Mesh Buildup Wizard.xrl**
   - Click **Open** icon in the top left corner.
   - Use the **Open Files** dialog to navigate to the folder where the training files reside. Select file **03a – Mesh Buildup Wizard.xrl** in the **Open Files** dialog.
   - Click **Open** to open selected file.

☞ *Both scan objects should be visible when the file is opened. If they are not turn them on in the **Point Clouds** group of the **Model Tree.***

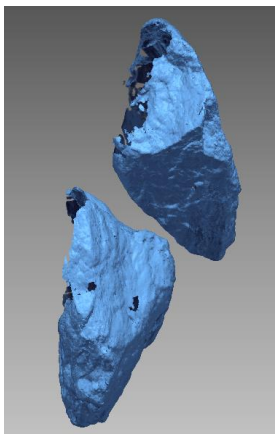


Figure 47

2. Begin the **Mesh Buildup Wizard**, located in the **POINTS** Tab>**Wizard** Group>**Mesh Buildup Wizard.**
   - The first stage of the **MBW, 1/5 – Data Preparation**, is for selecting the data and options for the wizard. If not listed under **Select** already, select the target scan data.
   - Under **Set Scanner Type,** choose **Unknown**.

☞ *Using **Unknown** will display all options in each stage.*

   - **Uncheck** any box for **Set Data Condition**

☞ *The data is not aligned already and we want to remove unwanted data.*

*The dialog box should now read **1/5 – Data Preparation** indicating all 5 stages are used.*

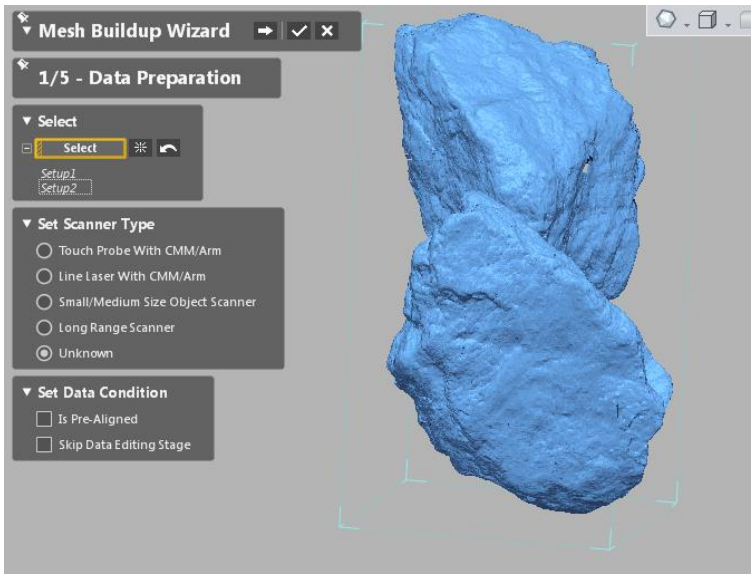


Figure 48

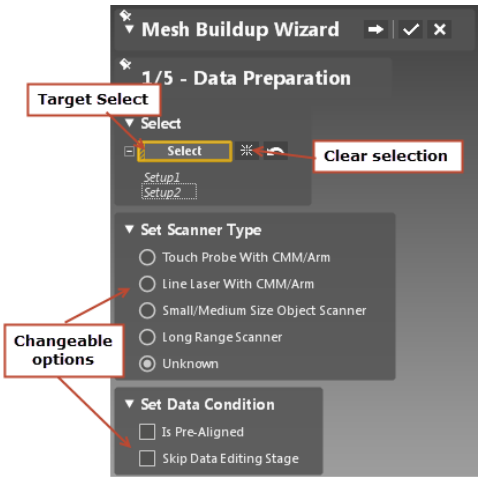


Figure 49

☞ *This dialog box has the standard set up used in the software.*

- *Select entities section*
- *Multiple roll-up groups of options*
- *Multi-stage command*

3. The **Entity Thumbnails** dock, below the Model View, allows for an organized way to view and select all the data available at the current stage.

- Click the **checkmark** next to **Setup1** in the Entity Thumbnails. The visibility of Setup1 is now off.
- Replace the **checkmark** to turn the visibility back on.
- Hover the mouse over the image of **Setup1** in the Entity Thumbnails. A small, independent, window will pop up. This view can be rotated separately from the other scans to view scan quality and completeness.

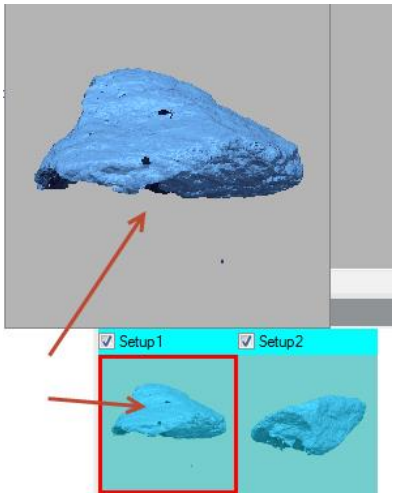


Figure 50

4. Go to the **Next Stage** of the **Mesh Buildup Wizard.**

5. Remove any unwanted points in the **2/5 - Data Editing** stage.

- Set the **Max. Poly-Vertices/Faces Count Per Noisy Cluster** to **100.**
- Click on **Find Noisy Cluster** radio button

☞ *Any groups of points, 100 or less, which are distanced from the main body, will be selected.*

- Inspect which points are selected, then press the **Delete** key or icon.

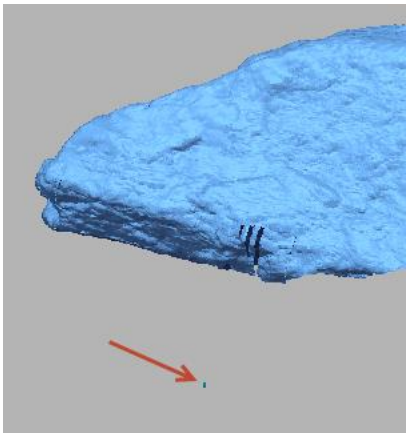


Figure 51

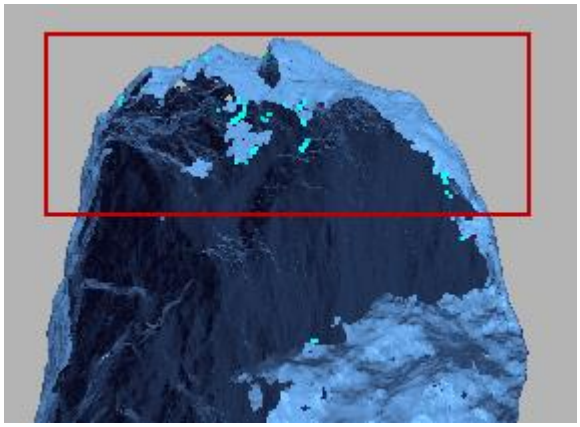


Figure 52

6. Click the **Next Stage** button in the **Mesh Buildup Wizard** dialog.

7. Use Stage **3/5 – Data Pre-Aligning** to align the two scans.

- Change the **Method** to **Local Based on Picked Point.**

☞ *The **Model View** will be divided into 3 view ports. The upper left view port is the **Fixed** view port and the upper right view port is the **Floating** view port. The wide frame at the bottom serves as a preview after registering the object in the **Floating** view port to the object in the **Fixed** view port.*

☞ *With proper scanning technique, the Auto Guess method should align the scans.*

- With **Reference** highlighted in yellow, choose one of the scans. Notice the scan now appears in the top right box labeled Reference.
- Manually select the **Moving** option and select the other scan.

☞ *One scan should be set as Reference, and one scan should be set as Moving.*

☞ *Setting one scan as **Reference** will keep it static. Scans that are pre-aligned during **LiveScan** should be set as Reference.*

- Orient each scan in the Reference and Moving boxes to have approximately the same positioning.
- Click on a point on the Reference scan that also exists on the Moving scan. A red point should appear.
- Click on the same point on the Moving scan. The red points should match.
- Repeat with at least two more points on each scan.

☞ *The order of points is Red, Green, and Blue. At least 3 points are needed to align scans.*

- Select the **Apply** check next to the 3/5 – Data Pre-Aligning line.

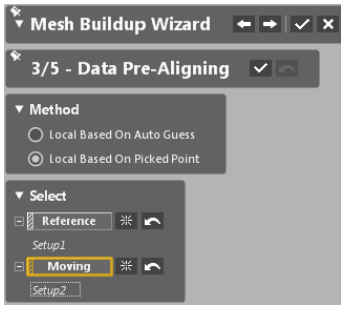


Figure 53

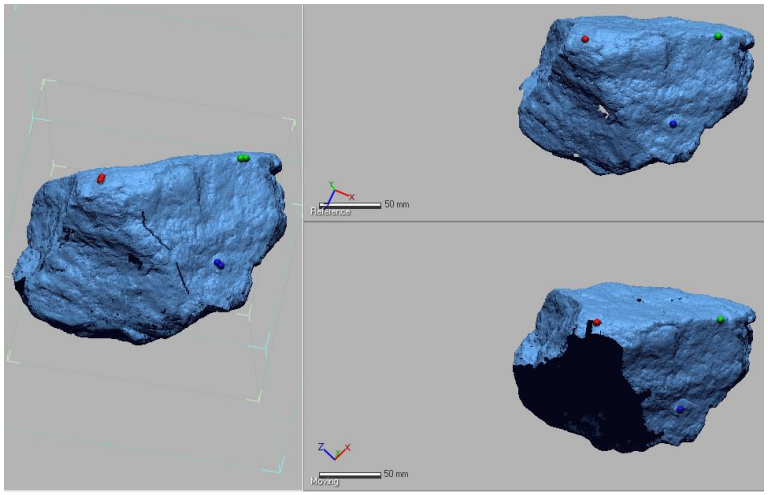


Figure 54

8. Go to the **Next Stage** of the **Mesh Buildup Wizard.**

9. Stage **4/5 – Best-Fit Aligning** will refine the alignment between the scans.

- Use a rectangular selection to highlight both scans for the **Moving** group.
- **Sampling Ratio** controls the percentage of data being used. Press the **Estimate** wand to see the amount used when set to **Auto**.

- Click the **Apply** check to complete the fine alignment of the scans.

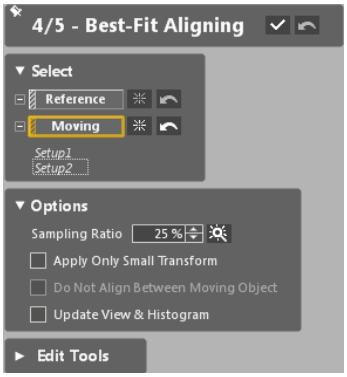


Figure 55

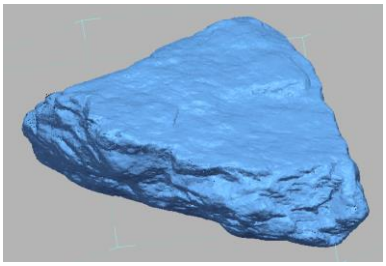


Figure 56

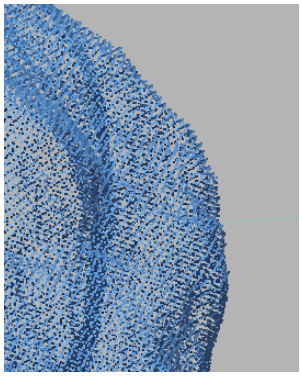


Figure 57

10. The **Best-Fit Aligning** can be run multiple times at different **Sampling Ratios**. When satisfied with the alignment, press the **Next Stage** button.

11. Stage **5/5 – Data Merging** will combine and triangulate the point clouds.

- The **MBW** has two different merging options. Select **Mesh Construction.**
- Place a **Check** next to **Reduce Noise**.
- Click the **Apply** check to create a single mesh out of the scan data.

12. Inspect the mesh, then press the overall **OK (Apply All and Finish)** check to finish the **Mesh Buildup Wizard.**

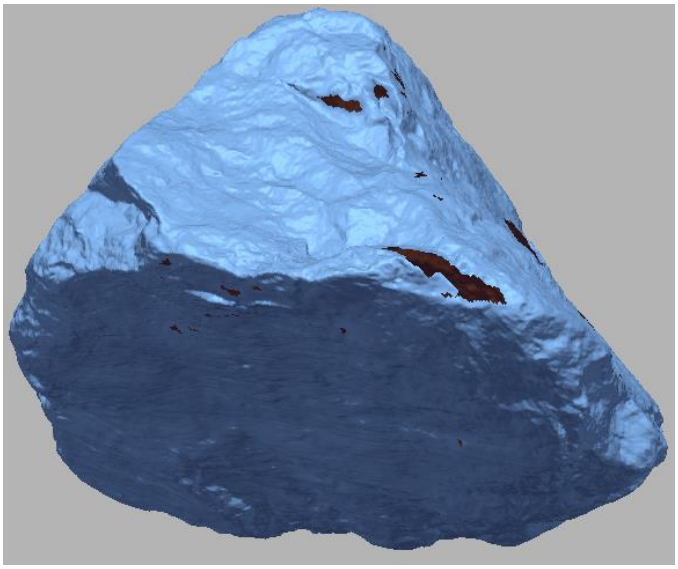


Figure 58

End of Activity

Mesh/Polygon Editing

Design X now allows for the mesh to be selected and deleted without entering any edit mode. All of the tools used to edit, optimize, enhance, and heal the mesh are in the **POLYGONS** tab.

ACTIVITY: Clean up mesh data

Objective

Create one clean, watertight, closed mesh using a variety of tools.

Highlighted Commands

- **Decimate**
- **Fill Holes**
- **Rewrap**
- **Enhance**
- **Healing Wizard**

Training File

Continue from previous or Open **03b – Mesh.xrl**

Reduce Polyface Count

After the Mesh Buildup Wizard, the resulting mesh may have more polyfaces, or triangles, than needed. The **Decimate** command can intelligently remove and merge polyfaces to have the lowest amount possible. This will allow for future processing times to be significantly reduced. Saved file sizes will also be smaller.

1. Select the **POLYGONS** tab to easily access all of the mesh editing tools.

2. Click on the **Decimate** command located in **POLYGONS>Optimize>Decimate** and progress through the dialog menu from top to bottom.
   - Select the mesh as the **Target Mesh**.
   - Change the **Method** to **Allowable Deviation**.

☞ *This method will remove as many polyfaces as possible without allowing the mesh to change at a greater distance than the set value.*

   - Under **Options** set the value to **0.0254 mm.**
   - Press the **OK** check to **Decimate** the scan.

☞ *The total triangles now should be under 1,000,000.*

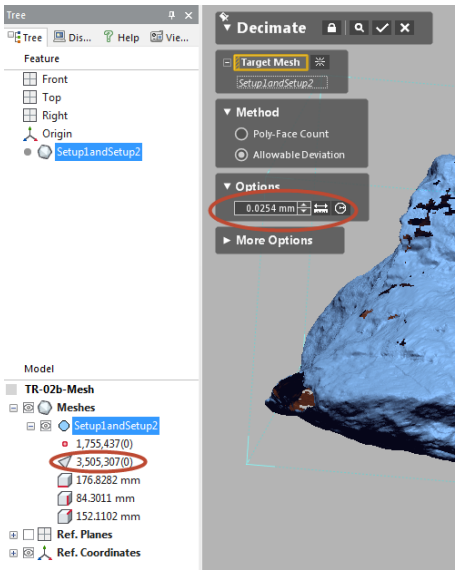


Figure 59

Create Manifold Mesh

Downstream applications, such as 3D printing, require a well cleaned mesh. A watertight mesh will produce best results for 3D printing applications. Geomagic Design X has tools that can fill in individual holes, and it can also wrap an entire part to fill in all the holes at once.

1. Navigate and zoom in to approximately the view shown in **Figure 60**.

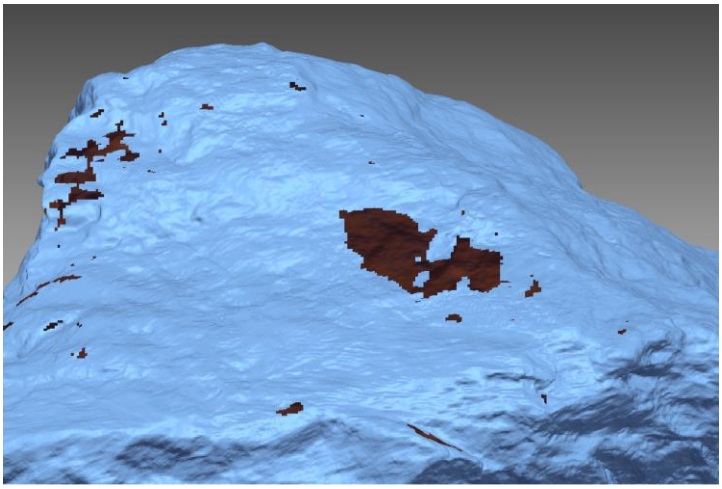


Figure 60

Fill Holes

1. Select the **Fill Holes** command located in **POLYGONS>Repair Holes/Boss.**

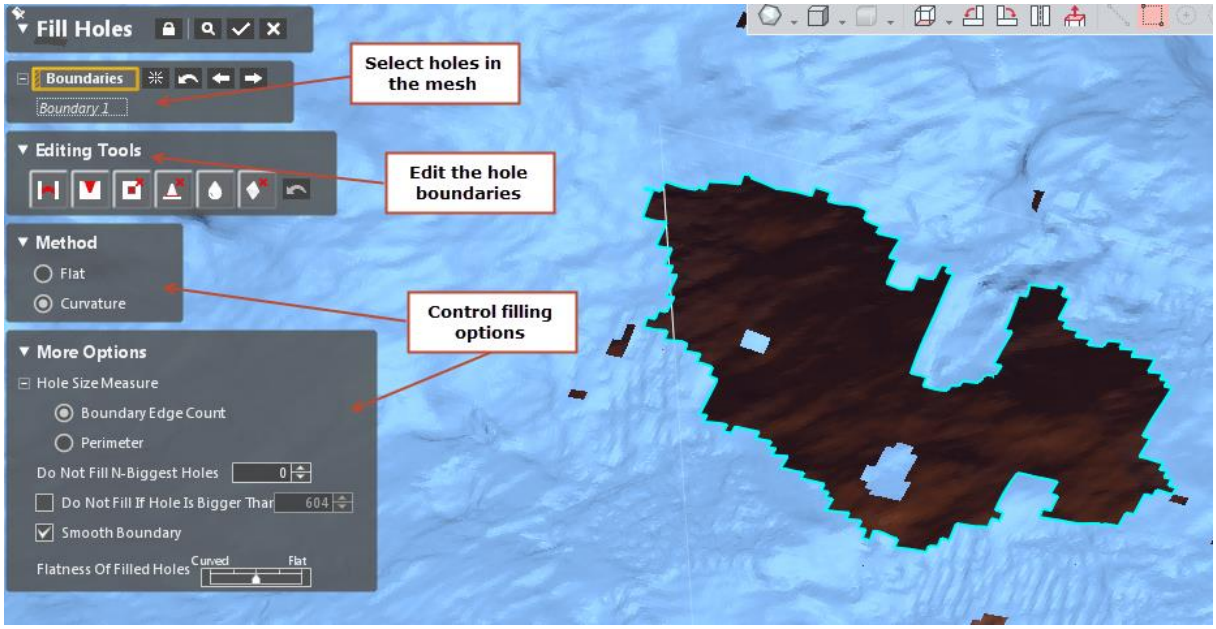


Figure 61 – Fill Holes Dialog Layout

2. Choose the large boundary shown above in **Figure 61**. Once the command is accepted, this hole will fill in.

3. Edit the boundary to make a cleaner perimeter. Select **Fill Gulf** from the **Editing Tools** rollup group.

- Left click one time on the top, left-most point of the gap shown below in **Figure 62**.
- Left click again on the opposite side of the gap to immediately fill in a portion of the hole.

☞ *Before making the second selection, a preview of the area to be filled will be shown.*

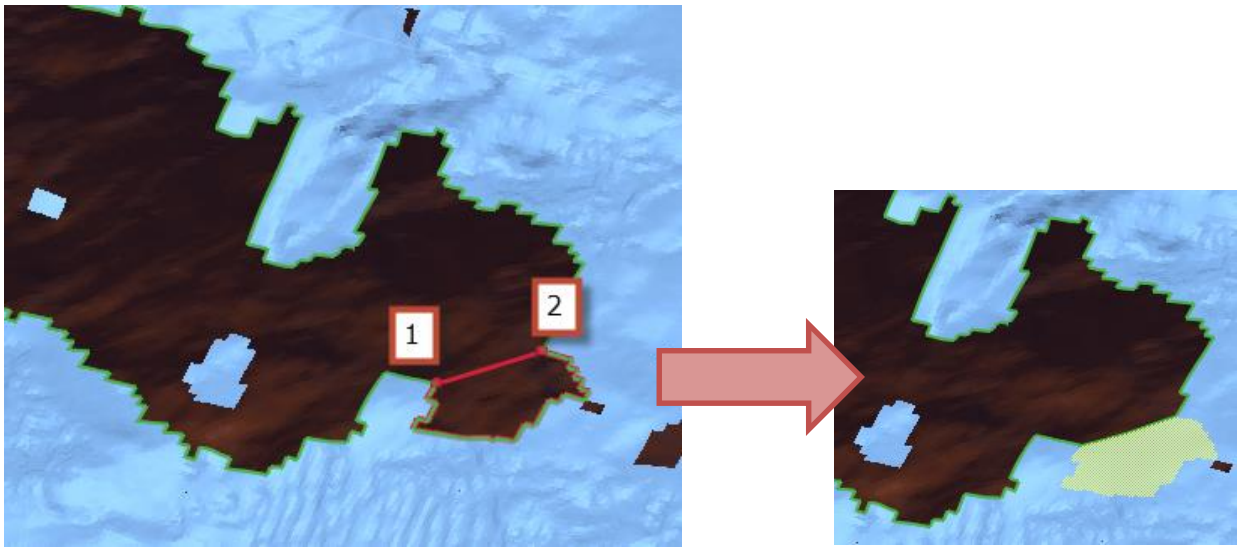


Figure 62 – Fill Gulf

4. Use **Remove Peninsula** to quickly delete excess, indented data. Edit the boundary to make a cleaner perimeter.

- Left click one time on the top, left corner of the area shown in **Figure 63**.
- Left click again on the opposite end to instantly delete this portion of data.

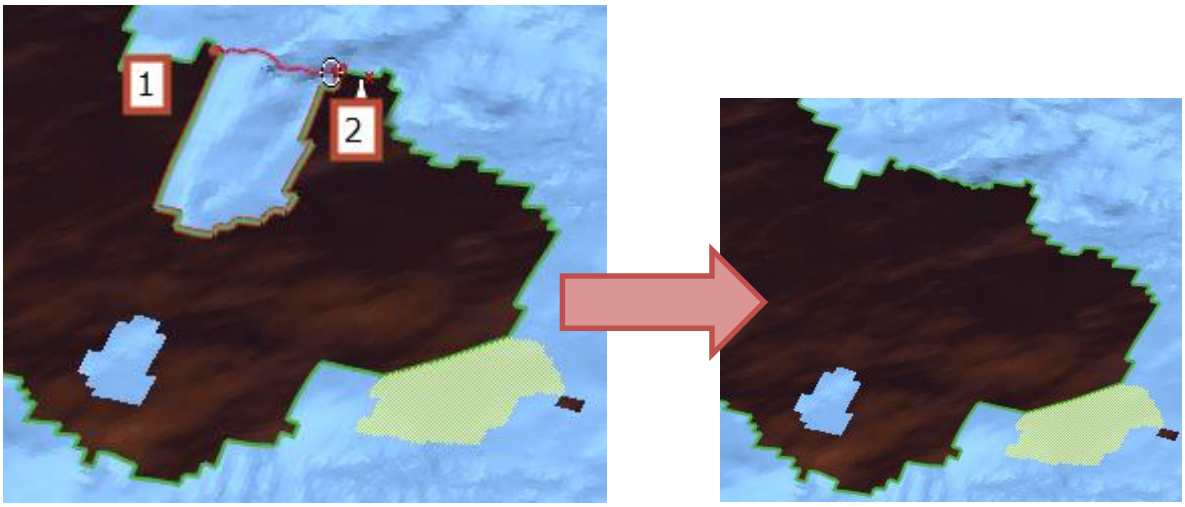


Figure 63 – Remove Peninsula

5. Select the **Remove Island** Editing Tool to delete floating areas of data.

- Left click on the boundaries of any islands to delete them.

*Do not click on any boundary of the main mesh. The entire mesh will delete. To **Undo** any Editing Tools, click on the left-pointing arrow at the end of the Editing Tools rollup menu.*

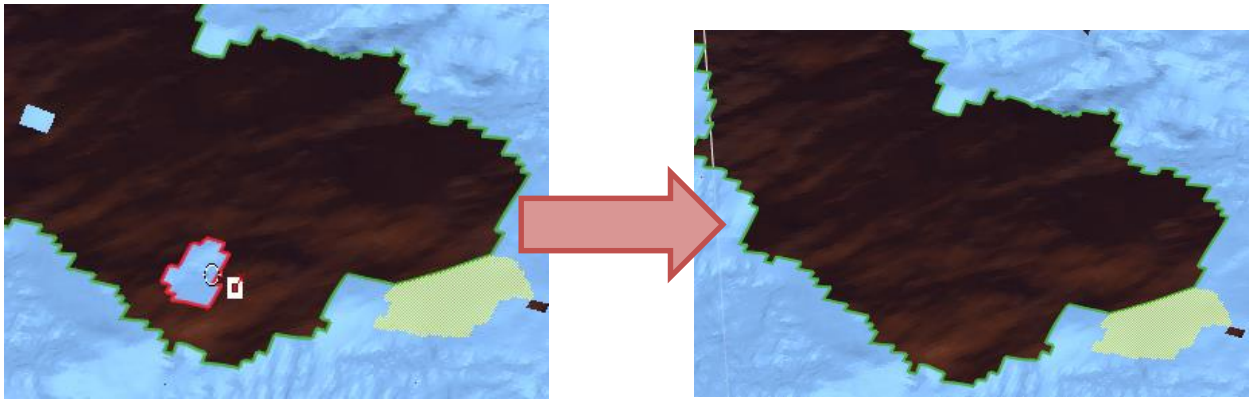


Figure 64 – Remove Island

6. Select **Add Bridge** to split the boundary into smaller sections.

- Left click and **hold** the mouse button down on one edge of one polyface.

☞ *The polyface edge will highlight when selected.*

- While holding the LMB, select a triangle on the opposite side of the boundary.

☞ *The original selected boundary is now listed as two boundaries.*

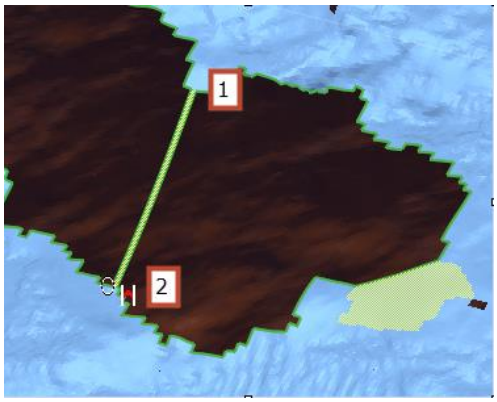


Figure 65

7. Adjust the **More Options** settings to match the previous **Figure 66**.

- Add a check next to **Smooth Boundary** in the **More Options** rollup group.

8. Accept the **Fill Holes** command by pressing the **OK** check.

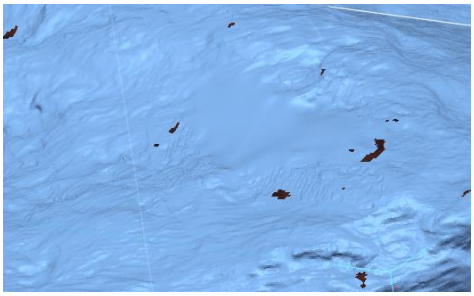


Figure 66

Rewrap

Instead of having to select every boundary in the part, the Rewrap command can extend all boundaries until there are no holes in the scan.

1. In the dropdown from **POLYGONS>Optimize>**Global Remesh, select **Rewrap**.
   - Under **Fitting Options**, slide the **Geometry Capture Accuracy** bar over to **Tight**. Move the **Overall Smoothness** bar to **Min.**

☞ *This will preserve the organic shape of the scan as much as possible.*

   - Place a check next to **Extended Boundaries** in the **More Options** group.

☞ *Extend Boundaries on will fill in every size hole. If it is deselected, the very large holes will not fill.*

   - Click **OK** to run the command.

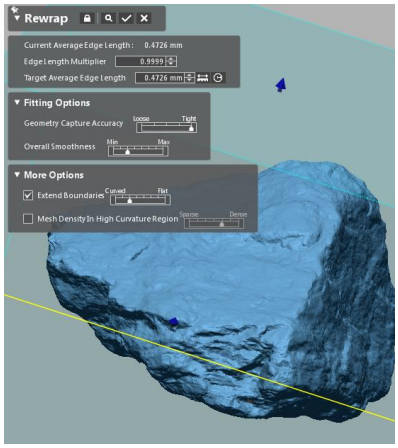


Figure 67

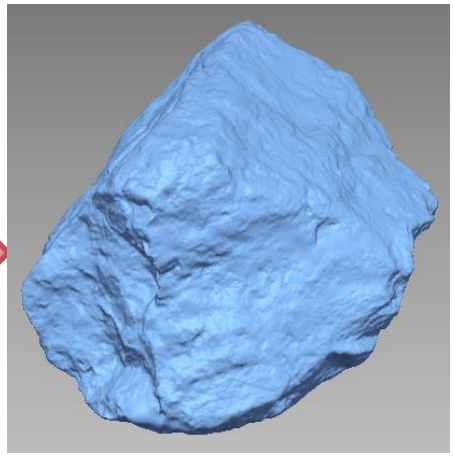


Figure 68

Optimize Mesh

For higher quality printing, the scan data should be enhanced and optimized further. For parametric modeling, the following steps are not generally needed. A higher quality mesh will create higher quality surfaces.

1. In the **POLYGONS** tab, select the **Enhance Shape** command in the **Optimize** group.
   - Increase the **Sharpness** to **Max**.
   - Decrease **Overall Smoothness** to **Min.**
   - The **Enhance Level** can be set to the **halfway** point.

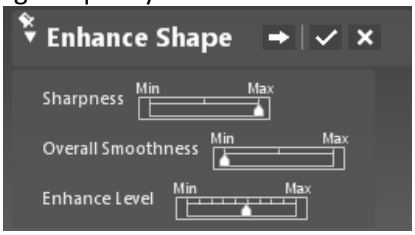


Figure 69

☞ *The **Enhance Level** can be altered in the second stage.*

   - Go to the **Next Stage** by clicking the arrow.

2. Adjust the **Enhance Level** to find optimal result.
   - Decrease the slide bar to **Min**.
   - Check the mesh to see the results.
   - Increase the slide bar to **Max**.
   - View the difference in the mesh.
   - Adjust the slide bar to the **third** mark.

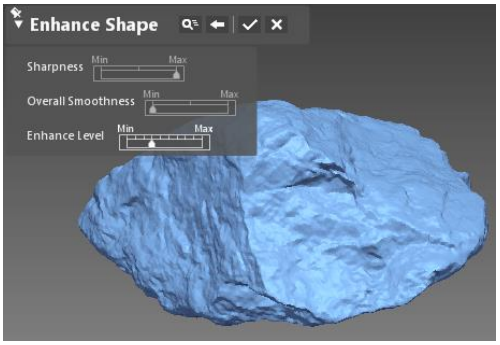


Figure 70

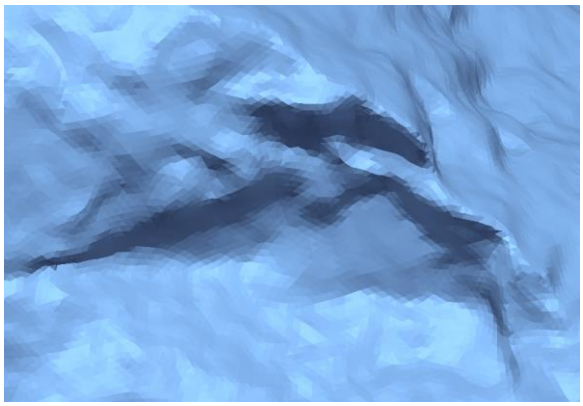


Figure 71 – Min Enhance Level

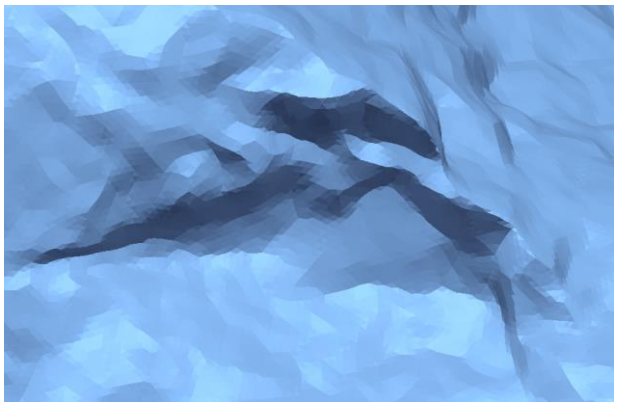


Figure 72 **– Max Enhance Level**

- **Accept** the Enhance Shape command with the **OK** check.

3. Select the **Healing Wizard** in **POLYGONS>Wizard>Healing Wizard** to remove the last erroneous triangles. This command finds these triangles automatically, and after accepting the command, they are removed.

4. **Accept** or **Cancel** the Healing Wizard to exit.

☞ *The Healing Wizard will find any small groups of floating triangles, small polyfaces, or dangling, folded, or crossing poly-faces automatically. The parameters for each of these categories can be modified as needed.*

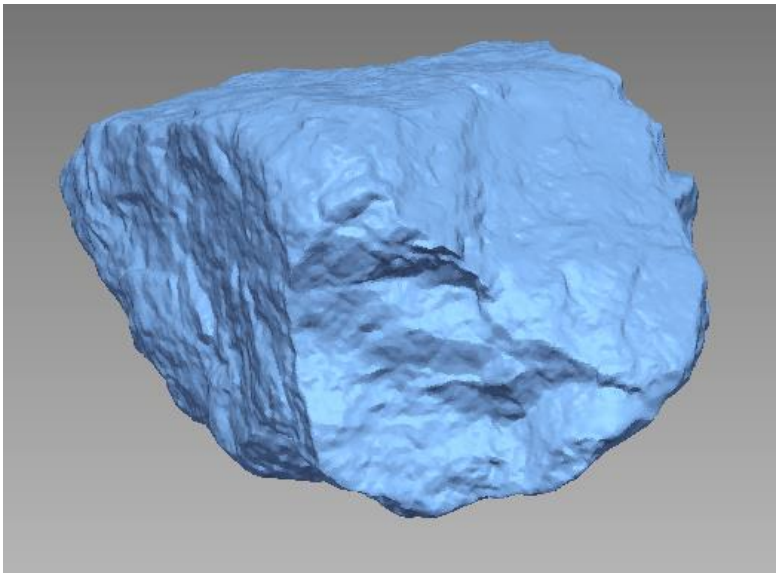


Figure 73

- The Mesh is now ready for 3D printing, surfacing, or other production methods.
- Additional mesh editing exercises can be found in the Appendix**.** Design X also has built in Tutorials.

End of Activity

Auto Surface

A quick, easy, non-parametric way to generate an IGES or STEP file from a cleaned Mesh is the **Auto Surface**. The mesh is analyzed and surface contours are automatically generated to create extremely accurate surfaces to the scan.

ACTIVITY: Generate CAD Body

Objective

Create an accurate CAD body to the scan data. The resulting body can be exported as a STP, IGS, x_t file, or other generic file types.

Highlighted Commands

- **Auto Surface**

Training File

Continued from previous or Open **03c - AutoSurface.xrl**

1. Open training file **03c - AutoSurface.xrl** or continue from previous exercise**.**

2. Enter the **Auto Surface** command located in the **SURFACING** tab.
   - Select **Organic** in the first option for patch structure.

☞ *The **Organic** method creates a curve network as evenly distributed as possible, and is good for models that have non geometric faces and with few distinct feature shapes. The **Mechanical** method will create a curve network more uniform and grid like.*

- Leave the **Patch Count** in **Patch Network Option** at **Auto Estimate** to let the software decide the amount of patches needed for surfacing.
- **Slide** the **Geometry Capture Accuracy** bar one away from Max.
- **Change** the **Tolerance** to **0.1285**.
- Go to the **Next Stage** in the command.

☞ *The first stage of the Auto Surface command sets how the software will construct its NURBS network. The Auto Surface command applies to the entire mesh.*

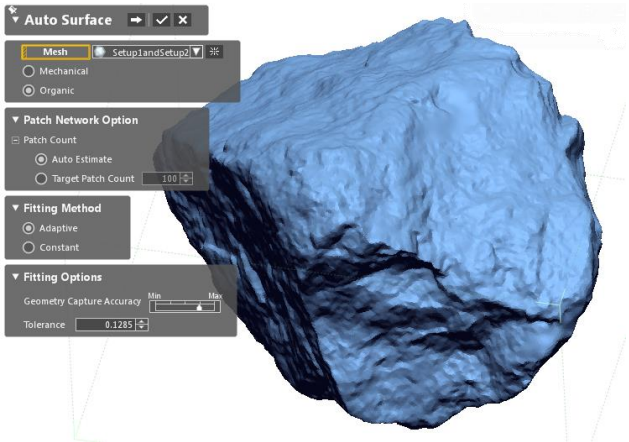


Figure 74 – First Stage

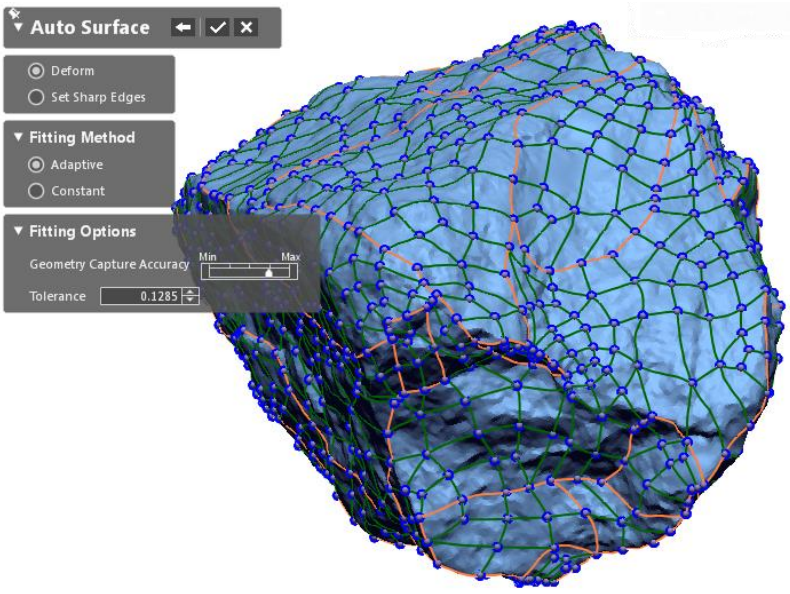


Figure 75 – Second Stage

3. The second stage allows for editing of the generated patch network.

- Select the **Deform** option.
- Zoom in to the area shown in **Figure 76**.
- Click and drag the center node of multiple patches to manipulate the form of all 5 quadrilaterals.

☞ *If a patch is not in a clear enough four-sided shape, the software might not fit a surface. Occasionally patches need to be edited and reshaped in order to build a better surface body.*

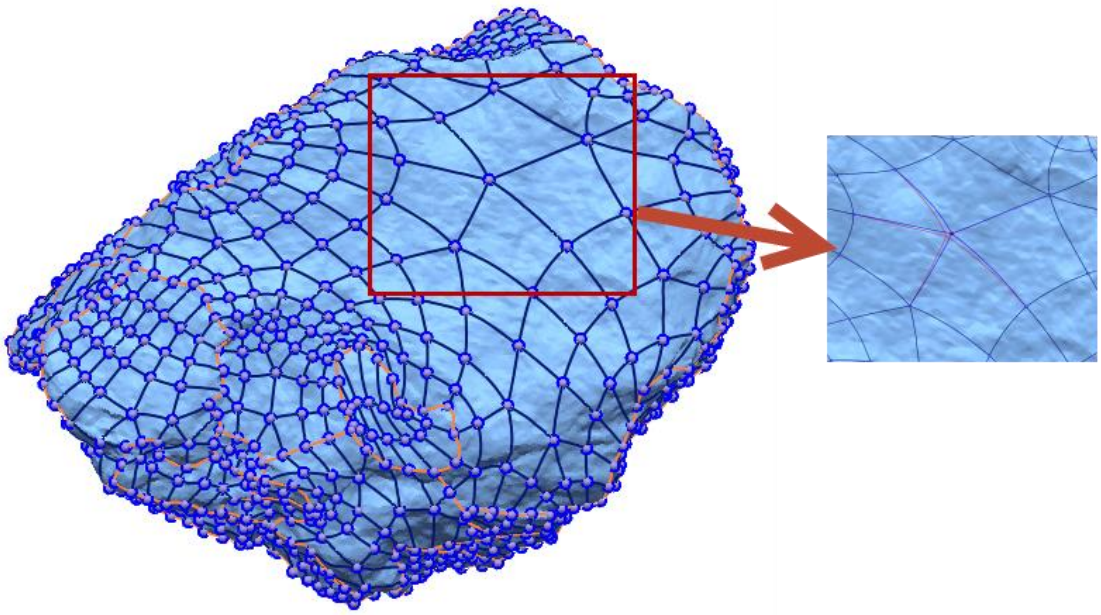


Figure 76

4. Edit the form of a single edge of a patch with the **Deform** option.

- Click and drag a node on a single spline to adjust the edge of a patch.

☞ *Straightening out a patch edge can help create better quality connections between the patches.*

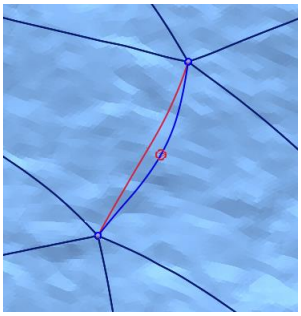


Figure 77

5. Press the **OK** checkmark to accept the **Auto Surface**.

☞ *If the resulting body is a manifold network of surface patches, the body will become a **Solid.***

☞ *Editing the network is generally only needed in the Organic method when patches fail. The Auto Surface can be edited after completion to go correct any failed patches.*

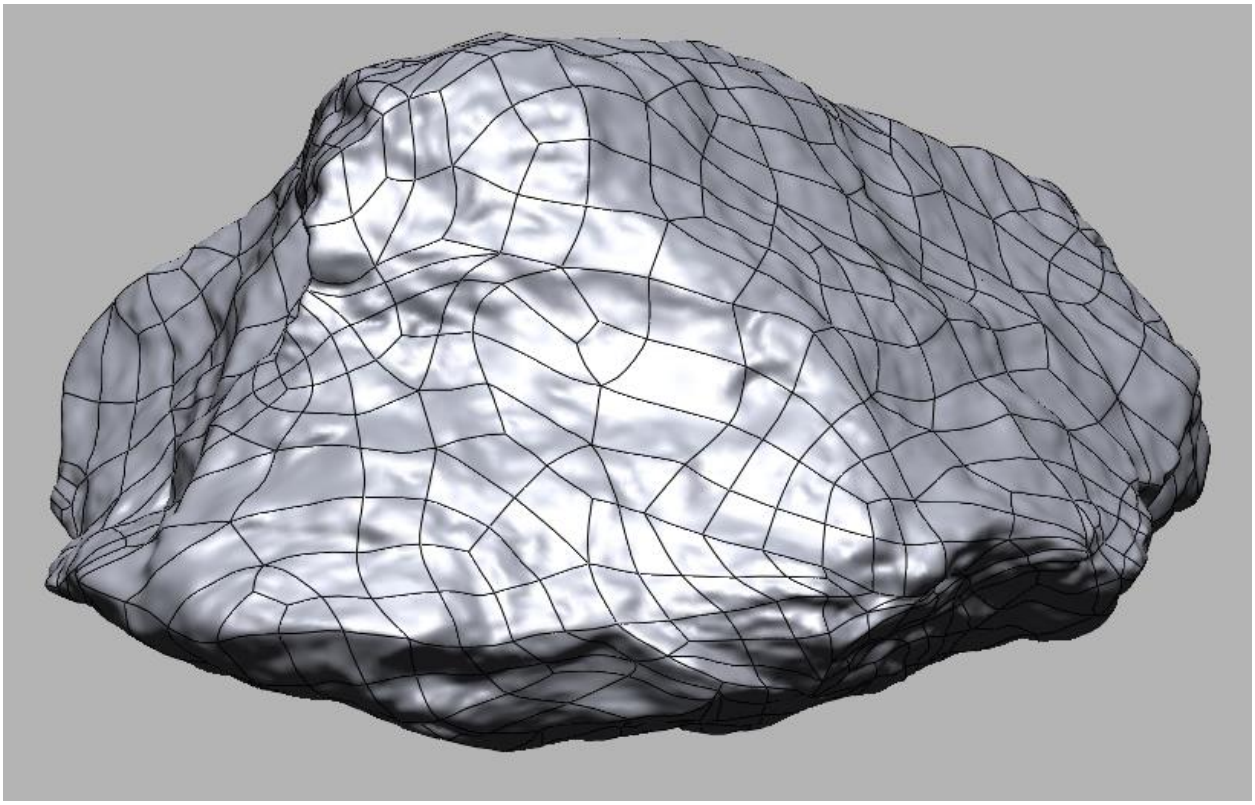


Figure 78

Inspect Accuracy

The Auto Surface creates an extremely accurate CAD body in relation to the scan data. Use the Accuracy Analyzer to verify the resulting body.

1. The **Accuracy Analyzer** is located in an unpinned dock on the right side. **Open** the **Accuracy Analyzer** dock.
   - Turn on the radio button for **Deviation for Body**.
2. Change the tolerance on the **color bar** to **0.05mm.** The default tolerance is set to **0.1mm**.
   - On the color bar, **double click** on the green **0.1**.
   - Set the value to **0.05.**
3. Hover the mouse over various areas of the mesh to see the deviations.

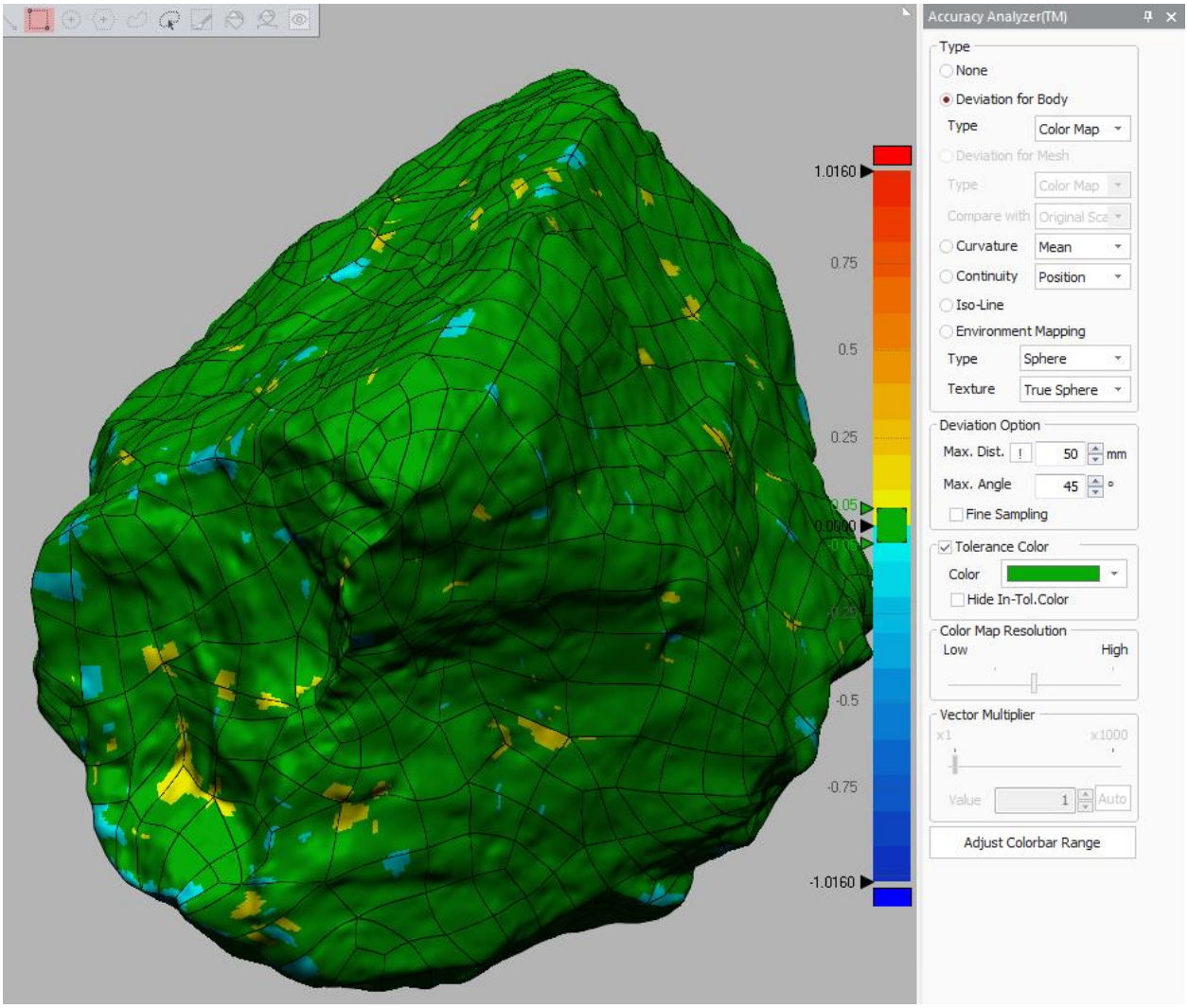


Figure 79

End of Activity

Regions

Regions classify areas of the mesh based on curvature and geometry features. An automatic segmentation of the scan data into regions is one of the first steps in the modeling processes for most standard workflows. Regions can assist with many processes, including:

- Aligning
- Quick Selection
- Wizard Tools

ACTIVITY: Understanding Regions

Objective

Create and edit regions on a mesh to assist in modeling.

Highlighted Commands

- Auto Segmentation
- Merge
- Split

Training File

04 – Regions.xrl

1. Open training file **04 – Regions.xrl**

☞ *No mesh editing will be necessary for the part.*

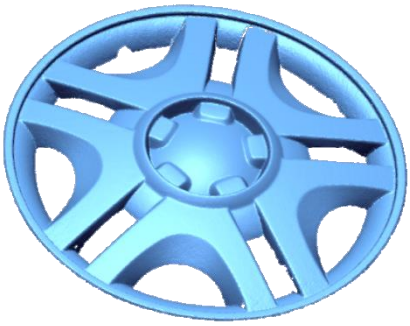


Figure 80

2. In the **REGION** tab, select the **Segment>Auto Segment** command.

☞ *Press the F1 key to read more about the Auto Segment*

- Set the **Sensitivity** to **5.**
- Click the **OK** button.

☞ *The sensitivity option classifies a shape as being round or prismatic. To make smoothly connected areas in a single region, move the slider to the left. To make strictly separated regions, move the slider to the right.*

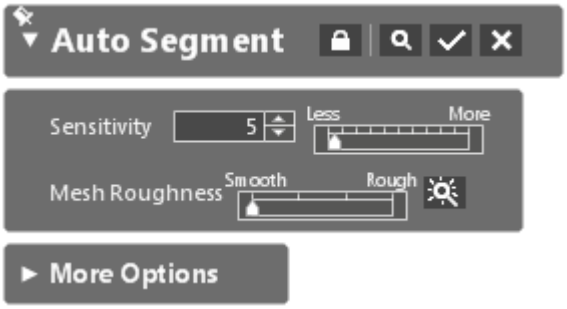


Figure 81

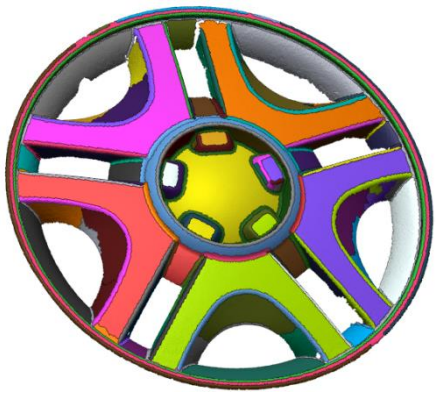


Figure 82

Editing Regions

1. Change the **Selection Type** to **Paint Brush Selection Mode** in the Upper-Side Toolbar.

☞ *To resize the paint brush tool, hold down the Alt key while left-clicking and moving the cursor.*

2. Paint across the regions shown below to select many regions with one click.

3. The **REGION>Edit>Merge** option will now be active. Select the **Merge** to combine the painted regions.

☞ *The Paint Brush can quickly select multiple smaller regions to merge together.*

Figure 83

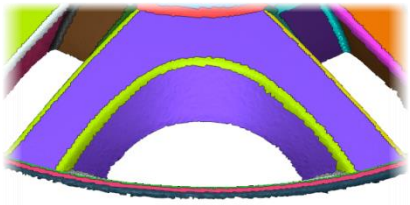


Figure 84

4. Change the Selection Type to the **Rectangular Selection Mode**.

☞ *The Rectangular Selection will allow for single click selections of entire regions.*

5. The **Merge** tool can combine regions that are not physicallly touching.

- Click on the five top **Sphere** regions show in **Figure 85**.
- Select the **Merge** command in the **REGION** tab.

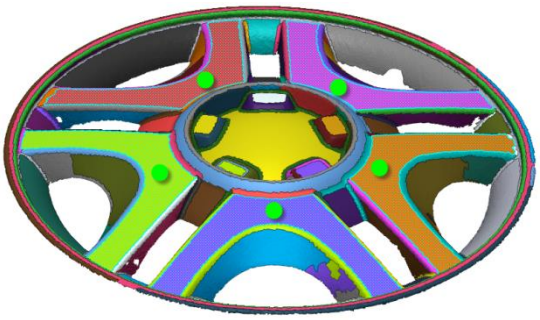


Figure 85

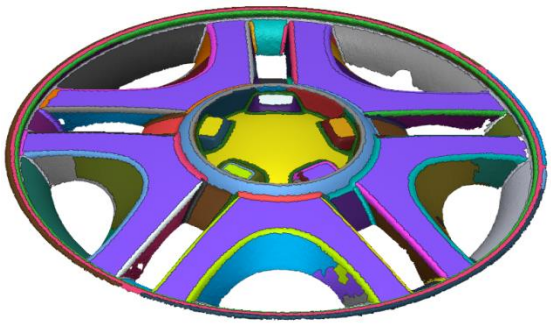


Figure 86

☞ *Non-adjacent, single regions can be separated back apart by selecting the region and then choosing the **Edit>Separate**command in the REGION tab.*

6. Select the **Edit>Split** tool to convert single regions into multiple separated regions.

- After selecting Split, the **Limit to Region** is automatically activated. **Select** the region shown in **Figure 87.**

- Switch the Selection Type to the **Paint Brush** again.
- **Paint** across multiple regions, including the previously selected.
- **Accept** the command.

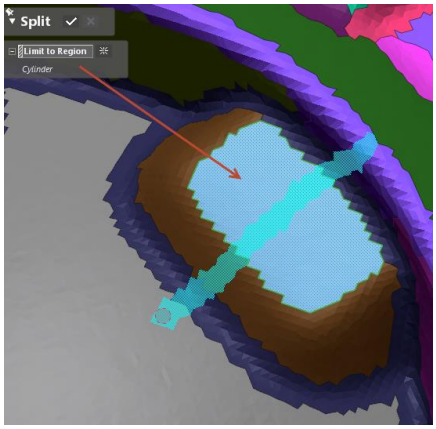


Figure 87

7. Use the **Insert** command to manually create a region.

- Select the **Extend to Similar Selection Mode** in the Upper-Side Toolbar.
- Click on the marked area to select the entire area in a single click.

- With the command now active, select **REGION>Edit>Insert** to create a region independent of the previously merged regions.

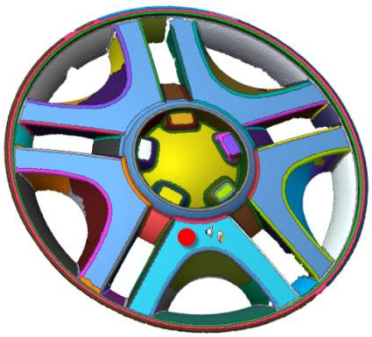


Figure 88

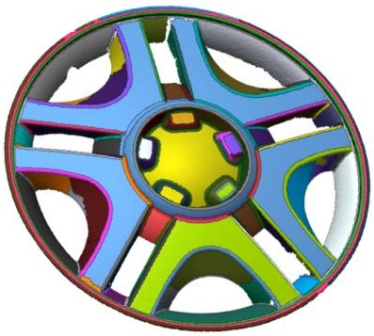


Figure 89

End of Activity

Align to Coordinate System

Scan data should be moved and aligned to a more useful position with the general coordinate system. Geomagic Design X has multiple ways to move

ACTIVITY: 3-2-1 Alignment using Regions

Objective

Learn the basics of aligning scans to the coordinate system using the regions of a mechanical part.

Highlighted Commands

- **Interactive Alignment**

Training File

05a – Align 321.xrl

Scan Setup

1. Open training file **05a – Align 321.xrl.**

☞ *No mesh editing will be necessary for the part.*

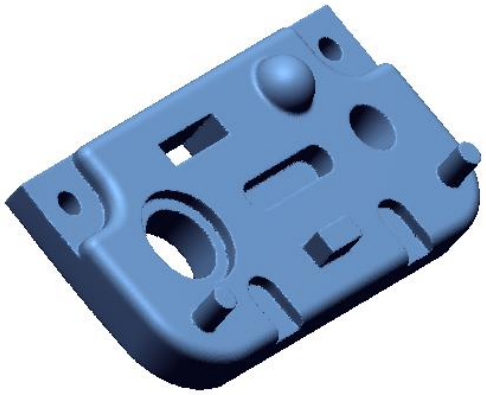


Figure 90

2. In the **REGION** tab, select the **Auto Segment** command.
   - Set the **Sensitivity** to **65.**
   - Click the **OK** button.

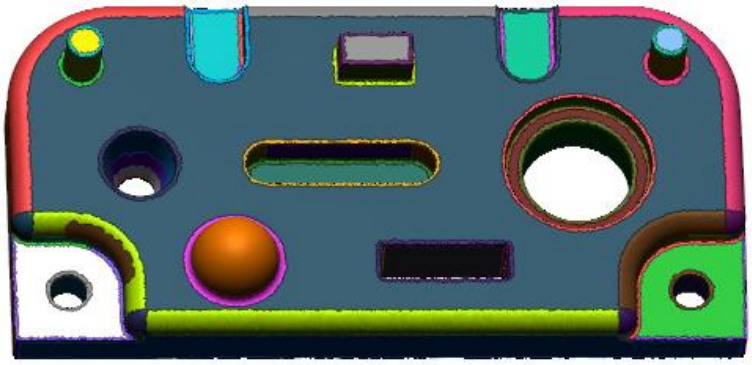


Figure 91

Interactive Alignment

3. Start the **Interactive Alignment** in the **Alignment>Scan to Global** group.
   - Click the **Next Stage** button.
4. Create a **3-2-1** alignment using three planar regions.
   - The **Plane** option will set the **Front Plane**, and therefore the **XY Plane** and the **Z Axis** direction. Choose the large **Plane Region** on the bottom of the part.
   - The **Vector** will set the **Top Plane** and the **Y Axis** direction. Choose the pink long thin **Plane Region** pointed at in **Figure 93**.
   - The **Position** will set the **Right Plane** and the location along the X Axis. Select the **Plane Region** shown in **Figure 94**.

   - Click **OK** to accept.

☞ *The origin will be located at the intersection of the three planes selected above.*

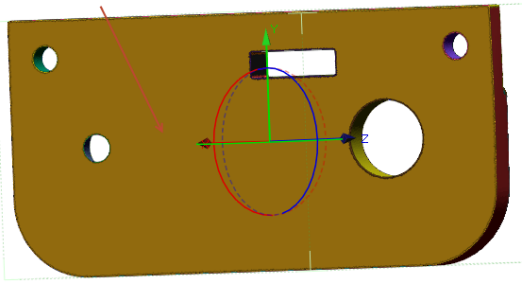


Figure 92

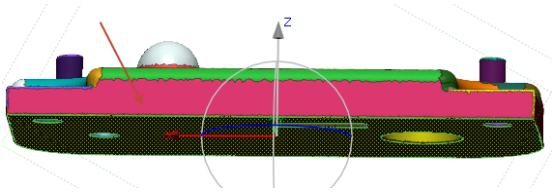


Figure 93

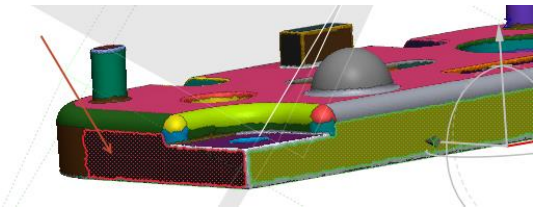


Figure 94

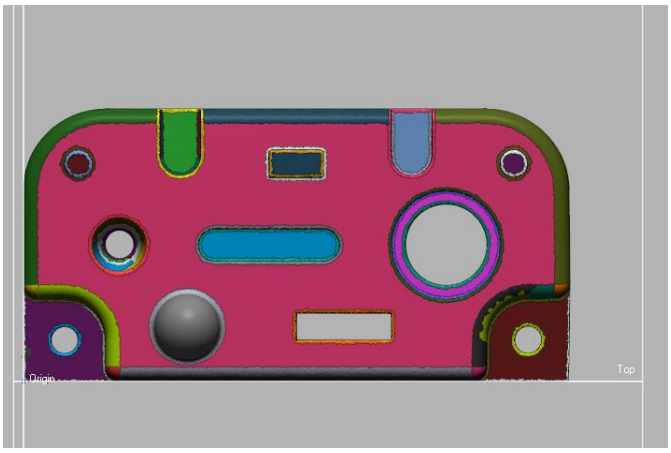


Figure 95

End of Activity

ACTIVITY: 3-2-1 Alignment using Reference Geometry

Objective

Learn how to create other types of reference geometry to use for an Interactive Alignment.

Highlighted Commands

- **Ref. Plane**
- **Interactive Alignment**

Training File

05a – Align 321.xrl

Scan Setup

1. Open training file **05a – Align 321.xrl.**

   ☞ *No mesh editing will be necessary for the part.*

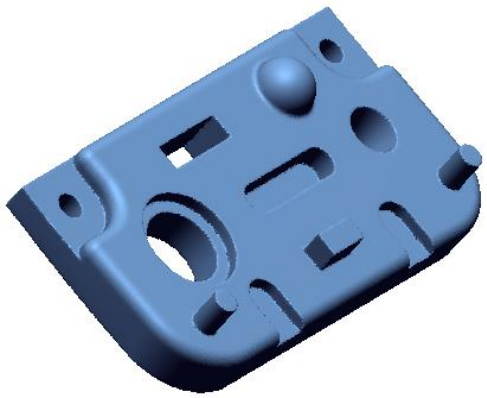


Figure 96

2. In the **REGION** tab, select the **Auto Segment** command.
   - Set the **Sensitivity** to **65.**
   - Click the **OK** button.

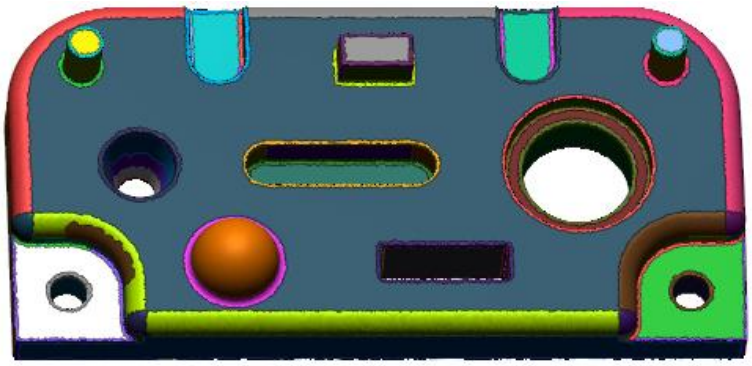
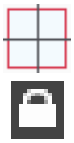


Figure 97

Reference Planes



3. Create a **Ref. Geometry>Plane** down the center of the part two directions.

- Select the **Lock** icon to stay inside the command and make multiple planes.
- Change the **Average**.
- Select two long planes shown to find the mid plane between them.
- Click the **OK** button to accept.

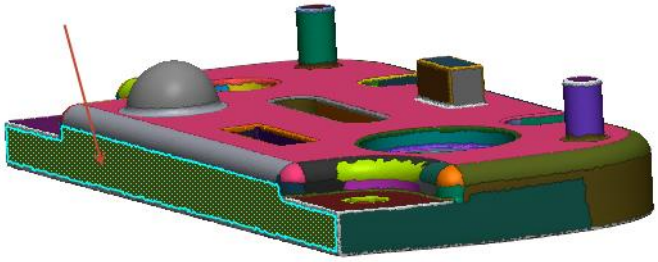


Figure 98

Figure 99

4. Create a symmetry plane going down the center of the part in the other direction.

- Change the **Method** to **Draw Line**.
- Draw a line roughly in the orientation shown in **Figure 100** to create a plane.
- Click the **OK** button to accept.
- Change the **Method** to **Mirror**.
- In the **Model Tree**, hold the **Ctrl** key and select the **Mesh (Align1)** and **Plane2**.
- Click **OK** to accept.

☞ *A plane representing the center of the mesh in the direction of the drawn plane will be found.*

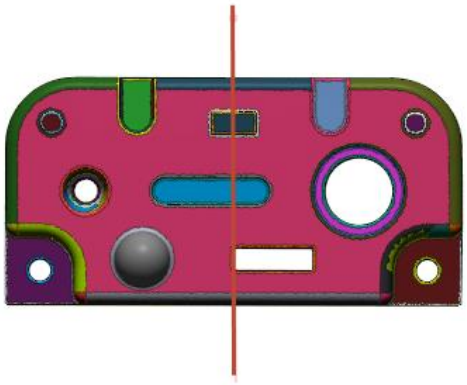


Figure 100

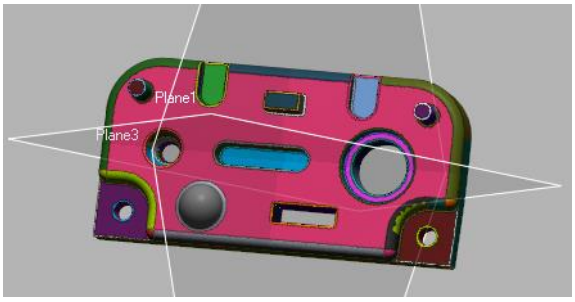


Figure 101

Interactive Alignment

1. Start the **Interactive Alignment** in the **Alignment>Scan to Global** group.
   - Click the **Next Stage** button.
2. Create a **3-2-1** alignment using the generated reference planes and a region plane.
   - The **Plane** option will set the **Front Plane**, and therefore the **XY Plane** and the **Z Axis** direction. Choose the large **Plane Region** on the bottom of the part.
   - The **Vector** will set the **Top Plane**. Choose **Plane1** to set the Y Direction normal to the plane.
   - The **Position** will set the **Right Plane** and the location along the X Axis. Select **Plane3**.
   - Click **OK** to accept.

☞ *The origin will now be set to the middle of the part along the bottom face.*

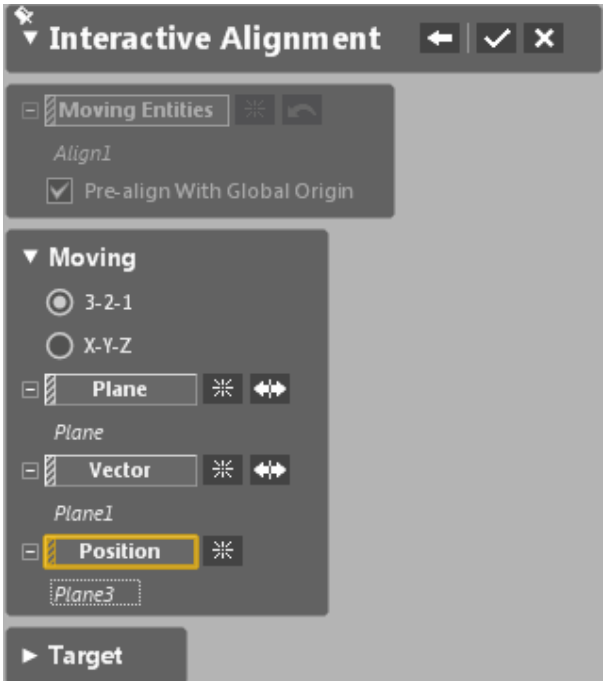


Figure 102

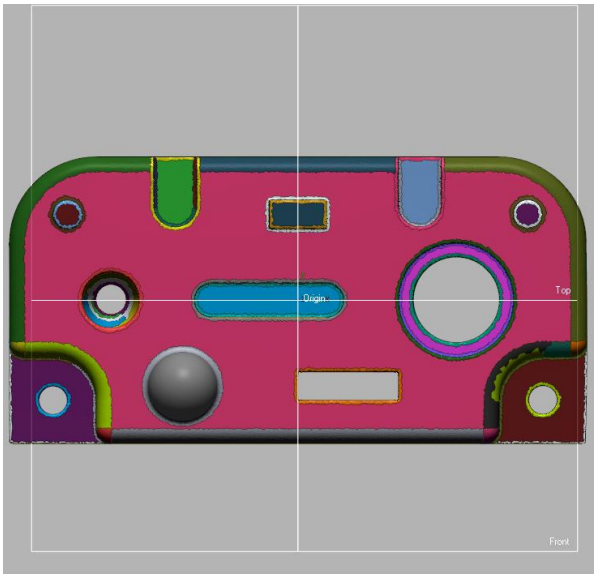


Figure 103

End of Activity

ACTIVITY: XYZ Alignment

Objective

Learn how to create other types of reference geometry to use for an Interactive Alignment.

Highlighted Commands

- **Ref. Vector**
- **Interactive Alignment**

Training File

05b – Align XYZ.xrl

Scan Setup

1. Open training file **05b – Align XYZ.xrl.**

☞ *No mesh editing will be necessary for the part.*

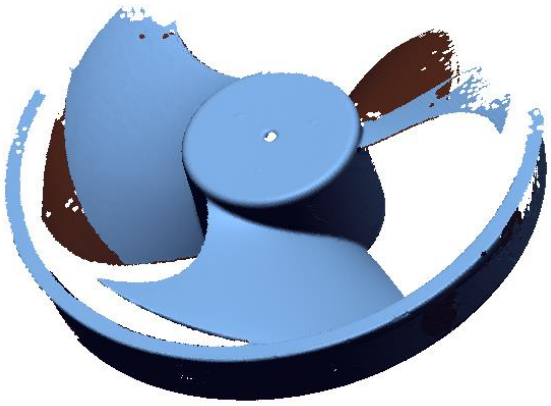


Figure 104

2. In the **REGION** tab, select the **Auto Segment** command.

- Set the **Sensitivity** to **50.**
- Click the **OK** button.

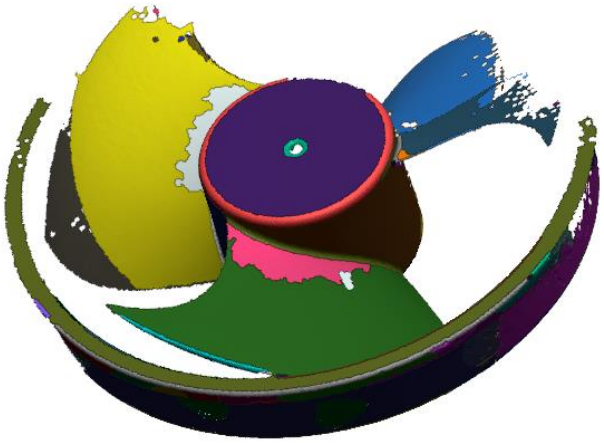


Figure 105

3. Create a **Ref. Geometry>Vector** down the center of hub of the fan.
   - Set the **Method** to **Find Cylinder Axis**.
   - Choose the two large cylinder regions around the central hub.
   - Finish the Vector by right-clicking and selecting the checkmark from the context menu.

☞ *Design X version 2016 and newer has many new right-clicking capabilities.*

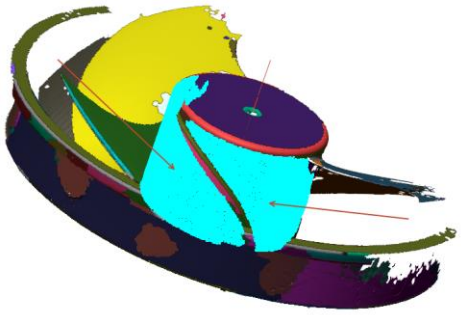


Figure 106

Interactive Alignment

1. Start the **Interactive Alignment** in the **Alignment>Scan to Global** group.
   - Click the **Next Stage** button.
2. Create an **XYZ** alignment using the generated reference planes and a region plane.
   - The **Position** will set the location of the Origin. Select the **Vector1** and Region **Plane** representing the top of the middle hub.

☞ *The intersection of a vector and plane represents a point, which can be used for position.*

   - To set the directions of the coordinate system, select **Axis Z** and choose the **Vector1**.
   - Click the **Flip Direction** arrow to adjust the positive Z in the desired direction.
   - Axis X and Axis Y do not need to be set. The blue circle can be twisted to manually clock the part to any location.
   - Click **OK** to accept.

☞ *The origin will now be set to the middle of the hub of the propeller with the Z Axis going through the middle.*

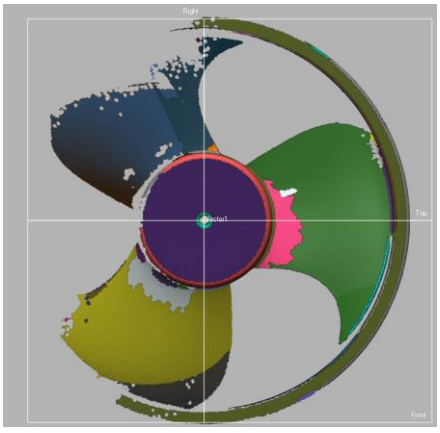


Figure 107

End of Activity

ACTIVITY: Scan to CAD Alignment

Objective

Learn how to import a CAD body and align the scan data to the CAD body.

Highlighted Commands

- **Quick Align**
- **Best Fit Align**

Training File

05a – Align 321.xrl
05 – Align CAD.igs

Scan Setup

1. Open training file **05a – Align 321.xrl**.
2. **Import** the IGES file, **05 – Align CAD.igs,** by selecting **HOME>Open/Import>Import.**

☞ *Importing a file will place it in the same open file.*

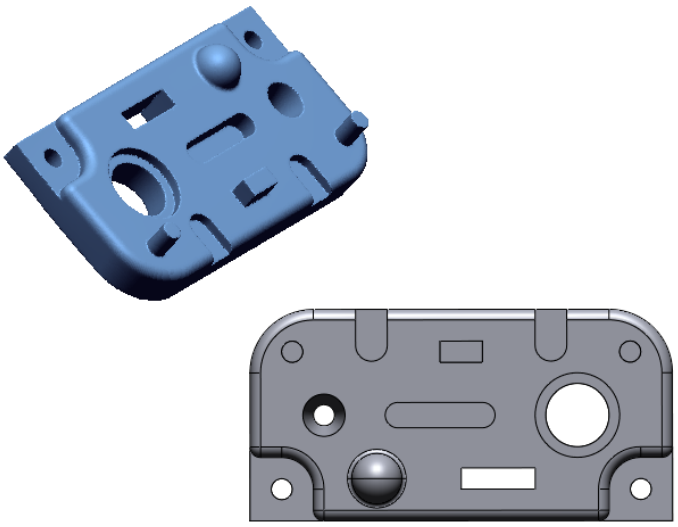


Figure 108

3. Choose the **ALIGNMENT>Scan to CAD>Quick Fit** command to align the scan data to the CAD data.
   - Set the **Target Body** to be the **CAD body.**
   - The Align1 scan should be automatically set as the Moving Entities.
   - Click the **OK** button to accept.

☞ *The Run Best Fit Alignment should be checked under More Options.*

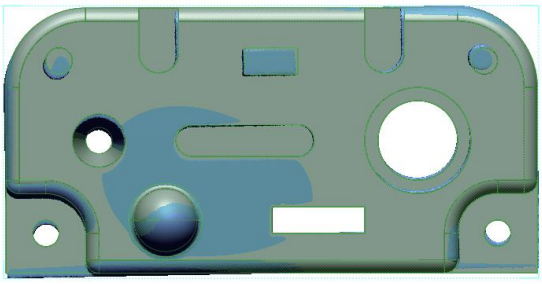


Figure 109

4. The **Best Fit Alignment** will automatically start and have all of the options populated. Press the **OK** key to accept.

☞ *The Best Fit Alignment will always need to be run after a Quick Fit. This will make a much better registration between the files.*

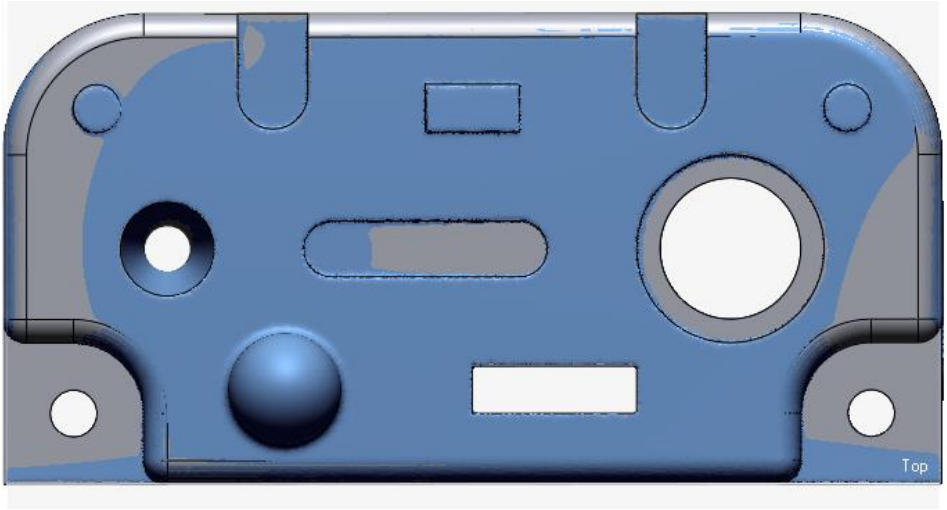


Figure 110

Aligning a scan to an already built CAD file can be used for multiple reasons, including:

- Check the accuracy of the scanned part to the CAD.
- Add features to a Solid Body that were not in the original file.

End of Activity

Parametric Modeling

The strength of Geomagic Design X is in its sketch based modeling. Any model can be created regardless of the scan quality. Design X utilizes sketches and standard CAD operations, such as Extrude, Revolve, Sweep, and Loft, to generate a parametric model.

ACTIVITY: Introduction to Sketching

Objective

Learn how to set up a Sketch and the basic principles of creating sketches. Creating sketch entities, constraining entities, and dimensioning are all introduced in this lesson.

Highlighted Commands

- **Setup Sketch**
- **Rectangle**
- **Parallelogram**
- **Fillet**

Sketch Entities

1. Create a **New File** by selecting the icon on the top-most toolbar.

2. Click **SKETCH>Setup>Sketch** to begin a 2D sketch.
   - Select the **Front Plane** from the **Model View**.

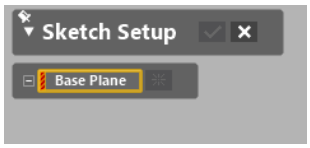


Figure 111

☞ *The sketch will start immediately after selecting the base plane.*

☞ *If a plane is already selected, choosing **Sketch** would directly start a sketch on that plane.*

☞ *The Sketch button has now changed to Exit in order to complete the sketch.*

3. Create a **Rectangle** using the **SKETCH>Draw>Rectangle** tool.
   - Left-click in the top-left corner of the origin, as seen in **Figure 113**, to begin the Rectangle.
   - Left-click the bottom-right corner to finish the Rectangle.

   - Finish the Rectangle by right-clicking in the Model View and selecting the **Checkmark** from the context menu.

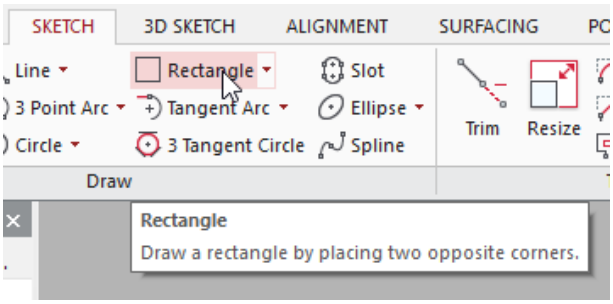


Figure 112

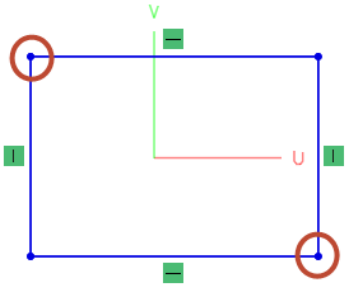


Figure 113

4. Drag a **Line** or a **Point** of the Rectangle to modify its size until it is roughly centered at the Sketch Origin (red U and green V axis).

☞ *By dragging a Line of the Rectangle you can alter one dimension (Length or Width of the Rectangle) at a time, while dragging a Point would let you alter both dimensions at the same time.*

Constraints

5. Double-click on the top-left **Point** of the Rectangle. The **Constraint** and information for that single point is now shown.

☞ *A Coincidence Constraint is applied on the end point of the two lines forming the corner.*

☞ *To see Constraints of another entity, only single left-click on the entity without leaving the Properties Dialog.*

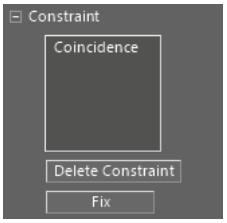


Figure 114

6. Show and add **Common Constraints** of multiple sketch entities.
   - **LMB Click** on the bottom line of the Rectangle to view the Constraints.

☞ *A Horizontal Constraint should be shown.*

   - **Shift+LMB Click** on the other **horizontal** line of the Rectangle (**Figure 116**).

☞ *Both lines will be selected to show the **Common** Constraint between all selected entities.*

   - Select the Independent Constraint **Horizontal** and click the button **Delete Constraint** to remove.
   - Select **Parallel** to set the **Common Constraint**.
   - Repeat with Vertical lines.

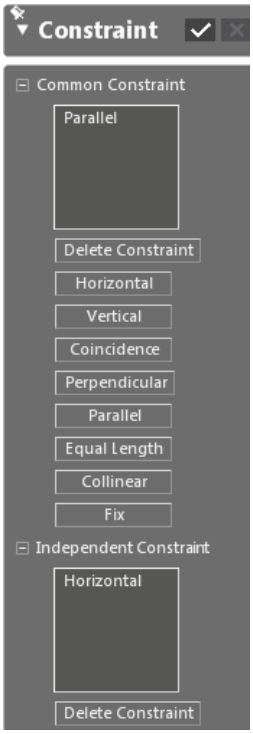


Figure 115

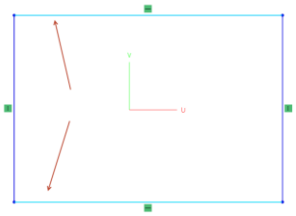


Figure 116

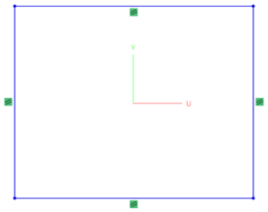


Figure 117

- Alternatively, a Parallelogram could have been chosen from the beginning by clicking the small triangle next to the “Rectangle” tool in the SKETCH Ribbon and selecting the “Parallelogram” in the tools list to produce the same result.
- The Ribbon always remembers the last method of a tool button, for quicker repeatability.

7. Apply a **Perpendicular** Constraint to the lower left corner of the Rectangle.

- Close the Constraints Dialog Tree with a RMB click check or the dialog checkmark.
- LMB click anywhere in the Model View background to deselect all entities.

☞ *After most operations, selections will persist after the dialog is closed.*

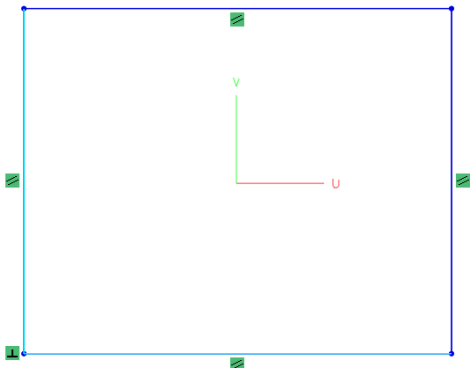


Figure 118

8. Create a pivot point for rotating the sketch by constructing two intersecting Centerlines with the **SKETCH>Draw>Centerline** tool below the Line tool.

☞ *The Line tool could have been used with the option For Construction selected.*

- Move the cursor over the middle of the left line, and LMB click at the **Mid-Point** Constraint to start the Centerline (**Figure 119**).

☞ *After clicking to make the point, move the cursor back over the left line to activate the Parallel and Perpendicular assist axes.*

- LMB click to place the Centerline when the yellow **Perpendicular** and white **Vertical** symbol are visible.

☞ *Creating the centerline perpendicular to the base will keep it related ta rotatable rectangle.*

- RMB click to finish the line.
- Move the cursor over the middle of the bottom line, and LMB click at the **Mid-Point** Constraint to start the Centerline.
- Finish the line at the end point of the first centerline, Perpendicular to the Mid-Point.

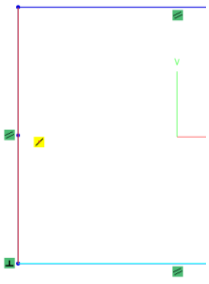


Figure 119

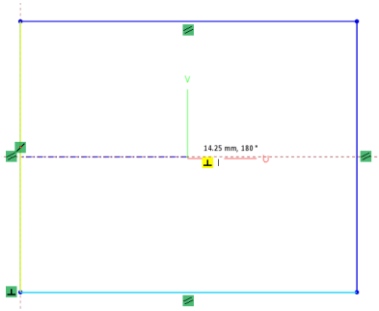


Figure 120

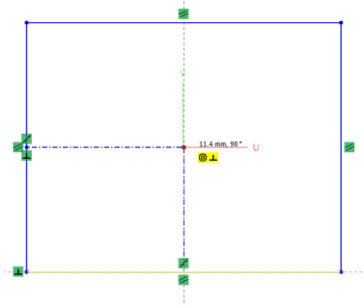


Figure 121

9. LMB Click and **Hold** on the intersection point of the two Centerlines and drag it onto the intersection of the red and green Sketch U/V-Axis to place the **Coincident** Constraint.

☞ *The center point should turn black signifying it is fully defined.*

☞ *Click and drag a mid-point of the rectangle lines and the rectangle will rotate and resize.*

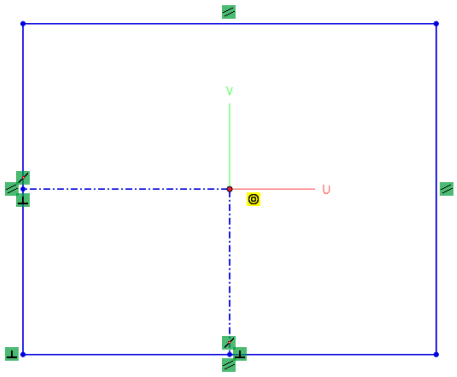


Figure 122

Dimensioning

10. Select the **Smart Dimension** tool from the **SKETCH>Draw** ribbon to apply the Linear and Angular dimensions as shown.

- LMB Click one line and then select the parallel entity. LMB Click in space to set the linear dimension.
- Repeat for the other set of parallel lines.
- Select the inferred horizontal horizon line and one of the created Centerlines. LMB Click in space to set the angular dimension.

☞ *Sketch entities will turn black when they are fully constrained by dimensions and constraints.*

- Click **OK** to accept the command.

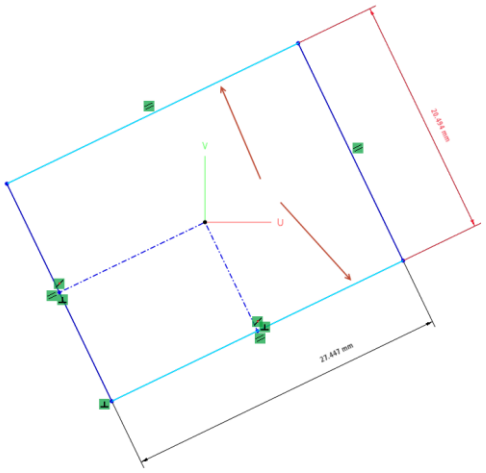


Figure 123

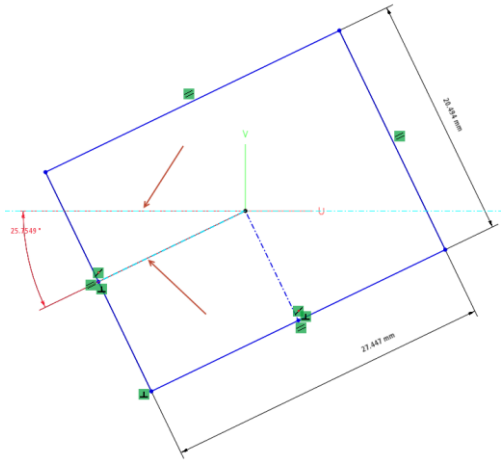


Figure 124

11. Edit the Dimensions by double-clicking on each value.

- Set the two linear dimensions to size the box.
- Change the angular dimension to rotate the box to a specific angle from the horizon.

12. Click the **Exit** button to finish the sketch.

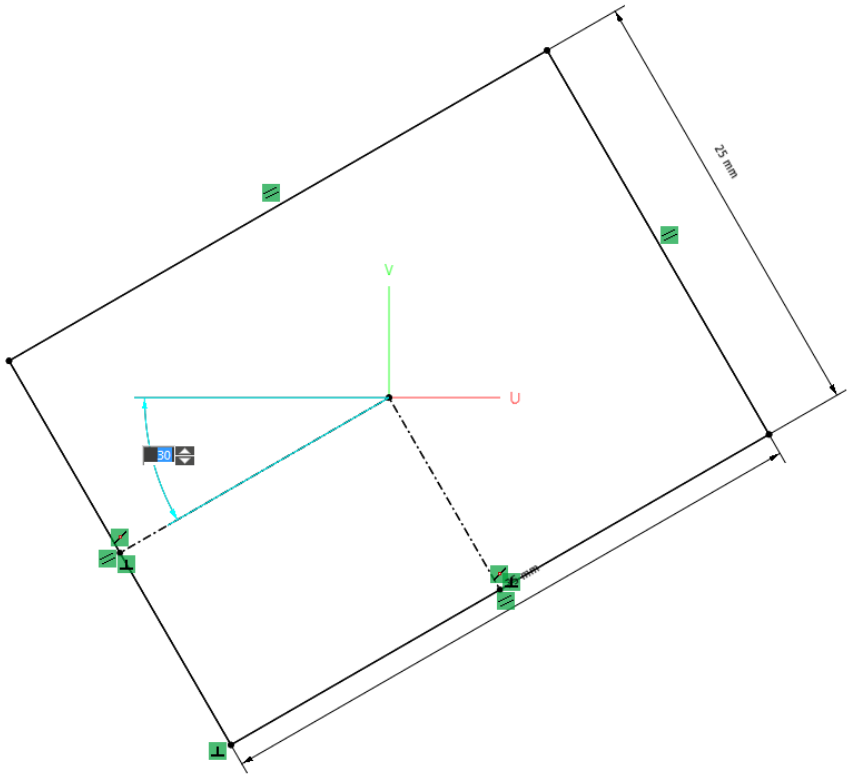


Figure 125

- This sketch can be easily controlled and updated for future uses in CAD.

End of Activity

ACTIVITY: Basic Concept

Objective

Learn the basics of sketching and extruding to create and cut solid bodies.

Highlighted Commands

- **Mesh Sketch**
- **Extrude**
- **Fillet**

Training File

07 – Basic Concept.stl

Scan Setup

1. **Import** training file **07 – Basic Concept.stl**.
   - Click the **Import** button on the Quick Access Toolbar.
   - Or choose MENU>Insert>Import from the dropdown.
   - Navigate to the file then choose **Import Only**.

☞ *Design X requires importing files that are not native Design X file types.*

☞ *No mesh editing will be necessary for the part.*

Figure 126

2. In the **REGION** tab, select the **Auto Segment** command.
   - Set the **Sensitivity** to **50.**
   - Click the **OK** checkmark to accept.

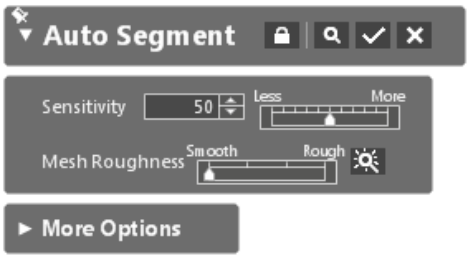


Figure 127

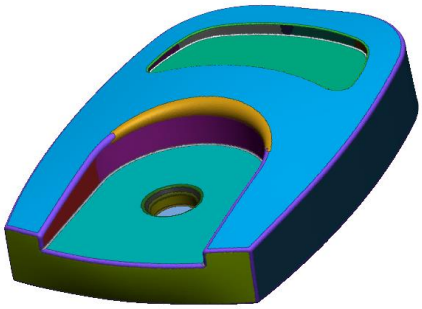


Figure 128

Interactive Alignment

1. Create a **Plane** through the middle of the part to use for alignment using the **MODEL>Ref. Geometry>Plane** command.
   - Change the **Method** to **Mirror**.
   - In the **Feature Tree**, hold the **Ctrl** key and select the **Right** plane and **Basic Concept** mesh.
   - Click **OK**.

☞ *The Mirror method will read the selected mesh and find the symmetry plane of the scan based on the given direction of the selected plane.*

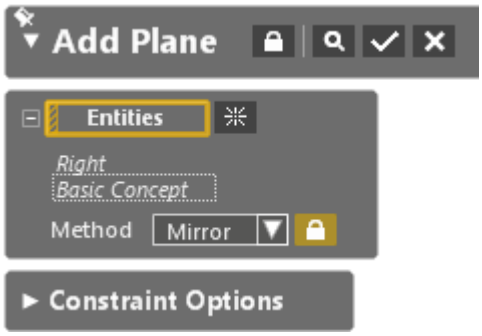


Figure 129

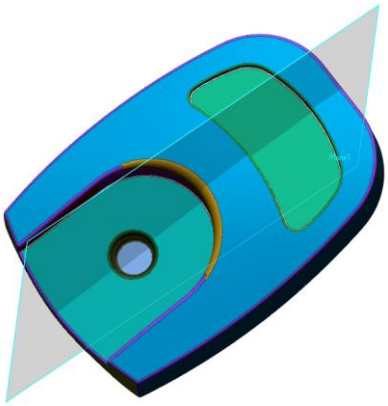


Figure 130

2. Begin the alignment by selecting **Interactive Alignment** from **ALIGNMENT>Scan to Global**.
   - Go to the **Next Stage**.
3. Change the **Moving** option to **X-Y-Z**.
   - Set the **Position** as the region **Plane** on the bottom of the part and the region **Cylinder** of the central hole.

☞ *The Origin will now be set to the point where the cylinder axis would intersect the plane.*

Figure 131

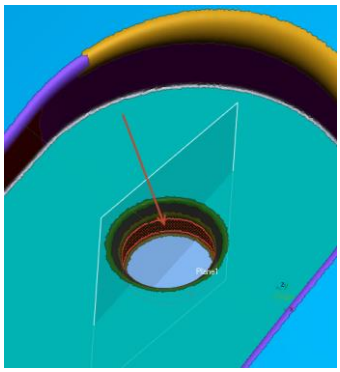


Figure 132

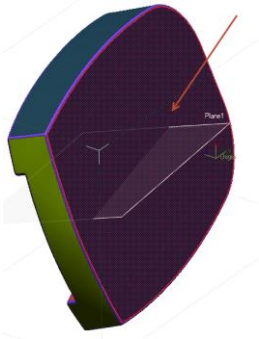


Figure 133

- Click on **Axis Z**, and then select the same region **Plane** as before to set the Z direction.
- Select the **Flip** button next to **Axis Z** to point the Z Axis through the part.
- Click on **Axis X,** and then select **Plane1**.

4. Click **OK** to accept the alignment based around the central hole.

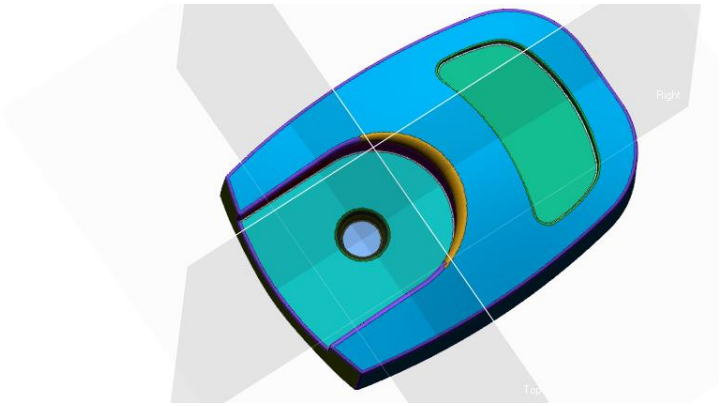


Figure 134

Create Main Body

1. In the **SKETCH** tab, choose **Setup>Mesh Sketch**.

- Select the **Front** plane as the **Base Plane**.
- Set the **Offset Distance From Base Plane** to **5mm**.
- Click the **OK** button to begin sketching.

☞ *The Offset Distance From Base Plane is only to set a cross-section distance of the model to be projected back to the originally selected Base Plane. All sketches will still be created on the Base Plane, regardless of the height of the Offset Distance.*

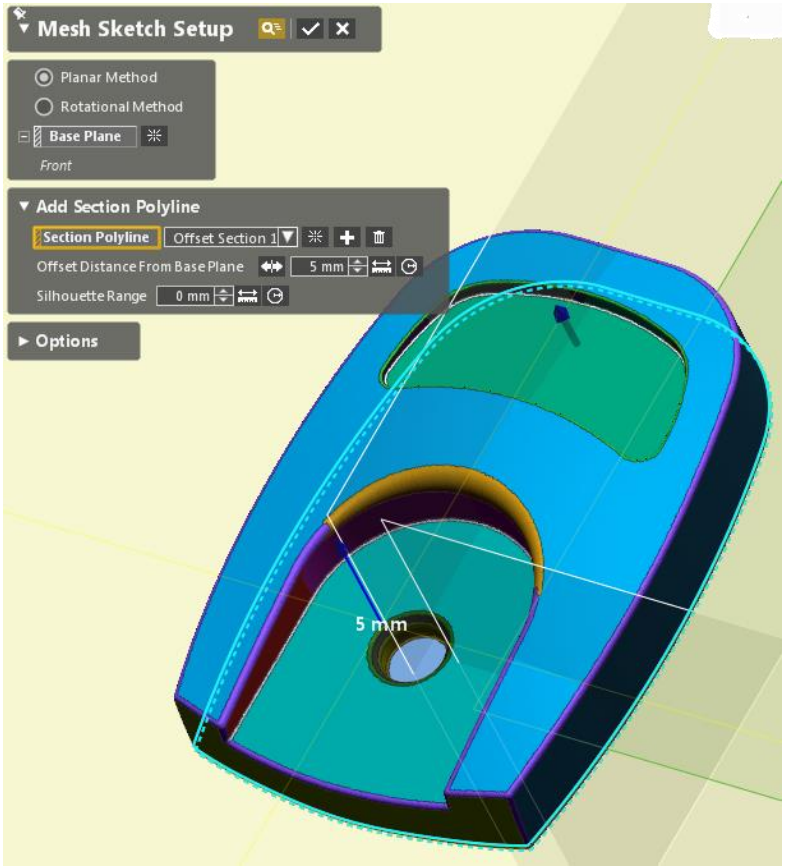


Figure 135

2. Use the **Model Tree** to **Hide** the Mesh.
3. Select the **Draw>Line** tool.
   - Click on the top-most solid pink line. A line will be best fit to the entire segment.
   - Click the **OK** button to accept the line.

4. Select the **Draw>CenterPoint Arc** command.
   - Click on the left-most solid pink line. An arc will be best fit to the entire segment.
   - Click on the **Accept Fitting** check.
   - **Double click** the right-most solid pink line. This will create and accept the arc while still staying inside the command.
5. Use the **Centerpoint Arc** to create the lower, segmented arc.
   - Single click on the five segments to create one best fit arc between the segments.
   - Double click anywhere in space to accept the fitting.
   - Click the **OK** button to accept.

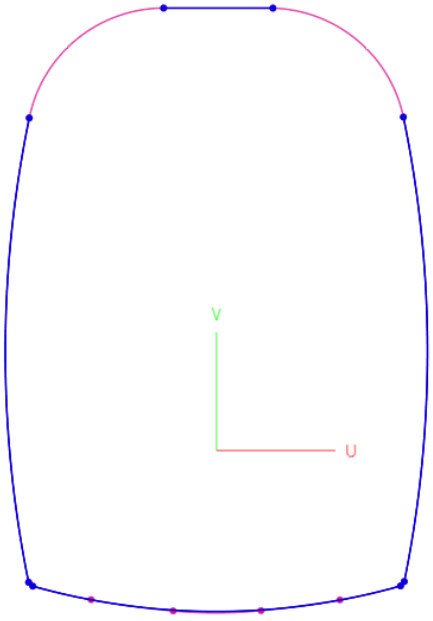


Figure 136

6. Open the **Accuracy Analyzer** on the right side toolbar to display all points of an open loop.
   - Place a checkmark next to the **Disjoined Ends** box.

☞ *Disjoined Ends will add a green end point to any point of opening in a sketch chain.*

7. Using commands from the **Tools** group allows for quick ways to close open sketch loops. Select the **Tools>Trim** command to close endpoints to a sharp corner.
   - Change the option the **Corner Trim**.

- Locate the two open ends shown in the picture below.
- Zoom in to one corner. Left click on one line segment surrounding one corner.
- Select the other line segment to join the two ends.

☞ *This command uses the sketch segments, not end points. The Trim will be directed based which side of the segment is selected.*

- Repeat on the other corner.
- Click the **OK** button to accept.

☞ *If the **OK** is greyed out, clicking the **Cancel** button will keep the Trim and exit the command.*

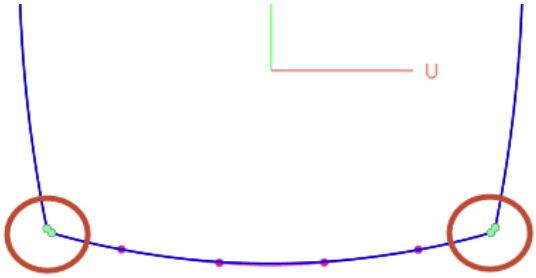


Figure 137

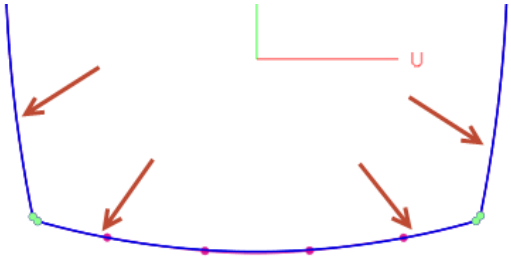


Figure 138

8. Use the **Tools>Fillet** command to close the other two open points with smooth, tangent arcs.

- Left click on one sketch segment near the missing arc segment.
- Click and **HOLD** the LMB on the second segment.
- While still holding the LMB, move the mouse around to adjust the size of the fillet.
- Release the mouse button when the arc is best fitting the pink polyline.
- Repeat to create the second arc.
- Click **OK** to accept.

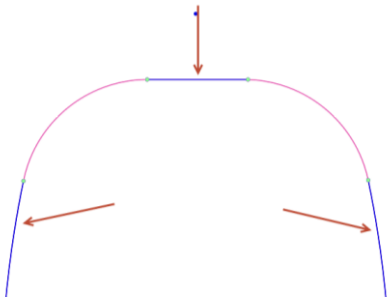


Figure 139

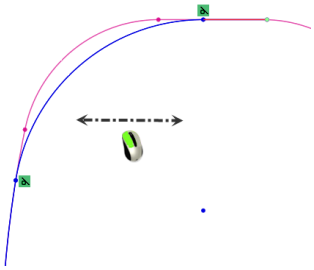


Figure 140

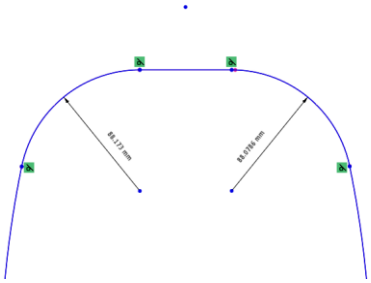


Figure 141

9. Double click on a dimension to edit the value.

- Using a design intent approach, change each dimension to **88 mm**.

10. Verify there are no openings in the sketch loop, then press the **Exit** button in the top toolbar.

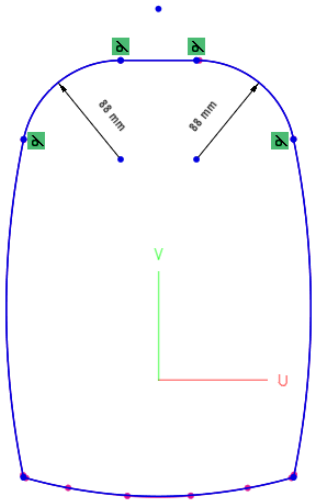


Figure 142

11. In the **MODEL** tab, select the **Create Solid>Extrude** command to generate the body.

- Turn on the **Mesh** and **Region** visibility by pressing **Ctrl+2**.
- The **Base Sketch** should automatically be selected as **Sketch1 (Mesh)**.
- Change the **Method** to **Up to Region**. The Sub Method should automatically set to **Trim with Region Fit Surface**.
- For the **Up To** option, select the top, Freeform region.
- Click the **OK** to accept.

☞ *The Up to Region>Trim with Region Fit Surface method will automatically fit the extrusion to the shape of the region.*

☞ *Press F1 to see all the different methods of Extruding.*

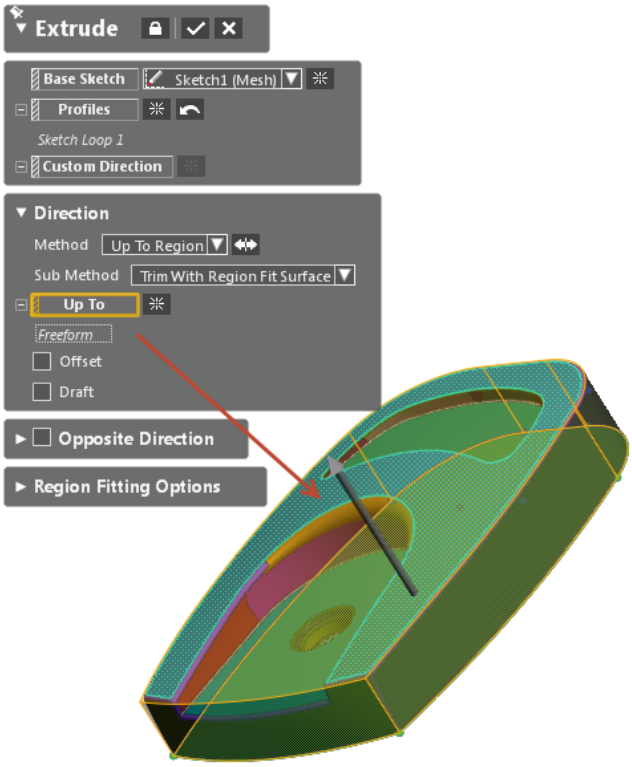


Figure 143

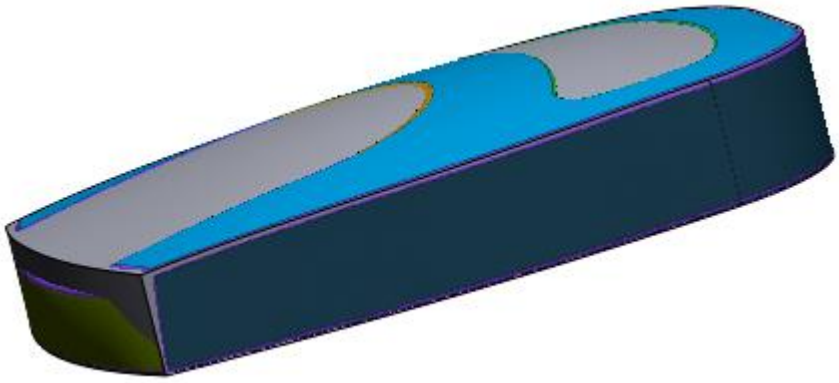


Figure 144

12. Turn on **Deviation for Body** inside of the **Accuracy Analyzer** dock.

- Rotate the body to inspect the accuracy of the model.
- The **Bottom View** will display the area of high deviation. This face shows the solid body is both too small in areas and too large.
- Select **None** to turn the deviations off.

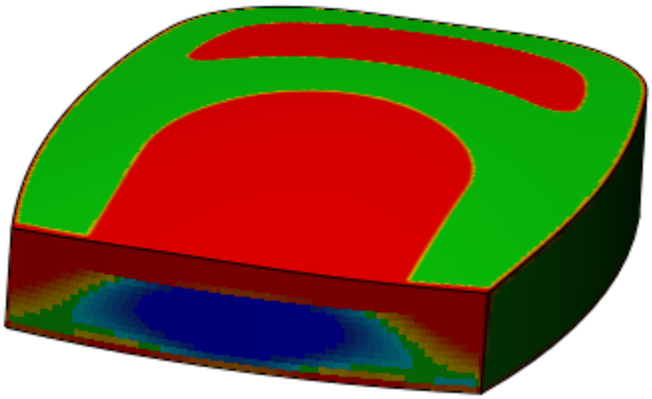


Figure 145

13. To fix the inaccuracies of the model, **Edit** the base sketch.

- **RMB** on **Sketch1(Mesh)** in the **Feature Tree**.
- Select **Edit** from the dropdown menu.

14. Press the **Normal To** button on the Upper-Side Toolbar, or **Ctrl+Shift+A**. This will align the sketch back normal to the screen.

15. Select the lower sketch arc and press **Delete**.

16. Using the **Draw>Line** command, reconnect the loop by clicking on one endpoint and then the other.

- Click the **OK** to accept.

☞ *A yellow and black concentric circle symbol will appear when selected on the end point.*

17. Click and HOLD the LMB on the new line and drag it below the pink polyline.

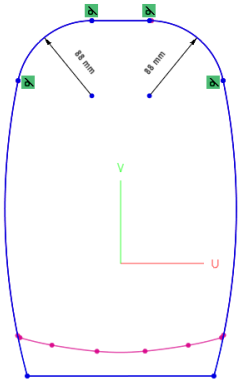


Figure 146

18. Click the **Exit** button to exit and accept.

19. Make sure the **Regions** are still visible, but **HIDE** the Solid Bodies using the Model Tree, or by pressing **Ctrl+5**.

20. Create an accurate, form-fitting surface along the back face of the mesh with the **MODEL>Wizard>Mesh Fit** command.

- Select the Revolution region that represents the inaccurate face.
- With the **Resolution** set to **By Allowable Deviation**, limit the **Max. No. of Control Points** to **20.**
- Click **OK** to accept the Mesh Fit surface.

☞ *A Mesh Fit surface will be created accurately to any mesh surface. This surface is not directly editable since it is not based off of a sketch. If changing the Mesh Fit parameters does not create the desired surface, the mesh itself can be optimized and editing to make a better fitting surface.*

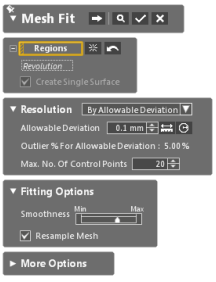


Figure 147

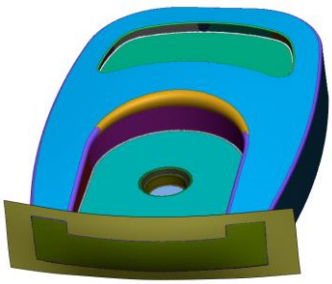


Figure 148

21. Turn back on the Solid Body visibility.

☞ *This surface will be used to cut off the excess solid body.*

22. Select the **MODEL>Edit>Cut** command to cut a solid body using a surface body.

- Select the **Mesh Fit1** surface as the **Tool Body**.
- Select the **Extrude1** solid as the **Target Body**.
- Click on the **Next Stage** arrow. Notice the OK check is not available until the next stage is completed.
- Click on the main portion of the Solid Body to select as the **Target Body**. This is the portion that will be kept.
- Select the **OK** check to accept the cut.

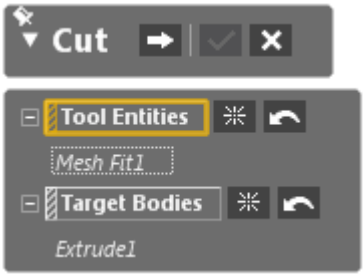


Figure 149

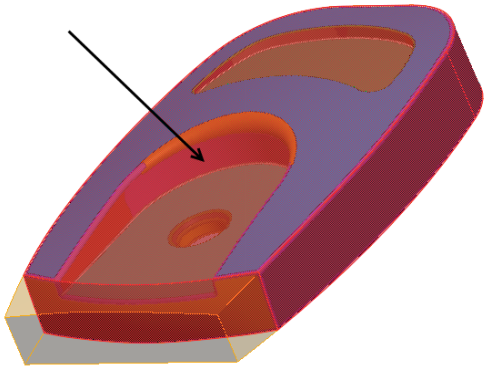


Figure 150

23. Turn the **Deviation for Body** back on in the **Accuracy Analyzer** dock to see the new resulting body.

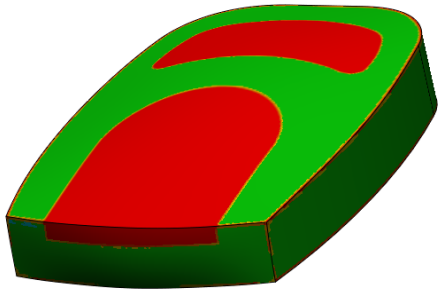


Figure 151

☞ *It is recommended to turn the deviations back off after viewing. This will speed up processing time for all proceeding steps instead of making the software rerun the analyzer.*

Extrude Cut

1. Turn on and off visbility of entities until only the **Mesh** and **Regions** are visible.

2. **Right Click** on the **Plane** region shown below.

- Select the **Mesh Sketch** icon to immediately enter the Mesh Sketch Setup up with the region as the Base Plane.

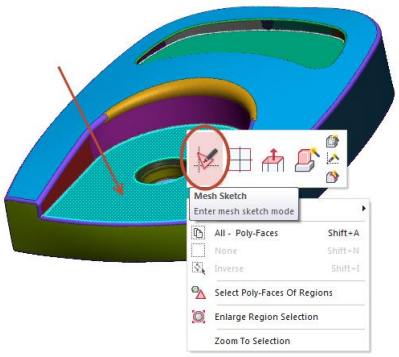


Figure 152

- Set the **Offset Distance From Base Plane** to **2 mm**, or drag the thin, blue arrow up until the outline of the cutout shown is visible.
- Place a Check next to **Draft Angle**.

- Select the **Measure Angle** button then click on the two planar regions shown in **Figure 154**.
- Set the **Draft Angle** to **1°**.

☞ *The total angle will be displayed. This will show if there is any draft from the base plane. The Mesh Sketch Setup can compensate for the draft when projecting the poly-lines back to the base plane.*

- Click the **OK** check to begin sketching.

☞ *Hiding the Mesh can make it easier to see the underlying polylines.*

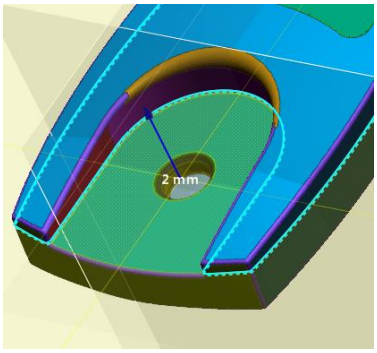


Figure 153

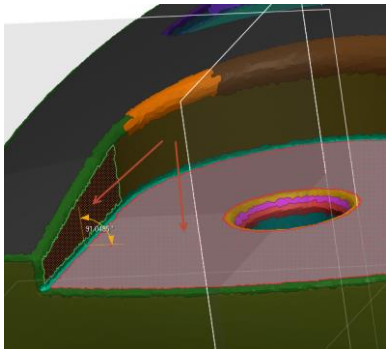


Figure 154

3. Sketch the inner cutout using the **Line** and **3 Point Arc** tools.

- **RMB Click** in the Model View. An array of common sketch tools will immediately appear.

- Select the **Line** icon and sketch the two vertical lines by double clicking the LMB on each polyline.
- Draw a **Line** between the bottom end points to close the loop.
- **RMB Click** and select the **Check** to accept the line command.

- **RMB Click**. Select the **3 Point Arc** icon. Click on both end points and then the height of the pink polyline radius.
- **RMB Click** and select the **Check** to finish the command.

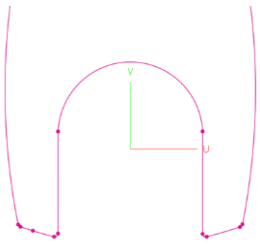


Figure 155

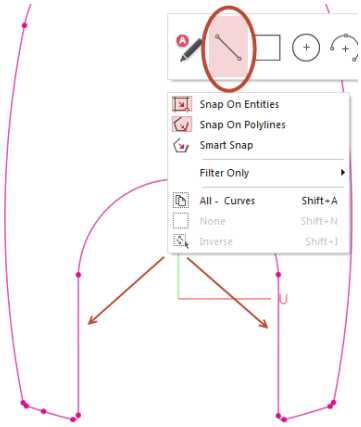


Figure 156

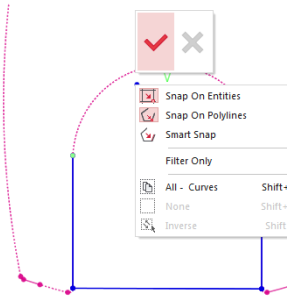


Figure 157

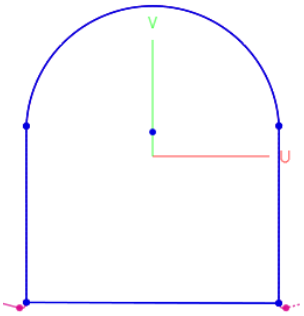


Figure 158

4. Edit and adjust the sketch for a better overall model.
   - **Double Click** on one of the vertical sketch lines.
   - A dialog box will appear with the editing and constraint options. Select **Vertical** from the **Constraints** box.
   - Hold **Shift** and select the other vertical line. Select **Parallel** from the **Common Constraint** options.
   - **Double Click** on the arc. Hold **Shift** and select one of the vertical lines. Select **Tangent** from the **Common Constraint** options. Repeat the process on the other line to create tangency with the arc.
   - **Adjust** the sketch entities until the sketch looks like **Figure 160.**
   - **RMB Click** and select the **Exit** button.

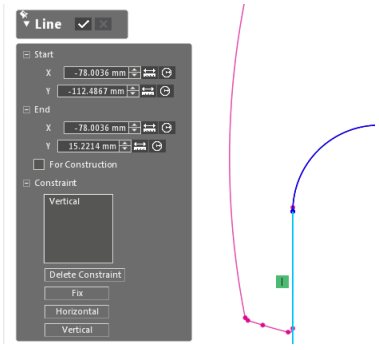


Figure 159

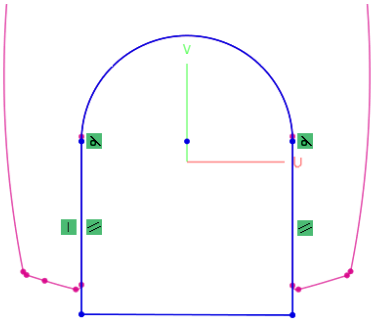


Figure 160

5. **RMB Click** in space on the Model View and select **Solid Extrude**.

- Turn on the **Mesh** and **Solid Body** visibility.
- Check **Sketch2 (Mesh)** is selected as the **Base Sketch**.
- Change the **Method** to **Through All**.
- Place a Check next to **Draft** and input the **1°** measured earlier.

☞ *Check the direction of the draft is correct. Use the Flip Direction arrow next to the Draft input.*

- Set the **Result Operator** options to **Cut.**
- Click the **OK** check to accept.

☞ *The sketch must still be highlighted, and the RMB click must not be on a plane to give the Extrude options.*

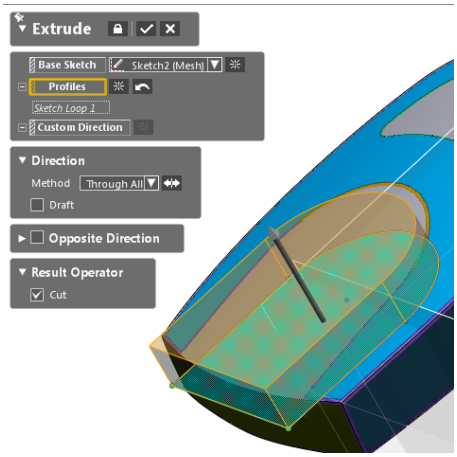


Figure 161

6. Hide the Solid Bodies, then **RMB Click** on the plane shown and start a **Mesh Sketch**.

- Set the **Offset Distance From Base Plane** to **2 mm**.
- Click the **OK** check to accept.

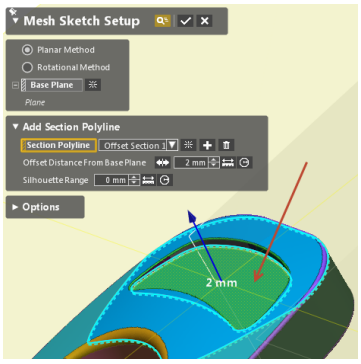


Figure 162

7. **RMB Click** on the Model View and select the **Auto Sketch** command.

- With the option set to **Make Selected Only**, select only the inner loop sketch.

- Click the **OK** check to accept.
- **RMB Click** and select the **Exit** button to finish the sketch.

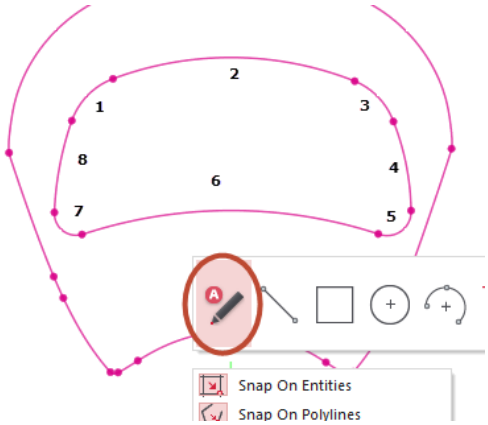


Figure 163

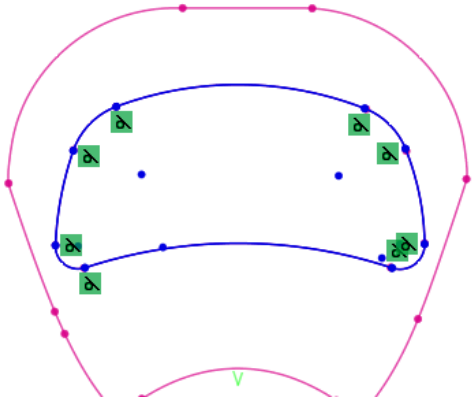


Figure 164

8. **RMB Click** in space on the Model View and select **Solid Extrude**.
   - Turn on the **Mesh** and **Solid Body** visibility.
   - Check **Sketch3(Mesh)** is selected as the **Base Sketch**.
   - Change the **Method** to **Through All**.
   - Set the **Result Operator** options to **Cut.**
   - Click the **OK** check to accept.

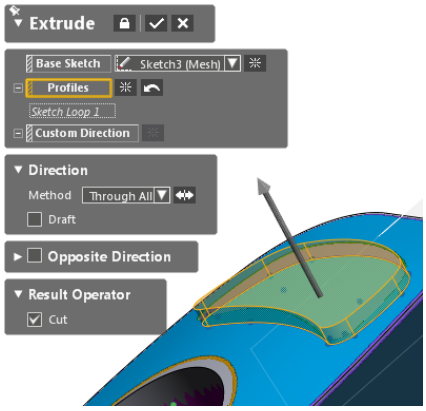


Figure 165

Revolve Cut

1. Create a **Vector** in the **MODEL>Ref. Geometry** group.
   - Select the **Right** and the **Top Planes**.
   - If not done automatically, set the **Method** to **Intersect 2 Planes**.
   - Click the **OK** to finish.

2. **Right-Click** on the **Right Plane** and select the **Mesh Sketch** icon to start sketching.
   - The sketch setup will already be located in the correct location. Click the **OK** to accept and begin sketching.

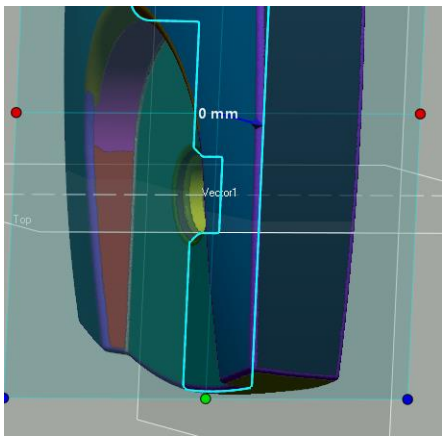


Figure 166

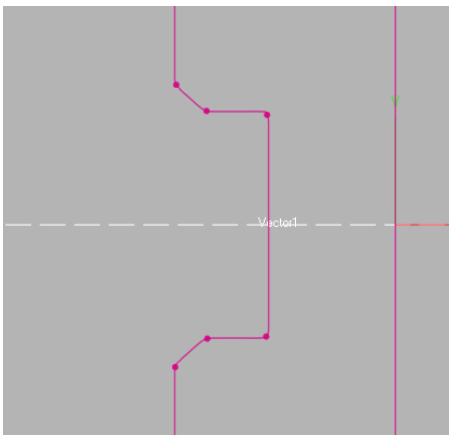
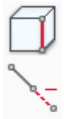


Figure 167

3. Use the **Model Tree** to **Hide** the **Mesh** and **Solid Bodies**.
4. Select the **Draw>Line** tool.
   - Use a box select method to create the 3 lines from the polylines.
   - Draw a vertical line starting from the vector to create a complete sketch.
   - Right-Click in space to select the **Check** and accept the command.



5. Right-Click on the Vector and select **Convert Entities**.
6. Select the **Draw>Trim** command.
   - Check it is set to **Corner Trim**.
   - Click on all of the lines surrounding each open connection to close.
   - Right-Click in space to select the **Check** and accept the command.

7. Set constraints and dimensions then **Right-Click** in space and **Exit** the sketch.

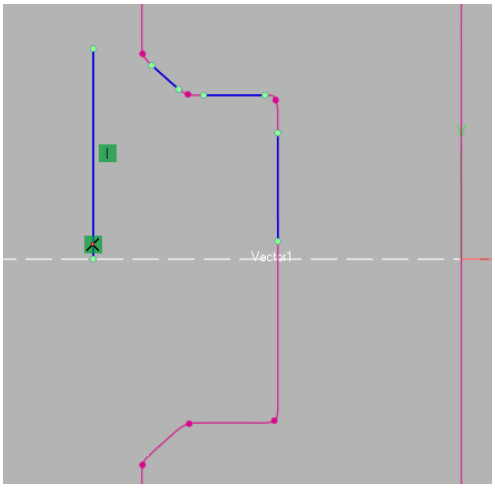


Figure 168

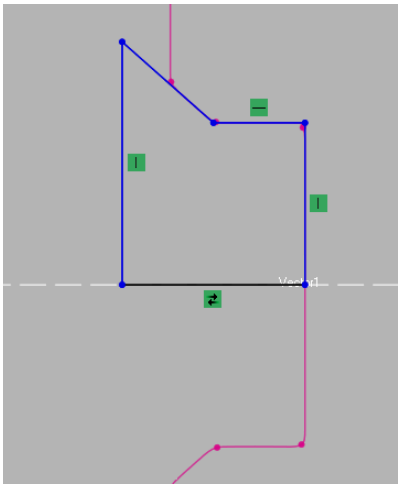


Figure 169

☞ *Only half of the figure needs to be sketched for the **Revolve** command.*

8. Choose the **MODEL>Create Solid>Revolve** command to cut out the section.

- With the previous sketch populating the **Base Sketch** and **Profiles**, chose **Axis** and select **Vector1**.
- Under Result Operator, select **Cut.**
- Click the **OK** to cut out the area.

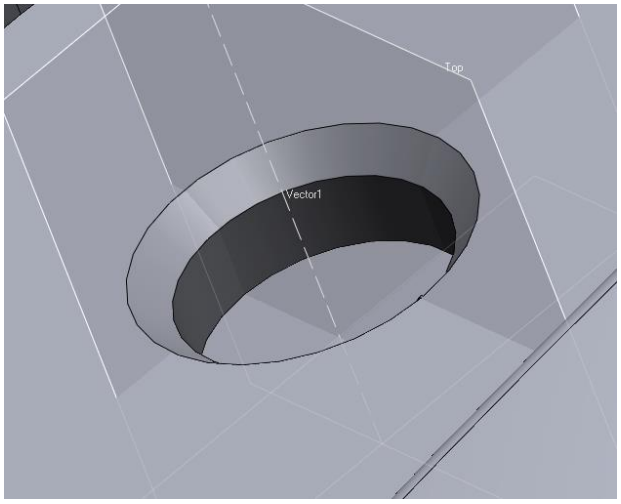


Figure 170

Create Fillets

1. Choose the **Fillet** command in the **MODEL>Edit** group.

- Create a **Constant Fillet** by selecting the edge shown in **Figure 171.**

☞ *The Tangent Propagation option will select all edges of the solid tangent to the first choice, reducing the need to manually select all edges.*

- Next to the Radius box, press the **Estimate Radius From Mesh** magic wand button.
- Adjust the value to **1.5mm**.
- Click the **OK** to finish.

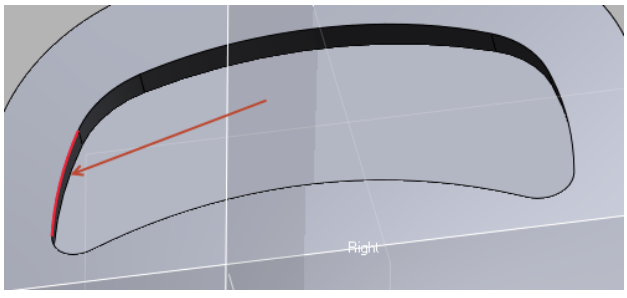


Figure 171

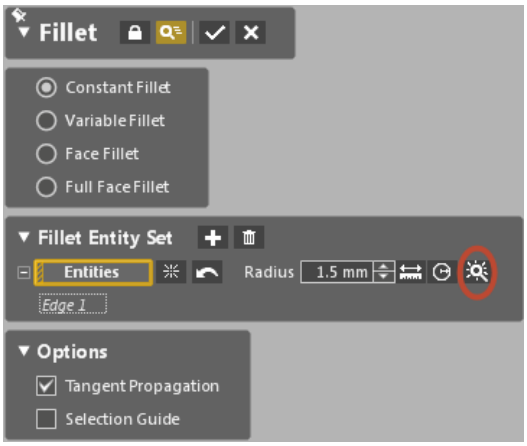


Figure 172

2. Repeat the process on the fillet around the outside of the mesh. Add all edges belonging to the same size fillet.

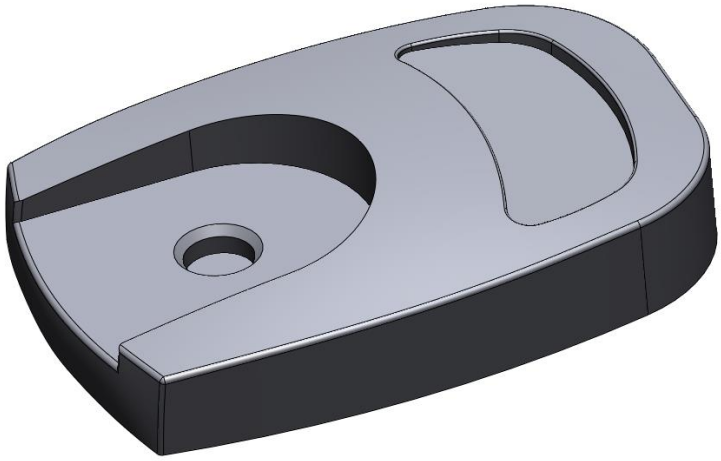


Figure 173

3. Right-Click the solid edge of the other cut out in **Figure 174,** and choose the **Fillet** icon.
   - Change the type to **Variable Fillet**.

☞ *The Profile View will appear below the Model View. This will be used to figure out the varying radius.*

   - Manually select all 3 edges used for the variable fillet then click the **Estimate Radii From Mesh Button.**
   - Look at the predicted fillet radius down in the Profile View.

4. Adjust the end points to match the pink polyline.
   - Set the **R(adius)** value to **4mm** at each end point.
   - Click on the middle section to re-zoom around this segment. Click again to place a new point in the approximate middle of the line.
   - Drag this point up to tallest peak of the pink polyline.
   - Click **OK** to finish.

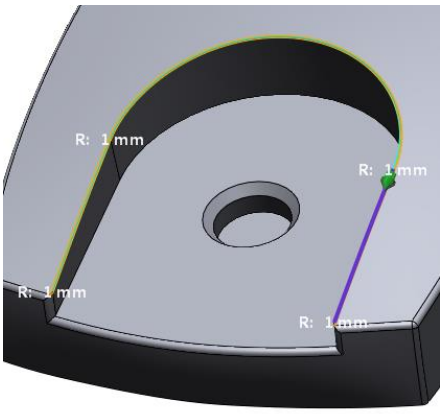


Figure 174

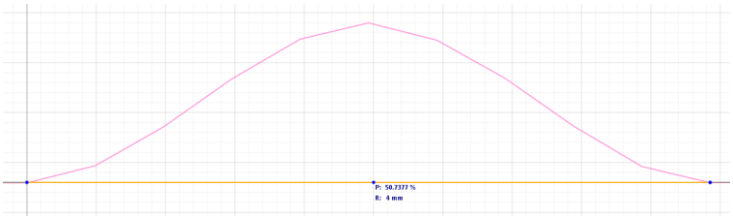


Figure 175

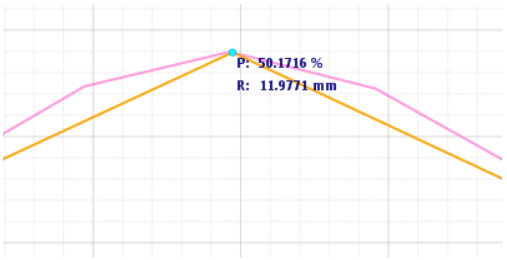


Figure 176

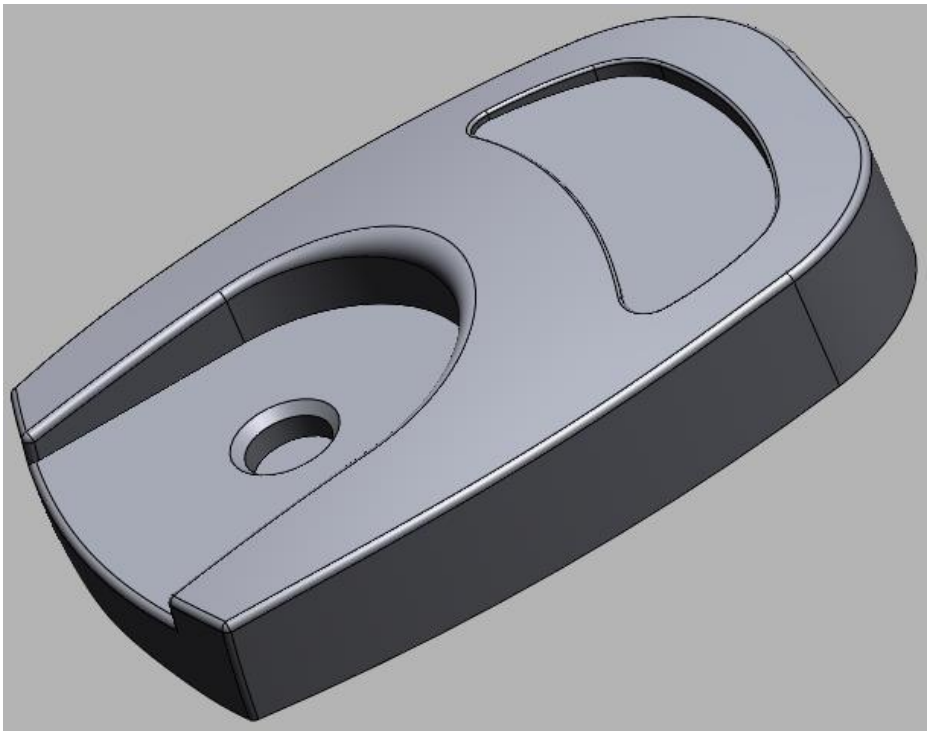


Figure 177

5. Insert any additional fillets to the model.

End of Activity

ACTIVITY: Mechanical Freeform Surface Modeling

Objective

Create a surface body using Modeling Wizard and Model Surface Features, and then combine the surfaces using 3D sketches and surface trimming tools. Analyze modeling result with Accuracy Analyzer™.

Highlighted Commands and Concepts

- **Surface Fitting**
- **Surface Editing**
- **Surface Loft**
- **3D Sketching**

Training File

08-Mech Freeform Surface.xrl

File Open

1. Open training file **08 – Mech Freeform Surface.xrl.**
   - Navigate to the file from the Open Files dialog and press the **Open** button.

☞ *The data is already aligned and classified with regions.*

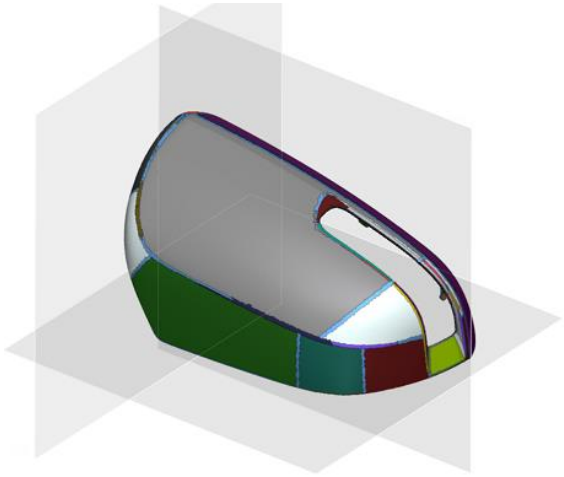


Figure 178

Upper Surfaces

1. Select the **Mesh Fit** button in the **MODEL>Wizard** group on the ribbon to create a surface on the large, upper face.
   - Select the large, upper target **Region** shown in **Figure 179**.
   - Set the **Resolution** type to **By Allowable Deviation**, and the **Allowable Deviation** to **0.1mm**.
   - **Rotate** the preview plane with a manipulator node until it is parallel to the x-axis.
   - Click the **OK** to accept the command.

☞ *The orientation of the preview plane will be the same as the finished surface. Having the surface more square to the part will allow for higher quality surfaces in the end.*

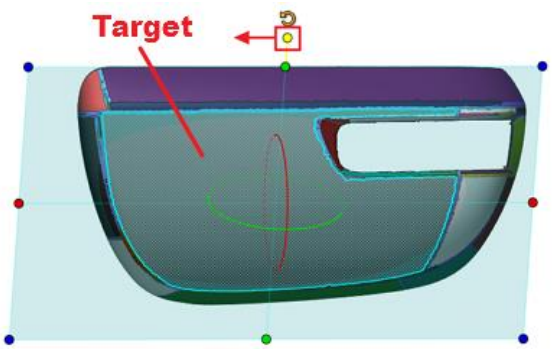


Figure 179

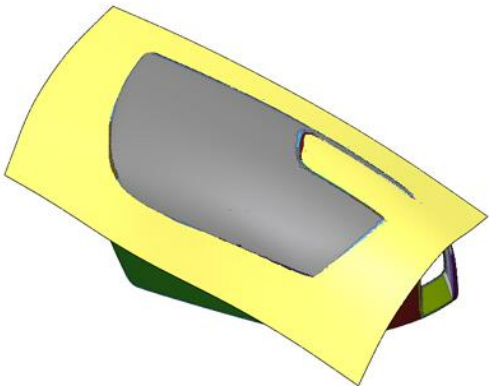


Figure 180

2. **Hide** only the **Mesh Fit(1)** surface body using the **Model Tree**.
3. Select the **Mesh Fit** button in the **MODEL>Wizard** group on the ribbon to create the adjoining surface on the upper face.
   - Select the target **Region** shown in **Figure 181**.
   - Click the **OK** to accept the command using the previous options.

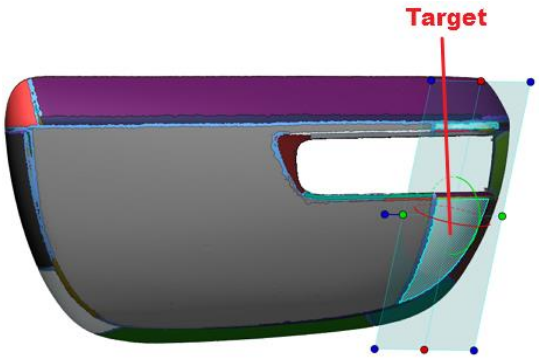


Figure 181

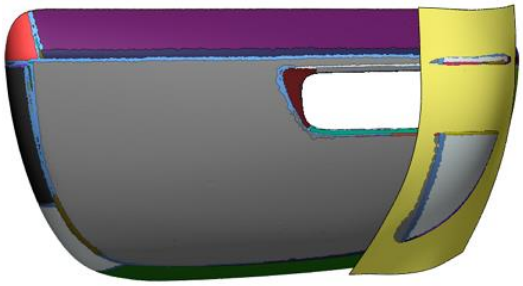


Figure 182

Join Surfaces

1. Click the **3D Sketch** button in the **3D SKETCH > Setup** group on the ribbon.
2. Select the **Spline** tool in the **Draw** group.
   - Draw a Spline from edge to edge of the Mesh Fit(1) surface body, as shown in **Figure 183**.
   - **Right-Click** to end the first spline.
3. Repeat Step 2 for drawing a Spline on **Mesh Fit(2)** as close to parallel as possible to the first spline, then Accept the Spline.

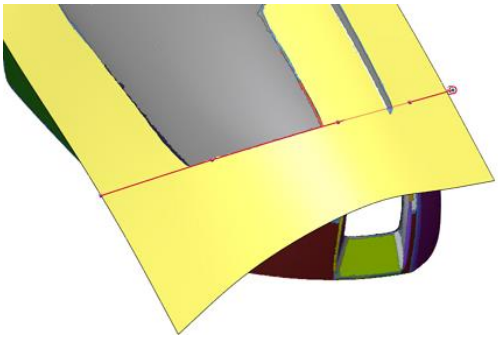


Figure 183 – Mesh Fit(1)

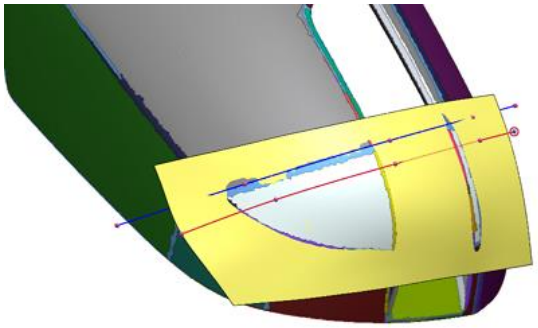


Figure 184 – Mesh Fit(2)

4. Click the **Split** button in the **3D SKETCH>Edit** group on the ribbon.
   - Split the **first curve** by clicking on two points near the edges of the **Mesh Fit(2)** surface.
   - **Accept** the command.

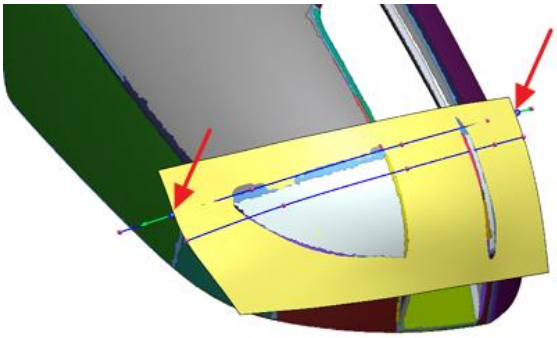


Figure 185

5. Click the **Exit** button in the top left corner to finish the 3D Sketch.

6. **Hide** Meshes and Mesh Fit(2), and **Show** Mesh Fit (1).

7. Select the **MODEL>Edit>Trim Surface** command to trim the surfaces with the 3D Splines.
   - Select the first **Spline** in 3D Sketch(1) as the **Tool Entities**.
   - Select **Mesh Fit(1)** as the **Target Entities**.

   - Click the **Next Stage** button to continue.
   - Chose the correct larger portion of the surface as the **Remaining Bodies**.

   - Click the **OK** button to accept.

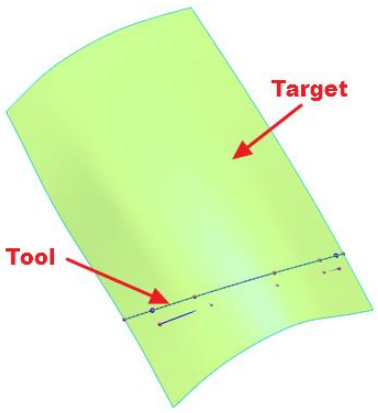


Figure 186

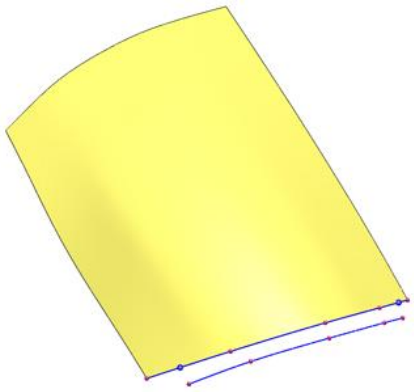


Figure 187

8. Repeat **Step 7** using the second drawn **Spline** as the **Tool Entities**, and the **Mesh Fit(2)** as the **Target Entities**.

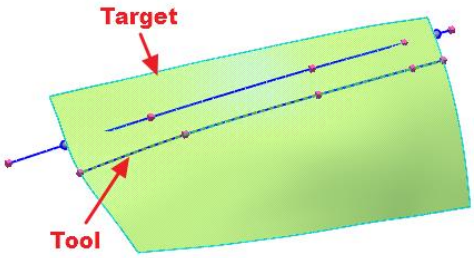


Figure 188

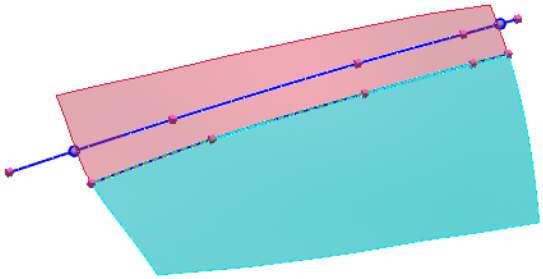


Figure 189

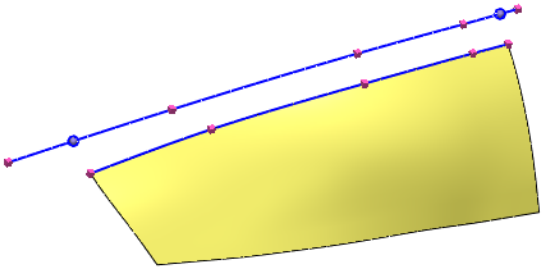


Figure 190

9. Turn on visibility for both **Trim Surface(1)** and **Trim Surface(2)**, and turn off visibility for **3D Sketch(1)**.

10. Select the **MODEL>Create Surface>Loft** command to join the surfaces.

☞ *Double check the Surface Loft command has been selected.*

- Select the **Edge1** of Trim Surface1, and **Edge2** of Trim Surface2 as the **Profiles**.
- Set the **Start Constraint** and **End Constraint** to **Tangency To Face**. This will apply tangency to the lofted surface where it connects to the original surfaces.
- Click the **OK** button to create the lofted surface.

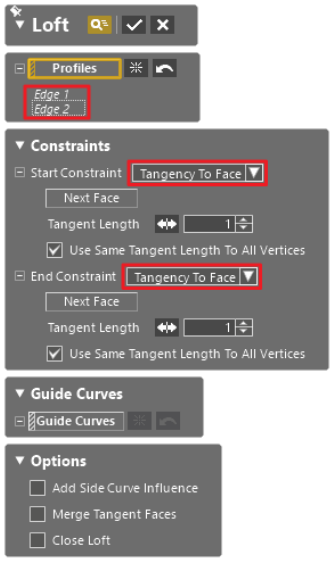


Figure 191

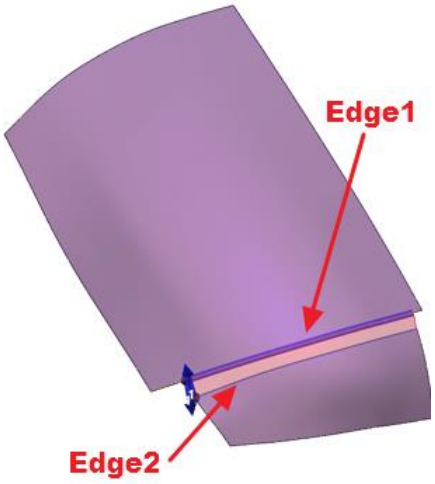


Figure 192

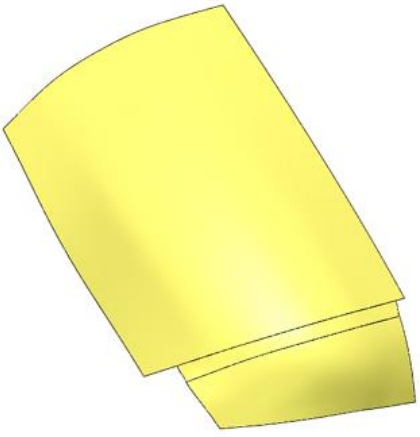


Figure 193

11. Finish joining all the surfaces into one body with the **Sew** command, found in the **MODEL>Edit** group.

- Select all **three** surfaces as the **Surface Bodies**.
- Click the **Next Stage** button.
- Click the **OK** button to accept the result.

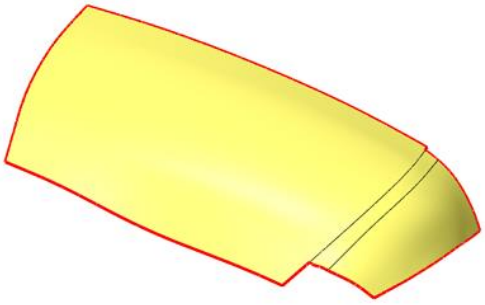


Figure 194

☞ *Alternatively, the Sew step can be skipped if the "Merge Tangent Faces" option is selected during the Loft command.*

Reasoning for splitting the spline

When trimming a surface by a curve, the split points on the curve will create a separate edge along the surface. Therefore, the surface edge length can be adjusted for lofting by editing the split points. If the edge lengths are of similar size when lofting, the resulting surface will be smoother and of higher quality.

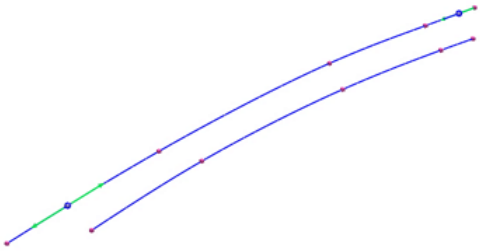


Figure 195 – 3D Sketch with Split

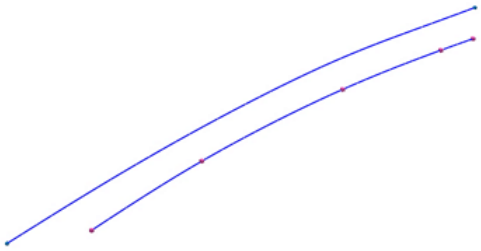


Figure 196 – 3D Sketch without Split

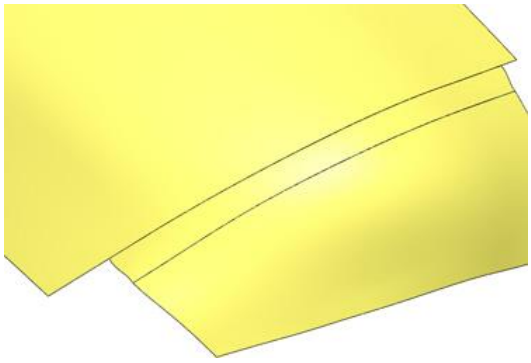


Figure 197 – Loft with Split

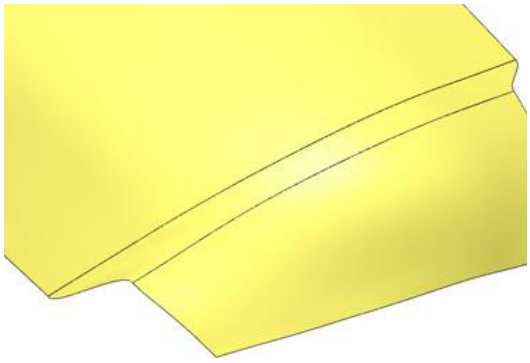


Figure 198 – Loft without Split

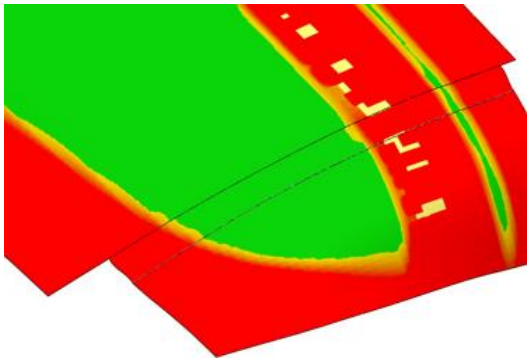


Figure 199 – Deviation with Split

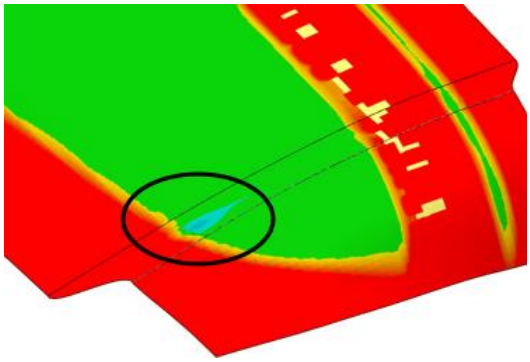


Figure 200 – Deviations without Split

Create Side Surfaces

1. Hide **Surface Loft1** to view the Mesh with Regions on.
2. Choose the **Loft Wizard** command in the **MODEL>Wizard**.
3. In the 1st stage of the Loft Wizard, set the location and orientation of the Loft.
   - Choose the three regions, shown in **Figure 202**, as the **Regions/Poly Faces**.
   - Change the **Section** option to **By No. Of Sections**, and set the value to **5.**
   - Check the options match **Figure 201**.

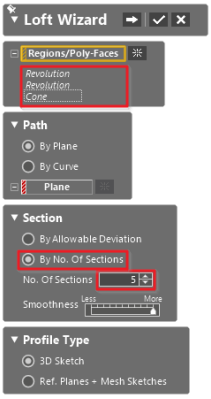


Figure 201

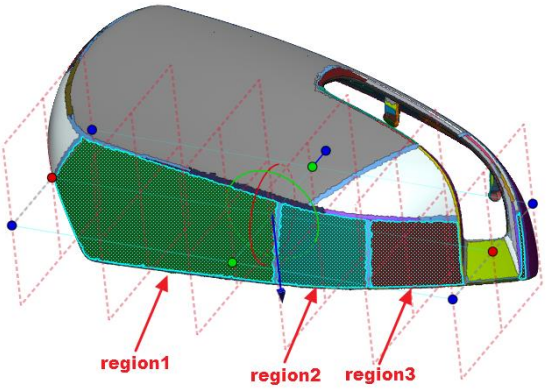


Figure 202

4. Rotate and adjust the manipulator to stretch the preview planes the long way across side body.

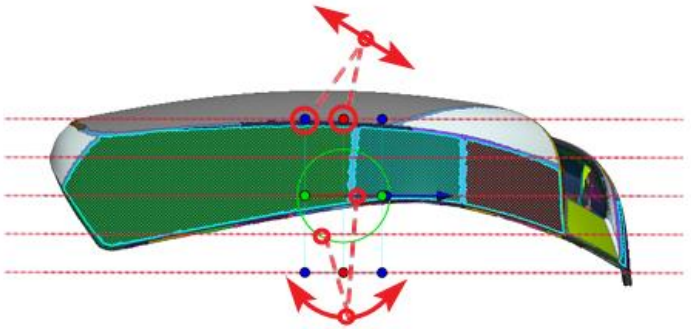


Figure 203

5. Click the **Next Stage** button to continue. A preview of the flow of the fitted surface body will be displayed.

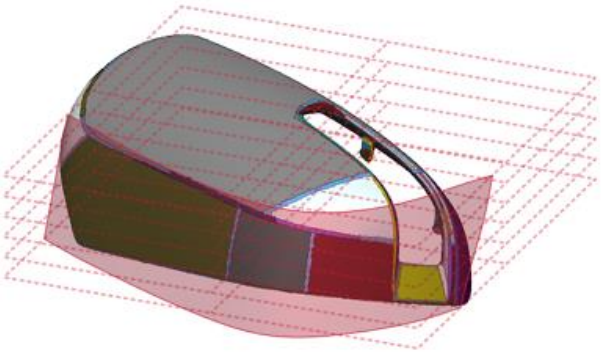


Figure 204

6. Turn on the **Deviation for Body** in the **Accuracy Analyzer™**, and use these deviations to add and move sections for a higher quality surface.

- *Sections can be moved by clicking and dragging. Sections can be added by holding Ctrl and clicking and dragging the section planes.*
- *These sections cannot be easily added and moved once the Loft Wizard has be completed.*

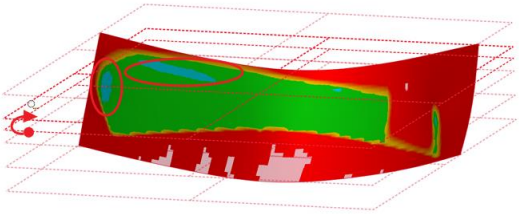


Figure 205

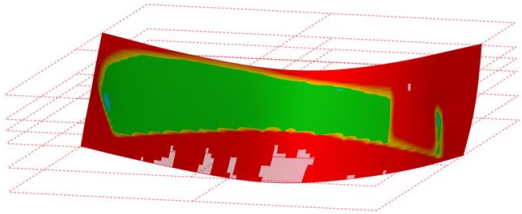


Figure 206

7. Click the **OK** button to finish the lofted surface.

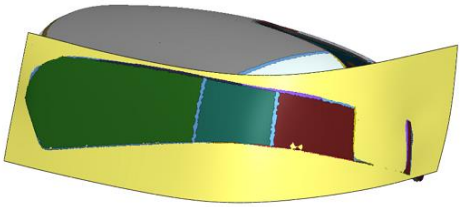


Figure 207

8. Create the adjacent surface with a **Mesh Fit** surface. Select **MODEL>Wizard>Mesh Fit** on the ribbon.

- Select the region shown in **Figure 208** as the Target **Regions/Poly-Faces**.
- Click the **OK** button to accept the surface using the previous settings.

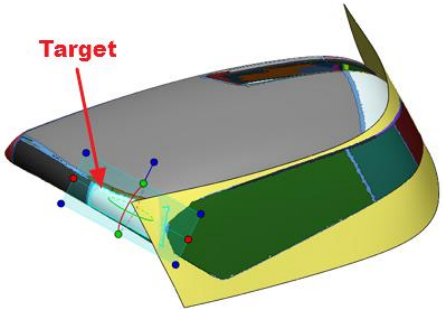


Figure 208

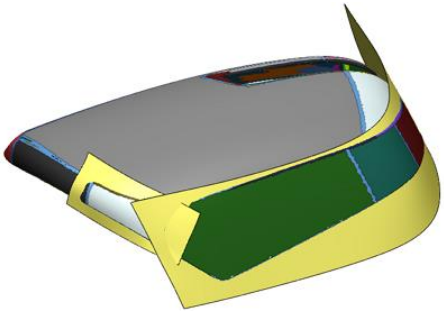


Figure 209

Join Side Surfaces

1. As done previously, enter into a **3D Sketch** to create clean trim lines for lofting between surfaces.

2. Hide Surface Loft1 and Surface Loft2 to only show the new Mesh Fit surface.

3. Select the **Spline** tool in the **3D Sketch>Draw** group.

- Draw a Spline on the Mesh Fit3 surface body (**Figure 210**).
- Hide the Mesh Fit3 surface and **Show** Surface Loft2.
- Draw a second spline on the Surface Loft2 body (**Figure 211**).

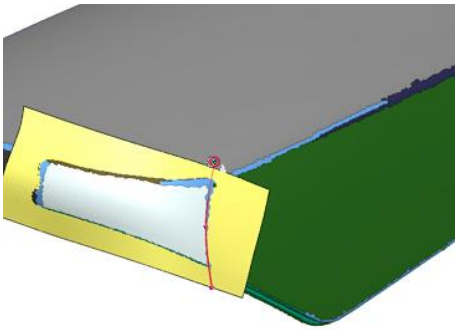


Figure 210

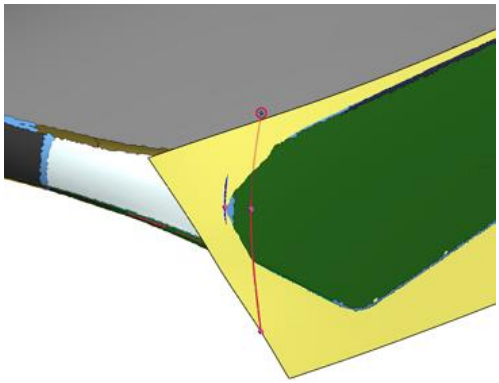


Figure 211

4. Select the **Split** button in the **3D SKETCH>Edit** group on the ribbon.
   - Turn on visibility of the Mesh Fit3 surface to see where the split points should be located.
   - Click on two points on the *second* spline to split the line.

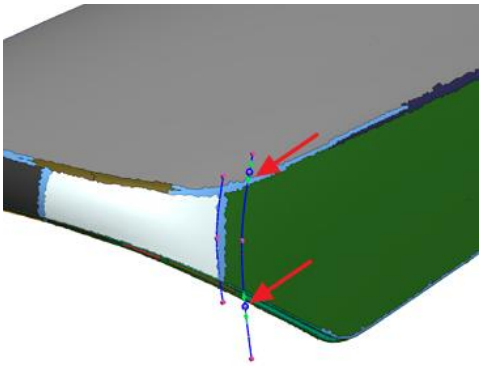


Figure 212

5. Click the **Exit** button in the top left corner to finish.

6. Turn on the visibility for Mesh Fit3 and Surface Loft2.

7. Select the **MODEL>Edit>Trim Surface** command on the ribbon.
   - Select the first spline in 3D Sketch3 as the **Tool Entities**, and **Mesh Fit(3)** as the **Target Entities**.
   - Click the **Next Stage** button to continue.
   - Choose the correct **Remaining Bodies**.
   - Click the **OK** button.

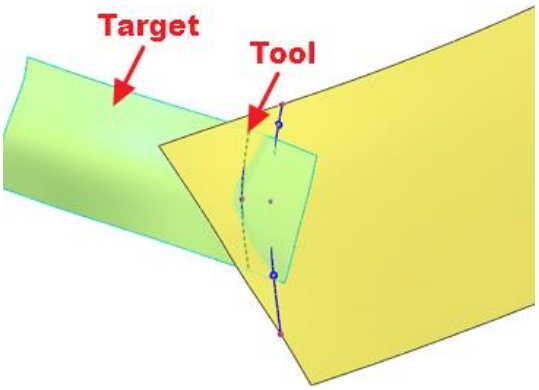


Figure 213

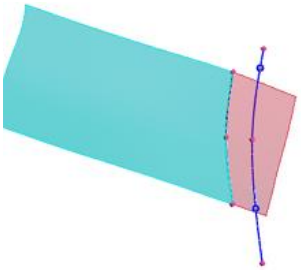


Figure 214

8. Repeat **Step 7** using the second spline as the **Tool Entity** and **Surface Loft2** as the **Target Entity** to Trim the lofted surface.

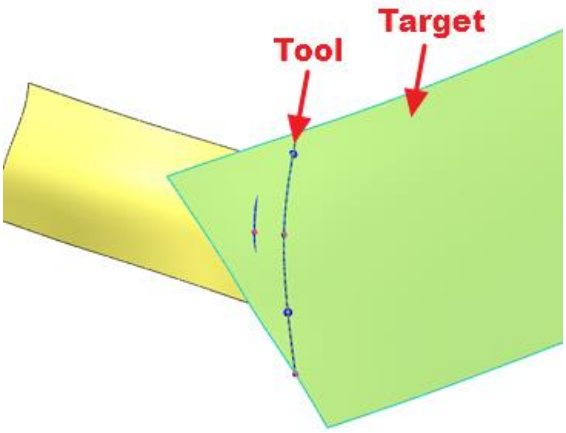


Figure 215

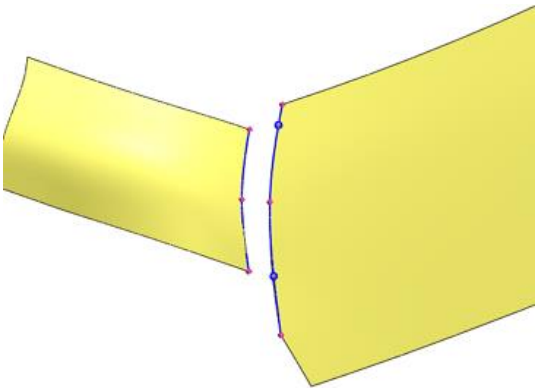


Figure 216

9. Hide 3D Sketch3, and Show Trim Surface3 and Trim Surface4.

10. Create a Surface Loft with the **MODEL>Create Surface>Loft** command to join the surfaces.

- Select the **Edge1** of Trim Surface3, and **Edge2** of Trim Surface4 as the **Profiles**.
- Set the **Start Constraint** and **End Constraint** to **Tangency To Face**. This will apply tangency to the lofted surface where it connects to the original surfaces.
- Place a check next to **Merge Tangent Faces** in the **Options** rollup.
- Click the **OK** button to create the lofted surface.

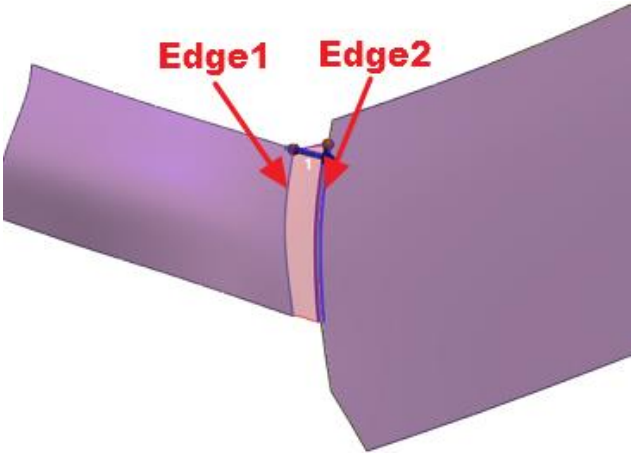


Figure 217

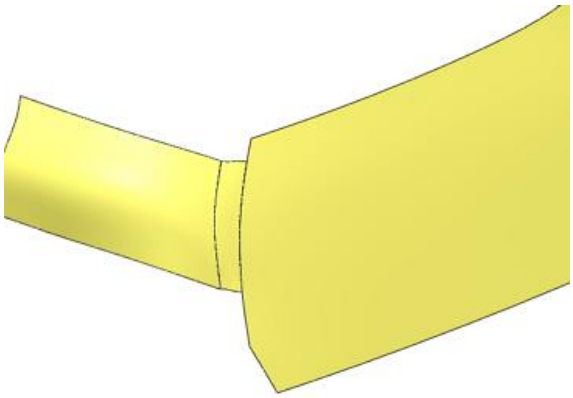


Figure 218

Finish Side Surfaces

1. Create the next with a **Mesh Fit** surface. Select **MODEL>Wizard>Mesh Fit** on the ribbon.
   - Select the region shown in **Figure 219** as the Target **Regions/Poly-Faces**.
   - Click the **OK** button to accept the surface using the previous settings.

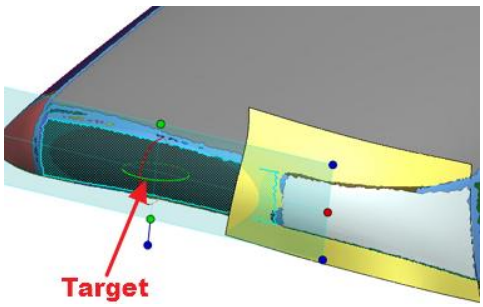


Figure 219

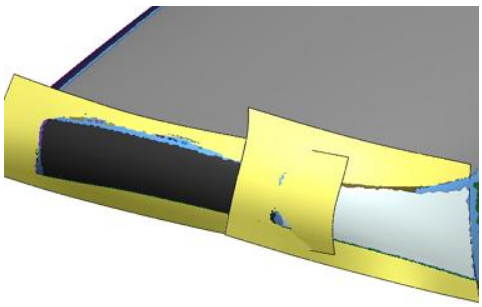


Figure 220

2. Enter into a **3D Sketch** to create clean trim lines for lofting between surfaces.
3. Hide all surfaces except the new Mesh Fit surface.

4. Select the **Spline** tool in the **3D Sketch>Draw** group.
   - Draw a Spline on the Mesh Fit4 surface body (**Figure 221**).
   - Hide the Mesh Fit4 surface and **Show** Trimmed Surface3.
   - Draw a second spline on the Trimmed Surface3 body (**Figure 222**).

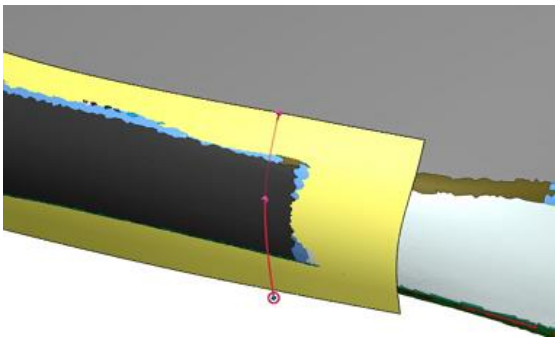


Figure 221

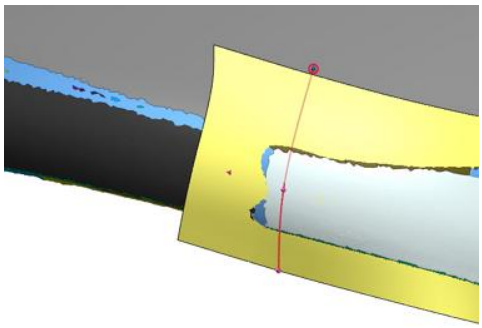


Figure 222

5. Select the **Split** button in the **3D SKETCH>Edit** group on the ribbon.
   - Click on two points on the second drawn, and longer, spline at approximately the length of the first spline.
   - Click **OK** to accept the Split.

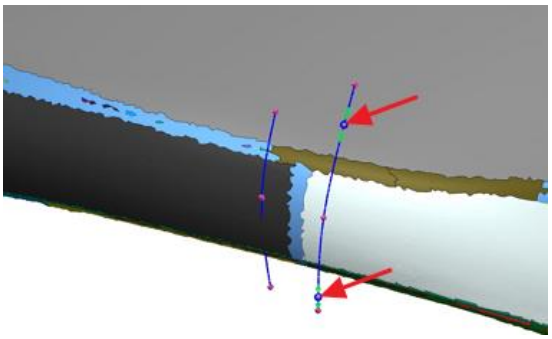


Figure 223

6. **Exit** 3D Sketch with the button in the top left corner.

7. Turn on the visibility for necessary surfaces.

8. Select the **MODEL>Edit>Trim Surface** command on the ribbon.
   - Select the split spline in **3D Sketch4** as the **Tool Entities**, and **Trim3** as the **Target Entities**.
   - Click the **Next Stage** button to continue.
   - Choose the correct **Remaining Bodies**.
   - Click the **OK** button.

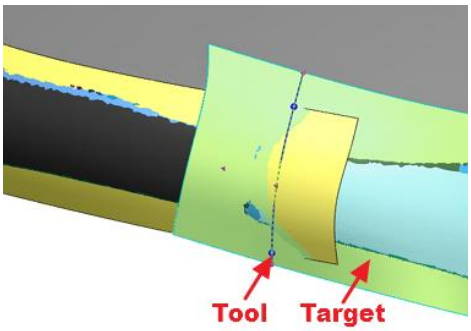


Figure 224

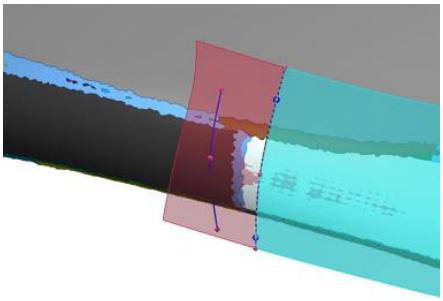


Figure 225

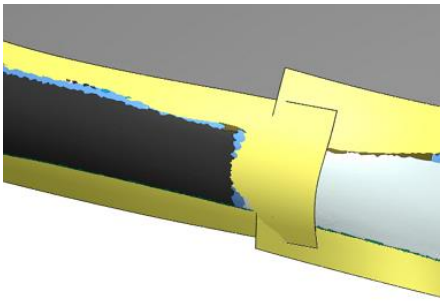


Figure 226

9. Select the **MODEL>Edit>Trim Surface** command on the ribbon.
   - Select the other spline in **3D Sketch4** as the **Tool Entities**, and **Mesh Fit4** as the **Target Entities**.
   - Click the **Next Stage** button to continue.
   - Choose the correct **Remaining Bodies**.
   - Click the **OK** button.

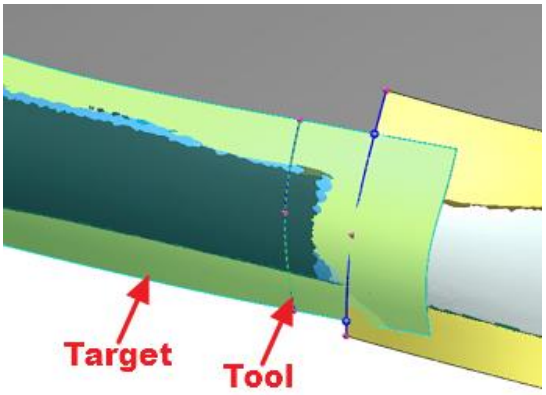


Figure 227

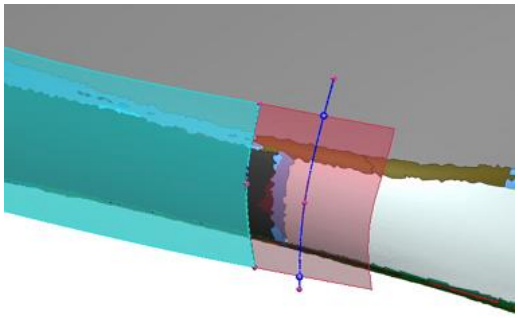


Figure 228

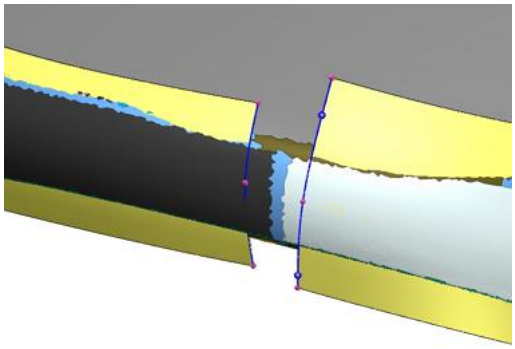


Figure 229

10. Hide 3D Sketch4 and show all the side surfaces.

11. Create a Surface Loft with the **MODEL>Create Surface>Loft** command to join the surfaces.

- Select the **Edge1** of Trim Surface5, and **Edge2** of Trim Surface6 as the **Profiles**.
- Set the **Start Constraint** and **End Constraint** to **Tangency To Face**. This will apply tangency to the lofted surface where it connects to the original surfaces.
- Place a check next to **Merge Tangent Faces** in the **Options** rollup.
- Click the **OK** button to create the lofted surface.

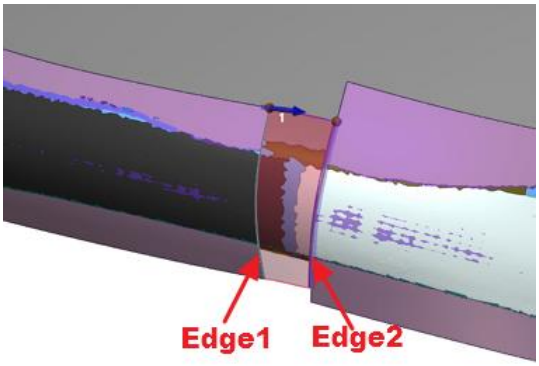


Figure 230

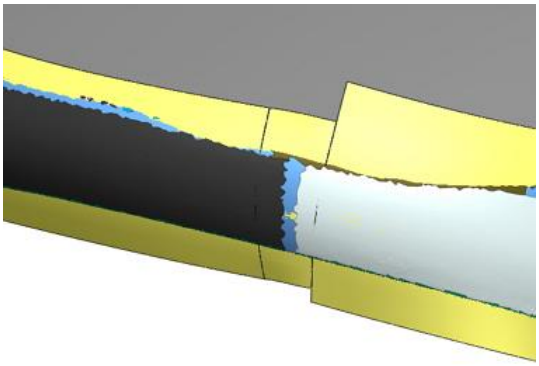


Figure 231

12. Click the **Sew** button in the **MODEL > Edit** group on the ribbon.

- Select all the side surface bodies as the target **Surface Bodies**.

- Click the **Next Stage** button to continue.

- Check the previewed result and then click the **OK** button.

☞ *This step is only necessary if there are multiple side surfaces.*

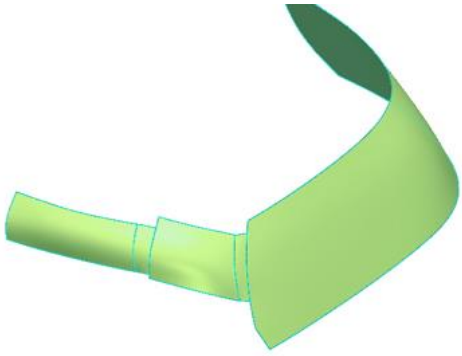


Figure 232

13. Click the **Environment Mapping** button in the **Accuracy Analyzer™**. Check continuity of the surface bodies.

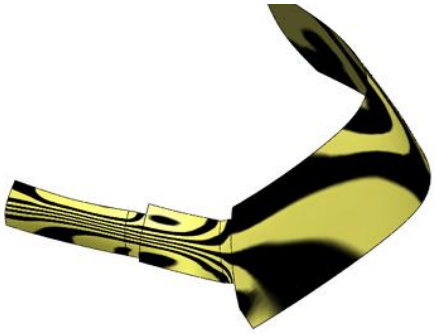


Figure 233

Combine Upper and Side bodies

1. Turn on the visibility of the upper and side surfaces (Surface Loft1 and Surface Loft4).

2. Select the **MODEL>Edit>Trim Surface** command on the ribbon.

- Select the **Loft1** and **Loft4** as the **Tool Entities**.
- **Deselect** the **Target** checkbox.

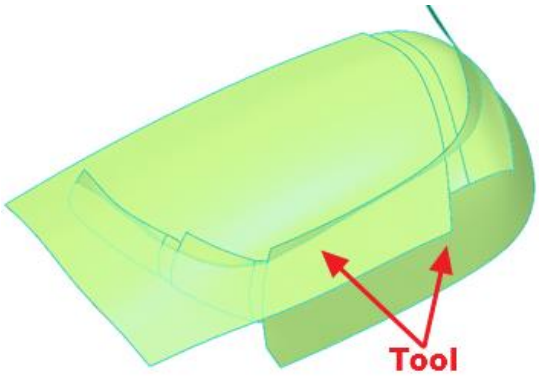


Figure 234

Figure 235

- Click the **Next Stage** button to continue.

- Choose the correct **Remaining Bodies**, as shown in the image below.
- Click the **OK** button.

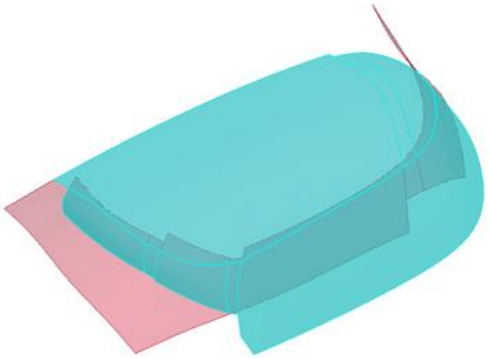


Figure 236

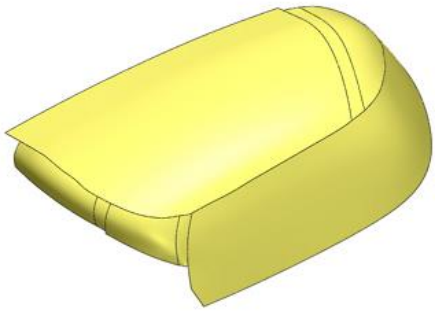


Figure 237

Add Fillet

1. Add a rounded, fillet edge to the connection of the two surfaces. Click the **Fillet** button in the **MODEL > Edit** group on the ribbon.

- Change the method to **Variable Fillet.**
- Select the edges between the two surfaces for the **Entities**, as shown in the image below.

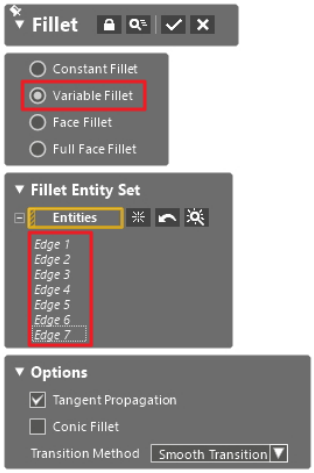


Figure 238

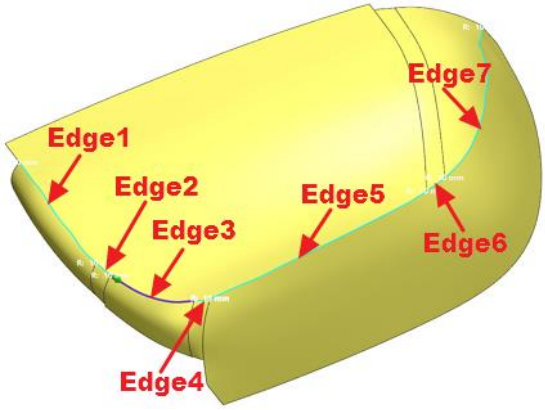


Figure 239

- Set **R** to **10-11mm** for each edge.

☞ *These values can be directly edited by clicking them in the Profile view or in the graphic window.*

- Click the **OK** button.

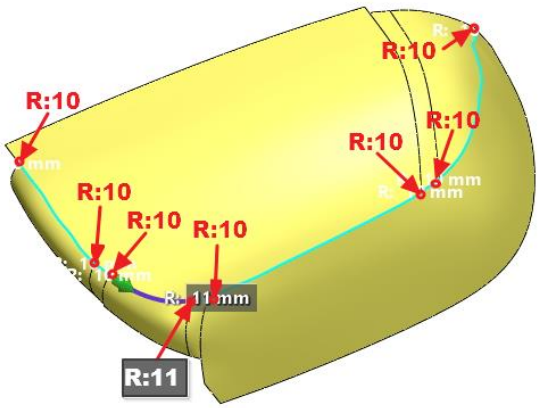


Figure 240

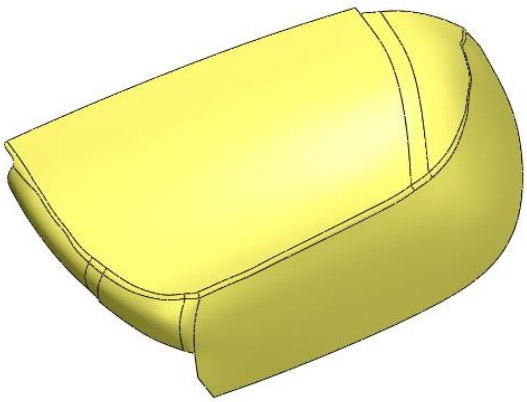


Figure 241

Finishing Features

1. Begin a **3D Sketch** by **Right-Clicking** in space in the Model View, and select the **3D Sketch** icon.

2. Draw a spline along either side of the fillet with the **Draw>Spline** tool.
   - Draw the first spline at the edge of the **Fillet1** surface body to the first seam. (**Figure 242**).
   - Draw a second spline below the crooked fillet (**Figure 243**).

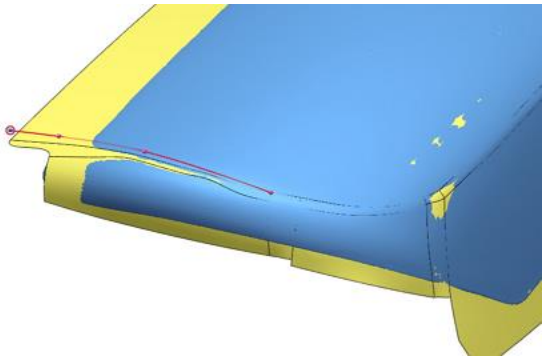


Figure 242

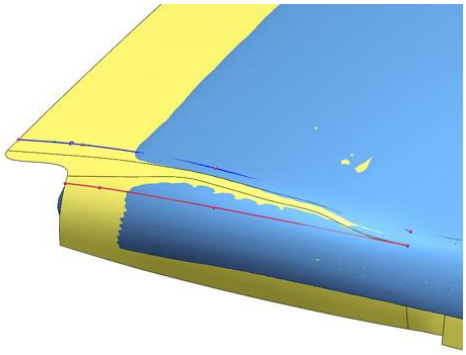


Figure 243

3. Click the **Split** button in the **3D SKETCH > Edit** group on the ribbon.
   - Select a point on the top spline near where the side surface ends.
   - Click the **OK** button to accept the command.

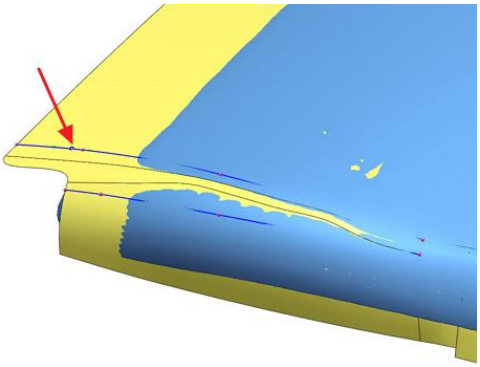


Figure 244

4. **Right-Click** and select the **Exit** button to finish the **3D Sketch**.

5. Hide the Mesh. Show the Surface Body and 3D Sketch 5 only.

6. Click the **Split Face** button in the **MODEL>Body/Face** group on the ribbon.
   - Select the split spline as the **Tool Entities**.
   - Select the upper surface face as the **Target Entities**, as shown in the image below.
   - Click **OK** button to accept the command.
7. Repeat Step 6 using the lower spline and the side surface face. (**Figure 246**).

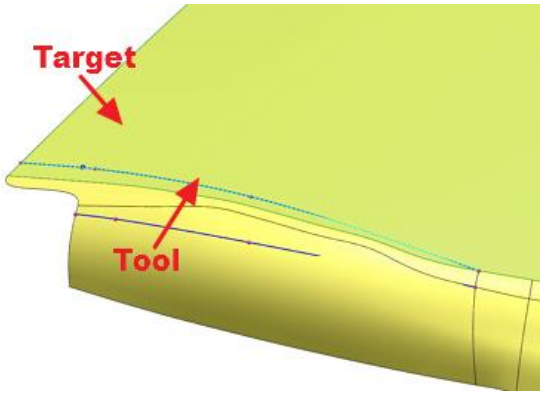


Figure 245

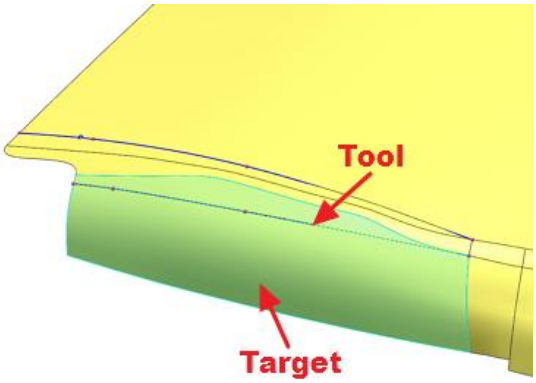


Figure 246

8. Remove the newly split area with the **Delete Face** command found in the **MODEL>Body/Face** group on the ribbon.

- Select the **Faces**, as shown in **Figure 247**.
- Click the **OK** button to remove the area between the splines. (**Figure 248**)

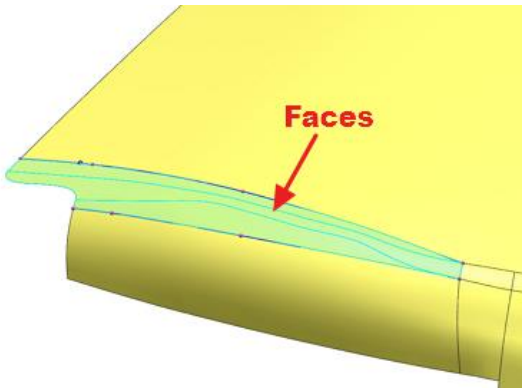


Figure 247

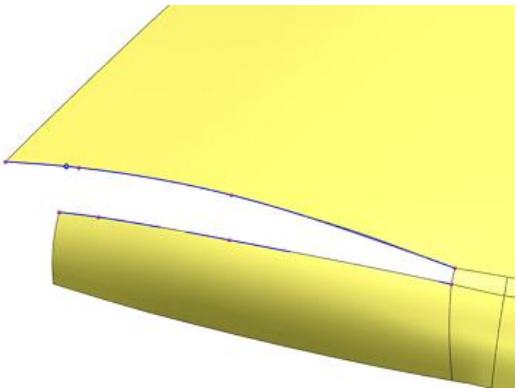


Figure 248

9. Hide **3D Sketch5**.

10. Create a Surface Loft with the **MODEL>Create Surface>Loft**.

- Select **Edge 1** and **Edge 2** as the **Profiles**.
- Apply the **Start** and **End Constraint** as **Tangency To Face**.
- Select the small **Edge 3** as the **Guide Curve** to create better connections with the surrounding surfaces.
- Click the **OK** button and a lofted surface will be created.

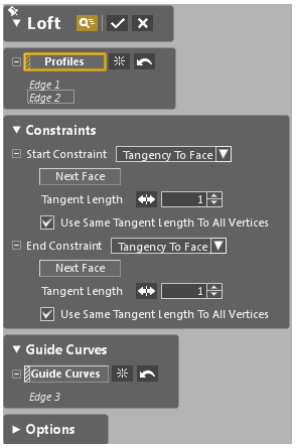


Figure 249

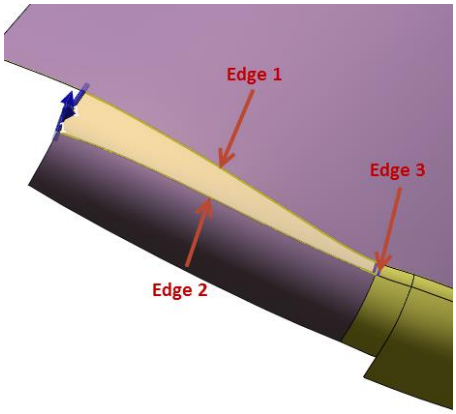


Figure 250

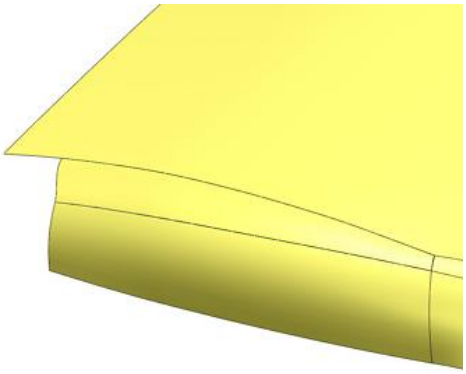


Figure 251

11. Repeat all of the previous steps on the other side where the fillet becomes unsmooth.

- 3D Sketch on either side of the fillet with a spline to connect
- Split Face of the surface using the 3D Sketch
- Delete Face
- Surface Loft

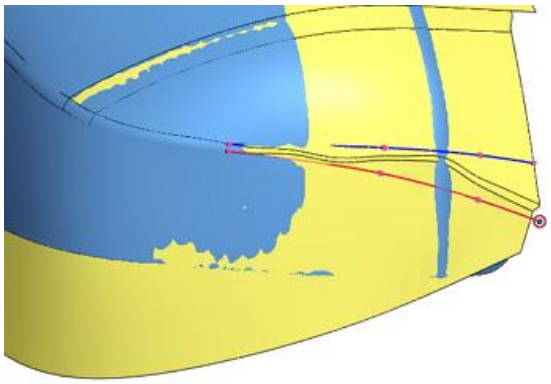


Figure 252 – 3D Sketch

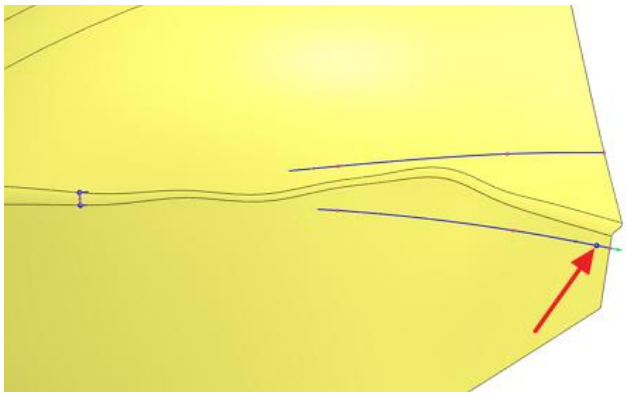


Figure 253 – 3 Splines with Split Point

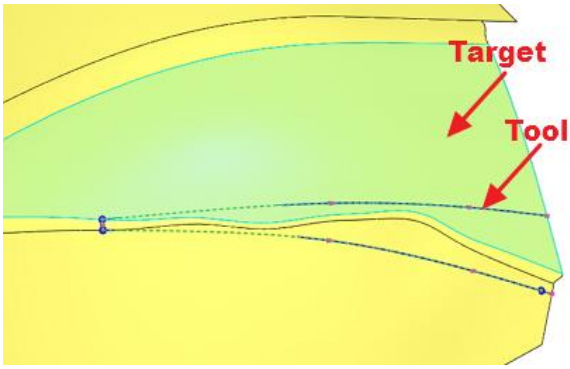


Figure 254 – Split Face

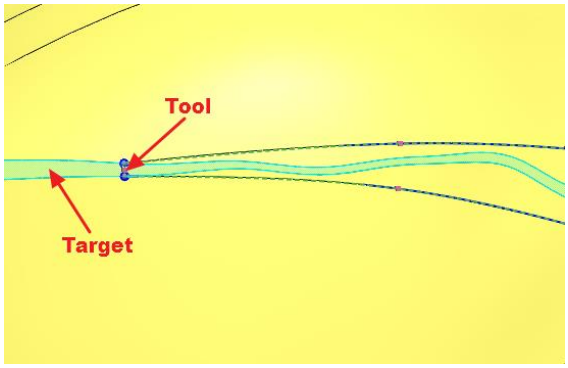


Figure 255 – Split Face

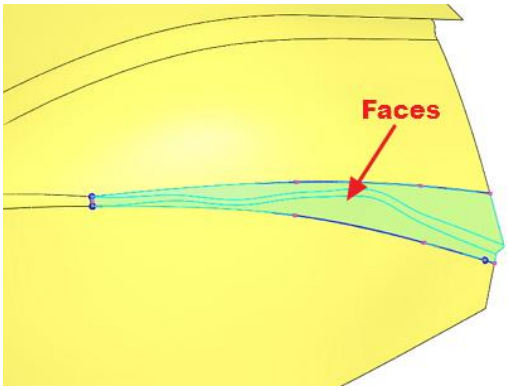


Figure 256 – Delete Face

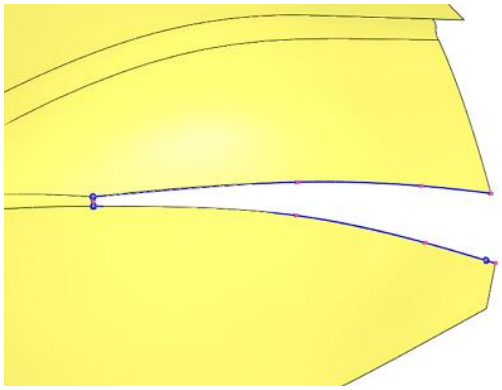


Figure 257 – Removed Face

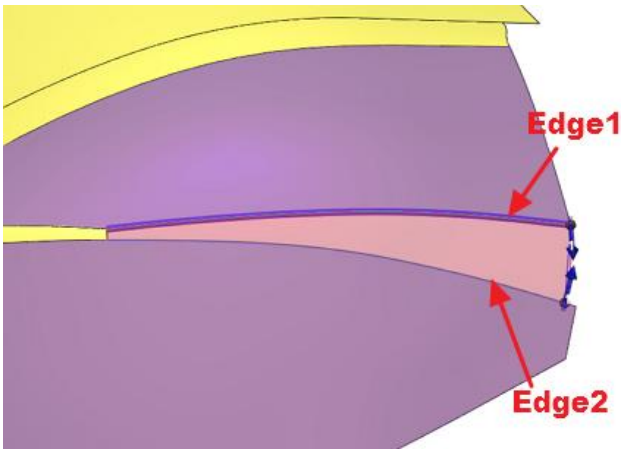


Figure 258 – Surface Loft

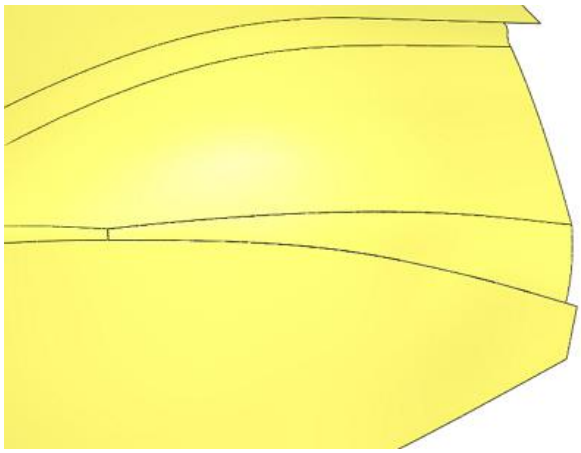


Figure 259 – Finished Surface

12. Sew the remaining surfaces together with the **MODEL>Edit>Sew** command.

- Select all the surface bodies as the target **Surface Bodies**.
- Click the **Next Stage** button to continue.
- Check the previewed result and then click the **OK** button to accept.

⦿ The model has more surfaces created and joined the same way the lesson demonstrated.

⦿ Continue creating surfaces or move to the Advanced Modeling section.

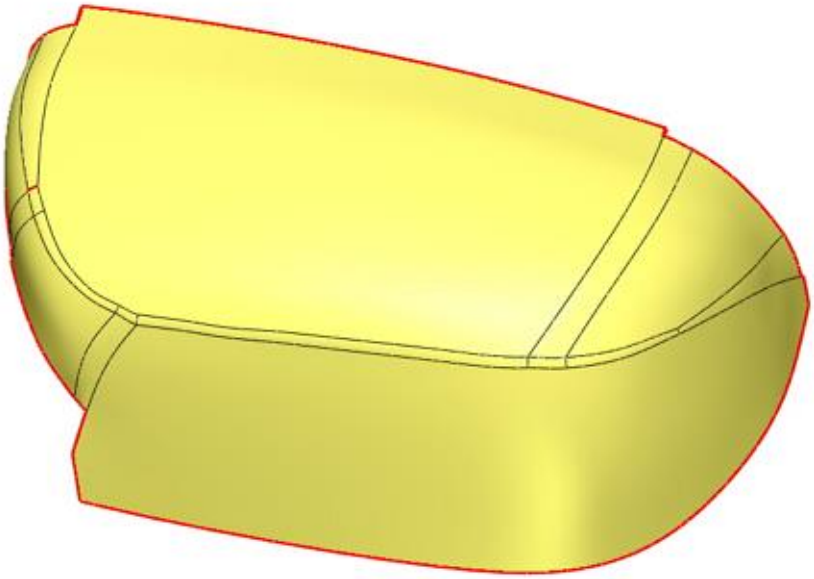


Figure 260

13. Click the **Deviation for Body** and the **Environment Mapping** buttons in the **Accuracy Analyzer™**.

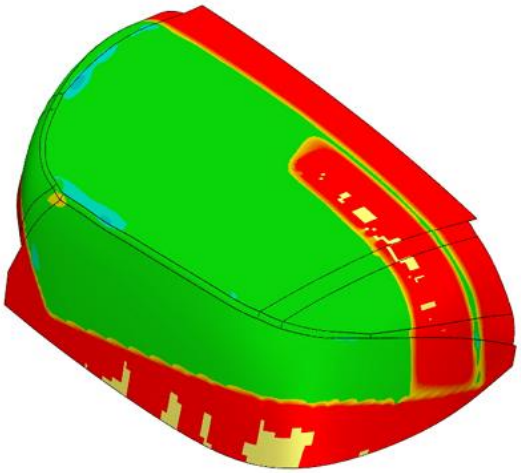


Figure 261

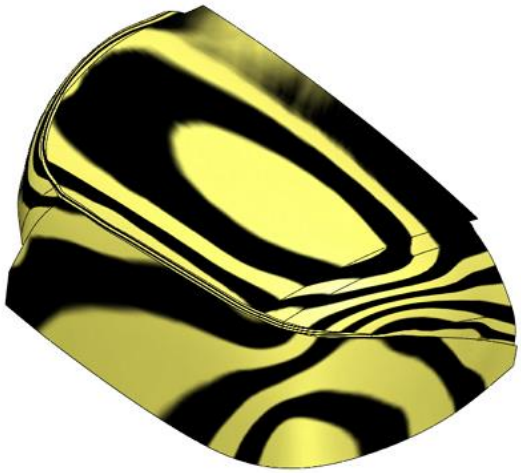


Figure 262

End of Activity

Advanced Modeling

ACTIVITY: Hybrid modeling

Objective

Create a model by combination of automatic and parametric modeling. Hybrid modeling allows for quickly creating a model which has accuracy and parametric features. The resulting model is helpful for checking assembly and analyzing.

Highlighted Commands

- **Global Remesh**
- **Enhance Shape**
- **Optimize Mesh**
- **Auto Surface**
- **Sketch**
- **Extrude - Cut**

Training File

09-Knuckle.stl

Scan Setup

1. Import training file **09-Knuckle.stl**.
   - Click the **Import** button or select **Import** from **Menu>Insert**.
   - Use the Import Dialog to navigate to the training files folder. Choose **Knuckle.stl** from the **Import dialog**.
   - Click **Import only**. The file is loaded and displayed in the Model View.

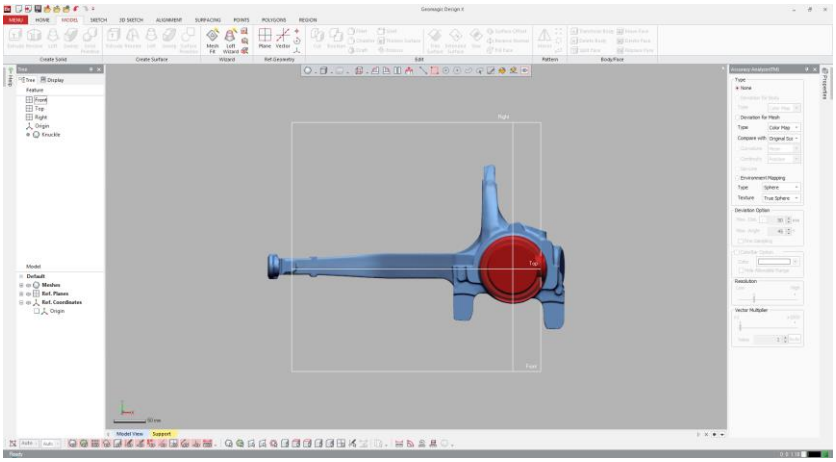


Figure 263

1. Select the **Auto Segment** command located in **REGION** tab.

☞ *The Auto Segment command can be activated by RMB context menu after selecting the mesh with a double click.*

- Set the **Sensitivity** to **10**.

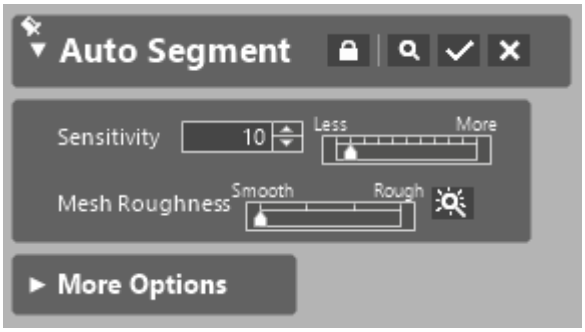


Figure 264

- Press the **OK** button to finish the **Auto Segment**.

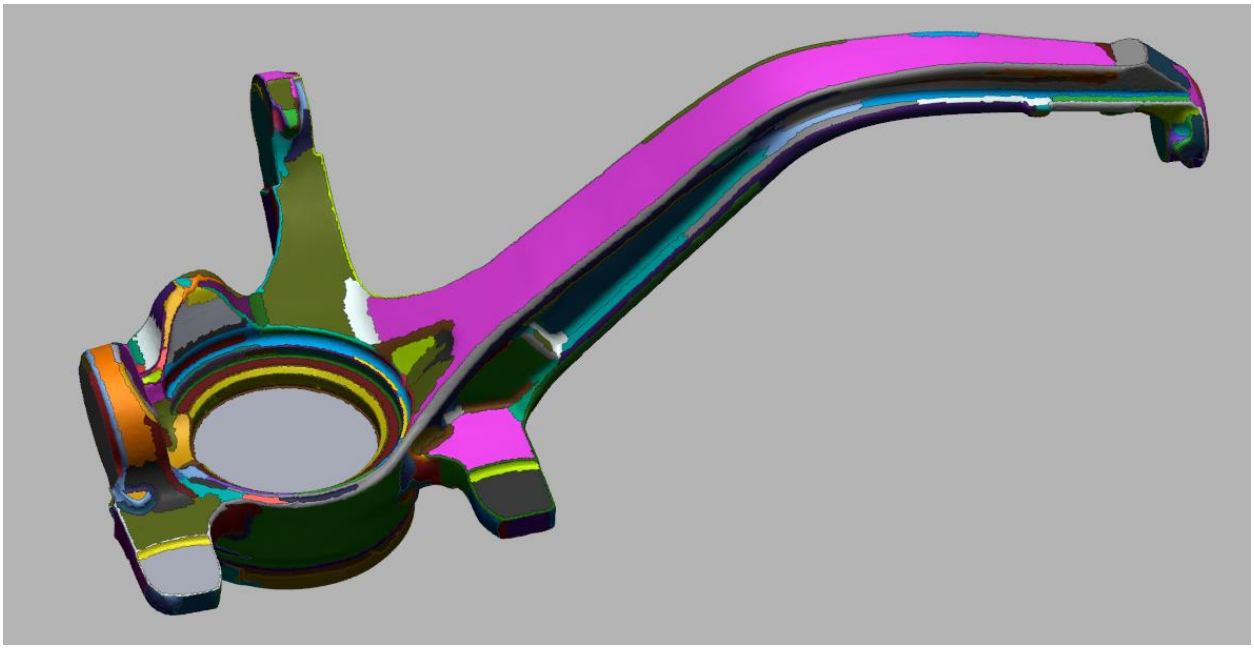


Figure 265

2. Click the *Knuckle* from **Feature Tree** and select **Copy** from **MENU>Edit** (Ctrl+C).
   - Click the **OK** button to copy the selected mesh data.

☞ *After editing a mesh to create a better Auto Surface, the original mesh will still be needed to show where cuts and features belong.*

Figure 266

- Select **Paste** from **MENU>Edit** (Ctrl+V) and paste the *Copied Mesh* in **Feature Tree**.

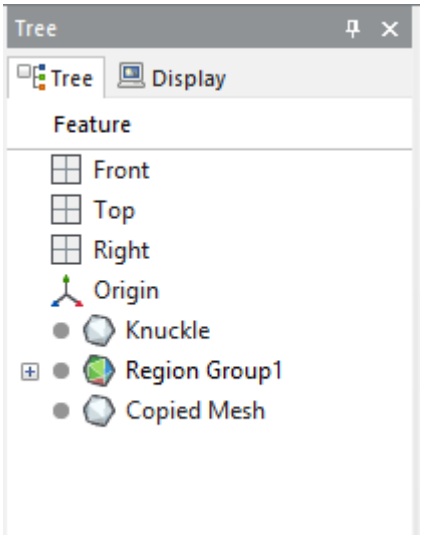


Figure 267

Edit and Optimize Mesh

1. Select *Knuckle* in **Feature Tree** to edit.

☞ *For a higher quality Auto Surface, details can be removed from the mesh and smoothed over. The Copied Mesh can still be used to track these features and shapes to use for sketch based modeling.*

☞ *When there are multiple meshes, a mesh needs to be selected before editing with the POLYGONS tab.*

2. Select **Offset** command located in **POLYGONS>Edit** group on the ribbon.

☞ *Create thickness beforehand to cut to plane.*

- Select ~8 regions while holding the **Shift** key (**Figure 268**).

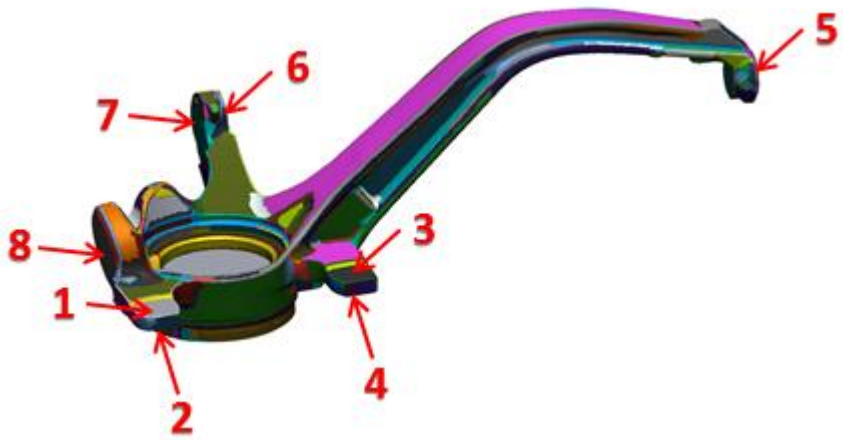


Figure 268

- Change the **Distance** to **1mm**.

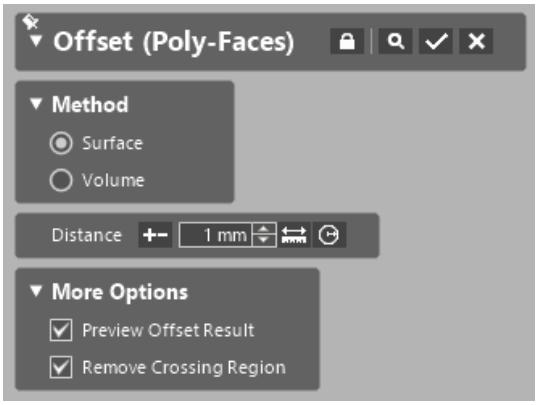


Figure 269

- Press the **OK** to finish the **Offset**.

Figure 270

3. Select **Global Remesh** command located in **POLYGONS** tab.

☞ *Recalculate the entire mesh to make watertight, and improve the mesh quality.*

☞ *Verify the mesh is still selected in the Tree if the command is not available.*

- Change the **Target Average Edge Length** to **0.5mm**.
- Place a **check** next to **Make Clean And Manifold Solid Mesh**.

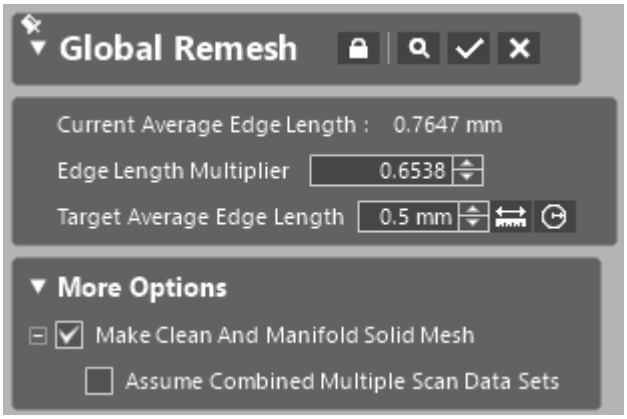


Figure 271

- Press the **OK** check button and finish **Global Remesh**.

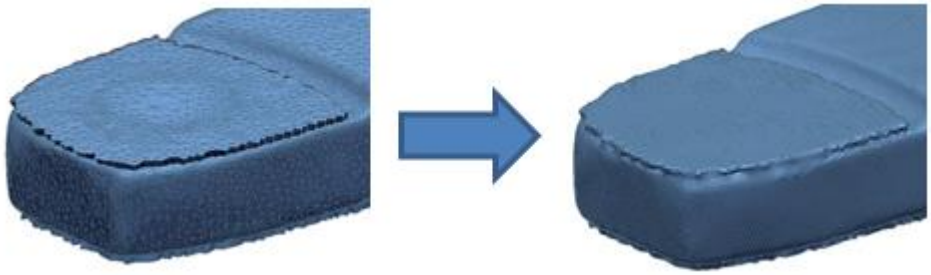


Figure 272

☞ *It is easier to see the mesh quality without Regions. Ctrl+2 will toggle Regions on and off.*

☞ *Select **Shaded With Edges** (F8) from **MENU>View>Mesh Display Mode** to show the mesh edges.*

4. Select **Enhance Shape** command in **POLYGONS>Optimize** group on the ribbon.

☞ *Improve the quality of the mesh by sharpening corners and smoothing flat or rounded areas.*

- Change the options like the figure below.

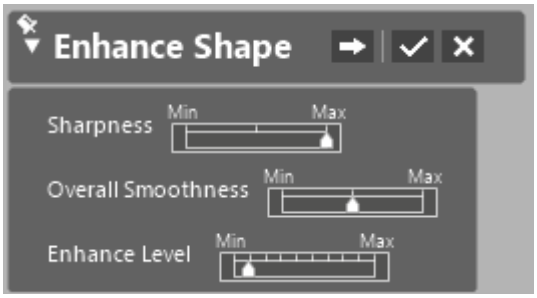


Figure 273

- Press the **OK** check button and finish **Enhance Shape**.

5. Select **Optimize Mesh** command located in **POLYGONS>Optimize>Global Remesh** dropdown.

☞ *Optimizes a mesh based on its feature shapes and improves the quality of the mesh. This command also offers advanced options to control the size of poly-faces and the effects of smoothing a model.*

- Change the **Method** to **High Quality Mesh Conversion**.

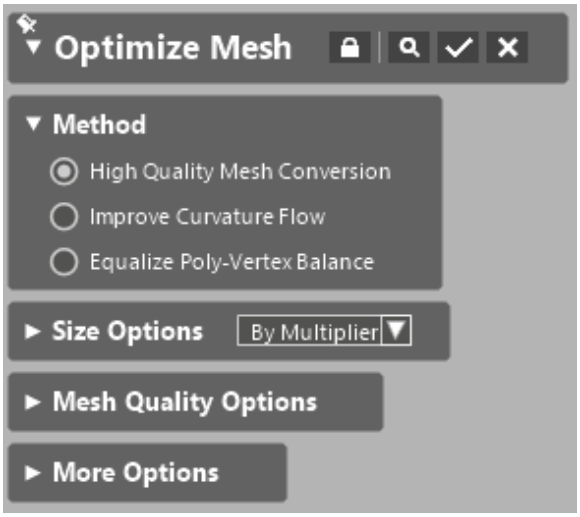


Figure 274

- Press the **OK** check to finish **Optimize Mesh**.

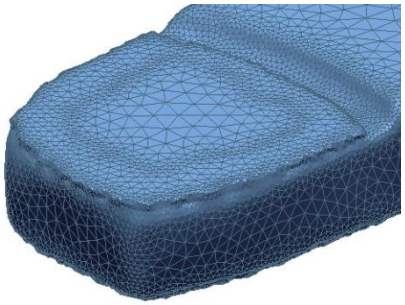


Figure 275

Create Solid Body

1. Select the **Auto Surface** command located in **SURFACING** tab.
   - Select *Knuckle* for **Mesh** and click **Organic** button.

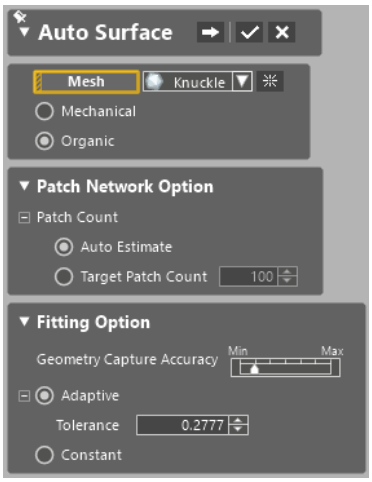


Figure 276

- Press the **OK** check button and finish **Optimize Mesh**.

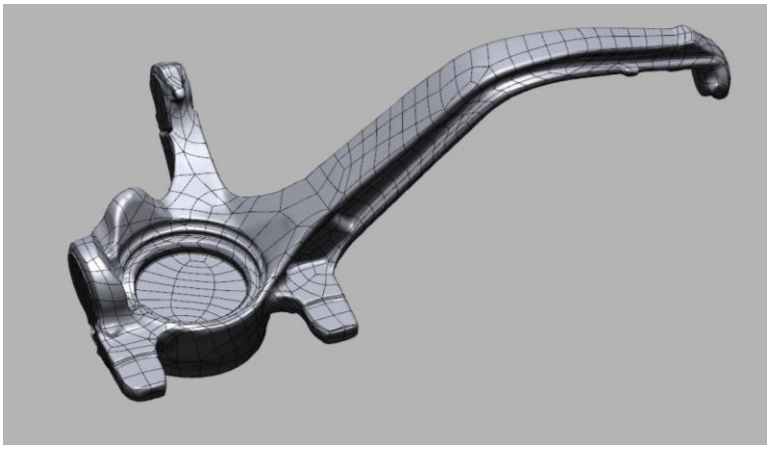


Figure 277

Create Cutting Planes

Create reference planes which are used as base sketch. These planes should be located at original mesh not on offset mesh.

1. Click *Copied Mesh* in **Feature Tree**.

☞ *Hide the visibility of the Mesh* **Knuckle**.

- Select the **Auto Segment** command located in **REGION** tab.

☞ *Use the previous settings.*

- Press the **OK** check button and finish **Auto Segment**.

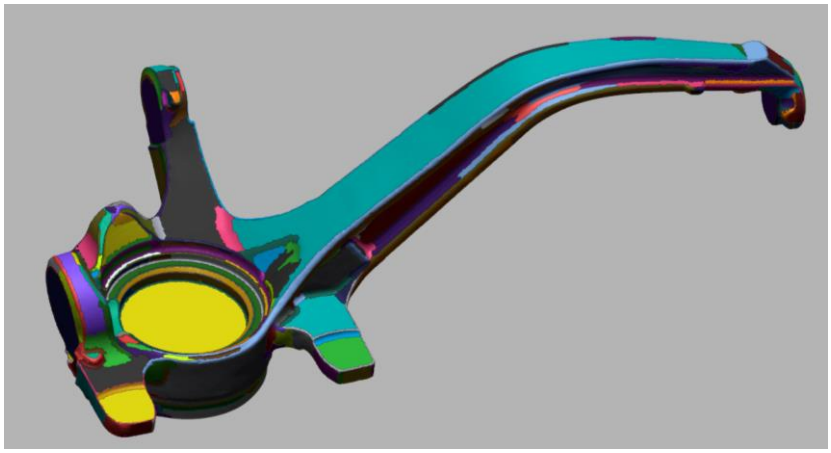
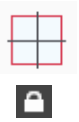


Figure 278



2. Select the **Plane** command located in **MODEL>Ref. Geometry** tab and group.

- Select the **Don't Quit Command With OK** Lock button to remain in the command after accepting.
- Select both the regions like the figure below.
- Verify the **Method** is set to **Average**.
- Press the **OK** check button.

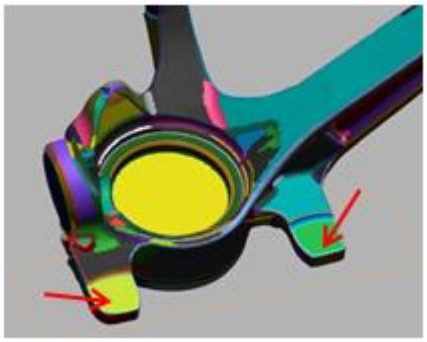


Figure 279

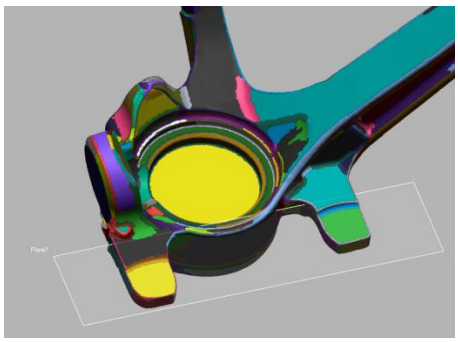


Figure 280

3. Immediately create a new **Plane**.

- Select the region as shown **Figure 281** below.
- Press the **OK** check button.

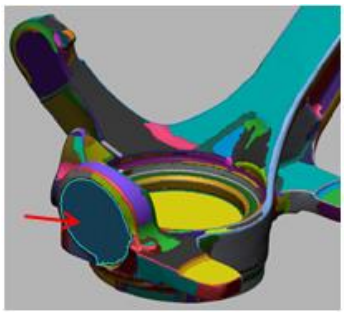


Figure 281

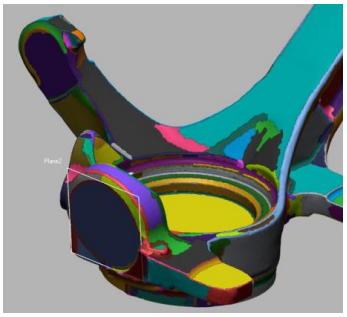


Figure 282

4. Select the region shown in **Figure 283**.
   - Press the **OK** check button.

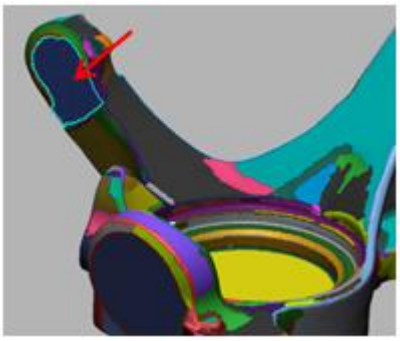


Figure 283

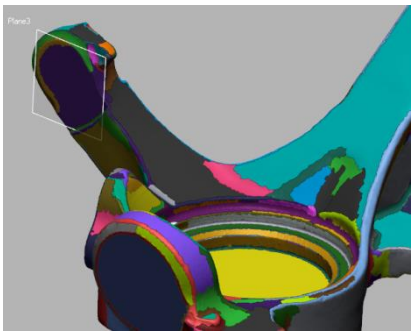


Figure 284

5. Select the region like **Figure 285**.
   - Click the **OK** check button.

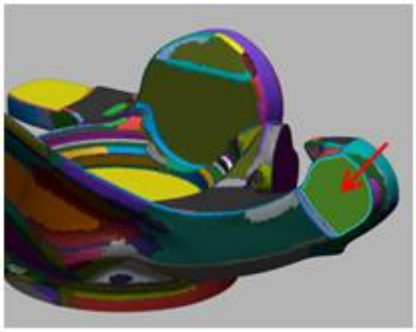


Figure 285

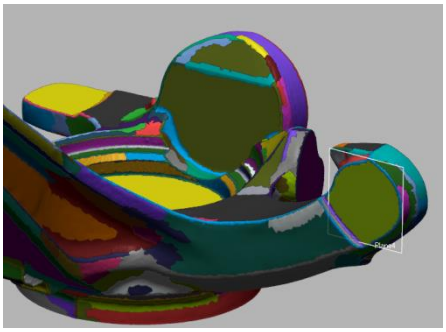


Figure 286

6. Select the region like **Figure 287**.
   - Click the **OK** check button.

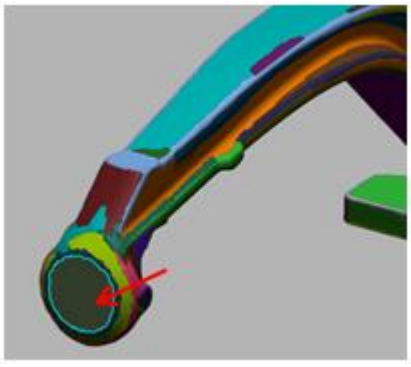


Figure 287

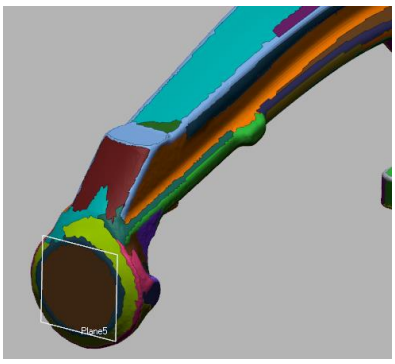


Figure 288

7. Select the region like **Figure 289**.
   - Click the **OK** check button to create the sixth plane.

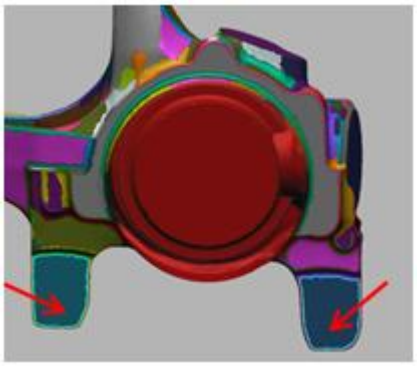


Figure 289

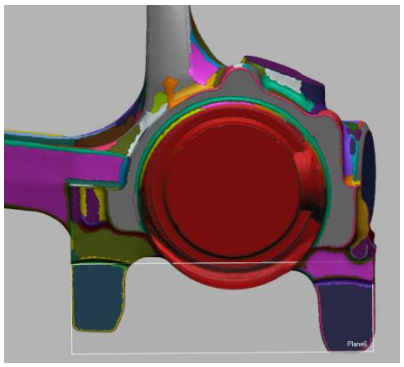


Figure 290

- Press the **Cancel** button and finish the **Plane** command.

Create Sketches

1. Create sketches to cut off redundant parts from solid model for machining.
   - Click **Knuckle** in Feature Tree.

2. Select **Mesh Sketch** command in **Setup** group located in **SKETCH** tab.
   - Select **Plane1** for **Base Plane** and press the **OK** check button.
   - Draw a sketch similar to **Figure 291**.

   - Press the **EXIT** button and finish the **Mesh Sketch**.

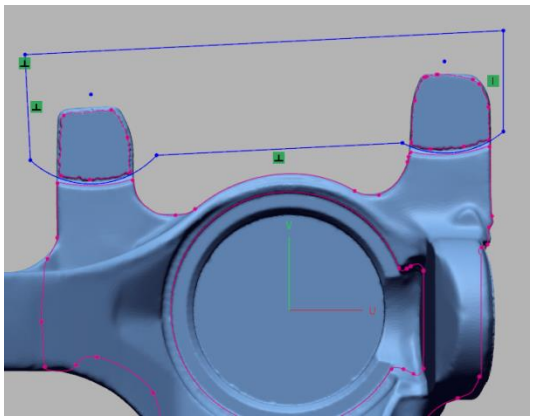


Figure 291

3. Click *Knuckle* in Feature Tree and select **Mesh Sketch** in the **SKETCH>Setup** tab.
   - Select **Plane2** for **Base Plane** and press the **OK** check button.

4. Draw circular sketch line like the figure.

   - Press the **Exit** button and finish **Mesh Sketch**.

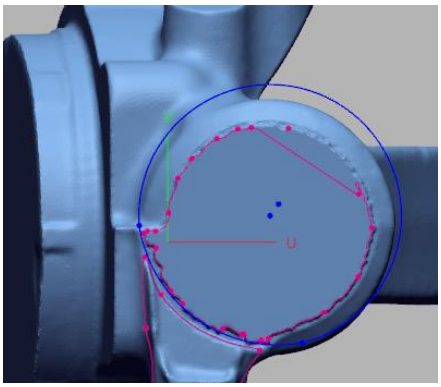


Figure 292

5. Click *Knuckle in the Feature Tree then* select **Mesh Sketch** command in **SKETCH>Setup** group.

- Select **Plane3** for **Base Plane** and press the **OK** check button.
- Draw sketch similar to shown in **Figure 293**.

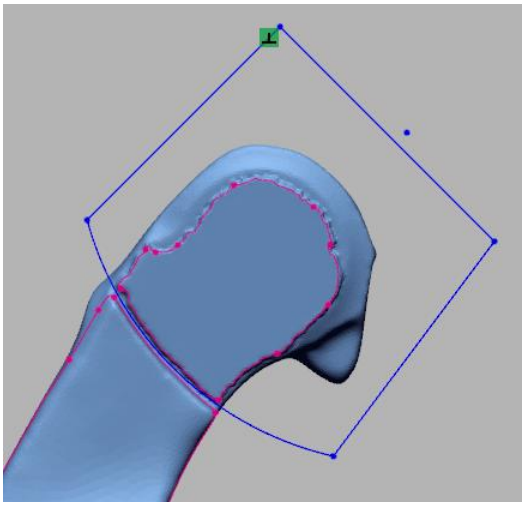


Figure 293

- Press the **EXIT** button and finish **Mesh Sketch**.

6. Click *Knuckle in the Feature Tree then* select **Mesh Sketch** command in **SKETCH>Setup** group.

- Select **Plane4** for **Base Plane** and press the **OK** check button.
- Draw sketch line like the figure.

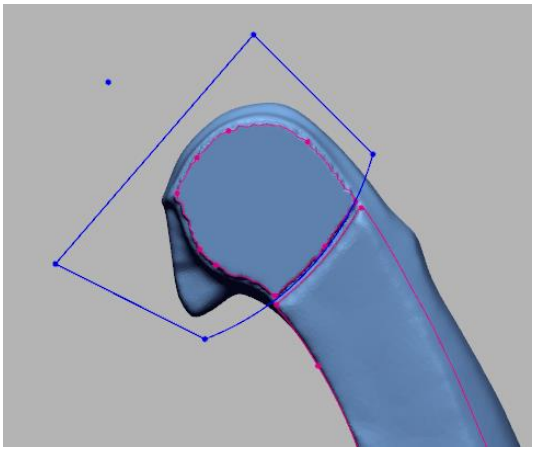


Figure 294

- Press the **EXIT** button and finish **Mesh Sketch**.

7. Click *Knuckle in the Feature Tree then* select **Mesh Sketch** command in **SKETCH>Setup** group.

- Select **Plane5** for **Base Plane** and press the **OK** check button.
- Draw sketch line like the figure.

- Press the **EXIT** button and finish **Mesh Sketch**.

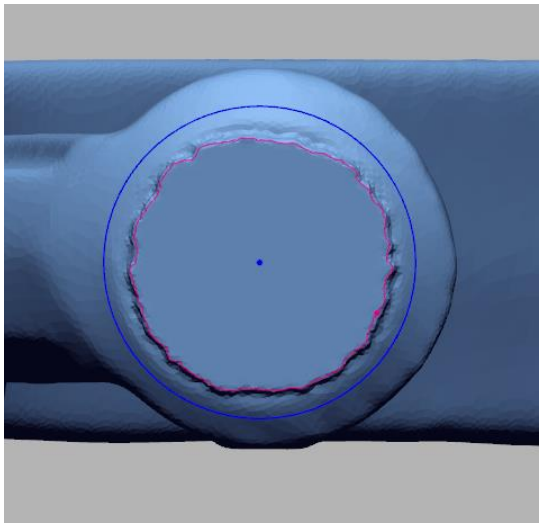


Figure 295

8. Click *Knuckle in the Feature Tree then* select **Mesh Sketch** command in **SKETCH>Setup** group.

- Select **Plane6** for **Base Plane** and press the **OK** check button.
- Draw sketch line like the figure.

- Press the **EXIT** button and finish **Mesh Sketch**.

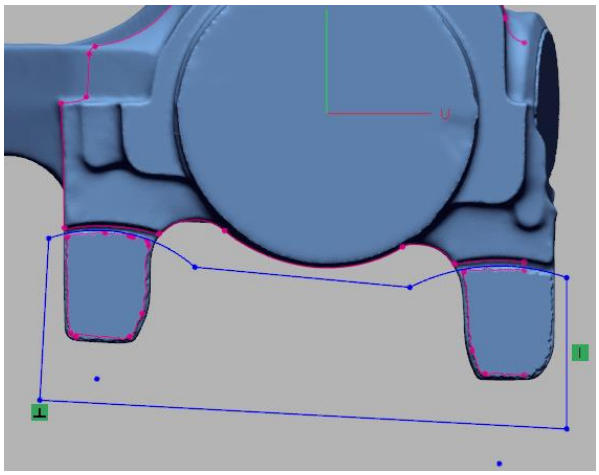


Figure 296

9. Show only Solid Bodies and Sketches in the Model View.

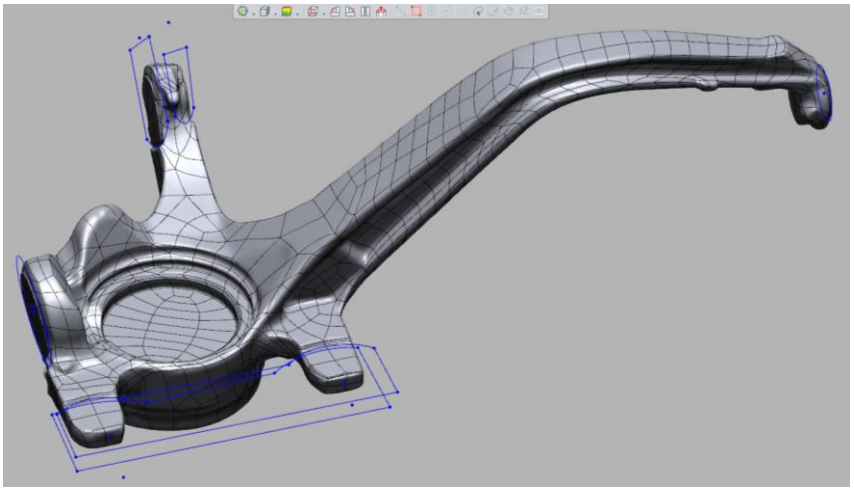


Figure 297

Cut Solid Body by Extruding Sketches

1. Extrude each sketch with the **MODEL>Create Solid>Extrude** command to cut the Solid Body.

- Select **Sketch1 (Mesh)** for **Base Sketch**.
- Set the **Method** to **Blind** with a **Length** of **20 mm**.
- Place a check in the **Cut** box in **Result Operator**.

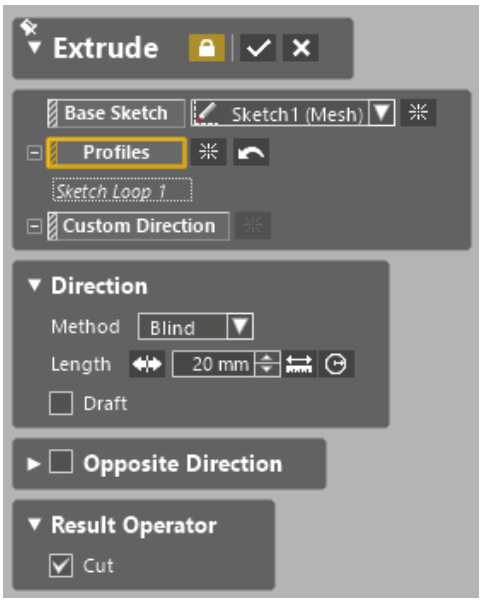


Figure 298

- Press the **OK** check button and finish **Extrude**.

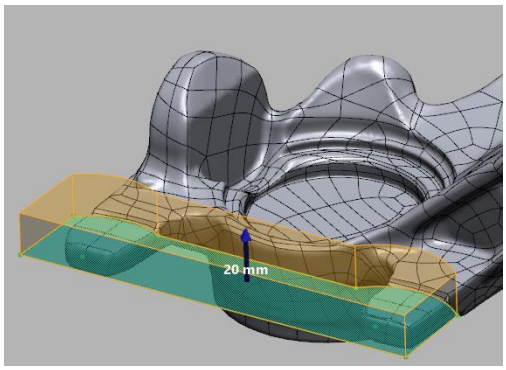


Figure 299

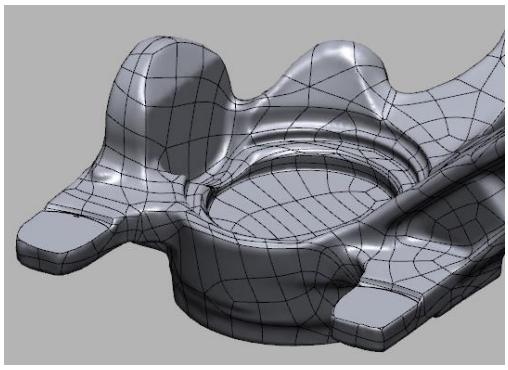


Figure 300

☞ *Repeat operation to cut the body.*

2. Select the **Extrude** command.

☞ *Use **Ctrl+Space** bar to execute previous command.*

- Select **Sketch2 (Mesh)** for **Base Sketch**.
- Place a check in the **Cut** box in **Result Operator**.
- Press the **OK** check button.

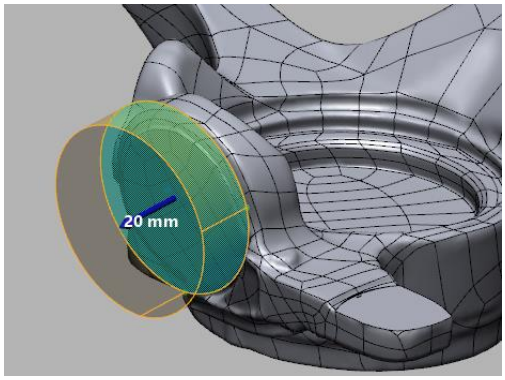


Figure 301

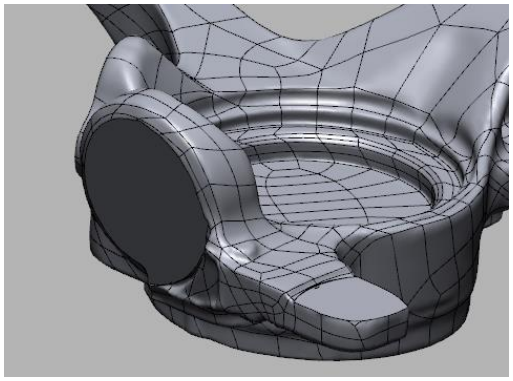


Figure 302

3. Select the **Extrude** command.

- Select **Sketch3 (Mesh)** for **Base Sketch**.
- Place a check in the **Cut** box in **Result Operator**.
- Press the **OK** check button.

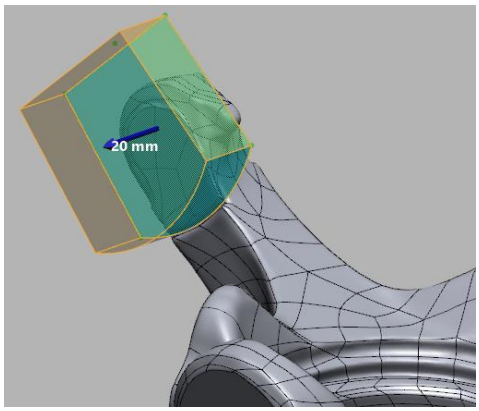


Figure 303

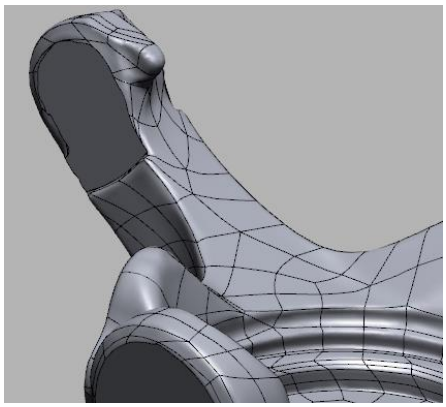


Figure 304

4. Select **Sketch4 (Mesh)** in the Feature Tree, then **Right-Click** in space and select the **Extrude** (Solid) icon.

- Place a check in the **Cut** box in **Result Operator**.
- Press the **OK** check button.

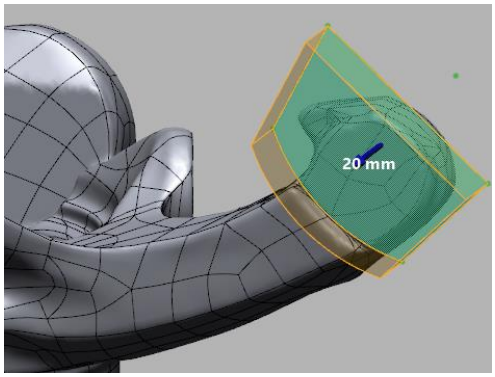


Figure 305

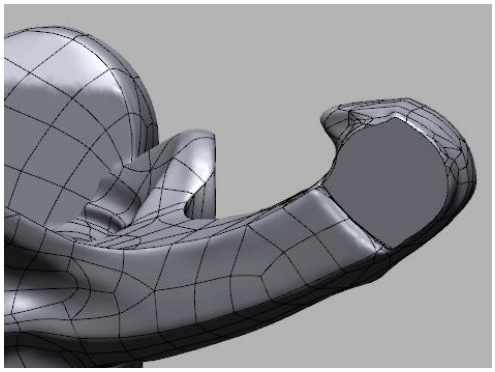


Figure 306

5. Select the **Extrude** command with any method.

- Select **Sketch5 (Mesh)** for **Base Sketch**.
- Place a check in the **Cut** box in **Result Operator**.
- Press the **OK** check button.

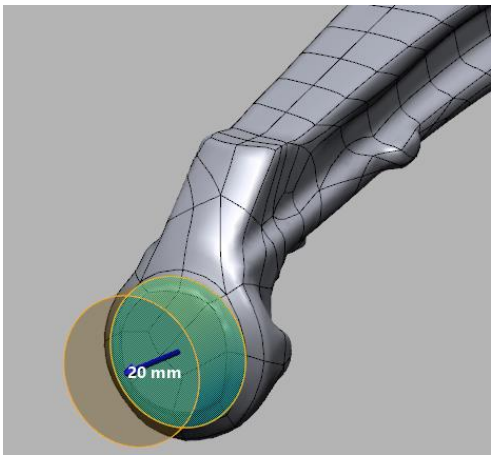


Figure 307

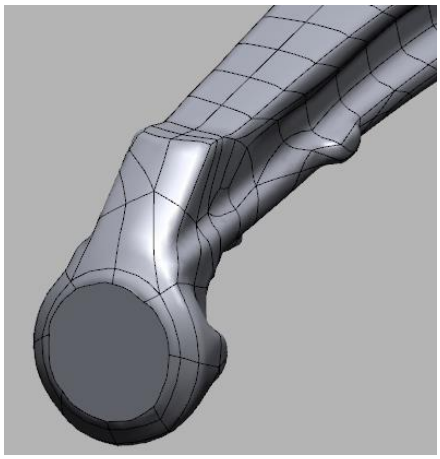


Figure 308

6. Select the **Extrude** command with any method.

- Select **Sketch6 (Mesh)** for **Base Sketch**.
- Place a check in the **Cut** box in **Result Operator**.
- Press the **OK** check button.

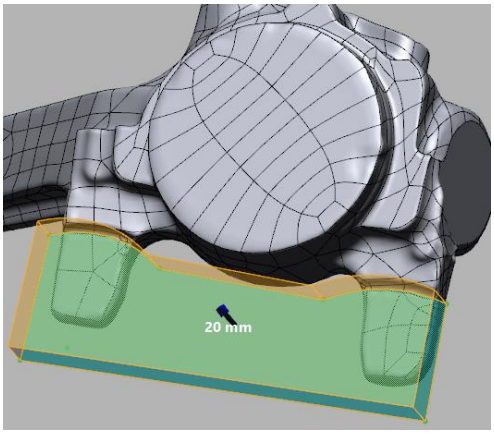


Figure 309

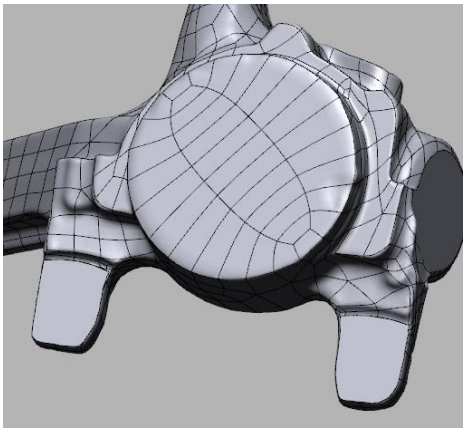


Figure 310

Make a Hole for Assembly

1. Select the **Vector** command located in **MODEL>Ref. Geometry** tab.

☞ *Create the axis of the Cylinder.*

- Select **Top** plane and **Right** plane for **Entities** and create a vector which will intersect both Planes.

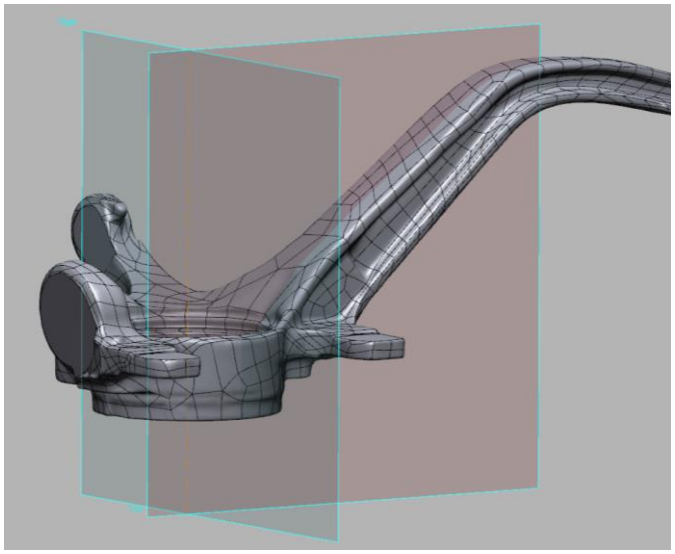


Figure 311

- Press the **OK** check button and finish **Vector**.

2. Click *Copied Mesh* in **Feature Tree**.

3. **Right-Click** in space, then select the **Mesh Sketch** icon.

- Select **Right** plane for **Base Plane**.
- Click the **Apply** check button.
- Draw sketch line as shown in **Figure 312**.

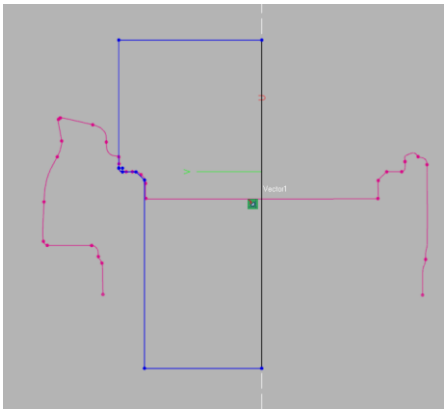


Figure 312

- Press the **EXIT** button and finish **Mesh Sketch**.

4. Select the **Revolve** command from **Create Solid** group located in **MODEL** tab.

- Select **Sketch7 (Mesh)** for **Base Sketch**,
- Select **Vector1** for **Axis**.
- Select the **Cut** check box in **Result Operator**.

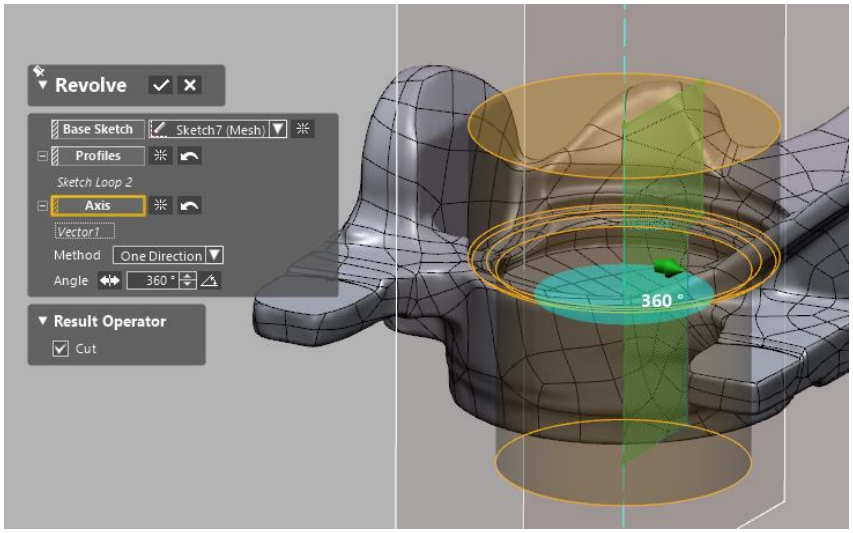


Figure 313

- Press the **OK** check button and finish **Revolve**.

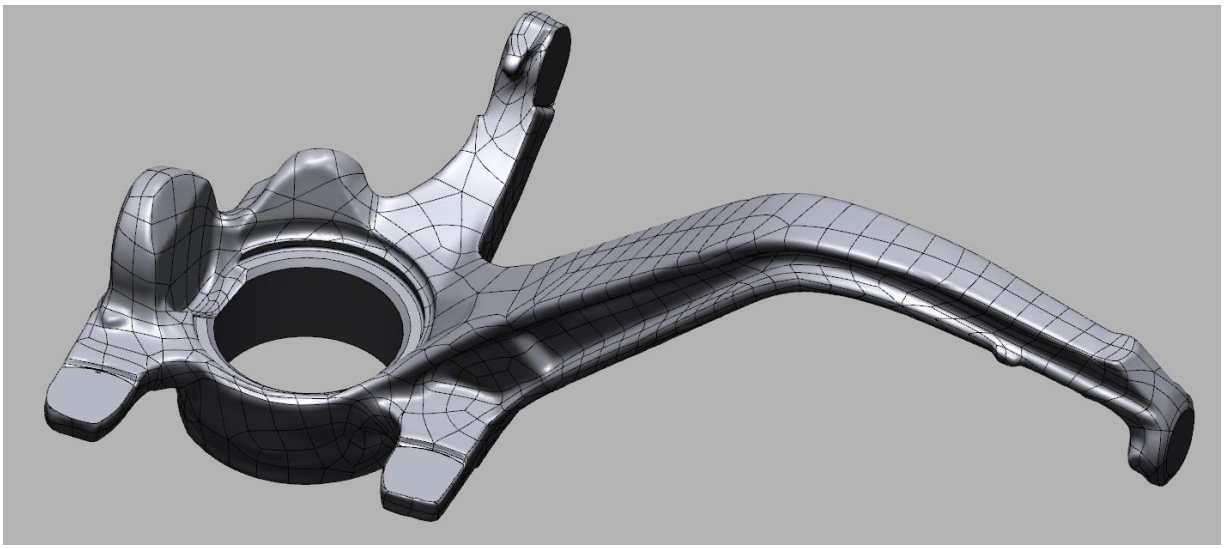


Figure 314

End of Activity

ACTIVITY: Impeller Workflow

Objective

Create an impeller using surfacing and solid tools.

Highlighted Commands

- **Revolving Silhouette Mesh Sketch Setup**
- **Loft Wizard**

Training File

10-Impeller.stl

Scan Preparation

1. Import training file **10-Impeller.stl**.

- Click the **Import** button or select **Import** from **Menu>Insert**.
- Use the Import Dialog to navigate to the training files folder. Choose **Impeller.stl** from the **Import dialog**.
- Click **Import only**. The file is loaded and displayed in the Model View.

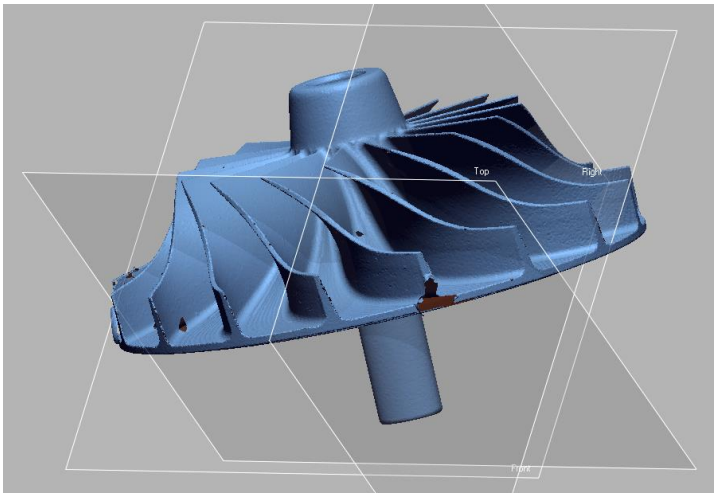


Figure 315

2. In the **POLYGON** tab, use the **Fill Holes** and **Smart Brush** command to clean and smooth a single vane of the impeller.

☞ *Assuming all blades are the same, only one will need to be edited for the modeling. The single modeled vane can be patterned.*

- With the **Repair Holes/Boss>Fill Holes** command, select all the holes on one vane to fill in with **Curvature**.
- Use the **Wizard>Smart Brush** command and the **Smooth** option to paint the specific areas that are rough.

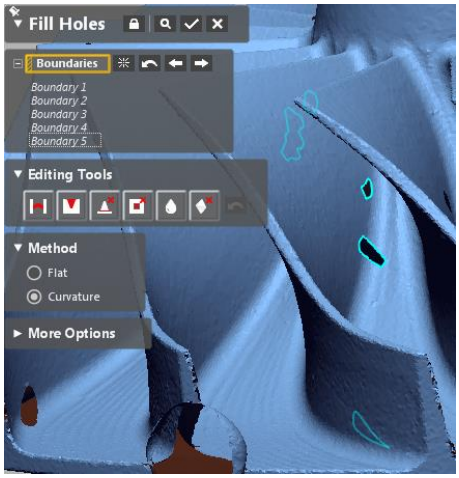


Figure 316

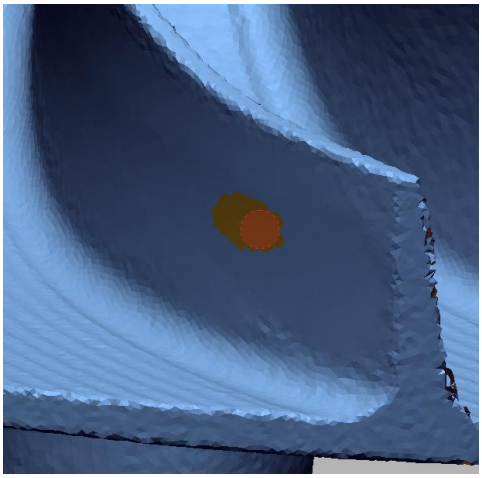


Figure 317

3. In the Tree dock, change the tab to **Viewpoint**.

- Orient the part to focus on the edited vane.
- Select the **Add Viewpoint** button to save the view.
- Toggle back to the **Tree** dock.

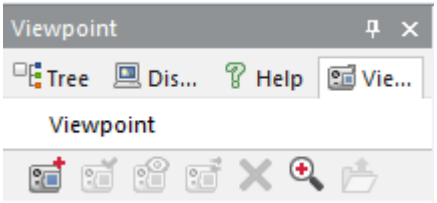


Figure 318

☞ *The Viewpoint can be saved for later use to be able to quickly move back to the edited vane.*

4. After editing the mesh, use the **Optimize>Decimate** command to reduce the triangle count.

- Select the **Allowable Deviation** method with a value of **0.01 mm** in the **Options**.
- Accept the command to see the new triangle count.

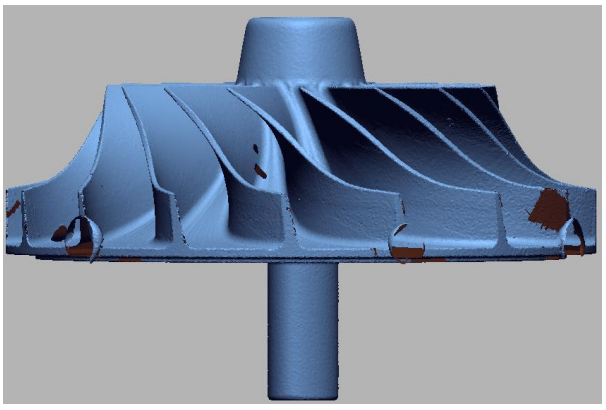


Figure 319

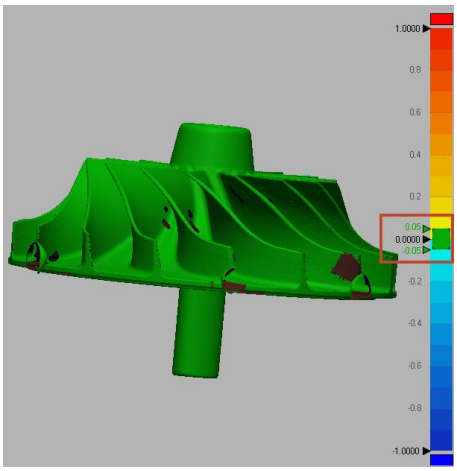


Figure 320

5. Select the **Auto Segment** command located in **REGION** tab.
   - Set the **Sensitivity** to **5** then accept.

6. Use the **Edit>Merge** command to merge the the regions on either side of the cleaned vane.
   - Hold **Shift** and select the multiple regions on a single face.
   - The **Merge** button is now active. Select.
   - Repeat process on other side face of vane.

Figure 321

Align To Coordinate System

1. Create **Ref. Geometry>Vector** to more accurately align the scan.
   - Select the **Revolving Axis** method.
   - **Select** all regions where the central axis would be the same as the center of the impeller.
   - **Preview** and then Accept the Vector.

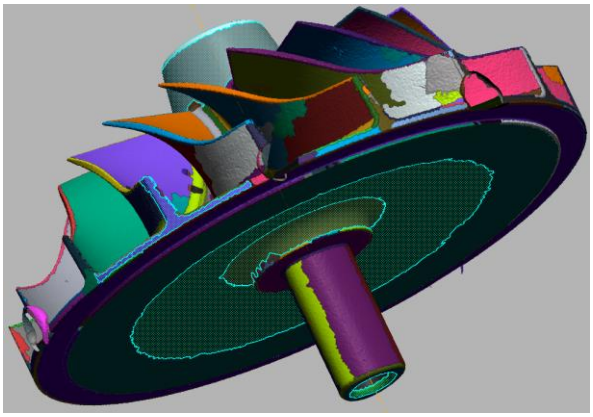


Figure 322

2. Align the scan to the coordinate system using **ALIGNMENT>Scan to Global>Interactive Alignment.**
   - Verify the mesh is selected then go to the **Next Stage**.
     - Change the **Method** to **XYZ**.
     - For **Position**, select the **Vector1** and a **Plane** Region on the bottom of the scan.
     - Set **Axis Z** to **Vector1**. If needed, Flip Direction of the positive axis.
     - Use the **Blue Circle** of the Manipulator to clock the part about the Z Axis. Rotate until the **X Axis** (Right Plane) is passing through the edited vane.
     - Check the alignment is in position on the right half of the screen then Accept.

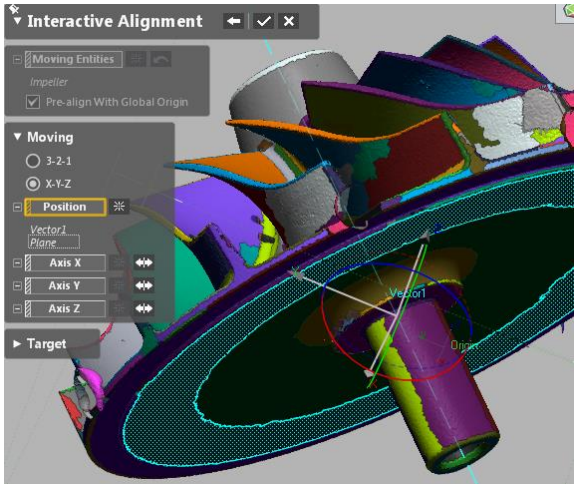


Figure 323

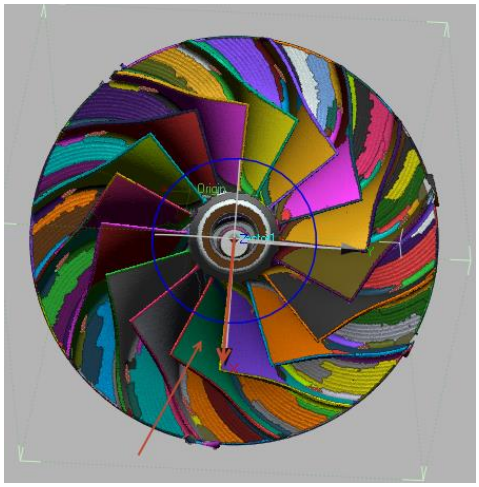


Figure 324

Main Body

1. Create a **Mesh Sketch** on the **Right** Plane.

- Change the method to **Rotational Method**.
- Use the **Offset Angle From Base Plane** or the Green Circle to rotate the base plane to find the best sketch angle for the main body of the impeller.
- Accept the sketch setup.

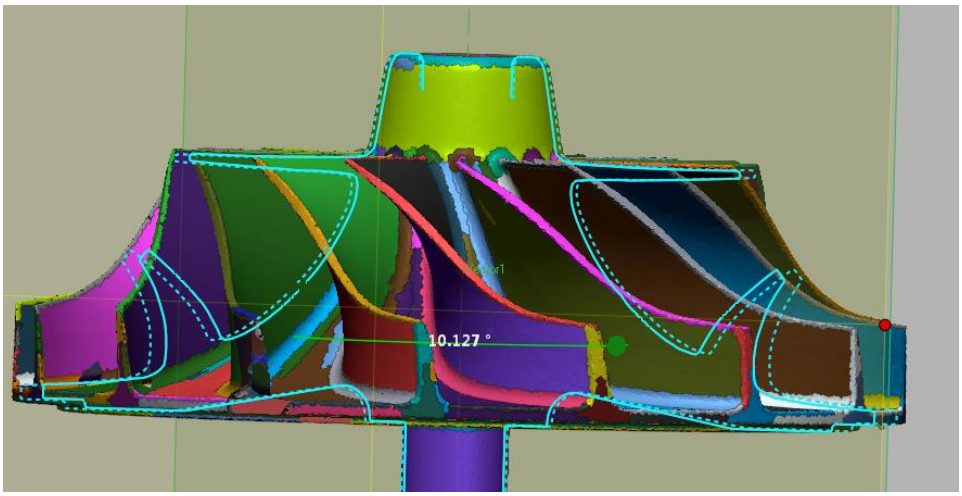


Figure 325

2. Sketch the inner profile shown in **Figure 326**.

- Use Arc and Line tools to create a sketch of the inside profile of the impeller.
- Exit the sketch when finished.

☞ *The bottom post is not rotational and does not need to be sketched.*

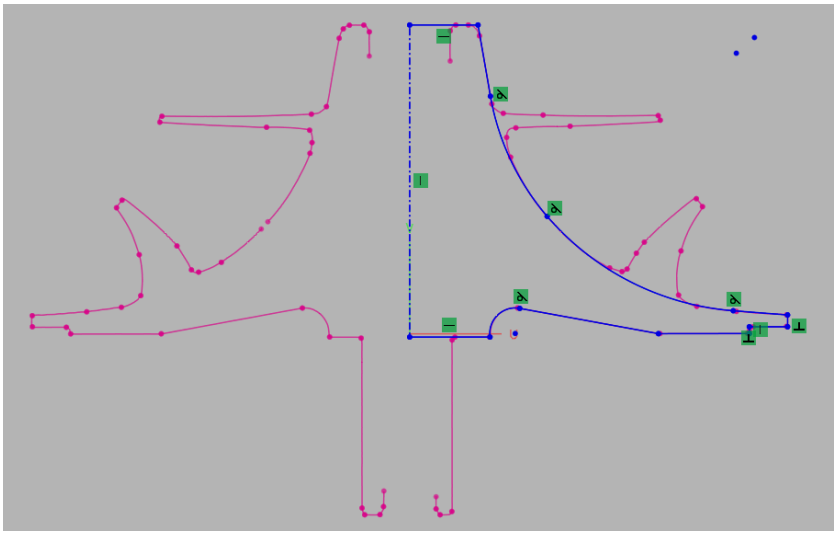


Figure 326

3. Create the body with **Create Solid>Revolve.**

- Set the previous sketch as the Base Sketch and Profile.
- Select the **Centerline** of the Sketch as the **Axis**.
- Accept the command.

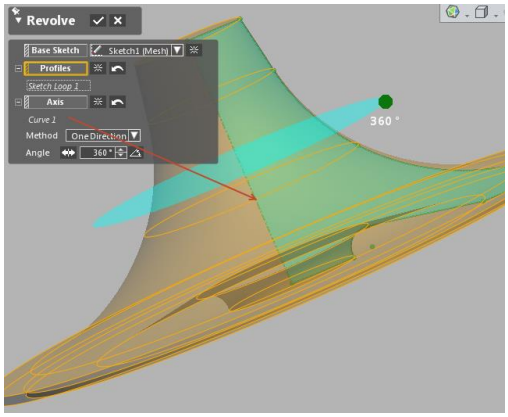


Figure 327

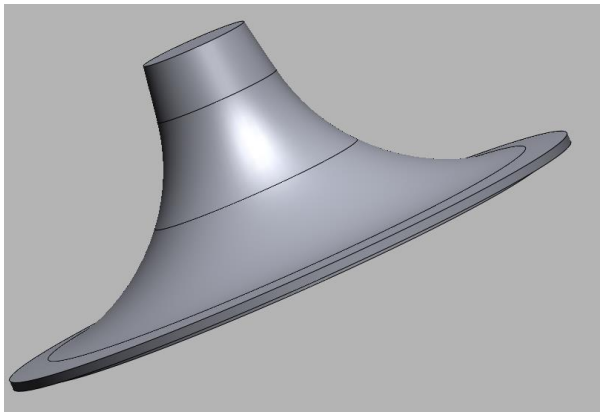


Figure 328

Vane

1. Create a surface on either side of the cleaned vane with a **Loft Wizard** or **Mesh Fit** command.

Option 1

- RMB on the region of one side of the cleaned vane scan and choose the **Loft Wizard** icon.
- In **Section**, select **By No. Of Sections**, then set the **No. Of Sections** to 7, and increase the smoothness.
- In **Profile Types**, select **Ref. Planes and Mesh Sketches**.

- Adjust the size of the bounding box of the surface with the red, green, and blue dots to be much larger than the mesh, then go to the **Next Stage**.
- Turn on the **Deviation From Body**.
- **LMB** and hold on the planes to readjust the position of the planes.
- **Ctrl+LMB** on a plane will add a new plane. Readjust the planes to reach the desired accuracy then accept the command.

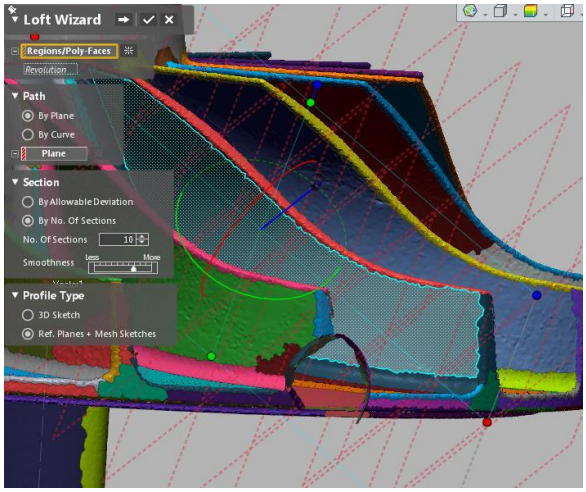


Figure 329

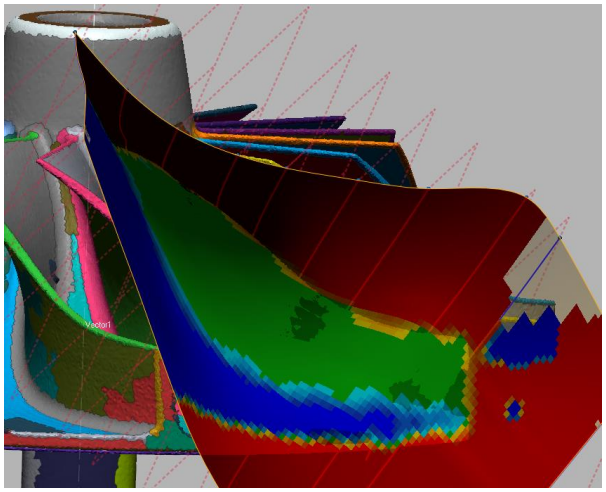


Figure 330

Option 2

- RMB on the region of one side of the cleaned vane scan and choose the **Mesh Fit** icon.
- Make any desired adjustments to the settings, or press the **OK** button to accept the command.

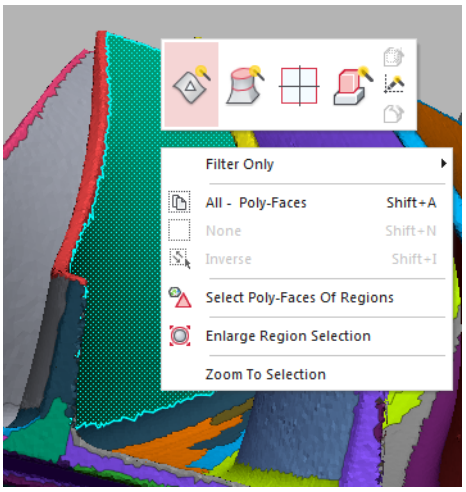


Figure 331

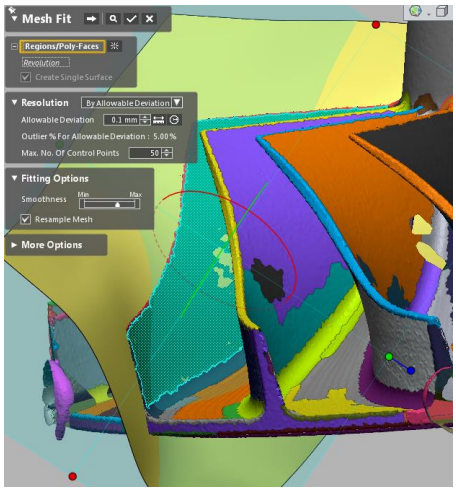


Figure 332

2. Repeat either option on the other side of the vane.
3. Edit any sketches created by the Loft Wizard to smooth and better the surface.
   - RMB on **Sketch 5(Mesh)**, or any other sketch created by the loft wizard, and choose **Edit**.
   - Select the generated spline.
   - In the **Rebuild Spline>No. Of Interp. Pts** box, enter **20** then press **Enter**.
   - Look at the spline, then edit the value again but to **10**.

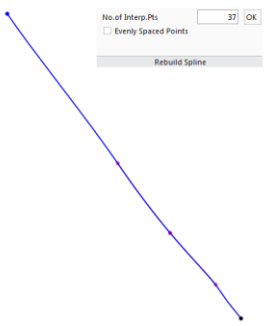


Figure 333

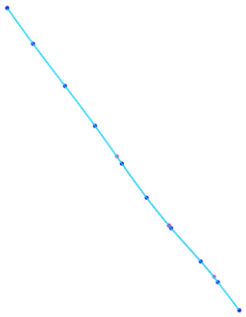


Figure 334

   - **Exit** the sketch to see how much smoother the surface is in one spot.
4. Repeat Step 3 on the other sketches as desired.

☞ *Reducing the spline points is one way to edit the loft wizard to change the resulting surface.*

☞ *Other ways could be to completely remove the spline and remake the sketch using lines and arcs.*

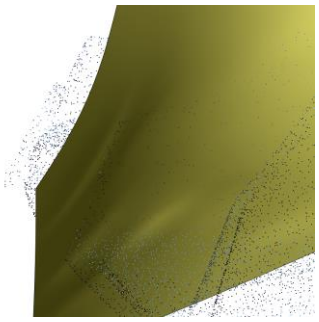


Figure 335- Original Surface

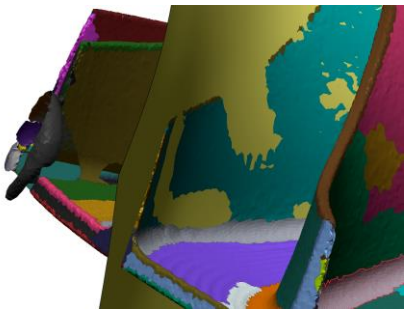


Figure 336- Edited Surface

5. Create a **Mesh Sketch** on the **Right** Plane to capture the outer profile of the vane.
   - Change the option to **Rotational Method** for the sketch setup.
   - **Pull** the Green Arrow or change the **Silhouette Range** to encompass the entire cleaned vane, then accept the command.

☞ *The Rotational Silhouette Range will create a wedge to contain the desired area, and then project by rotating about an axis.*

☞ *The **Offset Angle** can also be adjusted for a better cross section.*

   - Create the sketch of the outside vane profile using Line and Arc tools.
   - Use **Figure 338** to create the sketch then Exit.

☞ *An open profile sketch can be used only for surfaces. Close the sketch to create a solid revolve.*

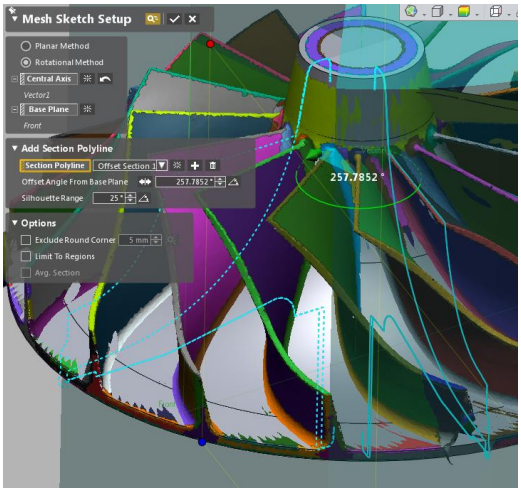


Figure 337

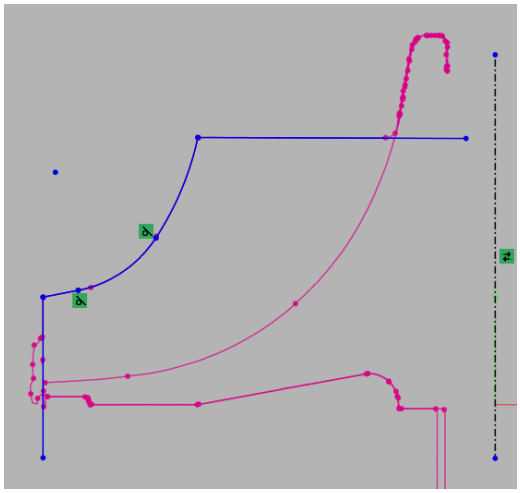


Figure 338

6. Use the **Create Surface>Revolve** to revolve the surface the necessary amout to enclose the surfaced vane.

☞ *The Sihlouette command will place the sketch near the vane or on the opposite side of the impeller.*

☞ *A small rotation in both directions could be needed, or a large rotation to encompass the entire vane.*

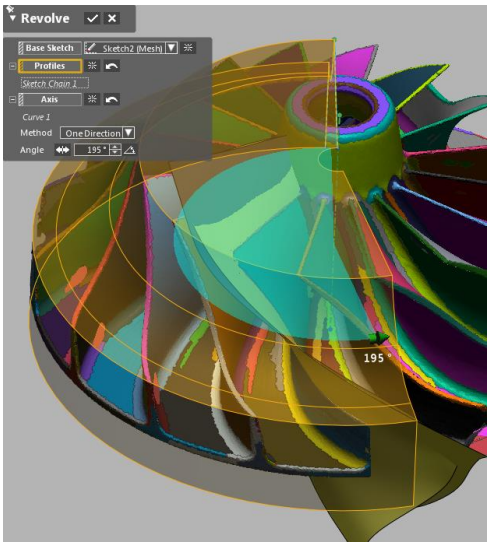


Figure 339

7. Create an **Edit>Surface Offset** of the original rotated body to completely enclose an impeller vane.

- Select all faces along the top of the solid.
- Change the **Offset Distance** to **0 mm** then accept.

☞ *An offset at 0 is duplicating and creating a surface on the solid face to use in more commands.*

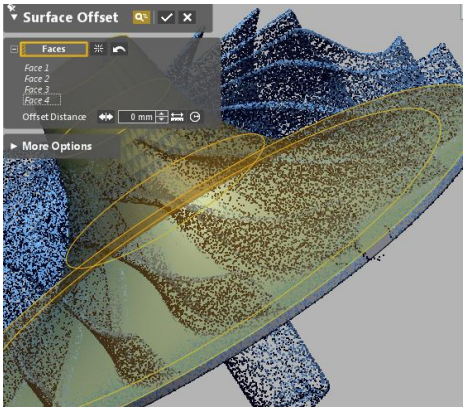


Figure 340

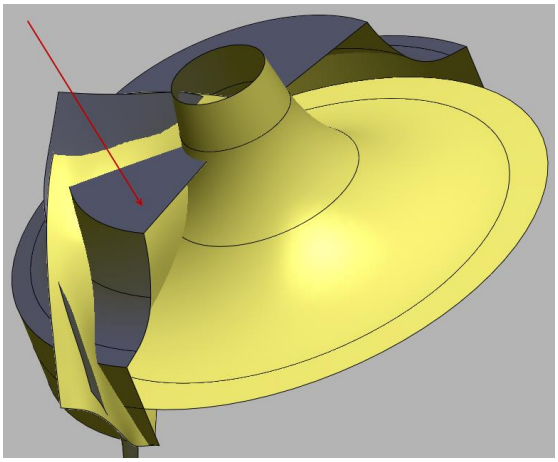


Figure 341– Enclosed Vane

8. Select **MENU>Insert>Surface>Solidify** to create a solid from the selected surfaces.

- Select the four surface bodies needed to enclose a vane and accept the command.

☞ *If an error occurs or a surface does not extend fully through the other surfaces, use the **Extend Surface** tool to lengthen the surface before repeating the Solidify command.*

- Check the Visibility of the Solid Bodies is on to view the main body and single vane.

9. Use the **MODEL>Edit>Fillet** command to place a small fillet on the top edges of the vane.

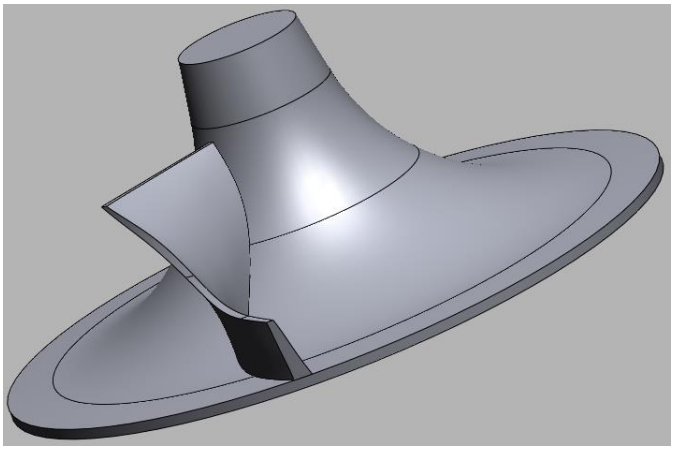


Figure 342

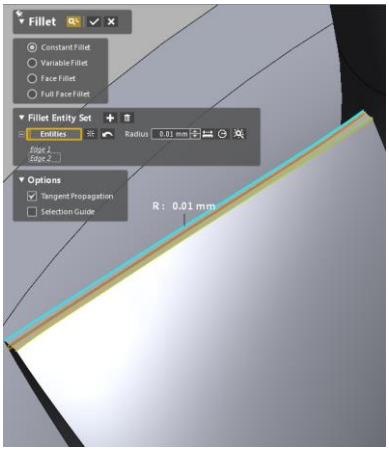


Figure 343

10. Select **Pattern>Circular Pattern** to create the remaining vanes.

- Choose the Vane as the **Bodies**.
- Select **Vector1** as the **Rotation Axis**.
- Set the **No. Of Instances** to **16**.

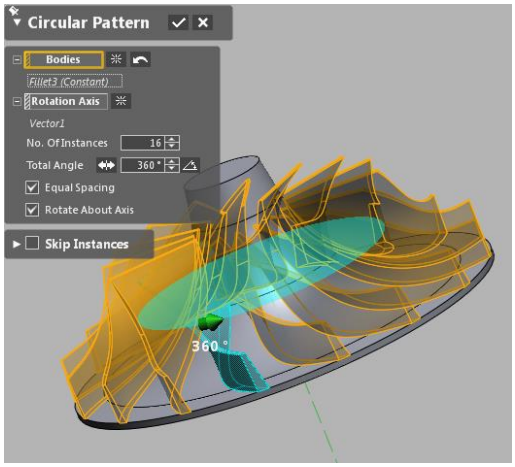


Figure 344

Edits and Shaft

1. Merge all solid bodies to one body with the **Edit>Boolean** command.

- Set the **Operation Method** to **Merge**.
- Select all 17 solid bodies as the **Tool Bodies** then accept to have a single solid body remaining.

2. Create the lower shaft using a **Mesh Sketch** on the **Front** Plane or any plane normal to the shaft.

- Offset the **Offset Distance From Base Plane** to cross through the triangular shaft and accept the set up.
- Use **Arc** and **Fillet** tools to create the sketch then **Exit** when completed.

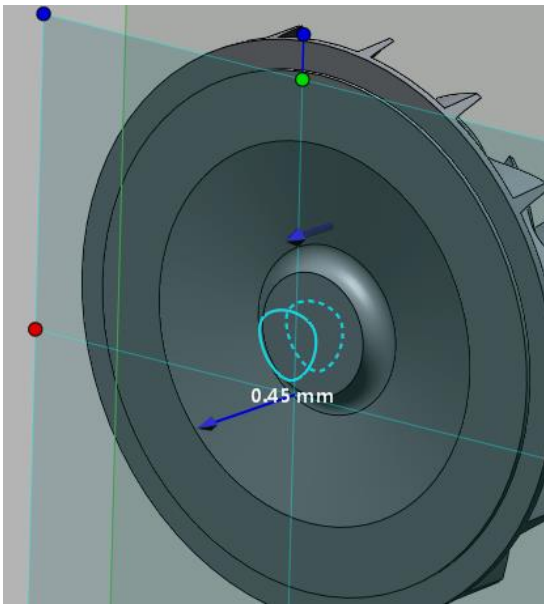


Figure 345

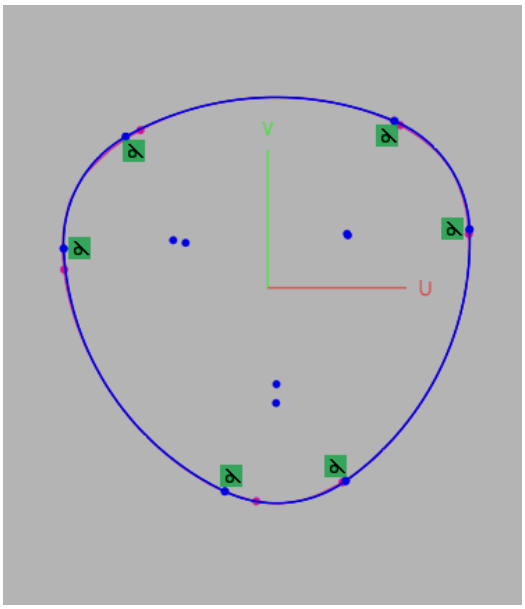


Figure 346

3. **Create Solid>Extrude** the sketch to create and merge the shaft.

- Use the **Up to Region** method with the **Max Position** sub method.
- Select the lower region of the scan shaft.
- Check the **Merge** option is selected under **Result Operator**, then Accept.

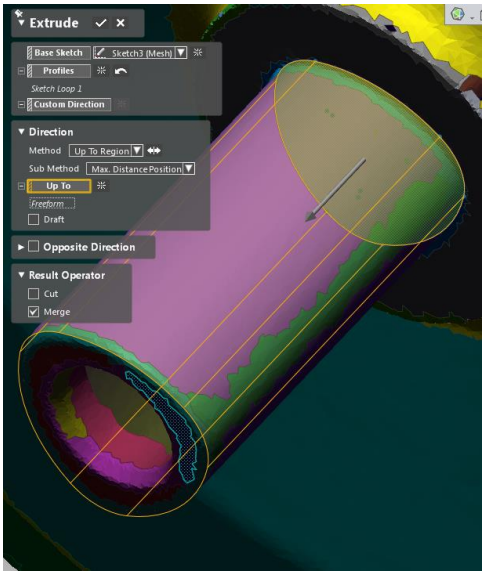


Figure 347

4. Create a **Mesh Sketch** on the top face of the shaft to cut out the top hole.

- Select the top solid face as the **Base Plane**.
- **Offset** the sketch profile down slightly to see the inside circle then accept the Mesh Sketch Setup.
- Sketch the inside **Circle** then **Exit** the sketch to cut out the hole.

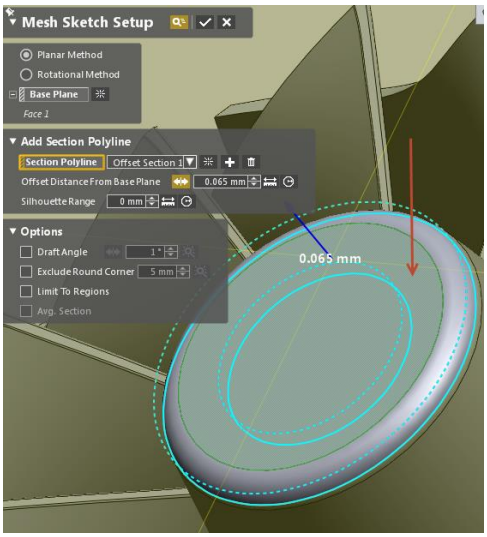


Figure 348

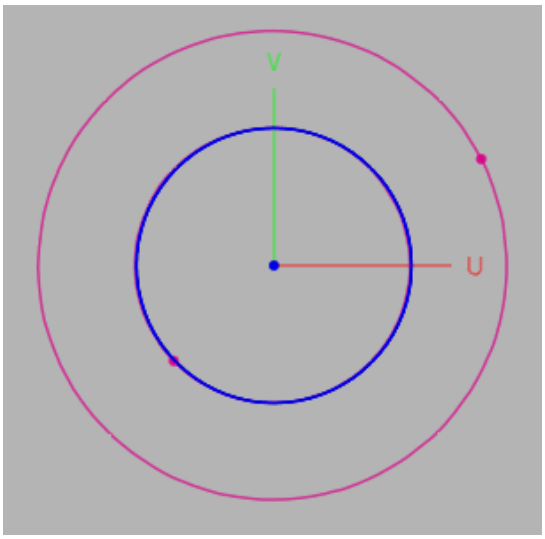


Figure 349

5. Cut out the hole with the **Create Solid>Extrude** command.

- Set the Method to **Blind** and cut down about **25 mm** in to the mesh.
- Set the **Result Operator** to **Cut**.
- **Accept** the command.

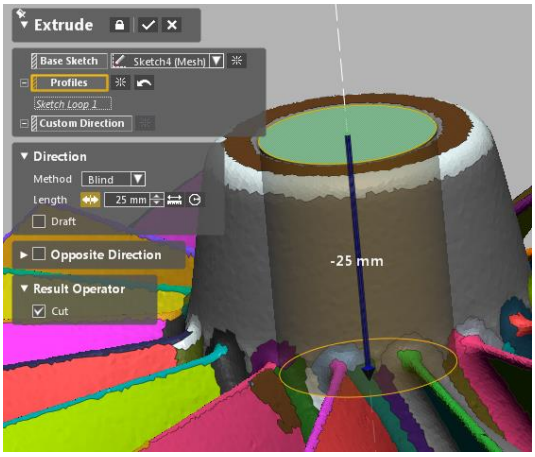


Figure 350

6. Repeat process for lower hole in shaft.

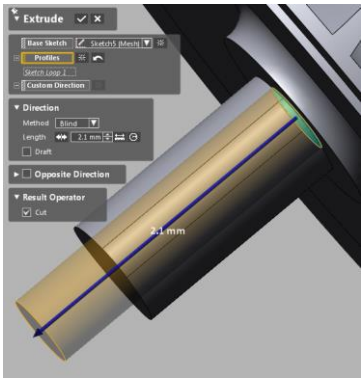


Figure 351

7. Add **Fillets** where needed on the model.

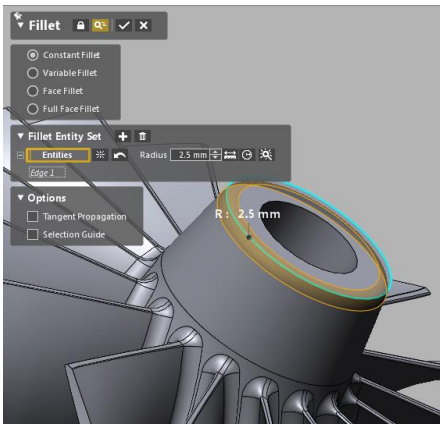


Figure 352

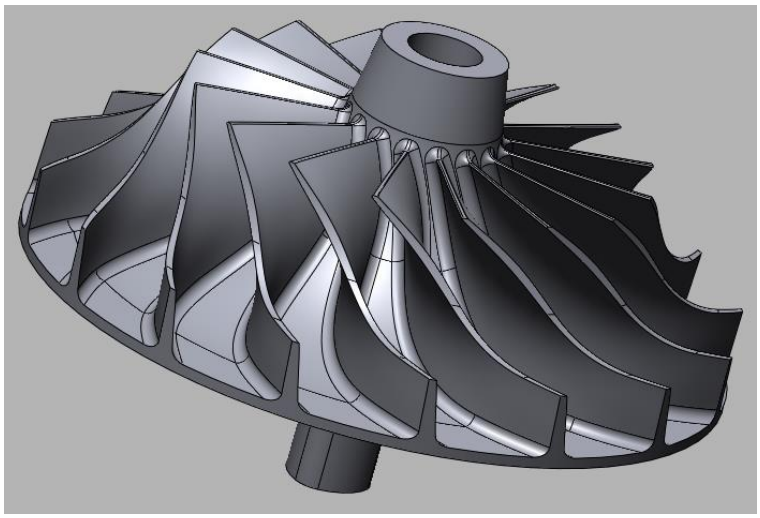


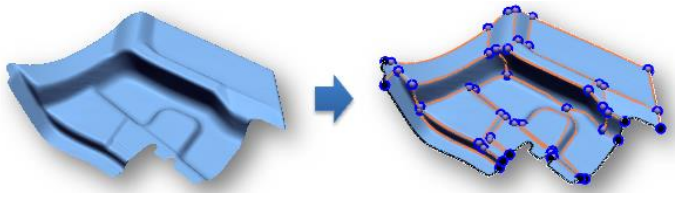
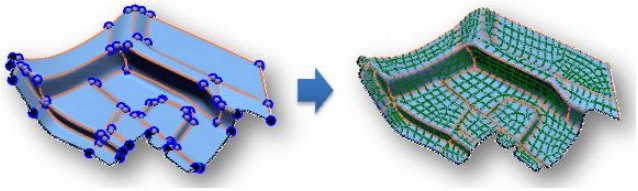
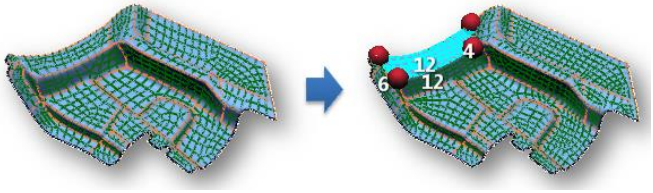
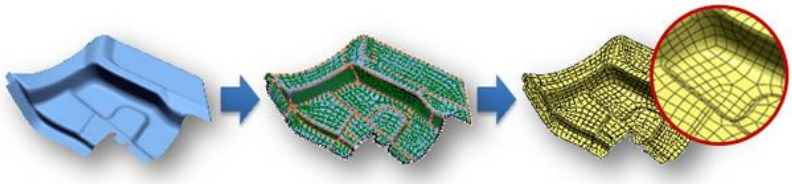
Figure 353

End of Activity

Exact Surfacing

Exact Surfacing is a workflow designated to create a very accurate surface on a complex scan. The new SURFACING toolbar walks through the steps needed to create a good patch network to generate the surfaces.

The Create/Edit Patch Network group in the SURFACING tab has all the tools needed to create the surface.

Step 1: Extract Contour Curves –Detect and automatically extract 3D contour curves for the area of high curvature on a mesh. These curves can be edited and adjusted manually to create a better Patch Layout.	 **Figure 354**
Step 2: Construct Patch Network – Automatically construct the patch network within the patch layout.	 **Figure 355**
Step 3: Shuffle Patch Groups – Reorient the patches within the panels in a 3D patch network for better continuity.	 **Figure 356**
Step 4: Fit Surface Patches- Create NURBS surfaces inside each patch of the 3D patch network. An accurate freeform surface body will be created.	 **Figure 357**

ACTIVITY: Exact surface

Objective

Learn about exact surfacing.
This exercise converts a polygon object to an exact surfaces object and activities the remainder of the exact surfaces tab. This workflow generates surfaces match the object exactly as it is constructed.

Highlighted Commands

- **Extract Contour curves**
- **3D curves**
- **Construct Patch Network**
- **Shuffle Patch Groups**
- **Fit Surface Patches**

Training File

12-Exact Surface.xrl

Import file

1. Open training file **11-Exact Surface.xrl**
   - Click the **Open** button or select **Open** from **Menu>File>Open**.
   - Use the Open Dialog to navigate to the training files folder. Choose **11-Exact Surface.xrl** from the **Open dialog**.
   - Click **Open**. The file is loaded and displayed in the Model View.

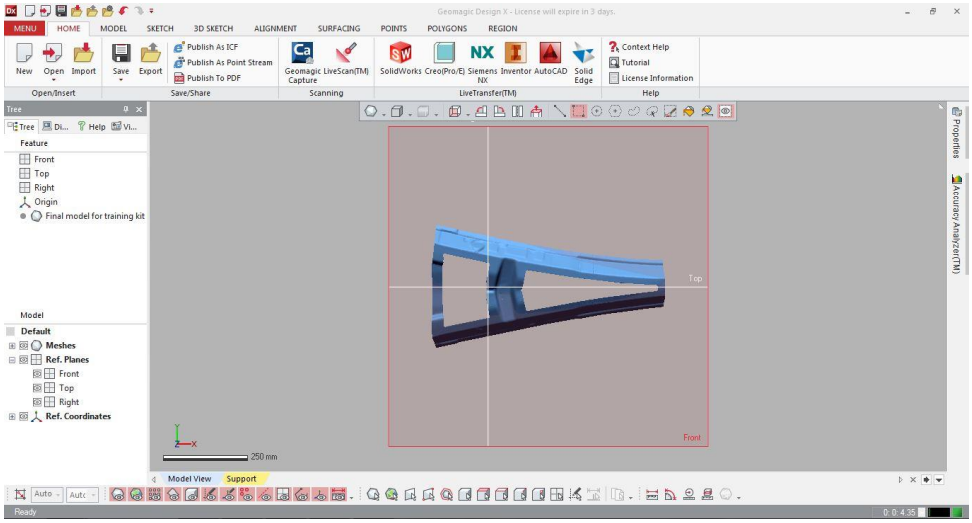


Figure 358

Extract counter network

1. Go to the **SURFACING** tab to begin the Exact Surfacing workflow. the Extract Contour Curves and set the curvature Sensitivity Potion to 60 and then click to next stage
   - Select the **Extract Contour Curves** command in the Create/Edit Patch Network group.
   - Set the **Curvature Sensitivity Position** to **60** then select the **Next Stage** button.

☞ *The Extract Contour Curves command automatically places red separators between regions of relative flatness. The Next Stage allows adjustment of those region separators and then places 3D contour lines inside the regions.*

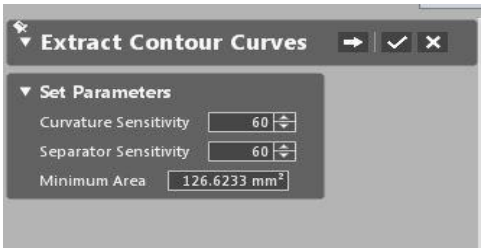


Figure 359

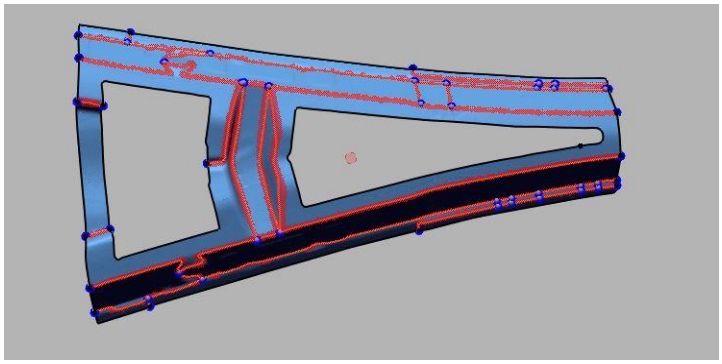


Figure 360

2. In the Next Stage, click the **Enable Real Time Preview** option to have live feedback of the editing of the separators.
   - Use the **Paint Brush Selection Mode** to add, remove, and clean regions.

☞ *Best practice will be to extend all separator regions to a scan boundary, similar to **Figure 457.***

   - Edit the Contour Curves regions as shown in **Figure 362**.

☞ *Hold **Ctrl** when selecting to remove separator regions.*

☞ *Other selection modes, such as the Line, Rectangle, and Circle can be used to edit separators.*

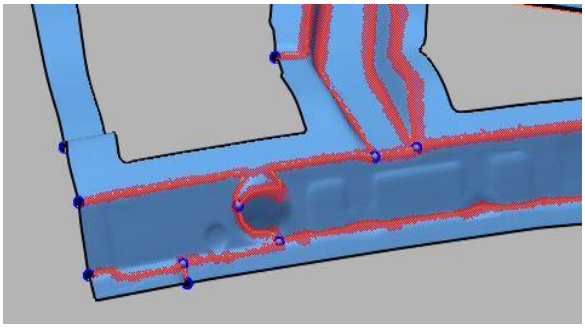


Figure 361 - Before Editing

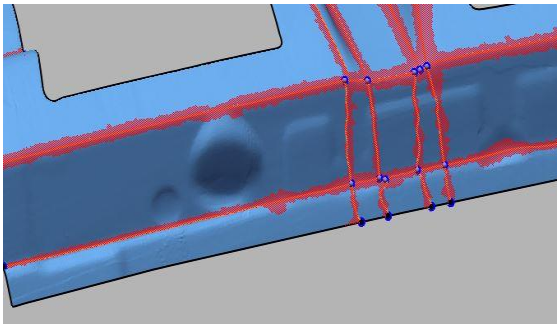


Figure 362 - After Editing

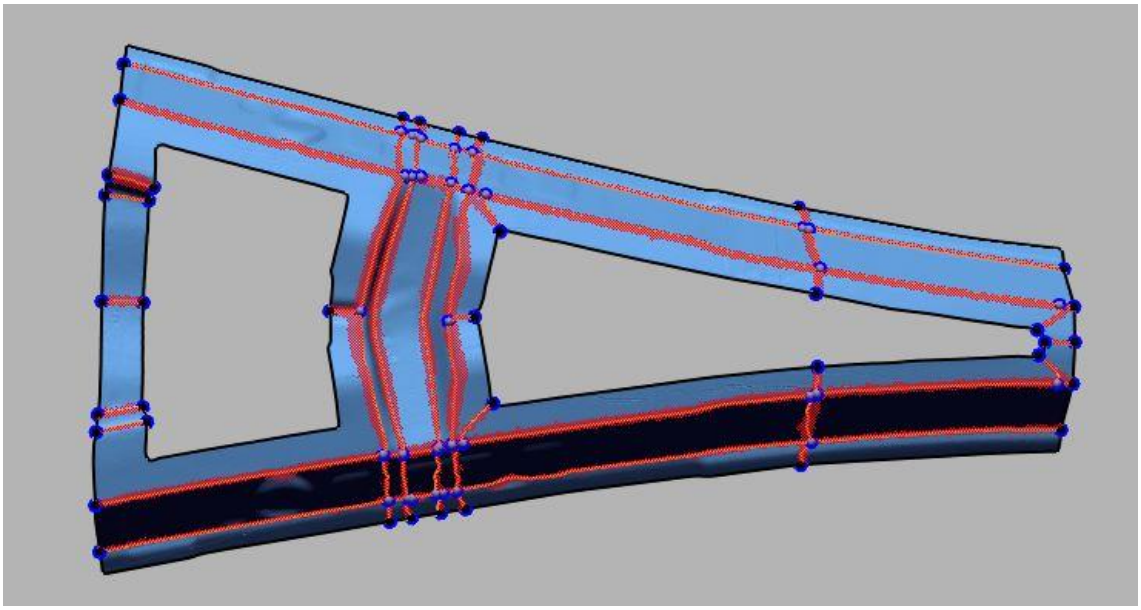


Figure 363 – Edited Separators

Edit Contour Curves on a Mesh

1. The generated contour curves are now a 3D Sketch. Go to the **3D SKETCH** tab to edit.
   - Select all of the Contour Curves by pressing **Ctrl+A** or box selecting the part.
   - In the ribbon, find the **Rebuild Toolbar**. Change the value of **No. of Interp. Pts** to **10** then click the **OK** button.

☞ *Each curve will now have 10 points; therefore, the contour curves are more easily controlled.*

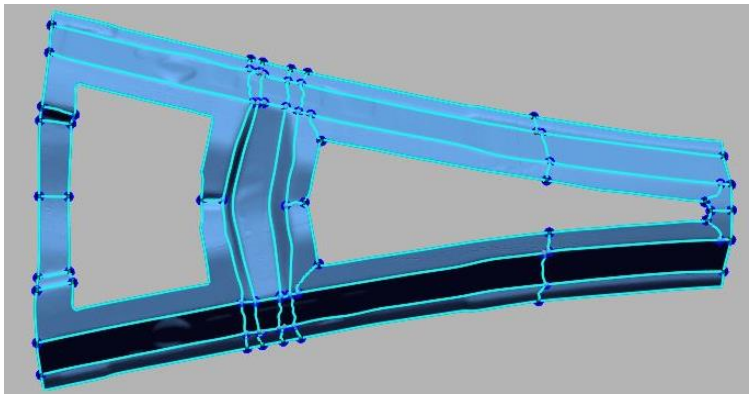


Figure 364

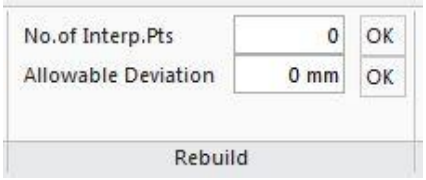


Figure 365

2. Select a junction point on the contour curves and swing the point back and forth so that the connected curves are smoothed.
3. Adjust all contour curves to be continuous and four sided. The **Edit>Trim** tool can remove extra segments.

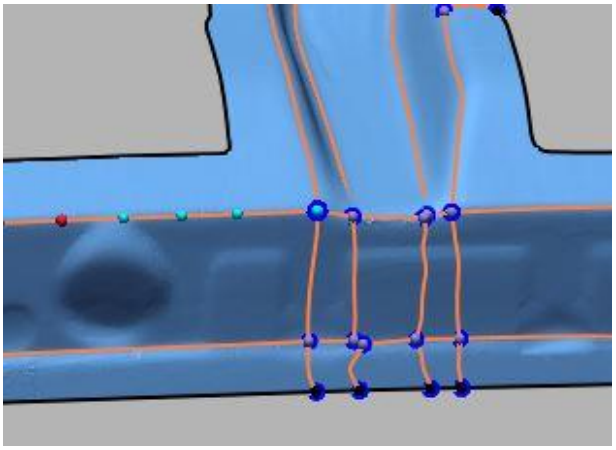


Figure 366 - Before Editing

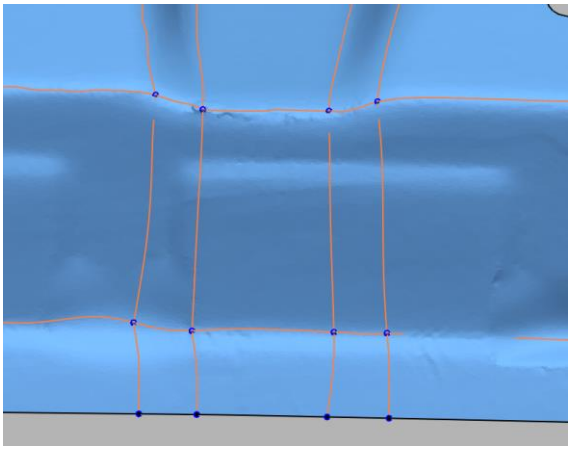


Figure 367 - After Editing

4. Select all of the Contour Curves and **Rebuild** them again.
   - In the ribbon, find the **Rebuild Toolbar**. Change the value of **No. of Interp. Pts** to **20** then click the **OK** button.
5. Verify all lines are straight and the patches seem continous.

☞ *Each of these four sided figures is a panel. The panels will dictate the layout of the patch network.*

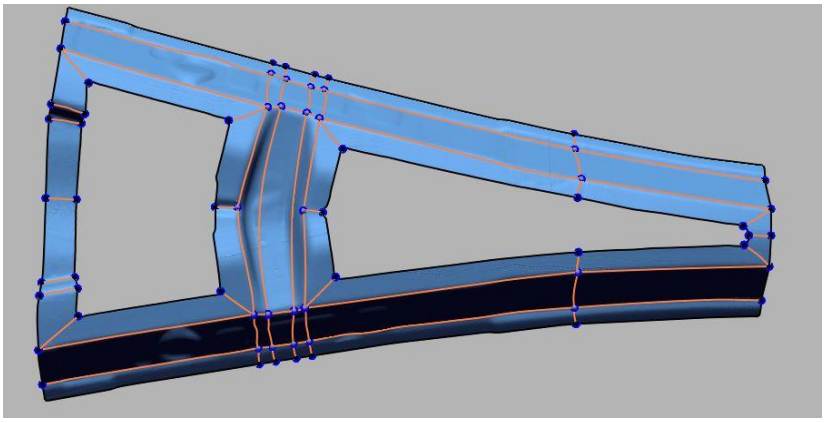


Figure 368

Constructing a 3D Patch Network

1. Allow the software to automatically construct a patch network from the panels created previously with the **Construct Patch Network** step. This is found in the **Create/Edit Patch Network** group.

☞ *Note that Construct Patch Network command can be found in the 3D SKETCHING and the SURFACING tabs. In both of these tabs, the command is located within the same group.*

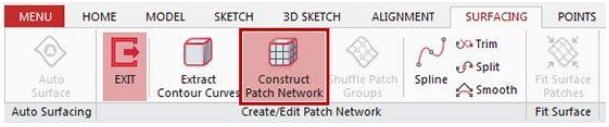


Figure 369

- Choose **Auto Estimate** option then **Accept**.

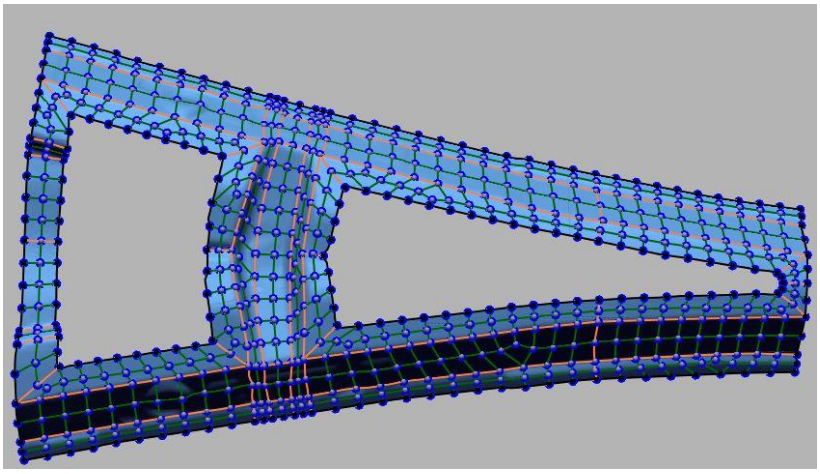


Figure 370

Edit 3D Patch Network

1. In the 3D SKETCH or SURFACING tab, choose the **Create/Edit Patch Network>Shuffle Patch Groups** command.
   - Click on a panel as shown in the **Figure 371**.
2. Select **Add/Del 2 Paths** in the **Action** roll up, and select the **Auto Fill Neighboring Panels** option in the **More Option** roll up.

☞ *The Auto Fill will match the number of paths in the adjacent panel when paths are added or removed.*

   - Select a point along the lower side of the panel to increase the number of paths along the edge until the edges match.

☞ *To decrease the number of paths in a panel, hold **Ctrl** when clicking on the panel edge.*

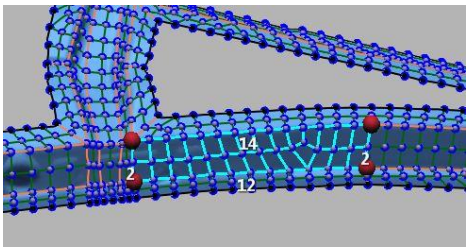


Figure 371

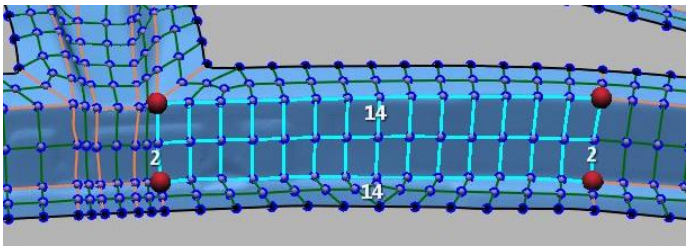


Figure 372

3. Click the **Deselect Current Planel** next to **Action** in the top roll up to clear the previous selection and make choose a new panel to edit.
   - In the **More Option** roll up, click the **Next** arrows to navigate through patch panels and find the ones needing rearranging.
4. Select a panel that needs to be shuffled and repeat steps 2 and 3 until all panels have equal patch sides.

☞ *While the define option is selected in **Action**, check the vertices of each panel is properly placed in each corner of the panel. If not, click the vertex to redefine and Accept.*

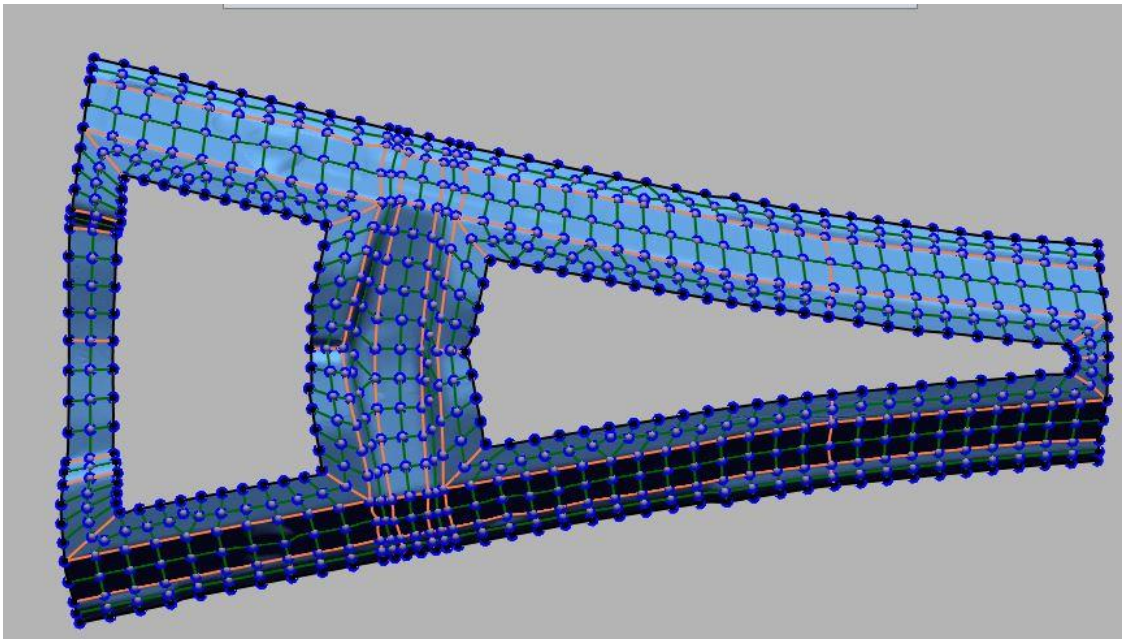


Figure 373

Create a NURBS Surface

1. In the SURFACING tab, choose the **Fit Surface>Fit Surface Patches** command.

- Select the **3D Sketch1(Mesh)** as the **Mesh Curves** base.
- Select the **Adaptive** option as **Fitting Method**.

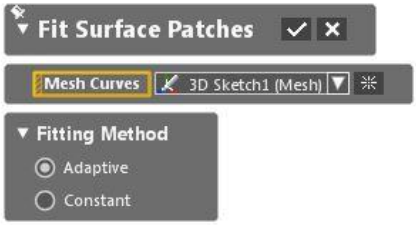


Figure 374

- Adjust the slider of the **Geometry Capture Accuracy** option to one notch away from **Max** to improve the accuracy of the fitting surface patches.

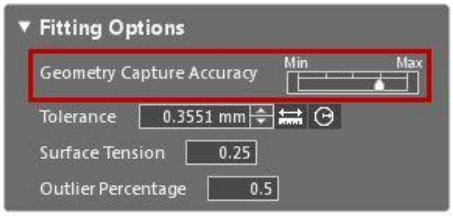


Figure 375

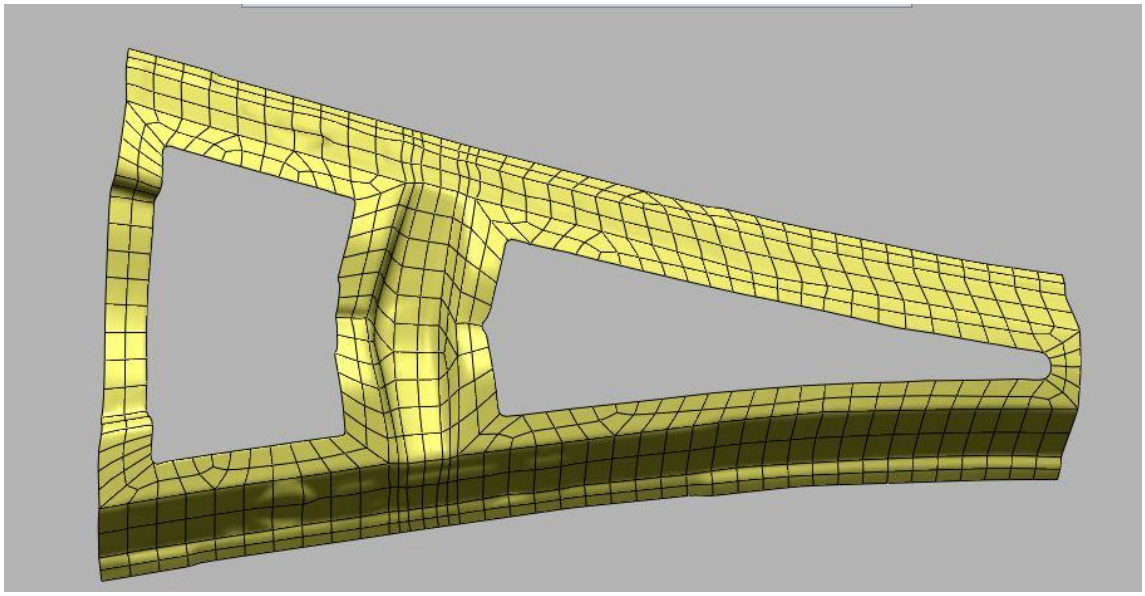


Figure 376

End of Activity

ACTIVITY: Self Exercise

Objective

Create a propeller using the skills learned throughout the training course.

Highlighted Commands

- **Interactive Alignment**
- **Mesh Sketch – Silhouette**
- **Loft Wizard/Mesh Fit**
- **Revolve**

Training File

12-SelfExercise.stl

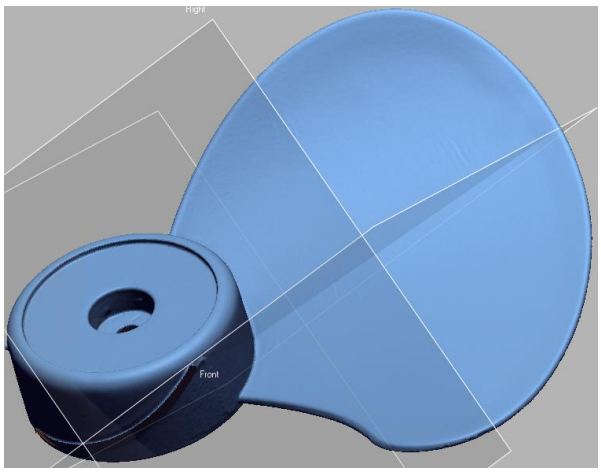


Figure 377

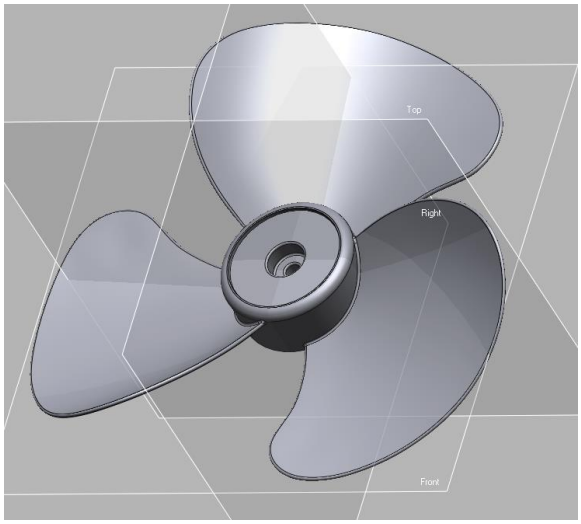


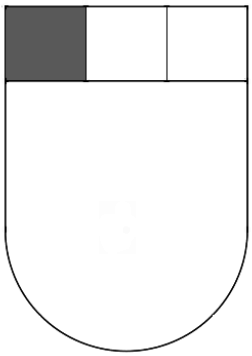
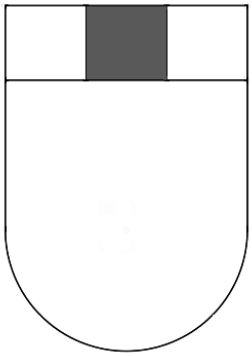
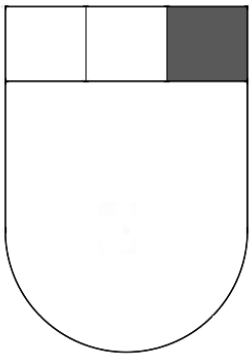
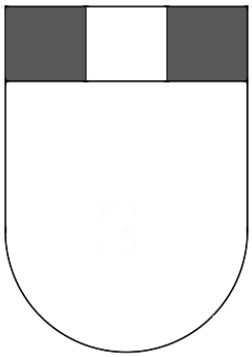
Figure 378

End of Activity

Appendix I – Mouse and Keyboard Control

Mouse Control

The default Right-Hand Mouse is shown.

	**LMB**	◉ Selection Button ○ Change tabs in ribbon ○ Activate commands ○ Select and activate entities ○ Make individual or box selections
	**MMB**	◉ Scroll Wheel ○ Zoom in and out of center of screen. Roll the mouse up and away to increase magnification; roll the mouse down to decrease magnification. ◉ Toggle Button ○ Activate second level of mouse buttons. Most buttons will rotate the part.
	**RMB**	◉ Rotate Button ○ Hold to rotate the view of the part about the center of the screen. ◉ Context Menu ○ Activate common commands based on the entity selected. ○ Accept and Exit Commands
	**LMB and RMB**	◉ Pan ○ Move the part laterally on the screen.

Shortcut Keys

Listed in the table are the commonly used default **Shortcut Keys**. These shortcut keys will allow quicker access to certain functions quickly without selecting the function from the Tool Bar.

Command	Shortcut Key
Menu	
New	**Ctrl+N**
Open	**Ctrl+O**
Save	**Ctrl+S**
Select All	**Ctrl+A, Shift+A**
Select Inverse	**Shift+I**
Undo	**Ctrl+Z**
Redo	**Ctrl+Y**
Repeat Last Command	**Ctrl+Space**
View	
Zoom Fit	**Ctrl+F**
Meshes Visibility	**Ctrl+1**
Regions Visibility	**Ctrl+2**
Point Cloud Visibility	**Ctrl+3**
Surface Bodies Visibility	**Ctrl+4**
Solid Bodies Visibility	**Ctrl+5**
Sketches Visibility	**Ctrl+6**
3D Sketches Visibility	**Ctrl+7**
Ref. Points Visibility	**Ctrl+8**
Ref. Vectors Visibility	**Ctrl+9**
Ref. Planes Visibility	**Ctrl+0**
Normal To	**Ctrl+Shift+A**

Appendix II - Mesh Editing

ACTIVITY: Mirroring scans

Objective

This training course is designed to show how to mirror half a scan and edit the mesh of a bottle scan to prepare it for either 3D printing or modeling.

Highlighted Commands

- **Mirror**
- **Thicken**

Training File

Bottle.stl

Import Scan Data

1. **Import** the **Scan Data** found at:
(~/Appendix/Bottle.stl)

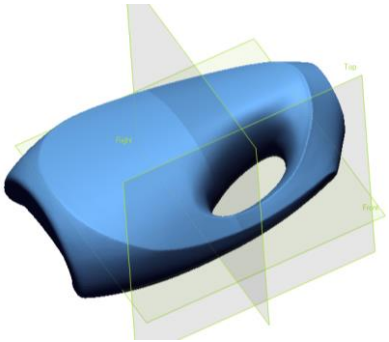


Figure 379

Add a Reference Plane

1. Hold down the **Ctrl** button and click-drag the **Right** plane to the neck of the bottle by **105mm**.

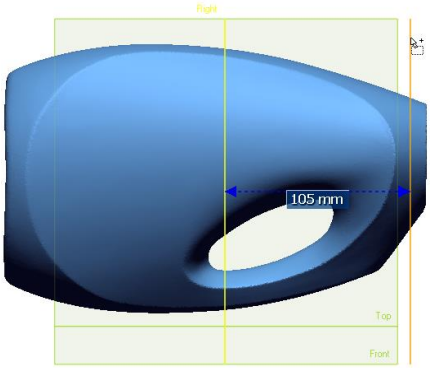


Figure 380

2. Click the **Visibility** icon next to the **Top Plane** and **Right Plane** entities in the **Model View** to hide them, leaving on the **Front Plane** and **Plane1** visible.

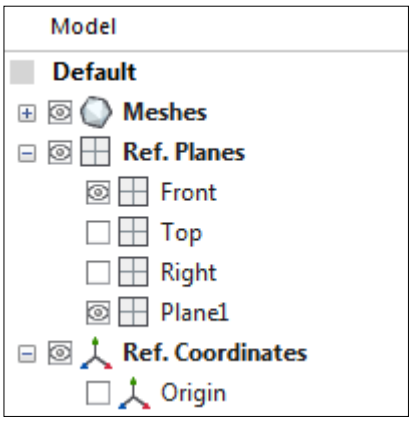


Figure 381

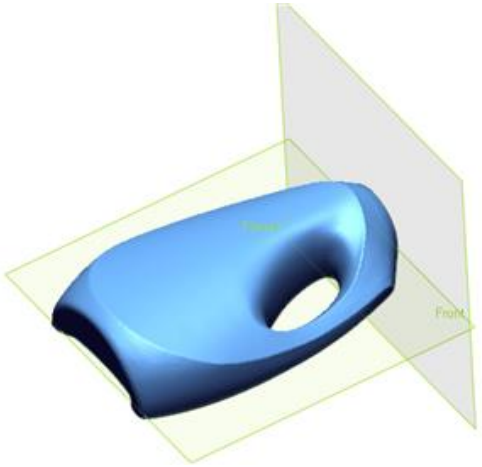


Figure 382

Splitting the Mesh

1. Click the **POLYGONS>Edit>Split** button in the ribbon.
   - Select the **By Entity** method.
   - Choose **Plane1** as the **Tool Entity**.
   - Do not use the **Cap Section Ends** option and toggle the **Don't Quit Command With OK** button to turn it off.

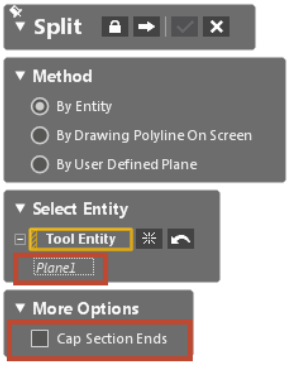


Figure 383

2. Click the **Next Stage** button.

3. Select the region to remain and click the **OK** button.

☞ *Make sure to select the larger section of the bottle and de-select the smaller section.*

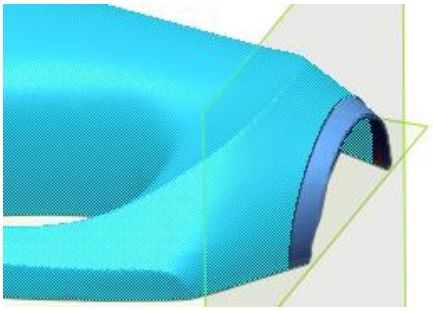


Figure 384

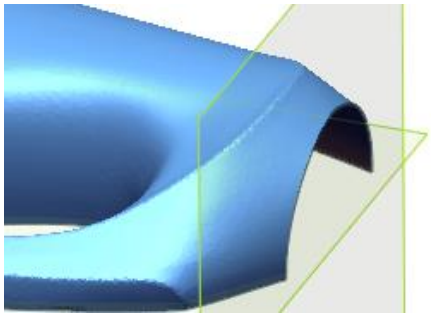


Figure 385

Mirror and Thicken

1. Click the **Mirror** button in the **MENU>Tools>Mesh Tools>Mirror** option.
   - Select the **Front** base plane as the **Position**.
   - Enable the Trim & Merge with the Remesh Poly-Faces Around Mirror Plane option enabled.

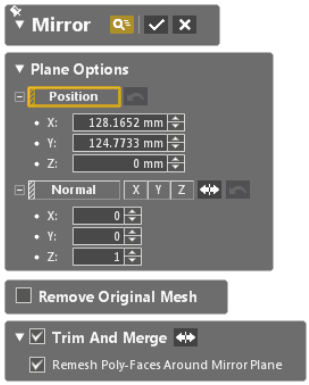


Figure 386

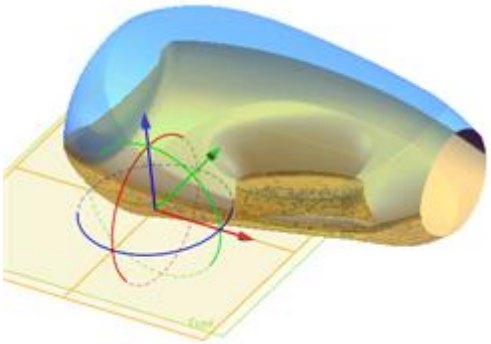


Figure 387

2. Click the **OK** button. The mesh will be identically mirrored across the **Front** base plane.

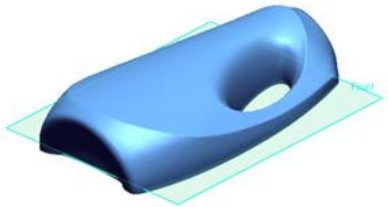


Figure 388

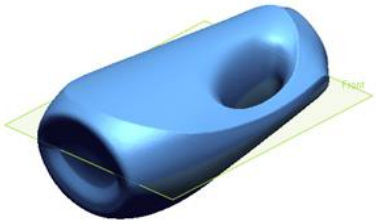


Figure 389

3. Click the **POLYGONS >Edit>Thicken** button on the ribbon. This command creates a constant thickness along the walls of the model.
   - Choose the **Surface** method, set the **Thickness** to **2mm** and choose the **Direction2** option.

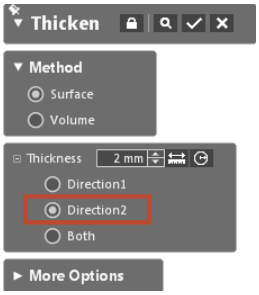


Figure 390

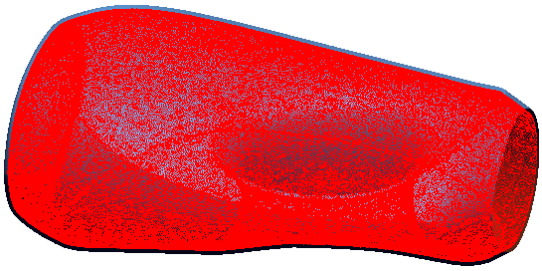


Figure 391

4. Click the **OK** button and the mesh will now have a thickness of **2mm**.

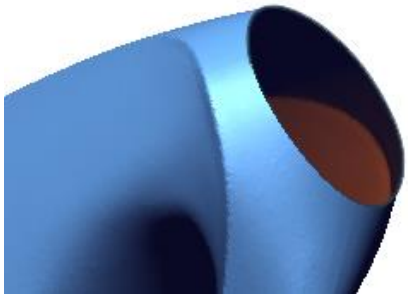


Figure 392

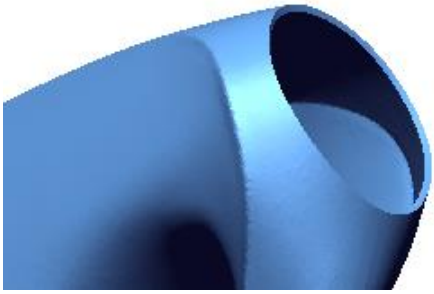


Figure 393

5. The mesh of the bottle is now complete and thickened and ready for the modeling process.

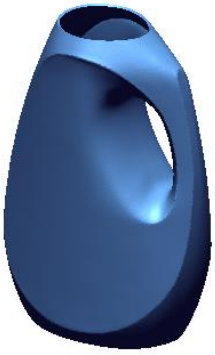


Figure 394

End of Activity

ACTIVITY: Global Mesh Enhancing

Objective

This training course is designed to show how to optimize and enhance messy scan data.

Highlighted Commands

- **Optimize Group**
- **Decimate**
- **Enhance Shape**

Training File

Grasshopper.stl

Import

1. **Import** the **Scan Data** found at:
(~/Appendix/Grasshopper.stl)

Figure 395

Hide Reference Planes

1. In this training the **Ref. Planes** will not be utilized, so they can be hidden from the **Model View** by pressing the **Ctrl + 0** keyboard shortcut or by clicking the visibility icon next to the **Ref. Planes** entity in the **Model Tree**.

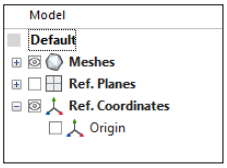


Figure 396

Change Mesh Color

1. Click the **Right Mouse Button (RMB)** on the **Grasshopper** entity in the **Model Tree** and select **Properties** from the pop-up menu if it is not already visibile on the right side toolbar.

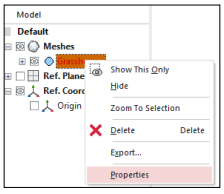


Figure 397

- Click the **Material** option in the **Appearance** options.
- Change the color to the **green** material.

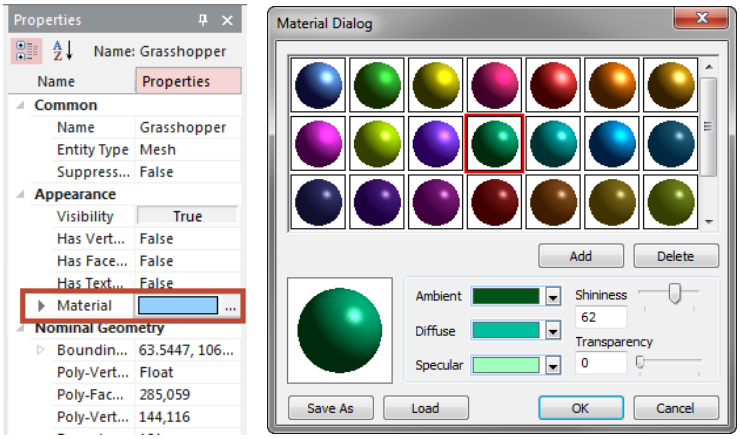


Figure 398

3. Click the **OK** button. The color of the mesh material for the model will change to green.

Figure 399

☞ *Changing the color of the mesh is used to help in recognizing the mesh in the application. It does not change the color information of the mesh file.*

Global Remesh

1. Click the **Global Remesh** button in the **POLYGONS>Optimize** group.
   - Set the **Edge Length Multiplier** to **1** to create similar edge lengths for the entire model.
   - Enable the **Make Clean And Manifold Mesh** option to generate a clean and closed mesh.

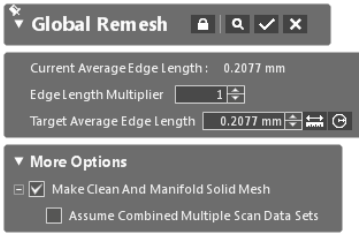


Figure 400

- Click the **OK** button.

☞ *The mesh will now be globally remeshed by regenerating mesh structures and removing defects from the curvature flow of the mesh. All poly-faces will be remeshed to have the same edge length, resulting in a more uniform mesh.*

Figure 401 – Before Remesh

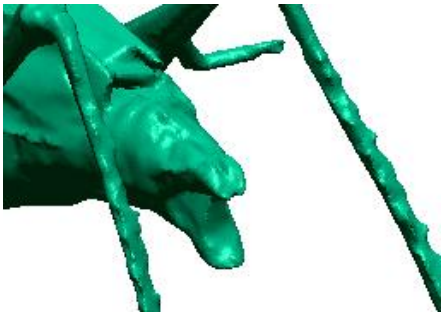


Figure 402 – After Remesh

Decimating the Mesh

Show Edge

1. Enable the **Edge** option under the **Mesh** section in the **Display** tab, or press the **F8** key on the keyboard.

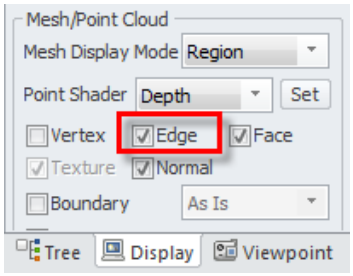


Figure 403

Decimate

1. Click the **Decimate** button in the **POLYGONS>Optimize** group.
   - Click the **Don't Quit Command With OK** button to continuously use the comm and without exiting it.
   - Set the Reduction Ratio to 70%.
   - Click the **OK** button to apply the command without exiting it.

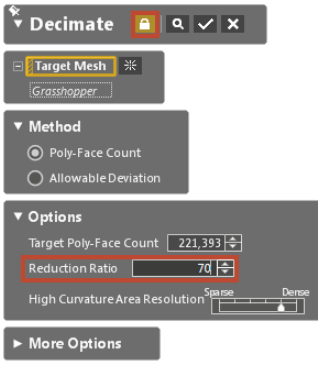


Figure 404

2. Apply the command again by clicking the **OK** button, then click the **Cancel** button.

☞ *The mesh will be reduced by 70% again to further reduce the poly-face count.*

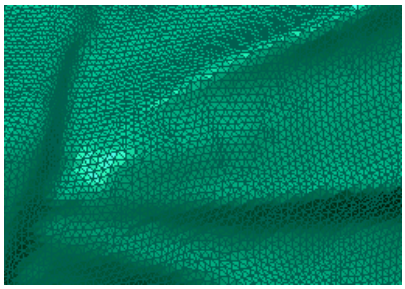


Figure 405

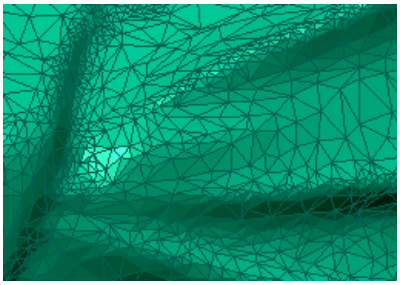


Figure 406

Optimize Mesh

1. Click the **Optimize Mesh** button in the **POLYGONS>Optimize>Global Remesh** dropdown group.
   - Select the High Quality Mesh Conversion Method.
   - Set the Min Edge Length Multiplier to 0.5 and the Max Edge Length Multiplier to 8.

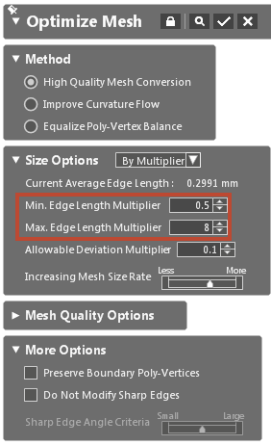


Figure 407

☞ *This method creates regular sized triangles from the original mesh by generating more triangles in high curvature regions and fewer triangles in low curvature regions.*

2. Click the **OK** button. The mesh will now be optimized.

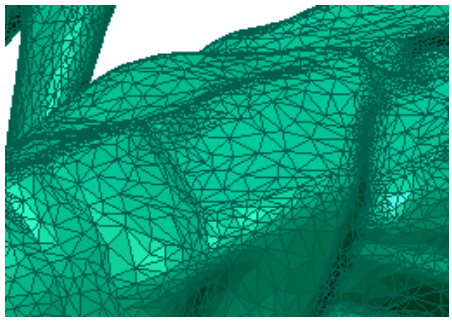


Figure 408

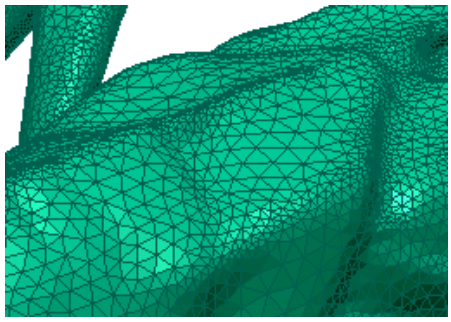


Figure 409

Enhance Shape

1. Click the **Enhance Shape** button in the **POLYGONS>Optimize** group.
   - Change the **Overall Smoothness** to **Mid** and the **Enhance Level** setting to **Max**.

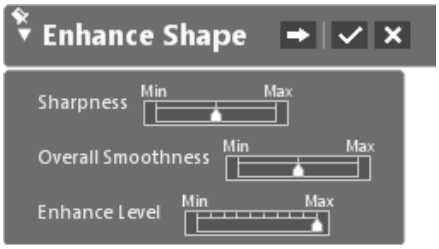


Figure 410

2. Click the **OK** button. Areas of high curvature will become sharper than the original shape. Apply this command again with the same options to emphasis sharper areas even more.

Smooth

1. Click the **Smooth** button in the **POLYGONS>Optimize** group.
   - Change the Smoothness Level to Max.

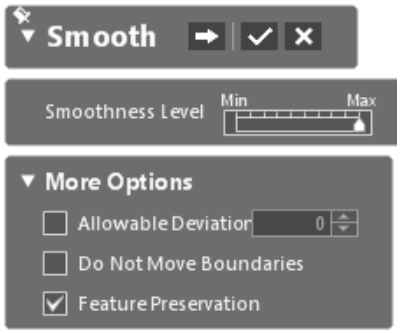


Figure 411

2. Click the **OK** button. The mesh will be smoother globally.

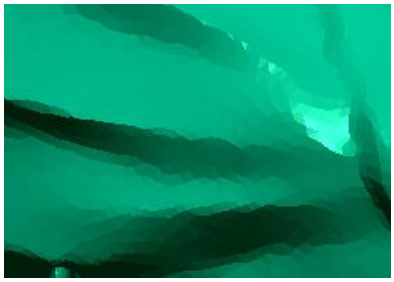


Figure 412

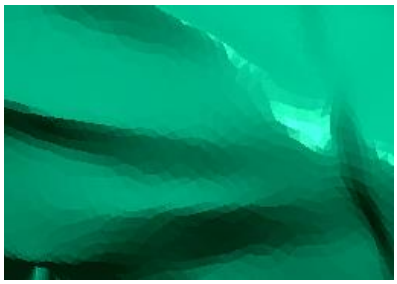


Figure 413

End of Activity

ACTIVITY: Mesh Editing

Objective

This training course is designed to show how to edit the mesh of a phone scan to prepare it for the modeling process.

Highlighted Commands

- **Defeature**
- **Fill Holes**
- **Edit Boundaries**
- **Healing Wizard**

Training File

Phone.stl

Import Scan Data

1. **Import** the **Scan Data** found at:
(~/Appendix/Phone.stl)

Figure 414

Defeature

1. Click the **Defeature** button in the **POLYGONS>Repair Holes/Boss** group.
    - Click-drag and release around an area near the top of the phone as shown in the image below.

Figure 415

☞ *The defeature command is useful for models where small irregular features exist that may hinder the modeling process.*

2. Click the **OK** button. The feature will be removed.

Figure 416

Fill Holes

1. Click the **Fill Holes** button in the **POLYGONS>Repair Holes/Boss** group.
   - Toggle on the **Don't Quit Command With OK** button to continuously use the command without exiting it.
   - Select the microphone hole boundary on the bottom of the phone (**Figure 417**) using the default settings.
   - Click the **OK** button.

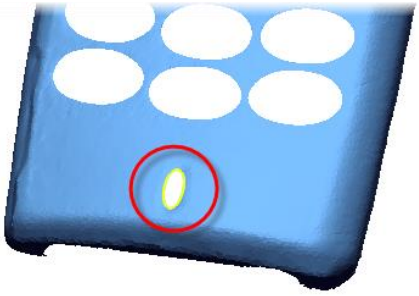


Figure 417

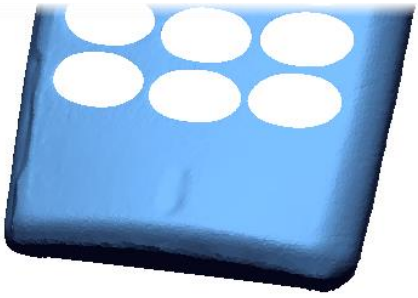


Figure 418

Add Bridge

1. Click the **Add Bridge** button in the **Editing Tools**.
   - Click on hold the LMB on the edge of one triangle on one side of the gap seen below.
   - Drag the mouse to the other side and release one an edge is highlighted.

Figure 419

☞ *Adding bridges is useful for filling in complicated holes or boundary areas that are irregular in shape.*

2. Click the **Boundaries** button and
   - Select the boundary created by the bridge shown in **Figure 420**.
   - Toggle off the **Don't Quit Command With OK** button to disable it and then click the **OK** button to complete the command.

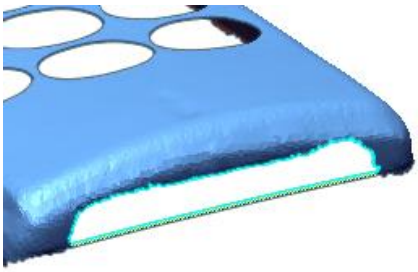


Figure 420

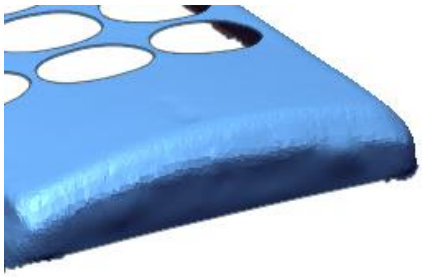


Figure 421

Edit Boundaries

1. Click the **Edit Boundaries** button in the **POLYGONS>Edit** group. This command enhances boundary shapes on a mesh.
   - Toggle on the Don't Quit Command With OK button.
   - Select the **Fit** method and change the **Fit Option** to **Circle**.
   - Select the largest circular boundary in the middle of the model shown in **Figure 423**.

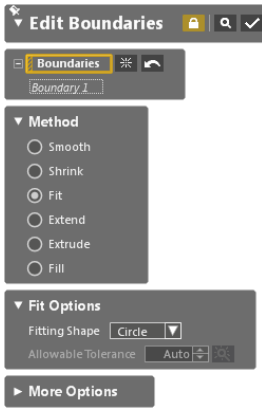


Figure 422

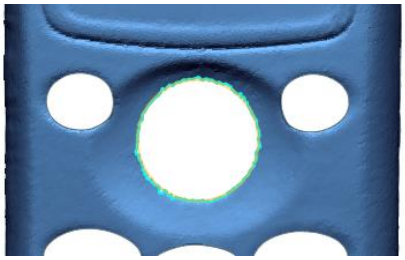


Figure 423

Figure 424

   - Click the **OK** button. The boundary will be changed to a circle.

2. Change the **Method** to **Smooth** and select the outer boundary of the mesh. Set the **Select N-Depth Boundary** to **3**.

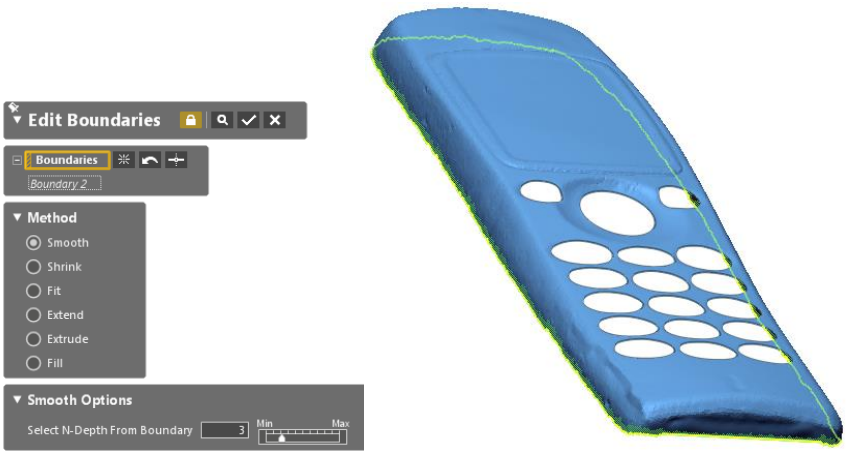


Figure 425

- Click the **OK** button. The boundary will be smoothed by 3 poly-faces from the original boundary.

3. Click the **Cancel** to exit.

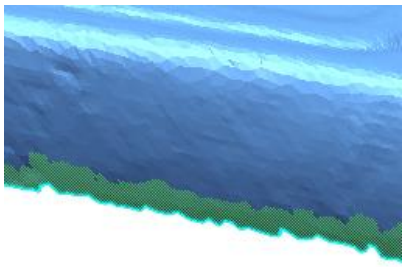


Figure 426

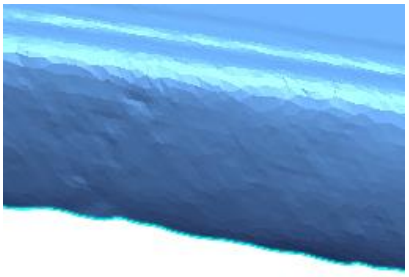


Figure 427

Healing Wizard

The wizard will remove unwanted and irregular poly-faces using an automated wizard process.

Remove Dangling Poly-Faces

1. To remove any remaining dangling poly-faces on the mesh boundary, click the **Healing Wizard** button in the **POLYGONS>Wizard** group.
   - De-select all options except for the **Dangling Poly-Faces** option.

Figure 428

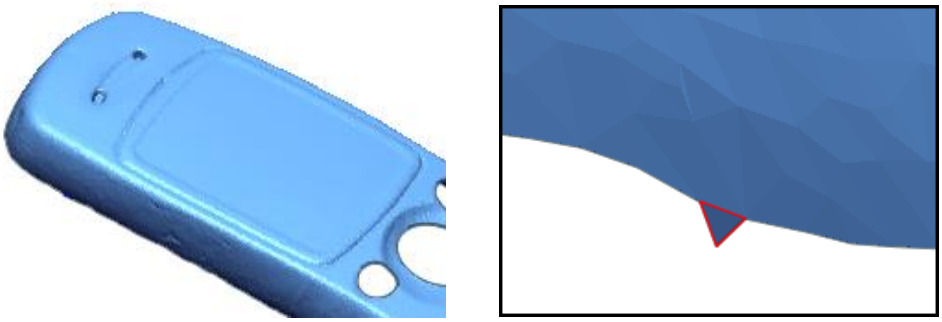
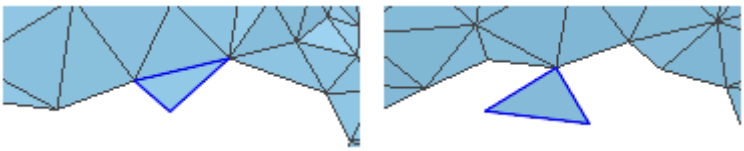


Figure 429

2. Click the **OK** button. The dangling poly faces on the mesh will be removed.

Figure 430

☞ *Dangling Poly-Faces are edges that are not connected to neighboring poly-faces.*



2 Sides Open **3 Sides Open**

End of Activity

Appendix III – Additional Models

ACTIVITY: Iterative modeling

Objective

Reinforce common skills, such as importing scan data, region segmentation and editing, and 2D sketching and basic modeling concepts to create a parametric model. Reverse model a casting and iterate the model to the desired result by modifying the parametric features.

Highlighted Commands and Concepts

- **Mesh Sketch Setup (Advanced)**
- **Convert Entities**
- **Iterative Modeling**

Training File

A1-Iterative Model.stl

Scan Setup

1. **Import** training file **A1-Iterative Model.stl.**

☞ *The model is not a complete scan, but all the information required to model is available.*

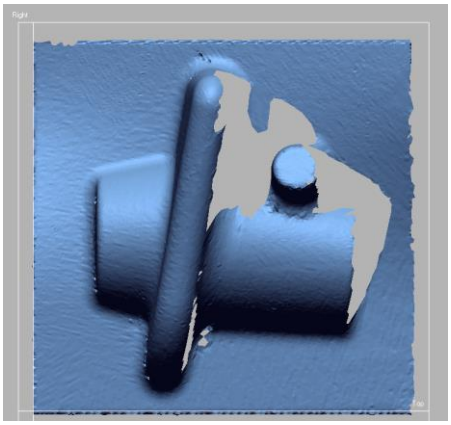


Figure 431

2. In the **REGION** tab, select the **Auto Segment** command.
   - Set the **Sensitivity** to **65.**
   - Click the **OK** checkmark to accept.

3. Switch to the **Geometry Type Display Mode** from the Upper-Side Toolbar.

☞ *Each specific geometry will have a specific assigned color. For example, all Planes will be blue.*

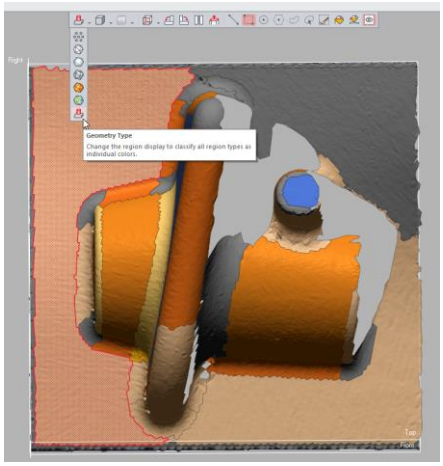


Figure 432

Method 1: Extracting Geometry

1. While still in the **REGION** tab, **Hold Ctrl** and select all three regions of the curved top face.

- Select the **Edit>Merge** command to merge the regions.

☞ *Certain commands in the extracting geometry workflow require the selection of a single region.*

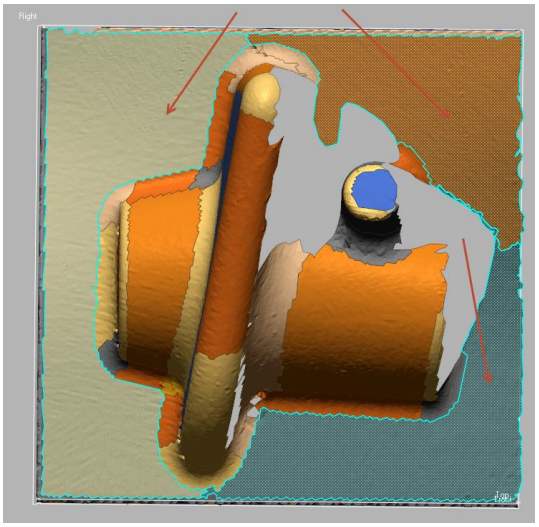


Figure 433

2. Use the **Extrusion Wizard** to quickly create a base body.
   - **RMB** click on the recently merged region and select **Insert/Merge**.
   - Click the **Select None** next to **Sides** then **Top**.
3. The automatically selected regions were not appropriate for the desired outcome.
   - For **Top**, select the **Freeform**, merged region on the mesh, and for the **Bottom** select the **Ref. Plane Front** plane.
   - For **Sides**, select the three **Plane** walls.
   - Set the **Custom Extrusion Direction** to the **Ref. Plane Front** to be able to select **Fix Sketch Plane With Selection**.

☞ *The extracted 2D Sketch for the upcoming Extrusion will be placed on the Front Plane.*

- **Uncheck** the **Draft Angle** option.

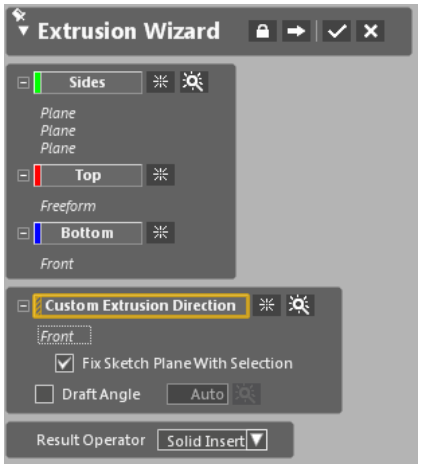


Figure 434

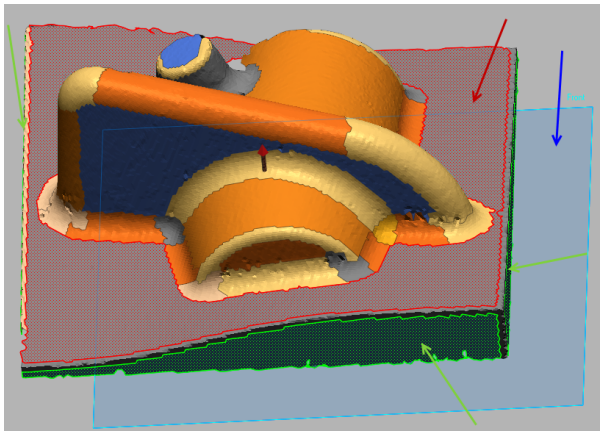


Figure 435

4. Select the **Next Stage** arrow to view and edit the generated sketch and extrusion.

- Slide the **Resolution** slide bar all the way to **Min** to avoid the noise of the scan data in the final sketch. It should appear as a rectangle.
- Set the **Geometry Capture Accuracy** all the way to **Loose** to allow for constraint (**Perpendicular, Tangent, etc.**) entities to be assigned to the sketch.
- Accept the Extrusion Wizard as long as the created body is previewed.

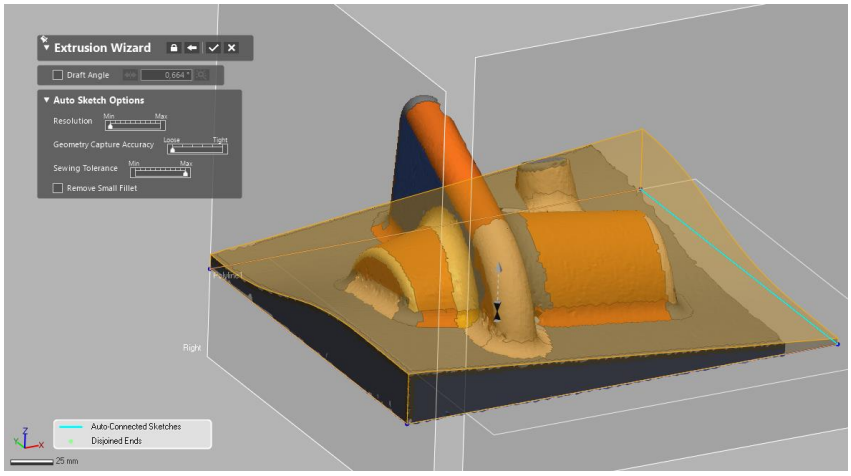


Figure 436

5. **RMB** click on the **Sketch1(Mesh)** that has been created in the Feature Tree, and select **Edit**.

☞ *Wizard parameterscannot be edited after acceptance because editable, parametric features are created in the Feature Tree.*

- Select the **Front Viewpoint** from the Upper-Side Toolbar.
- Adjust the right-most line to better fit the model.
- **Exit** the sketch to re-extrude the base.

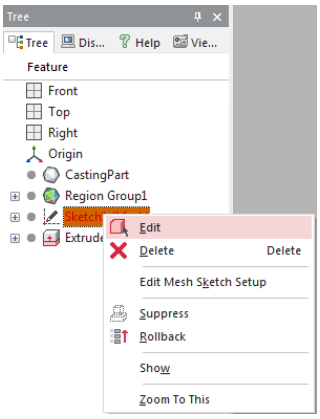


Figure 437

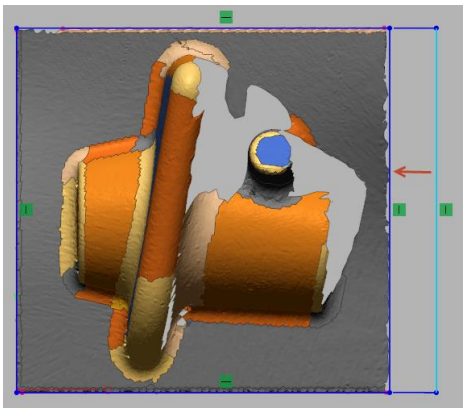


Figure 438

6. Inspect the quality of the surface by checking different **Surface Analyzing Modes**.

- From the Upper-Side Toolbar, or Accuracy Analyzer dock, select **Deviation For Body.** The majority of the surface is green so within our desired tolerance.
- Change the option to **Iso Line**. This will show the curvature of the freeform surface.

◉ Although this is a quick way of extracting the actual situation of the scan data, it is not always the desired method. The top, freeform surface is not editable in CAD. The surface is not easily reusable because its flow cannot be controlled by the user.

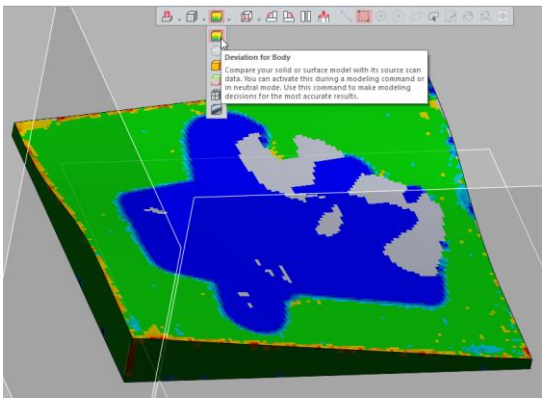


Figure 439

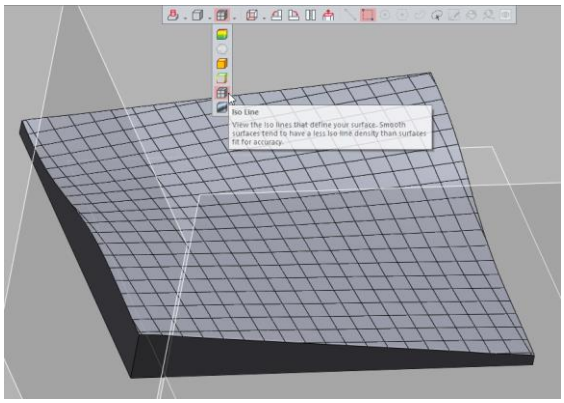


Figure 440

Method 2: Creating Geometry

◉ Looking and investigating the nature of the top surface, it is a surface that consists of two differently tilted areas that are blended between.

Do not delete or undo the previous body. It can still be utilized in the next method.

1. Create a **Mesh Sketch** on each of the side faces of the solid body.

- Select the actual face of the side wall shown in the **Figure** as the **Base Plane**.
- Set the **Offset Distance From Base Plane** to **1 mm** inward on the part.
- Click **OK** to begin sketching.

☞ *Tilted surfaces should ideally get a zero offset, but some cases require a slight offset to capture better data.*

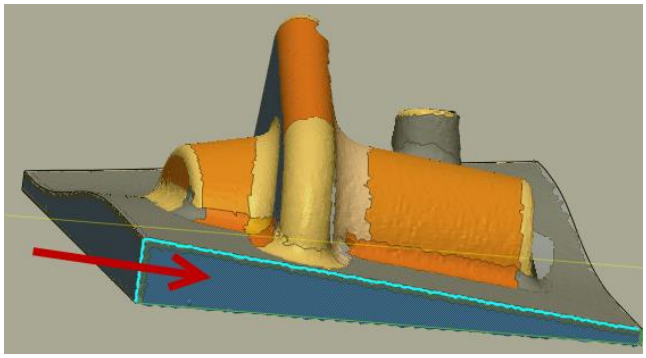


Figure 441

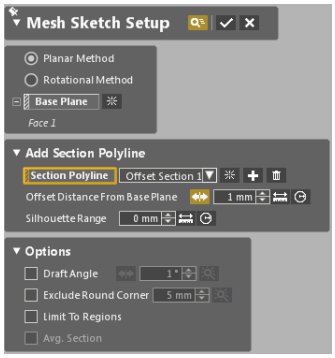


Figure 442

2. Set the left and right edges of the base by converting the extracted base.

☞ *Turn the visibility of the Solid Bodies ON and the visibility of the Meshes OFF.*

- **RMB click** on the left-most vertical edge and select **Convert Entities**.
- Repeat on the right-most vertical edge.

3. These lines have a Converted Constraint. Change the lines to **Construction Lines**.

- **RMB click** on each line and select the **Make As Construction Line** symbol.

☞ *Hide the Solid Bodies to avoid selecting when finishing the sketch.*

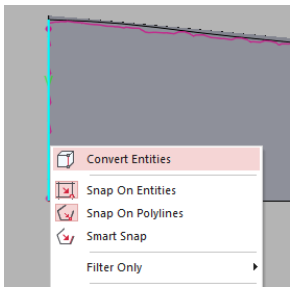


Figure 443

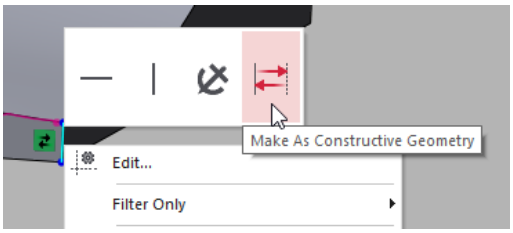


Figure 444

4. Finish the sketch with the **Line** and **Corner Trim** tools.

- With the **Line** tool, double click on the long diagonal pink profile line.
- Use the **Corner Trim** command to connect the diagonal to the vertical lines.

5. Add Dimensions using the **Smart Dimensions** command.

- Select the end point of the diagonal top line, vertically down to the U Axis to define the height.
- Repeat on the other vertical line.
- Turn on the **Deviation** from the Upper-Side Toolbar to adjust the best assumed heights.

☞ *In the Accuracy Analyzer dock, increase the Vector Multiplier to view the deviation easier.*

- Change the dimensions to the best fit height, or adjust to match **Figure 445**.
- **Exit** the sketch.

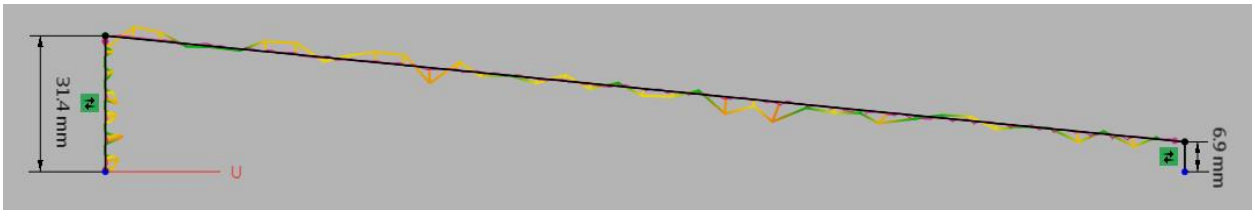


Figure 445

6. **Repeat** the same process for the opposite face.
   - Mesh Sketch on the solid face, offset 1mm inward.
   - Convert the vertical edges and make as construction.
   - Dimension to match **Figure 446**.

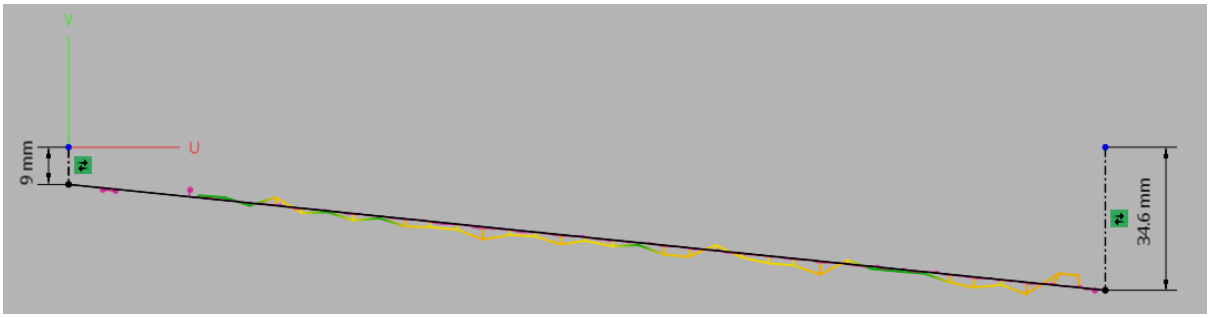


Figure 446

7. The other two sides needs to be created using Mesh Sketches and the previous sketches.

   - Set up the **Mesh Sketch** on a remaining face, offset inward **1 mm**.
   - Create a **Line** on a straight stretch of the pink profile line by using the rectangular selection method of drawing a line (**Figure 447**).
   - Double-click on the end point then **Shift+LMB** on the grey line of the previous sketch.
   - Select the **Pierce Constraint** from the Common Constraints List.
   - Repeat on the other side of pink profile lines.

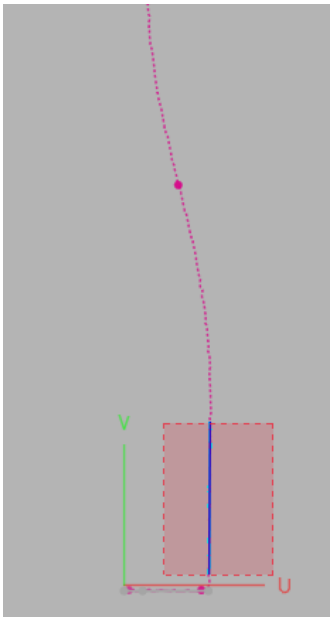


Figure 447

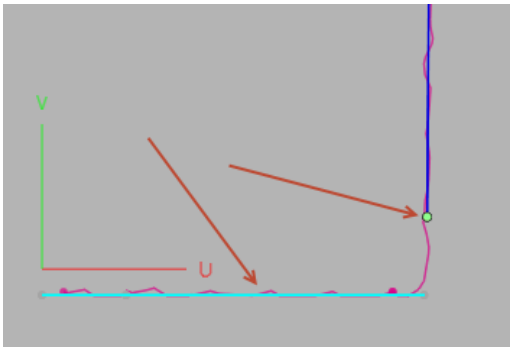


Figure 448

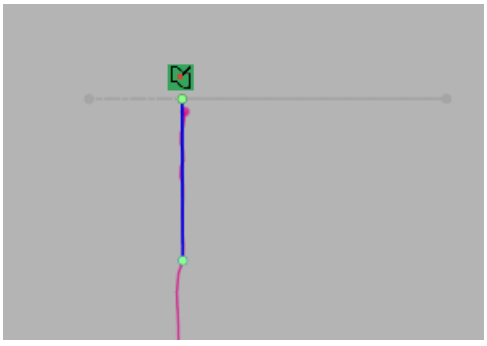


Figure 449

8. Use a **Spline** tool and only two points to connect the open entities.

- Add **Tangent** constraints to the connection of the spline and each line. This will allow the spline to bend and better fit the polylines.
- Use **Smart Dimensions** to define the sketch as shown in **Figure 450**.

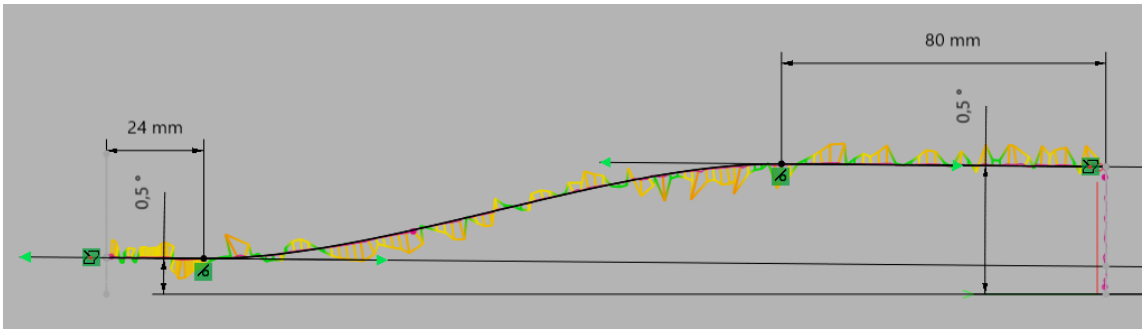


Figure 450

9. Repeat this process on the other side.

- Mesh Sketch on the face with the profile offset 1mm inward.
- Create two lines and Pierce them to the previous sketches.
- Use a spline to connect the two lines.
- Constrain and Dimension.

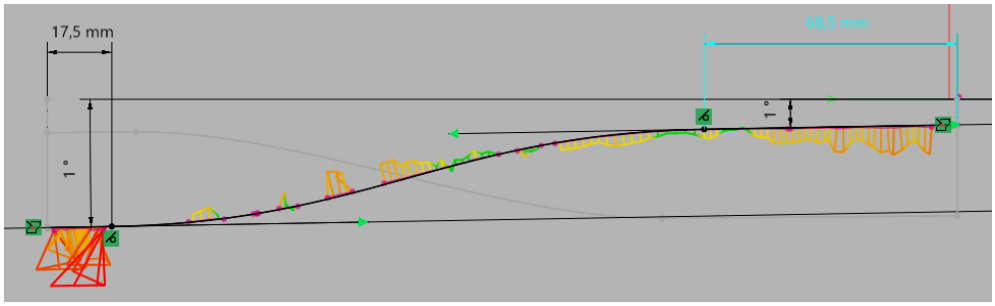


Figure 451

10. Create the top surface using the **MODEL>Create Surface>Loft** command.

- For the **Profiles**, select both sketch chains with the splines.

☞ *The Surface Normal depends on the order the profiles are selected. Choosing the profiles from right bottom to left bottom, or left top to right top, will create a surface normal pointed in the desired direction.*

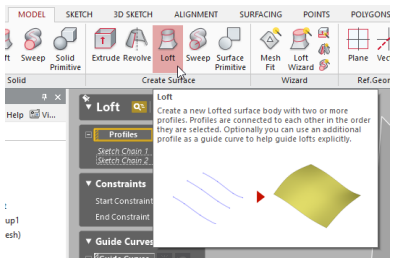


Figure 452

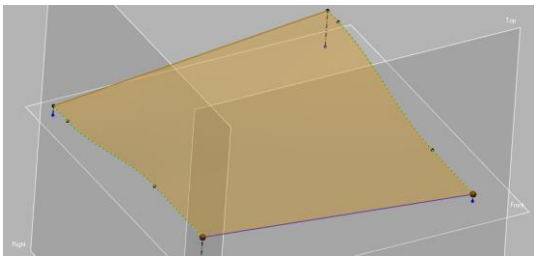


Figure 453

11. Use the **MODEL>Body/Face>Replace Face** command to use the newly lofted surface.

- For **Target Faces**, choose the extracted, curved top face of the solid body.
- For **Tool Entity**, select **Loft1** from the Model Tree.
- Accept the command to finish the solid.

Figure 454

12. Check the changes in the different deviations between the Extracted geometry and the geometry created with sketches and dimensions.

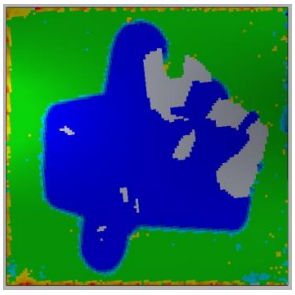


Figure 455 – Extracted Deviation

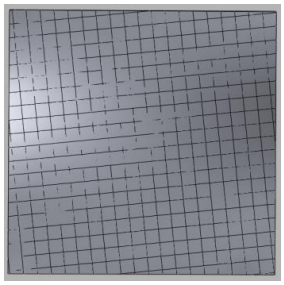


Figure 456 – Extracted Iso-Line

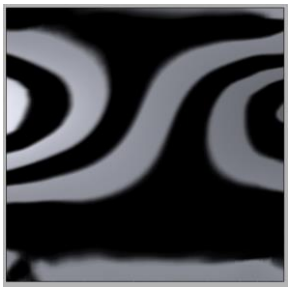


Figure 457 – Extracted Design Quality

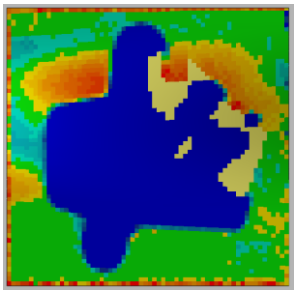


Figure 458 – Created Deviation

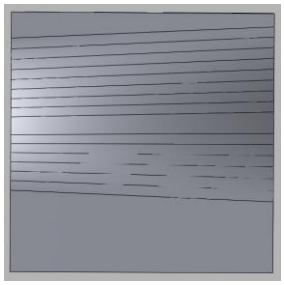


Figure 459 – Created Iso-Line

Figure 460 – Created Design Quality

Casting Body Features

1. Find the position and orientation of the rib feature.

- Method 1: Create **Ref. Geometry>Vector**, and select the **Cylinder** region shown and the **Front** plane.
- Change the **Method** to **Projection** then accept the command.

☞ *This method is most accurate to the scan data, but unable to change parameters of the vector. The created vector will not be used.*

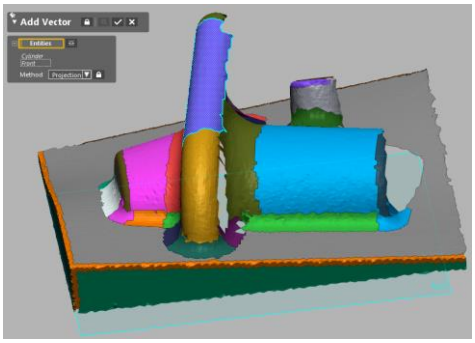


Figure 461

- Method 2: Create a **Ref. Geometry>Polyline**, and select the top fillet regions of the rib feature.
- Change the **Method** to **Fillet** then accept the command.

☞ *Polylines can be inserted into Mesh Sketches and sketch entities will snap to the lines to give full control of the shape and form.*

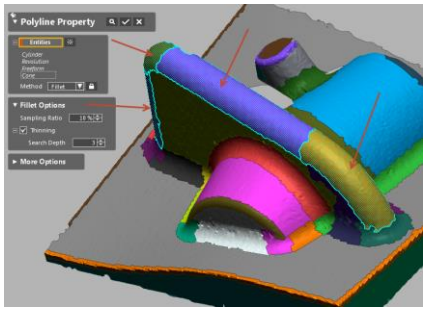


Figure 462

2. Create a **Mesh Sketch** on the **Front** Plane with 0 offset.

- Bring the Ref. Polyline into this sketch using **MENU>Tools>Sketch Tools>Insert Polyline.**

- Sketch a **Line** on the inserted polyline.
- **Convert** the intersecting edges of the solid base and **Make as Construction Geometry.**

- Use the **Edit>Extend** tool to extend the sketched line to the converted edges.
- Dimension the endpoints and angle of the line to the vertical V Axis.

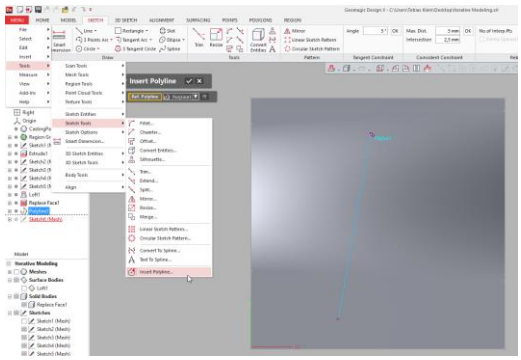


Figure 463

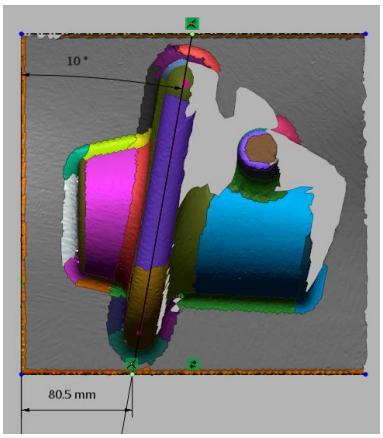


Figure 464

3. Create a **Ref. Geometry>Plane** on the sketch line.

- Choose the **Right** Plane and Sketch (**Curve 1**) as the **Entities.**
- Change the **Method** to **Projection** and accept.

☞ *A plane, orthogonal to the Front Plane, will now be set on the sketch line to use to create the rib.*

4. Create a **Mesh Sketch** with the new plane, **Plane 1**, as the **Base Plane**.

- To capture the maximum height and profile of the rib, set the **Silhouette Range** to be wider than the rib, about **38 mm** and accept the sketch setup.

☞ *The upper and lower boundaries of a volume around the section plane will be projected to the base plane.*

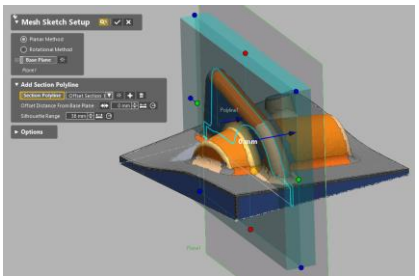


Figure 465

5. Sketch the profile of the rib as shown in **Figure 466**.

- Use the **Auto Sketch** tool to quickly extract sketch entities from the poly-lines.
- Adjust the **Resolution** slide bar to find the appropriate poly-line segments.
- With **Make Selected Only**, choose the top profile of the rib and accept the dialog.

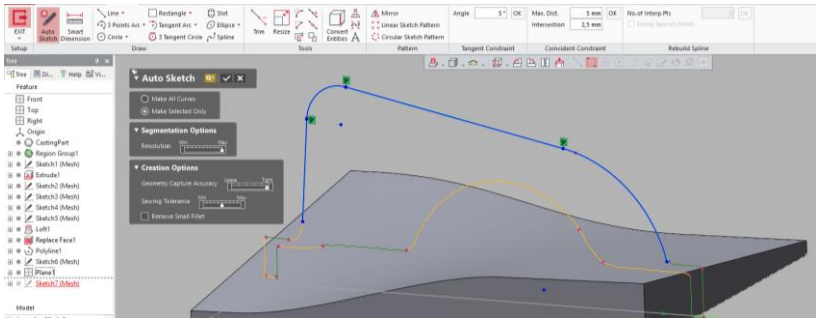


Figure 466

6. Use **Convert Entities** to bring in the lower line of the rib sketch profile.

- Select the line from the previous sketch.

7. **Trim** the profile to close the sketch loop.

8. **Smart Dimension** the sketch to define the rib, then Exit the sketch.

☞ *A 4.5° draft between the vertical and side lines of the sketch were added for manufacturing purposes.*

☞ *For better maintainability, sharp corners were created. Dimensions on the corner points, rather than the arc centers, will allow for easier radii changes.*

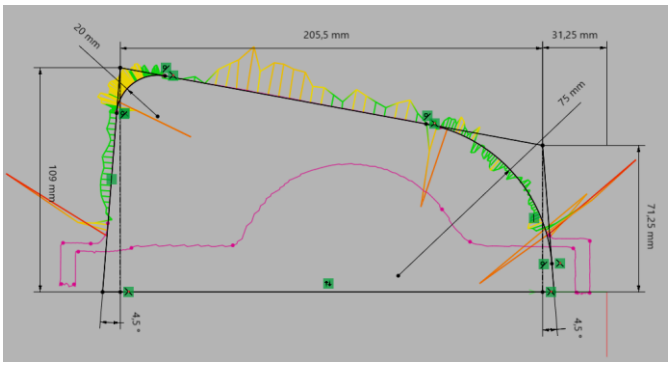


Figure 467

9. Select **Create Solid>Extrude** from the previous sketch to generate a solid model.

- Select all three closed sketch loops for the **Profiles**.
- Set the **Method** to **Mid Plane**. Drag the arrow until it snaps to the upper boundary of the planar side region.
- Change the **Length** value to a round number.
- Remove all checks under **Result Operator**, and then **Accept** the Extrude.

☞ *A draft is still needed for the extruded sides, but it will be added at a later step.*

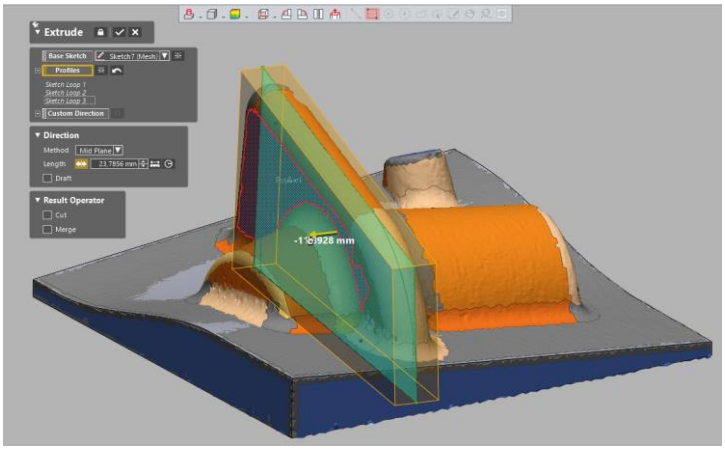


Figure 468

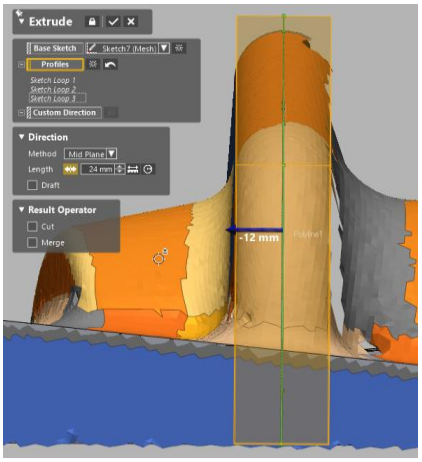


Figure 469

10. In the **MODEL** tab, select **Edit>Fillet**.

- In **Constant Fillet**, select the corner edge of the extruded rib.
- Use the small **Measure Radius** button next to the Radius input field.
- Click on the larger radius sketch entity.
- Verify the **Radius** is now set to **75 mm**.
- **Accept** the command.

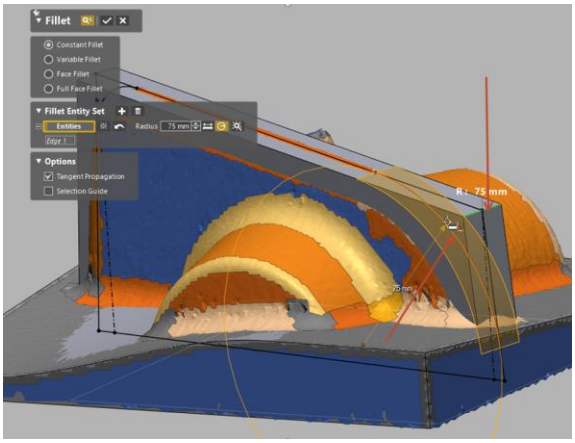


Figure 470

11. Apply the **Edit>Draft** command on the rib.

- Set the method to **Parting Line Draft**.
- Select the planar, top face of the rib as the **Pull Direction.**
- For **Parting Lines**, select 4 planar and 2 round edges along either side of the top rib face, with their respective arrows pointed down, in to the part.
- Adjust the draft **Angle** to **4°** with this arrow pointing up and away from the part.
- **Accept** the Draft.

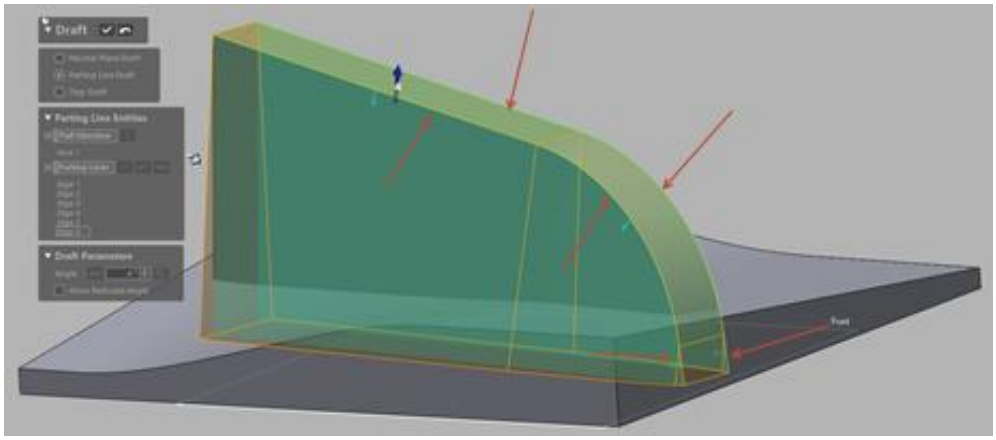


Figure 471

12. Apply a **Fillet** to the smaller edge of the rib, the same was as in Step 9.

- In **Constant Fillet**, select the corner edge of the extruded rib.
- Use the small **Measure Radius** button next to the Radius input field.
- Click on the larger radius sketch entity.
- Verify the **Radius** is set to **20 mm** then **Accept** the command.

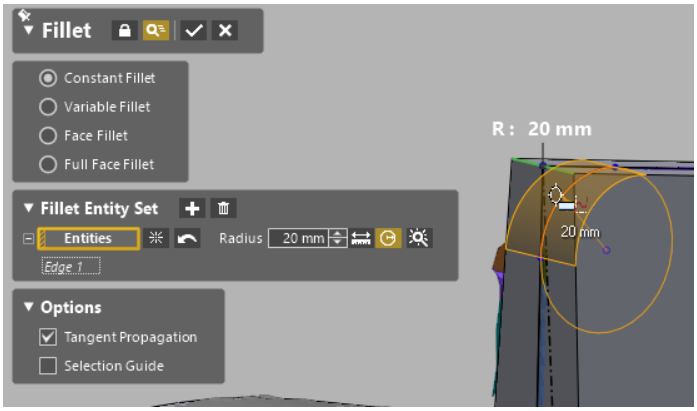


Figure 472

13. Press the **Ctrl+Space** keyboard keys to repeat the previous command of **Fillet**.

- Change the type to **Full Face Fillet** to round the entire rib face.
- Select one of the large side faces as the **Left**, and the other large face as **Right**.
- Select the top rib face as the **Center** and **Accept**.

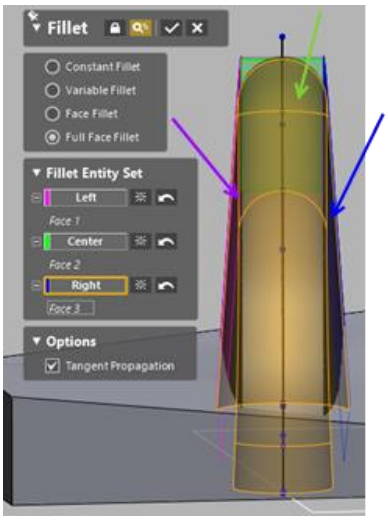


Figure 473

Adjustments and Deviation Checking

1. Turn on the **Deviation for Body** to see how the solid model matches the scan. Make adjustments as needed.
   - The four sketches (**Sketch2-5(Mesh)**) guiding the lofted face can be edited to minimize deviation of the top freeform loft face.
   - To edit the Spline Tangent Lengths, double-click on the Spline inside of the Sketch to enter the Properties Dialog Tree. Click, hold, and drag the Tangent Arrow around to create a smoother tangent transition. (**Figure 475**)
   - Edit the Extrusion width to fit the scan width better.
   - Adjust the position and angles of the rib with **Sketch6(Mesh)** and **Sketch7(Mesh)**.

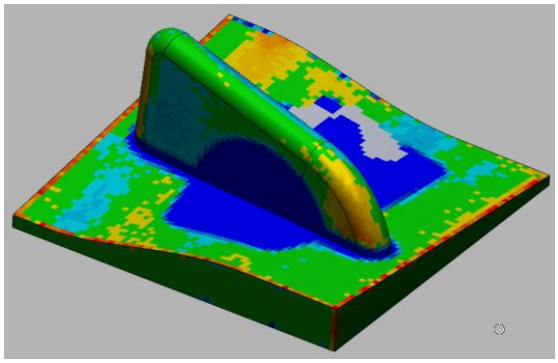


Figure 474

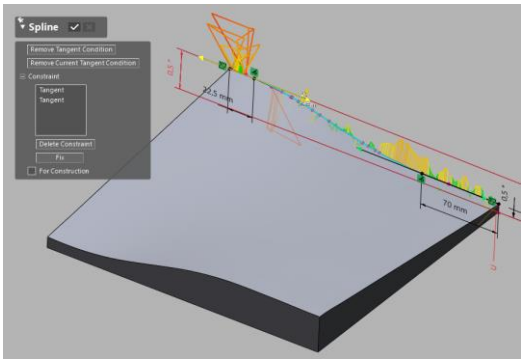


Figure 475

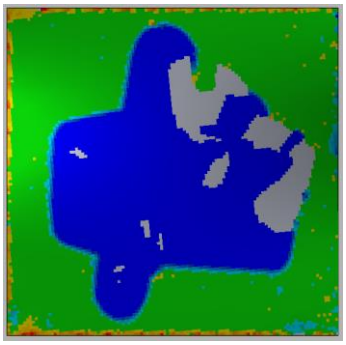


Figure 476 - Extracted

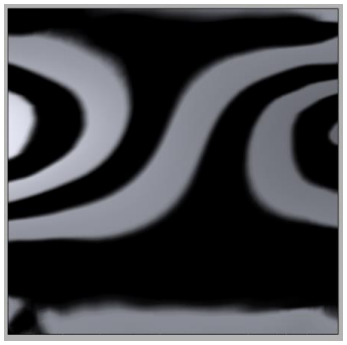


Figure 477 - Extracted

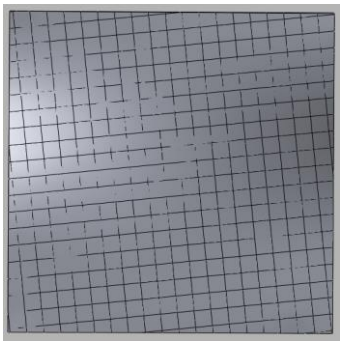


Figure 478 - Extracted

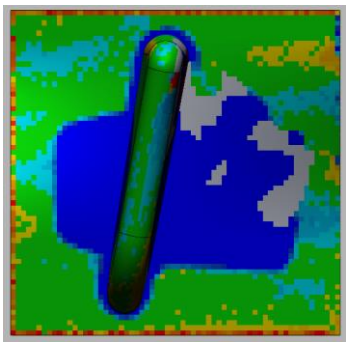


Figure 479 - Created

Figure 480 - Created

Figure 481 - Created

Cone Features

1. Generate an editable axis for one of the cone features by creating a **Mesh Sketch**.
   - Select the **Right** Plane as the **Base Plane**.
   - Offset the **Section Poly-line** until it is in the cone region (**Figure 482**), about **60 mm**.

2. Use the **Draw>Circle** tool to find a center point of the central cone axis.
   - Double click on the pink arc line to create the circle.

   - Use **Smart Dimension** to fully define the center point.

☞ *The Cone could be quickly and easily made with the RMB Insert/Merge command to extract from the scan data. Accurate to the scan geometry would be generated, but it would not be easily editable.*

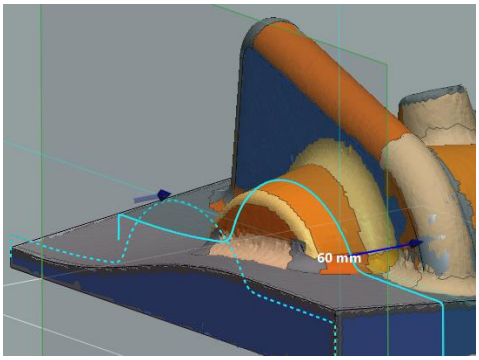


Figure 482

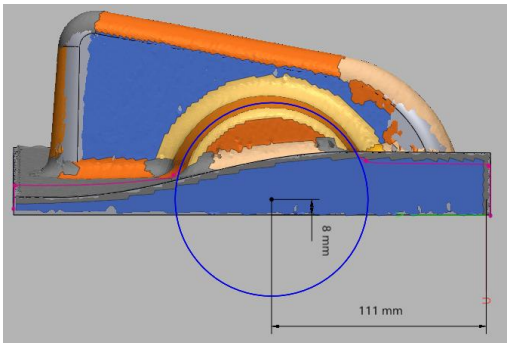


Figure 483

3. Create a **Ref. Geometry>Vector** for the central axis of the cone.
   - Select the Sketch **Circle Center** and the **Right** Plane.
   - Change, if not done automatically, the **Method** to **Pick Point and Line**.

4. Quickly generate the cone body with the **Revolution Wizard**.
   - RMB click on the **Cone** Region and select the **Insert/Merge** icon.
   - Add the fillet region and the planar face region to the **Target**.
   - Place a **Check** next to **Get Partial Shape** then go to the **Next Stage**.
5. The second stage allows for slight editing of the revolution and base sketch.
   - Modify the angle of revolution with the green and grey arrows.
   - Lower the **Resolution** slide for clean faces of the cone and no fillet in the sketch.

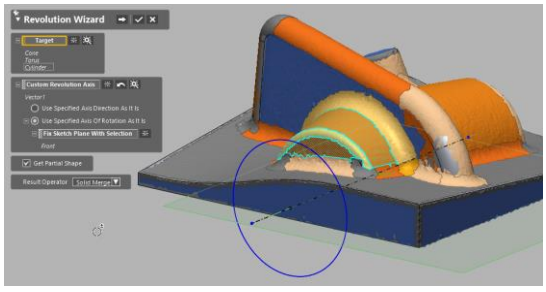


Figure 484

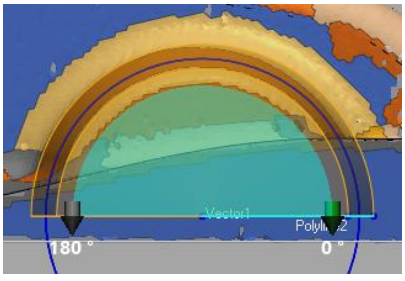


Figure 485

6. After accepting the wizard, investigate the revolved shape by looking from the **Front Viewpoint**. A gap between the cone and the rib is revealed.

- **RMB click** on **Sketch9(Mesh)** in the **Feature Tree** and select **Edit**.
- Click and drag the right edge further in to the rib section, then **Exit** the sketch.

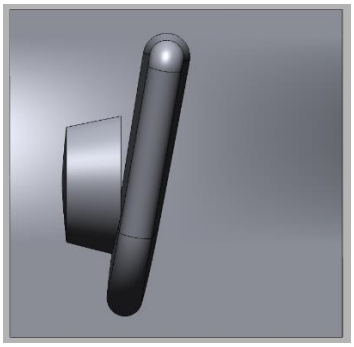


Figure 486

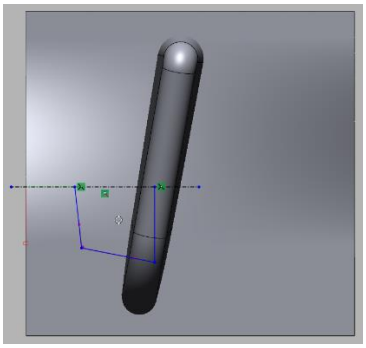


Figure 487

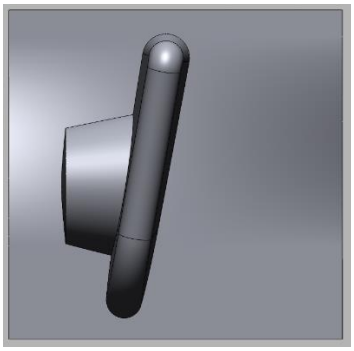


Figure 488

7. The previous cone was perpendicular to the ref. coordinate system. Create the axis of the second cone by again using a **Mesh Sketch**.

- Use the **Front** Plane as the **Base Plane**, and offset the section until it crosses through the entire conical face. They polylines will look like **Figure 489**.
- **Accept** the sketch setup and create the dimensioned sketch shown in **Figure 490**.
- Start with the **Parabola** tool found under **Draw>**Ellipse. Four points are needed to create the parabola.
- Use best judgement to choose the center point of the parabola. It should be near the apex. Use **Figure 489** to help place the other points of the parabola.

☞ *Just try to get the parabola close to the pink poly-lines. Dimensions and constraints will edit and move the entity.*

- Use the **Centerline** tool to draw a line between the center point and the apex point of the parabola. This line will be used to set angles and begin other dimensioning.
- Use the **Smart Dimension** to manipulate and find the central axis of the parabola.

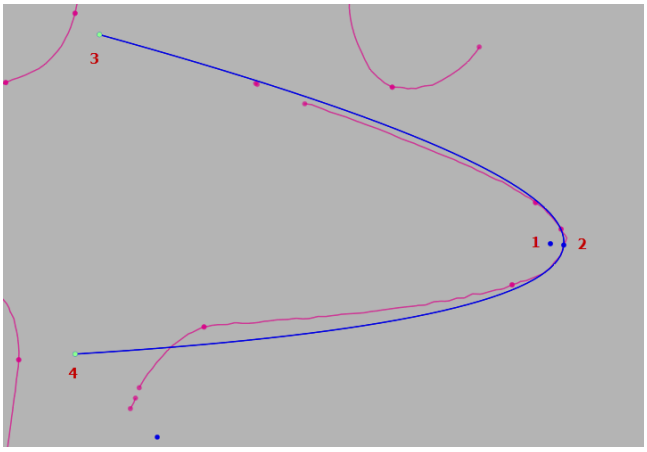


Figure 489

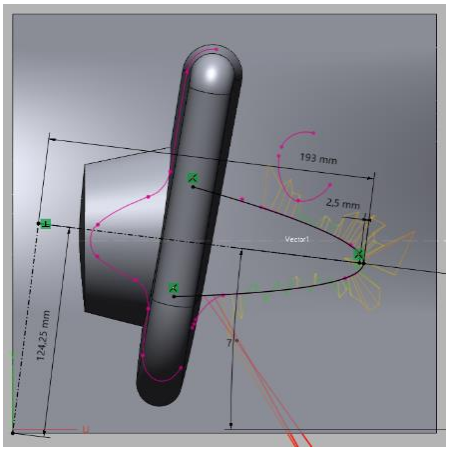


Figure 490

8. Create a **Ref. Geometry>Plane** with this center line.

- Select the **Top** Plane and the **Center Line** of parabola as the **Entities**.
- Change the **Method** to **Projection** if it does not change automatically.

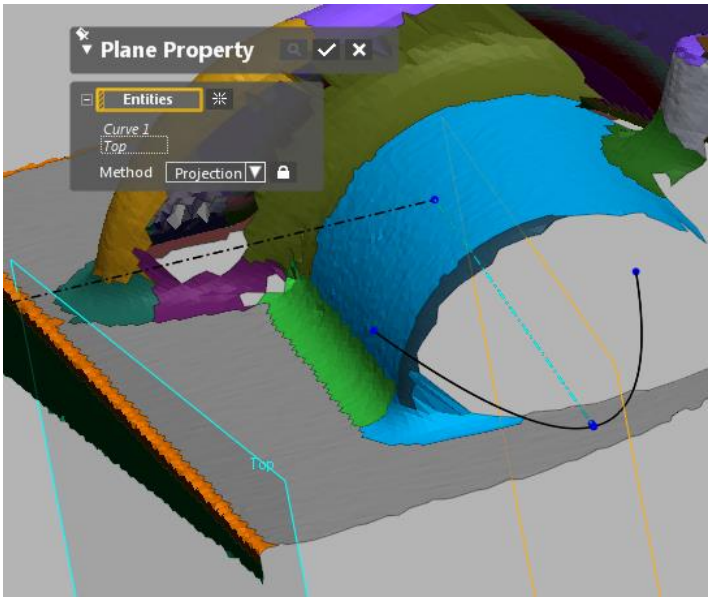


Figure 491

9. **RMB click** and select **Insert/Merge** on the **Cone** Region.

- Change the view to the **Bottom Viewpoint**.
- Notice the extracted axis generated by the **Revolution Wizard** is at about the same height as Vector 1.

☞ *This assumption can be used when manually generating this axis. The Revolution Wizard can show how the cone should be made, but it will be difficult to edit.*

- Cancel the wizard command.

10. Create a new **Mesh Sketch** on the new **Plane 2**.

- **RMB click** on **Vector 1** and select **Convert Entities**.

☞ *The center line is constrained to the first cone. Edits to this axis will change the axis of this cone.*

- To edit the center line independently, delete the **Converted** constraint and add a dimension and horizontal constraint.

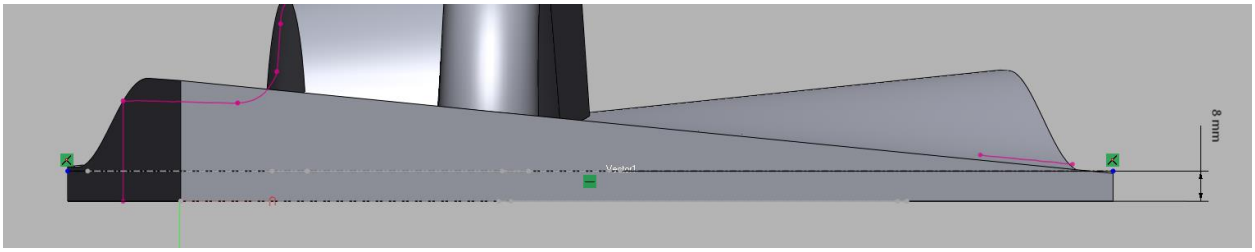


Figure 492

11. Create a **Ref. Geometry>Vector** on the center line from the previous sketch using the **Extract Method**.

12. Right-Click on the **Cone** Region and select the **Insert/Merge** icon.

- Check the **Revolution Wizard** has automatically found **Plane 2** and the **Vector 2** to use for the sketch and the revolution.
- Adjust the **Auto Sketch Options** to find the best fitting Cone.
- Increase the degrees of revolution by pulling on the arrows.
- Accept the wizard command.

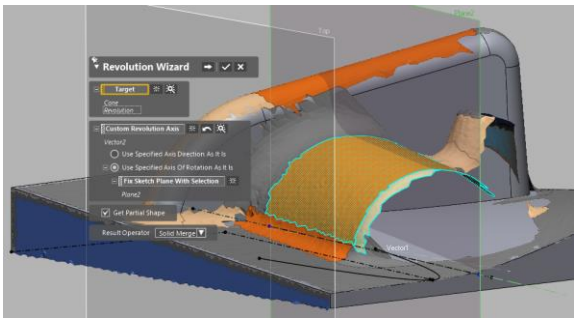


Figure 493

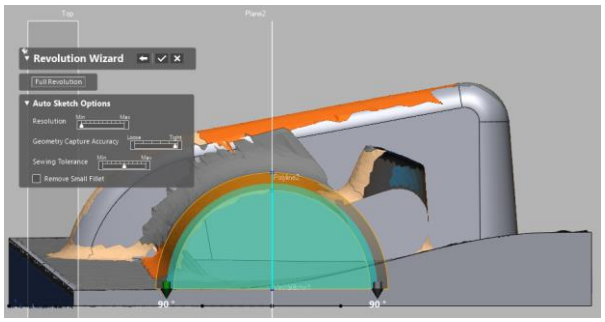


Figure 494

13. Right-Click on **Sketch12 (Mesh)** and select **Edit**.

- Adjust the length of the sketches to ensure it is long enough and extends in to the rib.
- Dimension the part to constrain the sketch and make it easier to edit in the future.

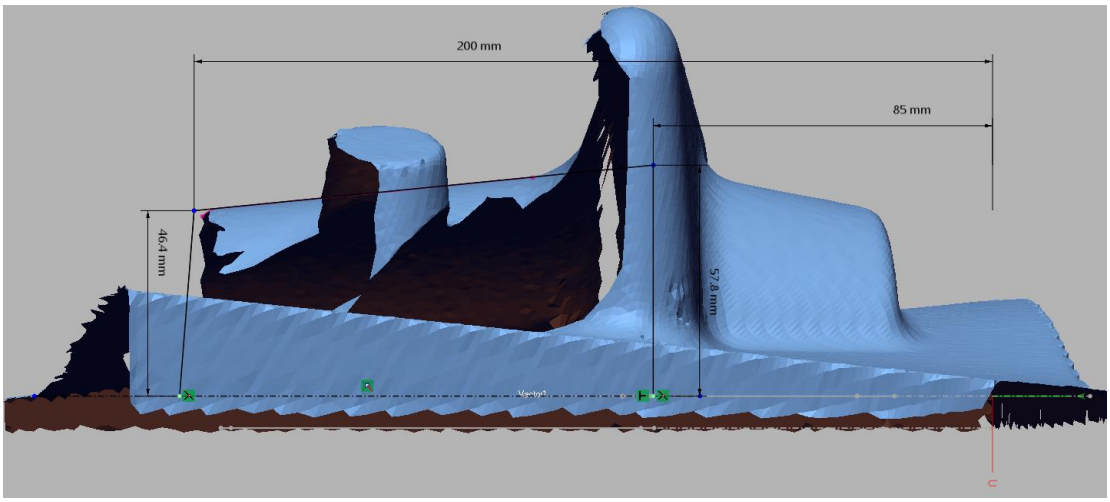


Figure 495 - Example of dimensioning

14. If needed, adjust any sketch or center lines of the first cone axis or parabola to fit better deviations.

☞ *Turn on Deviation for Body to check how each edit can affect the overall part.*

☞ *If entities are converted or tied to a previous entity, one edit could change multiple parts.*

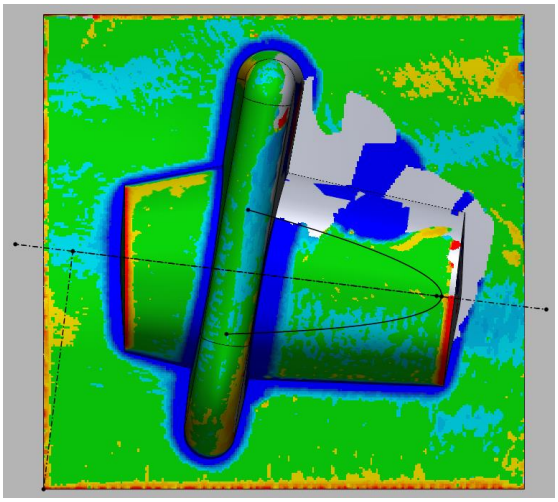


Figure 496

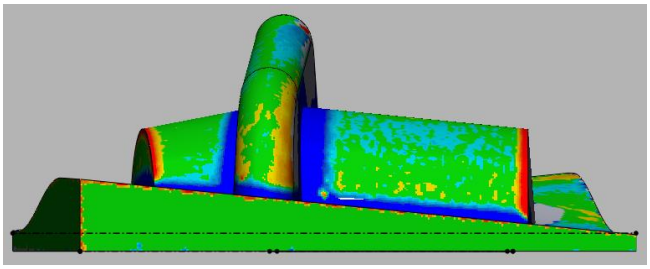


Figure 497

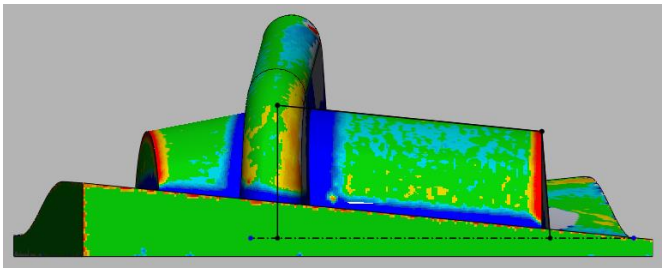


Figure 498

Pin Feature

1. Investigate the final pin feature.

- This feature has the least amount of information available from the scan.
- From the **Right Viewpoint**, the two different angles of the pin are visible.

2. Create a **Mesh Sketch** on the **Front** Plane.

- Offset the section to about **62.5 mm**; above the fillet in an area with the most possible amount of data.
- Create a sketch on the slot cross section. Draw the **Center Line** of the slot.
- **Exit** the sketch.

☞ *The information of the slot cross section can allow for the assumption that it is perpendicular to the Top plane.*

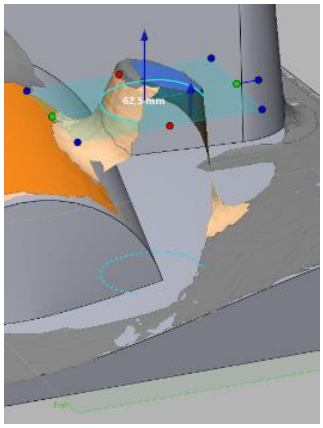


Figure 499

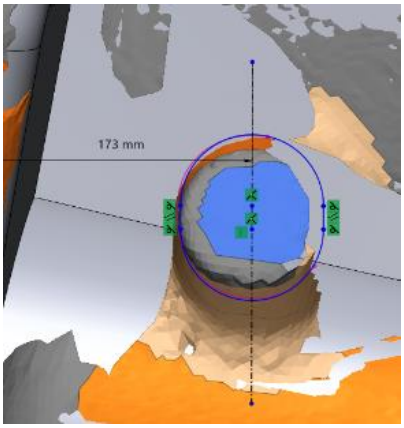


Figure 500

3. Select the **Ref. Geometry>Plane** tool to create the sketch plane for the side of the pin.

- Select the **Center Line** of the slot sketch, and the **Right** Plane.
- Change **Method** to **Projection** if necessary.
- **Accept** the command.

4. Create a **Mesh Sketch** on this new **Plane 3** and begin sketching.

- Since the left and right sides of the sketch have different draft angles, the best approach would be to create half the sketch and then mirror it about a center line.
- Create half of the sketch with the **Line** tool, and fully constrain with the **Smart Dimension**.
- Use the **Center Line** command and find the center of the feature.

☞ *This sketch will control the position of the pin, the tilt of the top and side faces, and the draft of the side faces.*

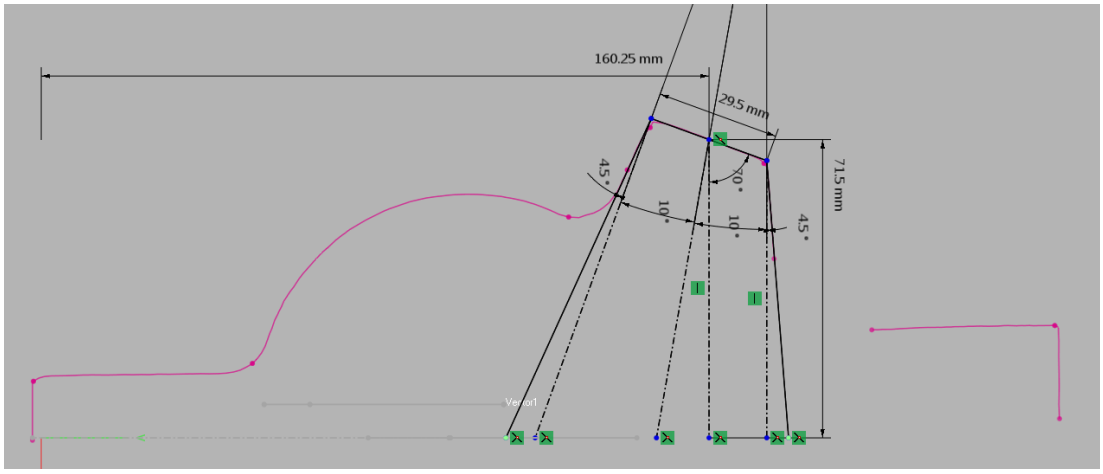


Figure 501

5. Create the sketching plane for the top shape of the pin by generating a new **Ref. Geometry>Plane**.

- Turn on the Visibility of **Sketch 14(Mesh)**.
- Select the **Front** Plane and the top **Line** of the last sketch.
- Change the **Method** to **Projection**.

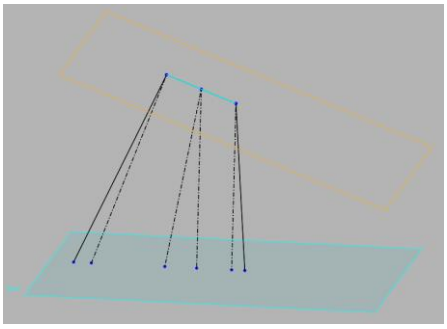


Figure 502

6. Create a **Sketch** on the new **Plane 4**.

- Use the **Circle** command to create a circle from the center point of the line from the last sketch to the end point of the sketch.
- Use the **Pierce** and **Converted** commands to constrain the sketch and pull in points needed for constraining.

- Use the **Center Line** and **Offset** commands to create split points for the circle.

☞ *The two arcs should be separate so different extrusions can be made to account for different draft angles.*

- Accept the sketch when it looks similar to **Figure 503**.

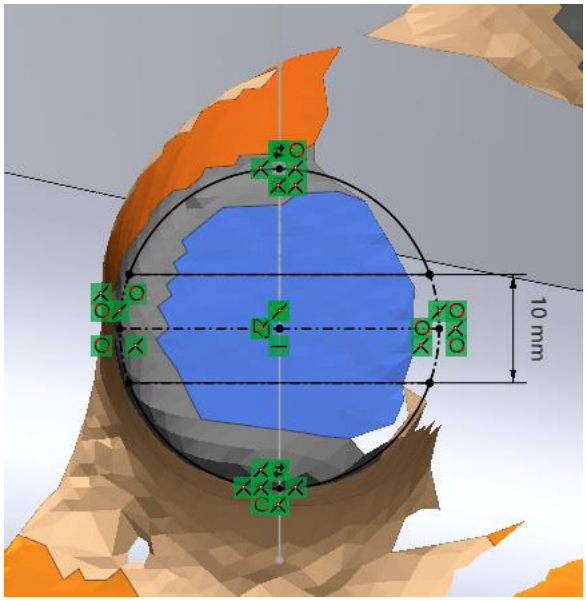


Figure 503

7. Insert a **Create Surface>Extrude** to generate one side face of the pin at a time.

- **Sketch 15** will be the **Base Sketch**, but only have one **Sketch Chain** as the **Profile** at a time.
- Select **Custom Direction** and choose the angled centerline from **Sketch 14(Mesh)**.
- Place a **check** next to **Draft** and set the angle to **4.5°**, as calculated in the sketch, so the extrusion will match up with the solid line.
- Set the **Method** to **Up To Vertex** and select the end point of the line used for Custom Direction.
- **Accept** the Extrude**.**

8. Repeat Step 7 to the other profile from Sketch 15.

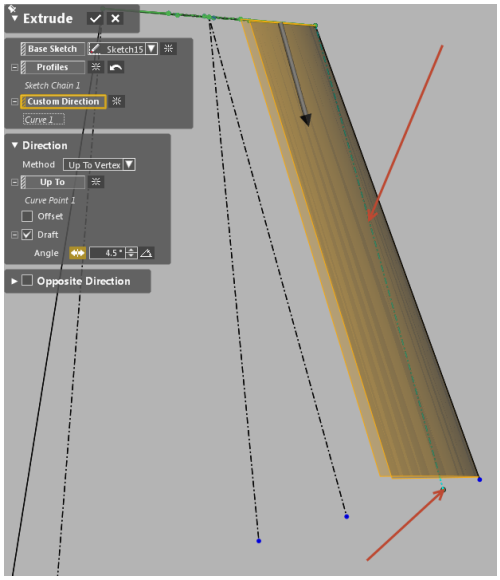


Figure 504

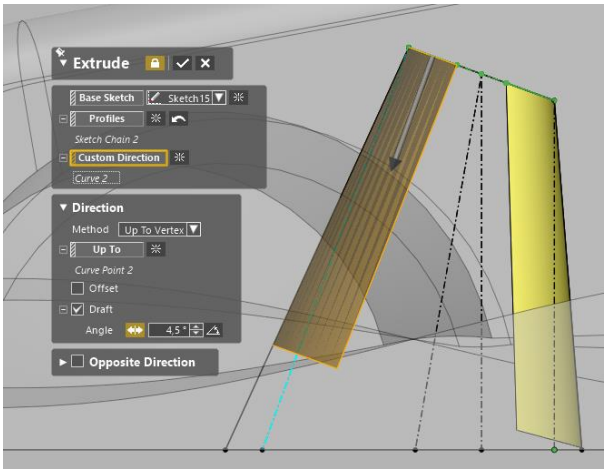


Figure 505

9. To join the two open surfaces, use the **Create Surface>Loft** command.

- The two facing edges will be the **Profiles**.
- Apply a **Curvature To Face**, **Start** and **End Constraint**.
- Add a **Check** to **Add Side Curve Influence**.
- Accept the Loft.

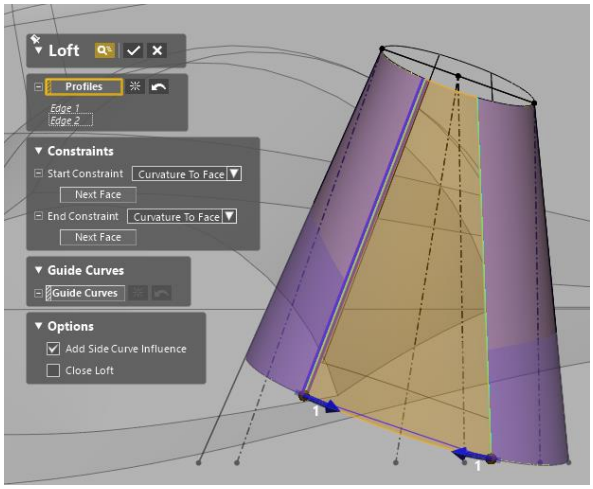


Figure 506

10. Use the **Edit>Fill Face** command to close the top and bottom faces of the pin and create a solid body.

- Select the four edges of the circular opening for **Edges**.
- Apply a **check** next to both options in **More Options** to create a closed entity.
- Accept the command.

11. Repeat Step 10 to the slot shaped opening of the pin.

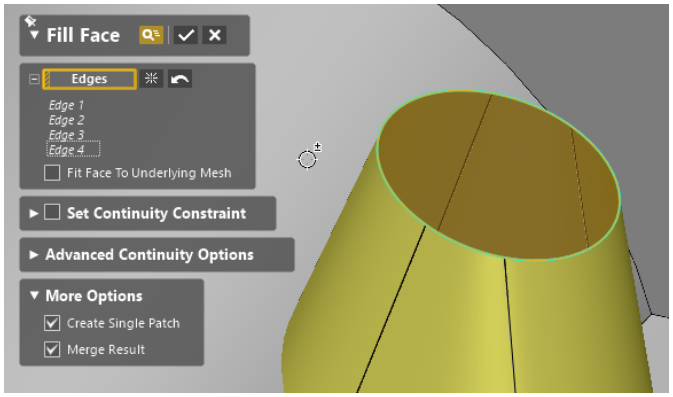


Figure 507

Fillets

1. Use the **Edit>Boolean** command to create a single solid body.
   - Set the **Operation Method** to **Merge**.
   - Select all solid bodies as **Tool Bodies** then accept the command.

☞ *In the **Model Tree**, only one Solid Body should remain.*

2. Turn on the **Deviation for Body** to see where all the Fillets are needed.

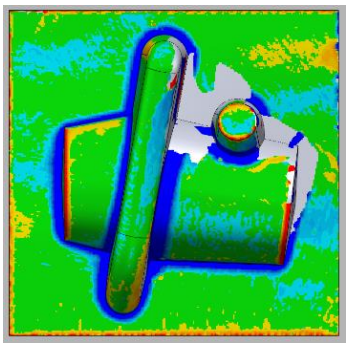


Figure 508

3. Choose the **MODEL>Edit>Fillet** command to generate the rounded edges.
   - Use a **Constant Fillet** on each of the fillets around the cones.
   - Select an edge of the Solid Body, then press the **Magic Wand** button to Estimate the Radius from the Mesh. This found value can be adjusted to the nearest round number.
   - Press the **Add Radius** plus sign button to insert a different size fillet in the same command.
   - Select a different edge and press the Magic Wand button again to read this fillet.
   - Repeat on the other cone fillets then accept the Fillet command.
4. Create a new **Fillet** for the fillet where the casting body meets the freeform face.
   - Select an edge then press the Magic Wand to find the value.

☞ *Since this fillet was done after the cone edge fillets, all the edges should be tangent making only a single click to select the edges necessary. Check all needed edges are selected.*

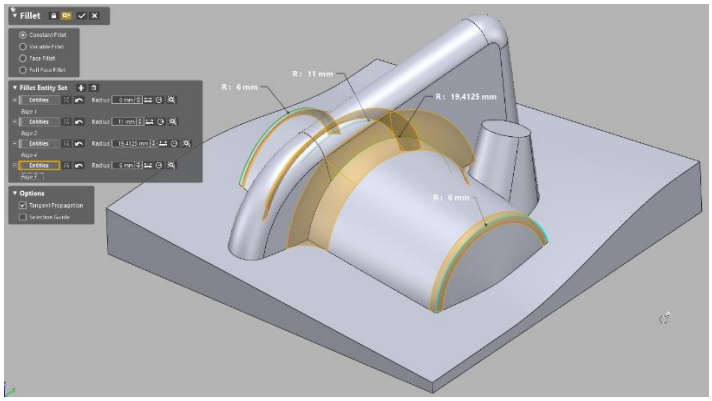


Figure 509

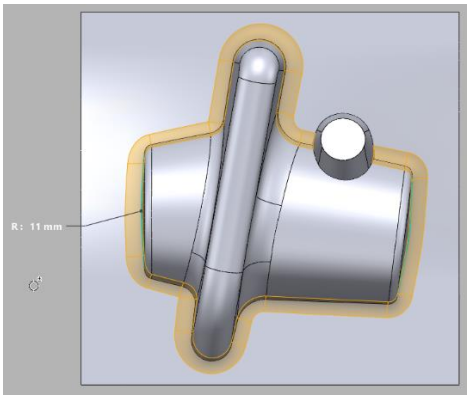


Figure 510

5. Create one more **Fillet** around the edges of the pin.

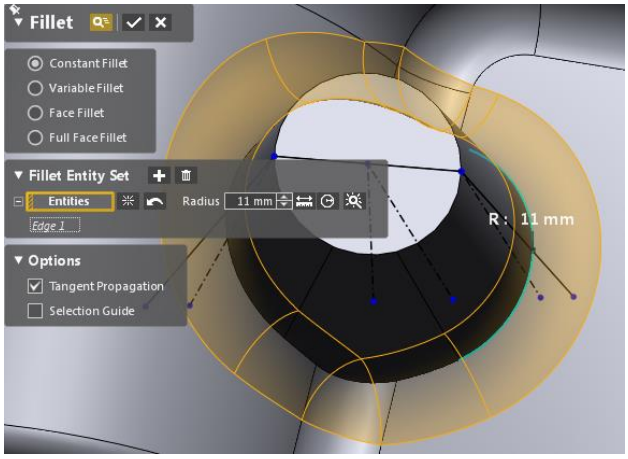


Figure 511

6. Use a **Fillet>Face Fillet** to define the top edge of the pin feature.

☞ *A width of the fillet segment would change with the angles of the faces of the pin in a constant fillet. The Face Fillet would keep a constant width even with changes to the face angles.*

- For the first set of **Faces**, choose only the top face of the pin.
- Select the green based **Faces** set and click on the side faces of the pin.
- Turn the Deviation for Body on to see what **Radius** would be best.
- Change the Radius to **5 mm** and accept the Fillet.

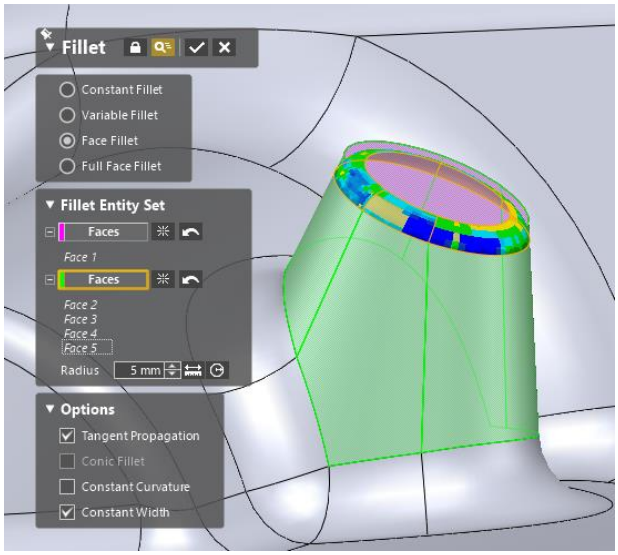


Figure 512

7. Check the different Deviations of the finished solid model.

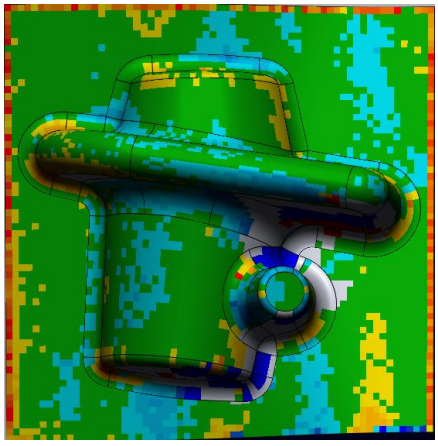


Figure 513

Figure 514

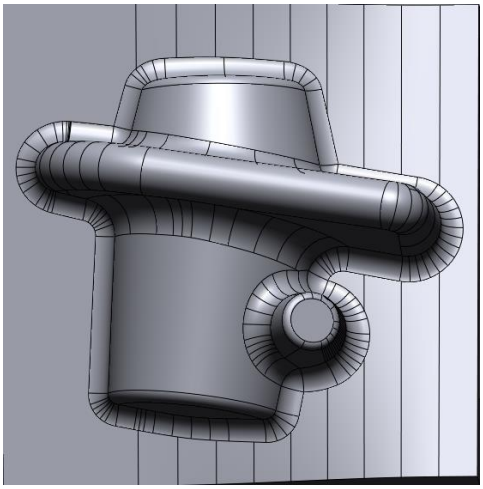


Figure 515

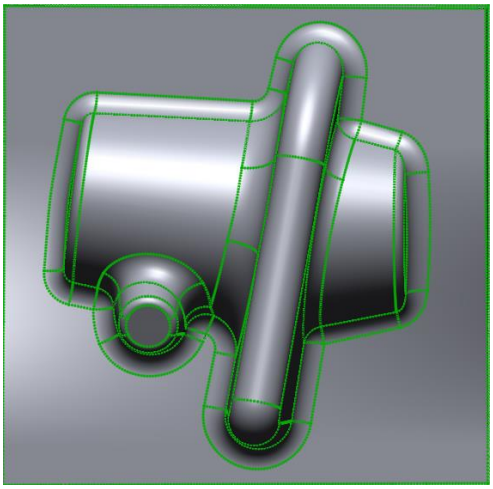


Figure 516

End of Activity