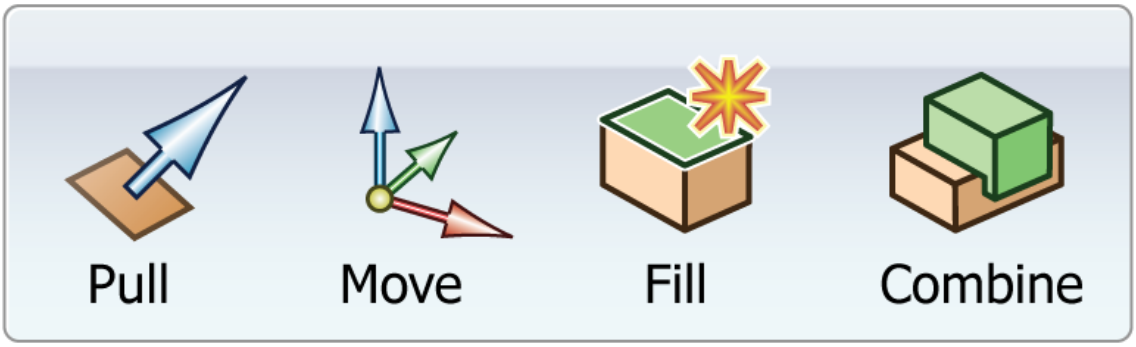
SpaceClaim Basic Training Notebook

SPACECLAIM
CORPORATION

Contents

Introduction

SpaceClaim is a Direct Modeler that is unlike any typical CAD tool. SpaceClaim is the easiest to use and fastest to learn 3D modeling tool. If you ever used a typical CAD tool, you may be familiar with features and having a feature tree to edit. SpaceClaim is a direct modeler and does not have features or a feature tree. Instead of editing the model by creating and changing features in a tree, you actually touch the model to make changes. Think of having virtual play-dough or building blocks on your desktop that you can edit with your hands. You select faces, edges, vertices etc. and primarily use 4 simple tools to edit them



Pull, Move, Fill and Combine are the 4 main tools of SpaceClaim. The vast majority of creation and editing in SpaceClaim is done with these 4 tools. Many of the other tools in SpaceClaim are just automated versions of what these tools can do manually.

This SpaceClaim Basic Training class covers the basics of using these 4 tools to create and edit, along with how to Select, Create assemblies, Drawings, Import, and Repair and edit other files types.

Quick Reference Card

Select

Single-click	Click to select one object
Double-click	Double-click to cycle through Face / Edge / Curve loops
Triple-click	Triple-click to select all the faces of a body
Ctrl +	Add or remove an object from the selection
Shift +	Select all objects between the original selection and this object
Alt +	Select a driving or alternate object for many tools (in blue)
+	Select all objects completely within the box; Ctrl + Drag toggles selection
+	Select all objects partially within the box; Shift + Drag adds to selection
Scroll	Select other objects under the cursor
+ Drag	• Click to get command menus • Drag to invoke gesture shortcuts
	Use the Select-Bounds toolguide to stop the propagation of selected faces and edges
	Revert to the last set of selected items
Esc	To exit current tool and return to selection

Orient

	Spin
Ctrl +	Zoom
+ Scroll	
Shift +	Pan
+	Snap View
Shift + Double-click	
H	Home View
+ V Plane selected	Plan View
Double-click	Zoom Fit selection
Anything selected + Z	
Previous view	Next view

Dimension

Spacebar	Edit dimension while dragging
Tab	Toggle between editable dimensions
Shift +	Hover over geometry to dimension from it while sketching
Enter	• Accept the dimension value • Complete most tools
Shift + Alt +	Create temporary objects
	Create a linear / angular ruler dimension
Ctrl + G	Create Group and store ruler dimension
14.75	Drive a ruler dimension with an Annotation dimension
U	Select an Up To reference

Files

Ctrl + S	Save
Ctrl + Shift + S	Save As
Ctrl + Alt + S	Save As New Version

Quick Reference Card (cont.)

Sketch & Section

- Select sketch curves and use model edges
- Select faces and edges by their section lines and points

Drag sketch and section curves

Move and Rotate curves

Splines

Select entire spline to move

- Add spline point
- Remove spline point
- Measure curvature
- Drag endpoint tangency

Scale/Rotate

Drag, Rotate, Re-center

Fix aspect ratio Reorient box

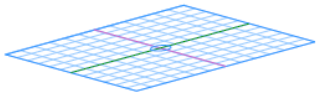
Ctrl • Press to copy in Drag and Move
• Toggle extension indicators

Shift • Toggle snapping to grid
• Grab dimension reference

Esc • Cancel an in-progress action

Snaps

On curve, Grid Point, Mid Point, End Point, Multiple snap types



Back to 3D, New sketch grid location (re-select), Move sketch grid, Plan View

Indicators

Tangent, Extension, Arc center, Right angle, Angle, Golden rectangle, Square

Dimension Anchors

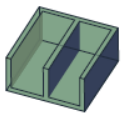
Select reference curve or point [also with Shift]

Move dimension base point [also with blue circle]

Change dimension reference angle [also with blue line]

Associations

Offset and Shell



Create with and tools

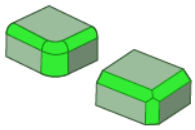
☐ Show offset baseline faces

Pull option: ☐ Maintain offset

Change face property: None (This face, Other face, Center)

Change face offset

Round and Chamfer

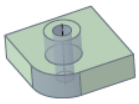


Create by pulling edges

Toggle with +

Remove with or by changing face property

Coaxial

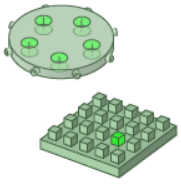


Create: + Create coaxial group

Remove: + Ungroup Coaxial Faces

Change: + Set Cylinder Diameter

Pattern

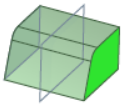


Create with Move + ☑ Create Patterns

Remove with + Unpattern Member

Works on:
- Solids and surfaces
- Protrusions and depressions
- Components
- Imported features of interest

Mirror



Set up: face + face +

Add to: Select plane then Body or Faces (Ctrl)

Remove: then Faces (Ctrl)

www.myspaceclaim.com

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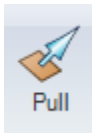
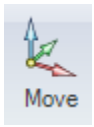
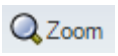
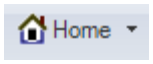
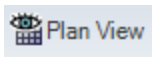
Common Shortcuts and Selection Highlights

	Pull (P)
	Move (M)
	Fill (F)
	Combine (I)
	Select (S)

	Undo (ctrl + Z)
	Zoom Selected (Z)
	Home (H)
	Plan (normal) View (V)
	Cancels Current Action
	Freezes Onscreen Dimensions

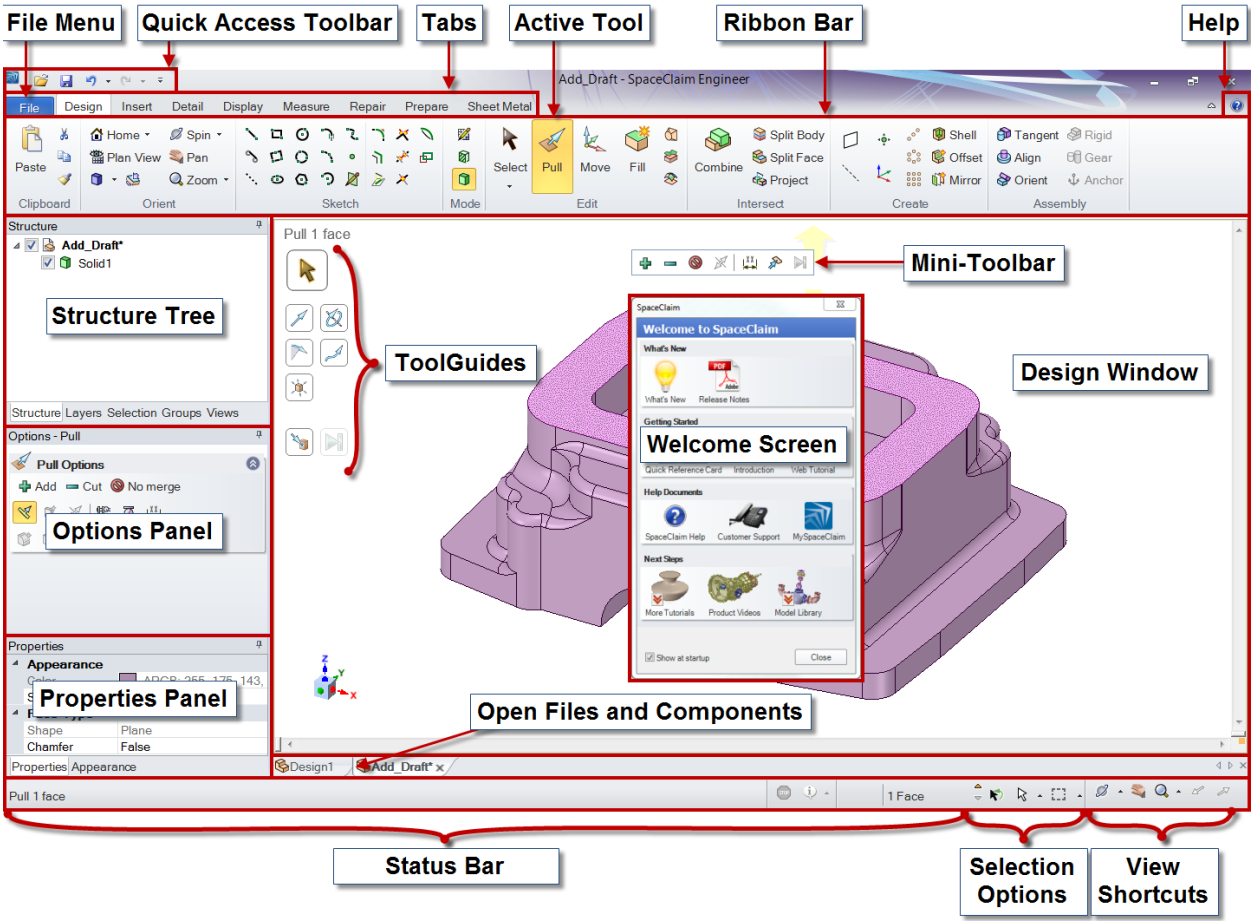
- Left click to select faces and edges
- Hold down the CTRL key to select multiple faces or edges
- Left mouse drag to box select
- Box select uses Smart selection filter
    - Left to right selects things entirely in the box
    - Right to left selects anything touching the box
- Query select by rolling the mouse wheel through the model
  Great for selecting hidden or small faces/edges

User Interface

SpaceClaim's user interface is designed to be very friendly and intuitive to find the tools and options needed. SpaceClaim has adopted the same Fluent User Interface that Microsoft Office made popular back in 2007, with a new Ribbon Bar and Tab style organization of tools. This section will explain where different tools are located, where to find different types of options, what all the different panels in SpaceClaim are, and will also introduce some very useful and unique features of SpaceClaim

Overview

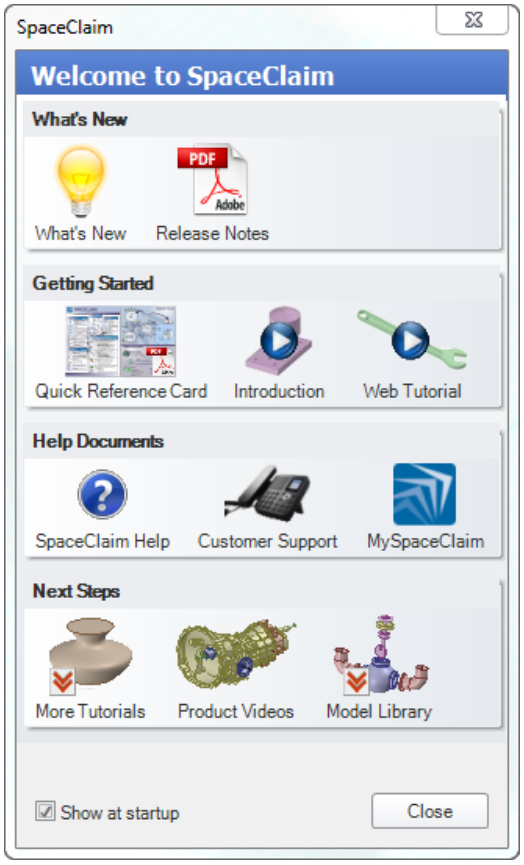
The image below shows an overview of the SpaceClaim User Interface. The following sections will provide more detail on each of these sections, what they do, and when to use them



The Welcome Screen

When you start SpaceClaim, a window will appear in the middle called the Welcome Screen. Once this window is closed, it can only be shown again by closing and opening SpaceClaim. Most of the buttons found on this Welcome Screen can be found within SpaceClaim or at www.spaceclaim.com. SpaceClaim Help can be loaded insides of SpaceClaim and some of the tutorials are found within the help. Customer support and product videos can be found on the SpaceClaim webpage.

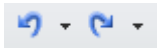
One of the most useful links in the welcome Screen is the **Quick Reference Card**, which is included on page 2 of this manual and the first page of the SpaceClaim Help. The Quick Reference Card is a 2 page document which is great when first getting started with SpaceClaim. It includes all the common Mouse and Keyboard commands, and explains how to do many things like Spin, Pan and Zoom a model, or how to select things.



Quick Access Toolbar

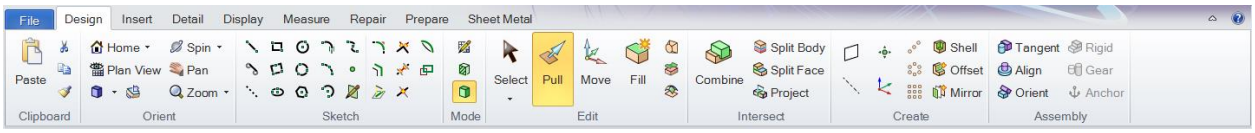
In the most upper left section of the SpaceClaim Window is the Quick Access Toolbar. It is a customizable toolbar where you can add the most frequently used tools. On the far right is a drop down menu that allows you to choose which tools is on the QAC, along with an option to change its location to below the Ribbon Bar

2 Buttons found in the QAC that aren't found elsewhere are the **Undo** and **Redo** buttons, the curved arrows next to save. Each has an arrow which is a drop down history list, so you can Undo or Redo more than 1 step at a time.



Ribbon Bar

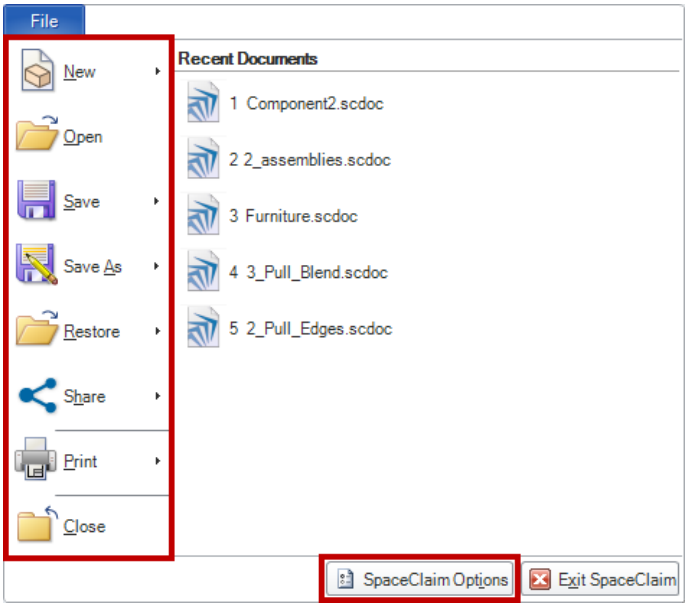
Below the Quick Access Toolbar are the Tabs and Ribbon Bar, which also includes the File Menu. The File Menu Tab is explained below, and the other Tabs will be explained later in this section



File Menu

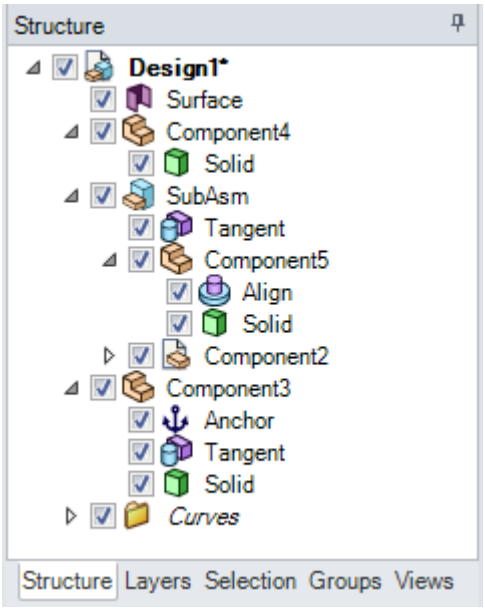
The File menu in SpaceClaim is just like the File menu I all Microsoft Programs. It's located in the upper left corner of the window, and clicking on it brings up a menu that allows you to Create, Open, Save, Share and Print files, parts, drawings and more.

At the Bottom of this window you will find the SpaceClaim Options, which are the type of options you change only once, or very rarely. The next page explains the Options Panel which has more frequently changed options.



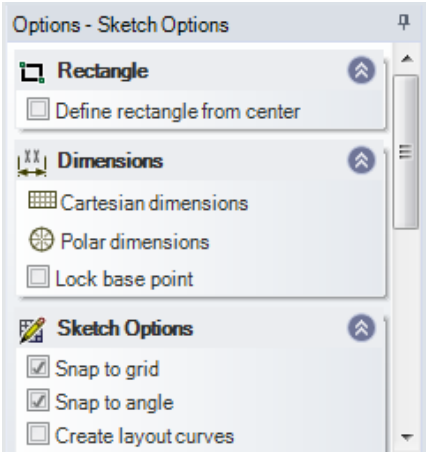
Structure Tree

Below the Ribbon Bar on the left side is the Structure Tree Panel. The Structure Tree contains and displays a list of all the Objects in the design. This can include, but not limited to Solids, Surfaces, Components, Subassemblies, Sketch Curves, Drawing Sheets and Assembly Conditions. Notice at the Bottom of the Structure Tree Panel are 4 other tabs that provide different functionality and are discussed in other training courses. There is an entire section dedicated to the Structure Tree in this manual.



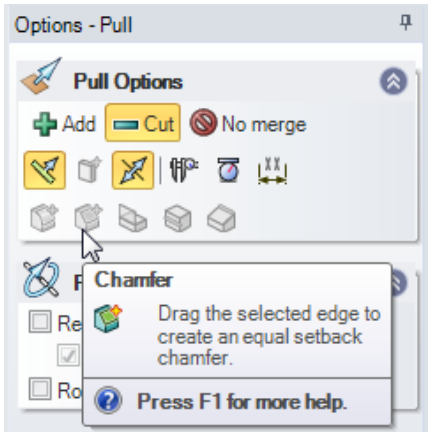
Options Panel

The Options Panel is found directly below the Structure Tree on the Left side of the window. This Options Panel is specific to whatever Tool is currently active. It typically displays the most common options that are changed on a frequent basis. Options that tend to be changed less often are found in File\SpaceClaim options, which will be explained later in this section. The Options in this panel are reset every time a tool is turn off and on. For some tools, there may be a scroll bar on the right to view additional options. Some tools have no options and this panel will be blank.



Shown to the right are 2 different options panels for 2 different tools: Rectangle and Pull. Notice that some of the options have check marks, and other options are toggles that get highlighted when active, and unhighlighted when not active.

There are often options that are greyed out. Hover of those options to get a Tooltip with an explanation of the option which should tell you why it's greyed out, and what will enable it. These Tooltips show up for every options and tool hovered over



ToolGuides

Inside the Design Window on the top left side next to the Structure Tree are buttons called ToolGuides. Some of the ToolGuides are the same in different tools and some ToolGuides are unique for certain tools. Some tools don't have and ToolGuides. There are 2 types of ToolGuides:

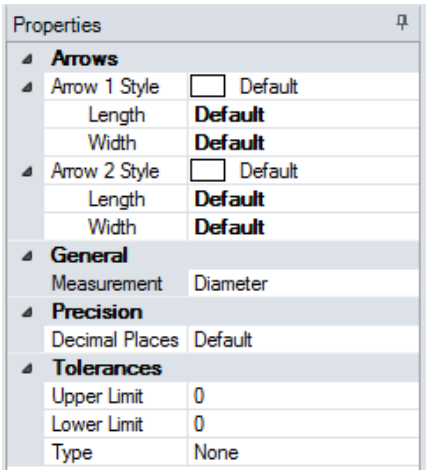
Some ToolGuides like the ones for Pull and Move **change the default behavior of the tool**, similar to the Options Panel. Notice Pull and Move have some of the same ToolGuides, and some that are different.

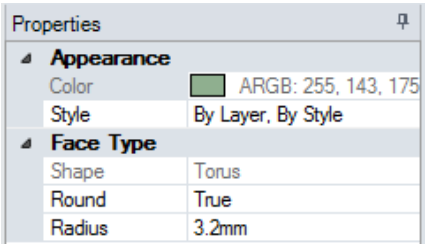
Some ToolGuides like for Combine **Guide you through the steps of the Tool**. Each ToolGuides is used in order to perform step 1, then step 2 etc. of the Tool.

Properties Panel

The Properties Panel can be found directly below the options panel on the left side of the screen towards the bottom. What is shown in the properties panel is based on what is selected. If no tool is on and nothing is selected, the properties panel will be blank. With a piece of geometry like a face selected, the properties panel will provide info like color and shape, often with additional info that can often be changed from the panel, like radius for a round, which can also be changed with the Pull tool.

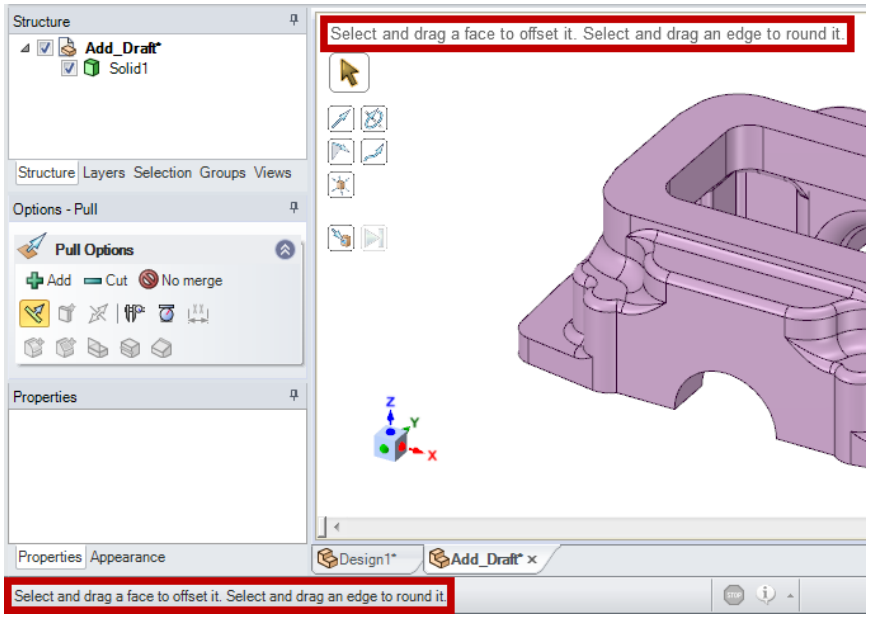
Some selected objects, like dimensions, have properties that can only be changed from the properties panel, like the tolerance type of weather a measurement is a diameter or radius.





Status Bar and Message

At the bottom of the screen below Properties is the Status Bar which displays a Status Message. The same status message is also in the top left corner of the Design Window, which can be turned off in SpaceClaim Options. Just like ToolGuides and Options, it changes with every tool. But more importantly, it tells you exactly what SpaceClaim is expecting you to do right now. With a tool like Pull that does so much, it will give you a hint to get started, but you are not limited to what the message says. For a Tool like Combine with a specific order of operations, it will tell you what step you are on.



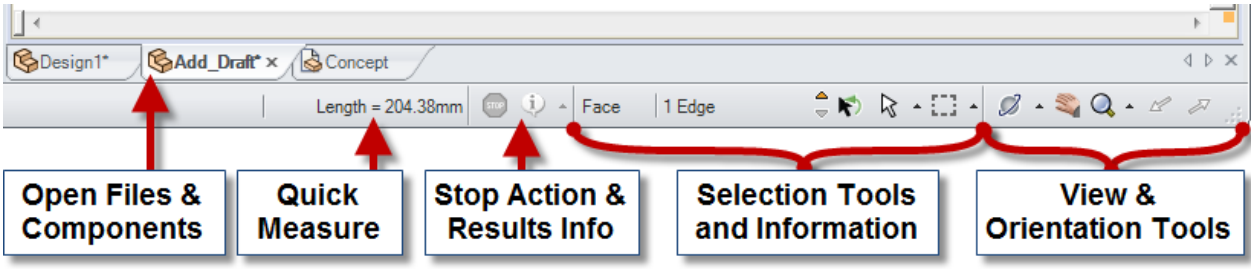
To the right of the Status Bar Message are the Tabs that represent all the open files and components. Also in the Status Bar is a Quick Measurement panel which can give you the length of an edge, distance between edges or even the XYZ coordinates of a selected point.

There is a Stop button that works just like pressing Escape to stop a current process, and right next to that is an information bubble that pops up to report errors or other results from a tool

To the tight of the results info are 2 selection panels. The 1st tells you what type of geometry the mouse is hovering over, and the 2nd tells you what it currently selected.

The orange, black and white arrows are tools for selection that are discussed in the Selection Section

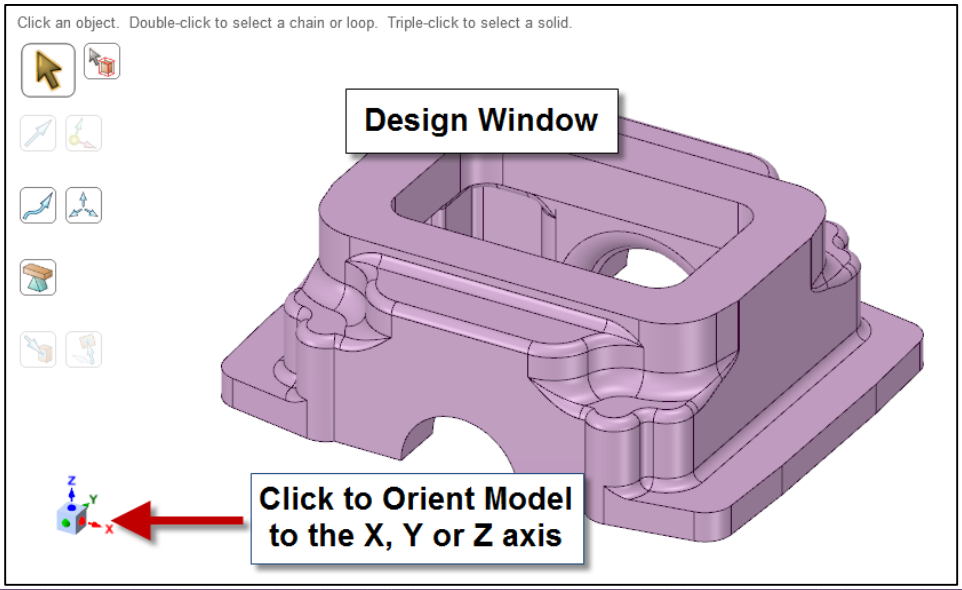
The last group of buttons are view and orientation buttons that are similar to the orientation group in the design tab. Many of these can be performed with just the middle mouse button.



The Design Window

Above the status bar and below the ribbon bar is the Design Window. The Design Window is the 3D space that the model resides in. Any 2D or 3D geometry like sketch curves, surfaces, solids components and assemblies will be displayed in the Design Window. The ToolGuides previous discussed are found in the design window on the left hand side

In the bottom left corner of the Design window is a useful widget that allows you to spin and orient the model to the X, Y and Z Axis. It looks like the world origin and you just click the arrows to orient



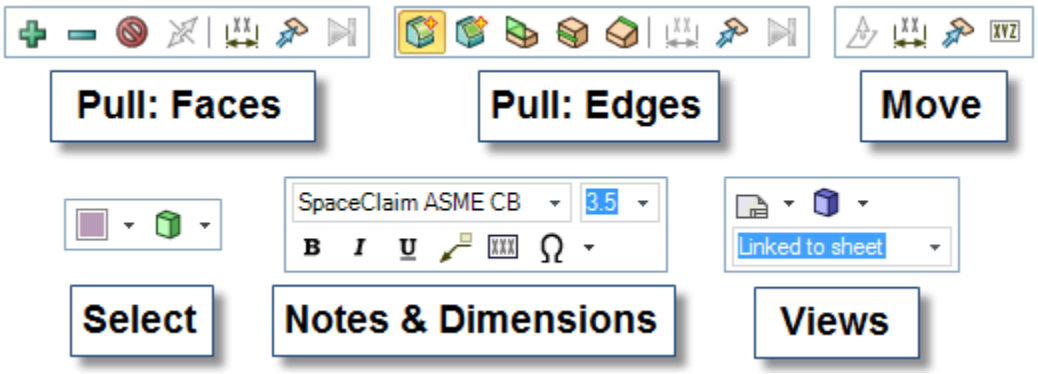
The Mini-Toolbar

The Mini-Toolbar is a menu of common options that appears in the design window when you click on something like a face, edge or dimension. The Mini-Toolbar that appears is based on both the tool you have turned on, and the geometry you have selected.

Everything in the Mini-Toolbar can be found in either the Options Panel or ToolGuides on the left, or in some cases the Ribbon Bar. Below are some examples of the most common Mini-Toolbars.

The Mini-Toolbar will appear near your mouse cursor every time you click something in the design window, when tools like Pull and Move are turned on. Move your mouse cursor near it to use it, moving your mouse away from it will make it disappear. Click on the geometry again to make it appear

Learning to use the Mini-Toolbar will save you a lot of time to use different options.



The Ribbon Bar and Tabs

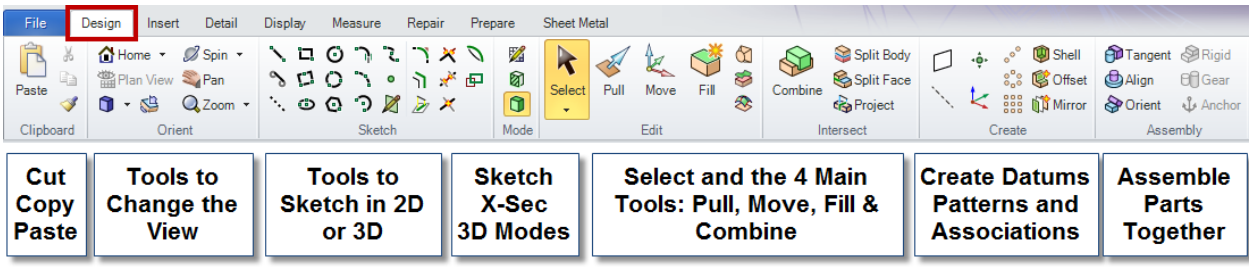
At the top of SpaceClaim you will find the Ribbon Bar and Tabs just like in current Microsoft Office Programs. The Ribbon Bar is where you will find the majority of the Tools you will use. The Tables at the top of the Ribbon Bar organize the Tools by Application and use. Clicking on the Tabs will change what tools are shown in the Ribbon Bar. Within each Ribbon, the Tools are broken up into Groups. The Group Names are found at the bottom of the Ribbon Bar, and each Group is separated from the next Group by a vertical line. A general overview of the Tabs and Groups will be given in this section. A more detailed description of the tools within a group will be provided in following sections as needed.

The main Tabs and Ribbon Bar are not customizable. You can add individual tools to the Quick Access Toolbar described on page 6.

SpaceClaim has an open API (application programing language) which allows you to create your own tabs using a .net based languages like C++. In the SpaceClaim install directory (usually c:\program_files\SpaceClaim) you will find SpaceClaim.Api.VXX folders for each of the SpaceClaim API versions. The SpaceClaim API lets you create your own Tabs of SpaceClaim tools, communicate with other programs and create custom add-ins with a little programming knowledge. Inside of the folder you will find a developers and style guide and a sample add-in

If you don't have the expertise to use the API, you can contact SpaceClaim or your reseller for support.

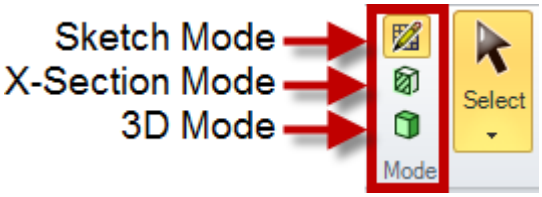
The Design Tab



The 1st tab of tools after the file tab is the Design Tab. This is where SpaceClaim users spend most of their time. The image below gives a brief description underneath each group. Most of these tools will be explained in greater detail in the following sections of this manual

Modes: Sketch, X-Section and 3D

Sketch is the first mode and every new design starts in sketch mode. Sketch mode also you to create sketch curves to start a design, or to sketch on and edit an existing design.

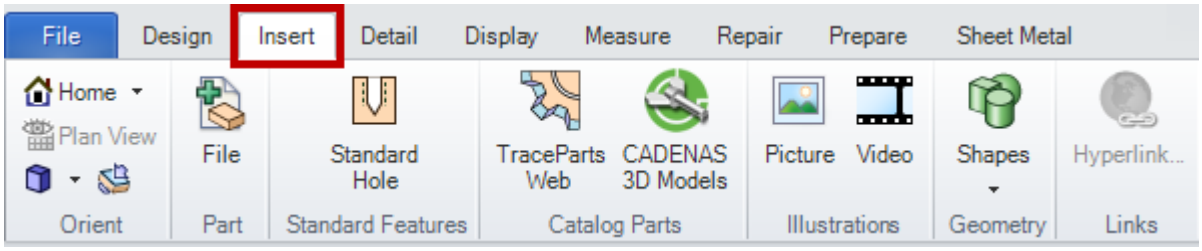


3D is the mode that most of the geometry editing takes place. Pull, Move Fill and Combine are most used in 3D. You can also sketch in 3D by snapping to existing objects.

X-Section is a mode that's a combination of 3D and Sketch. You are able to perform all the same operations as 3D, but through the cross section of a part. This is useful for working inside of a part when it's difficult to work in 3D.

1 of these 3 modes will always be on. Some tools will automatically switch modes: Sketch tools turn on Sketch Mode and Pull turns on 3D Mode.

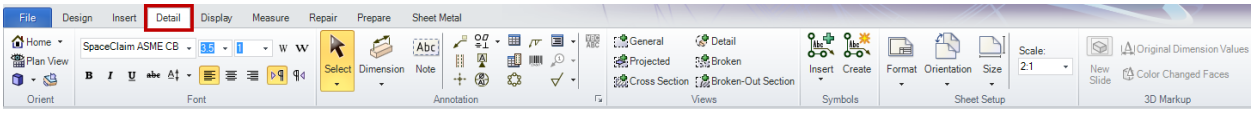
The Insert Tab



The insert tab allows to bring add files from your computer to an existing design, or search one of our online libraries. You can also insert standard holes with manufacturing info, images and some standard shapes. Each group is pretty self-explanatory.

The Detail Tab

All the tools needed to create a drawing sheet are found in the Detail Tab. There are tool to dimension

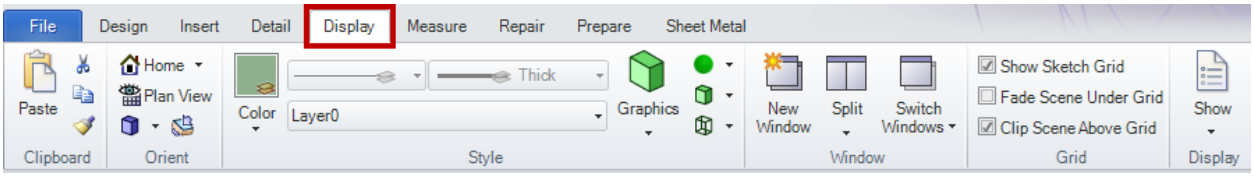


and annotate a part in a drawing sheet, in addition to create views, custom symbols and change the sheet setup.

The Detail Tab can also be used to add dimensions and annotations directly to a model in the design window in 3D. This is commonly referred to as PMI or MBD (Product Manufacturing Info and Model Based Definition)

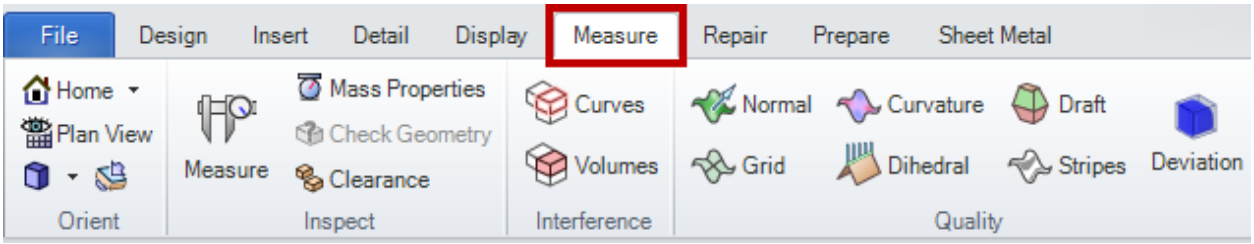
The Detail Tab is the largest tab and has the most tools. Some of the Groups may be condensed based on the size and resolution of your monitor

The Display Tab



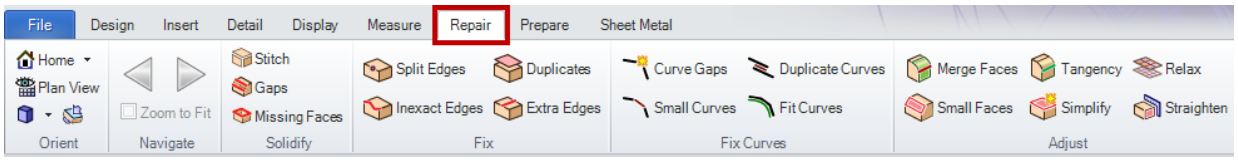
All of the visualization settings can be found in the Display Tab. You can change the color of solids, faces and edges, change layers and if a model is displayed at shaded or some version of wireframe. Edge display for tangency and meshes can be shown or hidden, along with other options like the world origin in the last Show group. The Grid group has 3 settings that are useful to have on or off while in Sketch and Cross Section Mode, depending on what you are doing in those modes. Most of the Style options will be greyed out unless you have something selected.

The Measure Tab



If you need to measure something that the quick measurement window in the status bar cannot measure, you will find the necessary measurement tool in the Measure Tab. In addition to a single Measure tool to find lengths, areas, distances, angles etc., there are tools to find Mass Properties, clearances, and also interrogate a model's surface quality. There is also a draft analysis tool and a deviation tool for comparing models or scan data to other models.

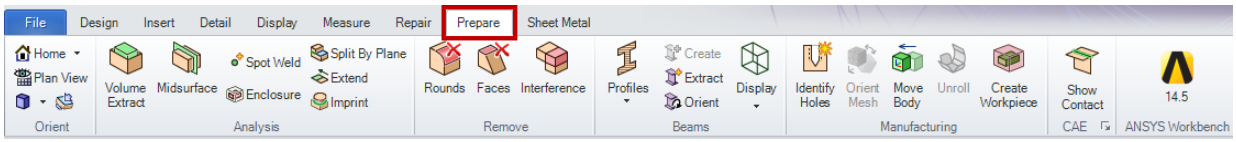
The Repair Tab



When importing data from other systems, whether using the native CAD file, or using a generic file type, many models import with corrupt geometry. The Repair Tab has tools to repair both 3D and 2D models, including solids, surfaces and sketch curves.

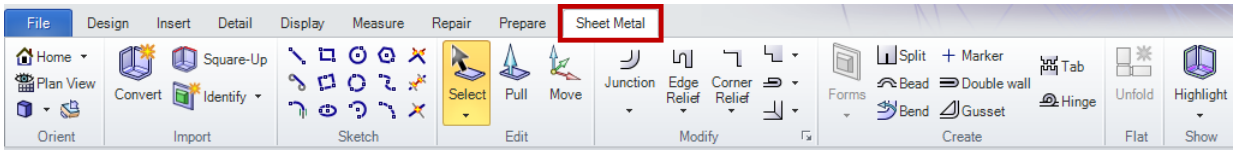
IMPORTANT: There are many tools in the Repair Tab, and each serves a very specific purpose. It is NOT recommended to import a CAD file and use every tool in the Repair Tab without understanding what issues the model has, and what each tool does to fix them.

The Prepare Tab



If you need to edit a part for Manufacturing or Analysis, the Prepare Tab is where you will find many of the tools needed. Some of the tools create bodies within or around existing models, and some remove unwanted geometry like rounds. Many of the tools in the Prepare Tab are very useful for applications outside of Manufacturing and Analysis, so don't be scared to check out what these tools do, as they are often useful for, but not limited to design, concept modeling, reverse engineering and 3D printing.

The Sheet Metal Tab



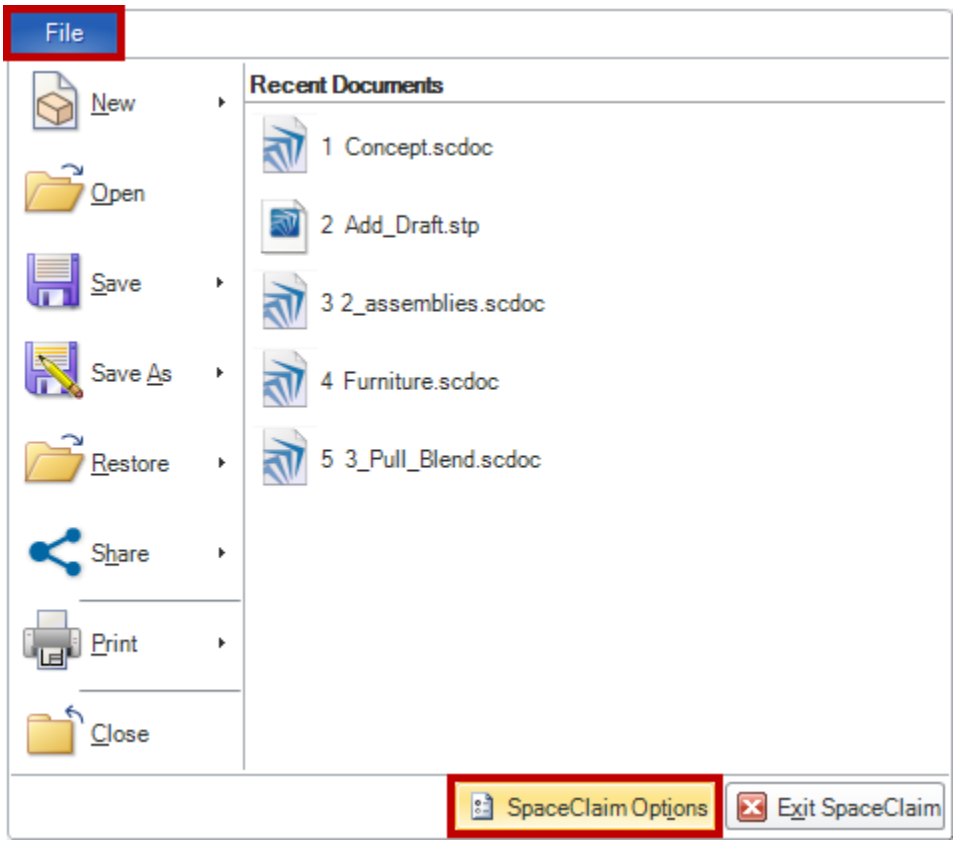
The Sheet Metal Tab is like the Design Tab but specifically for Sheet Metal. There is a group of sketch tools along with Pull and Move from the design tab. There are also tools to take an existing model created in SpaceClaim or other CAD program and convert them into a smart sheet metal part. SpaceClaim can easily add complex geometry like forms, gussets, double walls and hinges, and then unfold the model with a single click. The unfolded model can use a constant or variable K-factor or bend table, and provides manufacturing information like bend lines, bend dimensions and form location. The unfolded model can be exported to DXF for laser, plasma or any cutting method.

Additional Tabs

Any Tabs found past the sheet metal tab are add-ins. Add-ins can be created by users for free using the API described on page 11, by partners of SpaceClaim or any 3rd party program, in addition to being written by SpaceClaim or resellers. Keyshot is a popular 3rd party tab that creates photorealistic images of models from SpaceClaim. Another example is a gear wizard that automatically created gear models using inputs to drive the geometry.

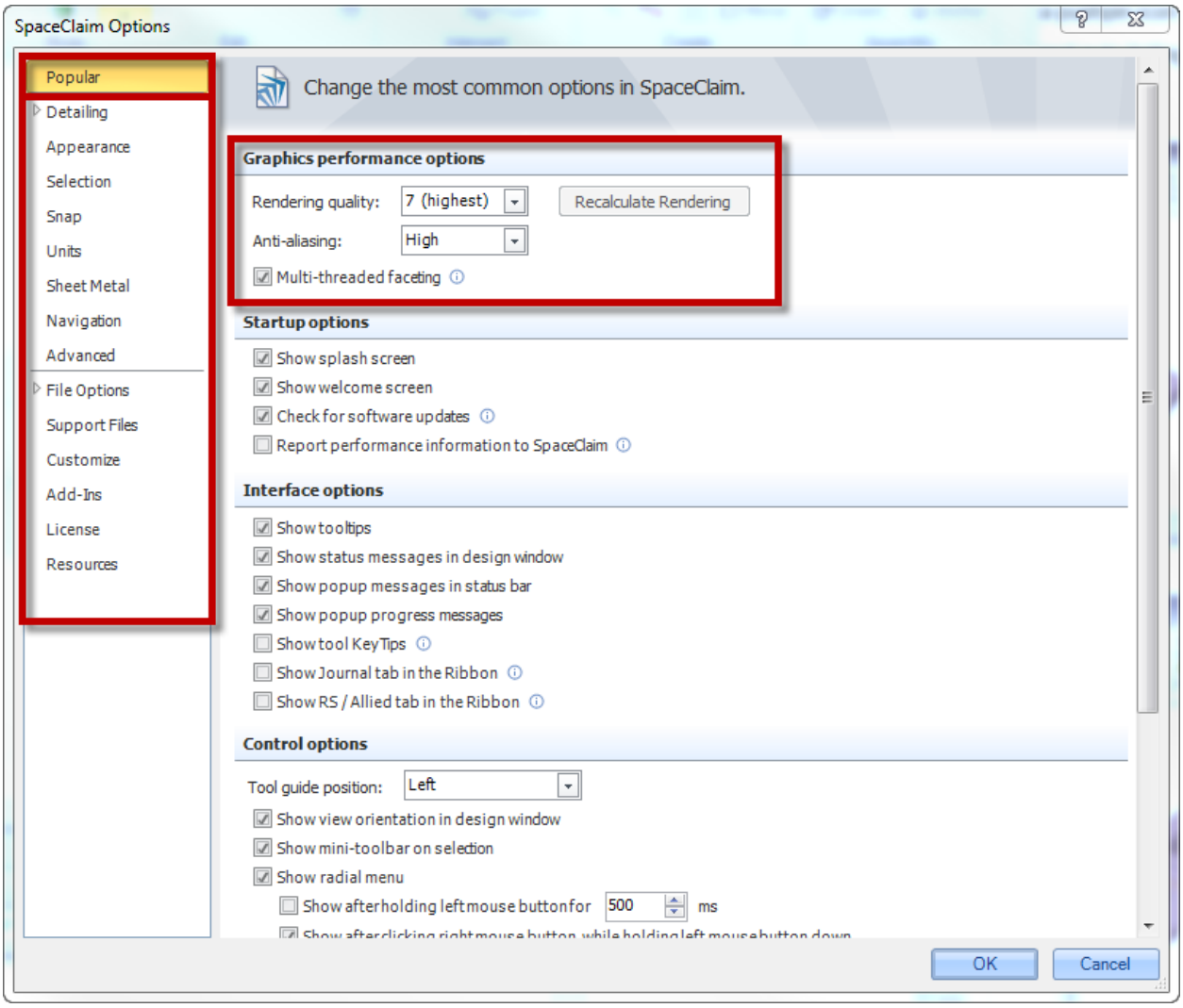
SpaceClaim Options

If you want to customize SpaceClaim, you want to visit SpaceClaim Options. In the File menu, there is a button at the bottom of the window to enter SpaceClaim options.



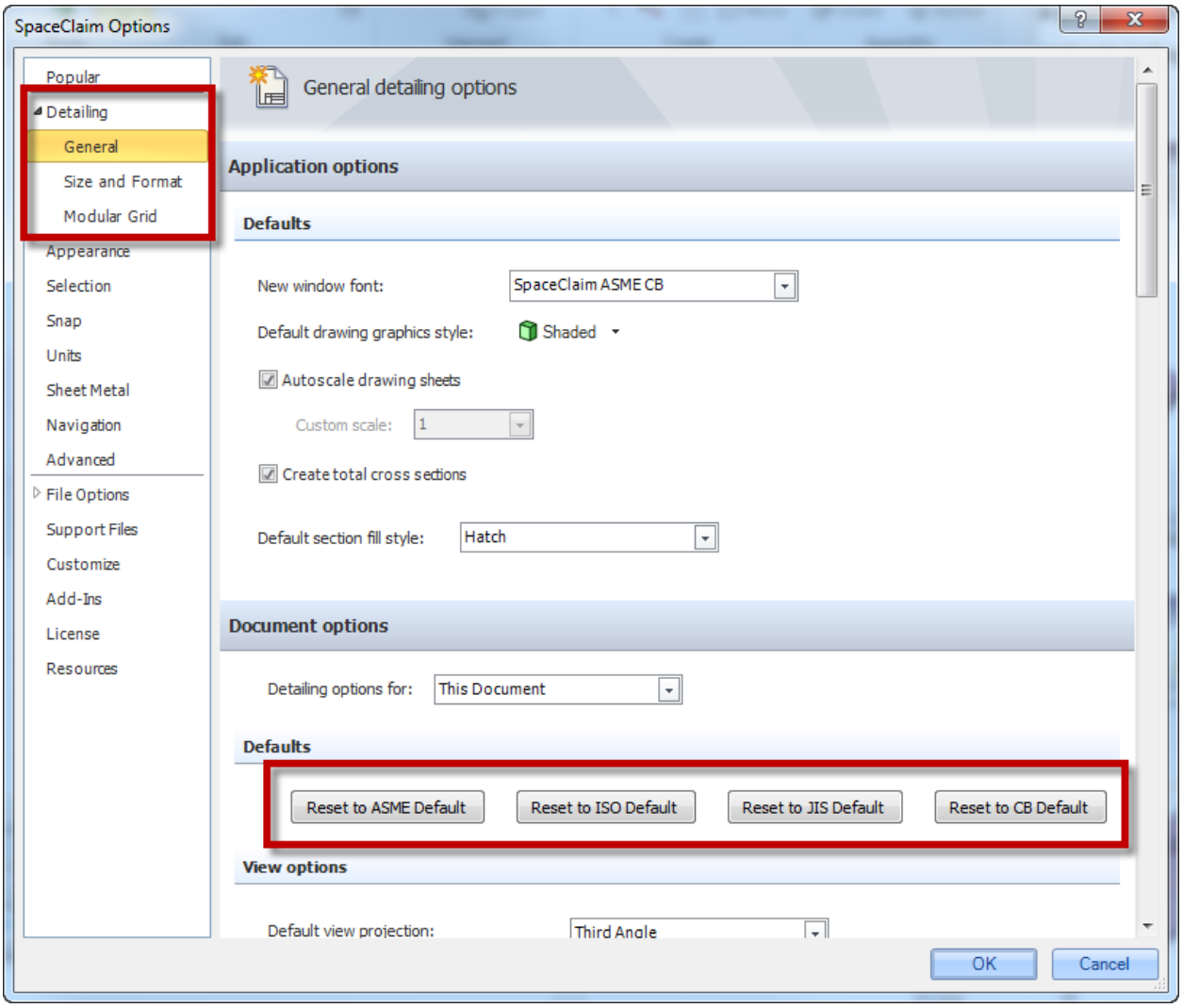
Similar to the ribbon bar, SpaceClaim Options is organized into Tabs, which are found on the left side of the SpaceClaim Options window. Every Option is explained in detail in SpaceClaim help, which will be discussed next. In this section some of the main settings that are changed most frequently will be discussed.

The first tab in Options is the **Popular Tab**. The very first option is the **Rendering Quality**, which determines how smooth rounded edges and faces look. If your model looked faceted, i.e. a cylindrical hole looks like a polygon, you probably need to turn up the rendering quality. Changing the rendering quality will change the size of the file, the higher the quality, the bigger the file



The Detailing Tab sets the defaults for annotating a drawing sheet or model, and many other settings for a drawing sheet. Notice there are 3 different tabs under Detail, General, Size and Format, and Modular Grid. General allows you to set presets like ISO or ASME, and below that is a long list of specific defaults.

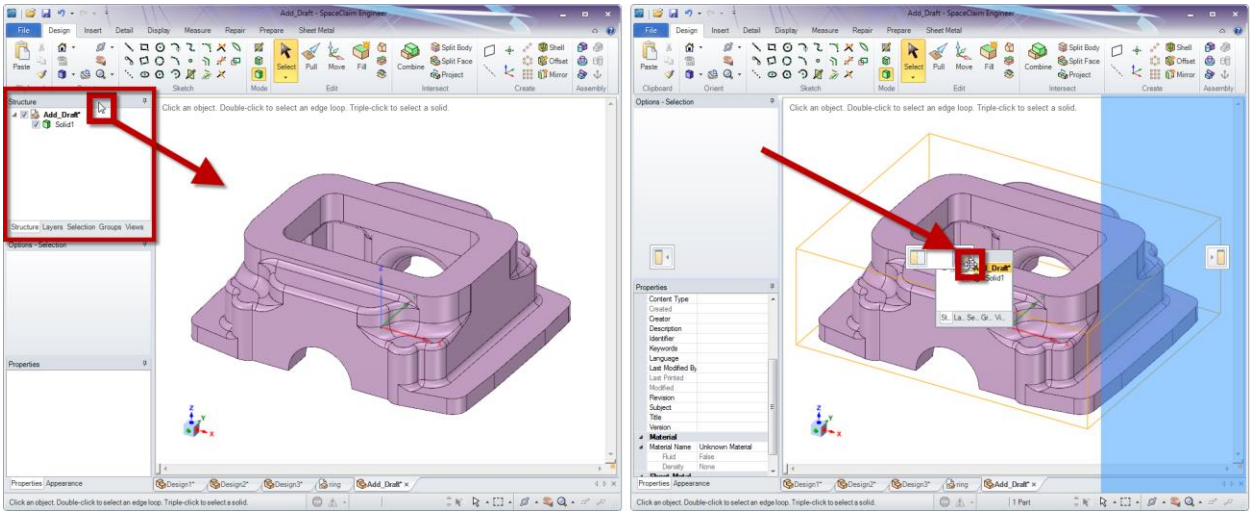
The Size and Format Tab is where you'd go to specify a default drawing template that you've created

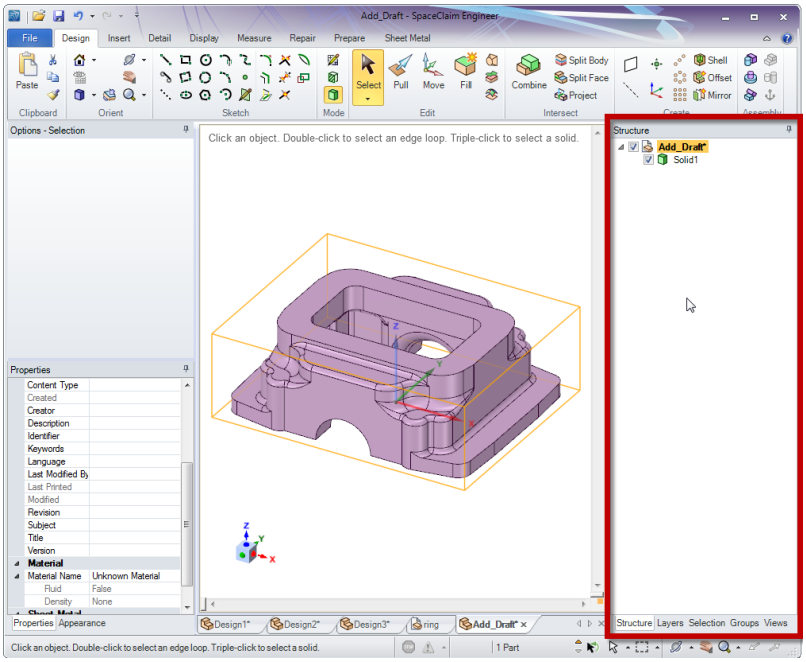


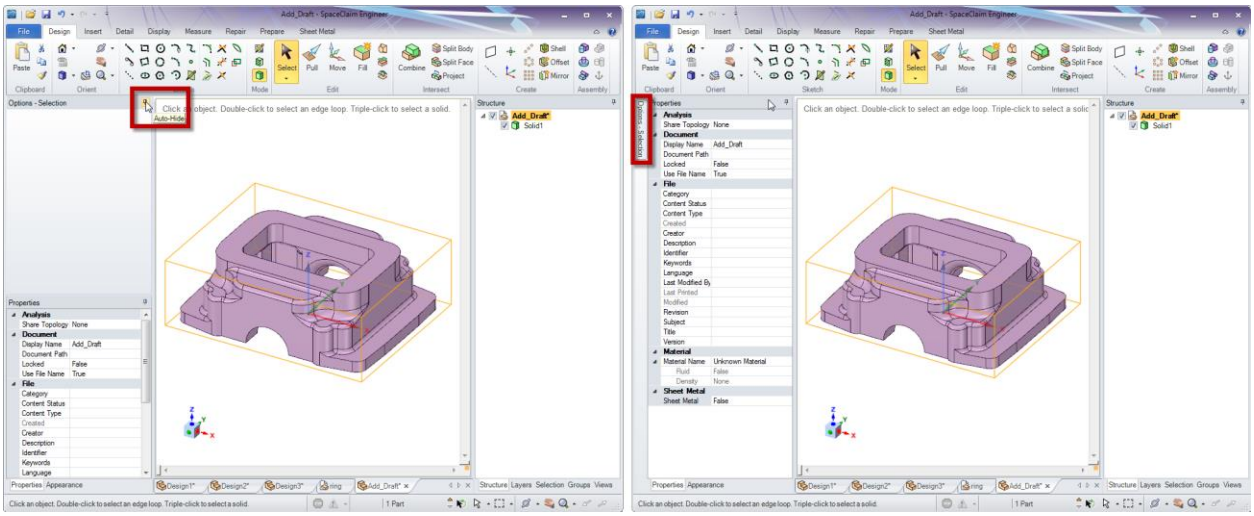
The Appearance Tab allows you to change all the colors in SpaceClaim, and there is one very important button on the right, Reset Docking Layout. The images on this page shows that you can change the location of the panels around the design window, by dragging the title bar of the panel around the screen. Once you start to drag any panel, boxes will appear on the left, right and center of the screen. If you drop the panel on one of these boxes, the panel will snap to the location.

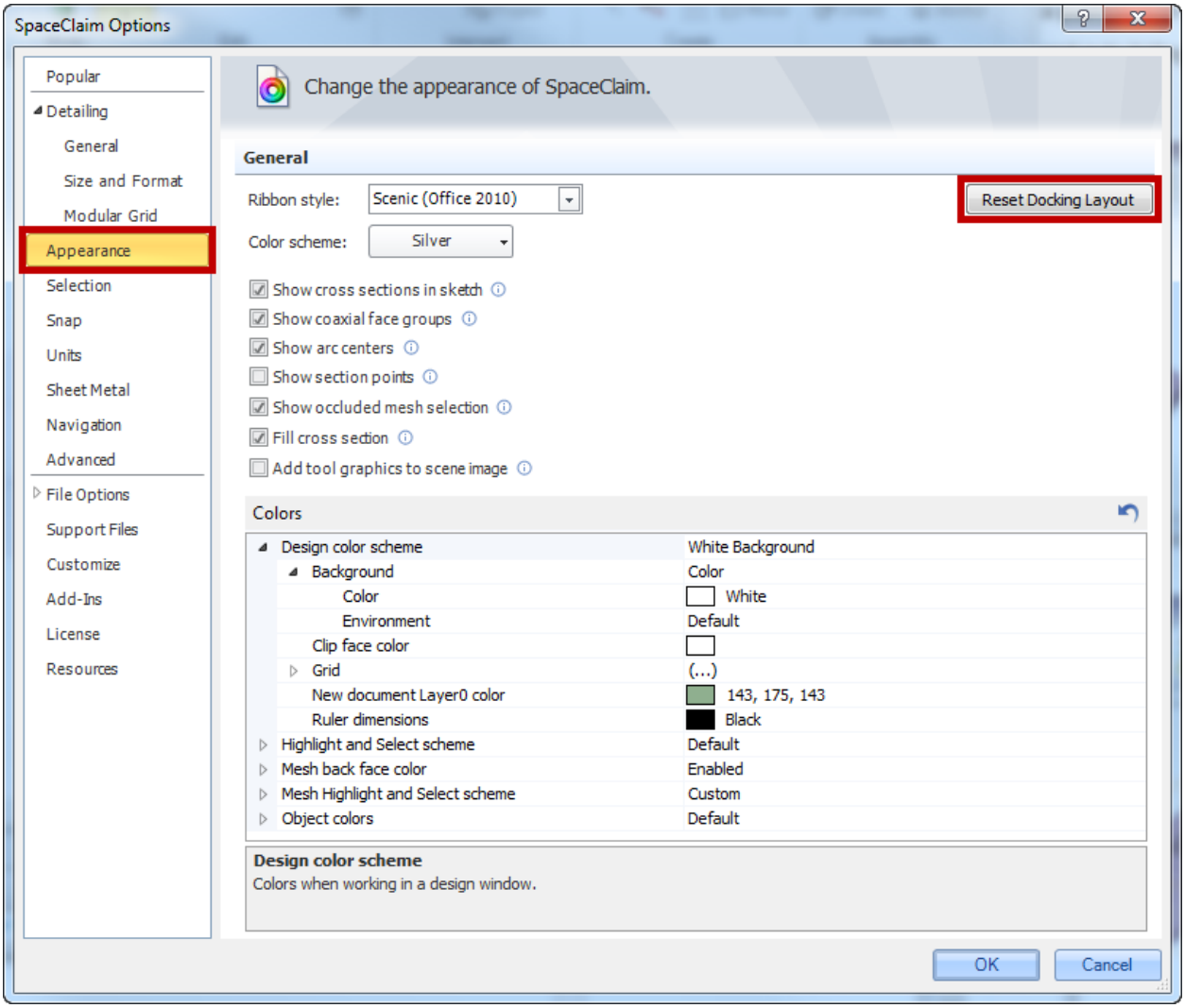
The image on the next page shows a pushpin icon on each panel that if clicked, auto-hides the panel.

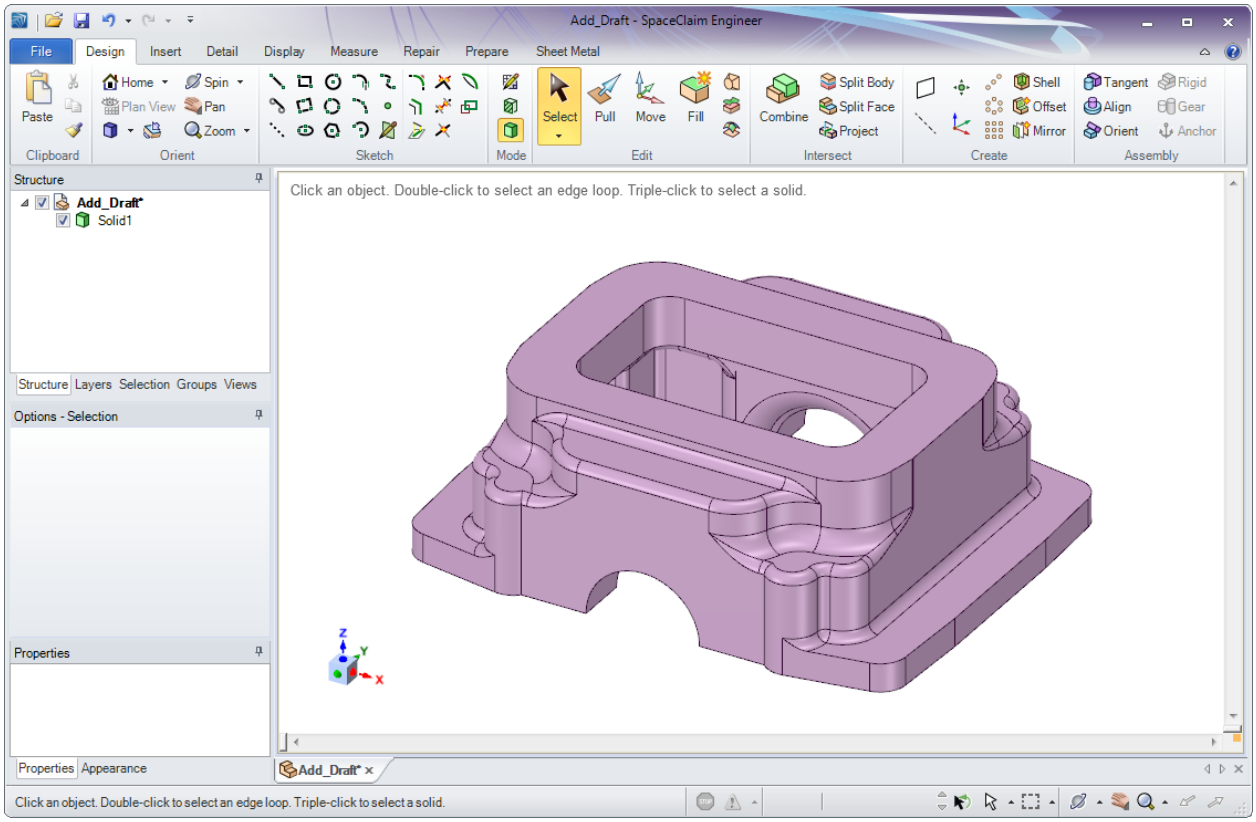
After moving the panels around and auto-hiding them, either intentionally or unintentionally, it can be difficult to drag them back into position. In file\options\Appearance, there is a button “Reset Docking Layout” that resets all the panels to default. The images below show rearranging the panels, auto hiding a panel, then using the Reset Docking Layout to change them back to default





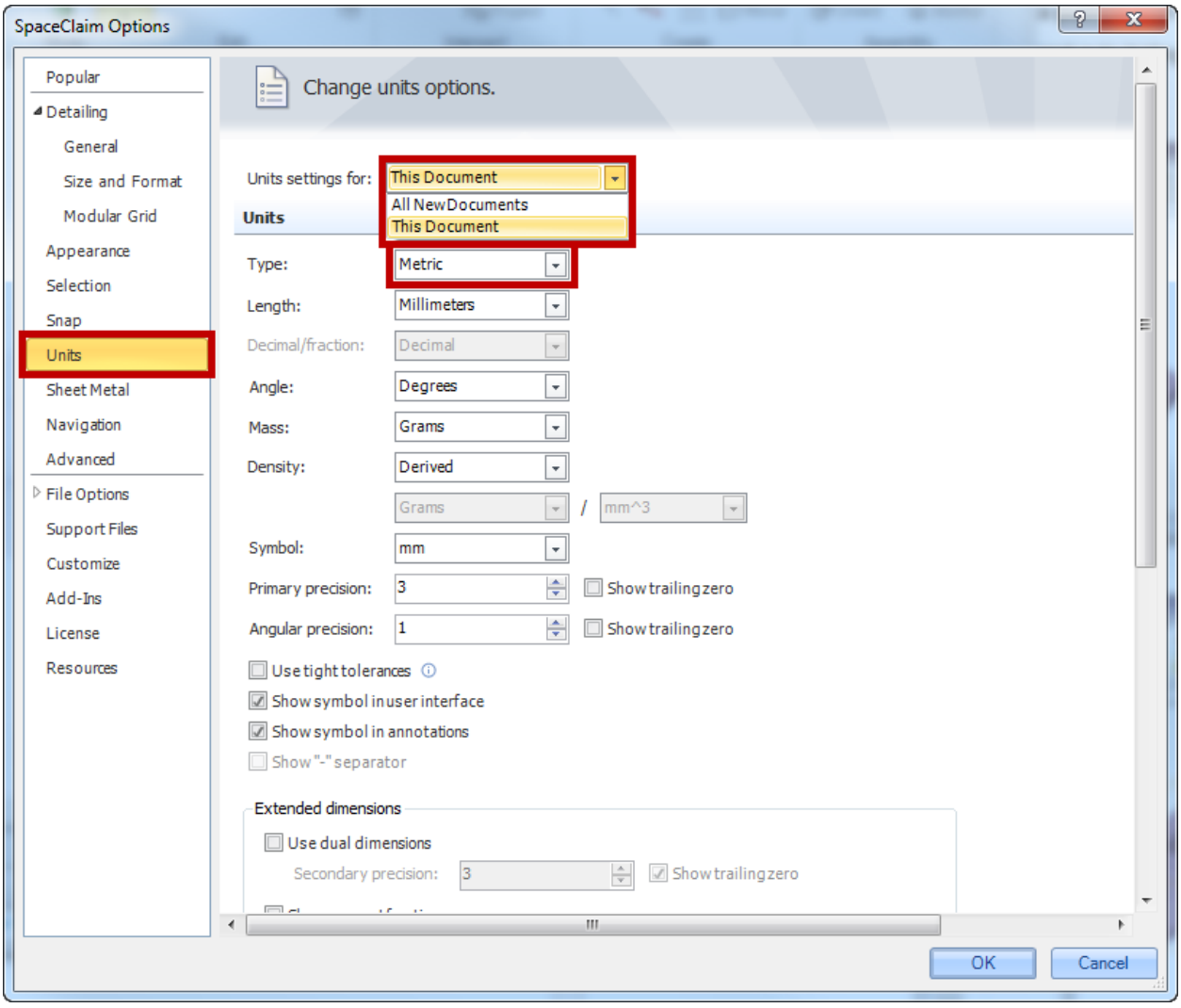



SpaceClaim Options
Popular
Detailing
General
Size and Format
Modular Grid
Appearance
Selection
Snap
Units
Sheet Metal
Navigation
Advanced
File Options
Support Files
Customize
Add-Ins
License
Resources
Change the appearance of SpaceClaim.
General
Ribbon style:
Scenic (Office 2010)
Reset Docking Layout
Color scheme:
Silver
Show cross sections in sketch
Show coaxial face groups
Show arc centers
Show section points
Show occluded mesh selection
Fill cross section
Add tool graphics to scene image
Colors
Design color scheme
White Background
Background
Color
White
Environment
Default
Clip face color
Grid
(...)
New document Layer0 color
143, 175, 143
Ruler dimensions
Black
Highlight and Select scheme
Default
Mesh back face color
Enabled
Mesh Highlight and Select scheme
Custom
Object colors
Default
Design color scheme
Colors when working in a design window.
OK
Cancel

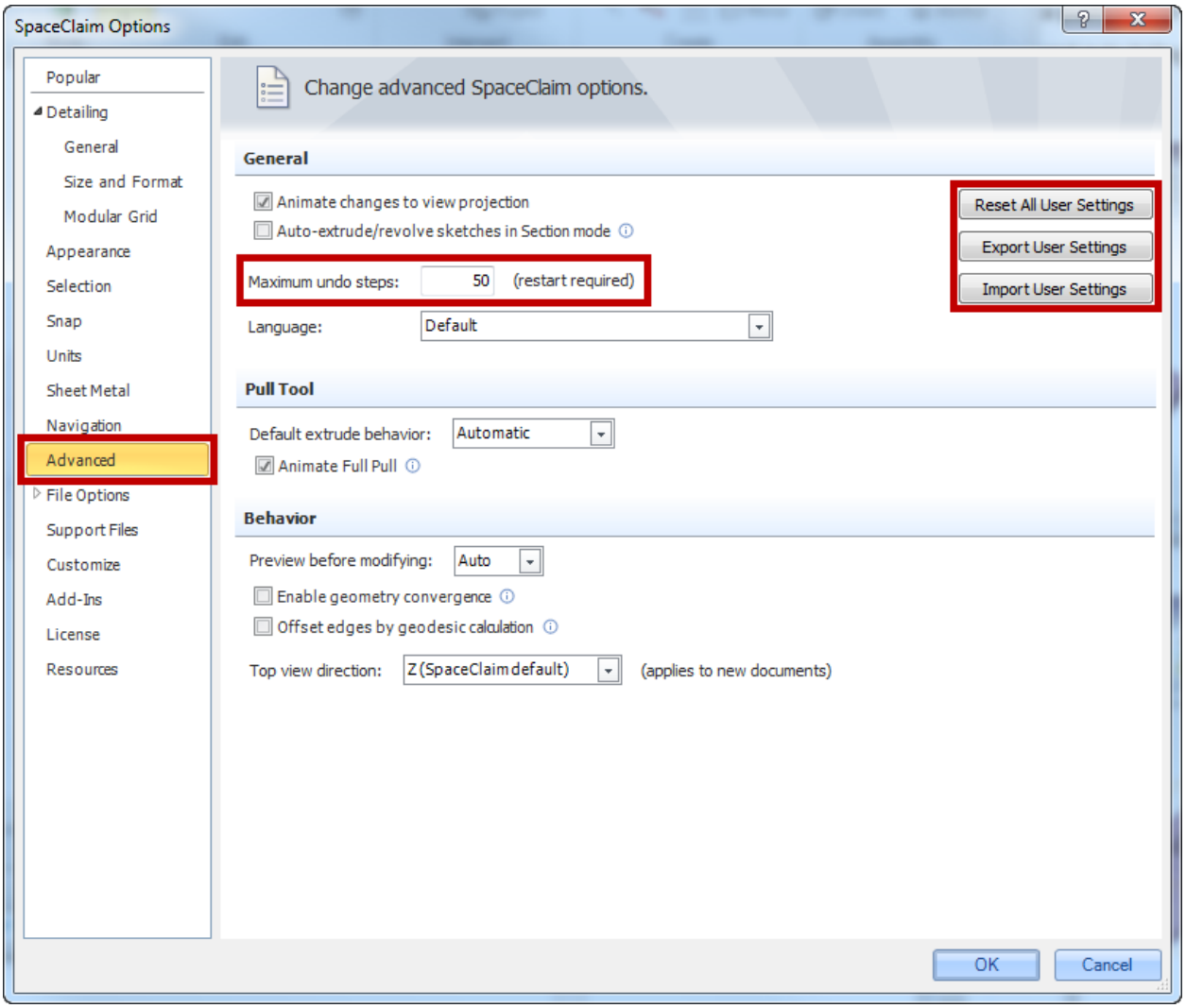

Add_Draft - SpaceClaim Engineer
File
Design
Insert
Detail
Display
Measure
Repair
Prepare
Sheet Metal
Paste
Clipboard
Home
Plan View
Zoom
Spin
Pan
Orient
Sketch
Mode
Select
Pull
Move
Fill
Edit
Combine
Split Body
Split Face
Project
Intersect
Shell
Offset
Mirror
Create
Tangent
Rigid
Align
Gear
Orient
Anchor
Assembly
Structure
Add_Draft*
Solid1
Structure
Layers
Selection
Groups
Views
Options - Selection
Properties
Properties
Appearance
Click an object. Double-click to select an edge loop. Triple-click to select a solid.
Add_Draft*
Click an object. Double-click to select an edge loop. Triple-click to select a solid.

The Units Tab allows you to change the units and other unit related options, either for the current document or all new documents. Notice the first drop down at the top starts off as This Document, and the other option is All New Documents. If you want to change both this Document and All New Documents, you need to change the settings for 1, press OK to close the Options window, open options again and change the other option.

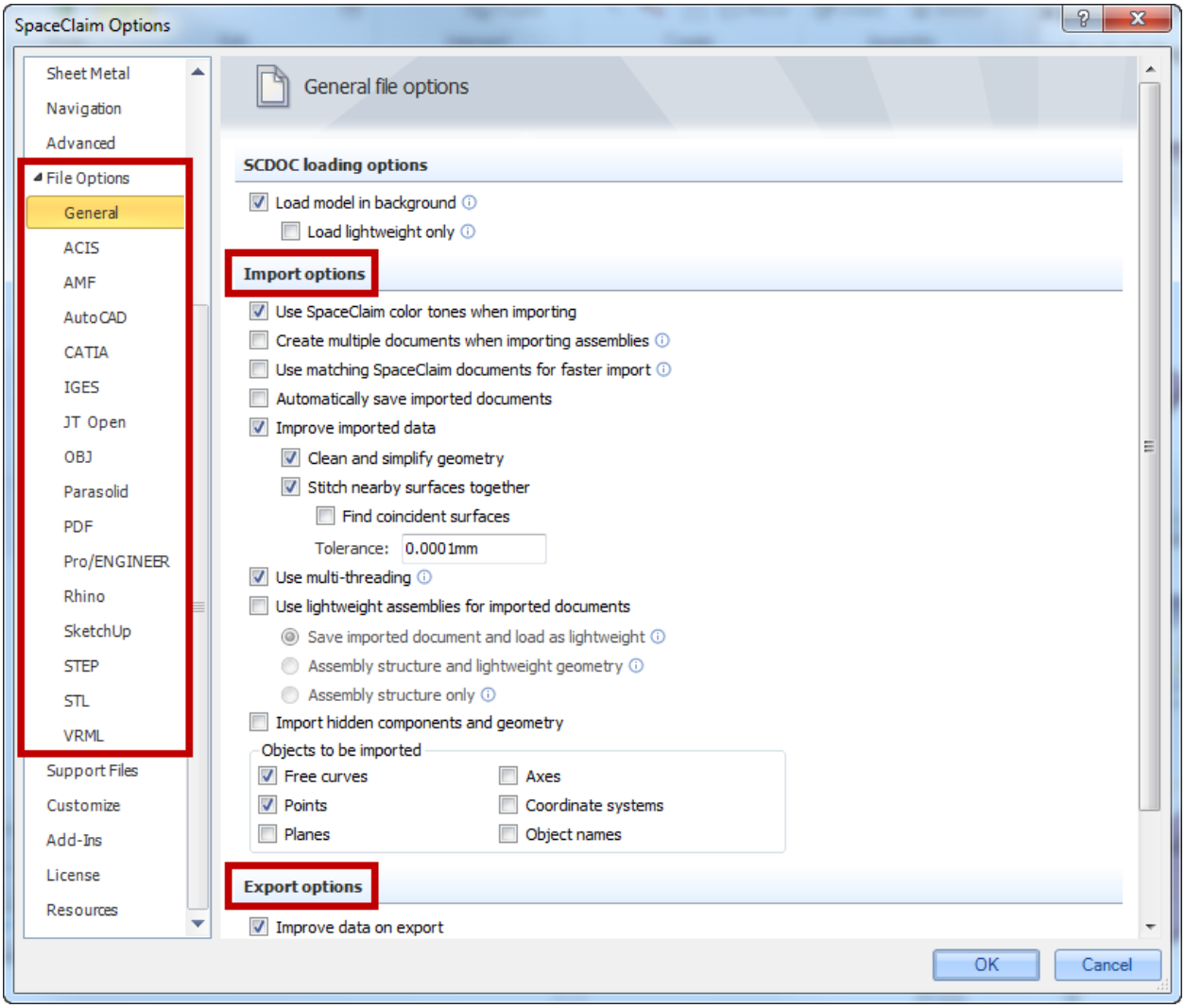
The Units Tab also lets you turn on Dual Dimensioning and change grid spacing.



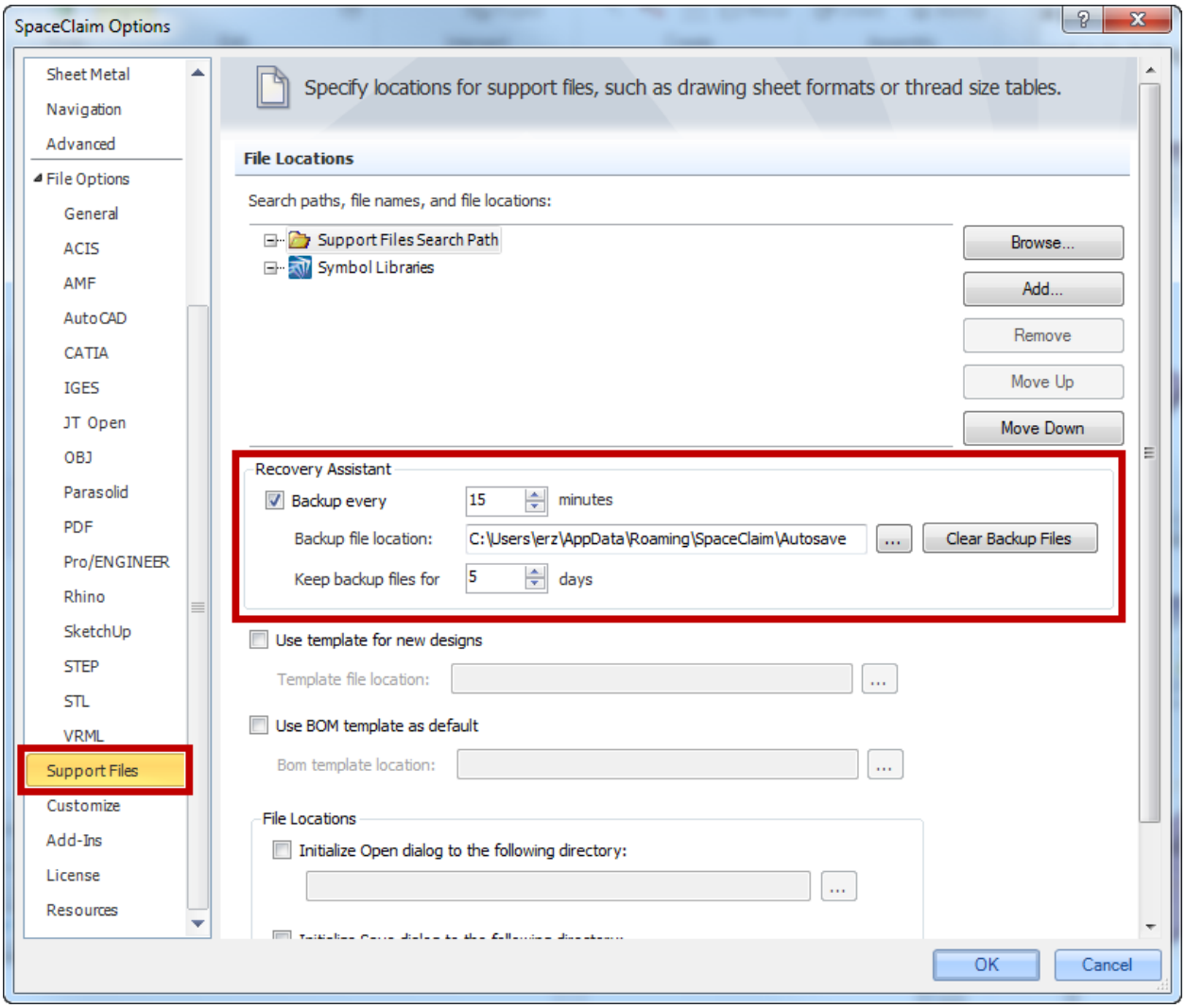
The Advanced Tab allows you to change the Maximum number of Undo Steps. Note this will use more RAM. Also notice the Reset, Import and Export User settings on the right. The Reset Button is the first troubleshooting step when SpaceClaim is not behaving normally. The import and export settings allows you to share your settings with other users, or are useful for transferring settings if you were to get a new computer.



The File Options Tab has a General Settings Tab which applies to all file types, and individual tabs for a variety of the file types SpaceClaim imports. Notice that both the General and Individual tabs have both Import and Export settings. First check the General Tab for a setting you want to change, then check the tab for the file type you are trying to import or export

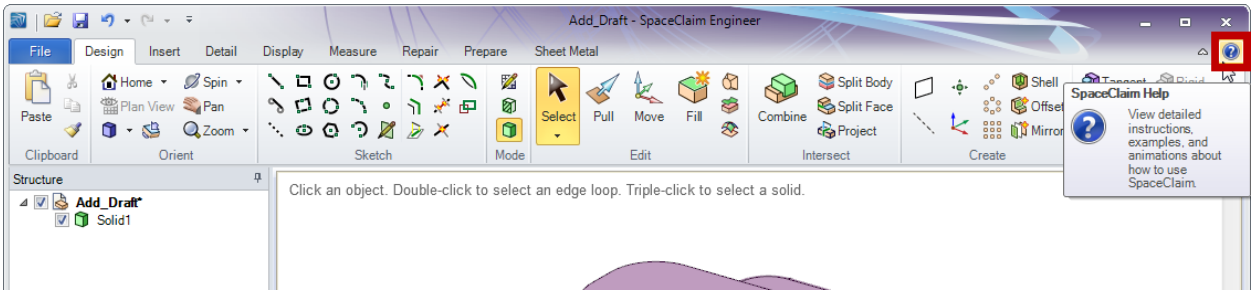


The Support Files Tab allows you to specify search paths or file locations like for drawing templates. Also notice SpaceClaim has a Recovery Assistant which auto-backups your design every 15 minutes by default. In the event of a software crash, when you try to open a file which has been backed up, SpaceClaim will display a message saying there is a newer version, and ask if you'd like to open the new version instead.

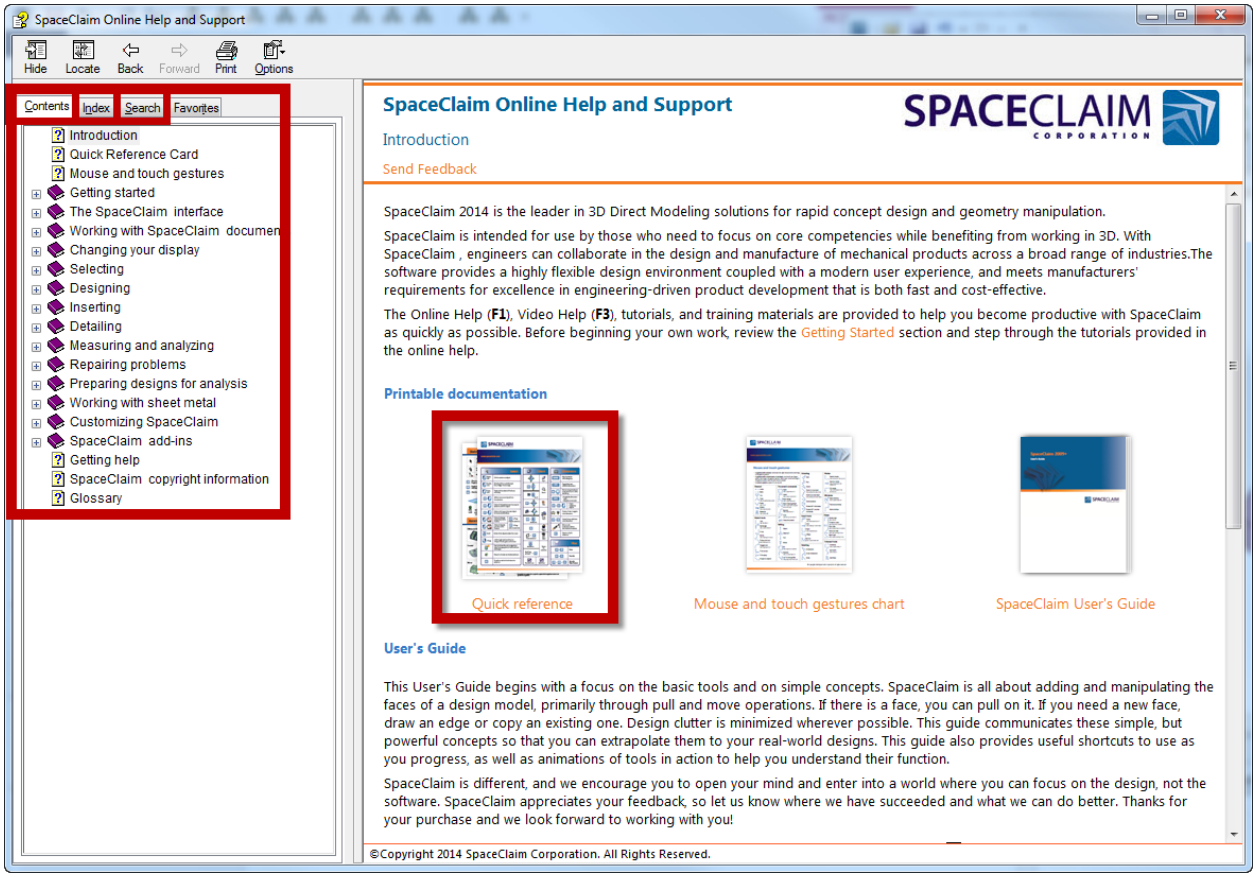


General Help in SpaceClaim

Help is also available directly in SpaceClaim. The first way to access SpaceClaim Help is to click the Question Mark in the far upper right of the Ribbon Bar

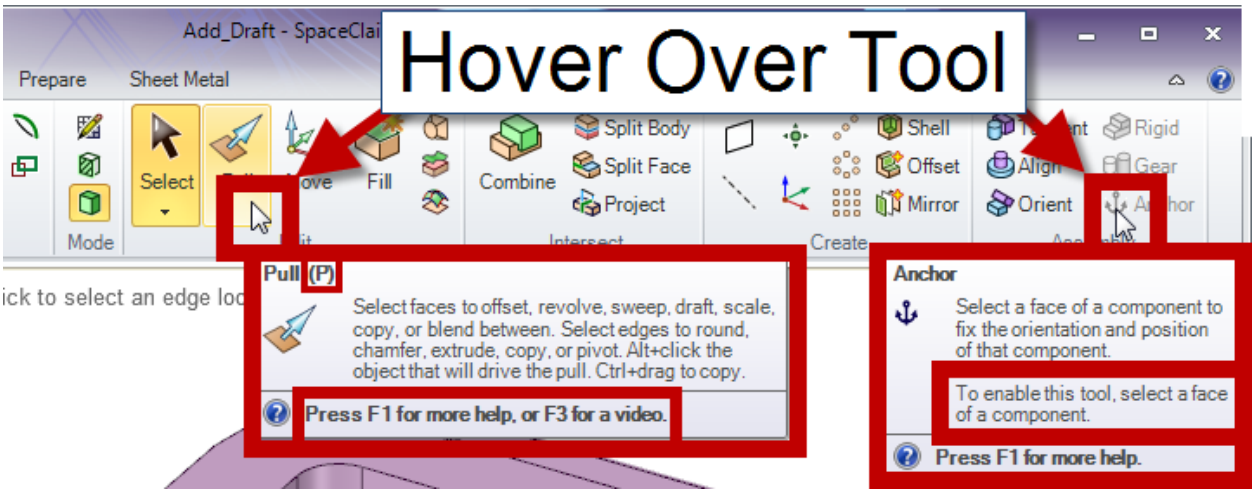


On the very first page of SpaceClaim help is the same Quick Reference Card found in the welcome screen and on page 2 of this manual. Notice on the left that the help is broken up into contents or chapters that help you find help on specific topics like Designing, Detailing or Repair. Also notice there is an Index and Search Tab. Search is great when you want to find all the topics related to something like dimensioning or the Pull tool.



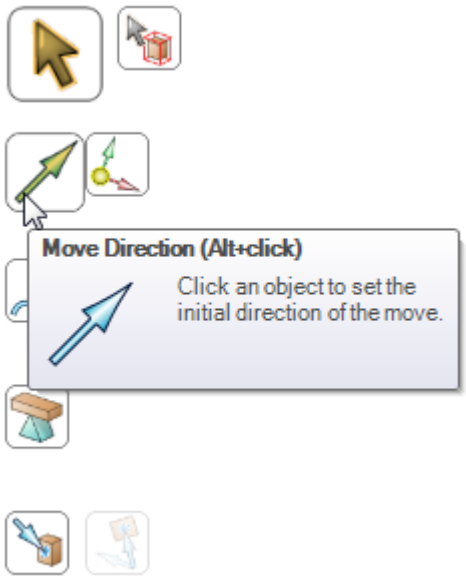
Context Sensitive Help.

At any time, you can easily find more information or help on a specific Tool, ToolGuides, Options or Button by Hovering over it. A ToolTip will show up by hovering over a Tool or Button

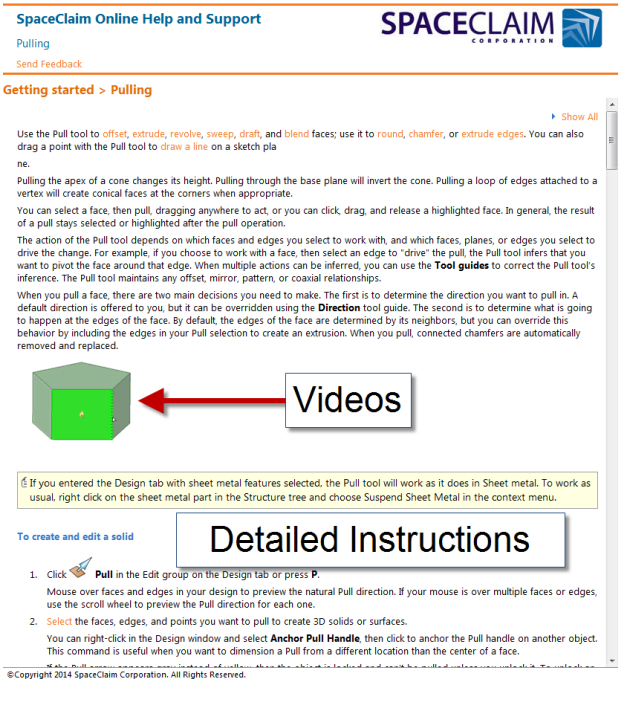


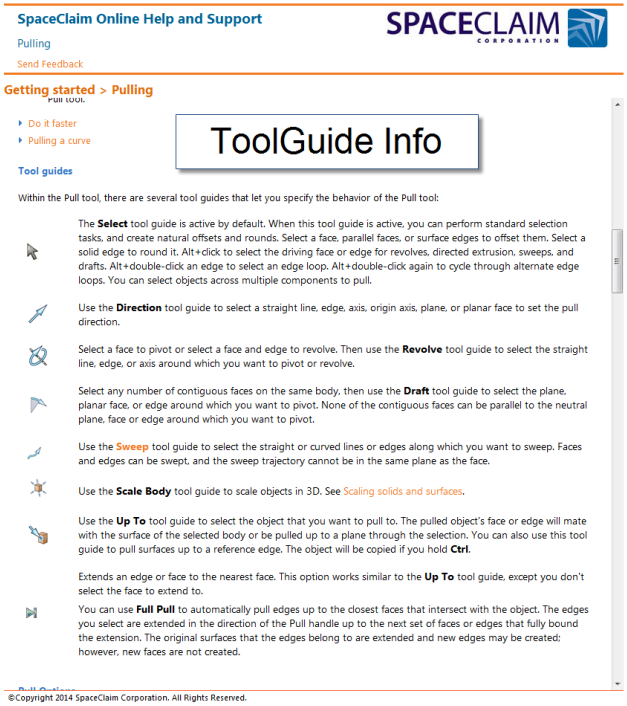
NOTICE that the ToolTip shows the name of the tool, if there is a keyboard shortcut it will be in parenthesis ( ) after the tool name, and a summary of what the tool is and how to use it will be displayed. If the tool is greyed out as the Anchor tool is above, it will tell you how to enable the tool. Also notice that the ToolTip says “Press F1 for more help” at the bottom of the ToolTip. Pressing F1 while the ToolTip is shown will bring you directly into the help page for that tool.

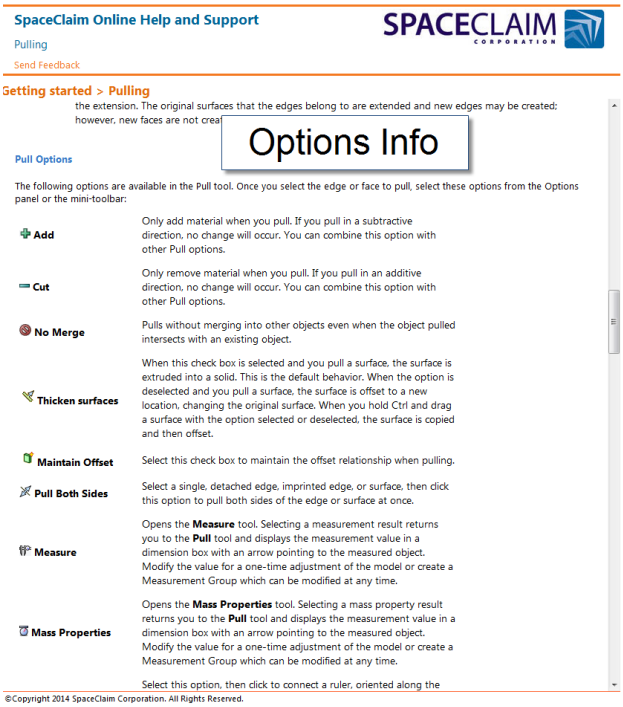
You can also hover over ToolGuides and Options to get a ToolTip, but there is no option to enter help for the ToolGuides or Option directly with F1. You want to hover over the active tool in the Ribbon Bar and press F1. That tool will have

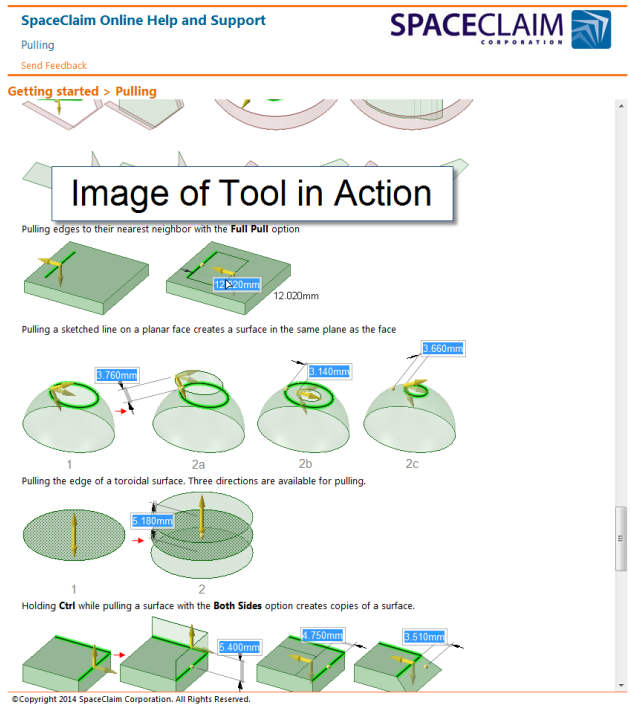


Below is the help page for the Pull tool. Notice a summary of what the Pull tool does at the top. Any orange word is a link to a related page. Many pages have a video showing the tool in action, and all pages have detailed instructions, info on the ToolGuides, options and images of the Tool in action.





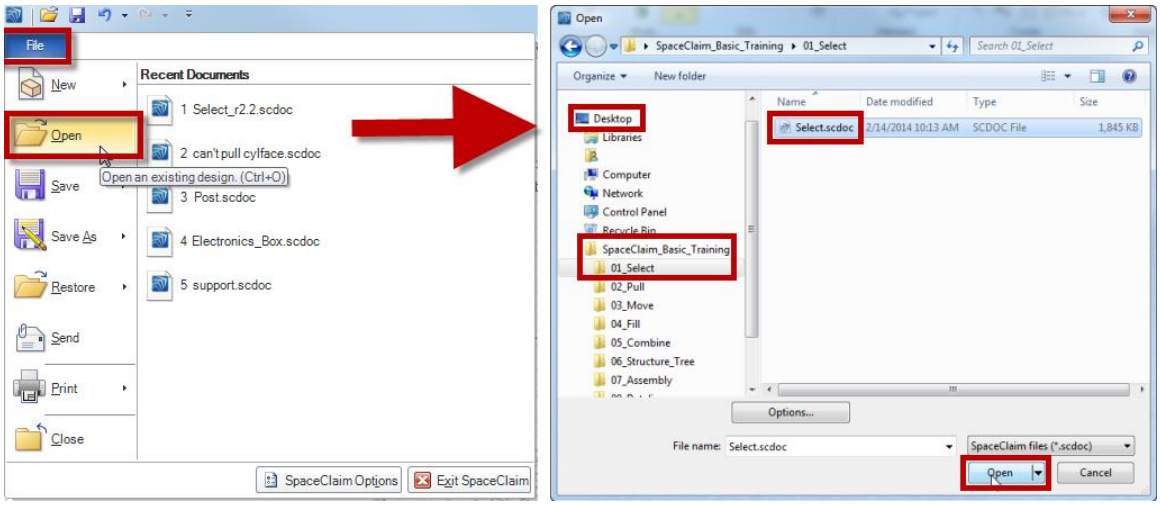




Select

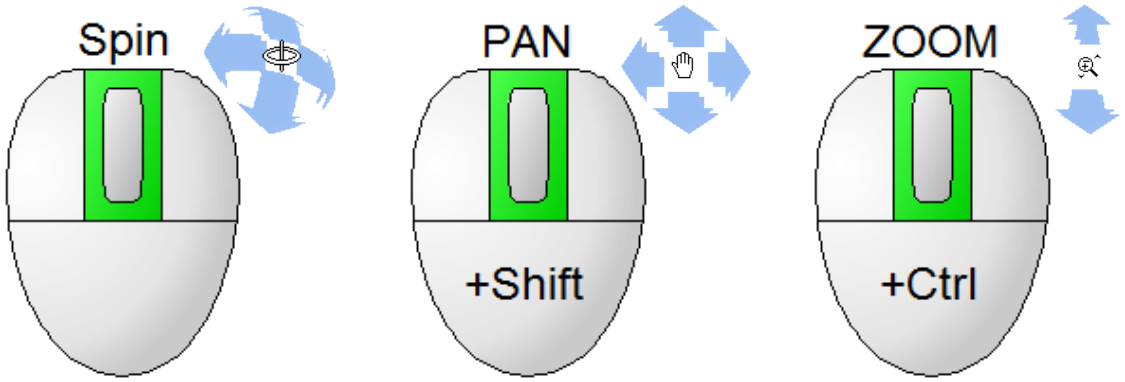
Selecting objects (solids, surfaces, curves, components, assemblies) in SpaceClaim is a lot like working in Microsoft products like Windows® and Office®. Common shortcuts like CTRL, SHIFT, Single Clicking, Double Clicking, Right Clicking, CTRL-C, CTRL-V & CTRL-Z all work the same way in SpaceClaim.

1. Click **File\Open**, and browse to **Desktop\SpaceClaim_Basic_Training\01_Basic_Select_2104.0** and Open **Basic_Select_2014.0.scdoc**



Spin, Pan & Zoom

2. Try **Spinning, Panning and Zooming** with the **middle mouse button (scroll wheel)**

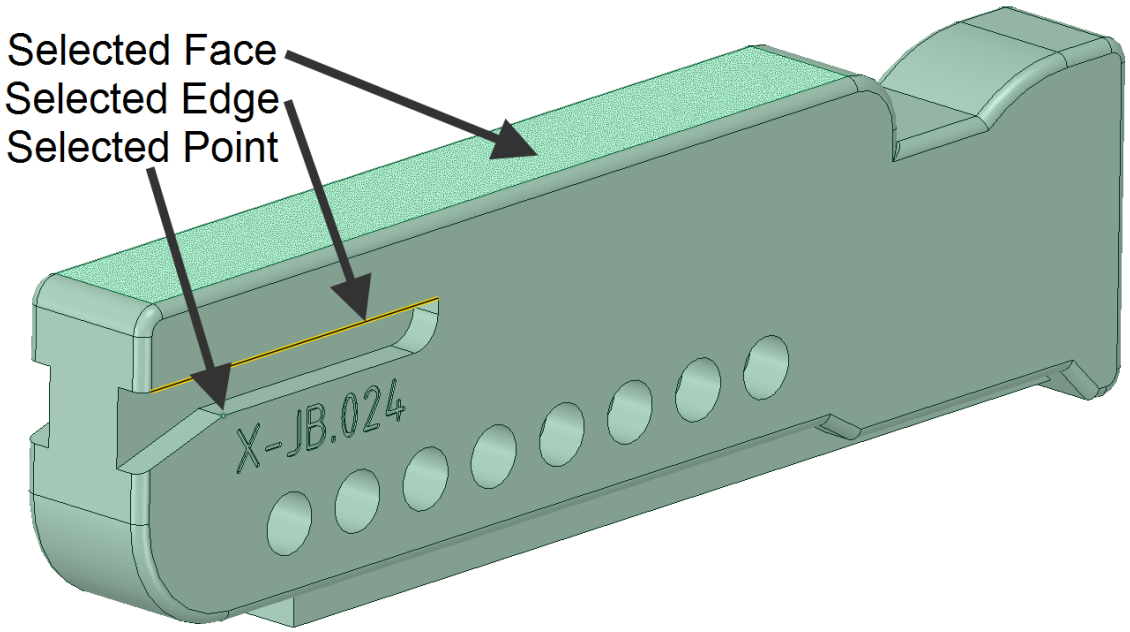


Tips:

1. Default **Spin/Pan/Zoom** can be changed in the SpaceClaim Options -> Navigation.
2. The Model will Spin around where your mouse cursor is when you start to spin
3. The Model will Zoom in on where your mouse cursor is when you start to zoom

Click to select

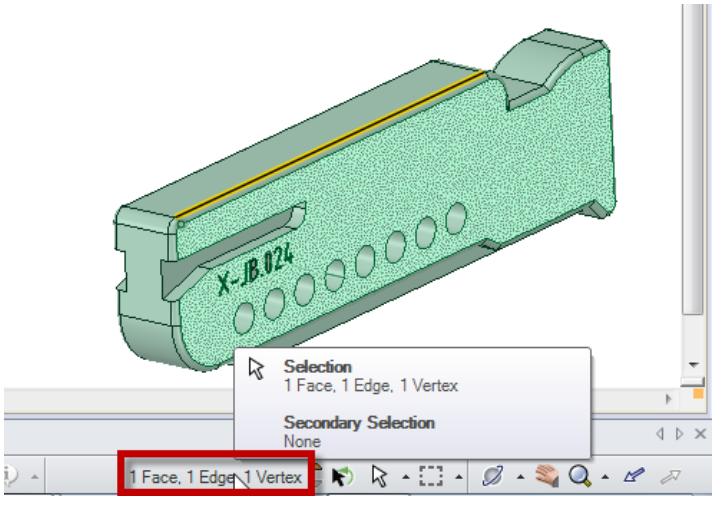
1. **Click a face…**
2. **Click an edge…**
3. **Click a point**…



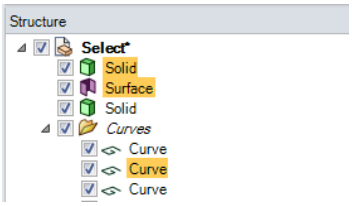
When you click on a piece of geometry, any previously selected geometry becomes deselected.

4. To **ADD** more geometry to your selection, **hold CTRL** while clicking
5. To **REMOVE** geometry from your selection, **hold CTRL** and click something already selected

NOTICE: a panel on the bottom of the screen that tells you what is selected. It's a good place to look if something doesn't work as expected.



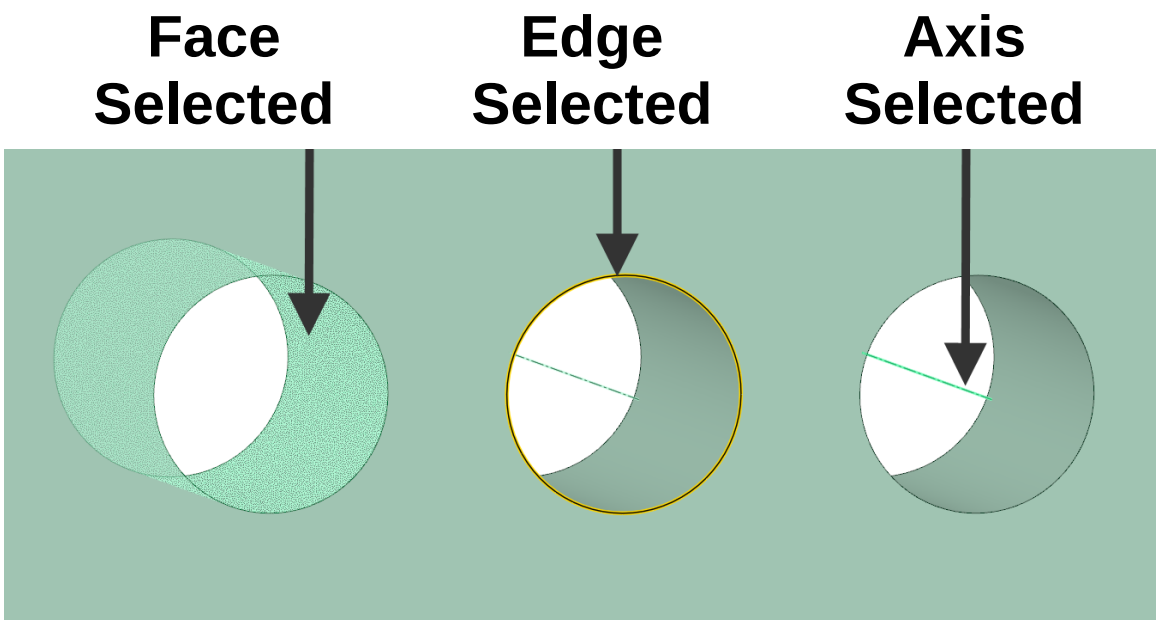
Also look in the Structure tree



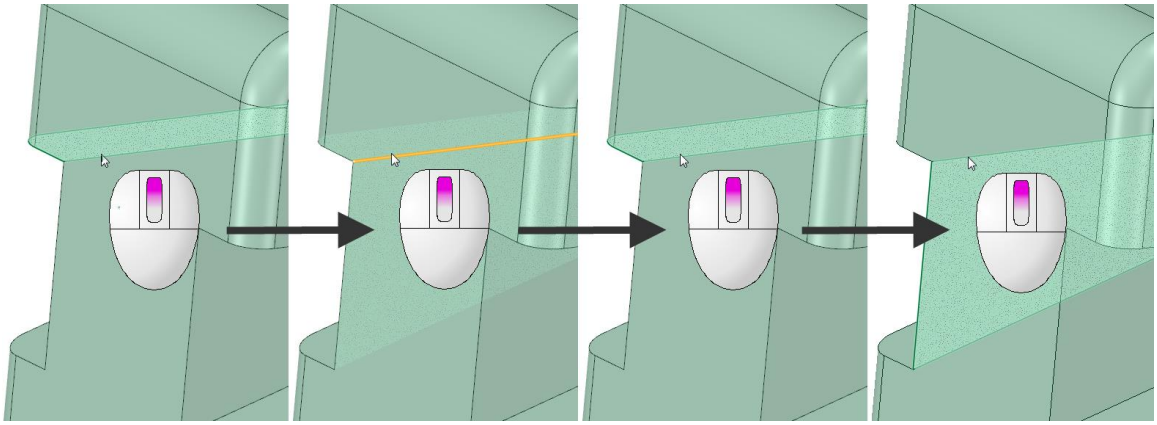
6. To **Completely Clear** your **selection**, either **Click in White Space** or **Press Escape**.

Selection TIPS and things to look out for

7. **ZOOM** in on an area to make it much **easier to** properly **select** things
   a. With **nothing selected**, press "**Z"** to **zoom OUT** to the extents of the model
   b. With **something selected**, press "**Z"** to **zoom IN** on selection
8. **Holes**: Most of the time, when working with holes, you want the face selected (Pull, Move & Fill sections). You may accidentally select the edge of the hole, or the axis of the hole
   To show the hole's axis, hover over the face of the hole



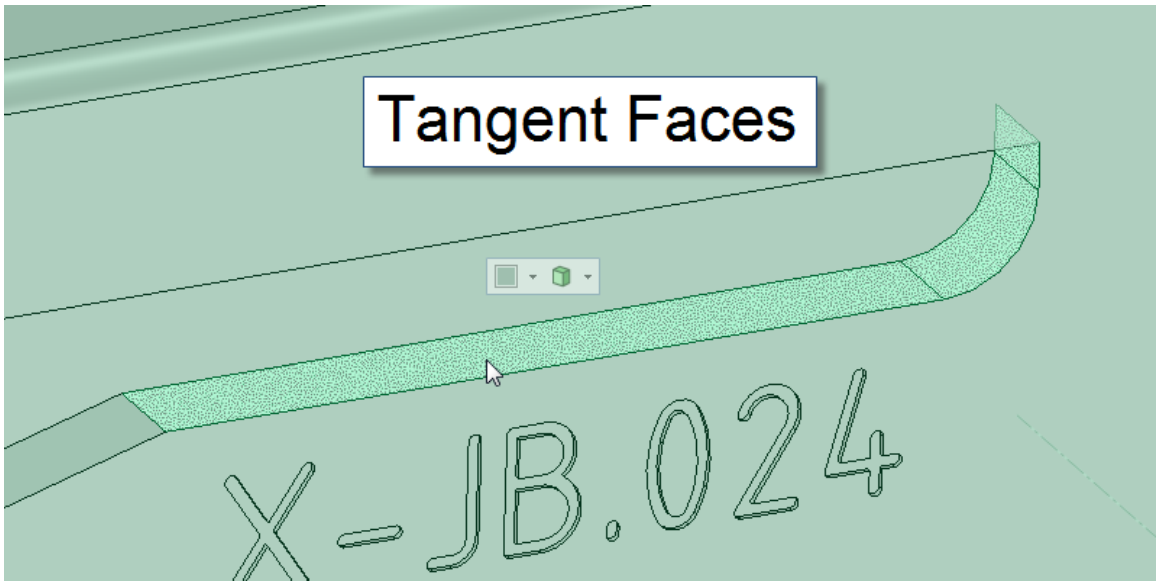
9. **Query Select**: To select something hidden, or hard to select due to a lot of geometry in an area, hover your mouse in the general vicinity, and scroll the mouse wheel up, 1 click at a time. Left click with the geometry highlighted to select it. Scrolling does the highlighting, not selection.



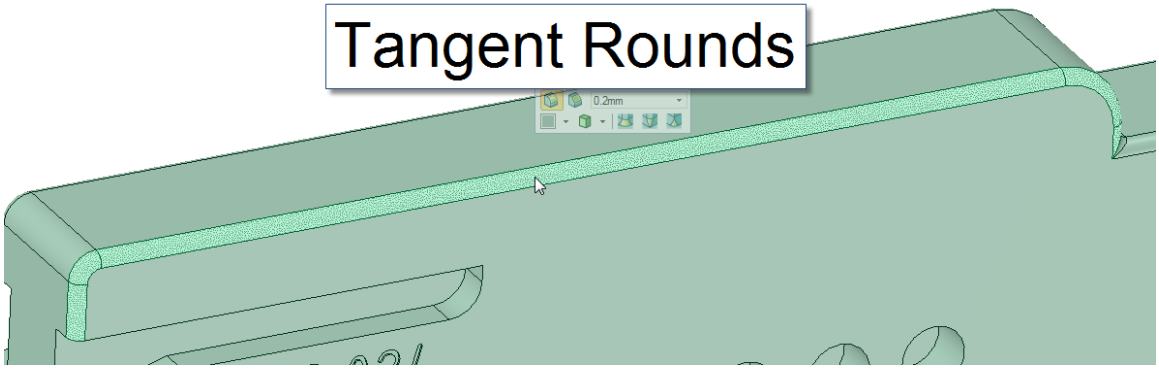
10. You can also **scroll down** if you scroll past what you want to select

11. To select a **Chain** or **Loop**: **Double Click**

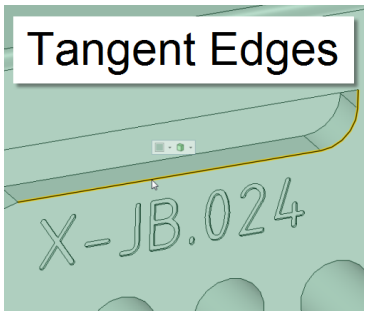
    a. **Face: Double click** to select faces **tangent** to selected face

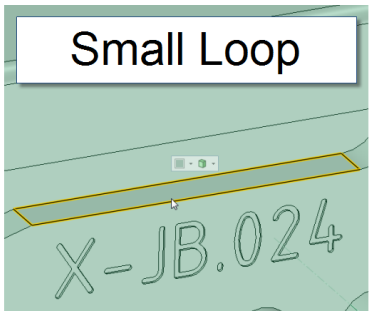


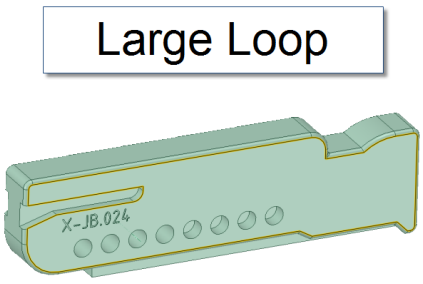
    b. **Rounds: Double click** to select **tangent** rounds



    c. **Edges: Double click** for **tangent chain, double click again** on same edge for **small loop**, **double click again** on same edge for **large loop**



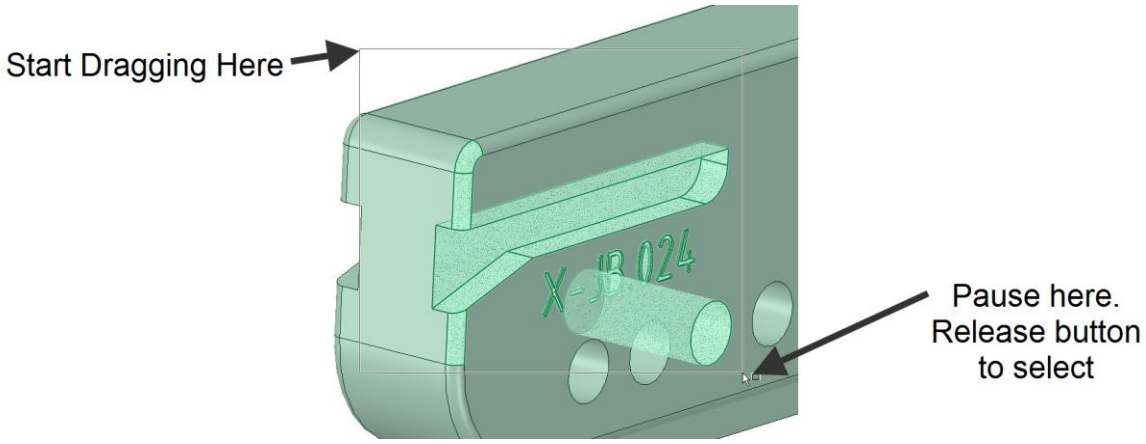




Drag to Select: Box Select

Box Select: Drag (hold down left mouse button and move mouse) **a box around** what you want to select

12. **<u>Left to Right</u> Box Select: Geometry COMPLETELY inside the box**



NOTICE: Only **geometry completely inside** the **box** is **highlighted/selected**

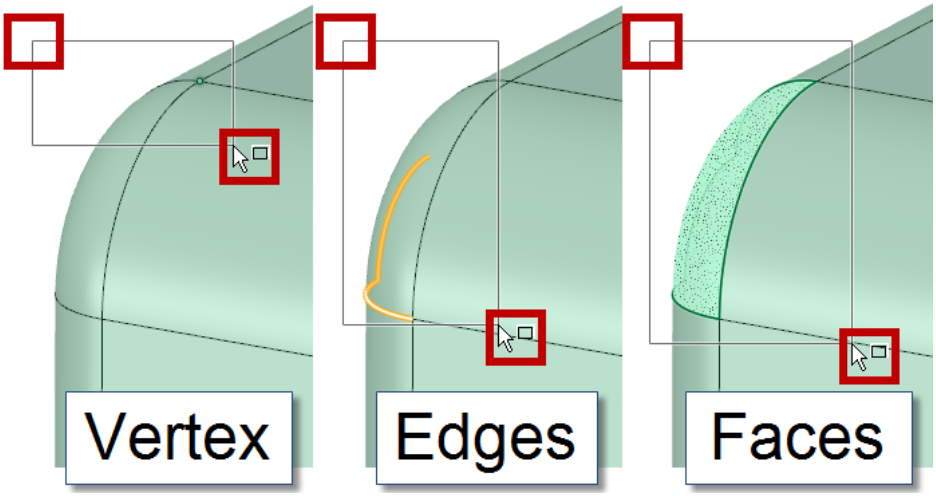
Left to Right Box Select Tips

a. If you **pause** while **dragging**, you get a **preview** of what will be selected.

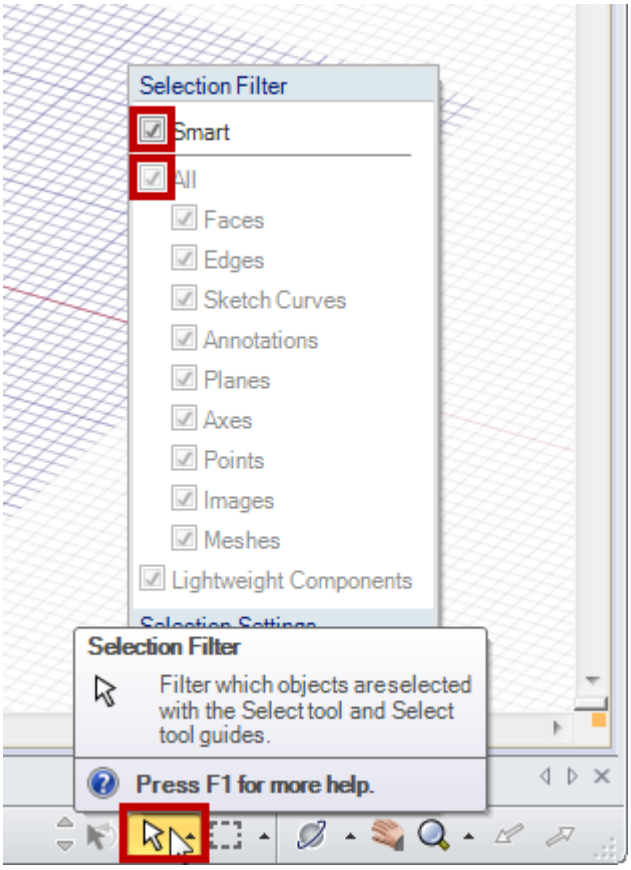
b. **Box select** can be **performed in any tool**, but the **safest place is the Select tool**

c. By default, the L-R box select uses a “Smart” selection filter.

- If the largest piece of geometry in a box is a Vertex, the point will be selected
- Once an Edge is inside the box, vertices are filtered out and only edges are selected
- Once a Face is inside the box, vertices and faces are filtered out and only faces are selected



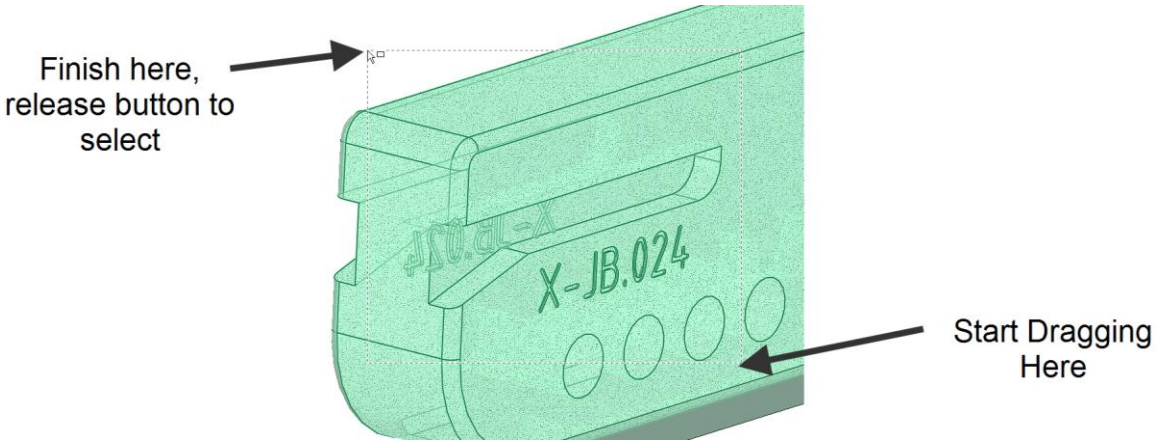
Selecting Filter



Selection Filter

13. To manually filter to any type of geometry, whether using box select or any section method, click the white arrow in the bottom right to show the Selection Filter. You can uncheck Smart, uncheck All and select any geometry or object to filter to.

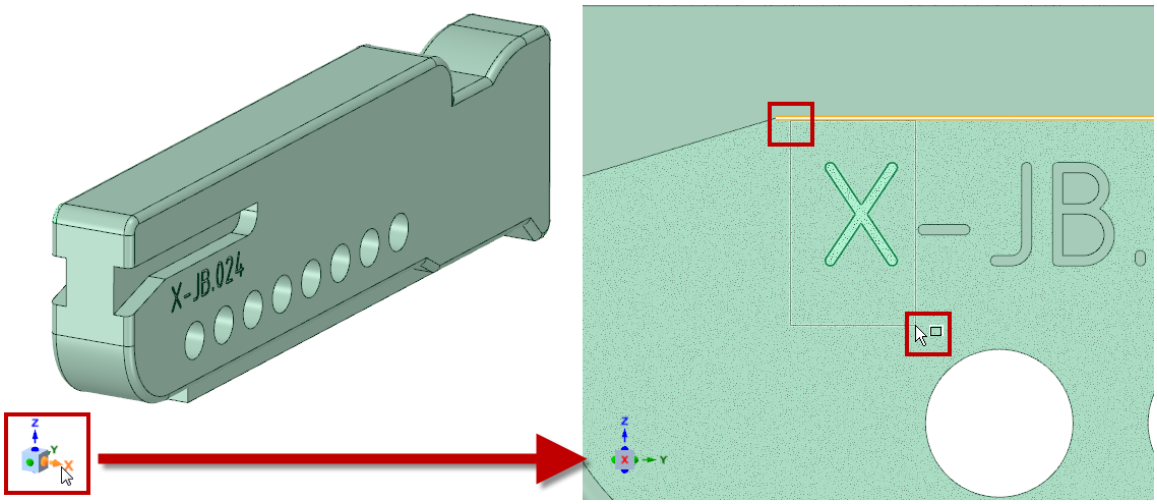
14. **<u>Right to Left</u> Box Select: Any Geometry the box touches**



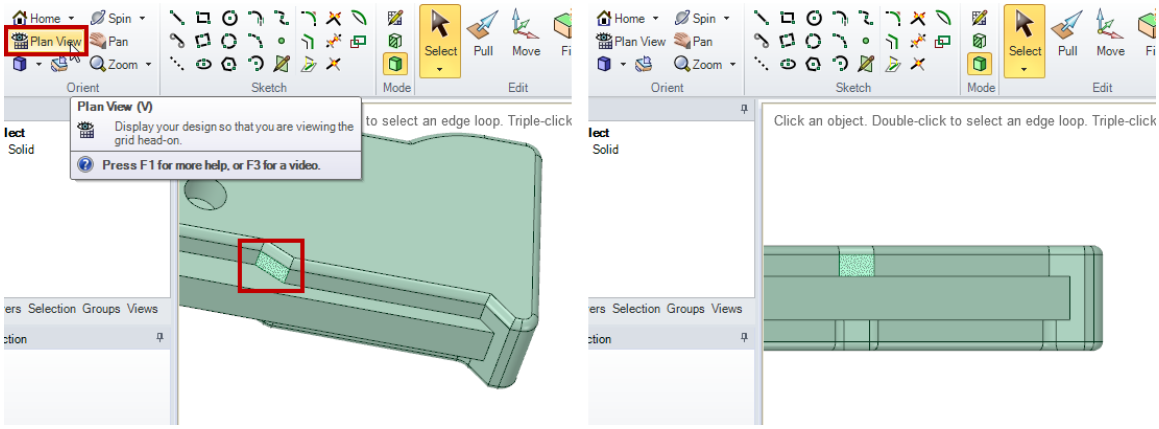
NOTICE: the **differences** between **right to left** and **left to right** box select

When using box select, it is important to be in a good view to make the box select easier. One method to get into a better view to box select is the orientation widget in the bottom left corner of the design window.

15. Click on any of the arrows to orient the view in that direction

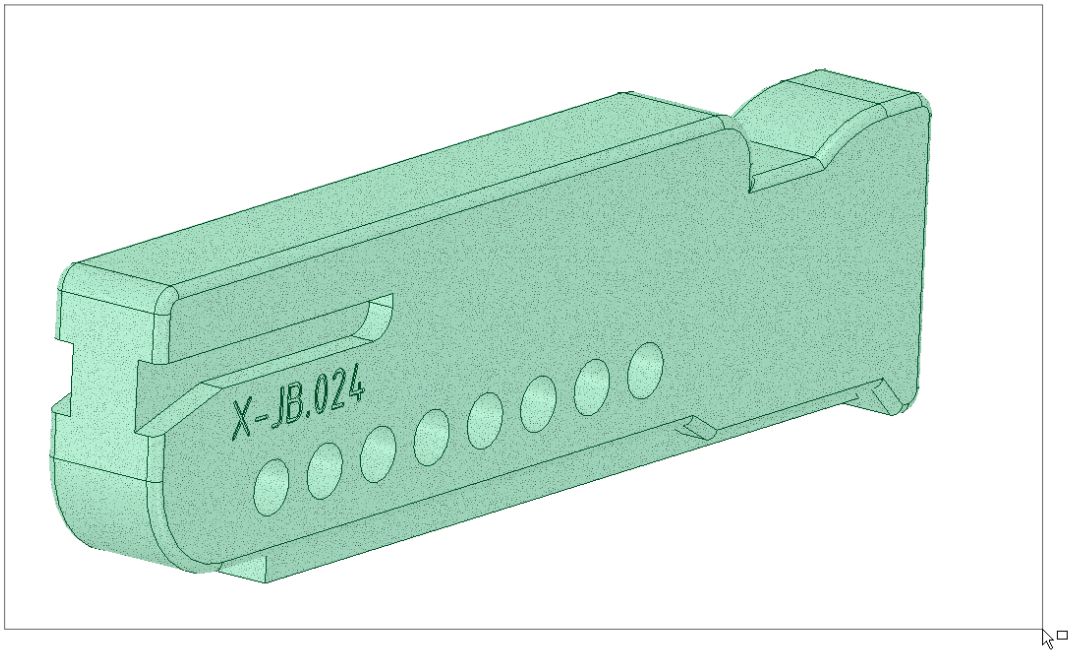


16. If the model is not oriented to the X-Y-Z axis, you can **select the face you want oriented** to the screen, and use the **Plan View** button in the Orient Group of the Design Tab to orient the model so the selected face is parallel to the screen



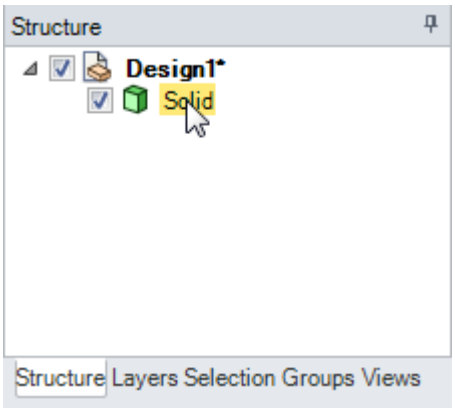
To select an entire object: Solids, Surfaces or curves

17. Drag a **box** around **entire object**: Left to right or Right to left



18. **Triple click** on a face (for Solids or surfaces) or triple click on a curve

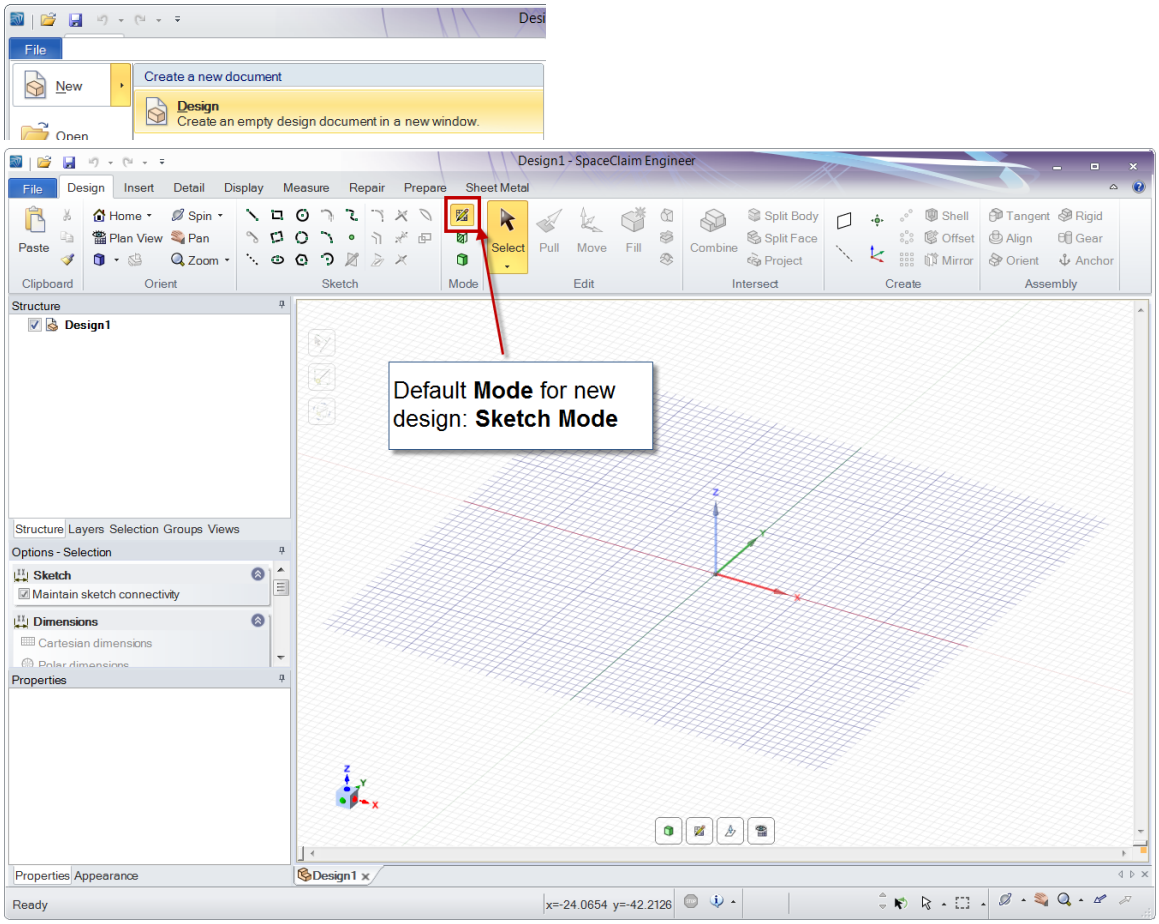
19. **Select the object in the structure tree**



Sketching

Sketching in SpaceClaim is similar to a freeform 2D sketching tool, as opposed to the sketch solvers in typical 3D CAD. Sketch curves (lines, arcs, splines etc.) are entities in space that can be selected, moved around, manipulated etc. Sketching is typically done in 2D. After completing the Pull and Move sections, you will notice that editing a 3D solid in SpaceClaim is a lot like editing a 2D sketch.

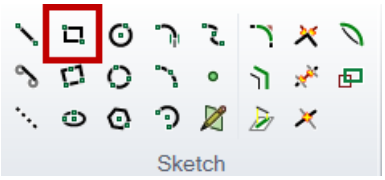
1. When you launch SpaceClaim, a new design will automatically load, with a sketch grid through the X-Y plane and in sketch mode.
2. If you want to start a new design, navigate to File\New\Design.



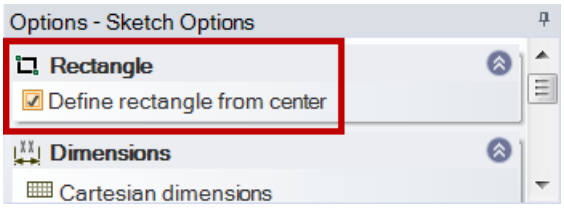
3. The **Sketch Tools Group** is found in the **Design tab**. They are split up into **2 sub-groups**: **Creation & Modification** as seen in the image below

Center Rectangle

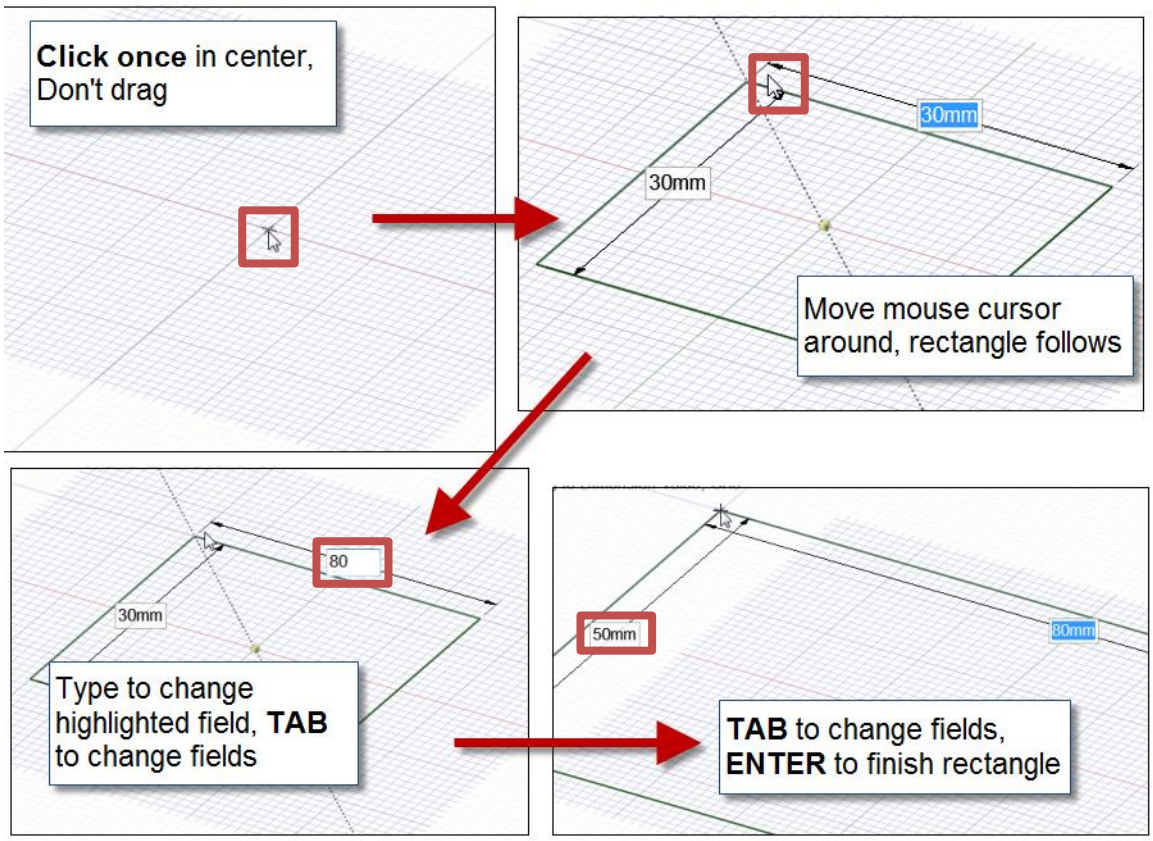
4. **Turn on the Rectangle Tool** in the Sketch group by clicking it.



5. In the Options panel on the left, **check the box for "Define Rectangle from Center"**.

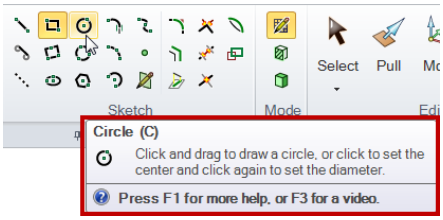


6. **Click and release once in the center**, don't drag.
7. **Move the cursor away from center**, but don't click a second time. A second click will complete rectangle.
8. **Type in 80** to change width, **press TAB**, **then 50** to change height, **press Enter** to complete.

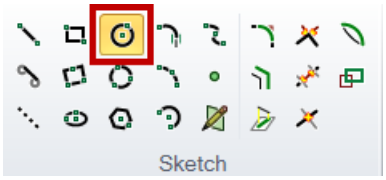


Circle

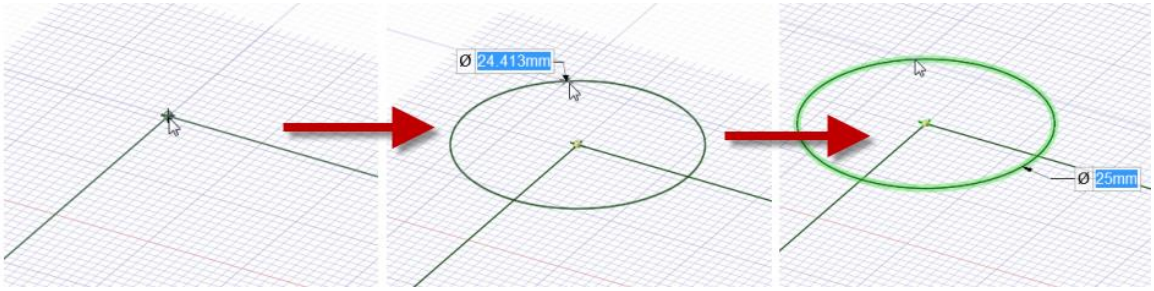
9. **Hover the cursor over the Circle tool** and a box called a **Tooltip** will appear. It provides information on the tool, like the name and keyboard shortcut. **Pressing F1** with the tip shown will open the SpaceClaim Help with Circle tool's help page loaded.



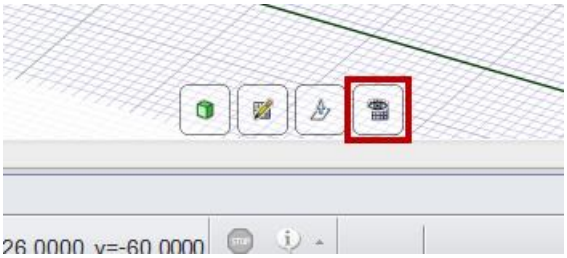
10. **Click the Circle tool**



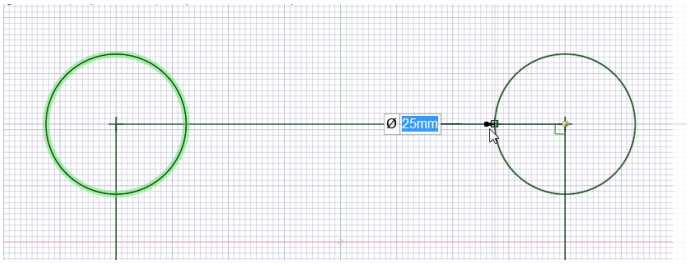
11. **Click the upper left corner of the rectangle**, which snaps the circle's center to that location.
12. **Move the cursor away from center**, the mouse will snap to the grid, **type 25 and press enter.**



13. Switch to a **Normal, Plan-View** with the fourth button in the Grid-Guides on the bottom of the screen.

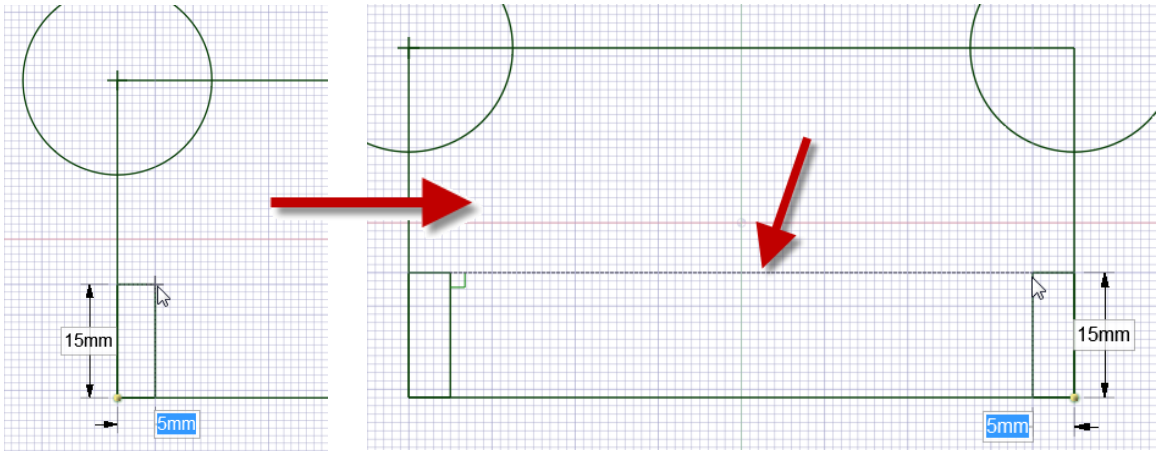


14. **Create** a 2nd **Circle**, **same size** as the first
    a. As you create the circle, it will highlight green and snap to the first circle you sketched. Click to complete the circle.



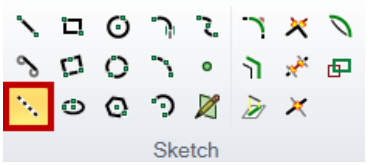
Corner Rectangle

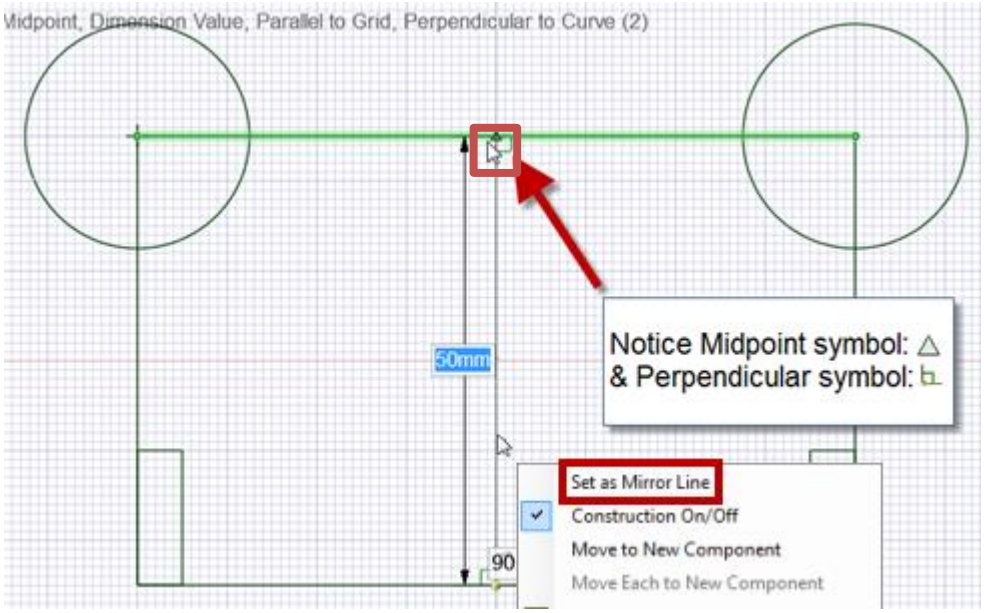
15. Turn on the rectangle tool, this will reset the options of the rectangle tool.
16. **Click in the bottom left corner of the first rectangle and move the cursor up and right**. The default rectangle is a corner to corner rectangle. **Type 5mm, TAB and 15mm.**
17. **Create an identical rectangle in the opposite corner**, **NOTICE** a dotted horizontal line, snapping to the height of the first line. This snap is temporary.



Center-Line

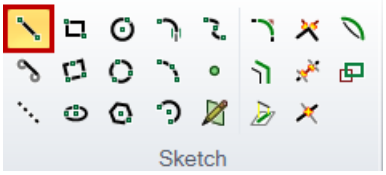
18. **Turn on the Construction Line Tool**.
19. **Click midpoint △ of the bottom line** to start the line.
20. **Click the midpoint on the line at the top** to complete the line
21. **Right-click on the new Construction line** and choose "**set as mirror-line"**





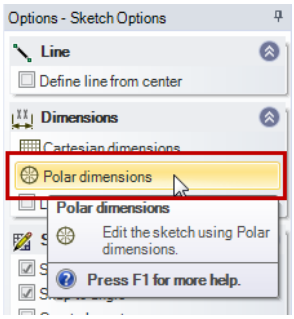
Line and Distance from a Point

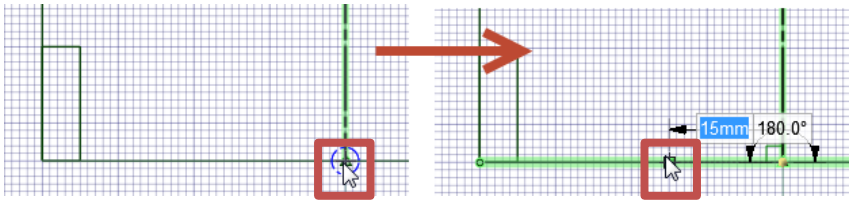
22. **Click the Line Tool**



23. In the options panel **click Polar dimensions**. These two options allow you to base a start point by Polar, or Cartesian dimensions.

24. **Click the intersection of the centerline and the bottom horizontal line** to set the base point.

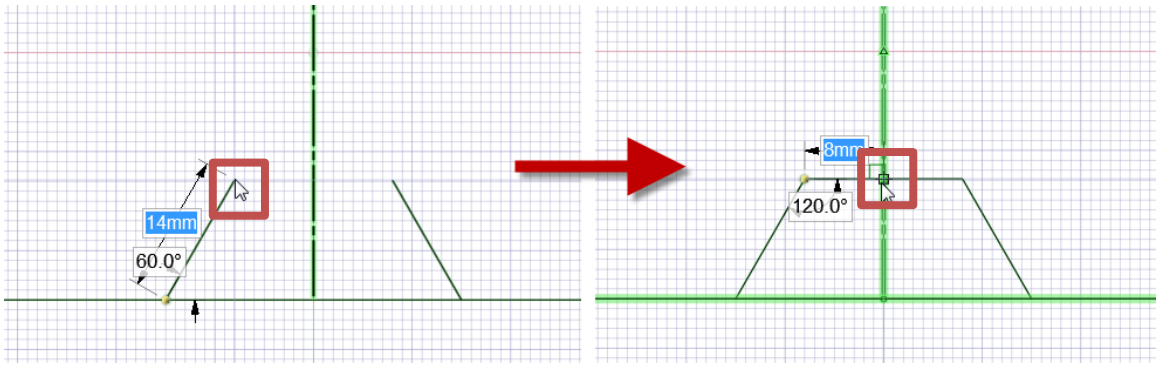




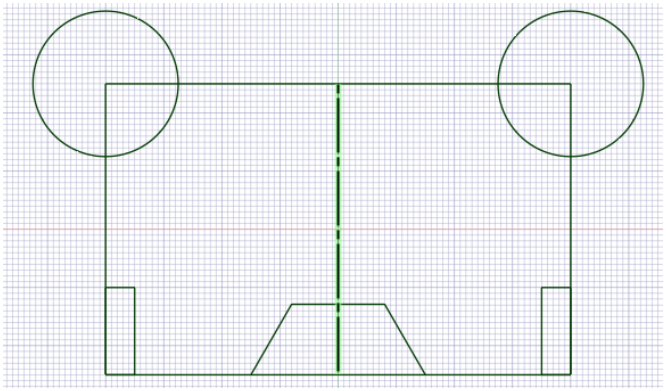
25. At 15mm to the left, **click to start the line**.

26. **At an angle of 60deg toward the centerline, make a line 14mm long**.

27. **Finish with a horizontal line that snaps to the center line**.

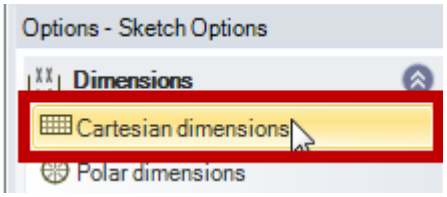


28. The sketch will now look like this.

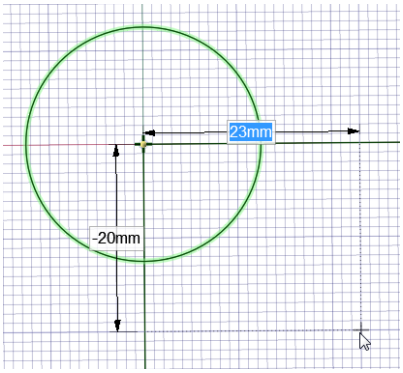


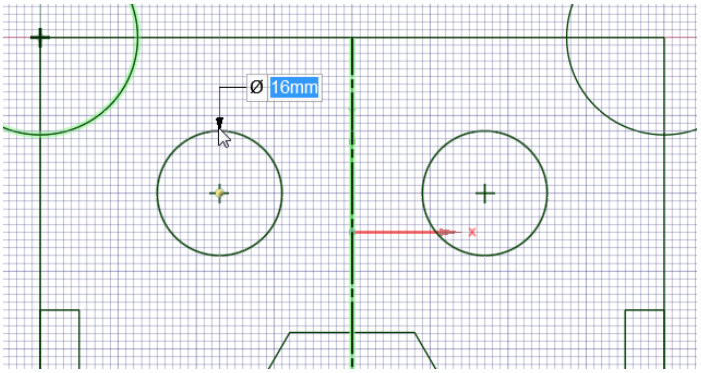
Create Circle at X-Y coordinates

29. **Turn on the Circle Tool**



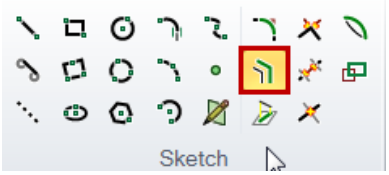
30. Click the Cartesian Dimensions button in options
31. **Hover over the center of the Left Circle and tap the Shift key**. This is a shortcut to shift the base point
32. **Move your cursor down and to the right** to see the X-Y dims
33. **Type 23mm for the X value, press Tab and -20mm** (negative) for the Y value and **press enter.**
34. **Or move mouse to that location and Click to start the circle**
35. **Move mouse to 16mm or type in a diameter of 16mm**



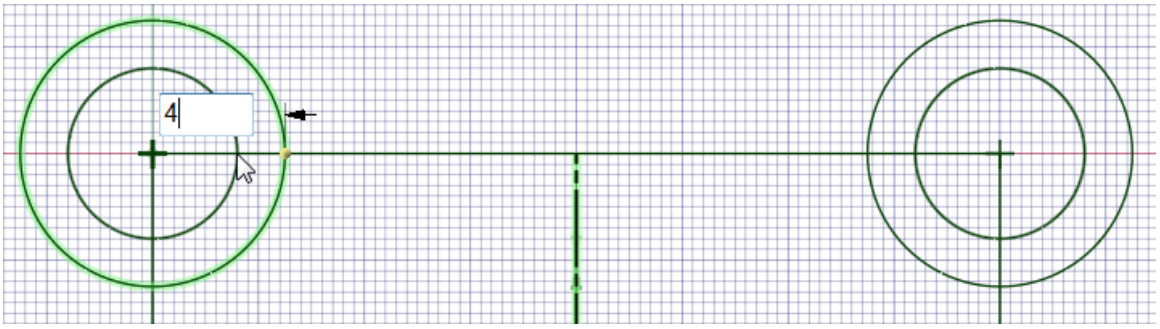


Offset Existing Curves

36. **Click the Offset Tool**.
37. **Select the first sketched circle**. Because the centerline is still set to mirror, the circle on the right will also offset.

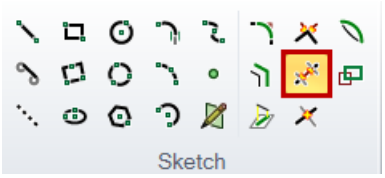


38. As you **move your cursor inward**, you will see the circles offset. **Clicking again** will finish the offset to the shown value. **Or typing 4 and pressing enter** will offset the circle 4mm.

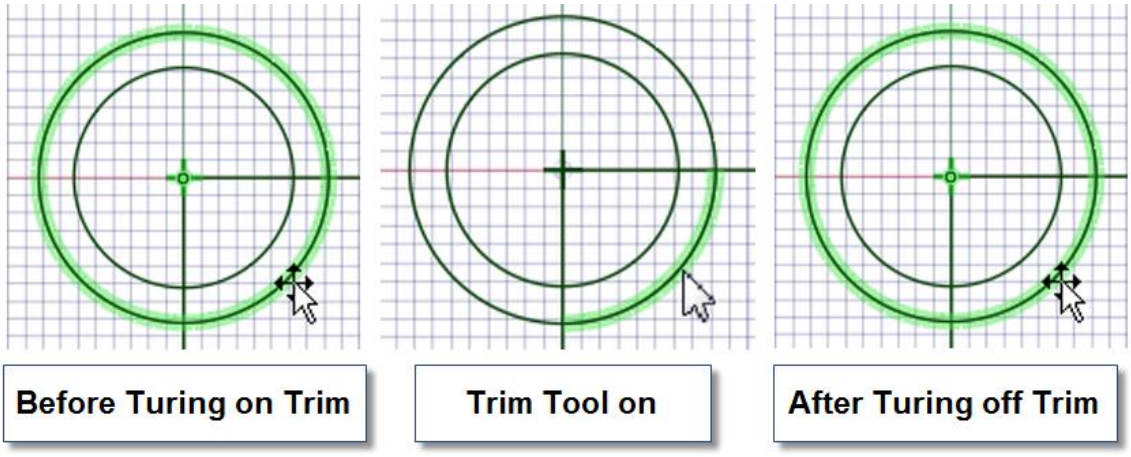


Trim Extra Curves

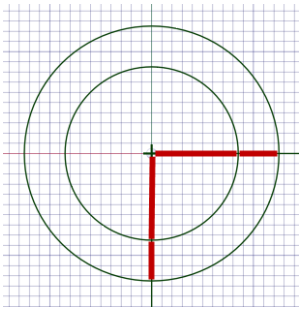
39. **Click the Trim tool**.

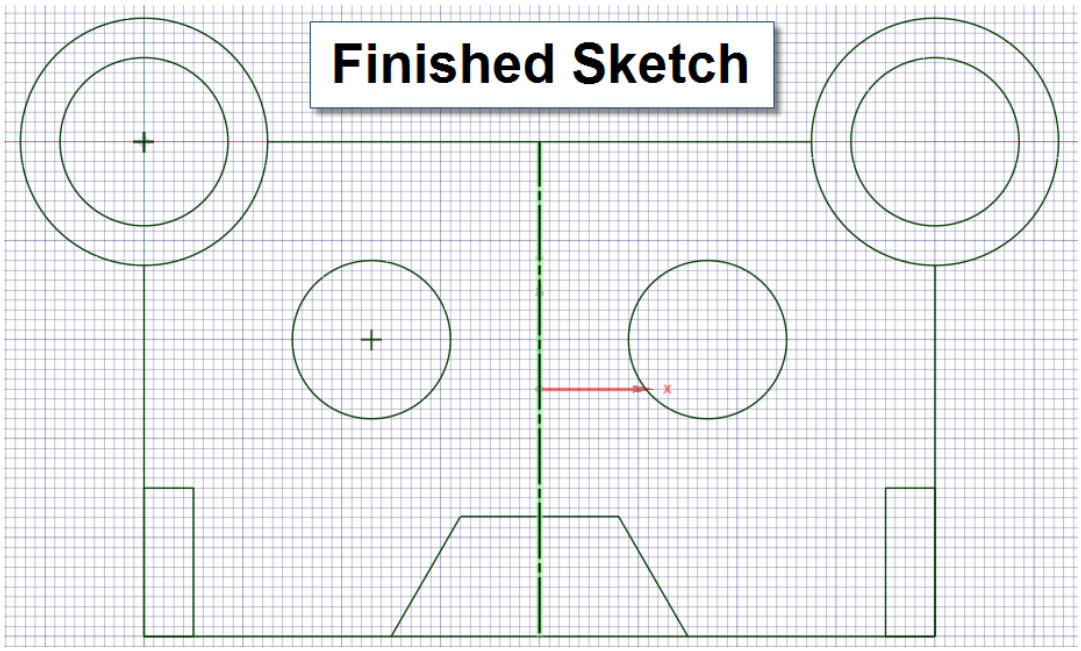


40. **Hover over different curves** (lines or arcs) in the sketch. **NOTICE** that the curves have been broken up into segments based on intersections with other curves.



41. C**lick the indicated curve segments in red** to remove them. Since the mirror line is still active, it will Trim both sides.

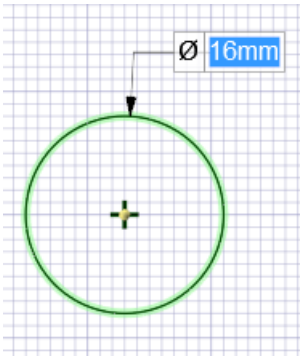




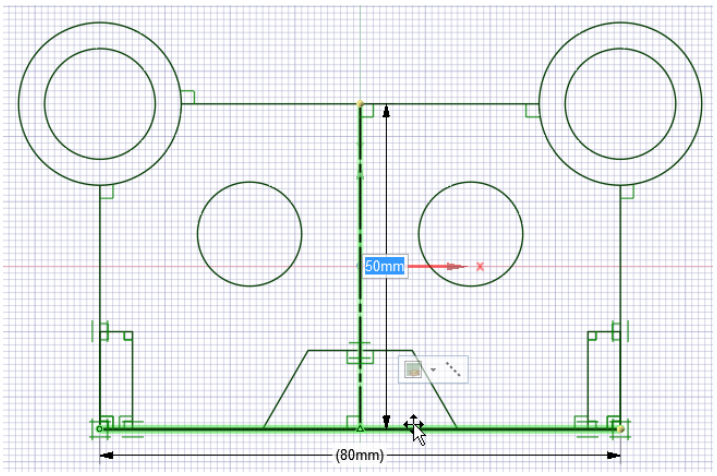
Editing Sketch Dimensions

42. Click the **Select tool**

43. While still in **Sketch Mode**, certain items like circles can be selected, and a dimension will appear that can be changed.

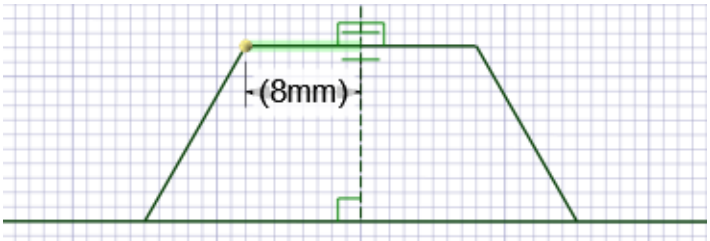


Certain items, like lines can be selected, but the length cannot be directly changed. **NOTICE** with the line selected below, the horizontal dimension of the length is a reference that cannot be edited. The length can be changed by moving the endpoint to the left, right, or both symmetrically.



NOTICE how the vertical dimension in the above image is editable. This dimension controls the location of the selected line. Editing the vertical dimension will move the selected line.

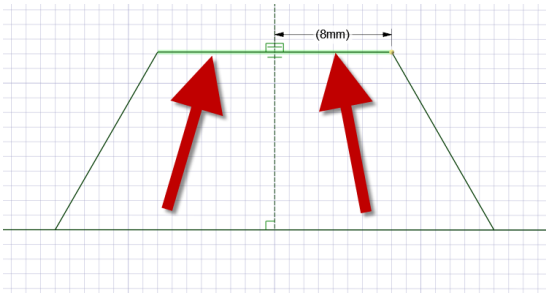
There will be lines like the one selected below that have no editable dimensions associated with them. The same is true for the location of the previously selected circle.

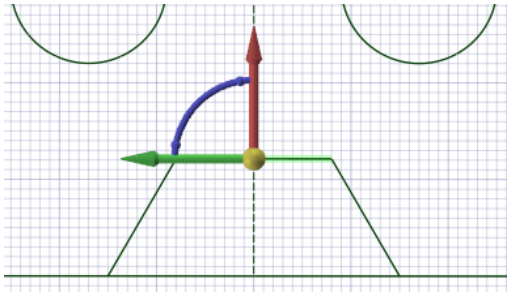


Moving Sketch Entities

44. Select the 2 lines in the image below by clicking one, then holding CTRL and clicking the other.

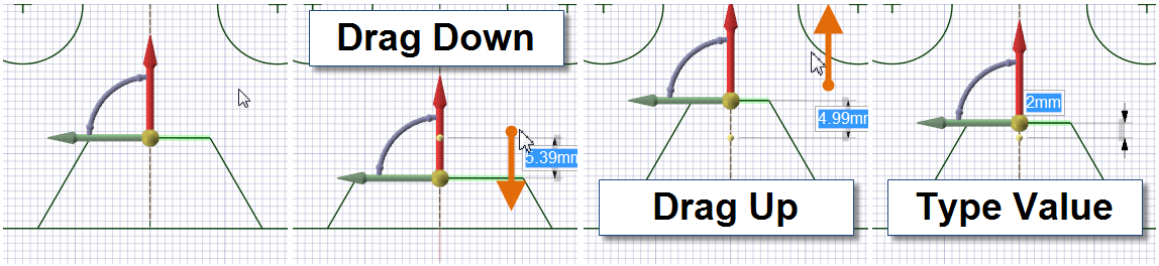
    Click the **Move Tool**. A move handle with 2 two linear arrows and 1 rotational arrow will appear. It will appear bigger or smaller to your sketch based on how far in or out you are zoomed.





45. **Click once on the vertical (RED) arrow.**

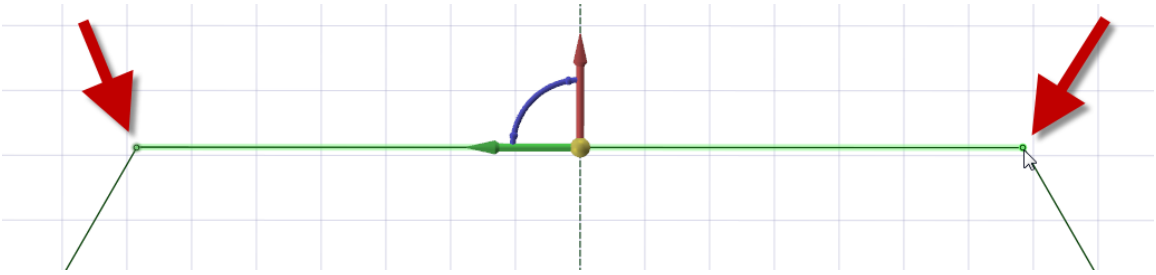
46. **Move your mouse away from the arrow**, (still in the design window) and **drag the mouse up and down**. **Type in a value of 2mm**.



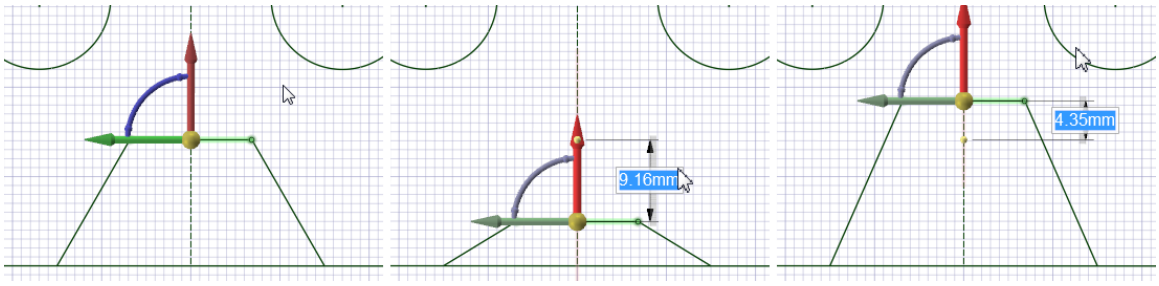
NOTICE the line's length gets bigger and smaller as it moved up and down.

NOTICE SpaceClaim pays a lot of attention to the neighbors/adjacent objects when moving.

47. **Zoom** in until the move handle becomes smaller than the line segments. **Hold CTRL and select the 2 endpoints shown below**. You should have 2 lines and 2 points selected.

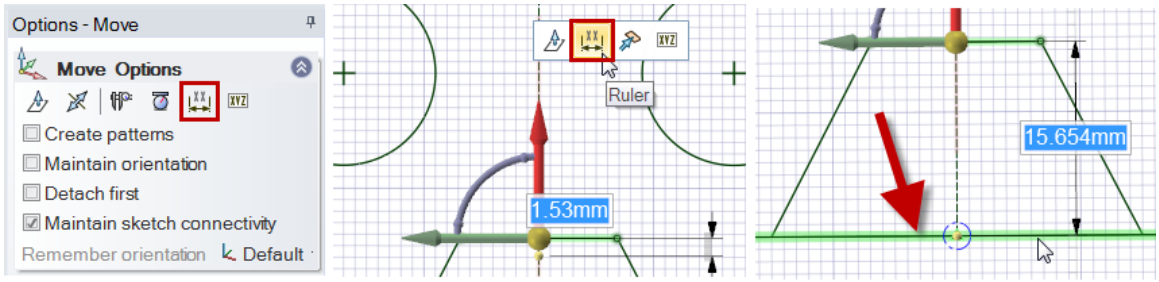


48. **Click the red arrow once**, **then click and drag the cursor anywhere on screen** to move the 2 lines and 2 points up and down by



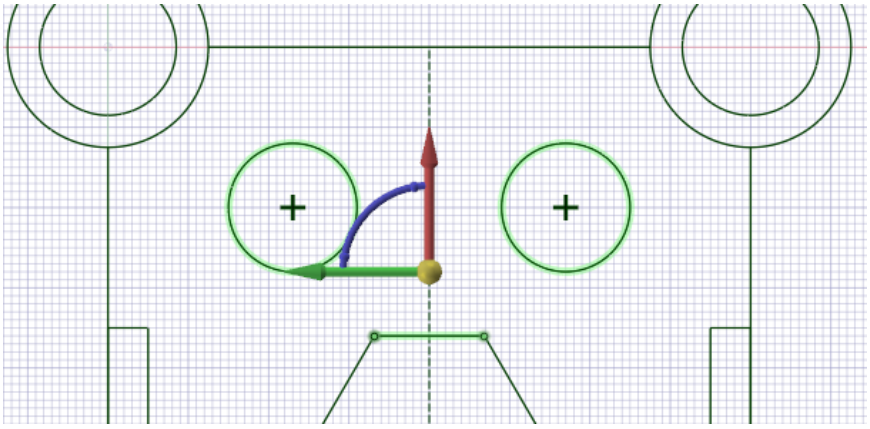
NOTICE that the lines stay the same length, and the neighboring/adjacent lines adjust and adapt their angles. This is because the points (boundaries between the lines) have been selected and are moving straight up and down with the lines.

49. To setup a dimension from a different reference, **click on the red arrow**, and a menu will pop up above the arrow. **Click** the 2nd button, called **Ruler**. A dimension will attach to the cursor as you move it around. **Click on the bottom horizontal line** to dimension from that location.

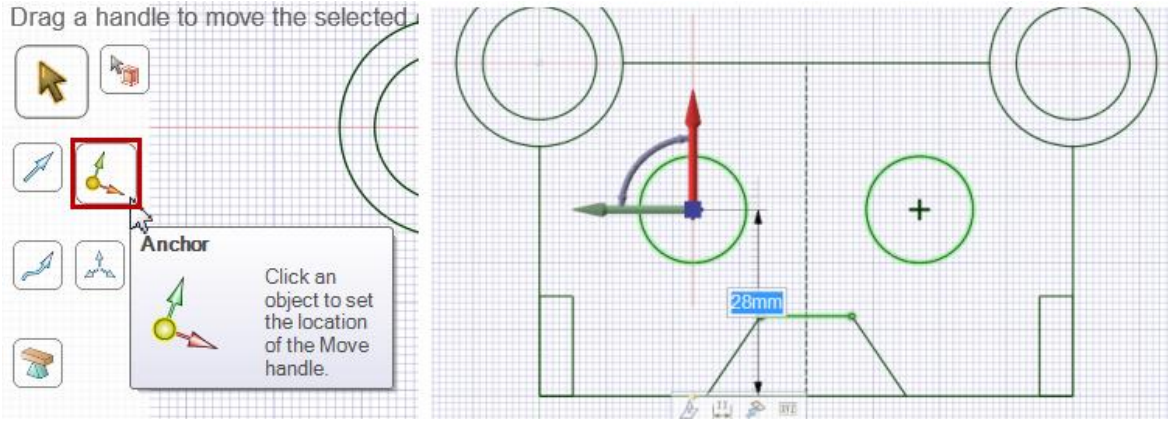


50. **Type in a value of 14 and press enter** to complete.

51. Add both nearby circles to your selection by holding **CTRL** and clicking the circle. **NOTICE** the **Move Tool** moves to the center of everything that is selected.



52. Click the Anchor ToolGuides on the left side of the design window, and click the center of one of the circles to snap the move handle to the circle. Click the vertical arrow of the move handle, and attach a ruler dimension to the bottom line, type 28 and press enter.

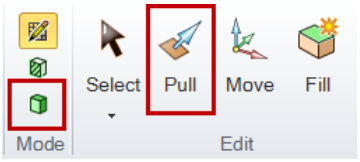


53. **Right Click** on the center vertical mirror line and select **Delete**

Sketch Mode to 3D Mode

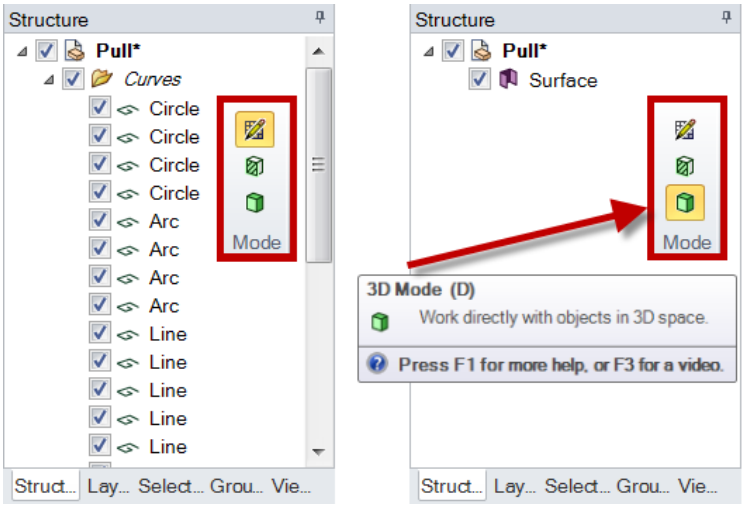
54. Click the arrow next to Curves in the Structure Tree on the left to see the folder of curves.

55. Click the **3D Mode** button or the **Pull** tool to exit out of 2D sketch mode, and into 3D mode.

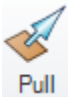


NOTICE in the Structure Tree that the folder of closed curves has automatically converted into a surface.

If you select the surface and click the **Sketch Mode** button, you will return to the sketch. Once you Pull the surface into a solid, you will **NOT** be able to return to the sketch.



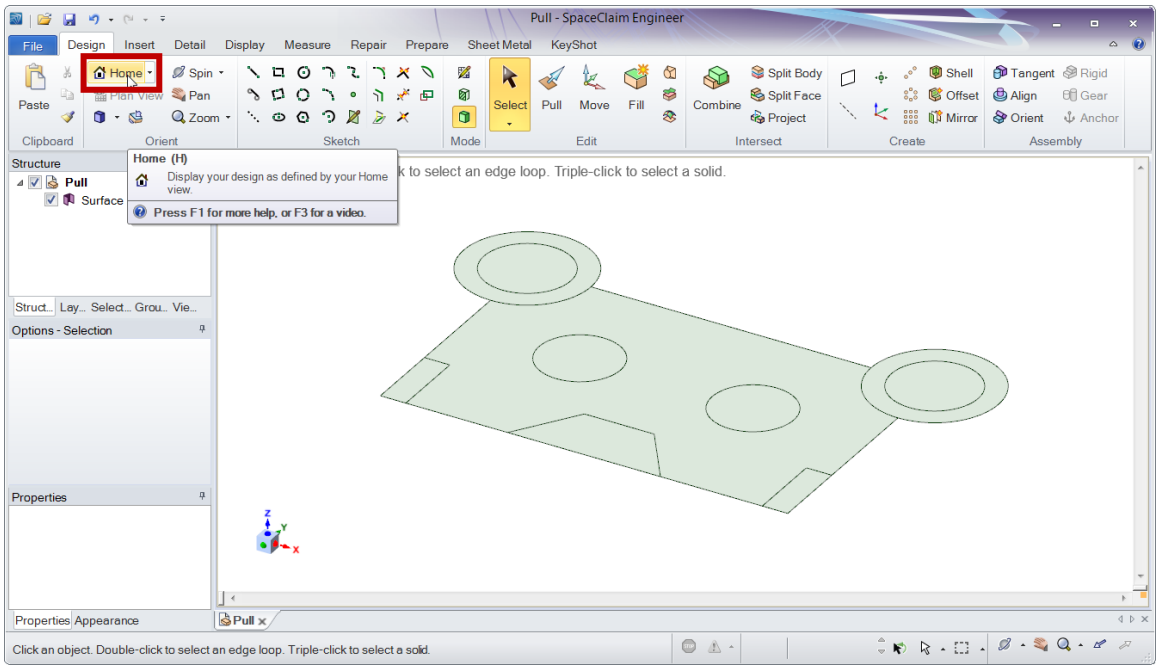
Pull



The Pull tool is the main and most frequently used tool in SpaceClaim. It is the primary tool used to create and edit geometry. In this course, we will use it to thicken a surface into a solid, and then edit the solid by pulling faces. We will also learn how to pull edges into rounds and chamfers, often referred to in other software as fillets or blends.

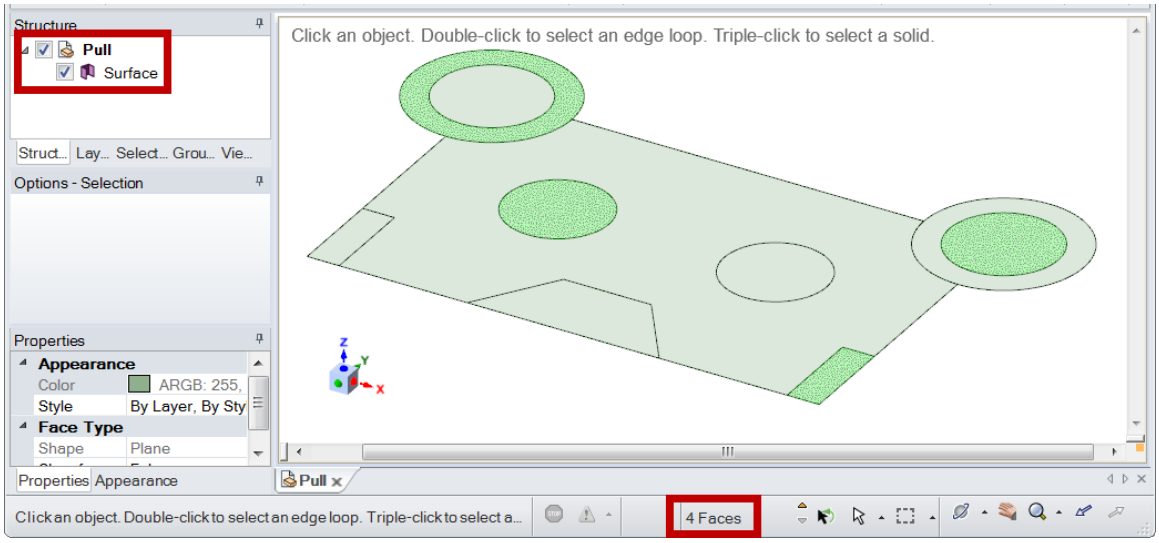
In later courses, Pull will be used to: Revolve, Draft and Sweep faces; Scale bodies; copy, and pivot edges.

1. **File\Open.** Navigate to Desktop\SpaceClaim_Basic_Training**\02_Basic_Pull_2014.0** and open **Basic_Pull_2014.0.scdoc**
   a. When you open a SpaceClaim file, the part will open in the same orientation and zoom it was saved as.
   b. If you open a non-SpaceClaim file, it will open in the default Isometric view, also called the Home View.
2. The Basic_Pull_2014.0.scdoc file was saved while looking straight at the sketch, in a normal or **Plan View**. It is very hard to Pull a surface or face that is normal to the screen.
3. **Click the Home** icon in the top right corner or type H to return to the default isometric or Home view.



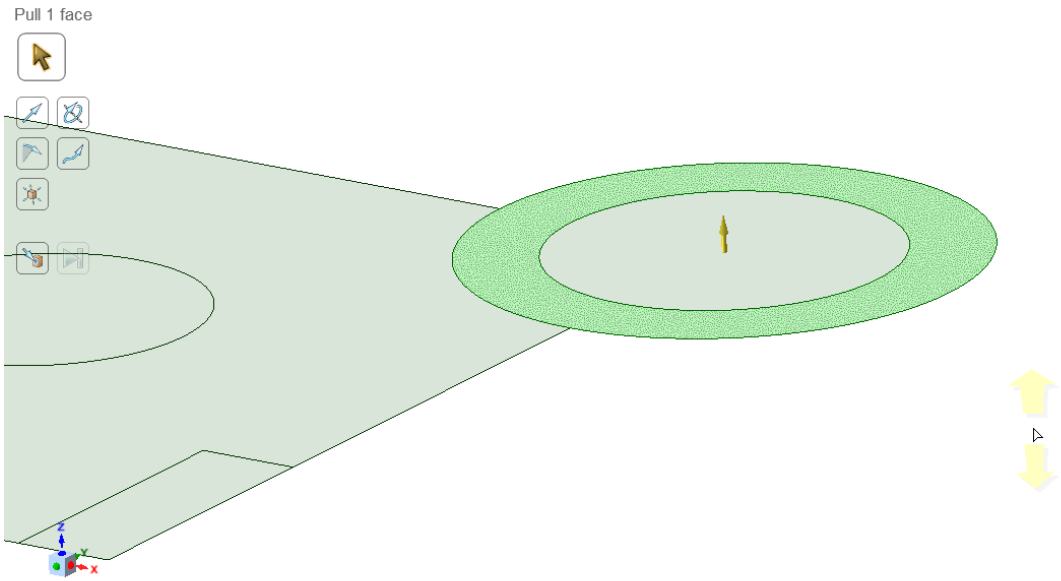
Pulling Planar Faces

Notice in the structure tree that this is 1 single body called a **Surface**. The surface has 10 regions referred to as **Faces**. The image below shows 4 faces selected, and the panel which confirms this.



4. Turn on the **Pull Tool** by clicking it.

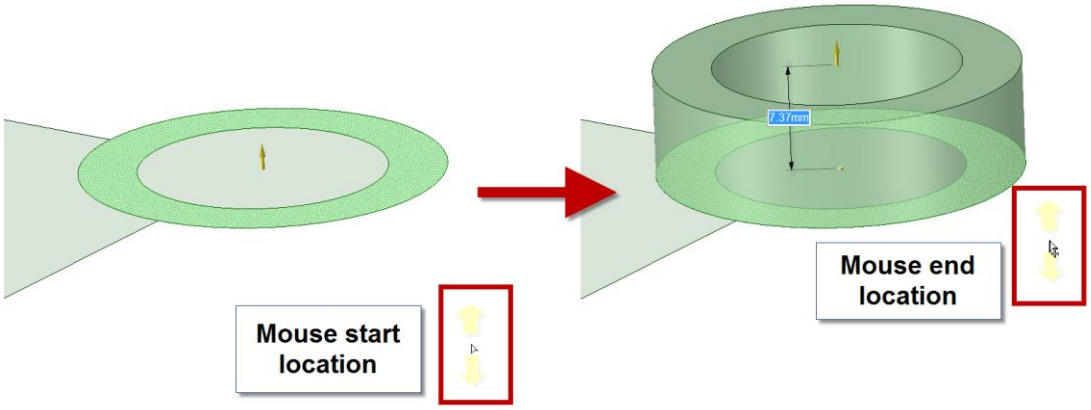
5. Spin (middle mouse button), Pan (Shift+middle button) and Zoom (CTRL+middle button) to get into the view below, then click the highlighted face.



.

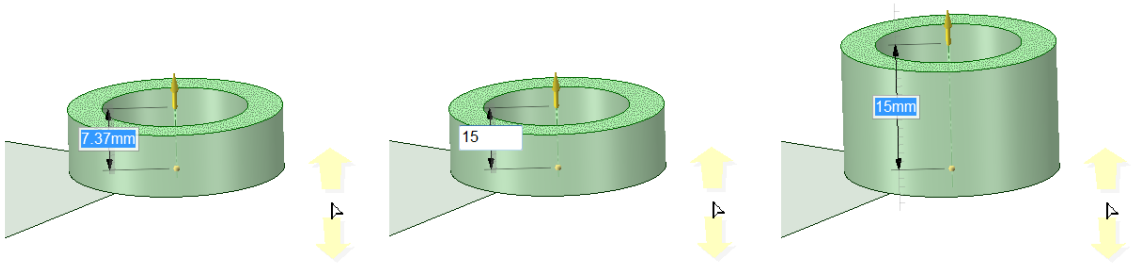
6. **Drag** (hold down the left mouse button while moving mouse) the mouse up and let go.

   NOTICE: The mouse does not need to be on the surface you are pulling\dragging, nor do you need to click the little yellow arrow pointing up from the center of the surface.

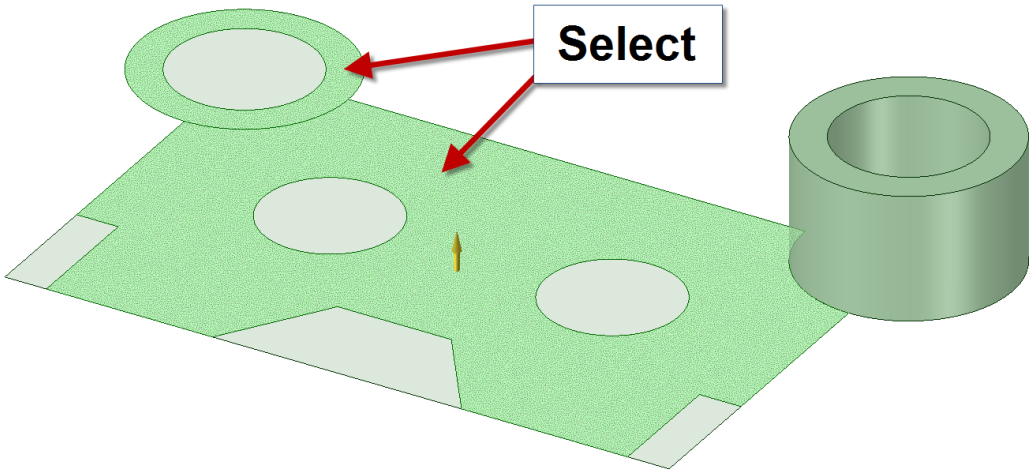


   After letting go of the mouse, the face you pulled will still be selected, and you can drag again to pull it up or down.

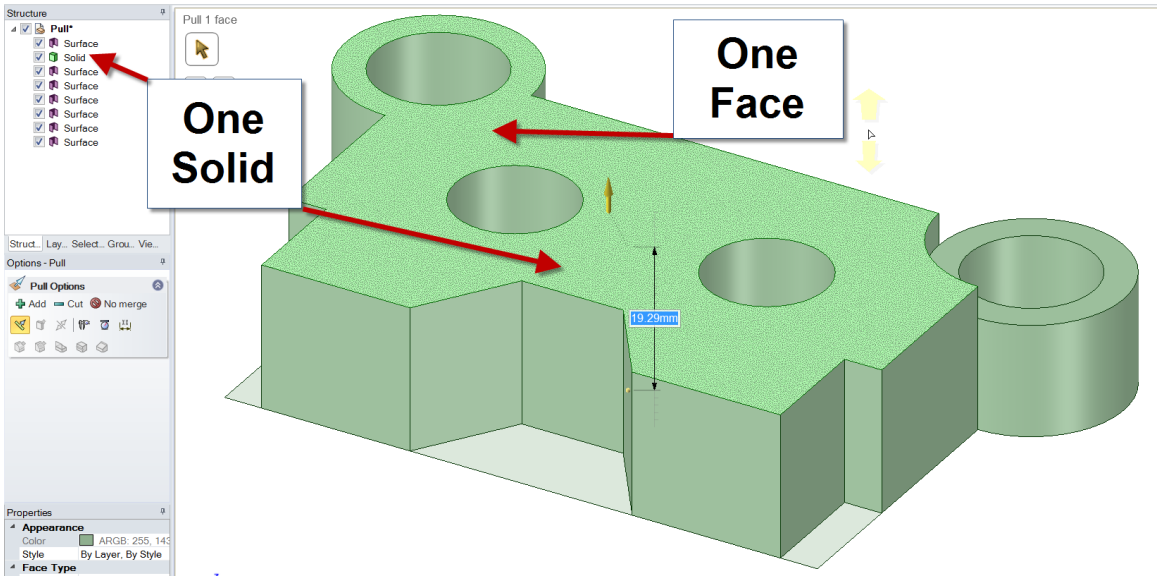
7. To enter a precise value, type 15 and press enter.



8. Press the **Home** button or H to return to the original Isometric view, and select the 2 faces below.

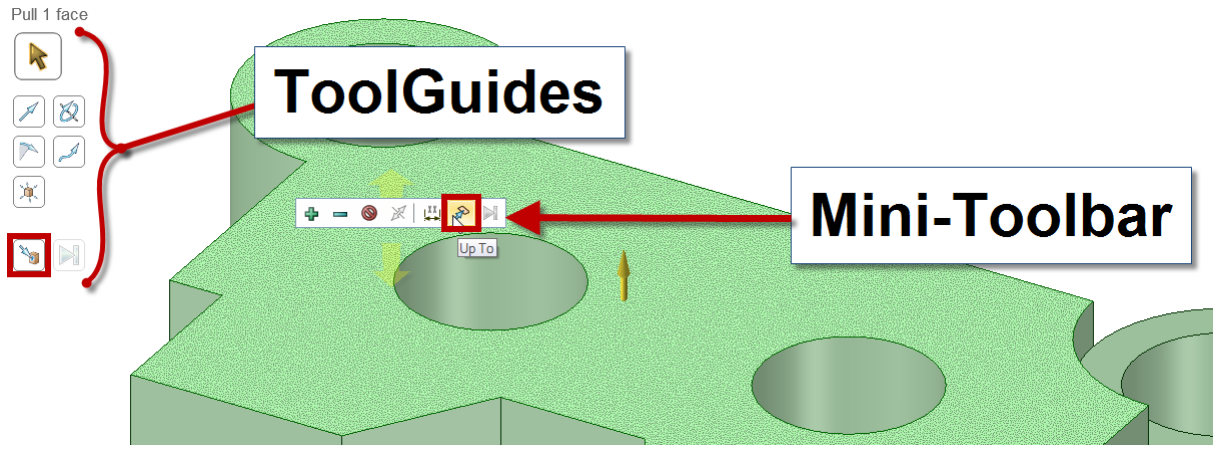


9. Drag the faces up any distance except 15mm and let go. **NOTICE** the 2 selected faces merge together into 1 face, and the new solid merges with the first solid

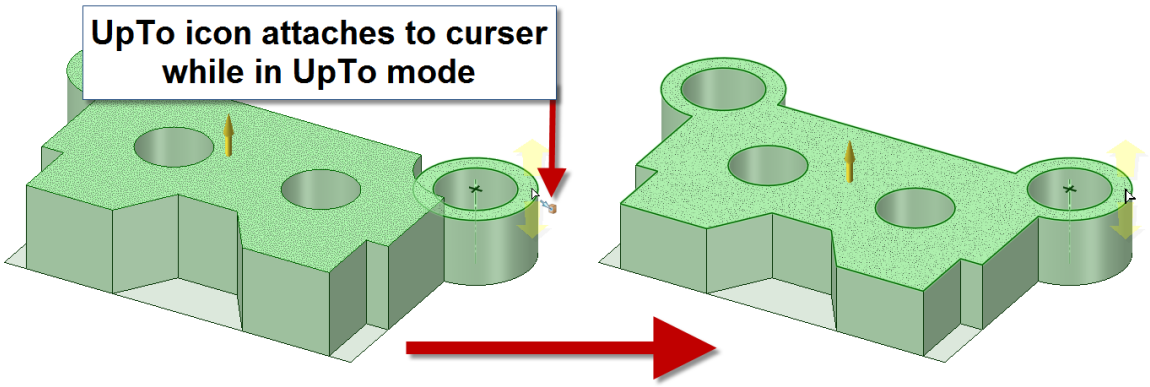


We want the new face to be the same height as the first face. You could type in the same 15mm value, but instead:

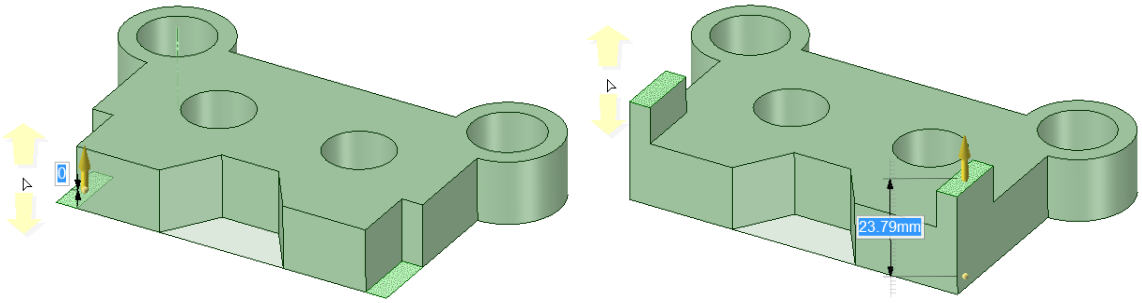
10. Click on the highlighted face below, and a Mini-Toolbar will start to appear near the cursor.
11. Move your cursor towards the **Mini-Toolbar** and click the highlighted button called **UpTo**. It allows you to pull a face up to another face, to snap them together. Even when the mini-toolbar is not shown, you can find **UpTo** in the **ToolGuides** on the left.



12. After clicking the **UpTo** button, click the first face you pulled up, to snap the faces together.



13. We've pulled a single face, and 2 faces that were touching. Now select the 2 small rectangular faces below that are not adjacent to each other, and pull them up as shown.



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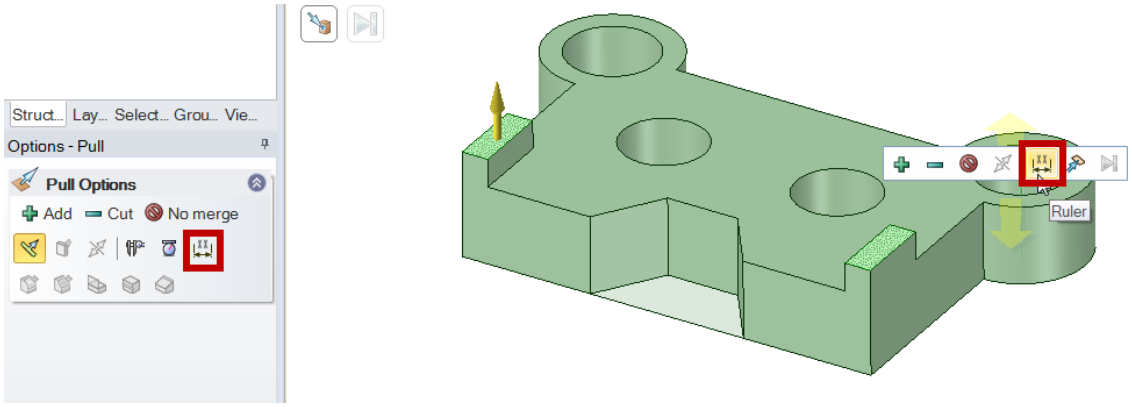
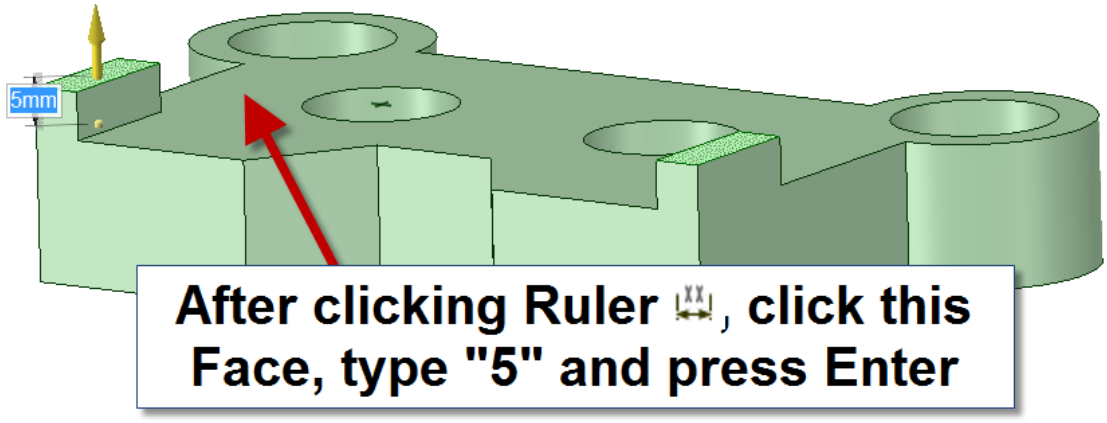
14. To reference a different face while dimensioning, click the **Ruler** button in the options on the left, or reselect the faces and click **Ruler** from the **Mini-Toolbar** to setup a Ruler Dimension from these 2 faces to the large top face.

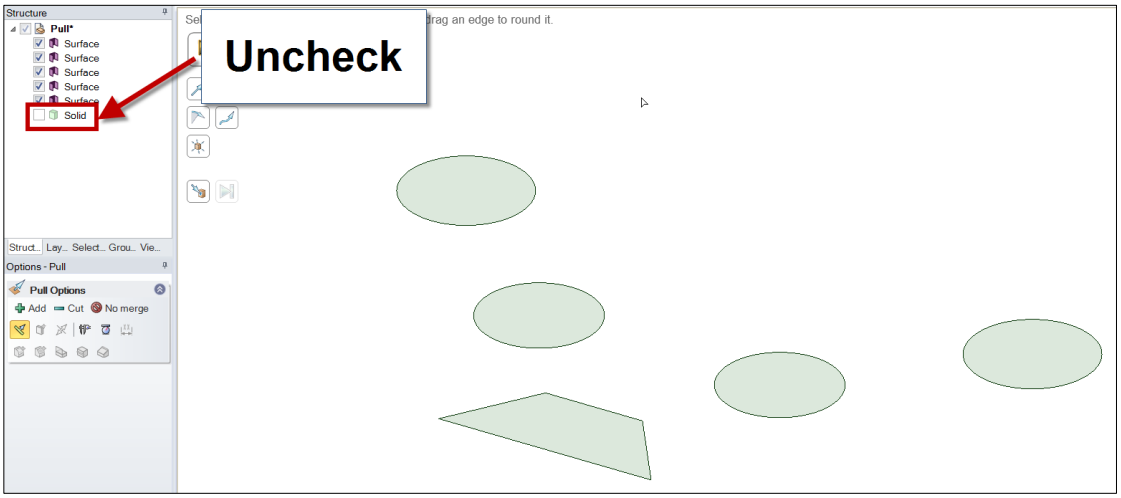


As you move your cursor around, the dimension will follow the cursor. The ruler dimension will attach itself to the next item you select.

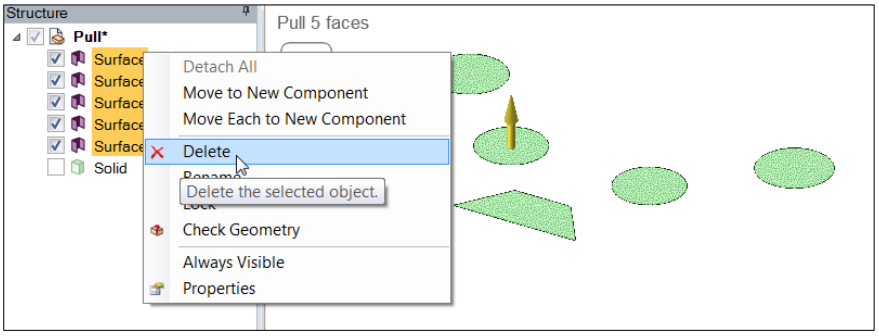
15. Click the top face to attach the ruler to it. Type 5 and press **Enter**.



16. Press **H** for home or click the **Home** button, and hide the solid by unchecking it in the tree.

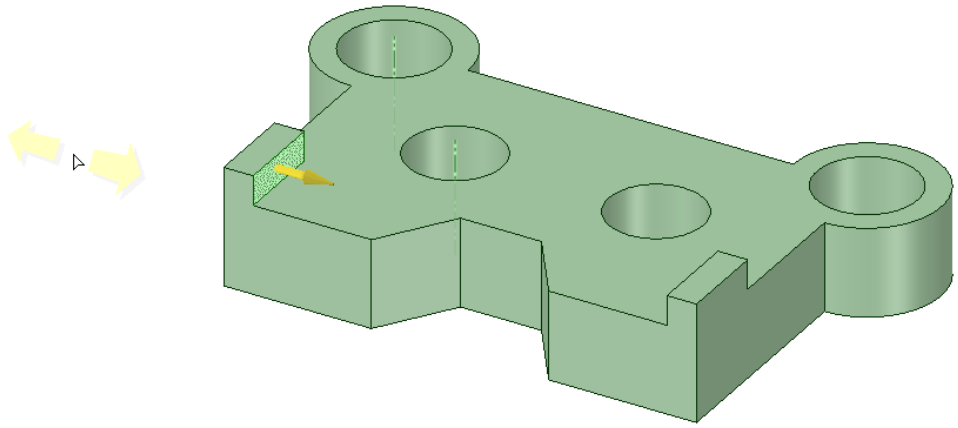


These remaining surfaces are the faces of the sketch we did not pull. They are not associated to the solid. If we change the size of one of the holes in the solid, the circular surfaces won't change. Having these extra surfaces won't hurt us, but to avoid confusion, select and delete them in the structure tree.



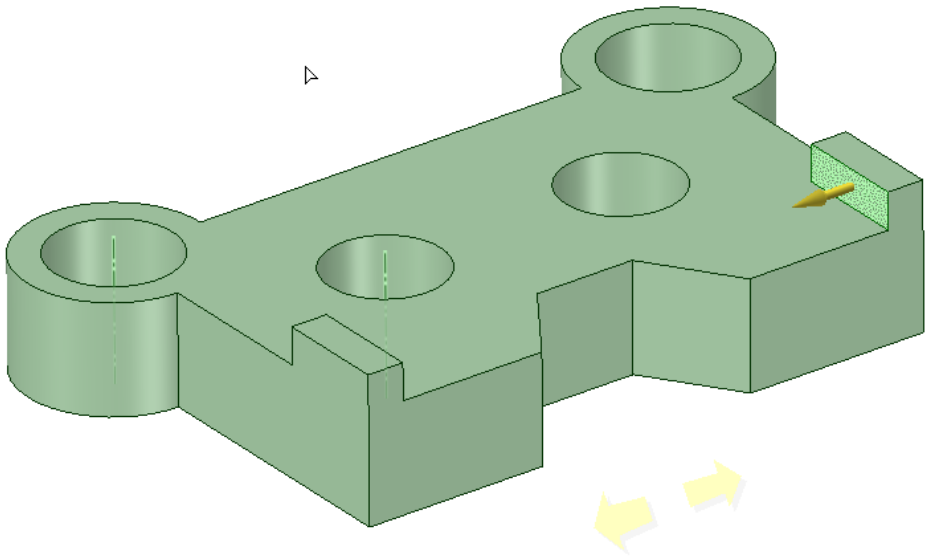
17. Unhide the solid by re-checking it in the structure tree.

Any face on this model can be selected and Pulled. Select a vertical face in the model like the one below. Notice the arrow is pointing to the right, and the arrows around the cursor are left to right, instead of up and down. The arrows tell you what direction to drag the mouse.



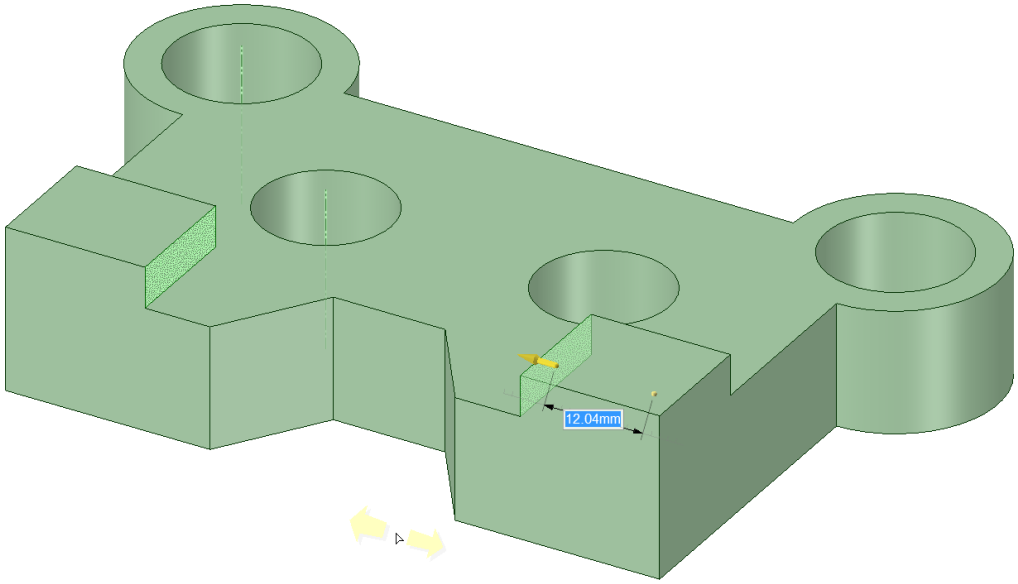
Drag this face right and left, noticing you can type in a dimension. After dragging it, press CTRL+Z to undo, until you get back to the image above.

18. Click the opposite face, and notice the arrow is pointing in the opposite direction. This arrow is the Normal direction of the Face.



You can drag 1 face, or multiple faces at the same time, regardless of how the model was created.

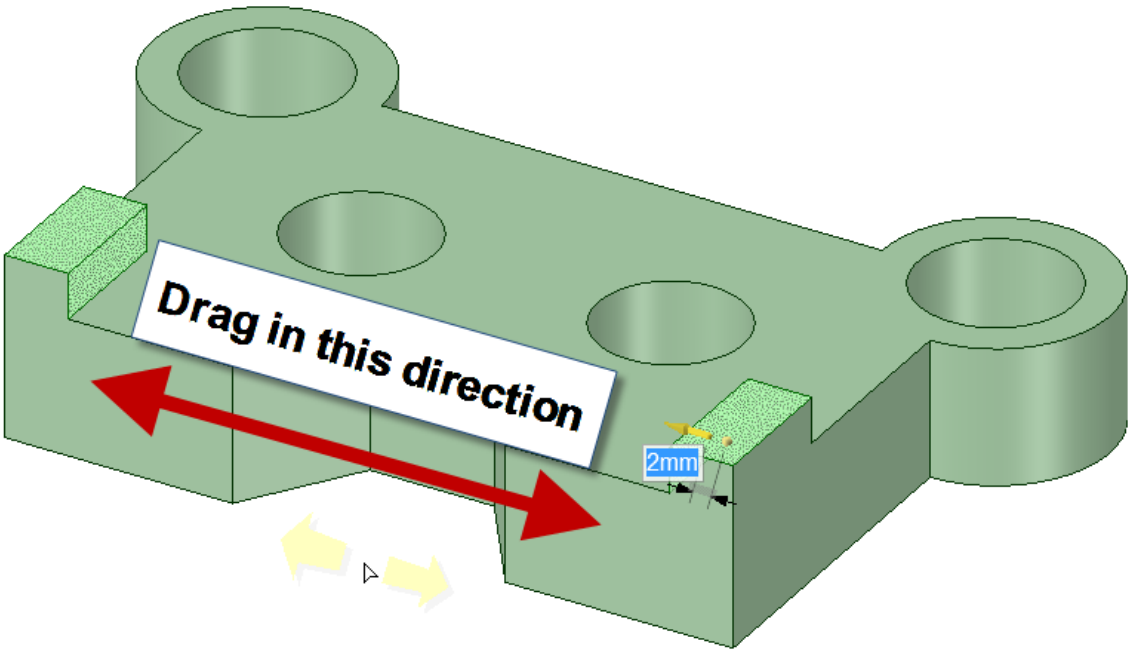
19. Select both faces previously selected (by holding CTRL when you click each face)



20. Drag left and right. The faces are being offset which means they are going in the positive direction of the normal, or the negative direction of the normal

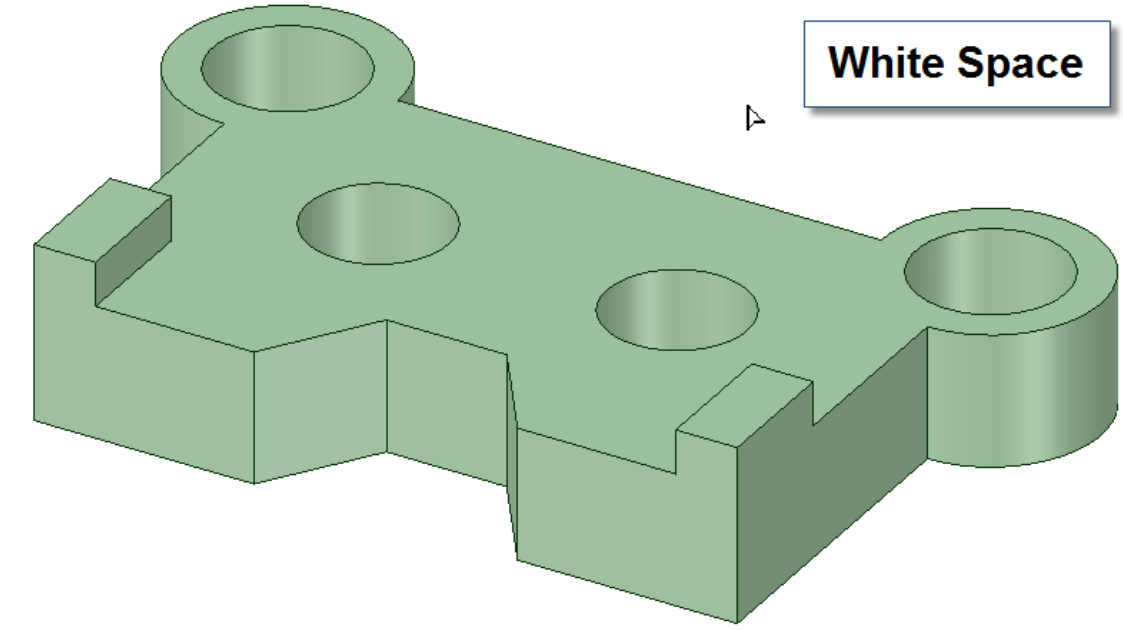
21. Type 2 and press enter to pull them both in 2mm.

22. Add the 2 faces above the 2 selected as shown below, and drag left and right. The face with the Yellow arrow is the first selected face, and tells you what direction to drag your mouse.

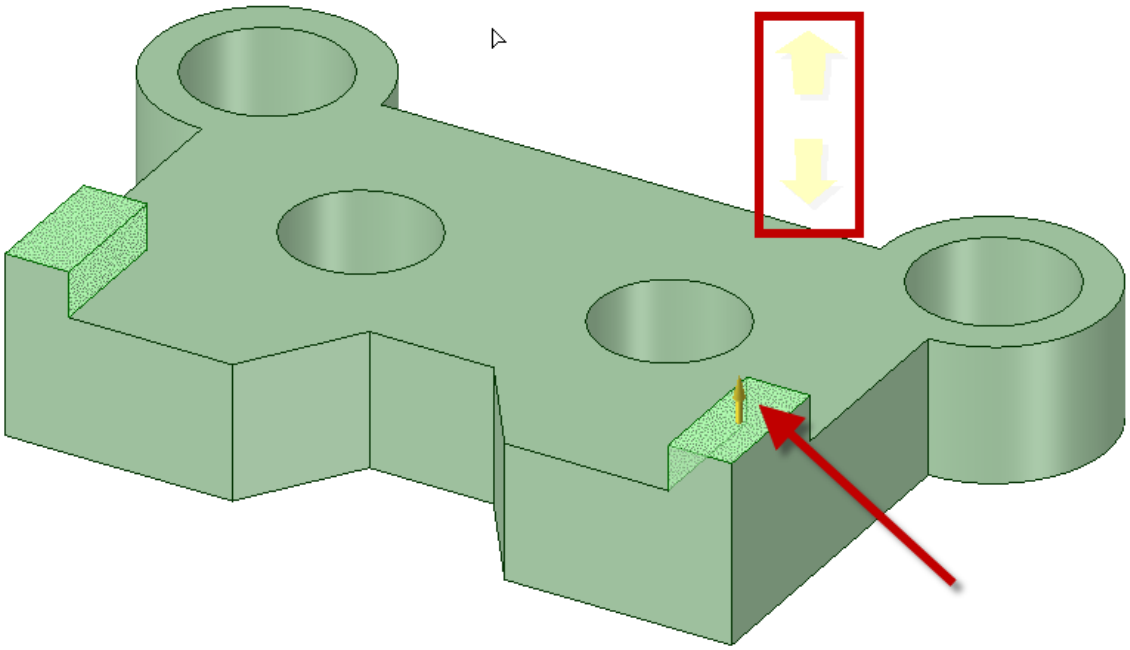


23. Drag left and right, then type 4 and press enter. This will pull all the faces an additional 2mm on top of the original 2mm. a total of 4mm from the original location for the first 2 faces.

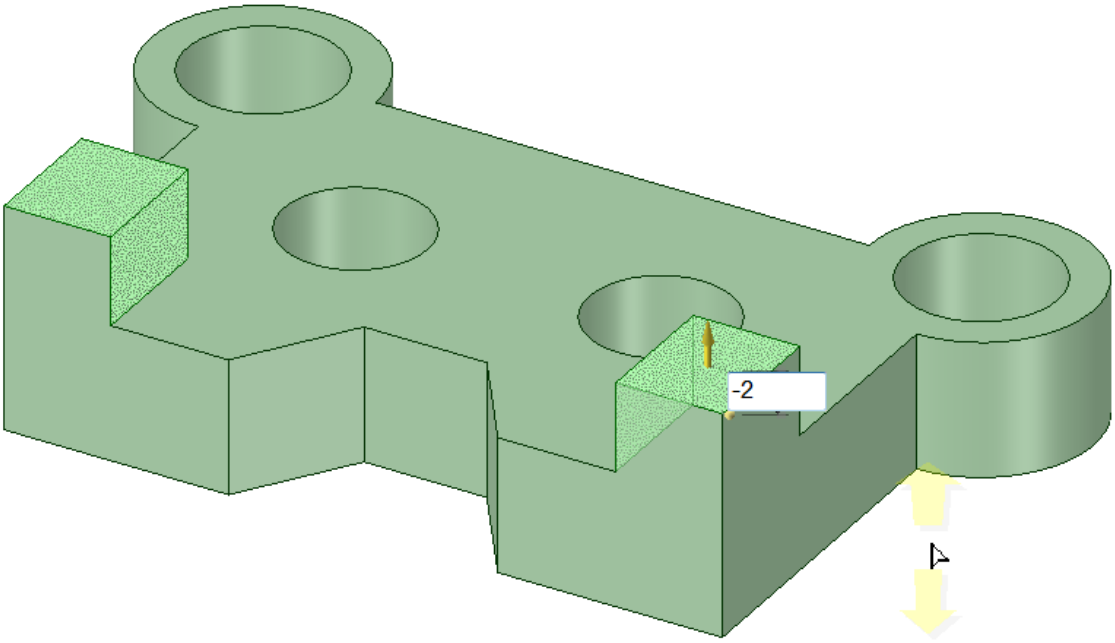
24. Click in white space (the space around the model, inside of the design window)
Clicking in white space clears the selection, and resets the dimension.



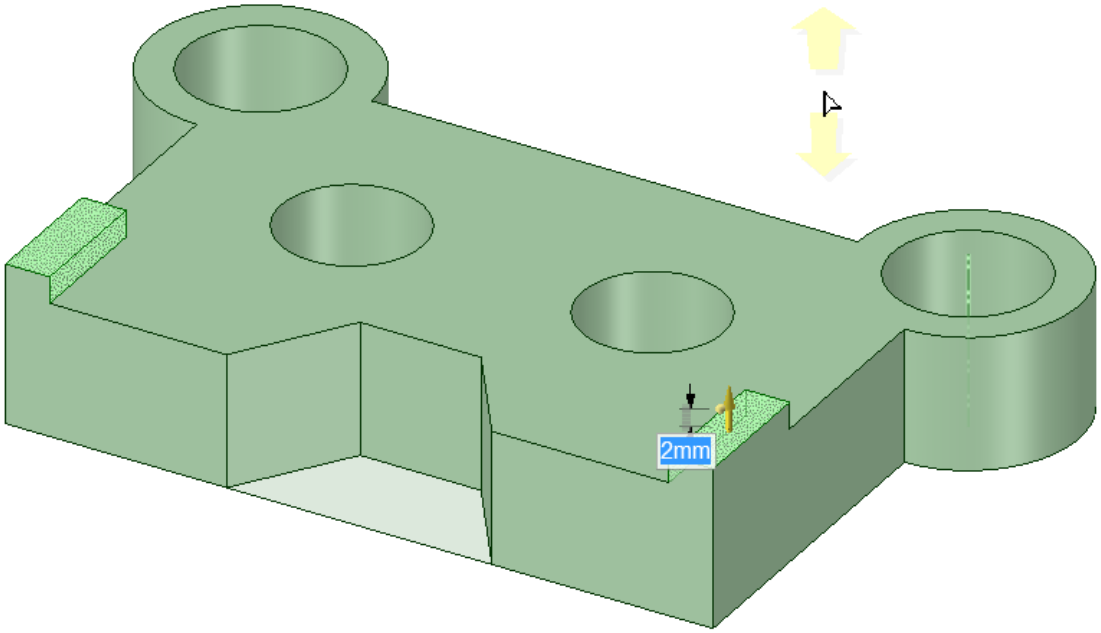
25. Select the same 4 faces, this time start with the one with the arrow indicated below. Notice the arrows around the cursor are straight up and down. In order to pull the same 4 faces now, you have to drag up and down instead of left and right.



Stop dragging while above the original location, i.e. adding material,



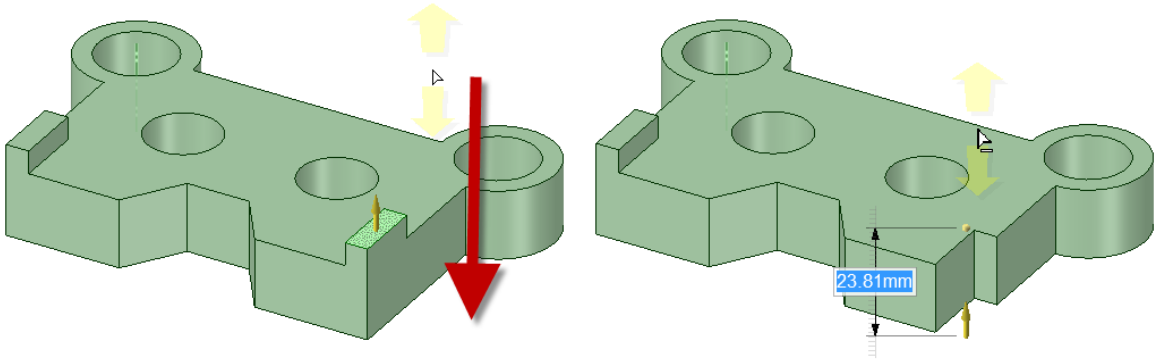
26. Type -2, and it will pull the faces until 2mm is subtracted from the original l



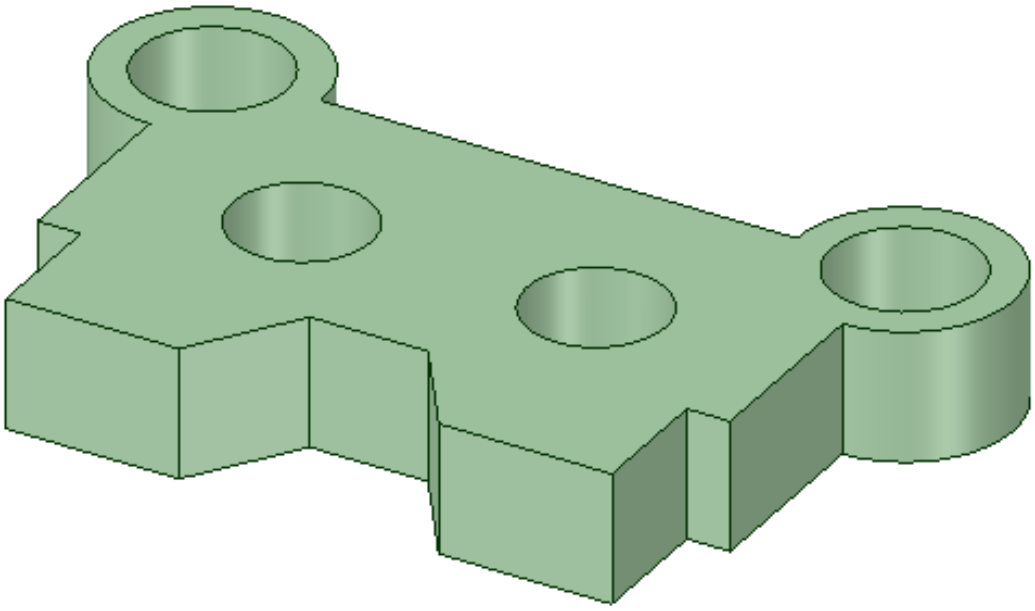
Notice the faces are pulled 2mm from the original location, not the location you dragged them to. To pull them 2mm from the location you dragged them to, you'd click in white space and reselect them

Design changes are often drastic. Maybe these 2 protrusions were supposed to be cut outs. No need to undo back.

27. Select the face below, and drag it all the way down to turn the protrusion into a cutout. **NOTICE**: even though the 2 blocks sticking up on the top were created together, they can be edited independently. They are not constrained together

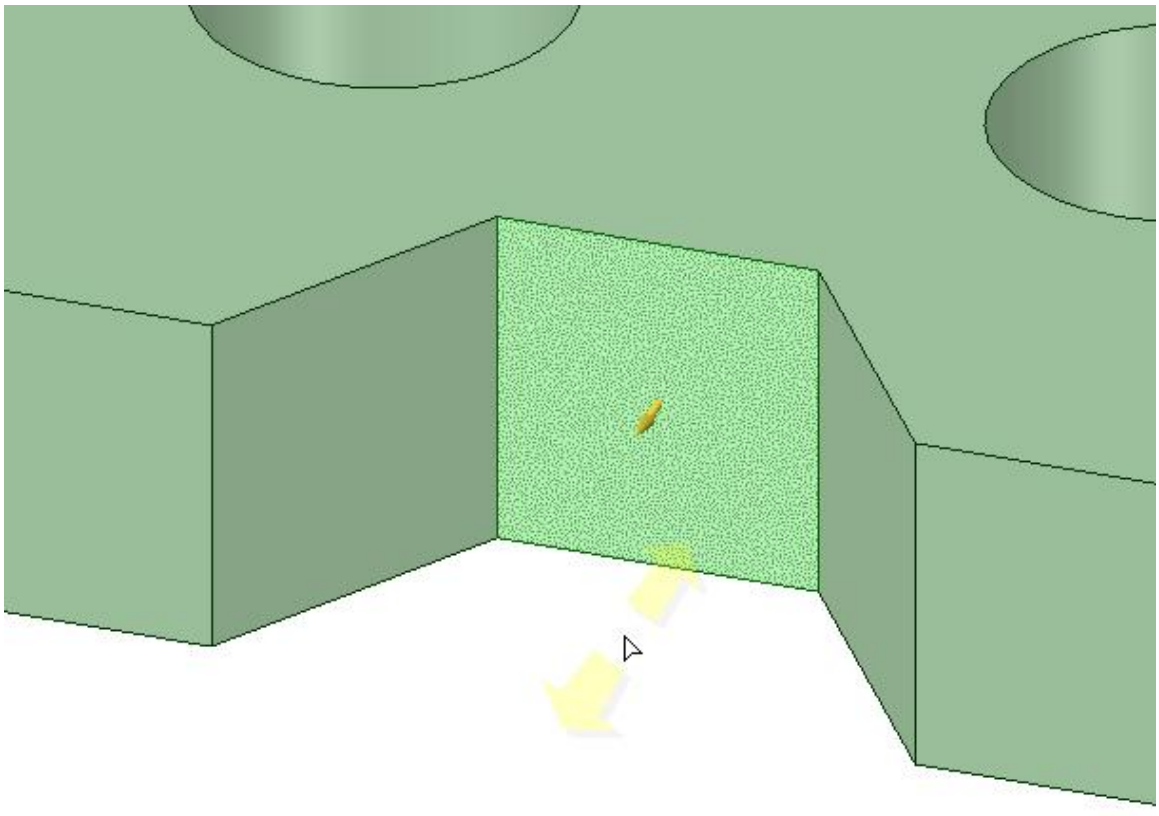


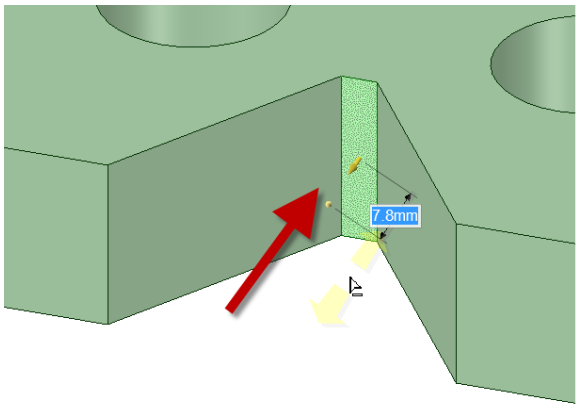
28. Repeat with the other face

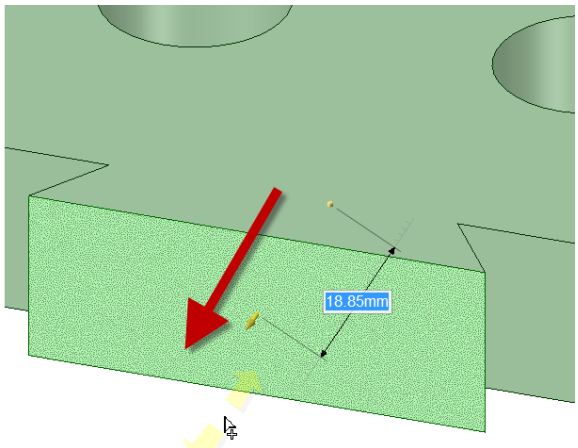


29. Select the face shown below, and Pull it in and out.

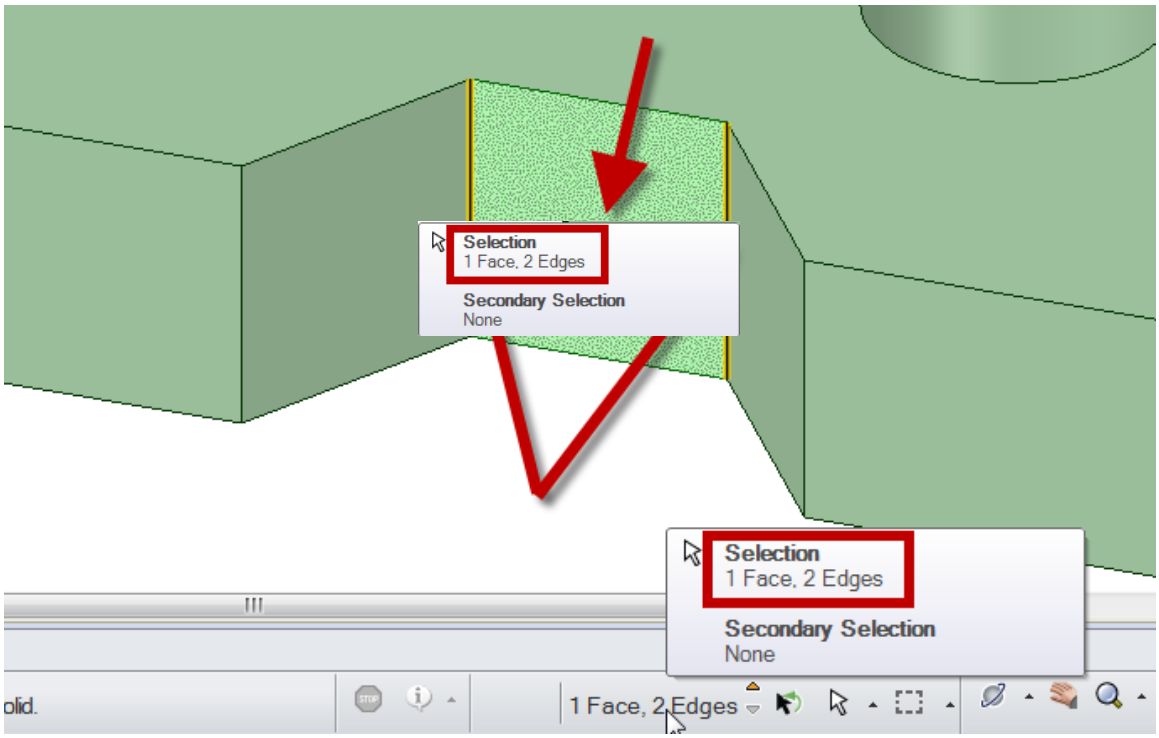
Notice how the select face is affected by the neighboring/adjacent faces as you pull. It “follows” the angle of the neighboring faces.



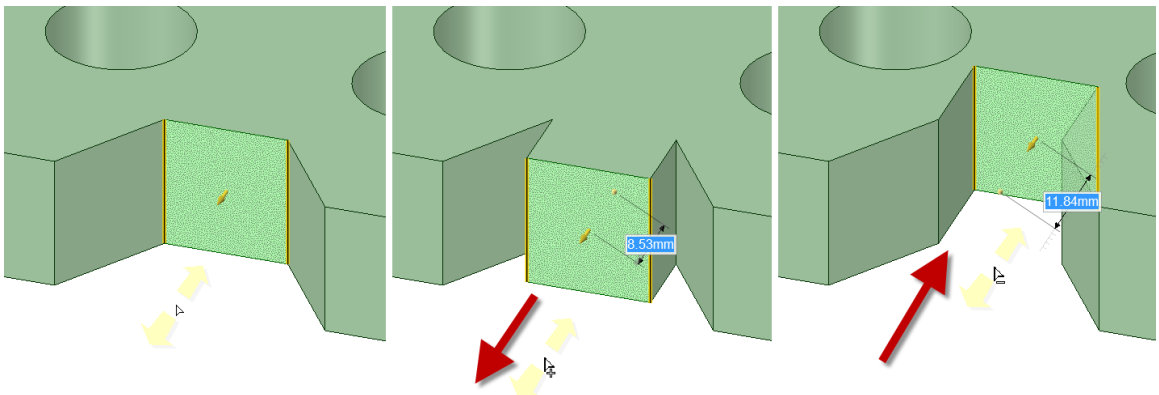




30. Undo (CTRL+Z) before pulling the previous face.
31. Select the face AND the 2 edges on either side of the face.

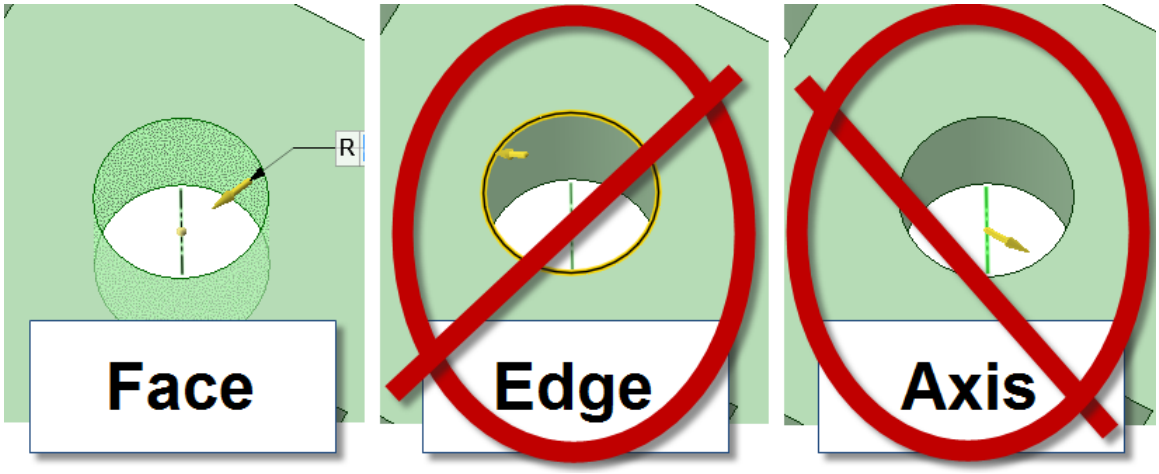


32. Pull in and out to add and cut, and make a cut of 5mm. Notice how by selecting the edges aka boundaries between the faces, you override the relationship to the neighboring/adjacent faces.



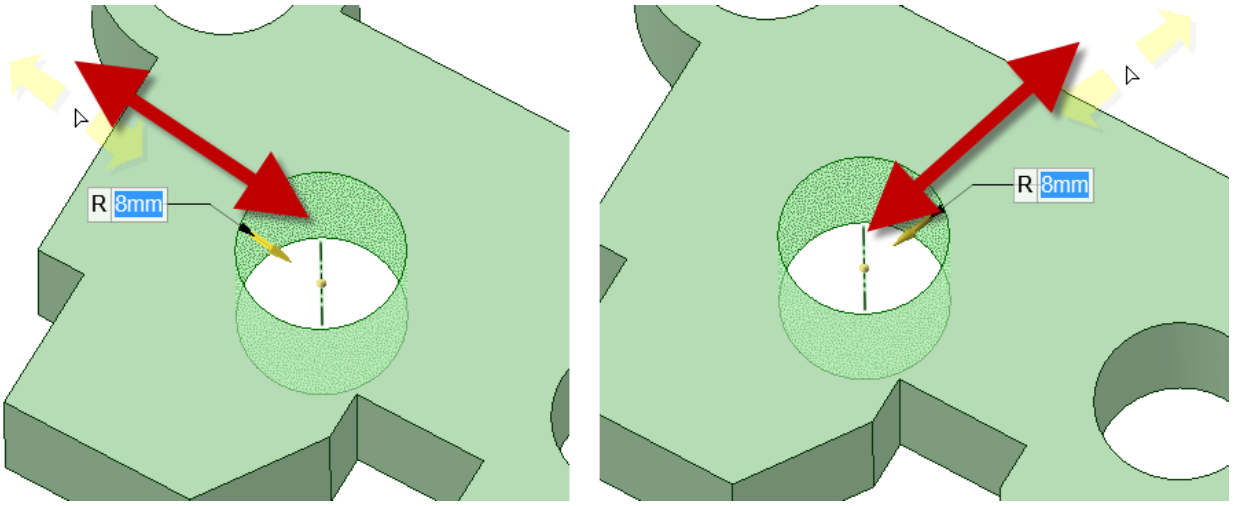
Editing Cylindrical faces

33. Spin the model so you can see through the hole, but not a straight view from the top.
34. Select the face of the hole. **Important**: Do not select the edge or axis of the hole.

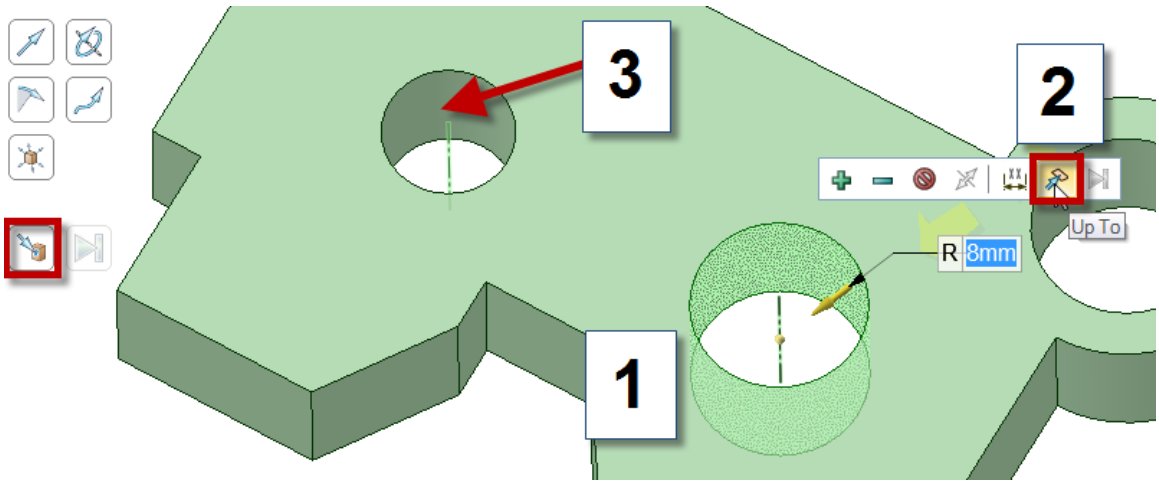


35. Confirm your selection with the panel on the bottom of the screen.

36. From wherever your mouse cursor is, drag towards and away from the center of the hole to make it bigger and smaller. Enter a value of 6mm for the radius and press enter.

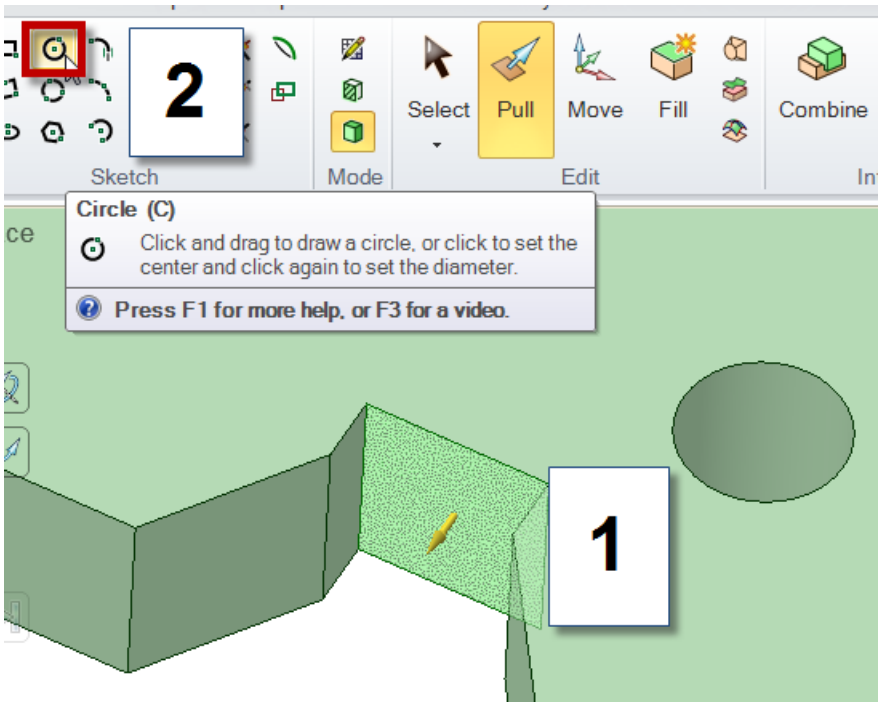


38. To set one radius equal to another, select the new hole, click **UpTo**, then select the existing hole.

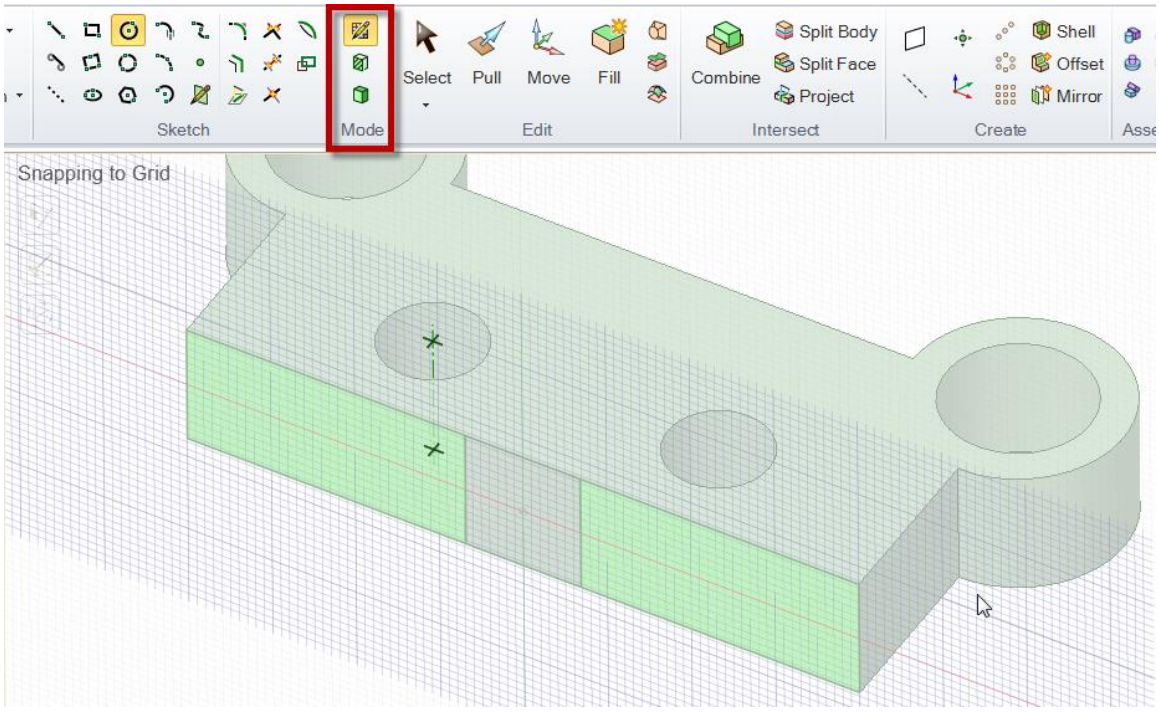


Sketching on a solid

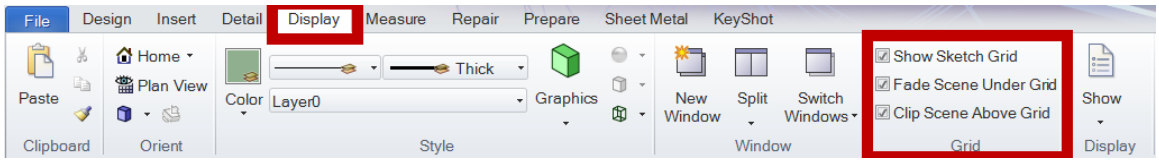
39. Select the face below, and click the circle tool in the sketch group. This will activate sketch mode, placing a sketch plane through the selected face.



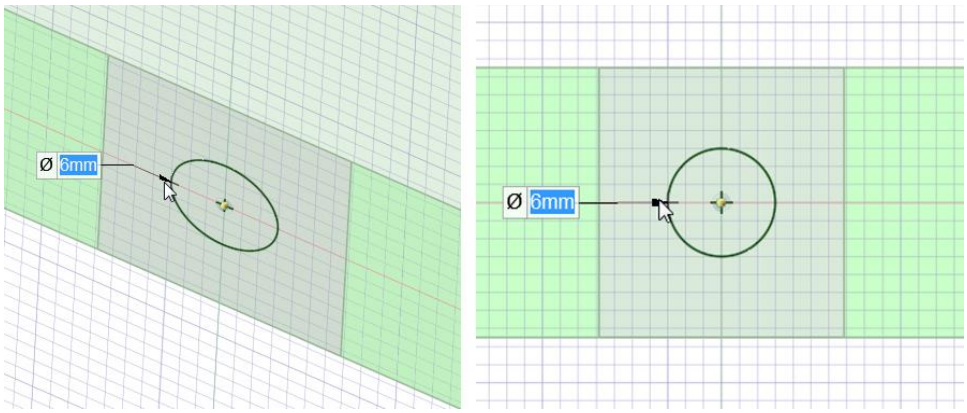
The image below shows SpaceClaim in sketch mode with a grid through the selected face.



40. Switch to the display Tab. There are three options controlling cross section and grid settings. Toggle each on and off in different combinations to see what they do, and which you like.

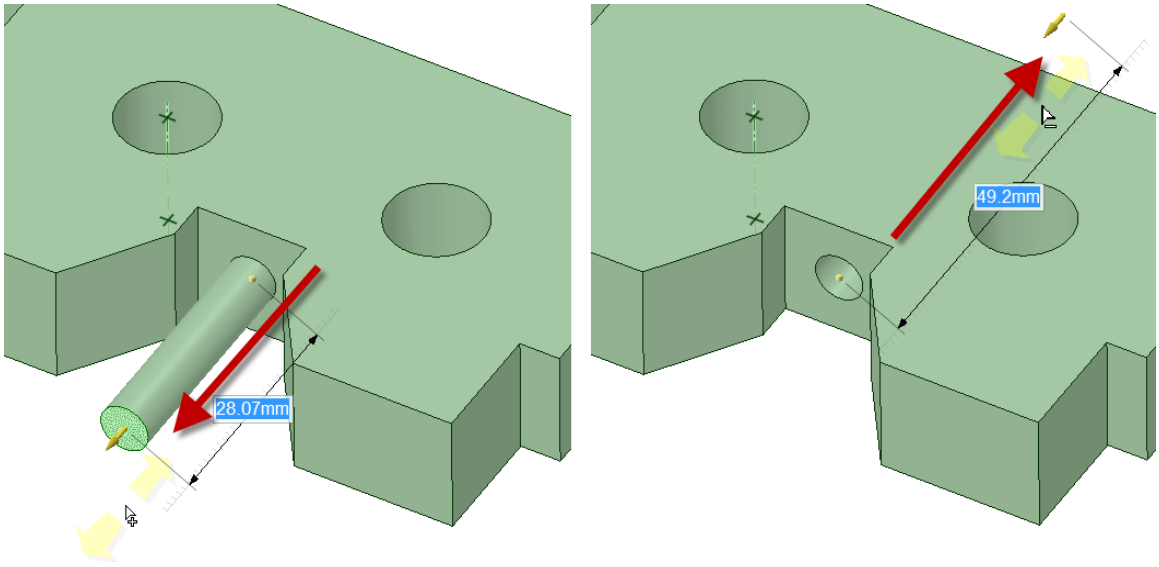


The sketch grid will be centered on the face. Click on the center of the face to start the circle. Type in a value of 6mm. You can sketch in the **isometric** or **Plan View**.



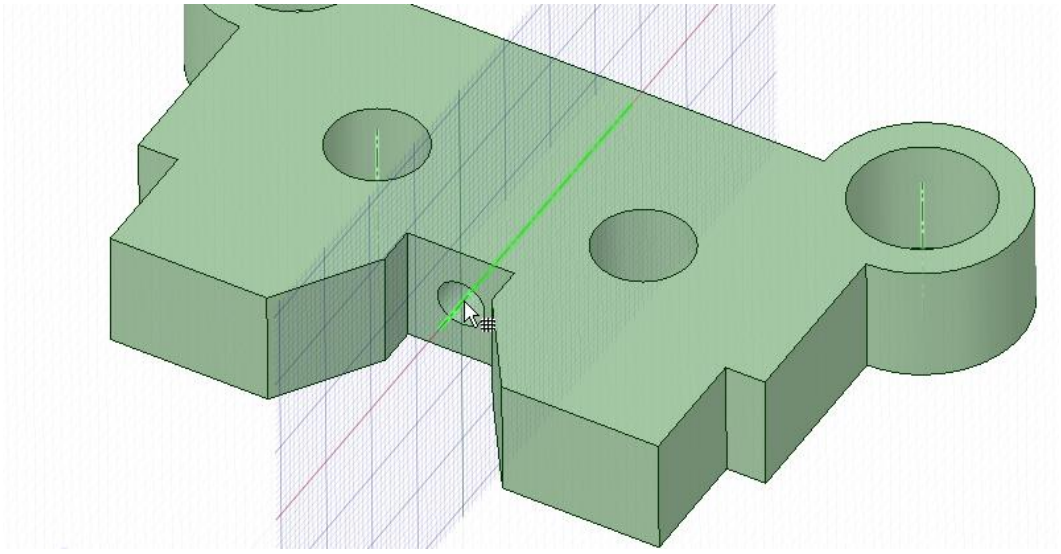
41. Click the Pull tool. SpaceClaim will automatically switch from sketch mode to 3D mode. The circle sketched on the face will automatically imprint a new onto the original face.

42. Select the new circular face, and drag in and out. End by pulling a through hole out the back of the part.

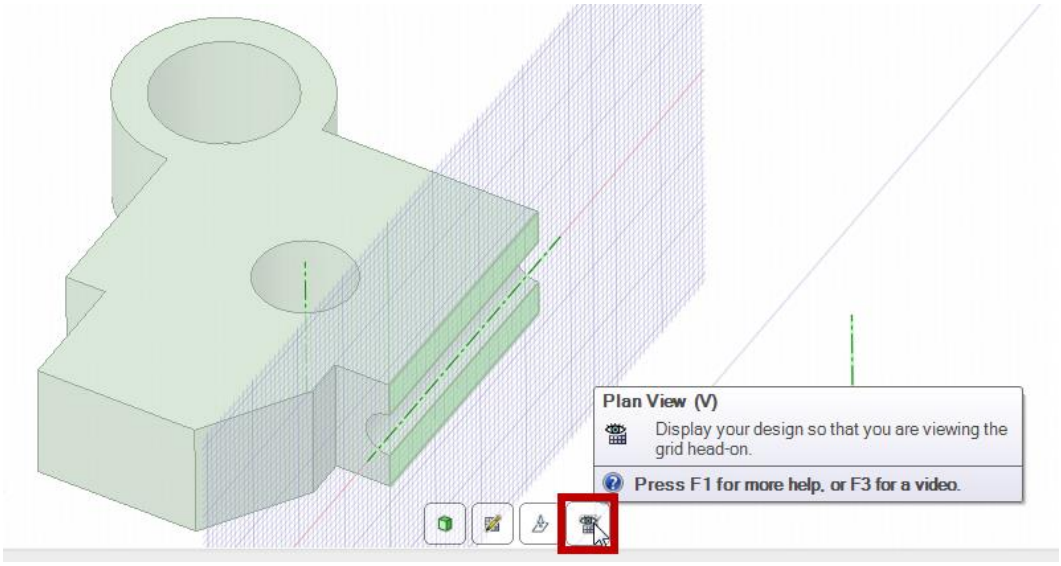


43. Click the Rectangle tool
44. Since nothing was selected when we turned on the Rectangle tool, SpaceClaim wants you to choose a sketch plane. Notice the grid icon attached to the mouse curser.
45. **Hover** (don't click) over different faces of the model, and SpaceClaim will **preview** the grid through the hovered geometry.

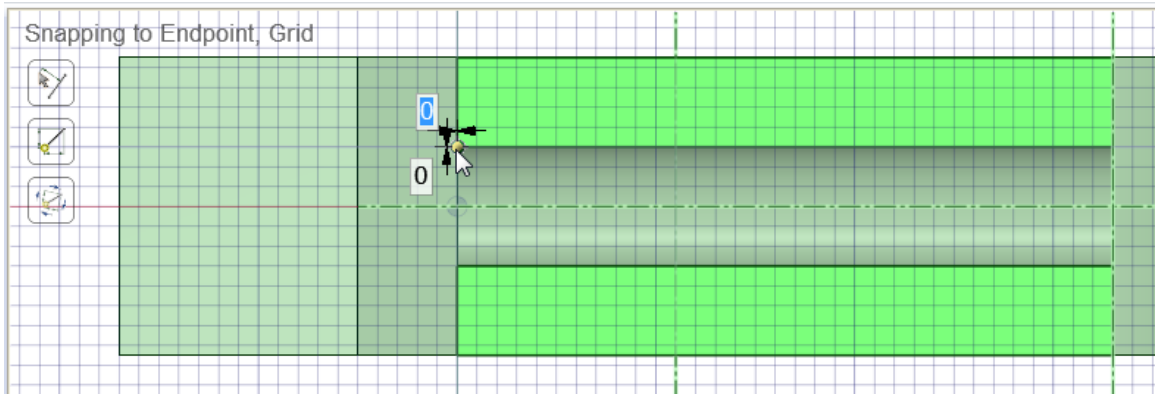
46. Bring the mouse to the center of the new hole, and a temporary axis will appear through the center of the hole. Click the axis to put the grid through the center of the hole.



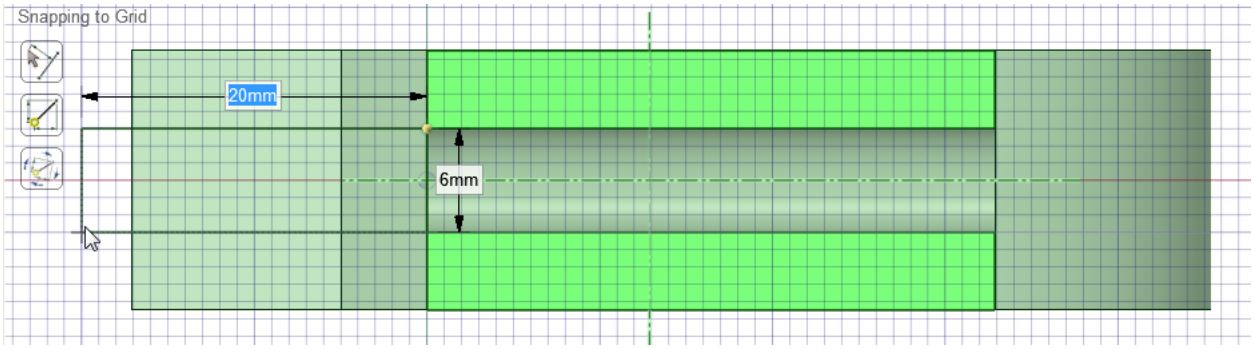
47. Click the Plan view button in the Grid-Guide buttons on the bottom of the window.



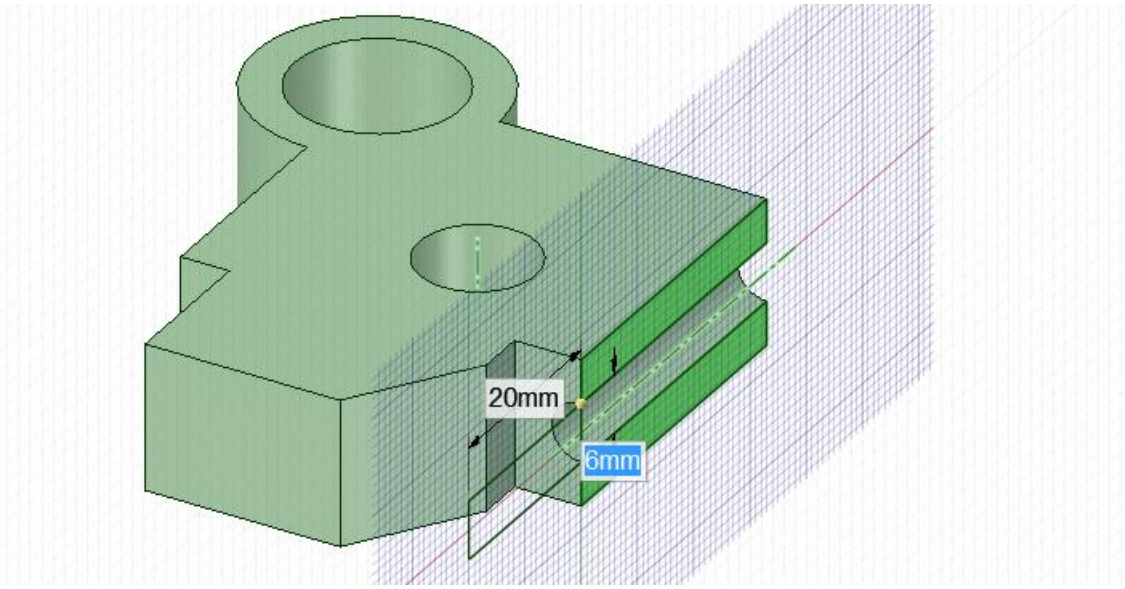
48. Start the rectangle, by clicking, on the upper left edge of the hole



49. Sketch a rectangle that is 6mm high and 20mm to the left.



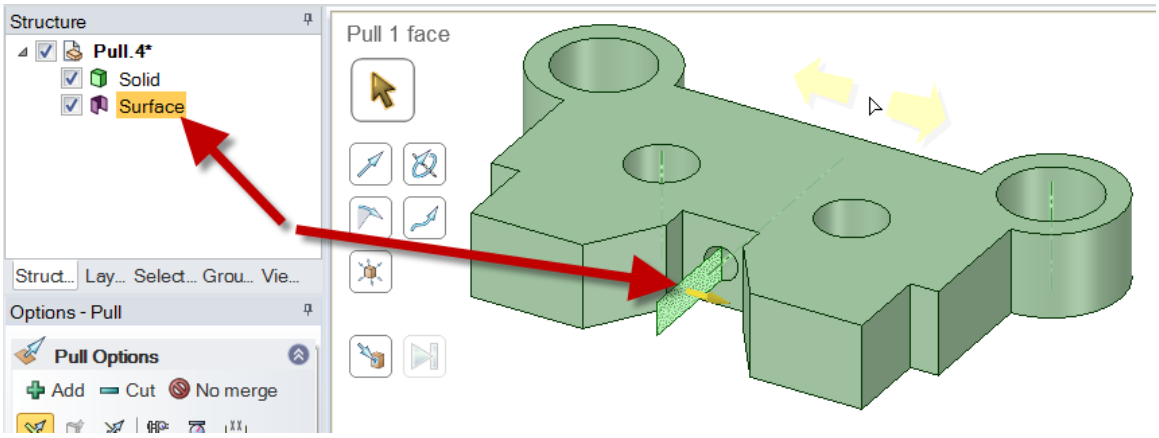
50. Press H or the home button to see the sketch in 3D



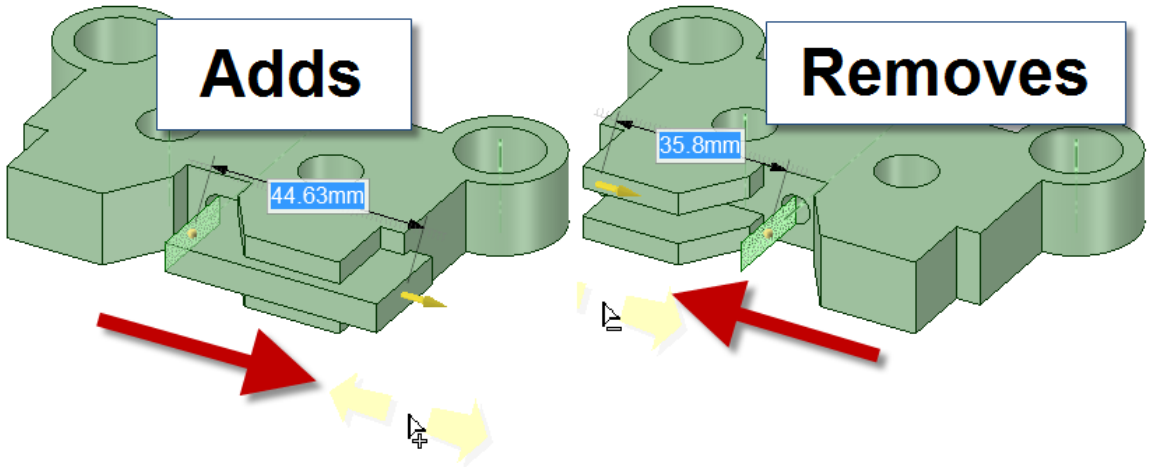
51. Press **P** or click the **Pull Tool**. SpaceClaim will switch to 3D mode.

    Notice the sketched rectangle turns into a surface and is listed in the Structure Tree.

52. Select the new surface, either by clicking the face of the surface in the design window, or by clicking the surface in the structure tree.



53. Drag the surface to the left and right.

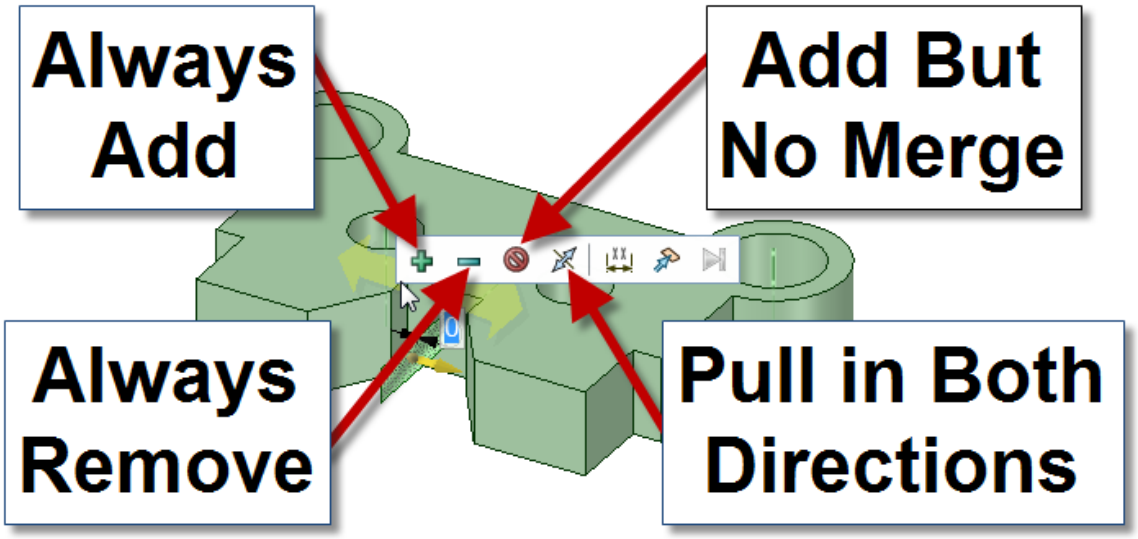


NOTICE when you drag in the direction of the yellow arrow, the Pull tool Adds.

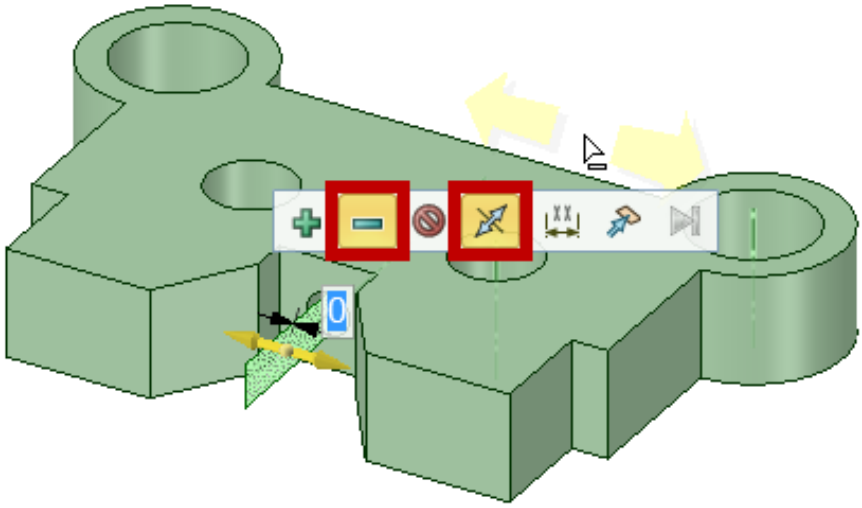
NOTICE when you drag in the opposite direction of the yellow arrow, the Pull tool Removes

54. While dragging, without letting go of the mouse button, press **ESCAPE** once. This will cancel the pull you are in. If you let go before hitting **ESC**, undo back to the surface (CTRL-Z).

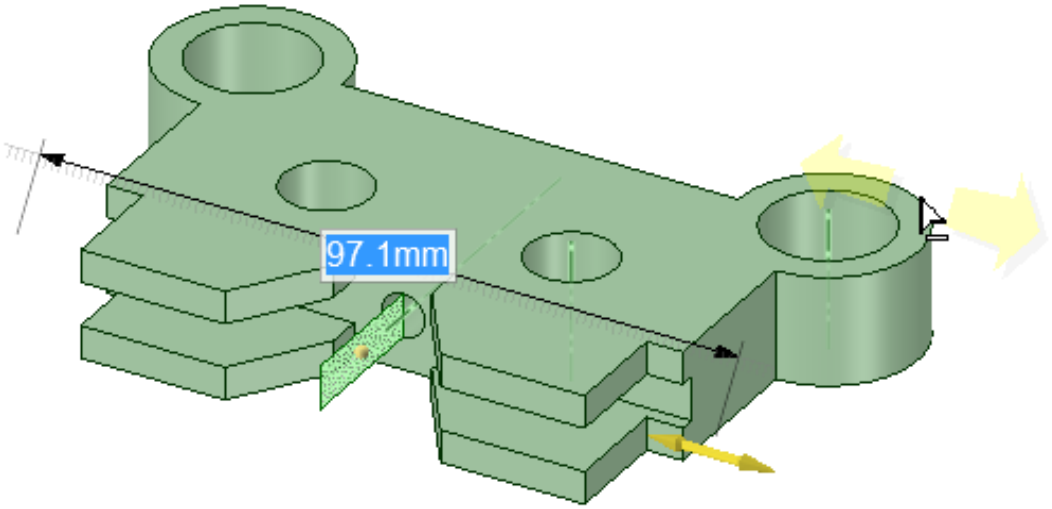
55. With the Pull tool on, click again on the face of the surface, and hover over the different buttons in the Mini-Toolbar



56. Click Add and pull both ways. **ESCAPE** or **Undo**
57. Click the face and click Remove and pull both ways, **ESCAPE** or **Undo**
58. Click No Merge, Pull and let go. Notice a 2nd solid in the structure tree. **Undo**.
59. Click the Face, click Remove and click Both Directions

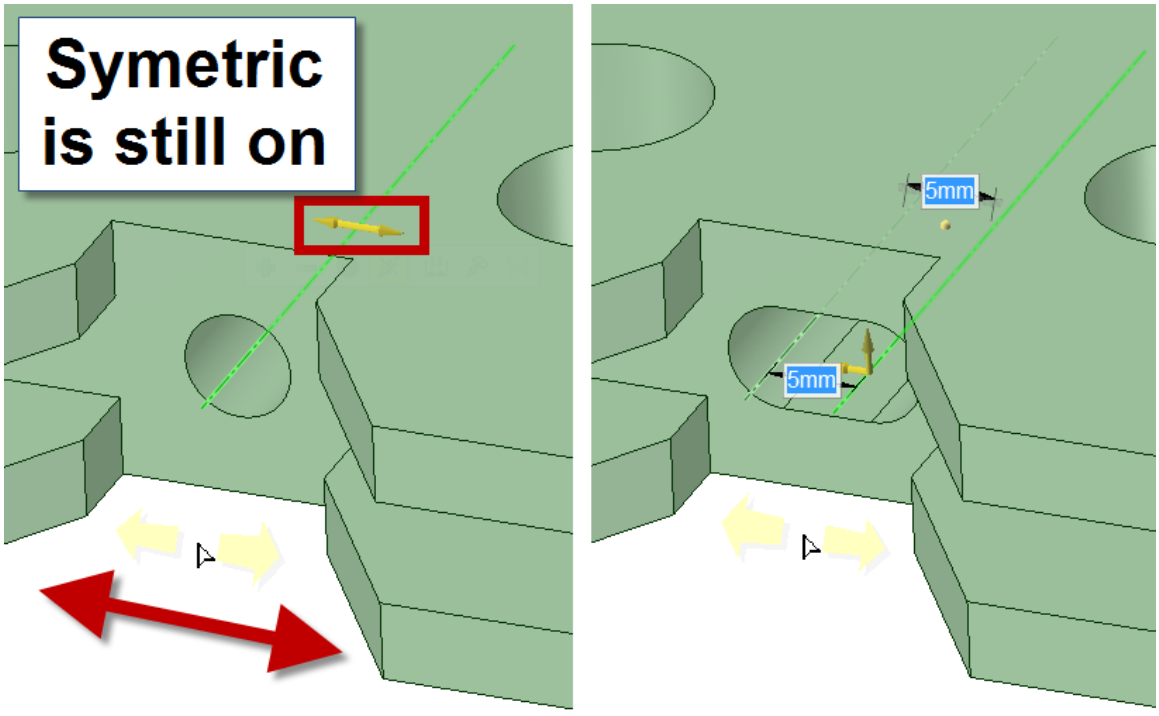


60. Drag left or right and cut all the way through



61. Select the axis through the hole, the symmetric options should still be on.

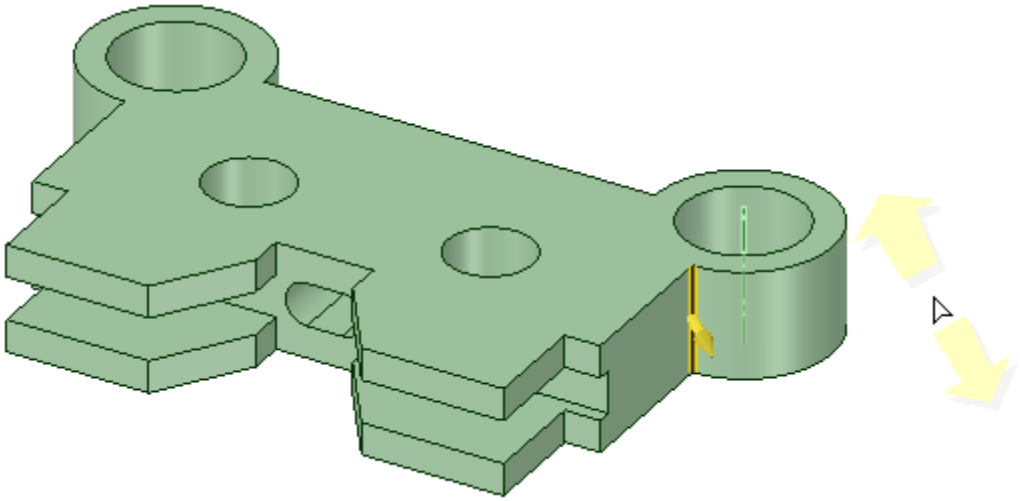
    Type in 5mm and press enter**. Pulling the axis of a hole Slots the hole**, in this case in both directions.



Pulling Edges: Rounds and Chamfers

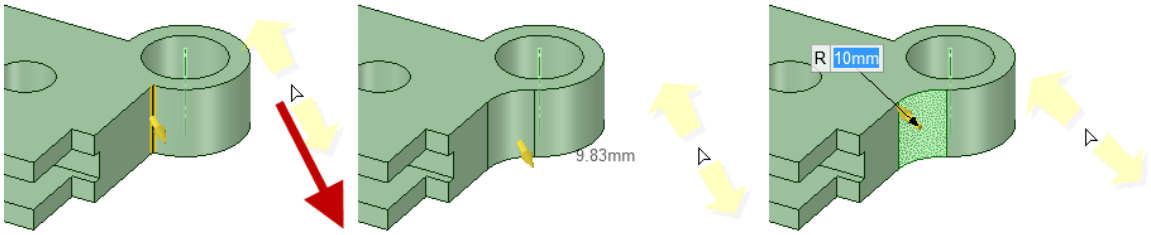
62. Return to a **home view** (H)

63. Select the concave edge below, with Pull on, and notice an arrow pointing away from the edge.



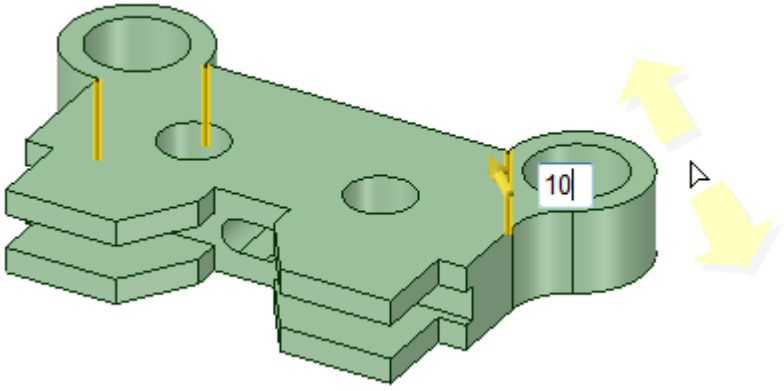
NOTICE: The arrow points away from this edge; when a concave edge is rounded material is added.

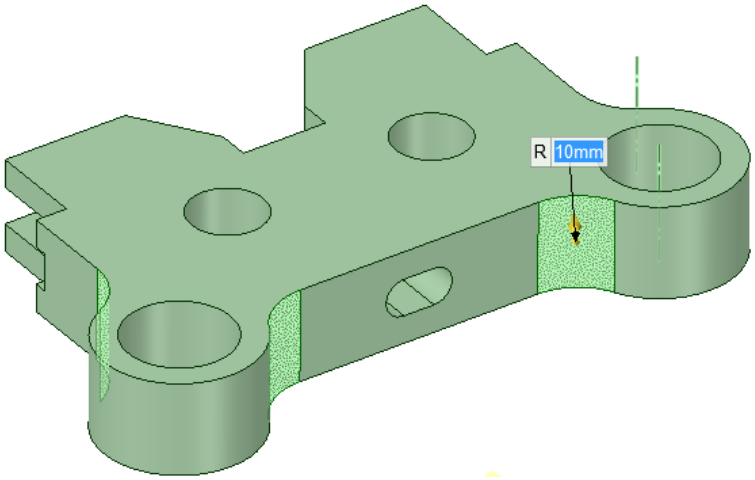
64. Drag down and to the right in the direction of the arrow - away from the solid - to add material and make a concave round.



65. Type in a value of 10mm to add a precise dimension.

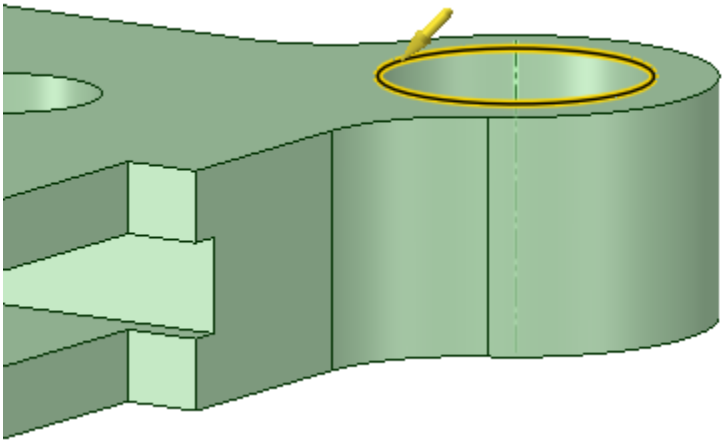
66. Select the 3 other edges show below, without dragging, type 10mm.





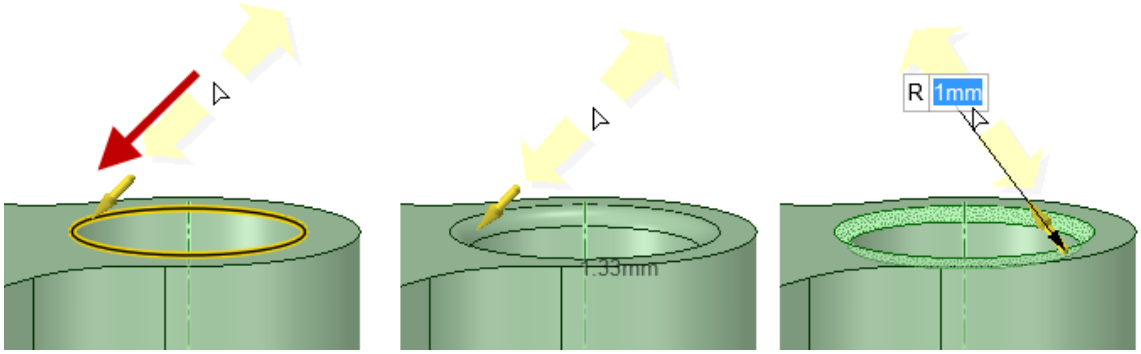
NOTICE: You can pull multiple edges to the same size without dragging, just by typing a number

67. Select the convex edge at the top of the hole shown.

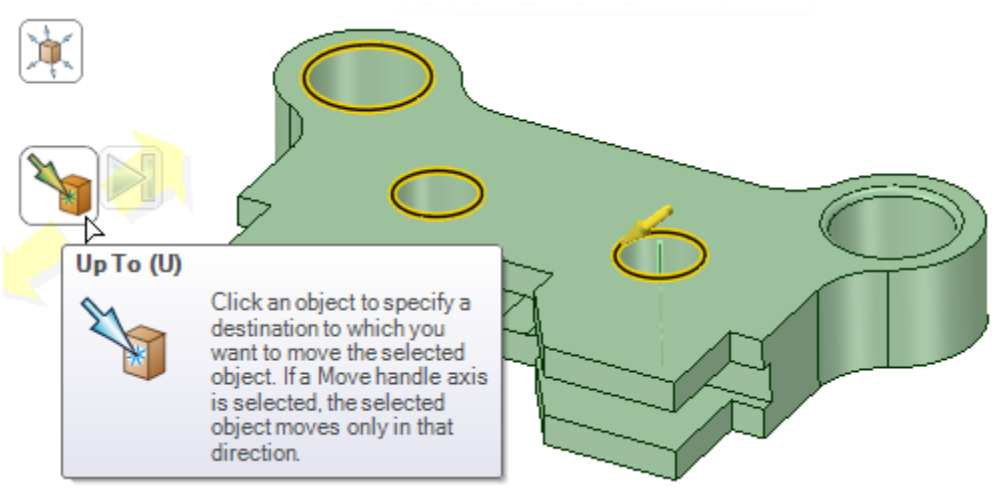


NOTICE: The arrow points towards the edge. When a convex edge is rounded, material is removed.

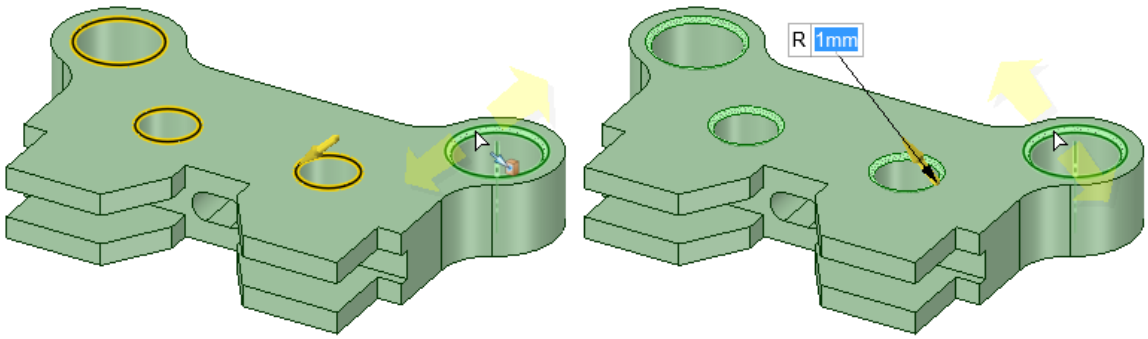
68. Drag down and to the left in the direction of the arrow - into the solid - to remove material and create a convex round. Type in a value of 1mm.



69. Select the 3 convex edges below, and click the **UpTo** button.

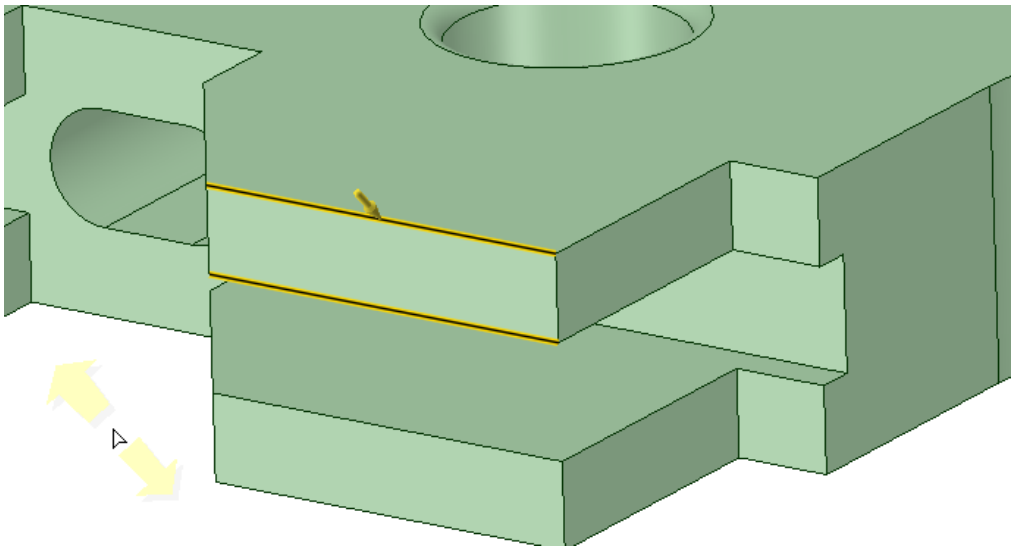


70. Click the first convex round shown below.



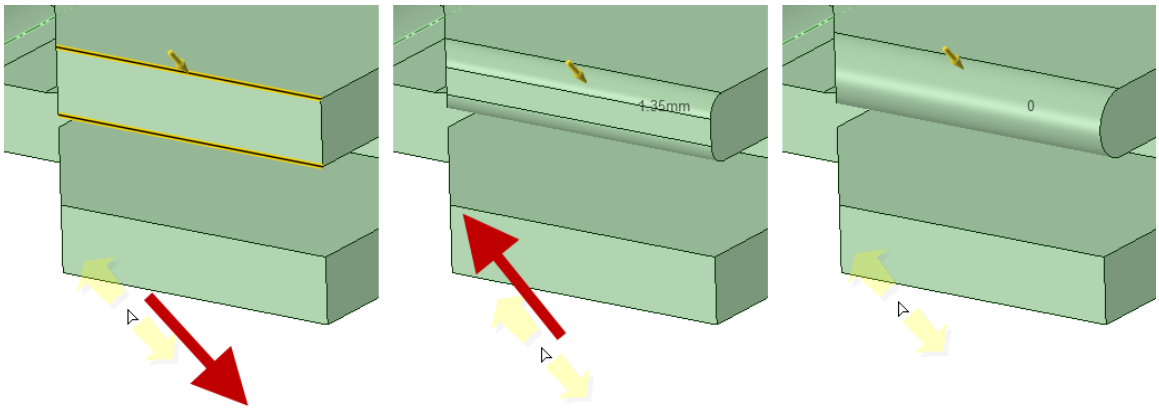
NOTICE: Pulling an edge up to a round, rounds that edge to the same size as the round.

71. To create a full round, **CTRL** select the 2 edges that will be turned into the Full Round.

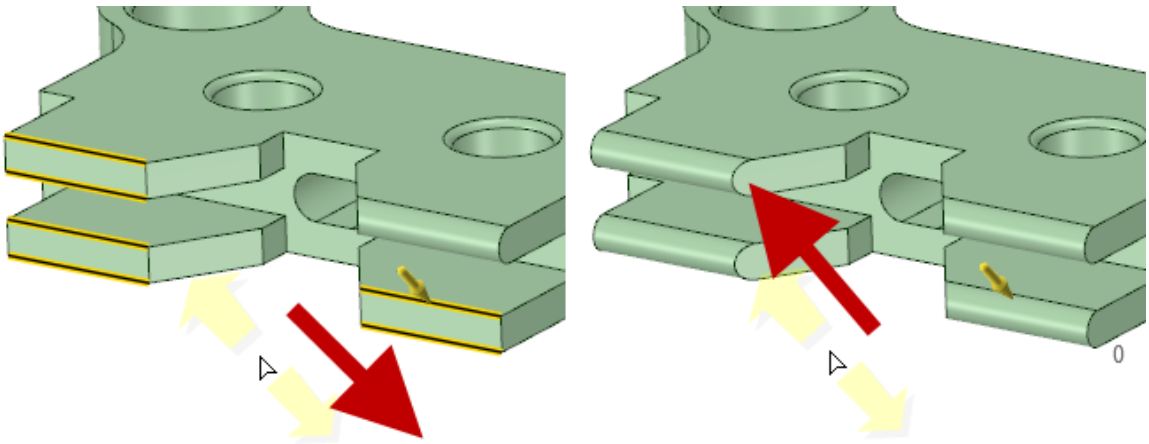


Full Rounds

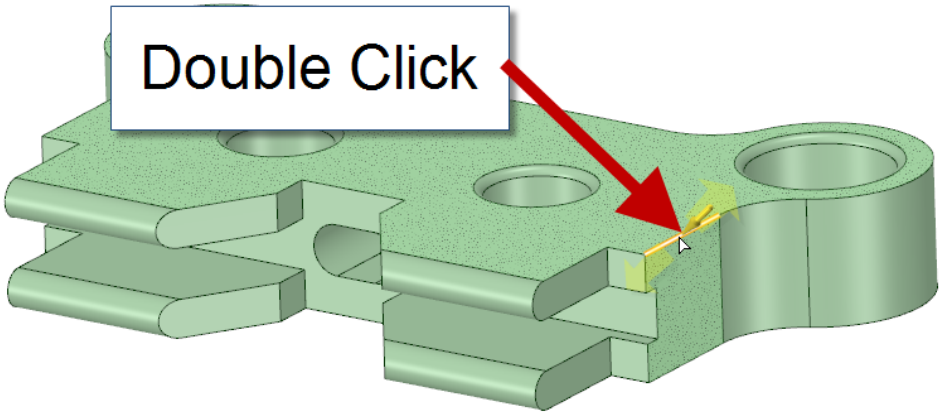
72. Start dragging into the model to start creating the rounds; at any point after seeing the rounds, drag in the opposite direction until the full round is created.

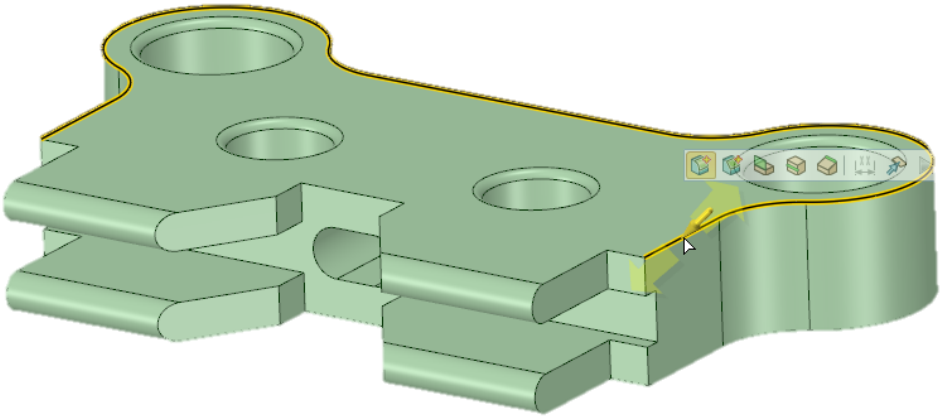


73. Select all 6 of the other similar edges. Pull in and out to create Multiple Full Rounds at the same time

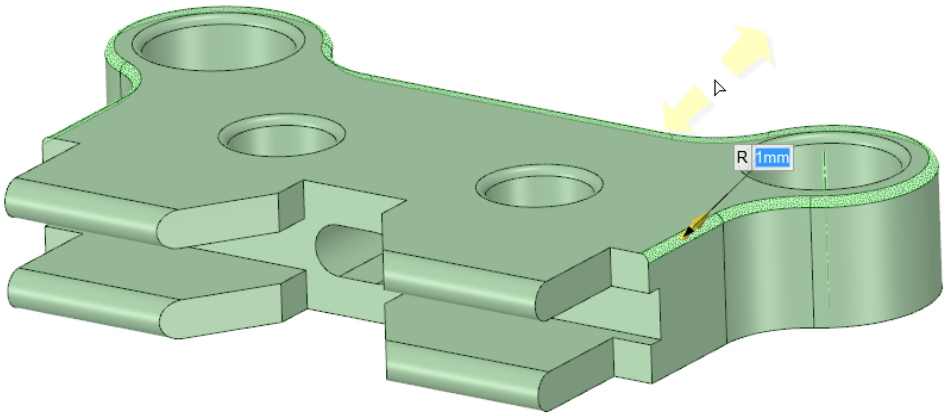


75. Double Click on the Edge below to select the tangent chain of edges.



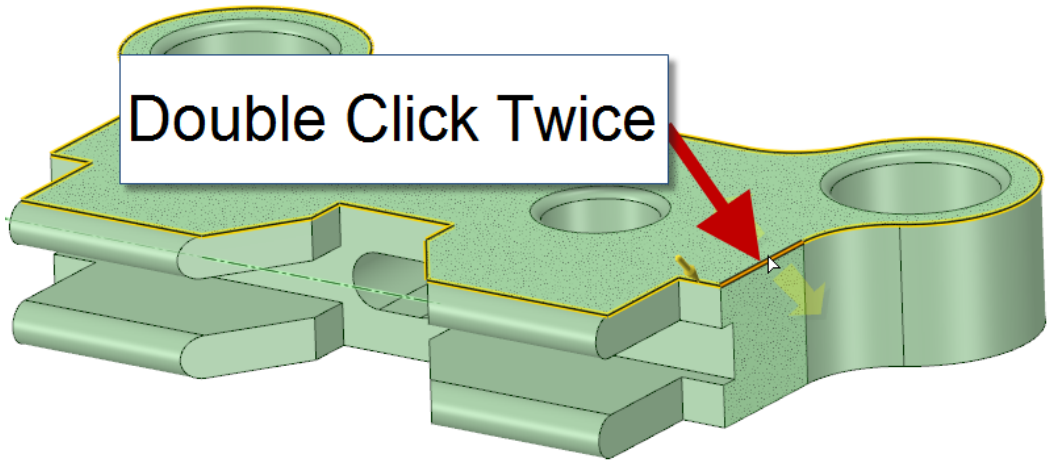


76. Pull the above edge chain to a Round of 1mm.



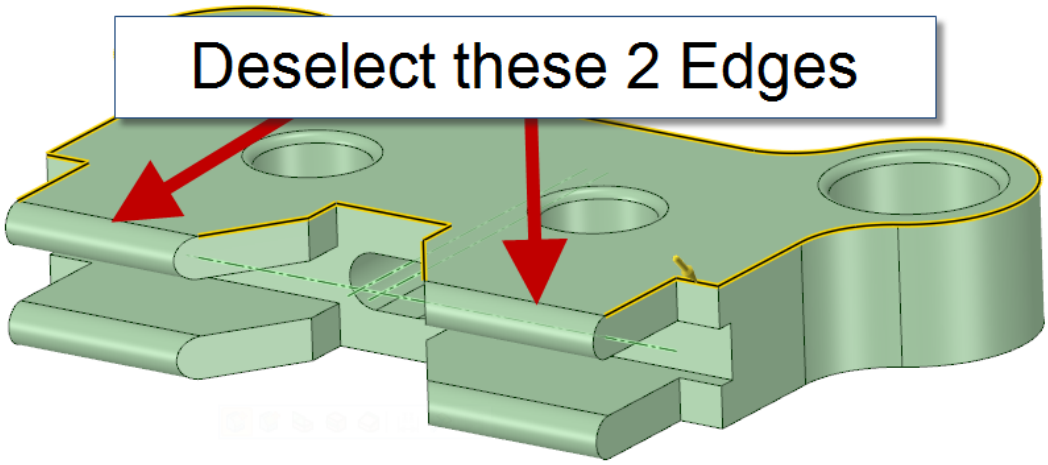
77. **Undo.** We actually want to round the entire top edge, not just the tangent loop. .

78. Double click the same edge, and then double click it again to get the full loop of edges.

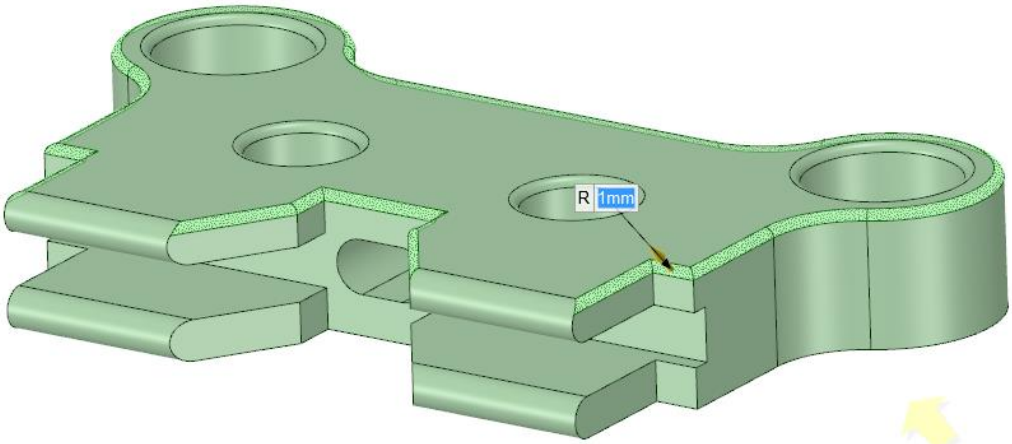


NOTICE: The first double click selects the tangent chain; the second double click selects a loop.

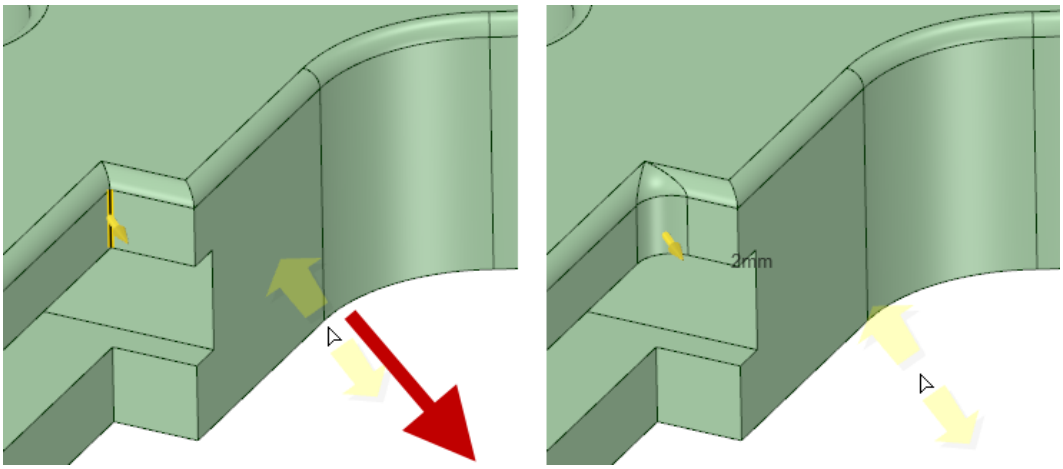
79. .Hold **CTRL** and click the 2 edges below to unselect them.



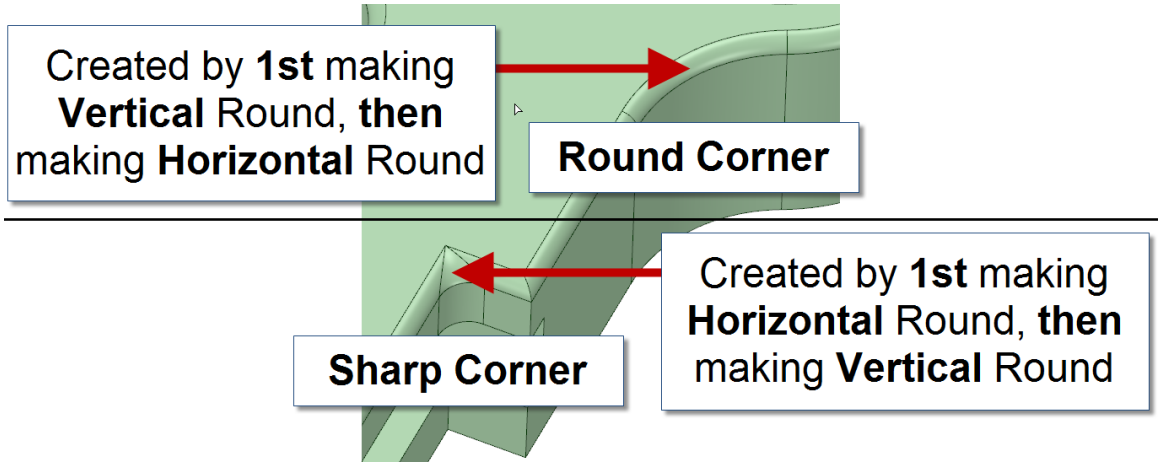
80. Pull these edges to a round size of 1mm.



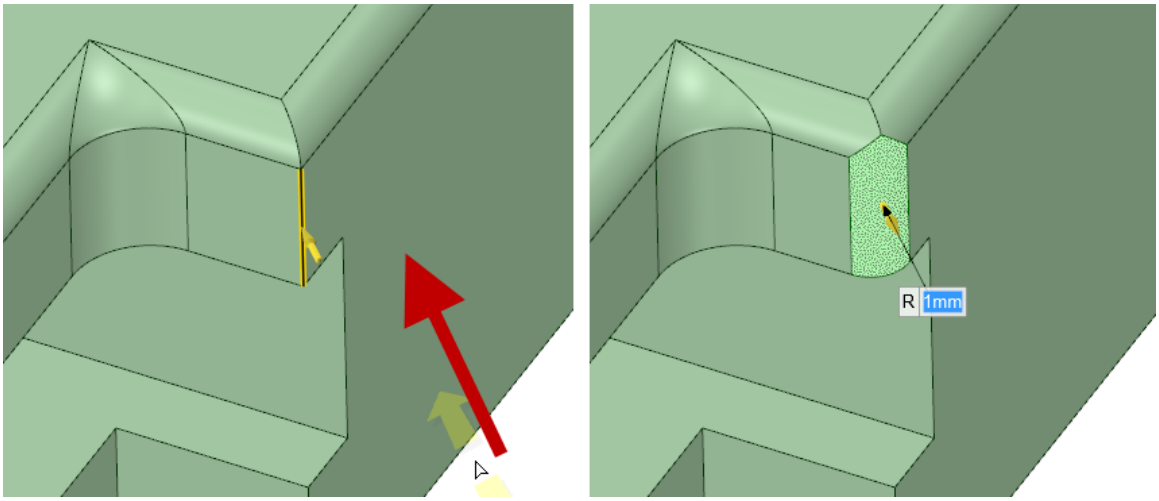
81. Select the vertical edge below on the front right side of the part, and pull it to 2mm.



NOTICE: It makes a difference with concave edges if you round the vertical or horizontal edges first.

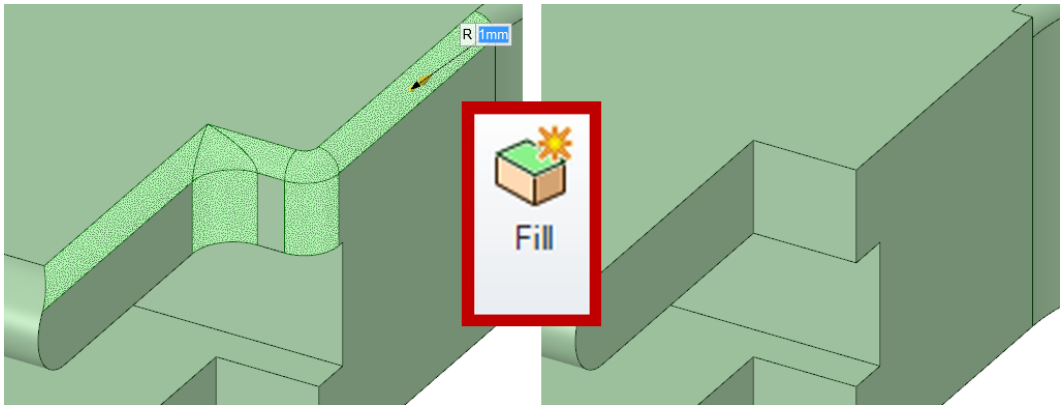


82. Select the edge below and round it to 1mm



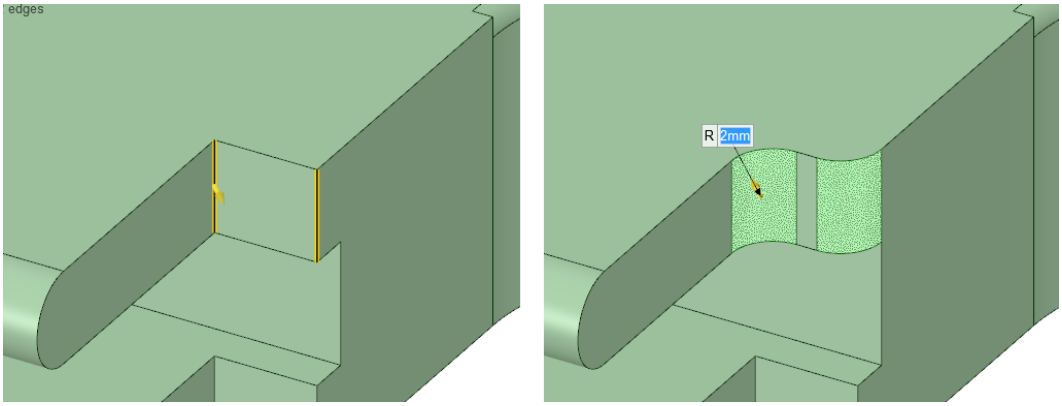
Both of the Round junctions shown in step 82 are not always desirable. Undo was previously used to remove the last round created, but sometimes the round was created many steps ago, or the round came from an imported file, and you can't undo

83. Select the 7 Faces highlighted below and click the Fill tool in the Ribbon Bar next to Pull.

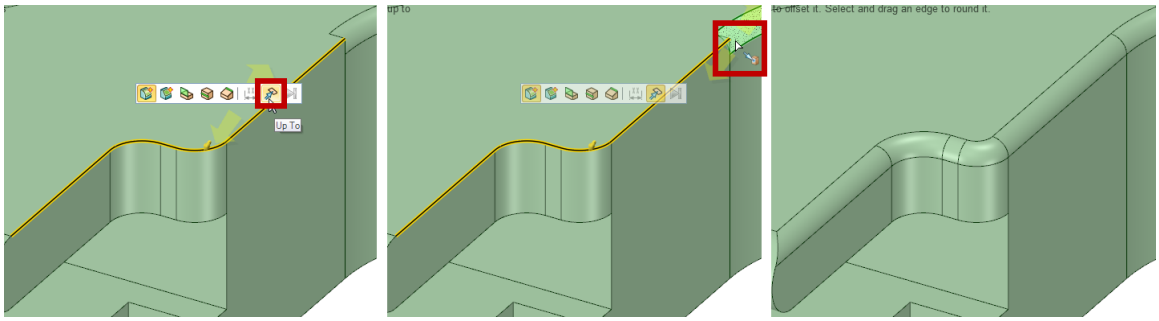


The Fill tool will remove whatever faces are selected, and extend the faces around them. The Fill Tool will be explored in more detailing in the following Fill Section.

84. Select the 2 edges shown below and Pull them to a 2mm round



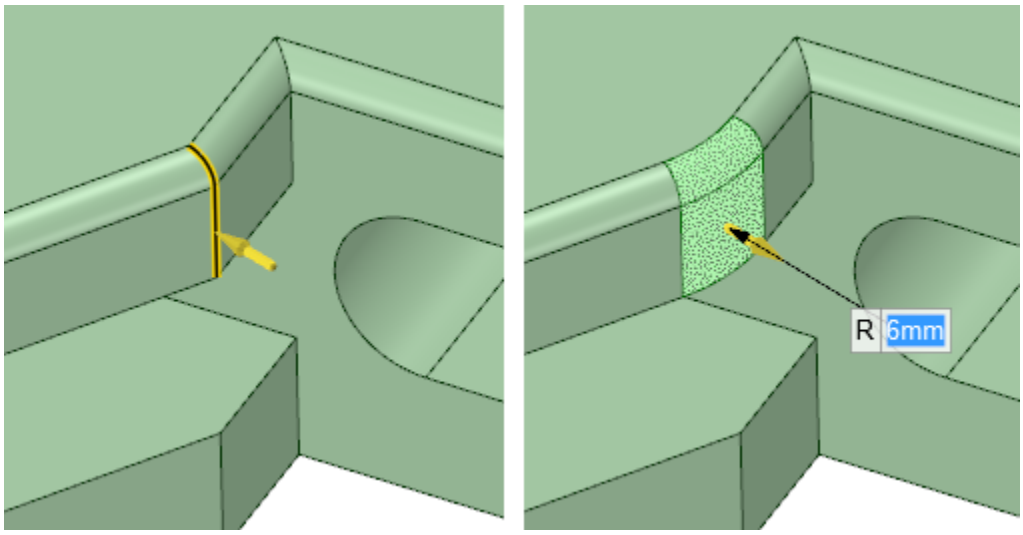
85. Select the Tangent Chain of Edges below, Click UpTo and select the round in the upper right to round the entire chain of edges UpTo the same value.



86. Zoom in on the front, center of the part.

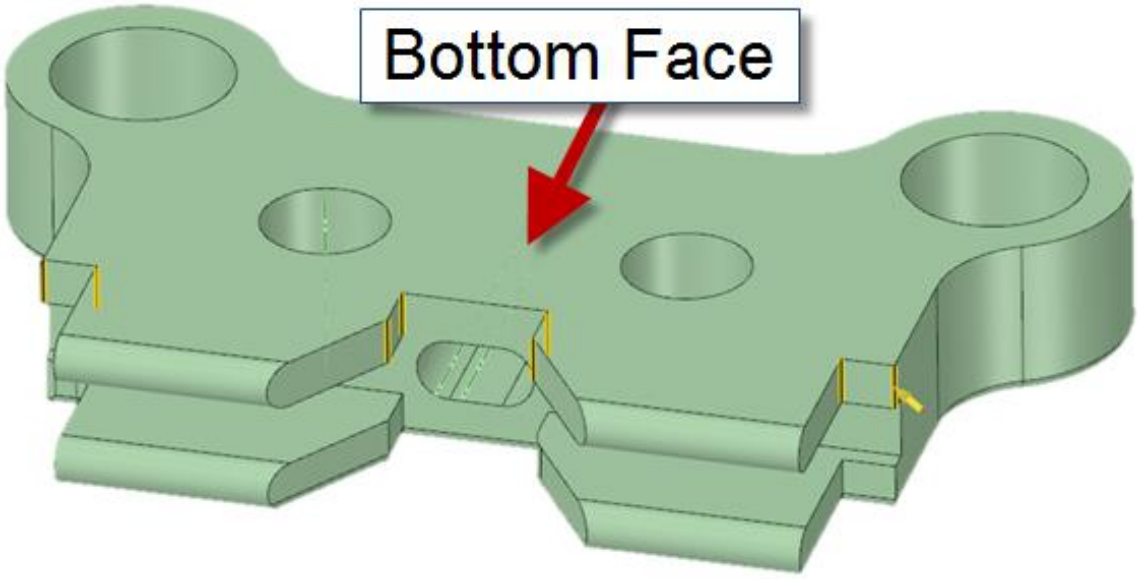
Typically you want to round the vertical edges and then the horizontal edges to get the most comment corner round types as shown above.

87. Double click on either edge below to select both edges, and pull to 6mm.

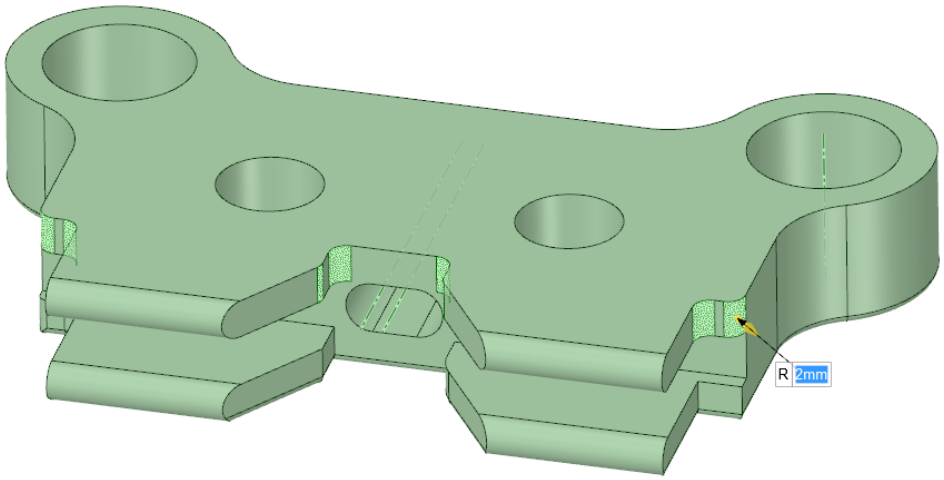


NOTICE: With convex (external) edges, you can create a smooth corner regardless if you round the horizontal or vertical edges first, but both edges above must be selected

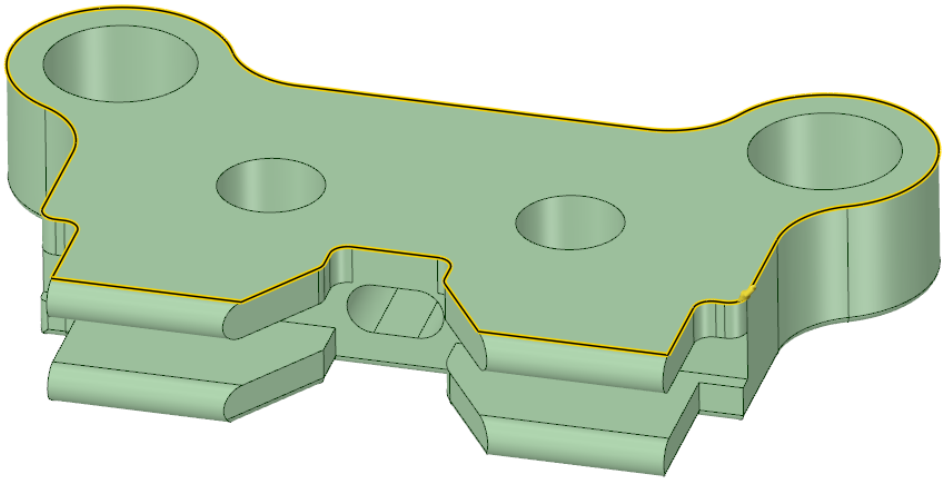
88. **Spin the model over**, so you are looking at the bottom of the part, select the edges below



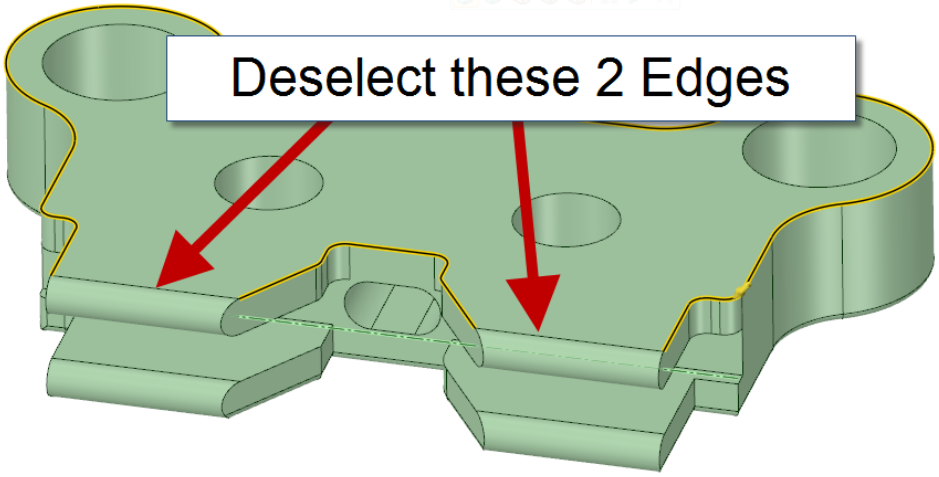
89. Round the below edges to 2mm.



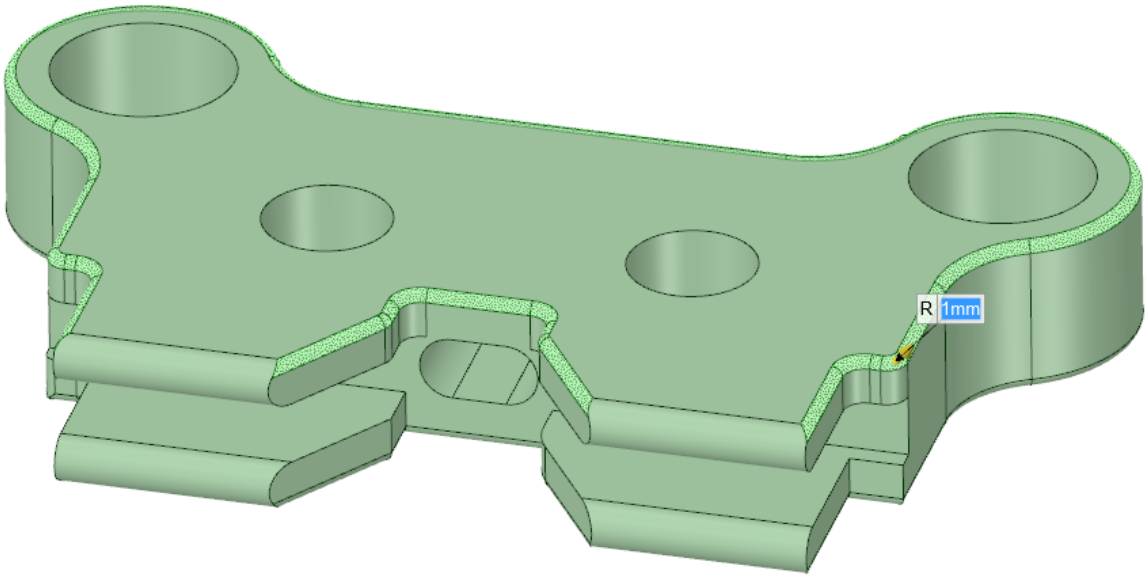
90. Double click on one of the top edges. If the edge loop below is not initially selected, Double click on the same edge again.



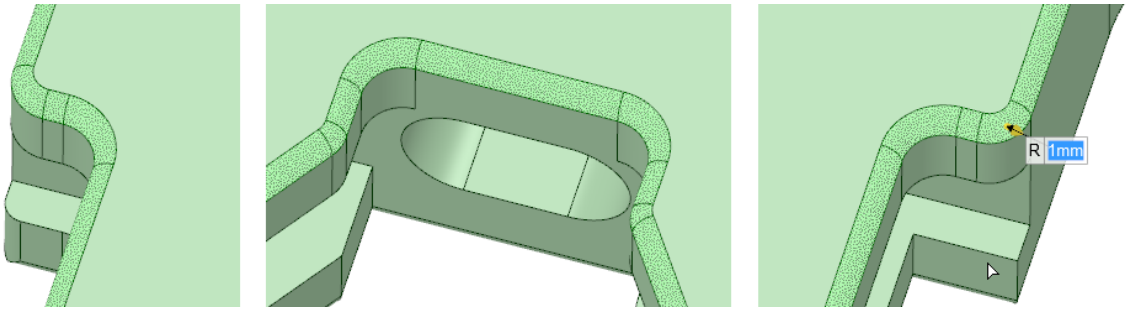
91. Deselect the same 2 edges as before.



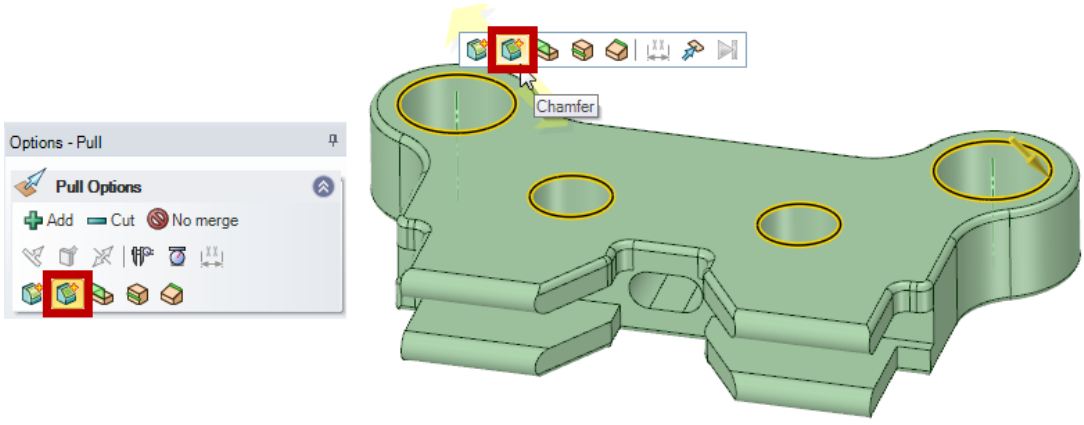
92. Pull the edges to a round of 1mm.



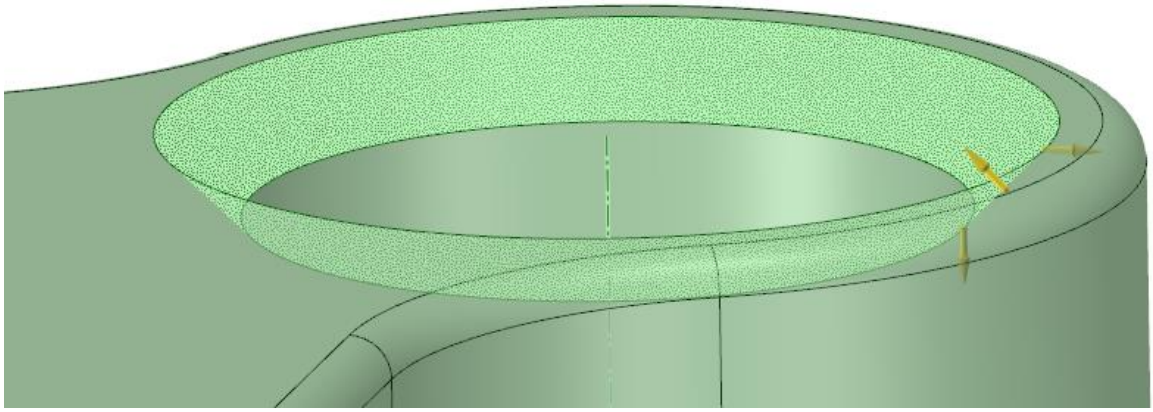
NOTICE the smooth corner rounds all around the upper edge



93. Select the 4 edges below, and from the **Mini-Toolbar** or Options on the left, click the **Chamfer** Button.

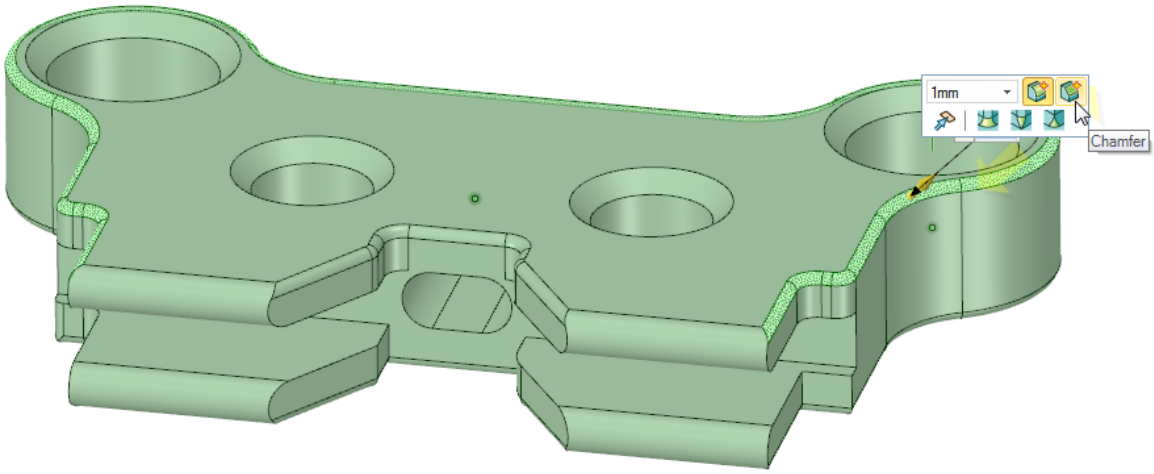


94. Chamfers are created just like rounds: After selecting the edge, and clicking the chamfer button, drag in the direction of the arrow and type in a value of 1mm.

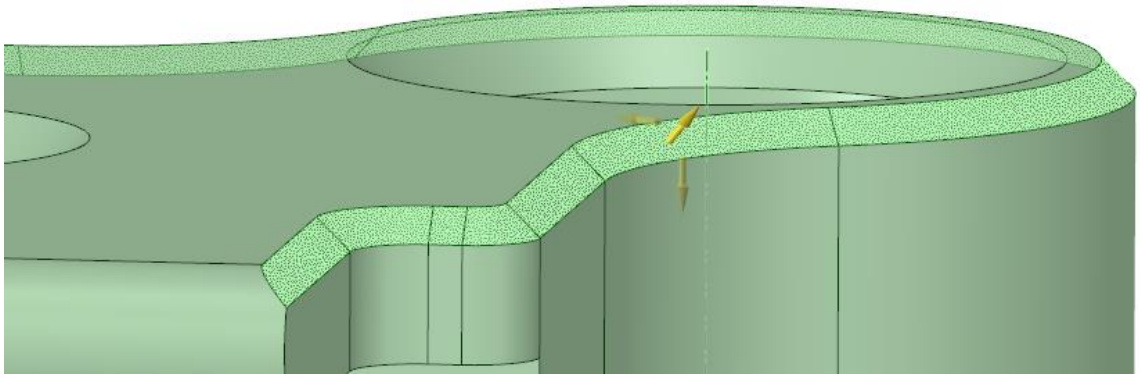


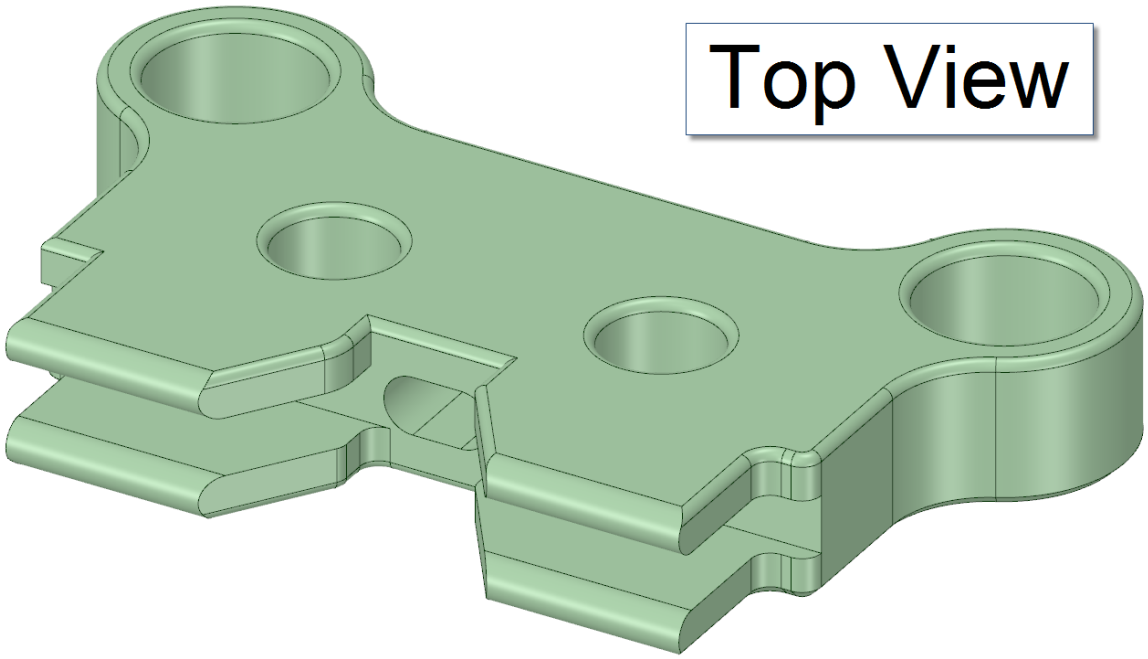
NOTICE: Instead of a rounded face, you will get a flat face at a 45degree angle.

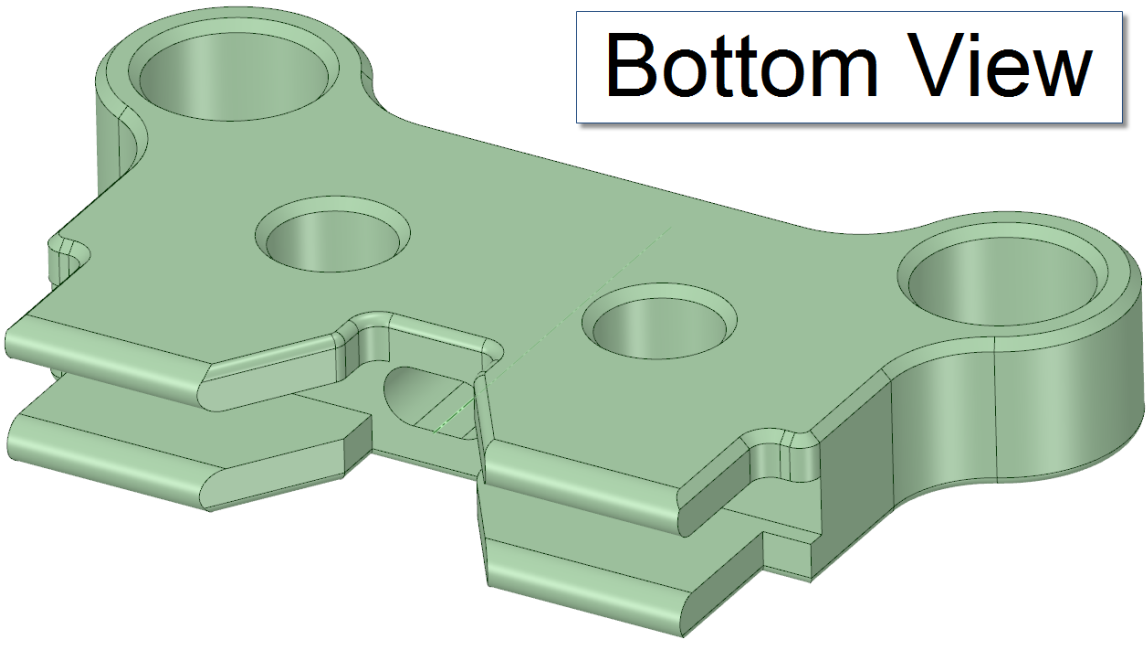
95. Double click one of the rounds on the top, and select **Chamfer** from the **Mini-Toolbar** or Options on the left middle side of the screen.



NOTICE how the rounds changed to chamfers



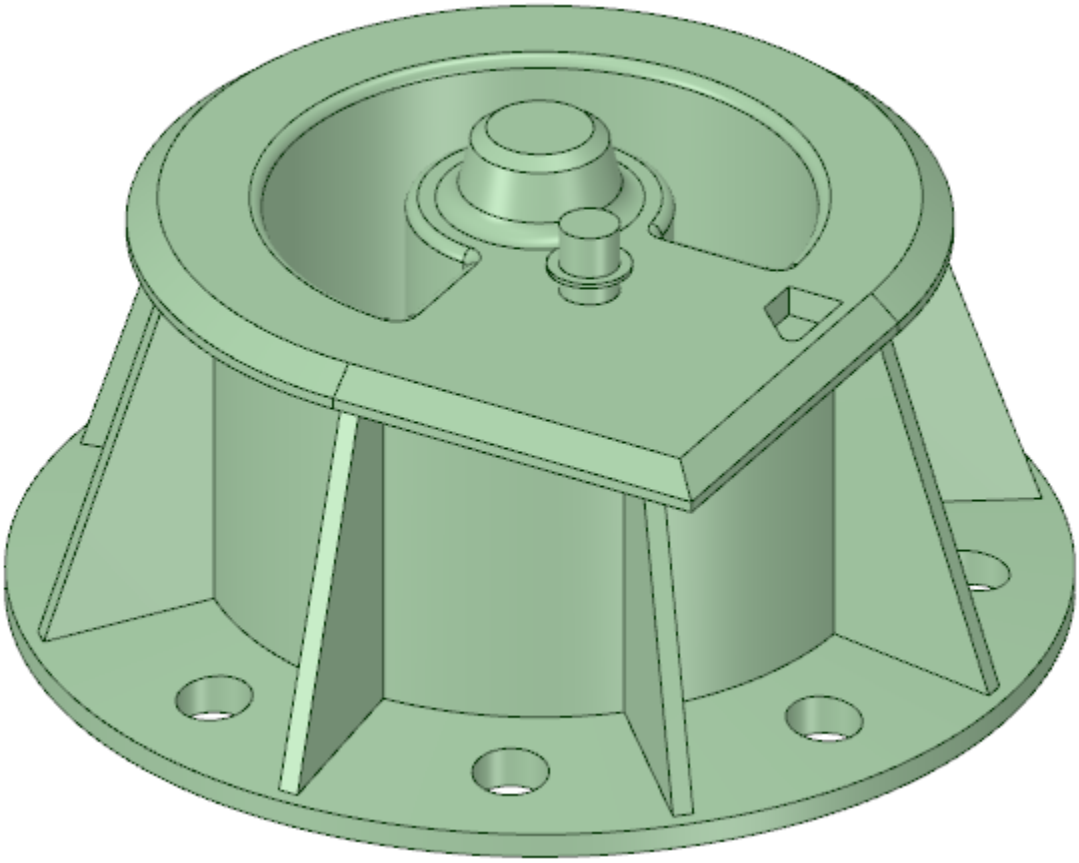

Top View


Bottom View

Move

In this section you will move things in 3D, which is similar to moving things in 2D like you did in the Sketch section. Move either translates or rotates a single object or multiple objects as a group. In some cases it will seem like Pull and Move tools do the same thing. In this section you will see when they perform the same action, and when they work differently. The Move tool can move points, curves, surfaces, solids, datums, faces, edges, components, assemblies and any combination of these.

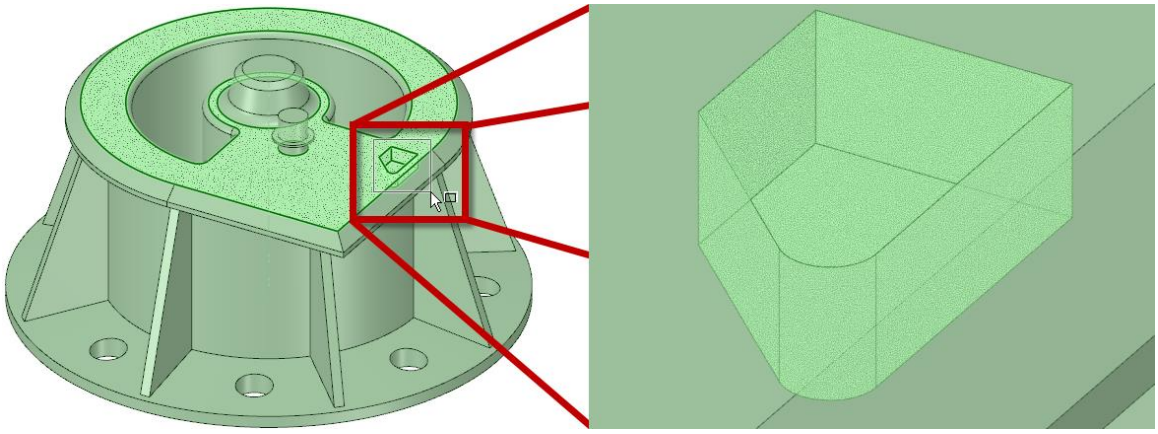
1. **File\Open**, Desktop\SpaceClaim_Basic_Training**\03_Basic_Move_2014.0** and open **Basic_Move_2014.0.scdoc**



NOTICE: the model for Move is a single solid.

We want to work on a specific section of the model which happens to be small.

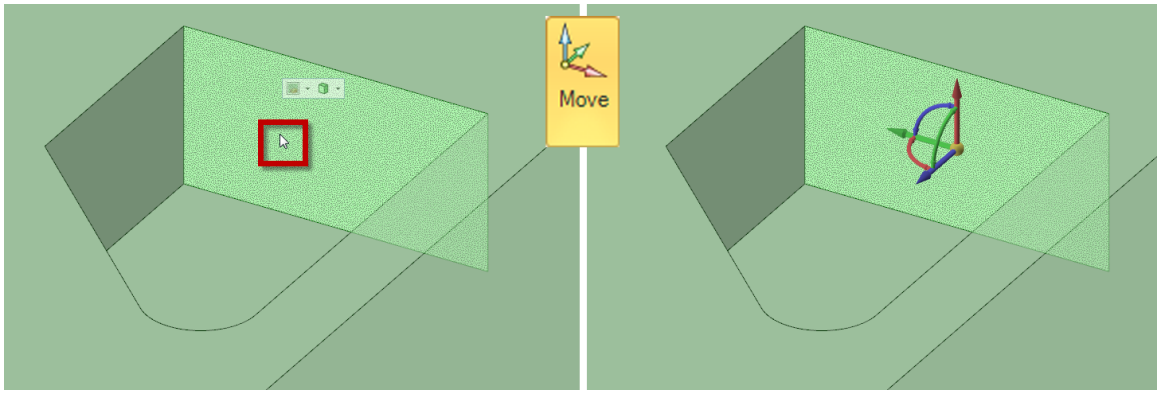
2. Zoom in on the region below by box selecting as shown and pressing **Z** on your keyboard.



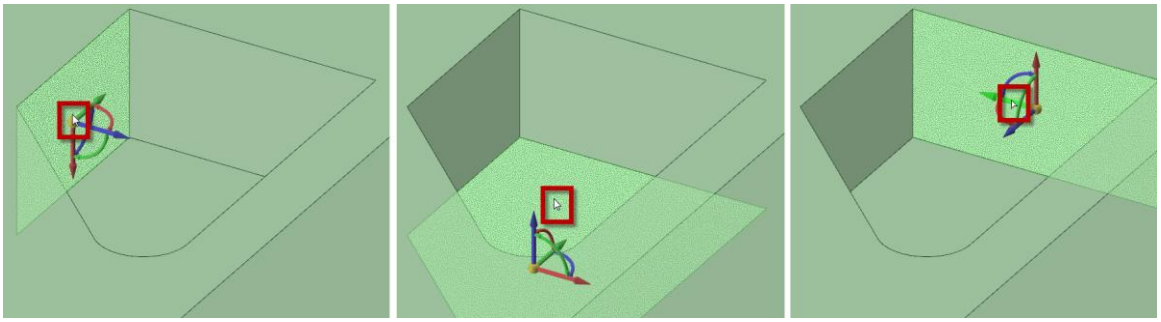
NOTICE: If you have anything selected and press **Z**, it will **Zoom** in on the Selection.

If you have nothing selected and press **Z**, it will **Zoom Extents**.

3. Click on the face indicated below
4. Turn on the **Move** tool in the middle of the **Design** tab.

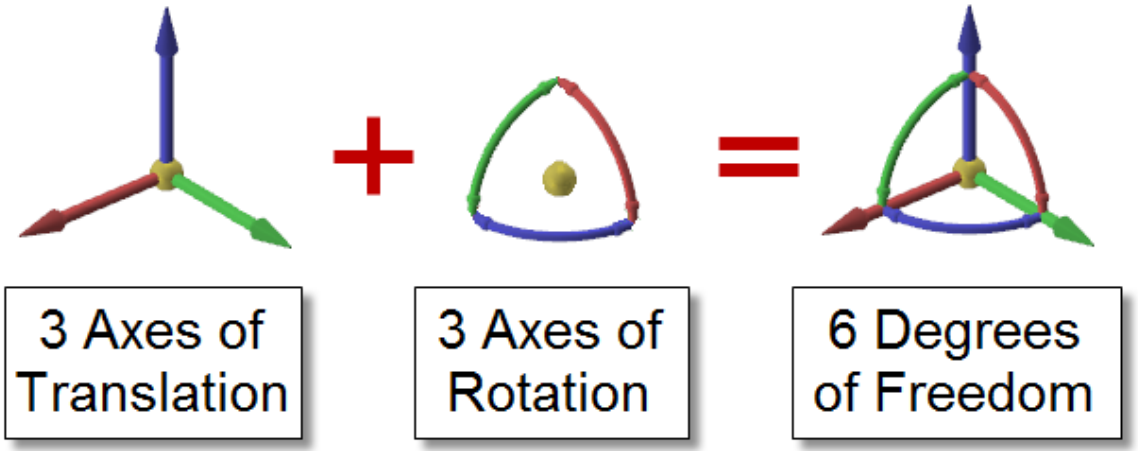


NOTICE: The Move handle appears at the center of the selected Face



NOTICE: You can select a face and turn on the tool, or turn on the tool and select a Face

The next step in moving something is to select the direction on the move handle.



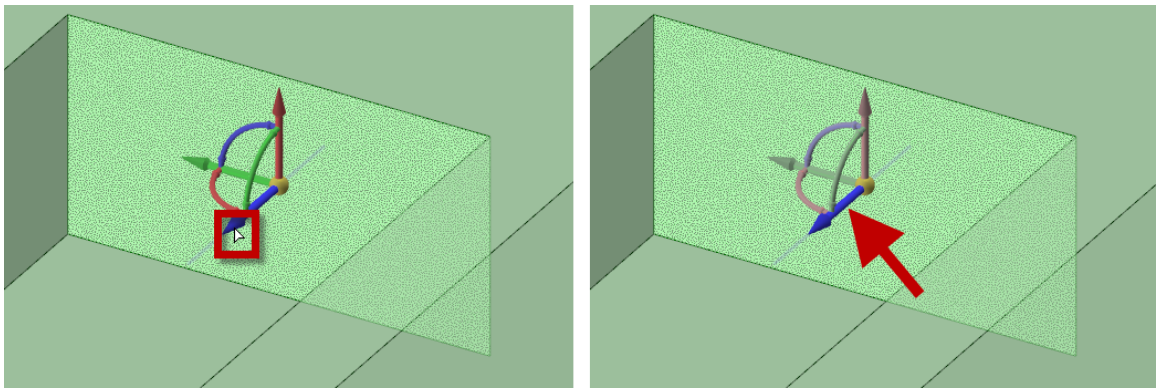
NOTICE: The move handle has three straight arrows known as **Translation Handles**, and 3 curved arrows known as **Rotation Handles**.

IMPORTANT: The direction of the arrows, left vs right, up vs down, is not important. The color of the arrows in not important either.

This document may reference an arrow color based on the images in this document. There are a number of reasons the blue arrow in an image may actually be a red arrow in your design.

Translating

5. Select the **Translation** handle that is perpendicular to the selected face, in this case blue.



NOTICE:

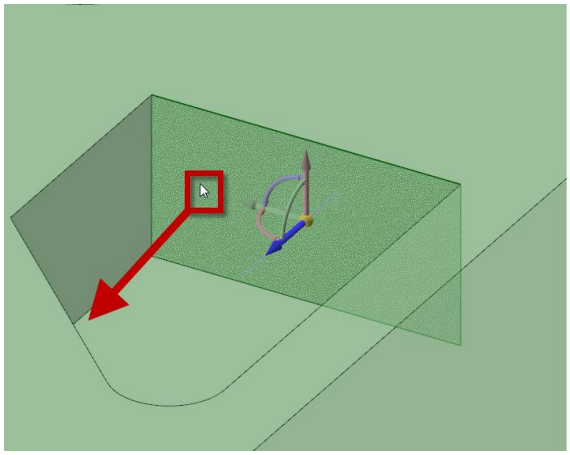
a. After clicking the Move Handle arrow, the other five arrow colors fade.

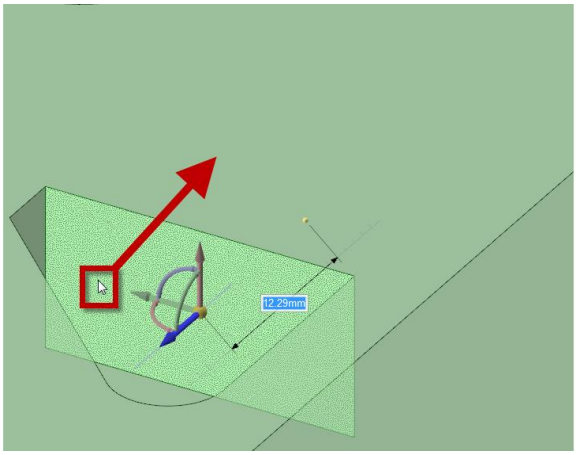
b. The bright colored arrow is the selected arrow for translation or rotation.

c. Accidental mouse movement when clicking can lead to incorrect arrow selection.

d. Always verify the correct move handle arrow has been selected.

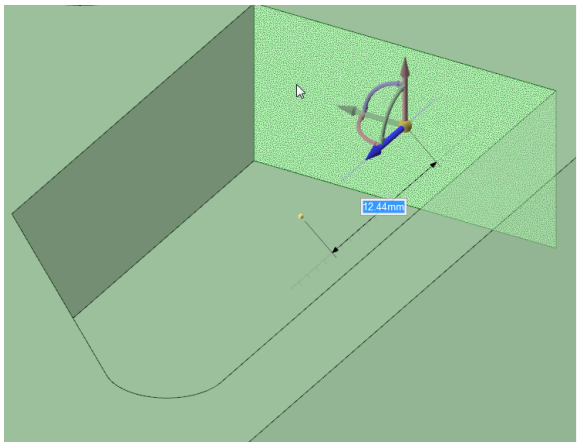
6. **Drag** anywhere in the design window to move the selected Face in the direction of the **Blue Arrow**.
7. **Drag** down and to the left to add material.
8. **Drag** up and to the right to remove material.

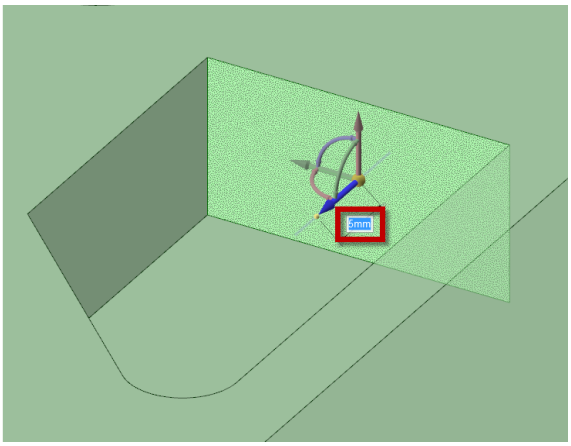




The goal is to move the face back 5mm, removing material.

9. **Drag** the face backward from the original location and type in 5 and press **Enter**.





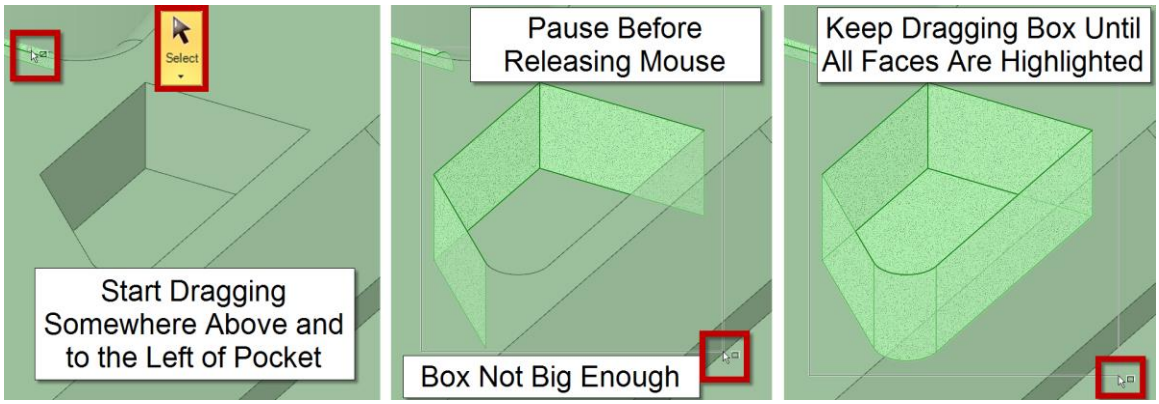
NOTICE: You can drag the face, let go, start dragging again, and each time you let go, you get a dimension from the original location of the face.

So what is the difference between Pull and Move?

When translating a planar face, there is no difference between Pulling and Moving.

Try pulling the face to see it is the same as Move.

10. Turn on the **Select** tool.
11. Box Select, from Left to Right, the entire pocket (6 Faces).



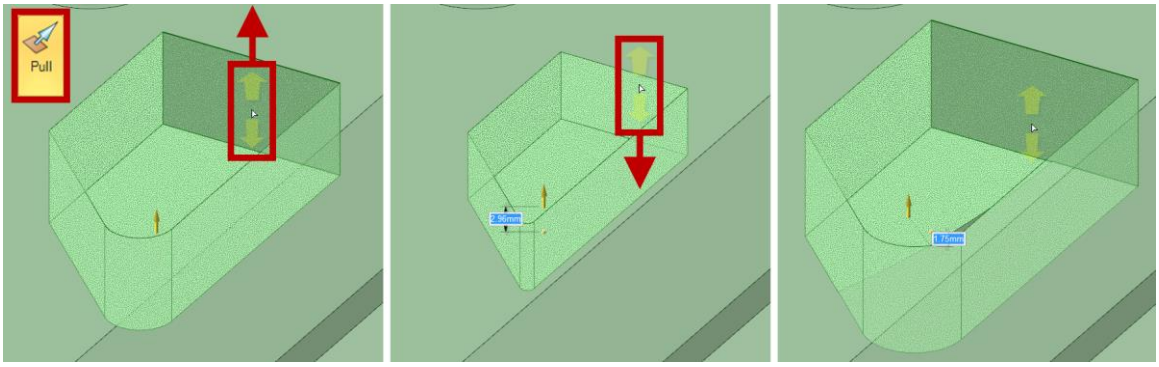
NOTICE: If the box isn't big enough, or if it isn't started far enough above or to the left of the pocket, the entire pocket won't be selected.

RECALL: You can pause while dragging in box select to preview what will be selected.

RECALL: The panel at the bottom right of the screen confirms what is selected.

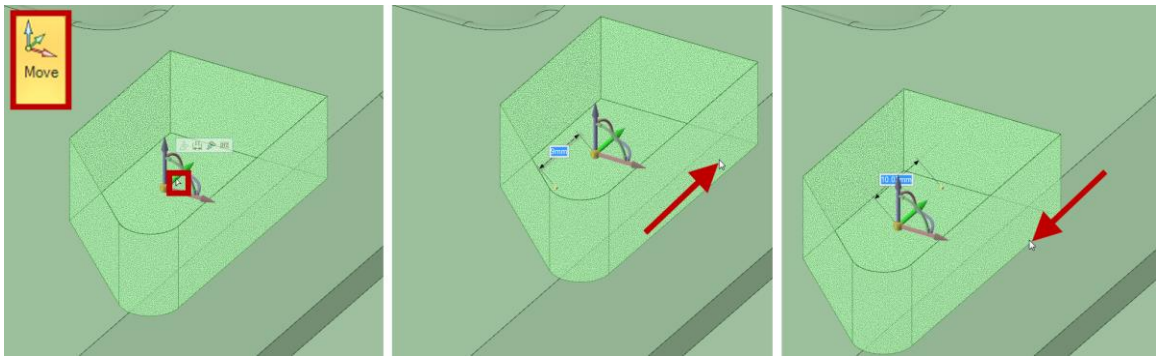
Let's refresh what happens when you pull a group of faces in different orientations.

12. With the 6 faces still selected, turn on the **Pull** tool and drag in the direction of the arrows.



RECALL: When you **Pull** multiple faces, each face **Offsets** in the direction normal/perpendicular to the face. Each face will offset in its own direction.

13. Either press **ESC** once while dragging to cancel the operation. If the command was completed, click **Undo (ctrl+Z)**.
14. Leave the 6 faces of the pocket selected (or re-box-select if not selected).
15. Turn on the Move tool.
16. Click the arrow corresponding to the length of the pocket, the green arrow in the image.
17. Drag anywhere in the design window in the direction of the arrow.



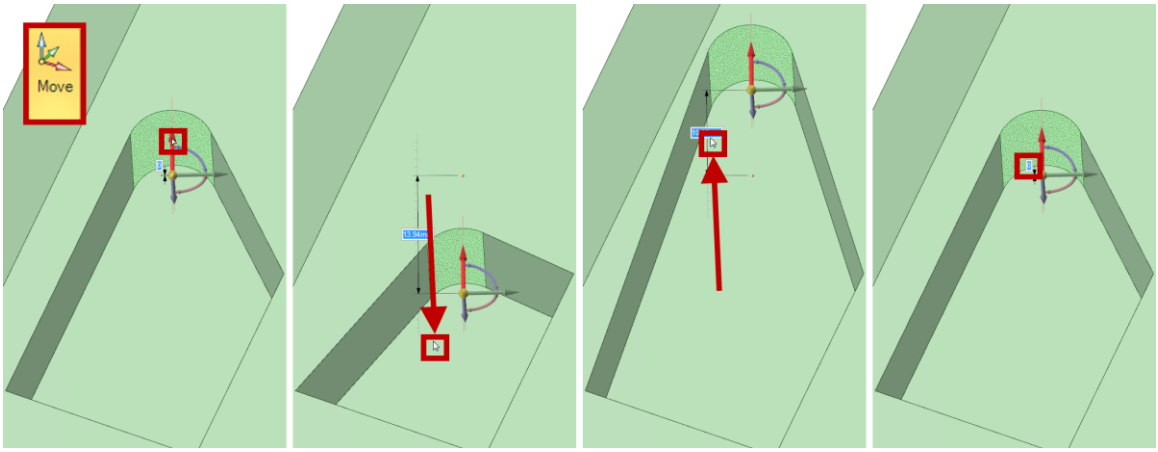
NOTICE: All 6 Faces Move or Translate in the same direction. The selected faces move together as a group, instead of being offset in independent directions with **Pull**.

Generally speaking, **Pull makes things bigger or smaller….Move changes the location**.

There are other differences between Pull and Move, even with a single face.

18. Rotate the model to the view below to clearly see the round on the pocket.
19. Turn on **Pull**, select the round and drag to pull it bigger and smaller.
20. Either press **ESC** mid-Pull or **Undo** after completing the operation.

21. With the round still selected, turn on the **Move** tool.
22. Click the arrow pointing at the round, red in the image below, and drag in the direction of the arrow.
23. Either **ESC** while dragging or **Undo** after completing the operation.

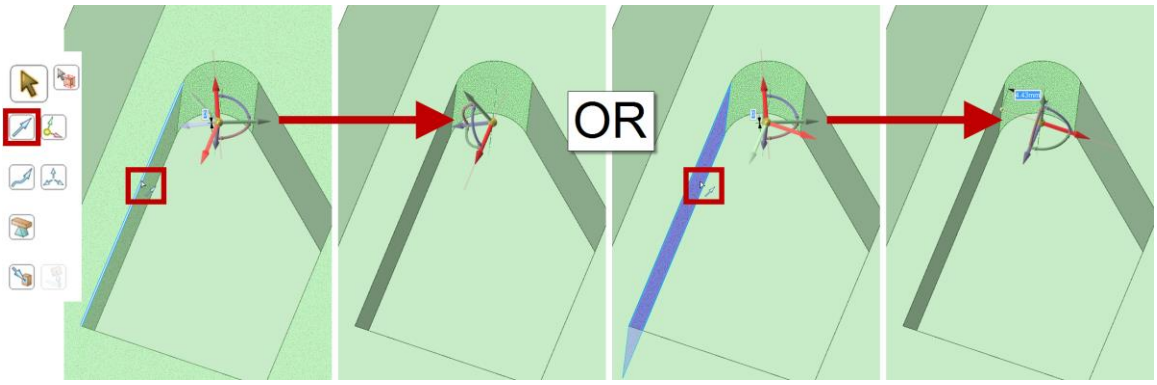


NOTICE: By default, the move tool snaps to the axis of the round and is directed at the center of the face of the round.

The default direction, in many circumstances may not be the direction you want to move something.

One of the **ToolGuide** buttons on the left is called **Move Direction** and lets you change the orientation of the move tool.

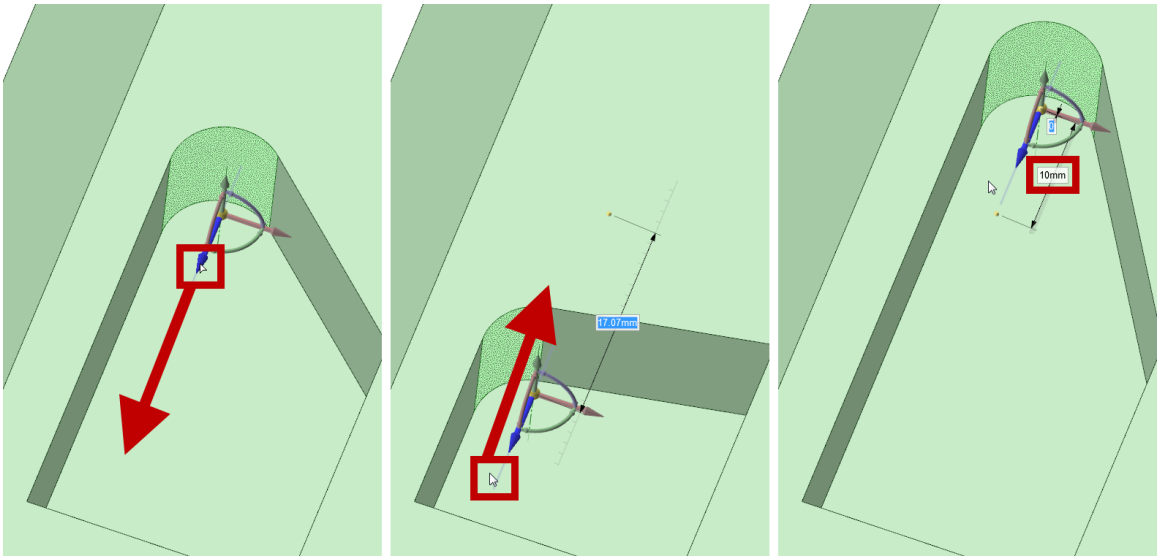
24. Click the **Move Direction ToolGuides**.
25. You can select the **Edge** or **Face** shown below. Hover over the edge and face to see a preview of how the Move tool will change directions after selecting them.



NOTICE: The shortcut for the direction ToolGuides, is **ALT + Click**.

NOTICE: After clicking the direction ToolGuides, edges and faces will prehighlight in **Blue**.

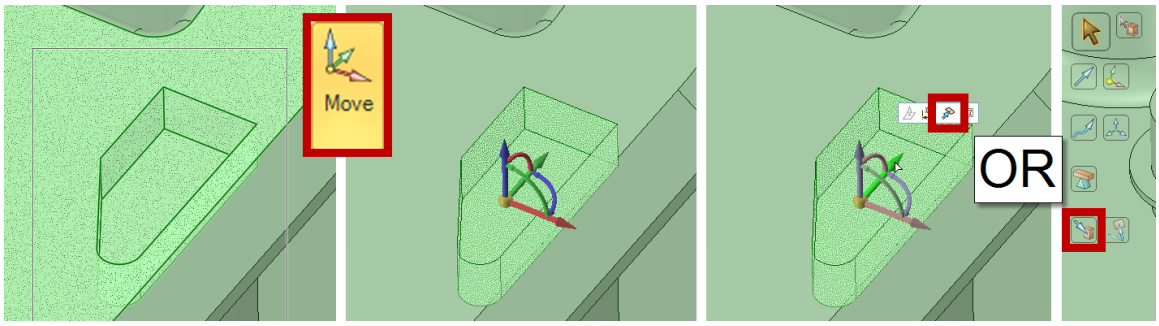
26. Click the **Move** handle arrow that's now aligned to the edge; blue in the picture below.
27. Now move the round in the direction of the edge.
28. Move the round upward and type in a value of 10.



NOTICE: The round moved along the selected direction, instead of the initial direction.

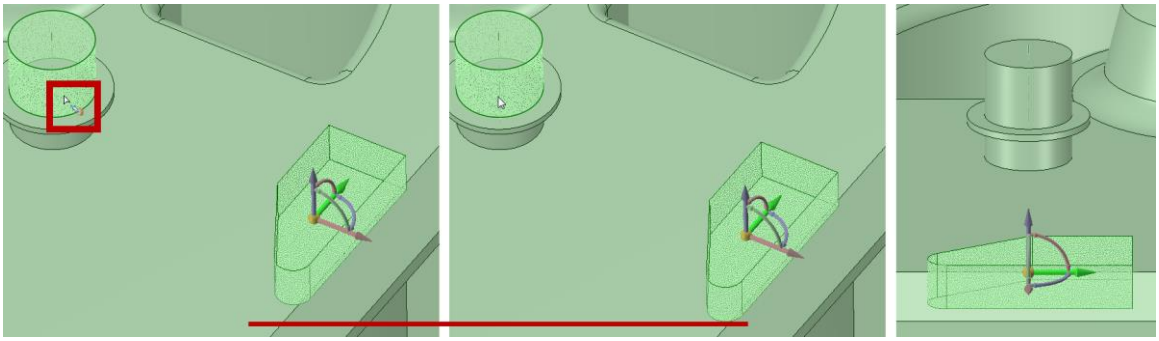
The direction ToolGuides (Keyboard shortcut **ALT + Click**) allows you to align the Move Handle parallel to an **Edge**, normal to a **Face**, or towards a **Vertex**.

29. Spin the model back to the original view.
30. Turn on the Select Tool and box select the pocket you've been editing.
31. Click the Green arrow in the image and click the **UpTo** button.



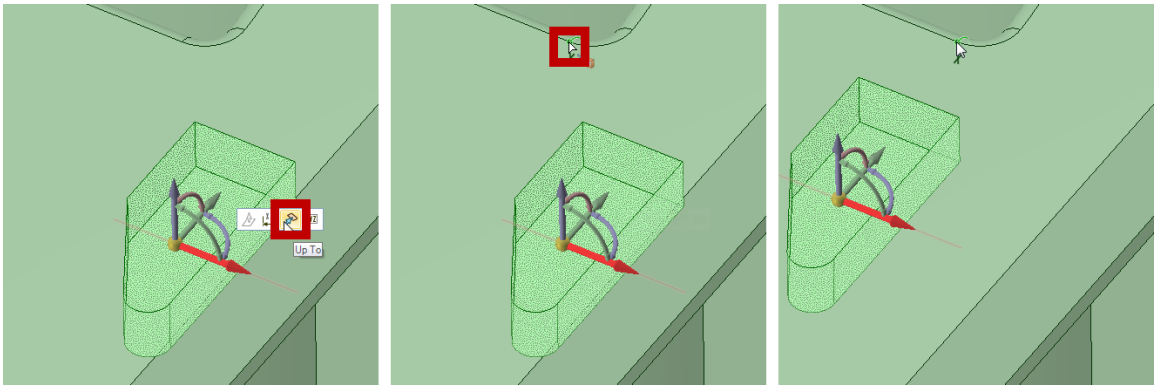
NOTICE: The **UpTo** button can be found in the **Mini Toolbar** and the **ToolGuides**.

32. Click one of the cylindrical faces on the protrusion in the image.



NOTICE: The pocket moved in the green direction so that the **Center** of the move handle is in line with the center of the cylindrical face.

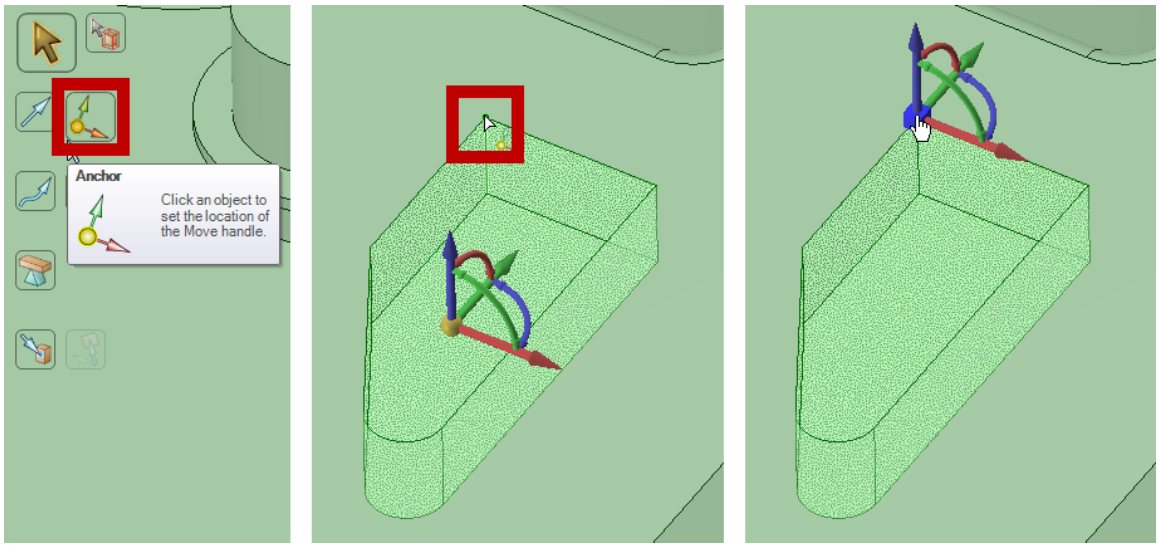
Click the red arrow in the image, click **UpTo**, and select the point shown.



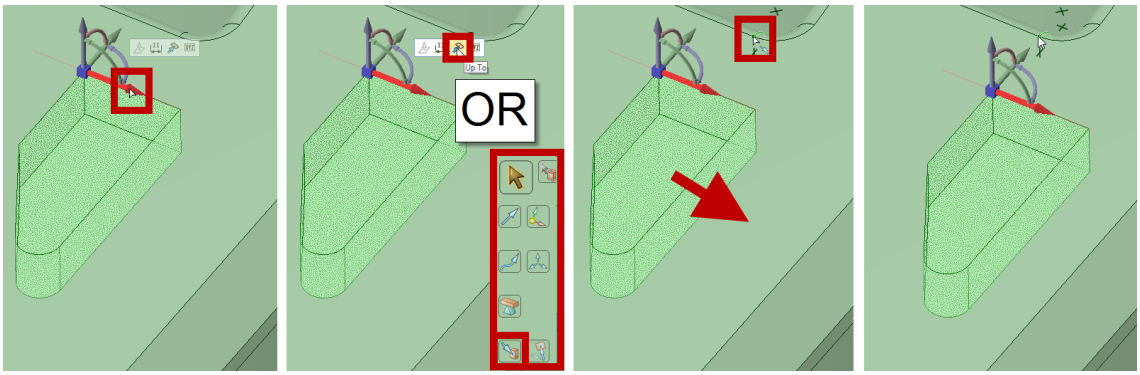
NOTICE: The move tool is centered to the selected faces. This is the default location of the move tool.

The center of the asymmetric pocket is not the important reference here.

33. Click the **Anchor ToolGuides** and select the vertex indicated below.

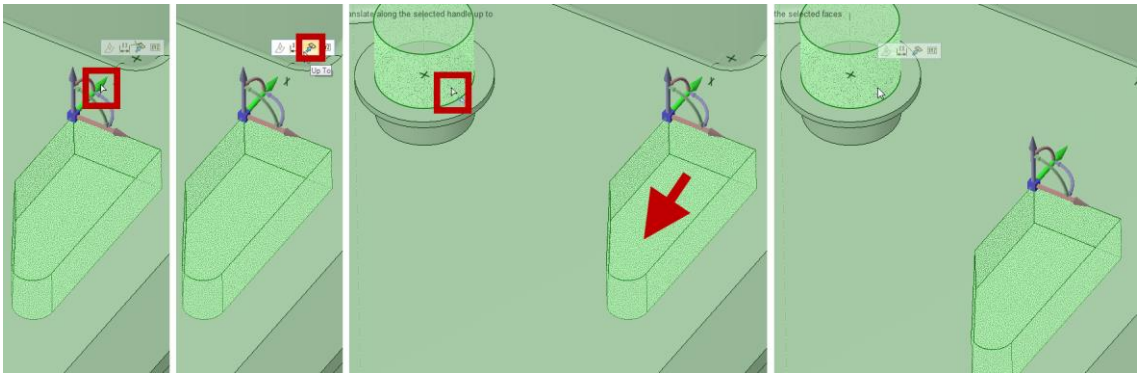


34. Click the red arrow of the move handle.
35. Click **UpTo** and click the same point in step 33.



NOTICE: You can use the **UpTo** button in the **Mini-Toolbar** that appears when you click the move handle, or the **UpTo** button in the **ToolGuides** on the left.

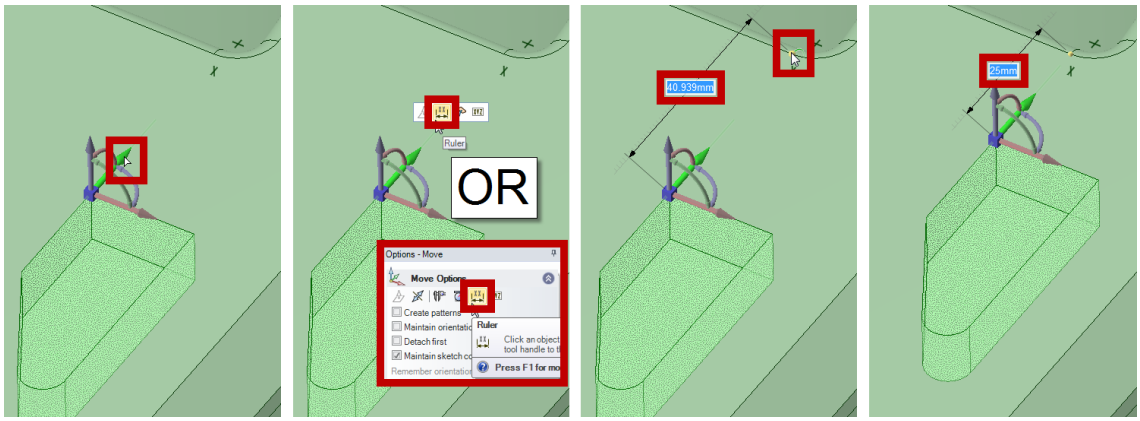
36. Click the green arrow of the move handle.
37. Click **UpTo** and click the cylindrical face you moved to in step 32.



In addition to moving UpTo, you may want to move something a distance from something else.

38. Click the green arrow, and either from the **Mini-Toolbar**, or the **Options Panel** on the right, click **Ruler**.

39. Click on the point shown to attach the ruler dimension to it, and type in a value of 25.



NOTICE: The entire pocket moved, so that the vertex the Move Handle is 25 mm from referenced location.

40. Click on the red arrow, click **Ruler**, and then click the edge shown in the image, and type in 25.

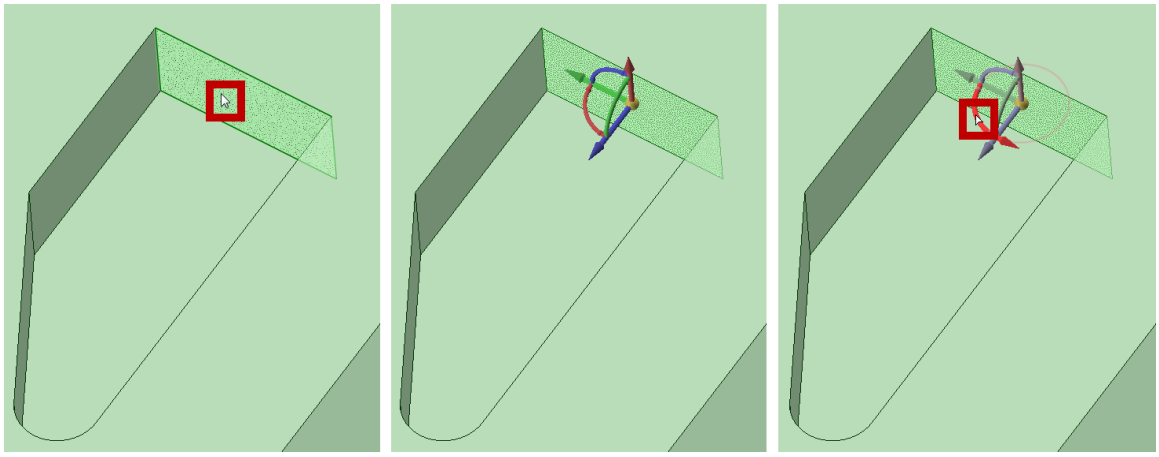
Notice that the Ruler Dimension can be setup to faces, edges, points, datums, or curves.

Remember to use selection techniques to select what to move, the **Anchor** tool to position the **Move Tool** in the correct reference position, **UpTo** to snap and **Ruler** to dimension. With these tools, you'll be able to move and position geometry quickly and easily.

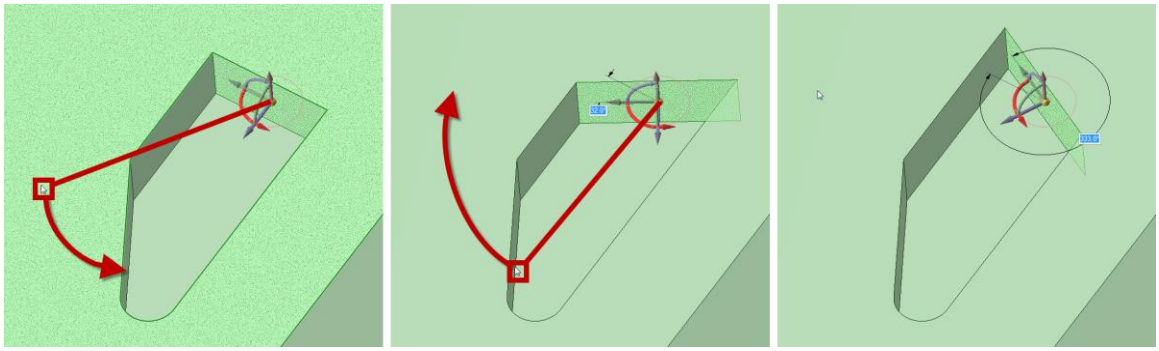
Rotating

In addition to translating geometry like faces, the move tool also rotates geometry. Let's look at how rotating a face is different yet similar to translating with the move tool

41. Zoom, pan, and spin to the view below.
42. Click the Move Tool to reset it.
43. Select the face shown below. The move tool will snap to the center of the face.
44. Click the **Rotational Arrow** shown below, red in the image.



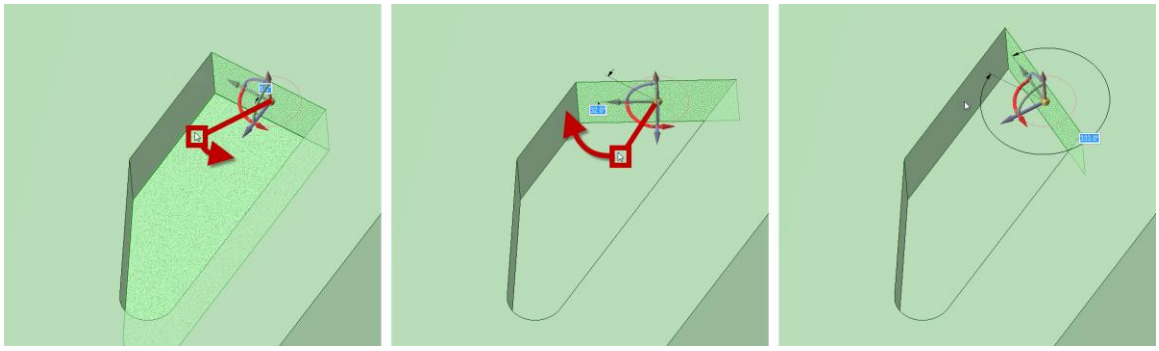
45. Position the cursor **away** from the move handle.
46. Drag in a circular direction around the **Move Handle** as seen in the image below.



NOTICE: The face will rotate around the center of the move handle as you drag the mouse.

47. Position the mouse cursor **closer** to the move handle.

48. Drag around the move handle as you did in step 45.



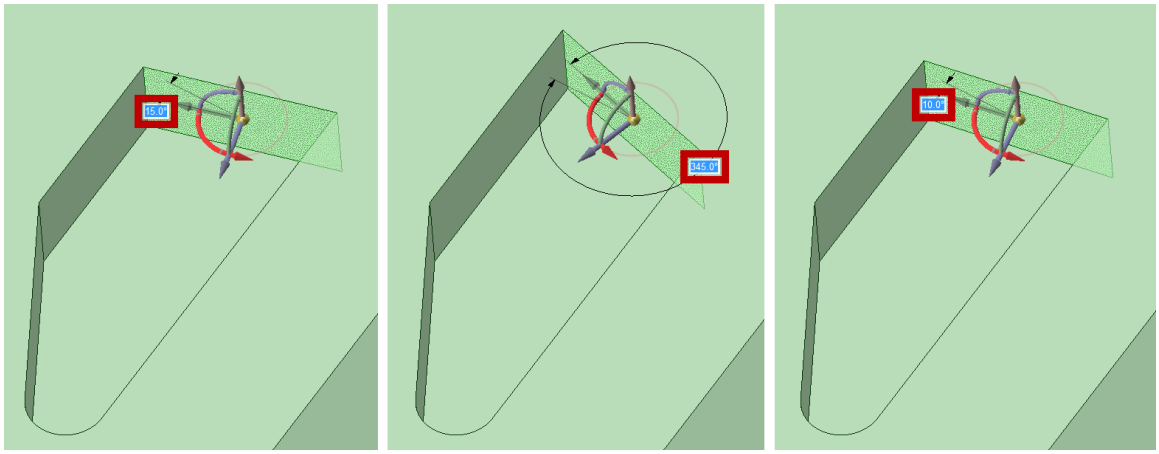
NOTICE: There is less movement needed when the mouse is closer to the move handle, compared to when it was farther away.

When rotating, consider the line between the mouse cursor and the center of the move handle to be a bar/lever used to rotate the selected object. The closer the cursor is to the move handle, the more the selected object will move when you drag the mouse.

49. Enter a value of 15 (Positive 15) and press enter.

50. Enter a value of -15 (Negative 15) and press enter.

51. Enter a value of 10.

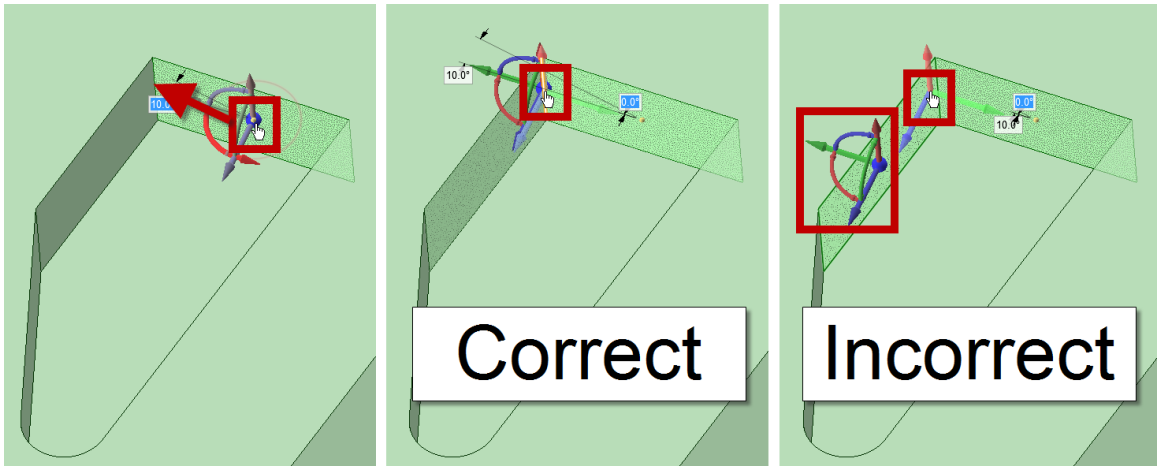


NOTICE: Whenever you enter a value for the rotation angle, it rotates the selected face to that angle from that starting point. It does not add on the amount entered to a previous rotation. Entering a negative value subtracts the amount from 360 (360-15=345).

NOTICE: The default location when you select something, whether to translate or rotate, is the center of the selected geometry.

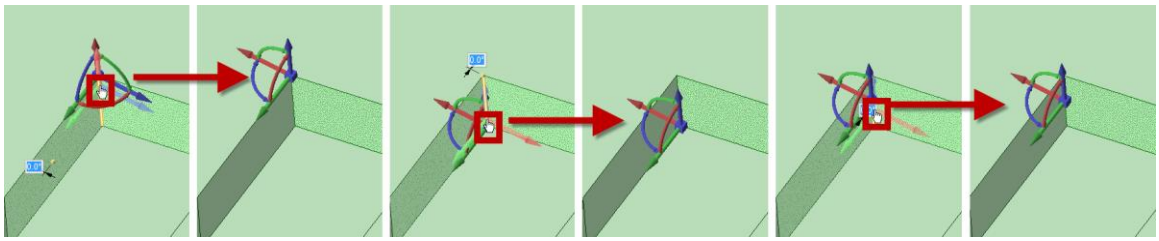
Dragging the **Yellow Center** of the move handle (or using the **Anchor ToolGuides**) can change the center of rotation.

52. Drag the center of the move handle (**Yellow Ball**) over to the edge on the left. **BEFORE LETTING GO** of the mouse, watch the preview of where the Move handle will go when you let go of the mouse. If you are not close enough to the edge, the move tool will snap to the center of the adjacent face.



NOTICE: Before you used the **Anchor ToolGuides** to change the location of the move handle. Both methods works great, and you can typically use either method interchangeably.

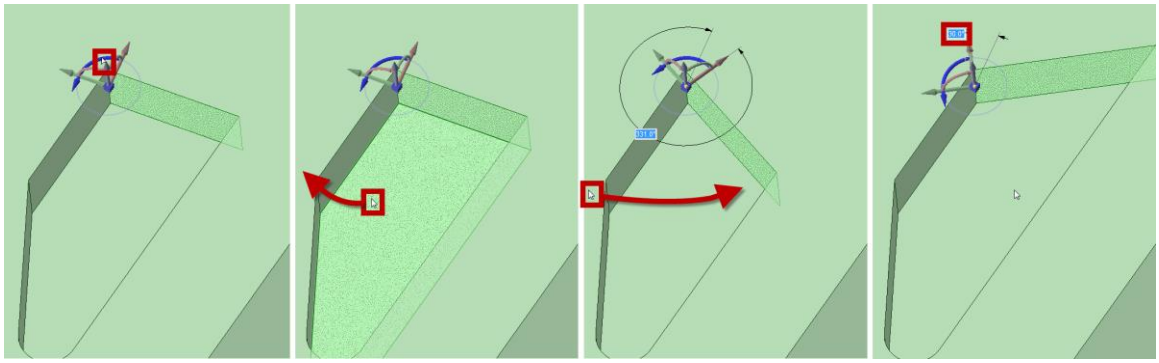
NOTICE: The center of the move handle is a **Blue Square** now. That means the original location of the move handle has been changed.



NOTICE: The location along the edge you drag the move handle to, (Top, Middle or Bottom) will determine where the move handle will snap.

For the next rotation, it does not matter where along this edge the move handle is located.

53. Click the curved move handle in the horizontal direction (Blue in the image).
54. Drag around the move handle the same was as when previously rotating the face.
55. Rotate to a value of 30 degrees.

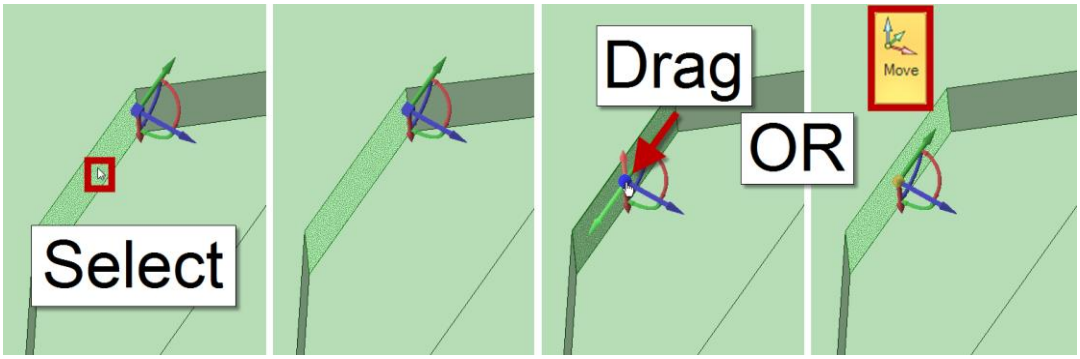


Moving Faces and Edges

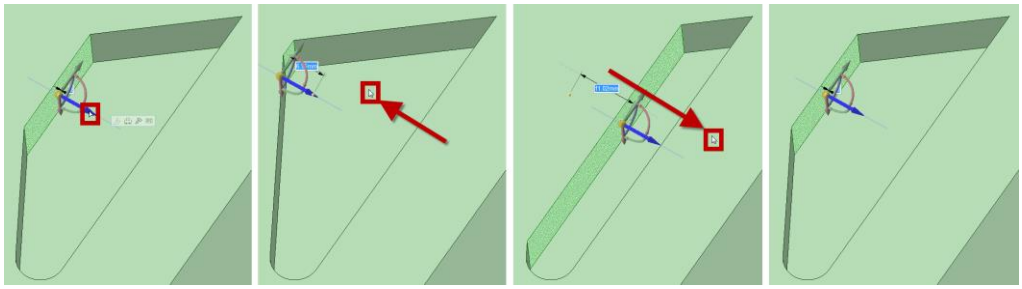
56. Select the face shown below.

NOTICE: The Move handle stays where it was previously dragged.

57. Click the **Move Tool** in the Ribbon Bar. This resets the tool and puts the Move Handle at the center of the selected geometry. You could also drag the **Move Handle** or use the **Anchor** ToolGuides.

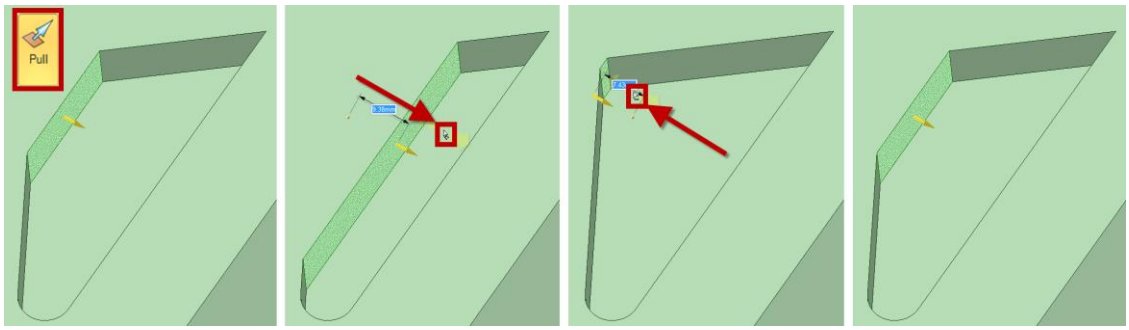


58. Select the arrow normal to the Face (Blue) and move the face in that direction. ESC or Undo.



NOTICE: The behavior is similar to pull. The moved face follows the angle of the neighboring faces.

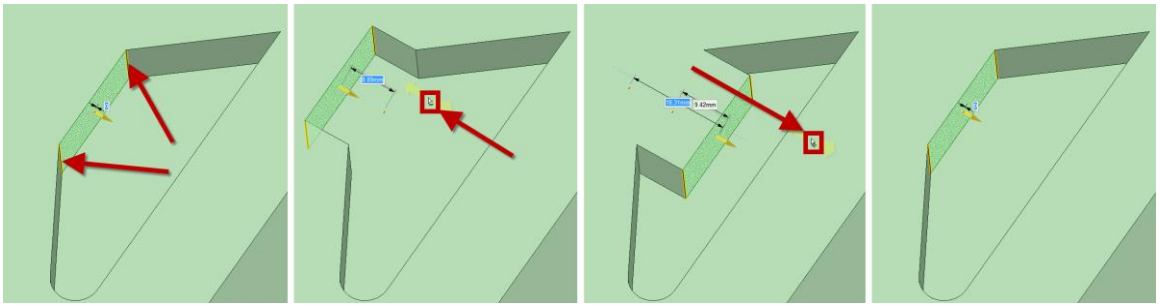
59. With the face still selected, turn on the **Pull** tool and Pull the Face to see the same behavior



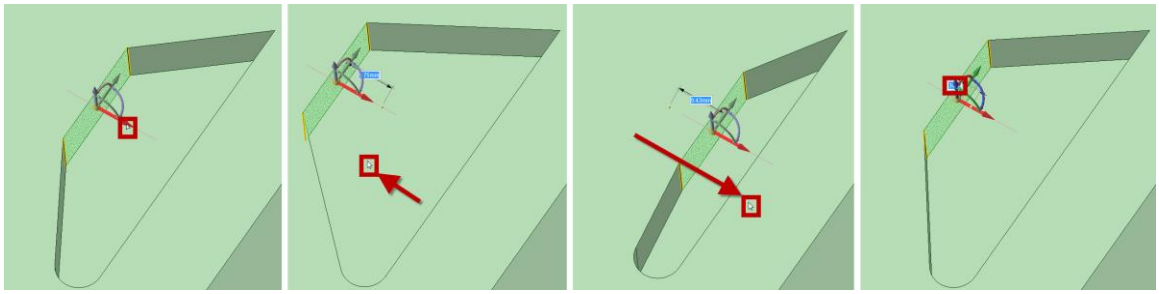
RECALL: This behavior can be overridden by selecting the **face** and **CTRL** selecting the adjacent **edges**.

60. Hold **CTRL** and add the **2 edges** on either side of the face into the Selection.

61. Pull in both directions to recall what happens when pulling. **ESC** to cancel the command or **Undo** if you finished the operation.



62. Turn on the **Move** tool. Select the arrow normal to the face (Red) and move the faces and edges in both directions. Entering a value of 3 when the face is moved up and to the left



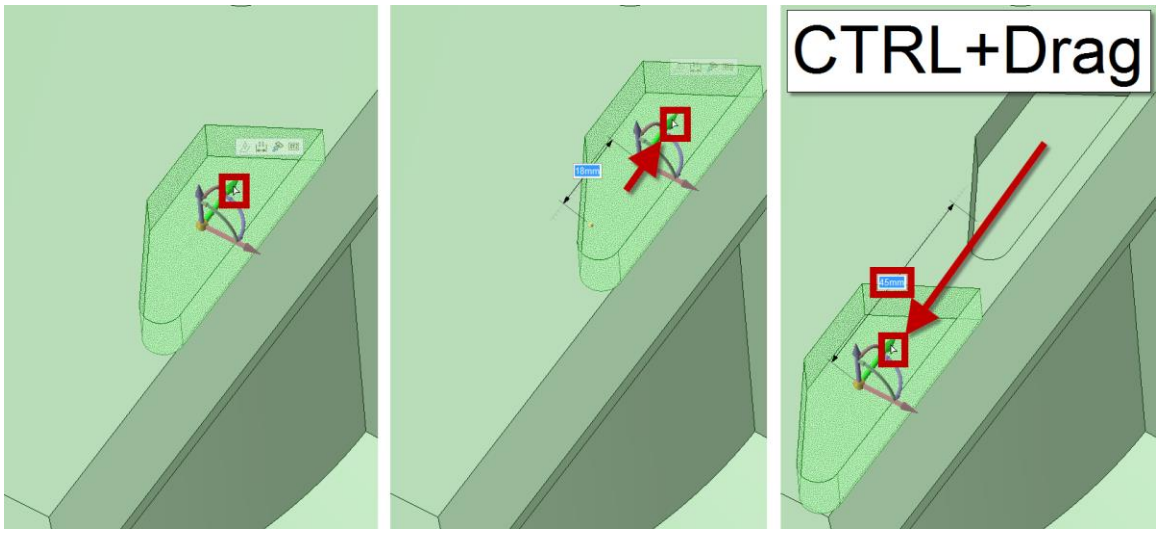
NOTICE: Similar to the Pull Tool, having the **Face** and the **Edges** selected modifies the way a face is moved.

NOTICE: Unlike the Pull tool, which is creating new geometry/faces when the **Face** and **Edges** are Pulled, the adjacent walls adapt to the face while moving.

This is useful when a face needs to remain the same size, but the location needs to change.

Copying Faces and Geometry

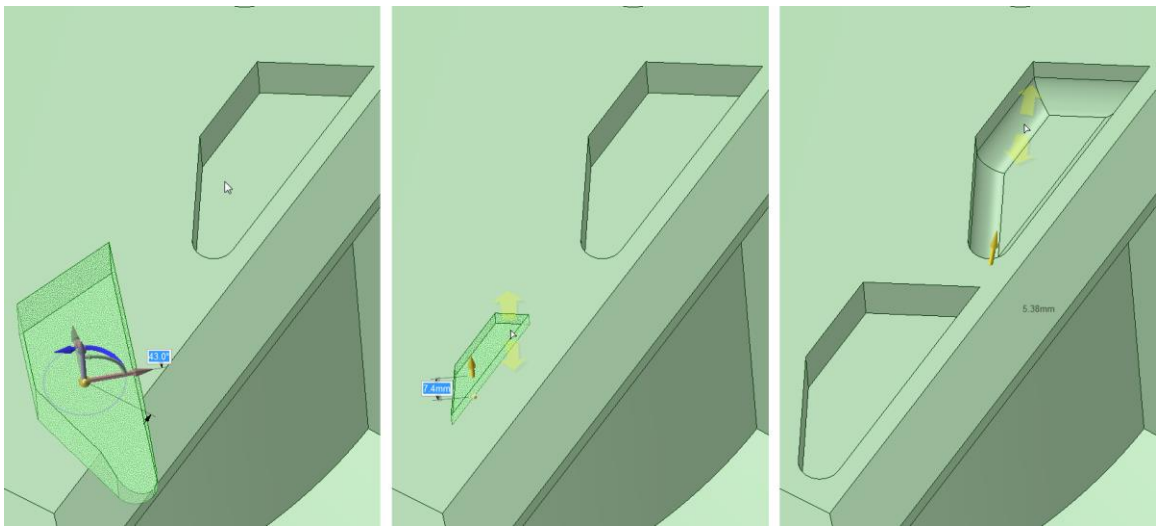
63. Zoom to the view below and box select the entire pocket.
64. Click the Green arrow, and first move the pocket up to the right 18mm.
65. **HOLD CTRL** and move the pocket in the opposite direction, 45mm.



NOTICE: Holding **CTRL** while you move something will copy it in that direction. Make sure to drag the copy past the first hole before letting go of the mouse to type in 45mm.

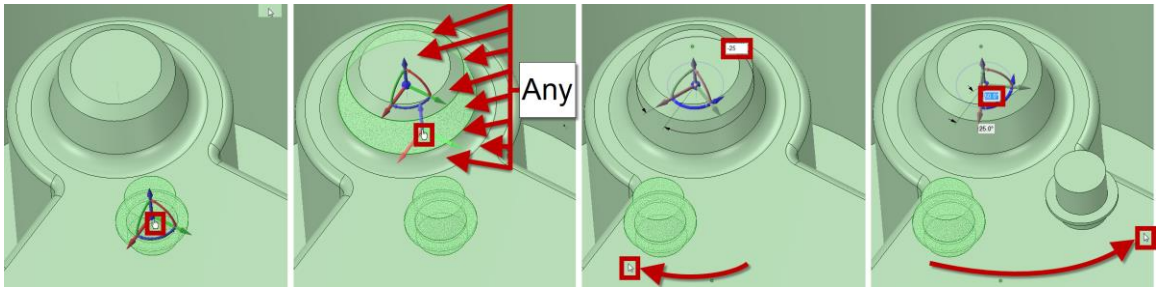
If you let go of the mouse before moving the copy completely past the original, Undo and repeat.

66. Edit either pocket with Pull or Move. `
67. Undo.



NOTICE: Each depression is independent of the other. Editing one pocket will not change the other.

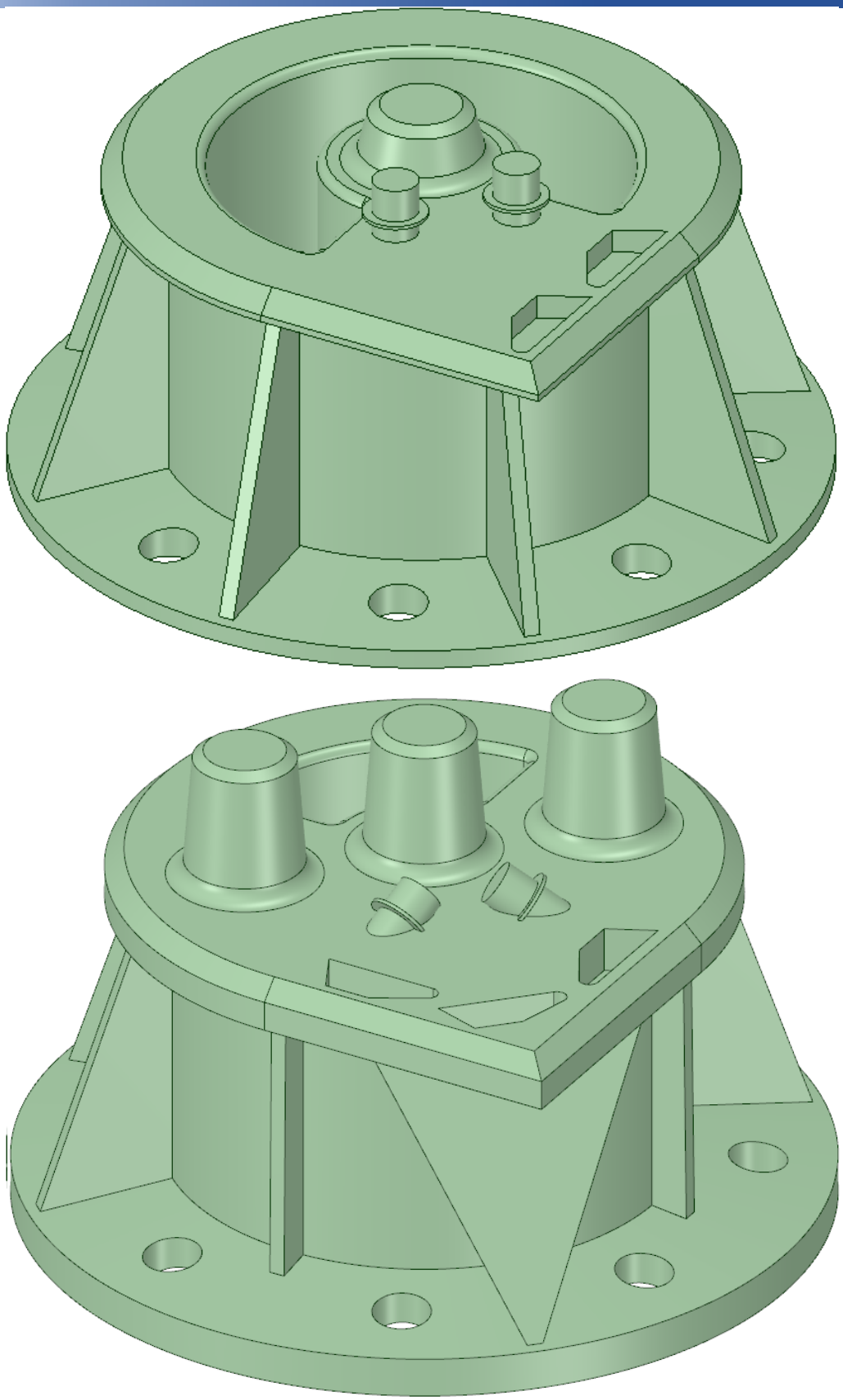
68. Zoom out and Pan to the center as below.
69. Box Select the cylindrical Pin in the center.
70. Drag the **Yellow Ball** on the move handle from the pin to any of the face or edge on the center protrusion (see Image).
71. Rotate the Pin around the center (**Curved Blue Arrow**) -25 degrees (Counter Clockwise).
72. Hold **CTRL** and drag clockwise, 50 degrees total (25 from the original location).



Anytime you Pull or Move while holding **CTRL**, it will copy whatever is selected in that direction.

You can copy faces, solids, and components while holding **CTRL** with the Move tool, while translating and rotating.

Try out moving different combination of Faces and Edges on the model.

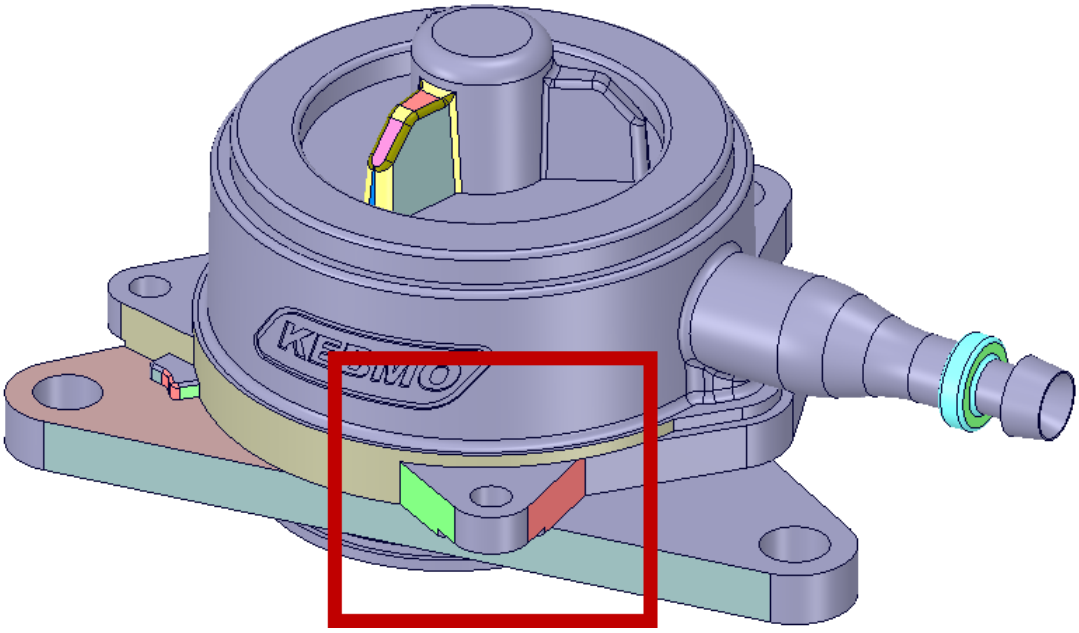


Fill

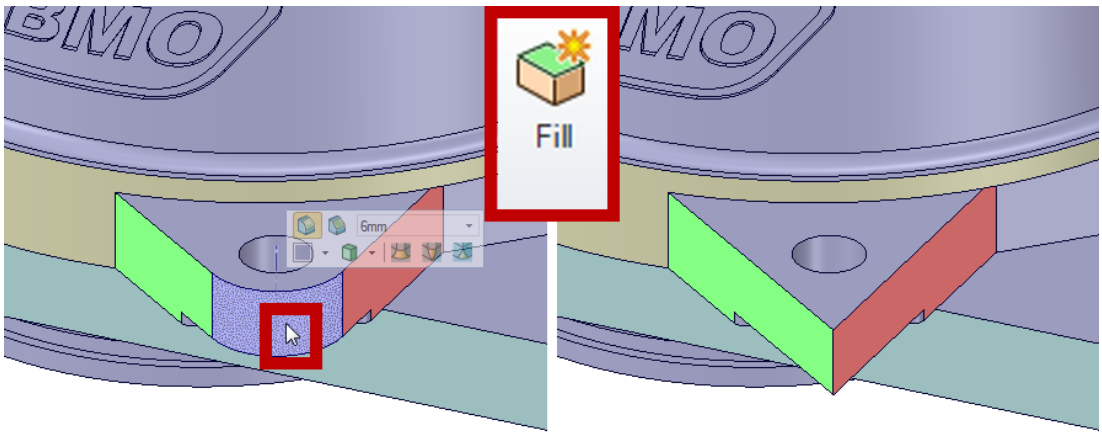
The Fill tool removes the selected geometry from the model and extends the faces which are not selected. The Fill tool can remove a Single Face, multiple faces, edges and vertexes. The Fill tool can also create geometry like Surfaces or Faces within the boundary of edges and curves.

Filling Faces

1. **File\Open**, Desktop\ SpaceClaim_Basic_Training**\04_Basic_Fill_2014.0** and open **Basic_Fill_2014.0.scdoc**,

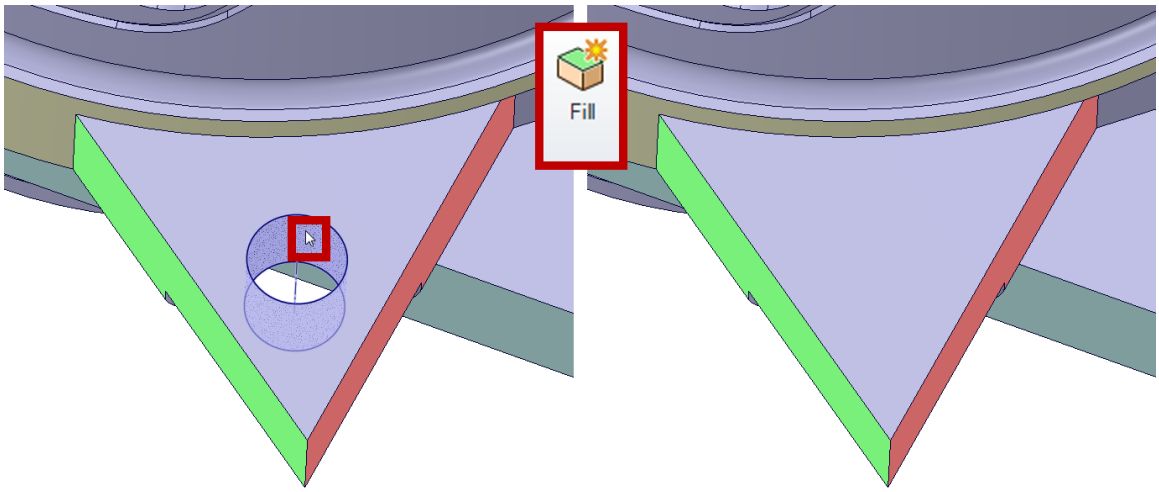


2. Zoom in on the Area above. **NOTICE:** Some of the faces have been colored for reference.
3. Select the round bellow, and click the **Fill tool**.



NOTICE: The selected Face/Round is removed, and the Green and Red Faces are extended to each other, keeping the model a water tight solid.

4. Rotate the View like below, and select the **face** of the hole shown. Click **Fill**.

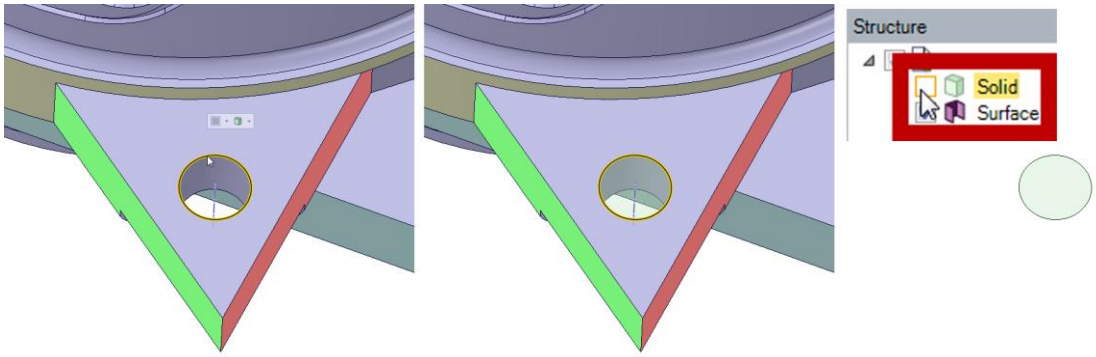


NOTICE: The face of the hole is removed, and the opening in the top and bottom face of the tab is closed off.

NOTICE: When you have something selected and click Fill, the Fill tool does not remain on.

It is very important to select the correct geometry when Filling.

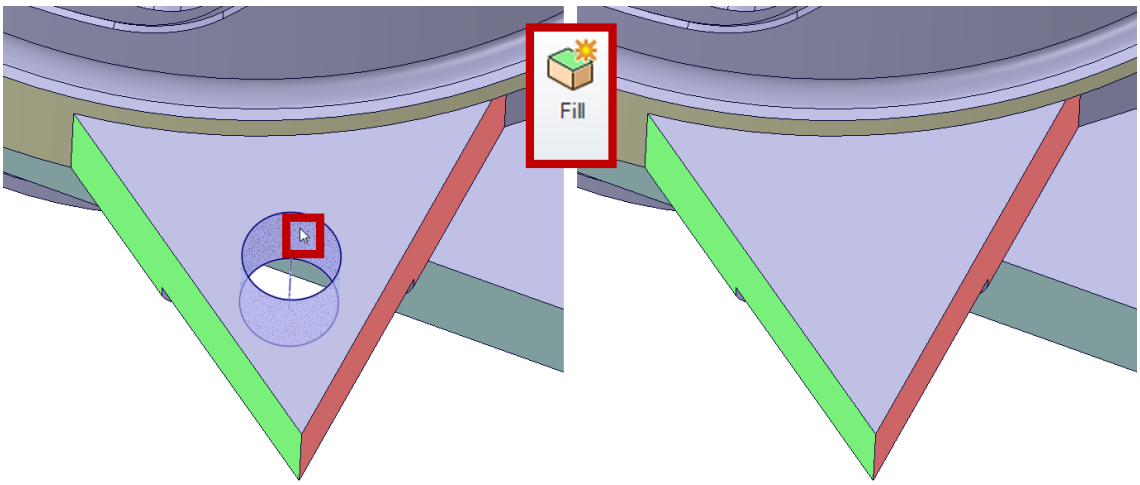
5. **Undo** the filling of the hole.
6. Instead of selecting the face of the hole, select the **top edge** of the hole.
7. Click **Fill**. Hide the solid in the Structure Tree by unchecking the box next to Solid.



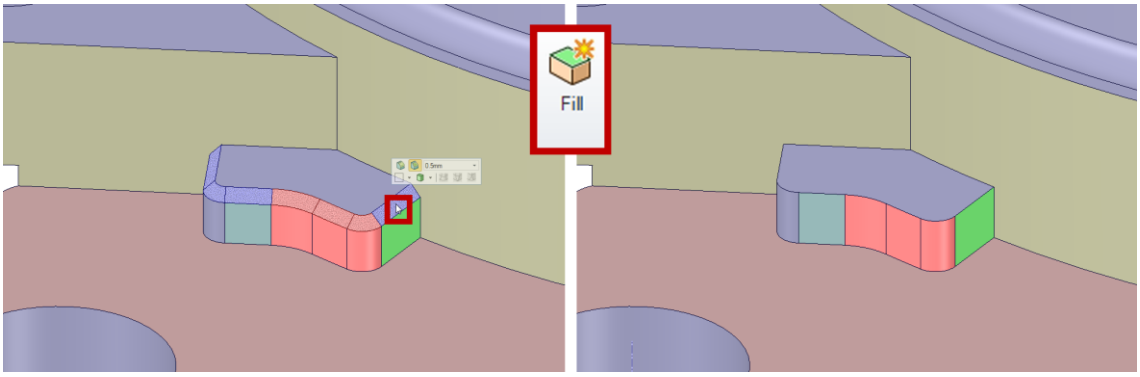
NOTICE: There is a new surface in the Structure Tree. This surface was created because Fill, fills the space within the edge with a new surface.

This new surface can be pulled downward halfway to the bottom face to make a blind hole. The Surface can also be pulled past the bottom face to turn the hole into a pin.

8. **Undo** the creation of the surface. Select the face of the hole and **Fill** as done in step 3.

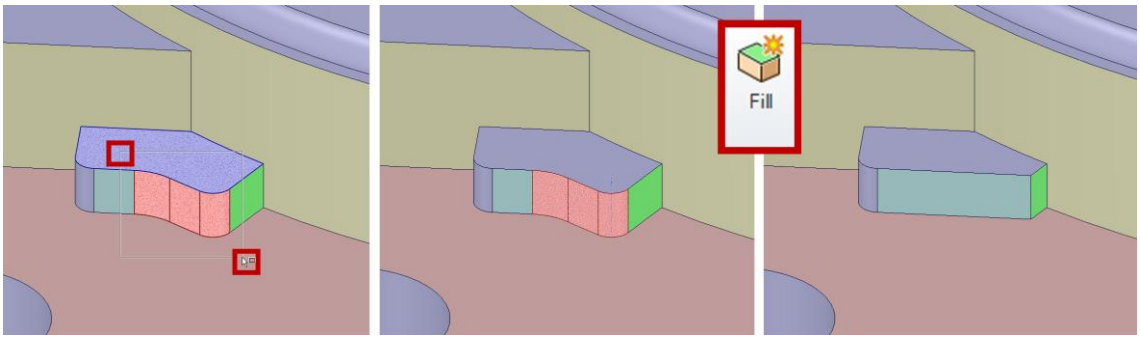


9. Navigate to the left side of the model as shown with spin/pan/zoom.
10. **Double click** any of the chamfer faces to select the entire chain of tangent faces.
11. Click **Fill**.



NOTICE: The chamfer faces are removed. The colored faces on the side are extended up to the top face, and the top face is extended out to the colored faces.

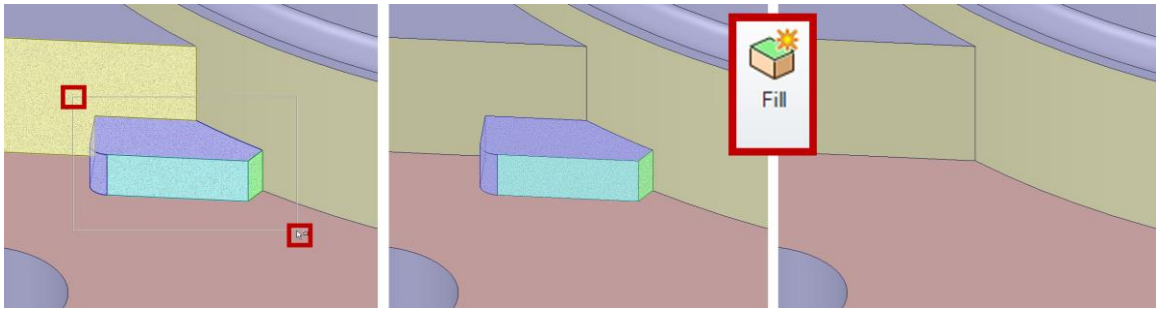
12. **Box Select** the red faces and click **Fill**.



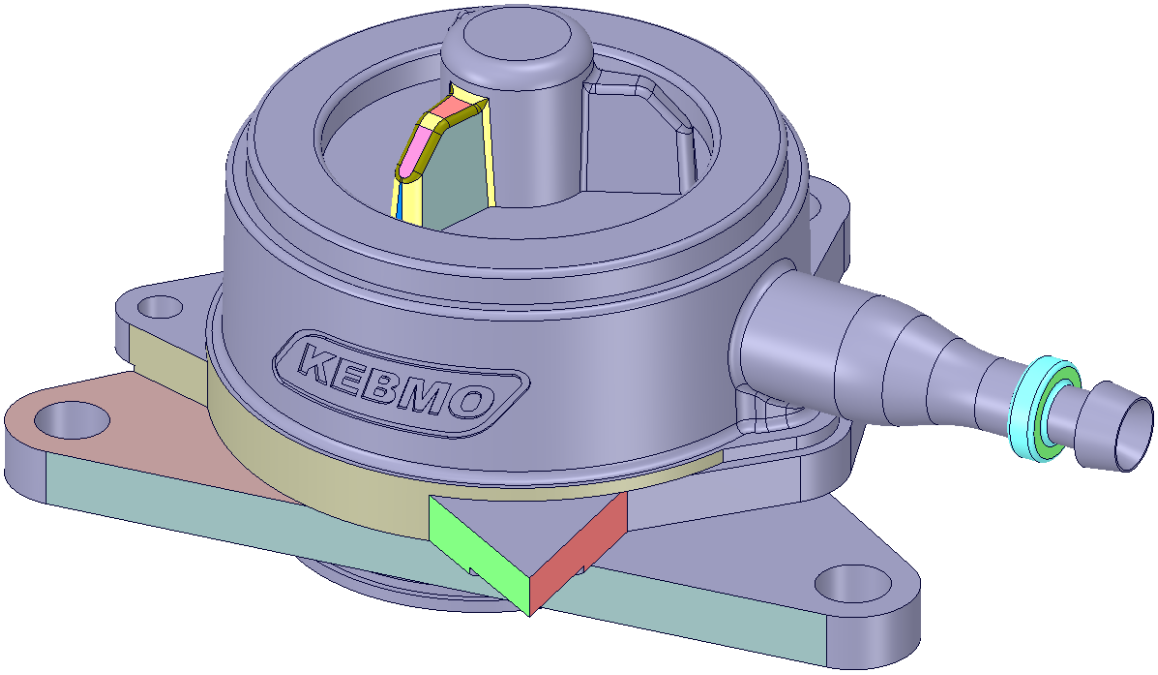
NOTICE: The **red faces** are removed, and the green and blue faces on the side are **extended**.

When using the Fill tool, it's important to thick about which faces you'd like removed, and which faces should be extended. Selected faces are removed, remaining faces are extended.

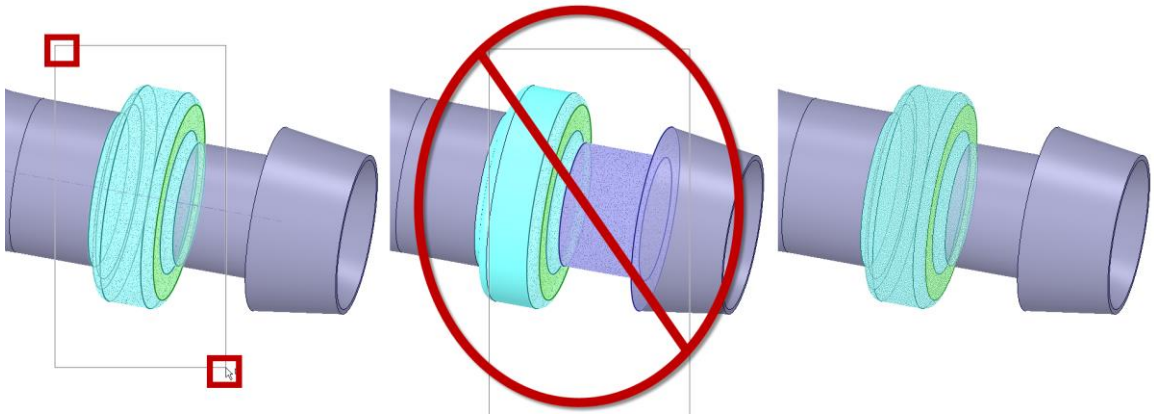
13. **Box select** the rest of the faces of this protrusion.
14. **Imagine** what faces will disappear, and what faces will stay and extend.
15. Click **Fill**.



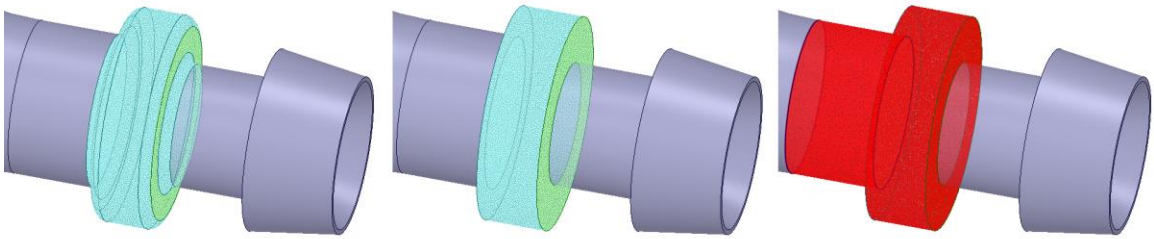
16. This is what the Model should look like now if you return to a Home View.



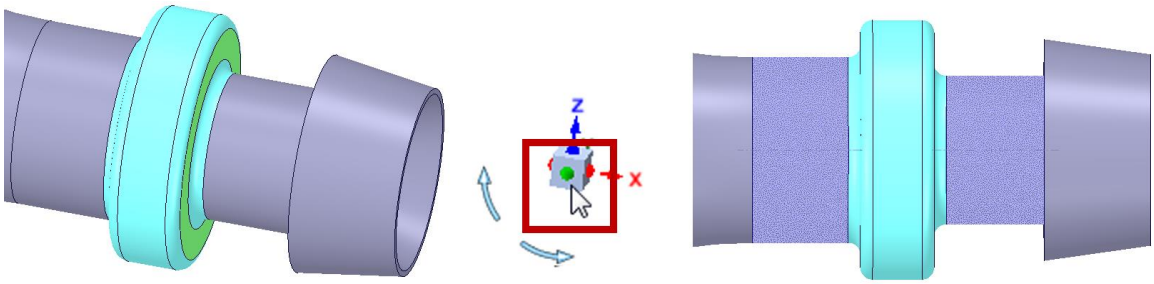
17. Navigate to the right size of the model as show below with spin/pan/zoom.
18. **Box select** the blue and green faces shown. Make sure to start and end the Box Select in the correct area as shown below. Pause during the Box Select Drag to see a preview of what will be selected.



19. Click the **Fill tool once**. Only the rounds are removed.
20. Click the Fill tool a **second time**. The faces blink red and nothing is removed.

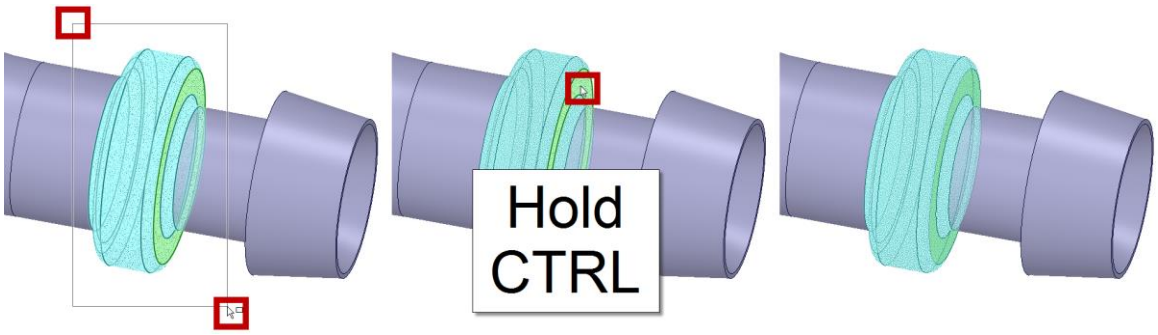


21. **Undo** to before the rounds are removed.
22. Click the green dot in the **Orientation Widget** in the bottom left corner (front view).



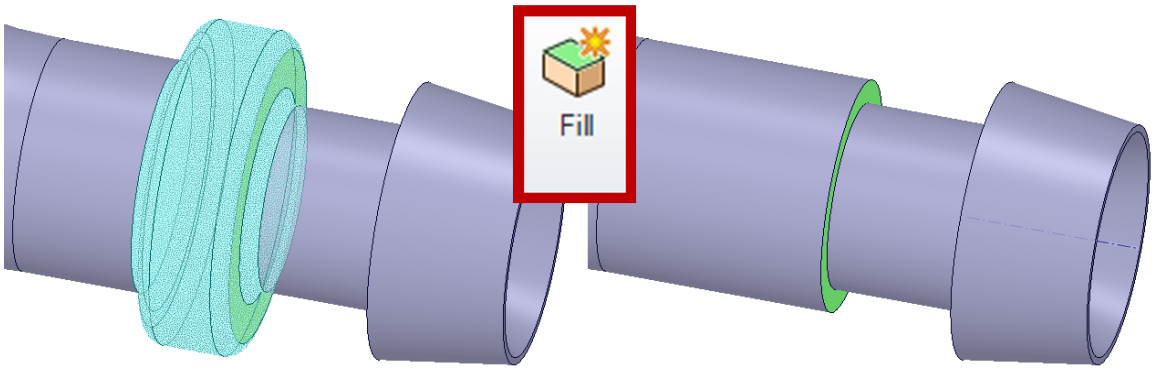
NOTICE: The reason the blue and green faces could not be removed, is because the fill tool could not extend the remaining neighboring faces (the 2 selected Purple Cylindrical Faces above) to intersect. The 2 purple Faces never extend to each other.

23. **Spin** the Model back to the view below and **box select** the blue and green faces.
24. Hold **CTRL** and click the Green Face to **deselect** it.



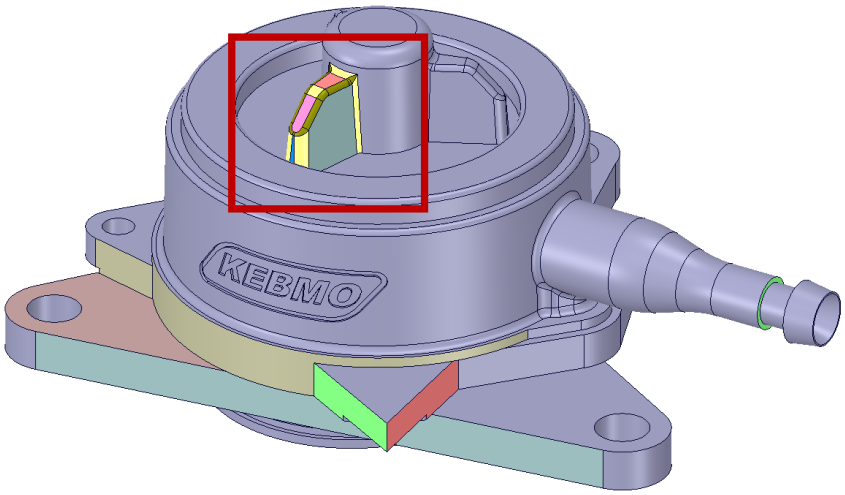
Just the Light Blue faces should be selected now, and in the Bottom Right it should say 6 Faces.

25. Click Fill.



NOTICE: The blue faces are gone, and the 2 cylindrical faces have extended to the green face.

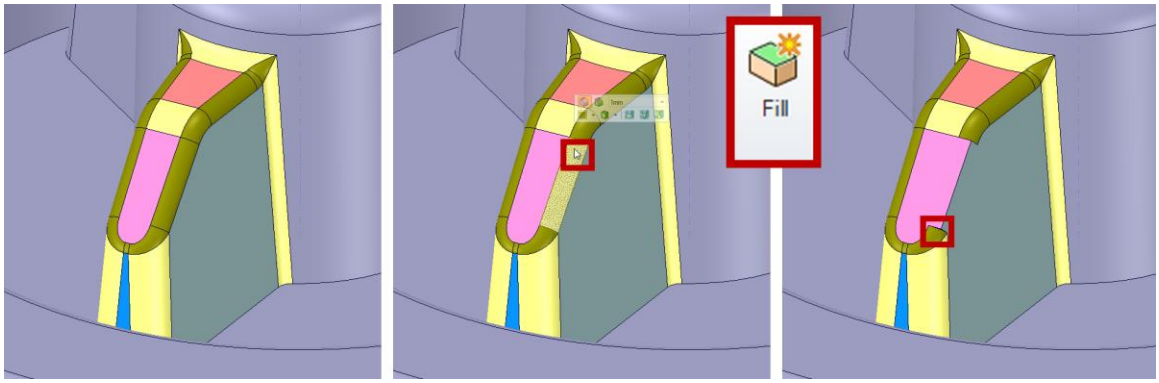
This is what the model should look like now.



Filling Rounds

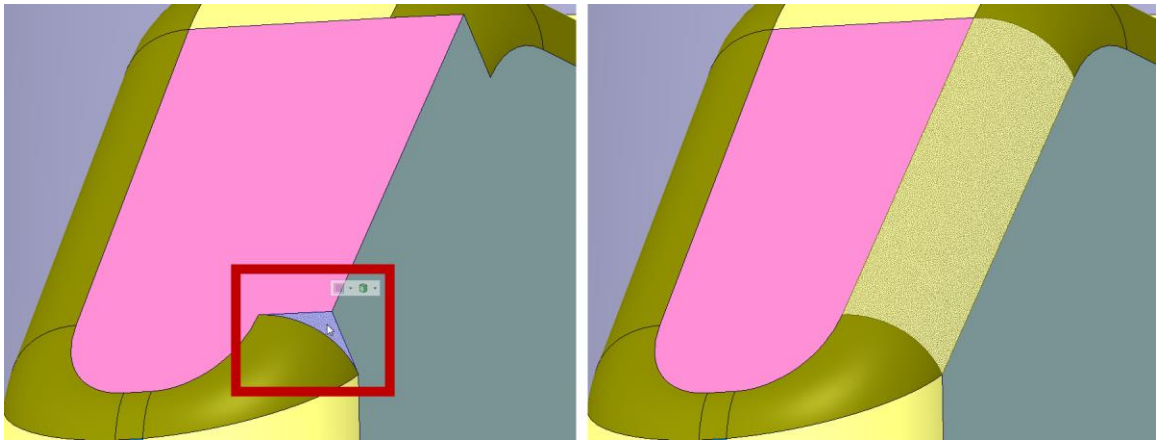
Many Rounds consist of chains of faces, making up a **round chain**. You will now explore how to Fill more complicated rounds.

26. Navigate to the region highlighted above with spin/pan/zoom.
27. Select the Round shown and click **Fill**.

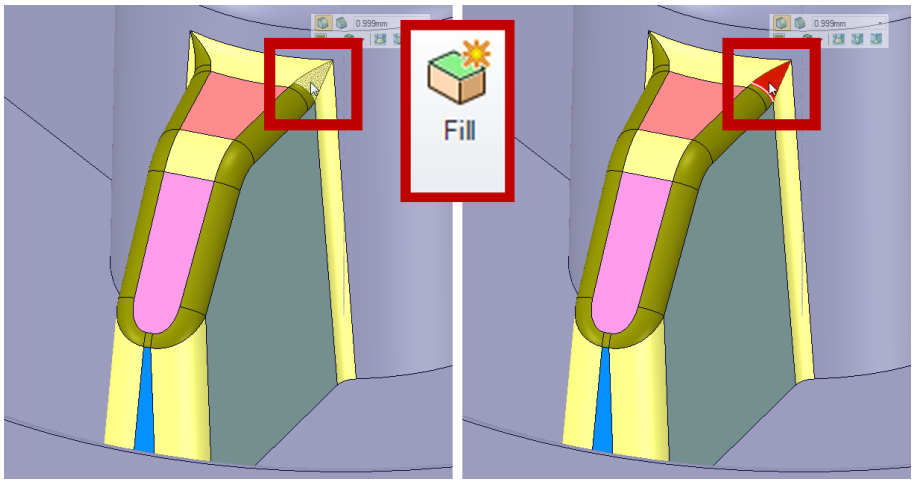


NOTICE: The 1 selected face of the Round was removed. The neighboring Pink and Blue faces extended together to make an edge. When removing a single round from a chain, new faces are created to keep the model water–tight.

28. Zoom in on the area below to see the newly created face. Typically, it's a good idea to first try to remove entire chains instead of removing rounds one at a time.
29. **Undo** the filling of the single round.



30. Zoom back out to the last view, select the face below, and Click Fill.

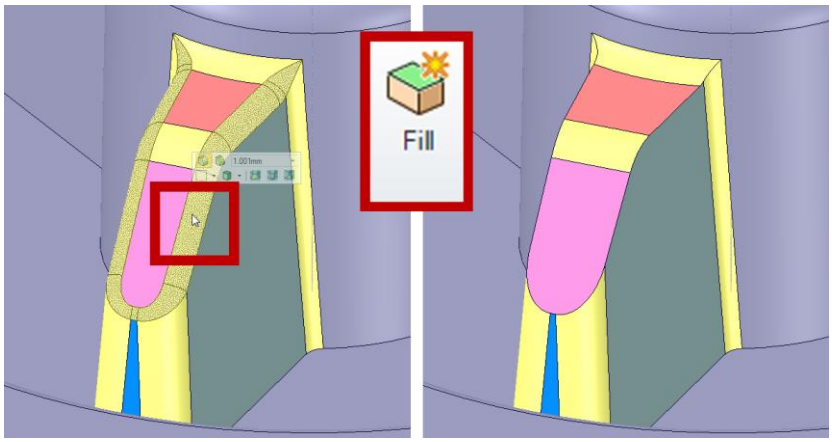


NOTICE: The face blinks red, and it is not removed. Try to imagine how this 1 face could be removed and the neighboring faces extended. If it's hard to imagine the extension, SpaceClaim will most likely not be able to remove the selected faces.

When a command like Fill can't be completed, SpaceClaim will temporarily highlight the Faces of issue in red. This becomes very useful when attempting to fill multiple Faces in different areas at the same time.

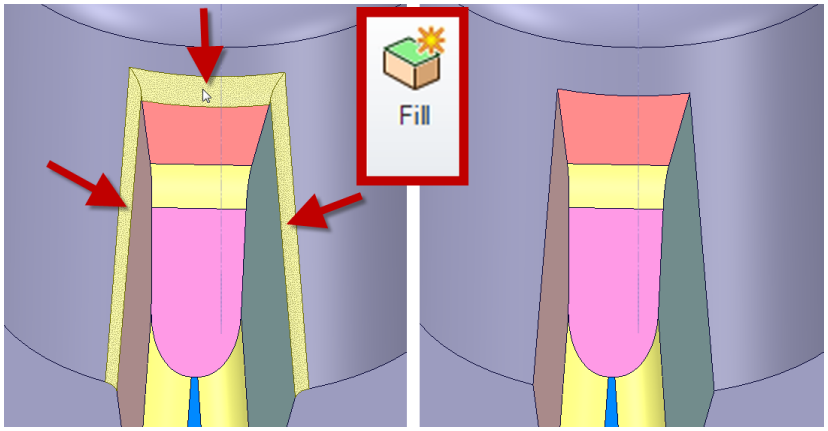
31. **Double click** on the round indicated below. Click Fill.

RECALL: Double clicking on a round will select a round chain.



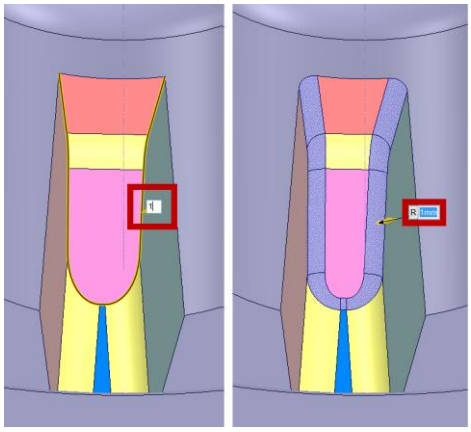
NOTICE: The entire round chain is removed, and all the neighboring faces are extended to create the edge that was originally rounded when the model was created.

32. Select the 3 round faces shown below and click Fill.

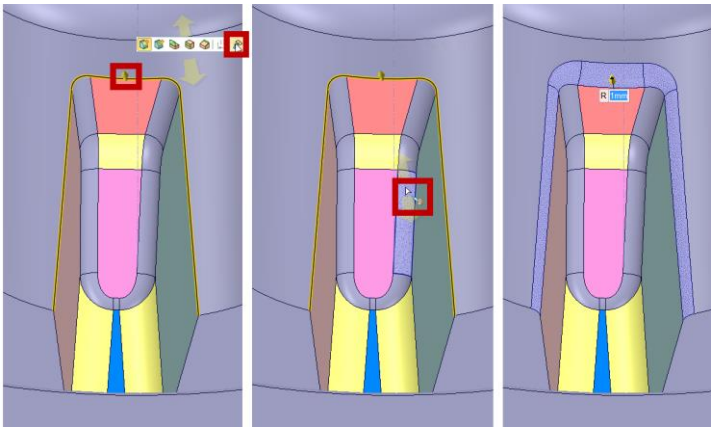


One of the main reasons to remove rounds is because they were not initially modeled in a way that can be manufactured. Fill can remove incorrect rounds, and then Pull can recreate the rounds in the proper order.

33. **Double click** on the edge below, **click Pull**, type 1 and press **Enter**.

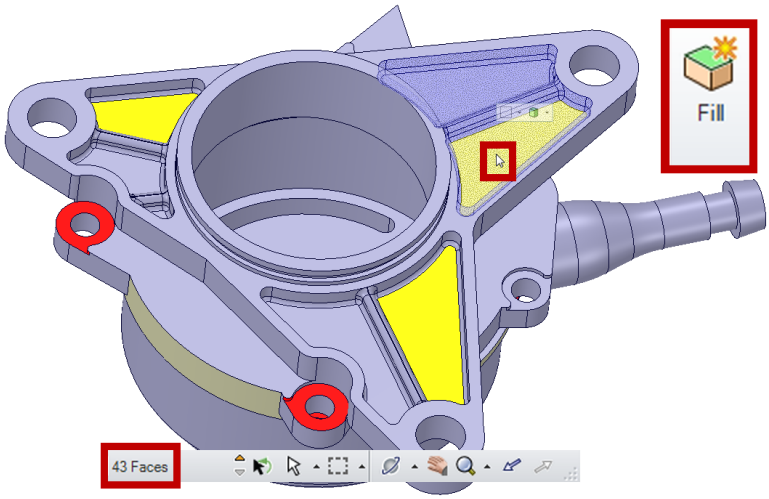


34. **Double click** the edge shown, and either type 1 and press Enter, or use the **UpTo** Command and select the first round pull a round of the same size.



Removing Depressions

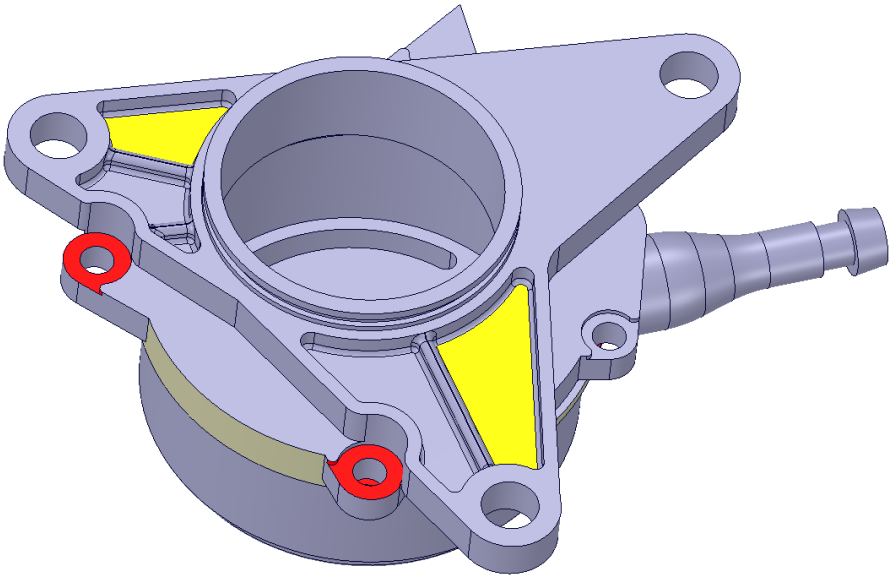
35. **ESC** out of Pull from the last step, or switch to the **Select Tool**.
36. Navigate to the view below with spin/pan/zoom.
37. **Double click** on the yellow face shown below.



NOTICE:

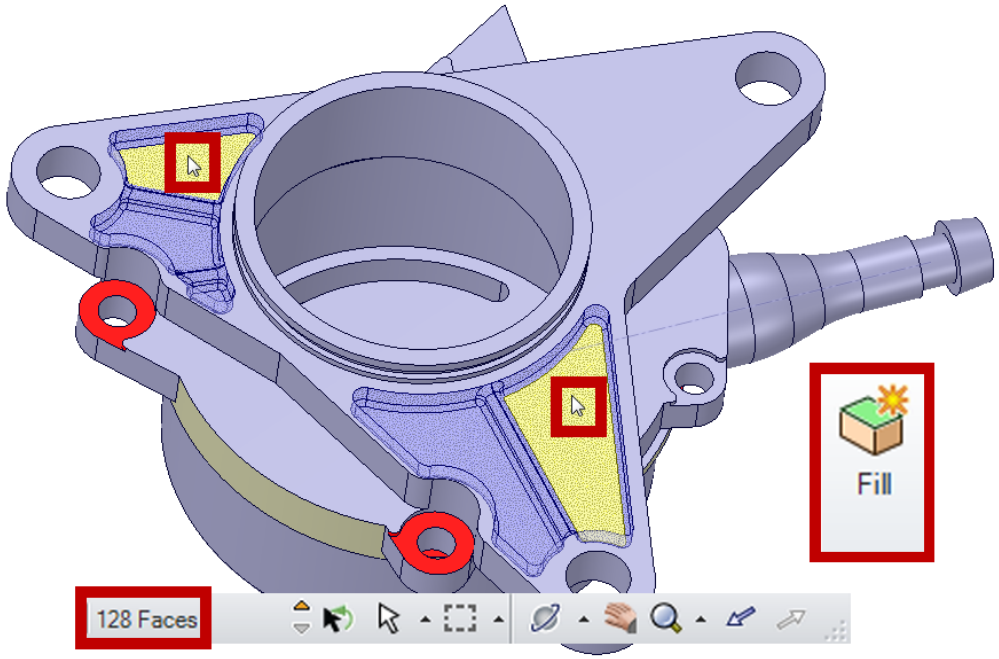
- The bottom middle of the screen shows that 43 Faces are selected.
- The yellow face was double clicked so it selected the entire pocket because all the faces in the pocket are tangent to each other.

38. Click **Fill**.

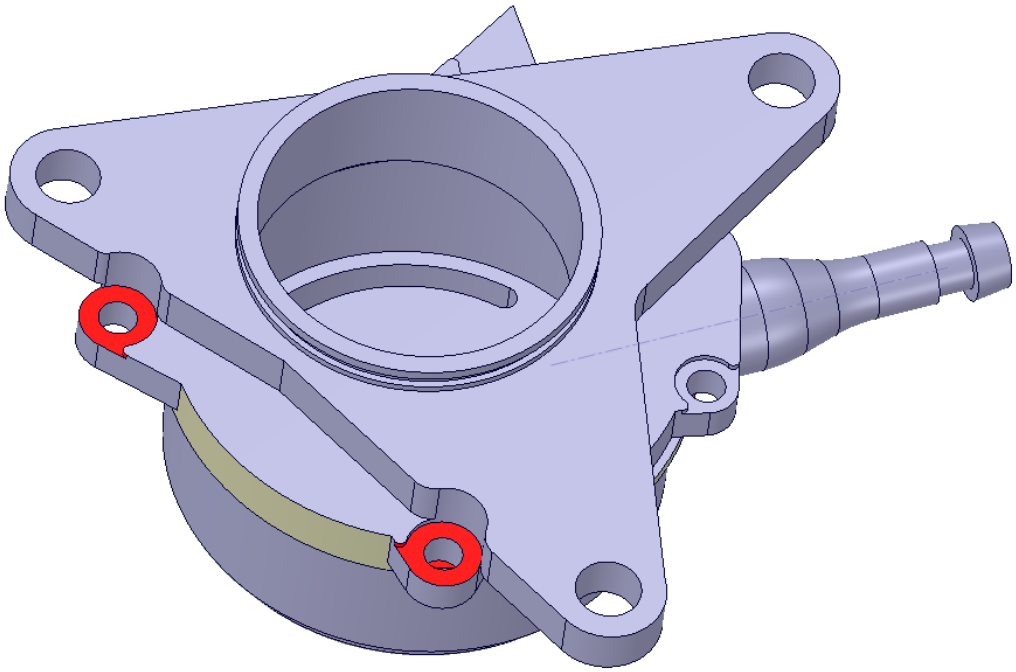


Even though a large number of Faces was selected, the Fill command finished very quickly. What takes computing power is figuring out the extending faces. With only one planar face to extend, the operation finishes extremely fast.

39. **Double click** one of the remaining yellow faces, hold **CTRL** and **double click** the other yellow face.



40. **Click Fill**.

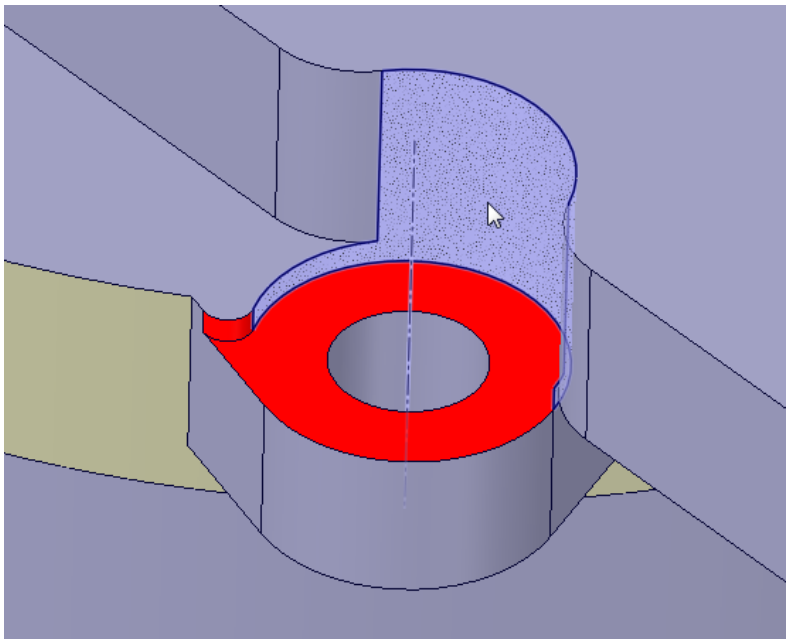


Pull for Filing

Fill is a very powerful tool for cleaning up a model by removing faces. The geometry in the model may actually require extra steps to be performed for the Fill tool to be able to remove what you want.

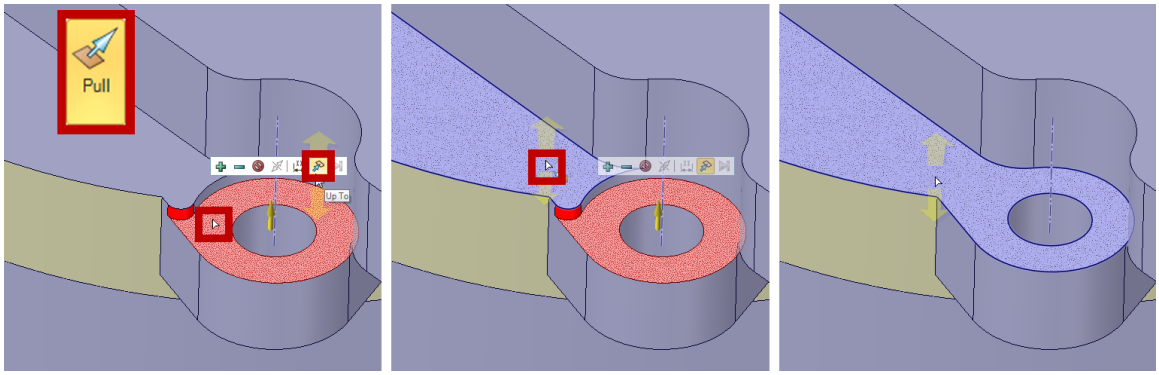
Sometimes the Pull tool can do a quicker and easier job of removing something than Fill.

Zoom in on the Lower Red Faces.



The goal is to remove the step where the red faces are. In order for Fill to remove the step, the highlighted Purple Face would have to be split into 2 faces, that's extra work.

41. Click **Pull** in the middle of the **Ribbon Bar**, select the large red face.
42. Click **UpTo** and select the Face shown below.

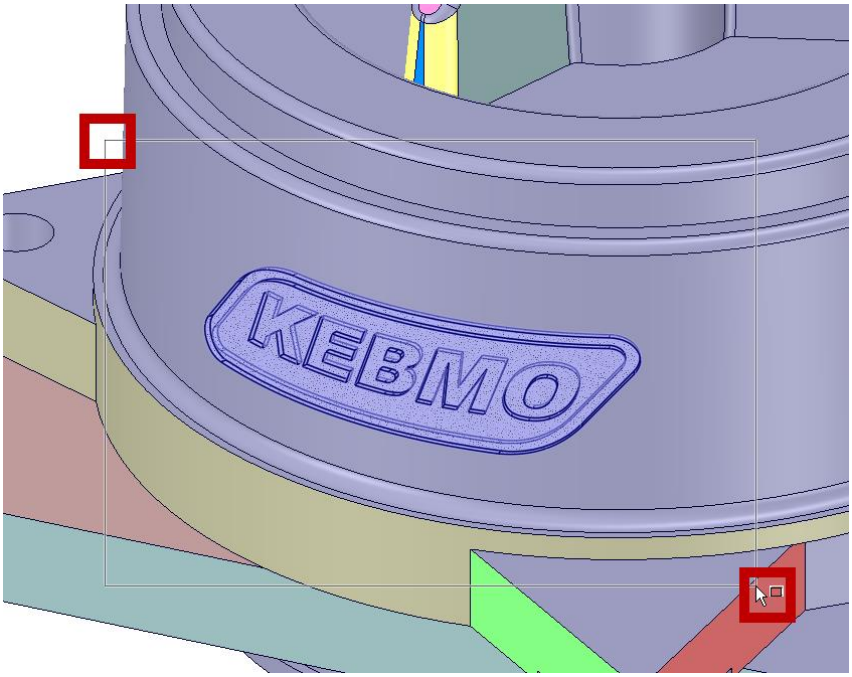


NOTICE: With just a few clicks, the Pull tool removed the red faces faster than the Fill tool could, which would have also involved using the split face tool.

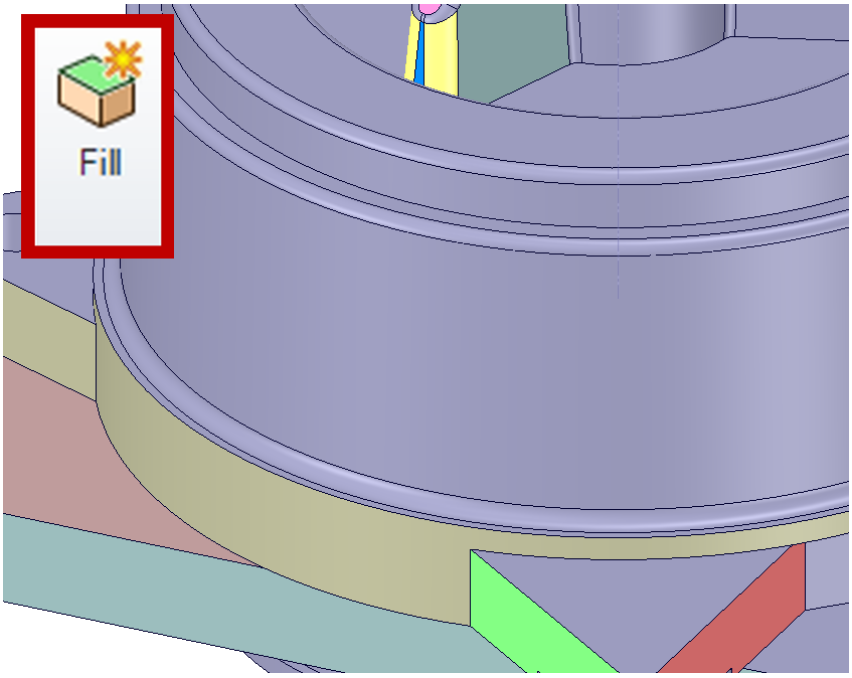
The Fill tool is use frequently to remove text from models for a variety of reasons.

43. Return to a home view (H)

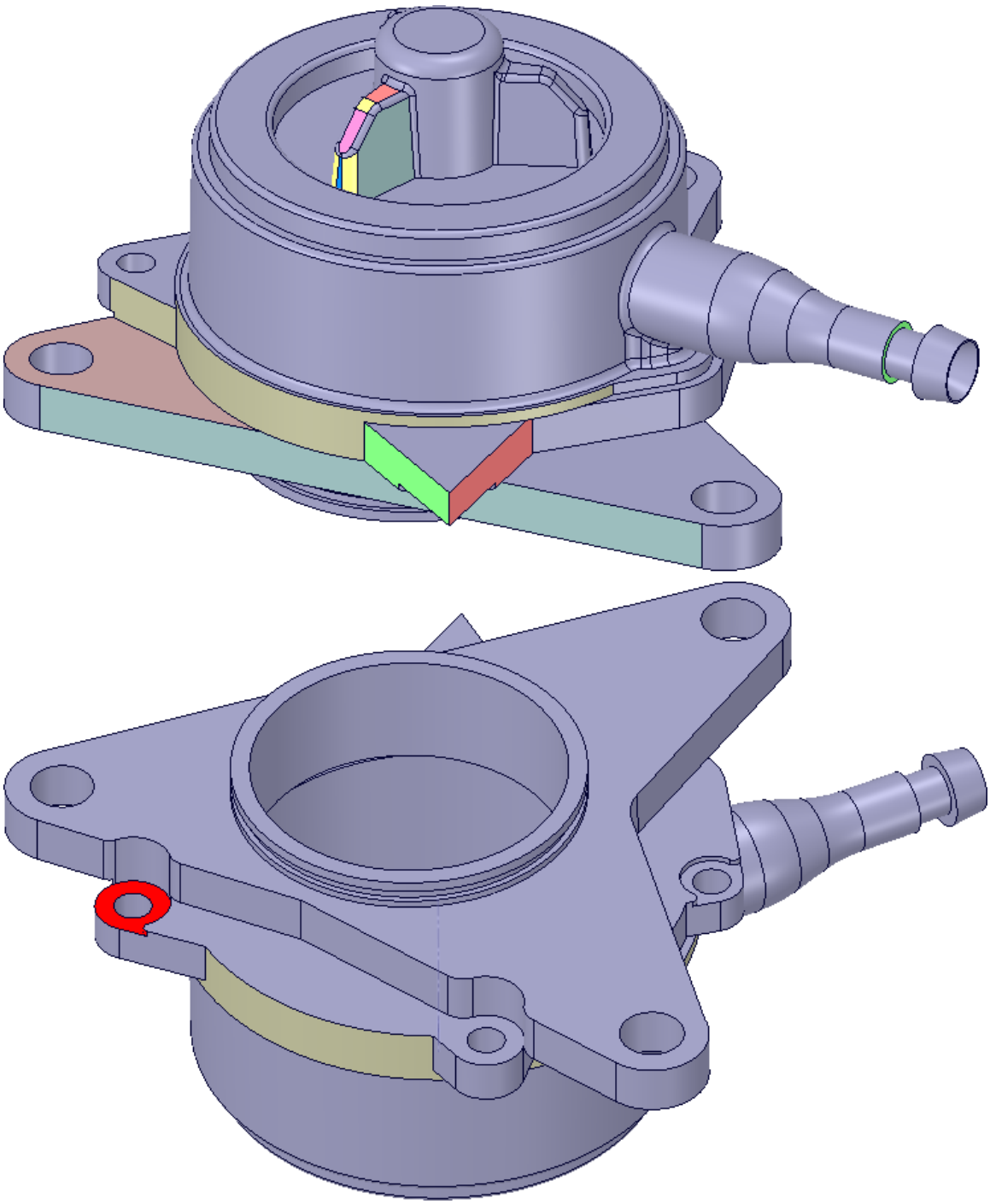
44. Box Select the Text on the front Cylindrical face



45. Click Fill



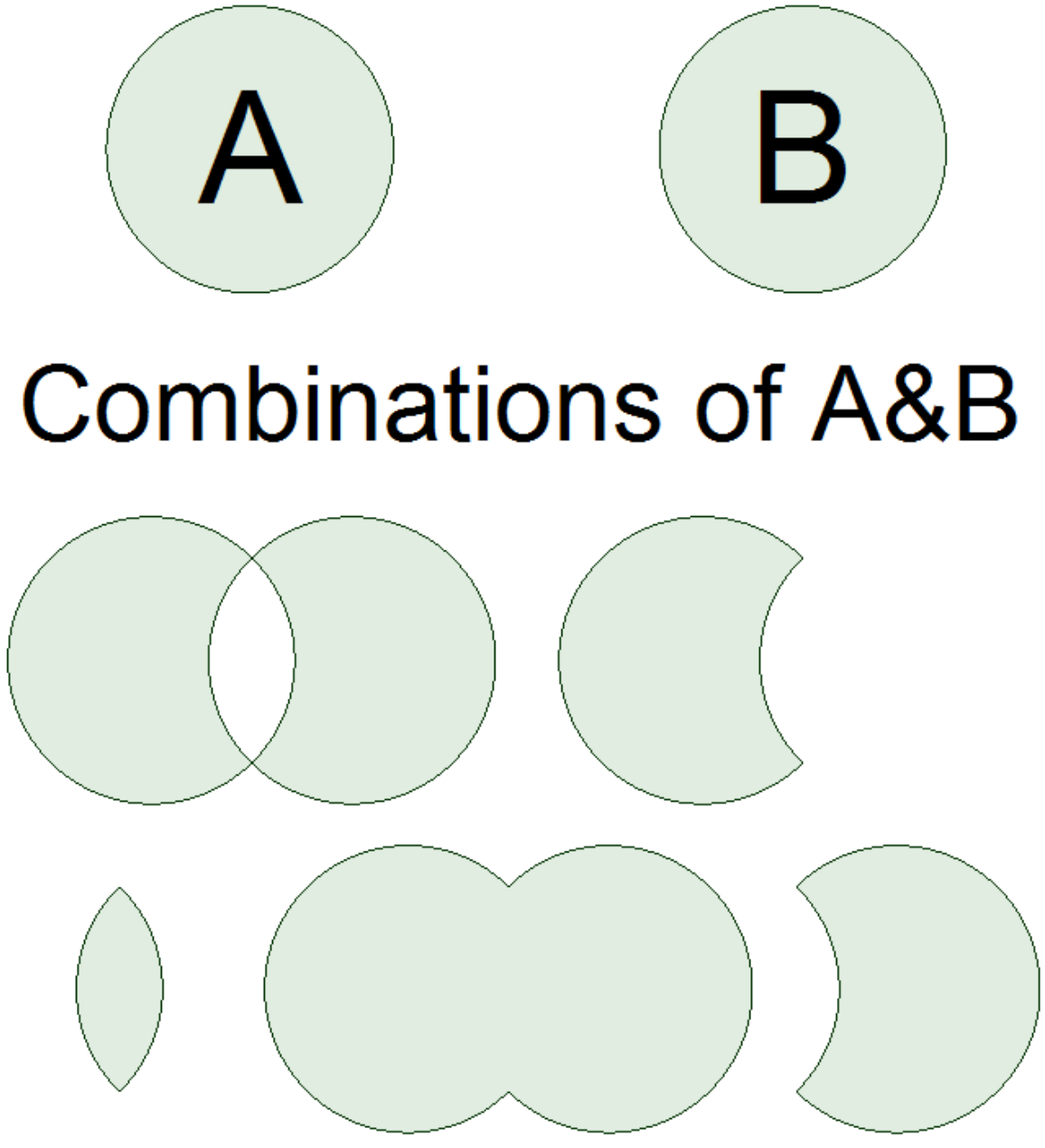
Box Select may not always be the best way to select text. You can use any selection method



Combine

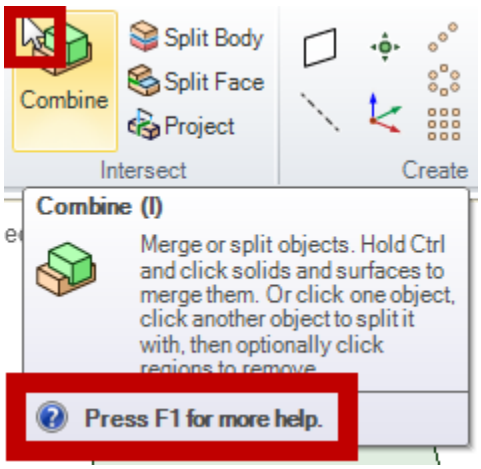
Combine is short for Combination. This is very important, because the Combine tool sounds like it only merges objects together. But a Combination of objects could be merging or splitting any number of objects.

A common term for what the Combine tool does is Boolean. Think about a Venn diagram, and how many combinations of A and B there are:



The Integrated Help in SpaceClaim is always very useful for getting more information on a tool

RECALL to open the Help page for any tool, hover over the tool, and once the tooltip shows, press F1



NOTICE that besides providing detailed instructions on using the Combine tool, the Help page shows images of the tool in action, showing what the tool is capable of

The Combine tool is used to make combinations of objects. You can add (or merge) objects together and you can subtract (or split) objects from each other. These actions are also known as Boolean operations.

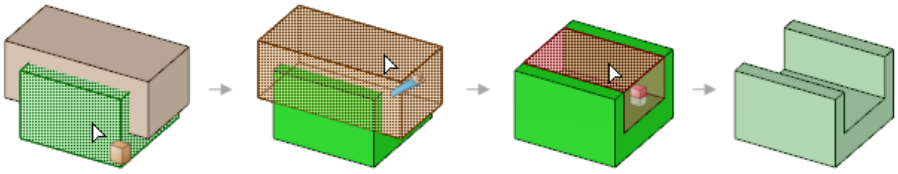
Tool guides for the Combine tool are sticky, and appear with a double outline when you click them. The tool guide remains selected so you can perform the same action repeatedly without holding the Ctrl key. To unstick a tool guide, click it again, click another tool guide, or click an empty place in the Design window.

When you use a pattern with the Combine tool, the entire pattern is merged or used to cut the target.

To split solids and surfaces

1. Click **Combine**.
2. Select the target solid or surface that you want to split.
3. Select one or more object to use as the cutter.
4. (Optional) Select the split region(s) that you want to delete.

▸ Detailed instructions

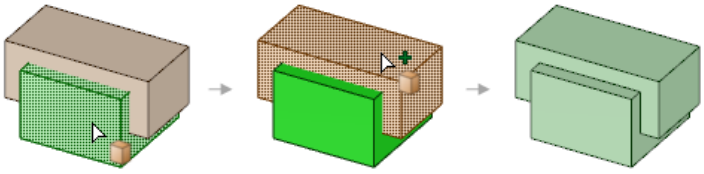


To merge solids and surfaces

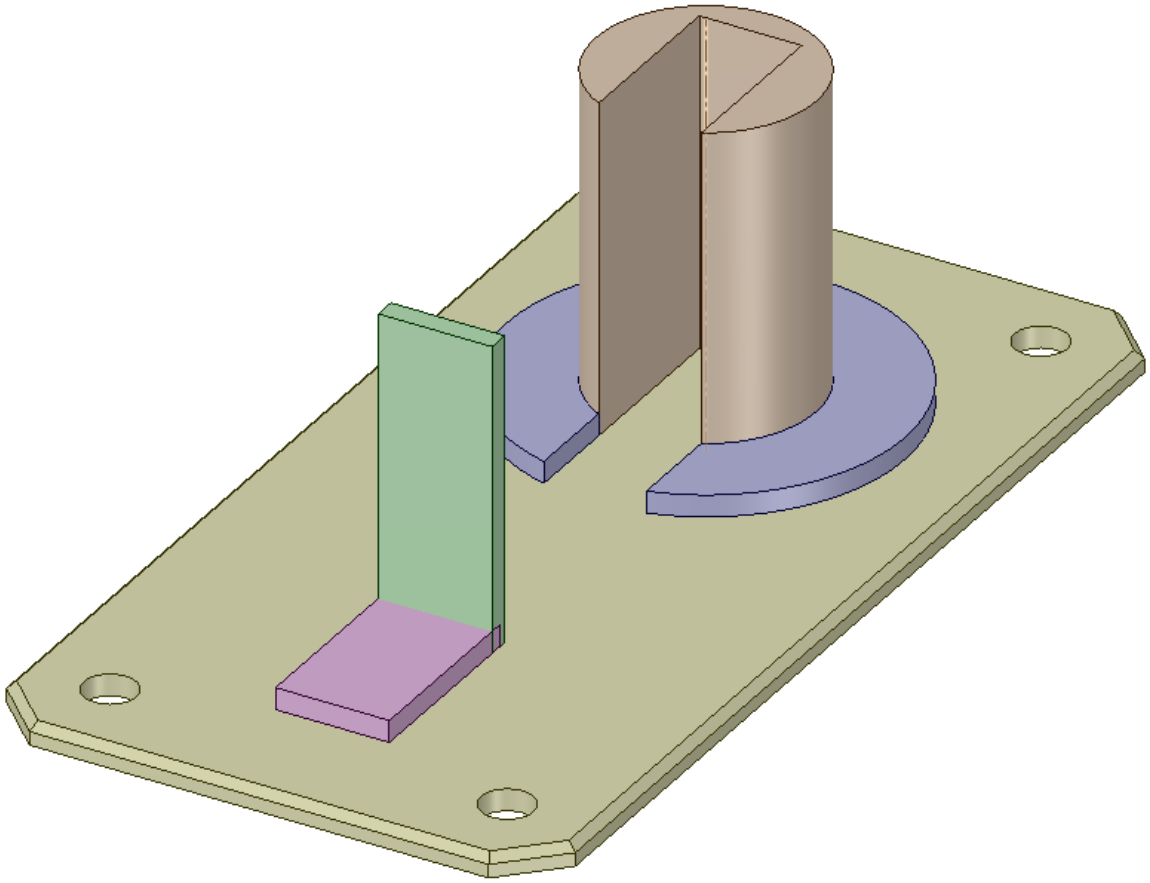
1. Click **Combine**.
2. Select the target solid or surface.

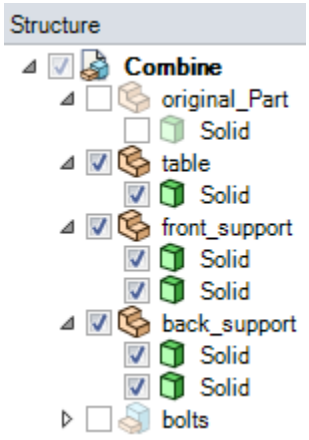
   You can select objects for Combine in the structure tree. If you select a component, all objects belonging to that component will be selected. You can also box select multiple solids or surfaces to merge them in one operation.

   Solids can be merged with solids, and surfaces with surfaces. Solids and surfaces can only be merged if the surfaces make a region that can be added to or cut out of the solid.
3. Click the **Select Bodies to Merge** tool guide or hold the **Ctrl** key.
4. Select the solid(s) or surface(s) that you want merged with the target.



1. Close the SpaceClaim help window
2. **File\Open**, Desktop\ SpaceClaim_Basic_Training**\05_Basic_Combine_2014.0** and open **Basic_Combine_2014.0.scdoc**

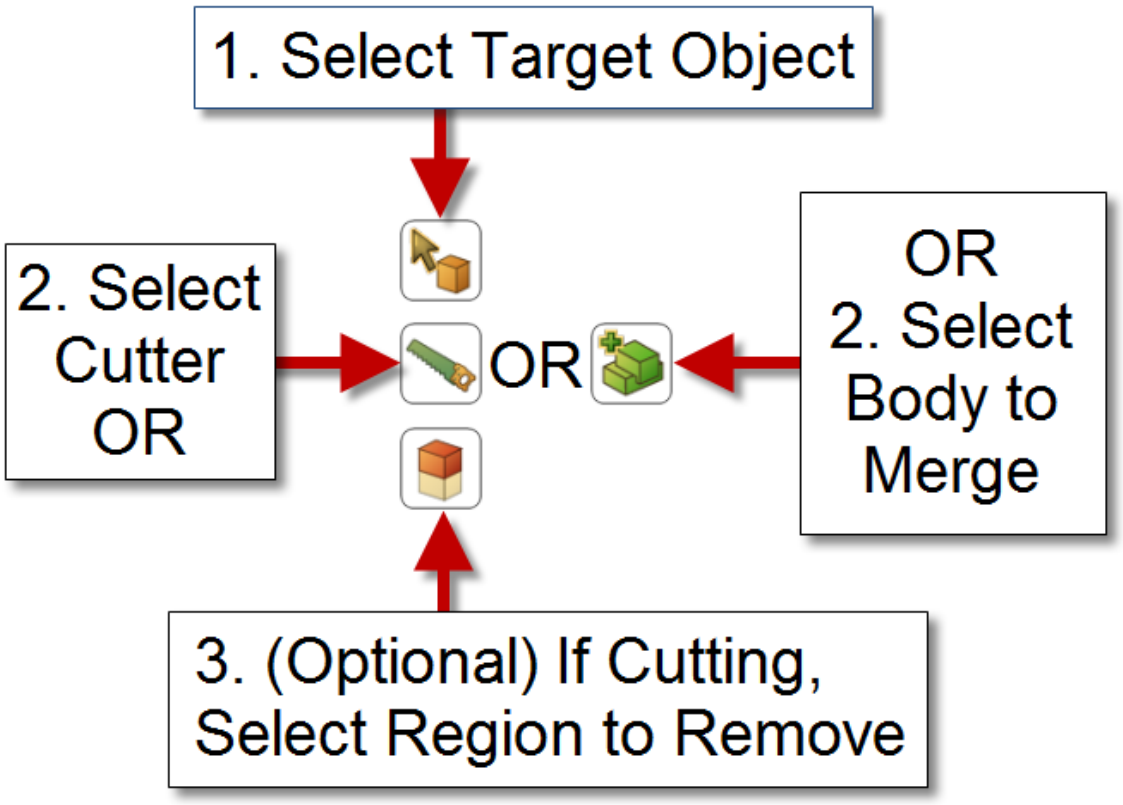




NOTICE the layout in the Structure Tree. Expand out the components by clicking the small triangle in each row and notice that there are multiple solids for both the front and back supports. There are a total of 5 solids for the fixture, 1 for the original part, plus some bolts.

Merging Solids

3. Make sure nothing is selected (click in white space), and **turn on the Combine tool**
4. First take a look at the **ToolGuides** (default location is Left side of window, but could be right, top or bottom)

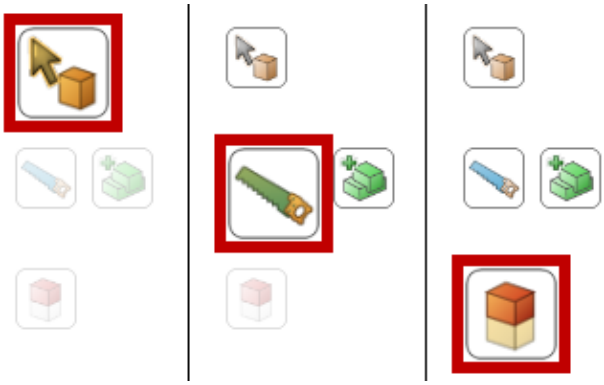


RECALL that Pull has 7 ToolGuides that were not used in this manual. Move has 8 ToolGuides and you selectively used 2 of them when needed (anchor and direction).

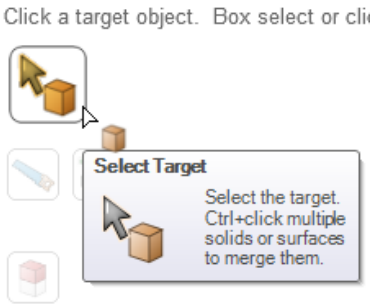
IMPORTANT: For many tools like Pull and Move, the ToolGuides are optional and used as needed.

The Combine ToolGuides literally Guides you through the steps of the tool. The 1st and one of the 2nd ToolGuides is always used when you use combine.

NOTICE that at any given point in time, one of the ToolGuides is bigger than the rest. **The Big ToolGuides is the Active ToolGuides.**



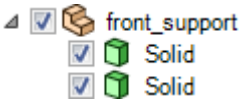
The first step of the Combine tool is to select the target object (Status message near the ToolGuides says this too)



When merging 2 objects, which should be chosen as the target object?

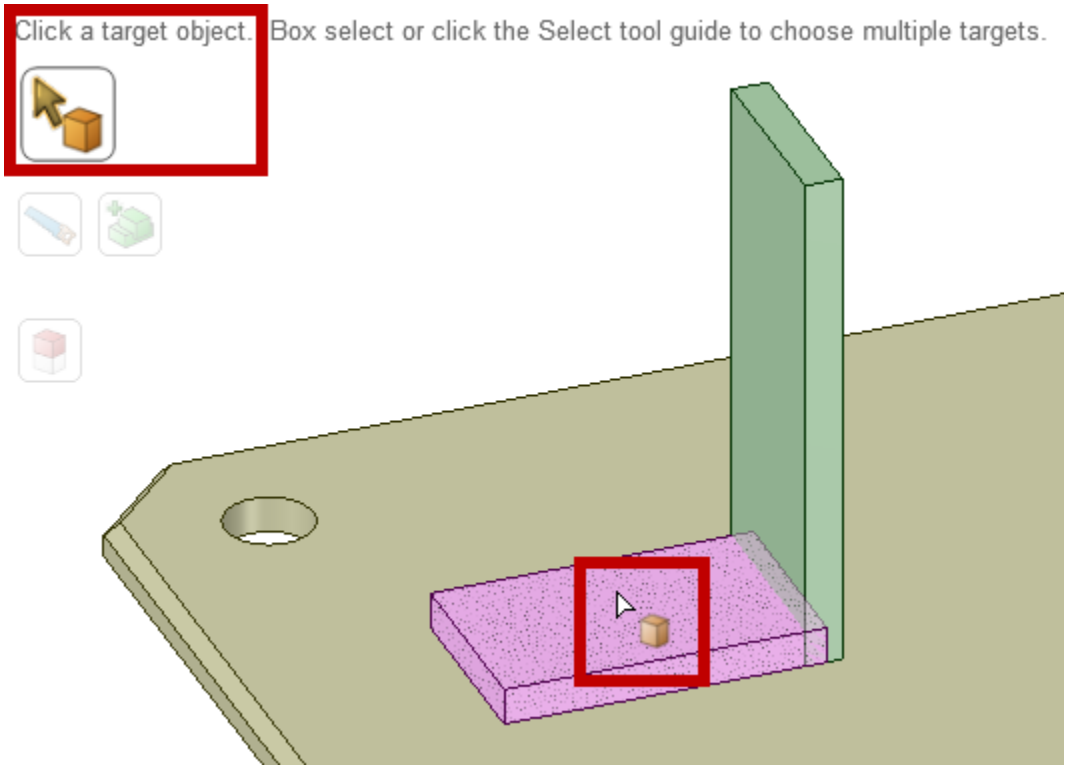
When merging, ask yourself 2 things:

- **What should the color of the merged object be?**
    - If the objects are the same color, doesn't matter which is chosen first
- **Which Component should the merged object be in?**
    - If both objects are in the same component (like they are for front_support) then it doesn't matter which is chosen first



You may find that Object 1 has the color you want, but object 2 is in the component you want. You have to choose which is more important (the color and component can always be changed. Components are covered in the Structure Tree section, which is next)

5. **Click the Pink Solid** to select is at the target object (**DO NOT CLICK THE TOOLGUIDE**)

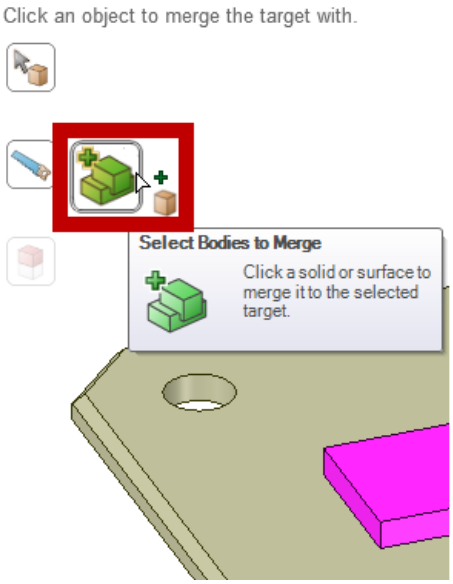


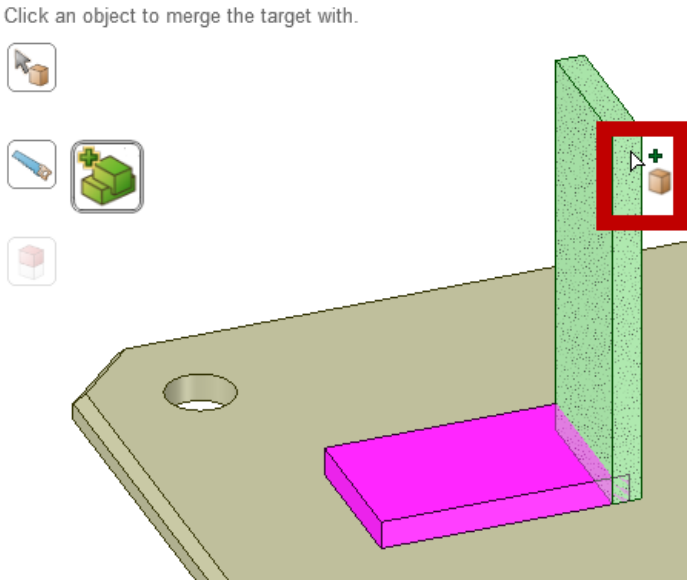
NOTICE as you hover over the Pink Solid, that the entire solid is highlighted, instead of just a Face like in the other Tools. The Combine tool works with entire objects, not individual Faces or Edges.

After clicking\selecting the Pink Solid, the Combine tool will automatically advance to the next ToolGuides.

NOTICE the target object is now brighter, and Combine switches to cutting mode. The Cutter ToolGuides is active (large), and a saw is attached to the mouse cursor.

6. **Click** the Right ToolGuides in the 2nd row of ToolGuides: the **Select Bodies to Merge ToolGuides**
7. **Click the Green Solid**.

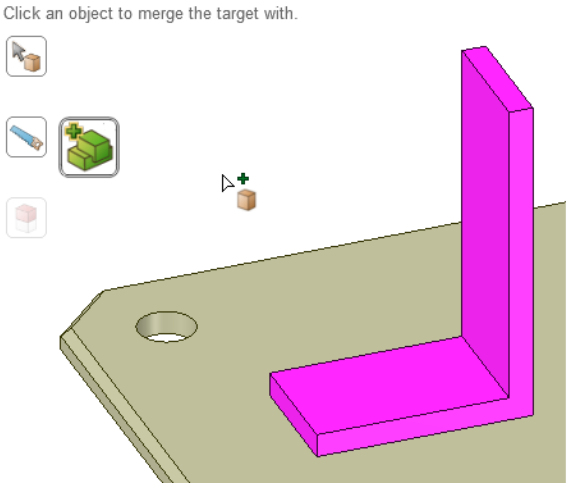




NOTICE

a. After you click the Green Solid and it merges into the Pink Solid, the newly merged solid is still highlighted, which means it is still the Target Object.

b. The Select Bodies to Merge ToolGuides is still active (Large) and there is still a solid with a "+" symbol attached to the mouse cursor. This means if you click something that is touching the Pink Solid, it will merge that body into the Pink Solid.

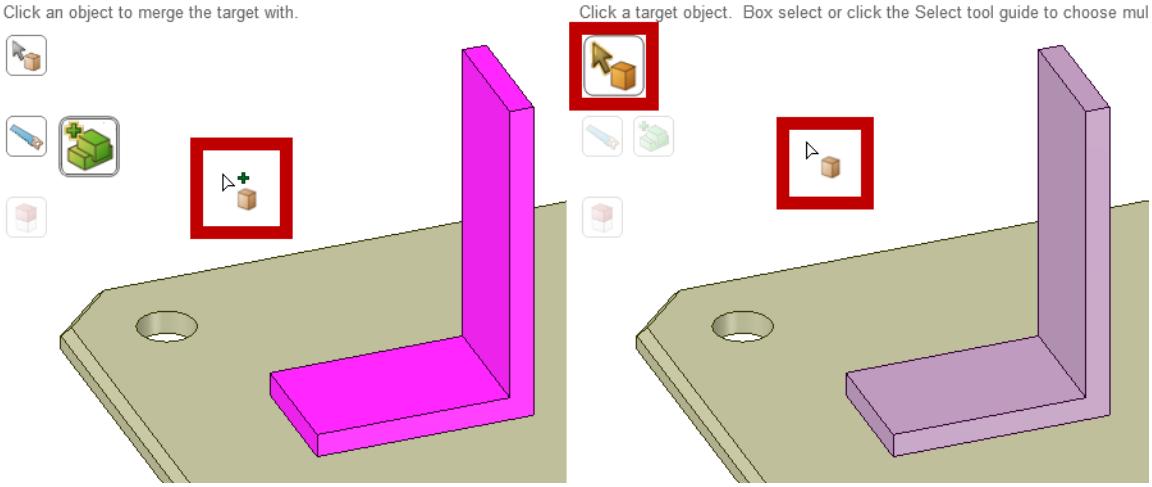


c. You will only be able to Merge bodies that are Interfering or have Faces that are touching the target, like the Base\Table

d. The Pink and Green Bodies were interfering\Overlapping before merging with Combine. After Merging, the interference is removed. There is no history of the interference in the new Solid.

NOTICE that the back_support was 2 solids, and is now 1 solid

8. **Click in White Space** to start the Combine tool over.



NOTICE

a. The ToolGuides has changed back to the first ToolGuides: Select Target Object.

b. The "+" symbol is no longer attached to the cursor.

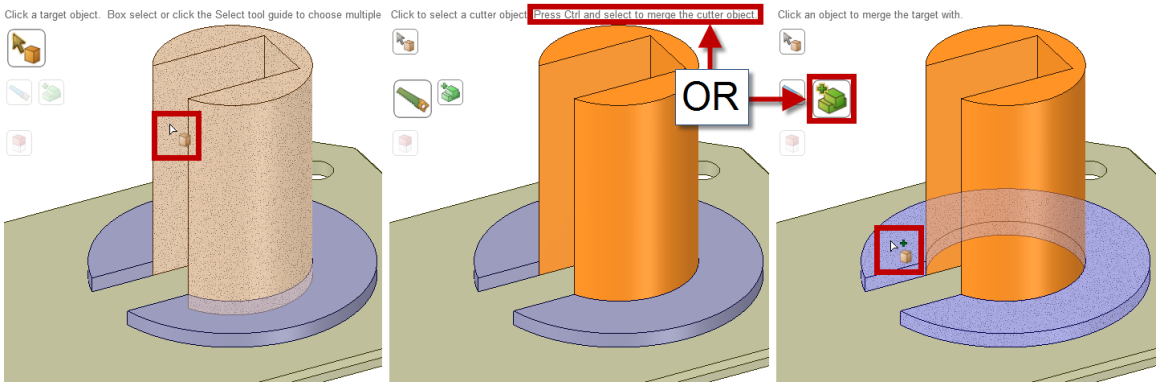
c. The Pink solid is no longer highlighted, meaning it's no longer the Target object.

Clicking in White Space resets the combine tool just like turning on the combine tool.

9. **Select the Cylindrical solid** as the Target Object

NOTICE the Status message says you can hold CTRL to merge

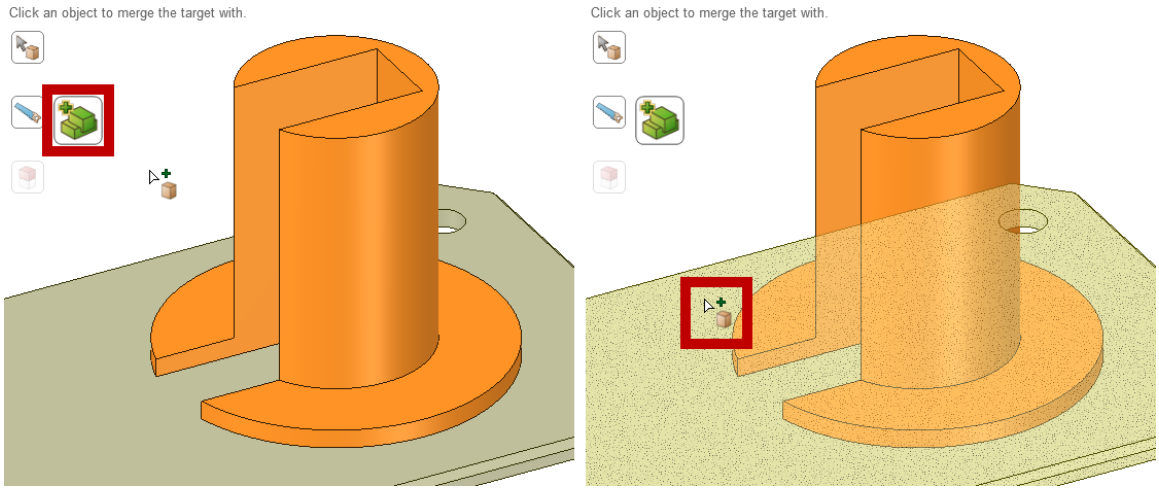
10. **Hold CTRL** and **click the Blue Solid** to Merge it into the Cylinder



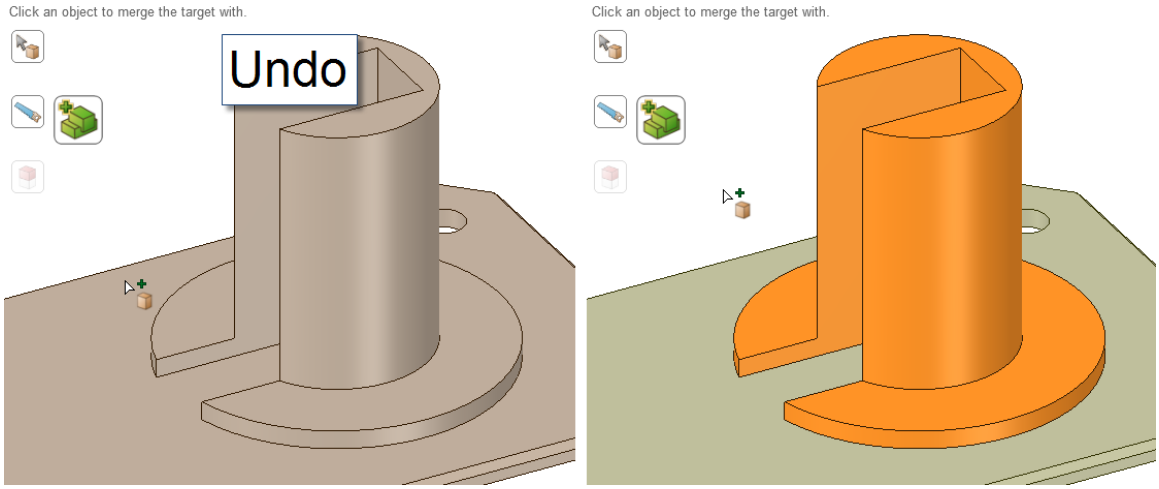
NOTICE that when you hold down CTRL, the Select Bodies to Merge ToolGuides becomes big/active.

NOTICE that after merging the Blue Solid into the Cylindrical Solid, the Select Bodies to Merge ToolGuides is still Active, and there is still a "+" symbol attached to the mouse cursor. This means SpaceClaim thinks you may want to merge additional bodies into the new solid, and makes it easier

11. **Click on the Base\Table**. It will Merge into the Target Object,



12. **Undo**

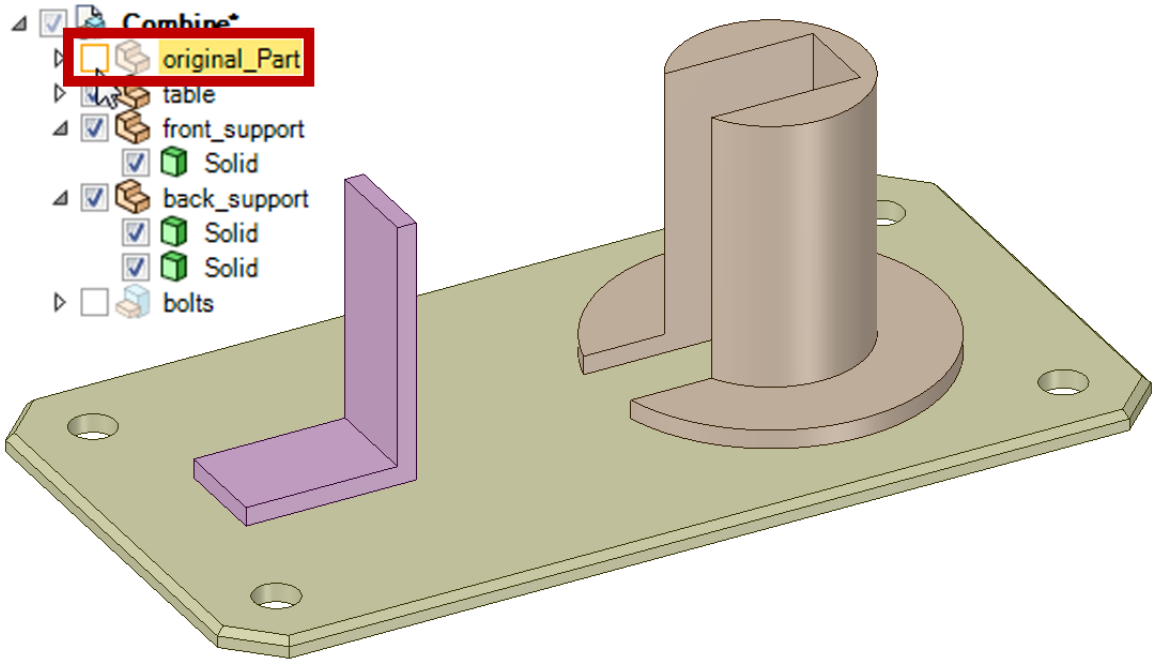


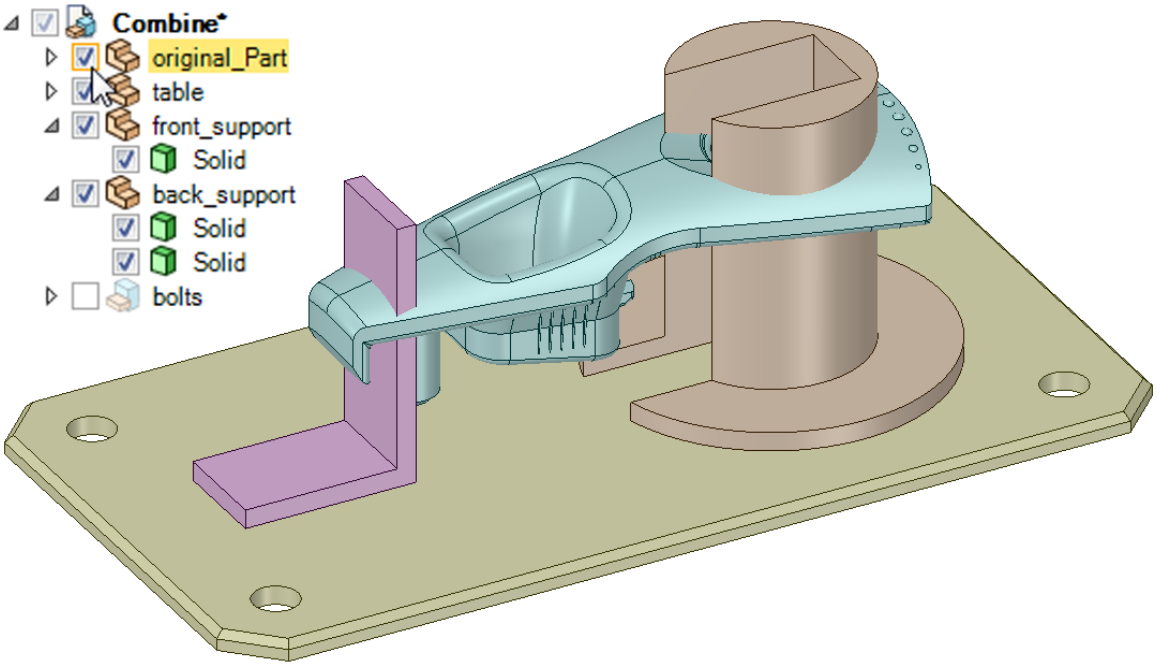
NOTICE that by clicking Undo once, only the last step of the combine tool is undone. The entire Combine operation is not undone, just the last step.

13. **Click in White Space** to reset the Combine Tool.

IMPORTANT It's usually good practice to click in white space before turning on Combine, or when you want to restart the Combine Tool

15. Click the Check Box in the Structure Tree next to Table to Show the Table





NOTICE how the original_part intersects\interferes with both the front_support and back_support.

The goal is to cut the front_support and back_support with the original_part, so that both supports have matching faces to the original part, to hold it perfectly for fixturing

Cutting\Splitting with the Combine Tool

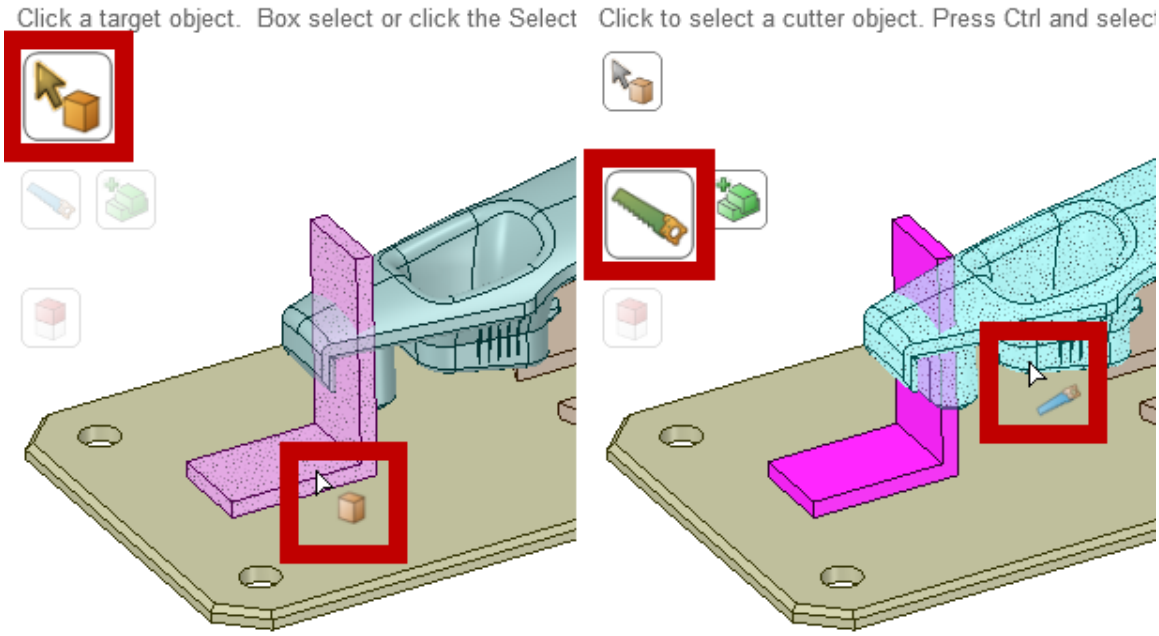
Which is the Target Object when Cutting\Splitting?

Just ask yourself: **Which Object do you want to cut? That is the Target Object**

IMPORTANT: do not click the ToolGuides unless you are instructed to do so.

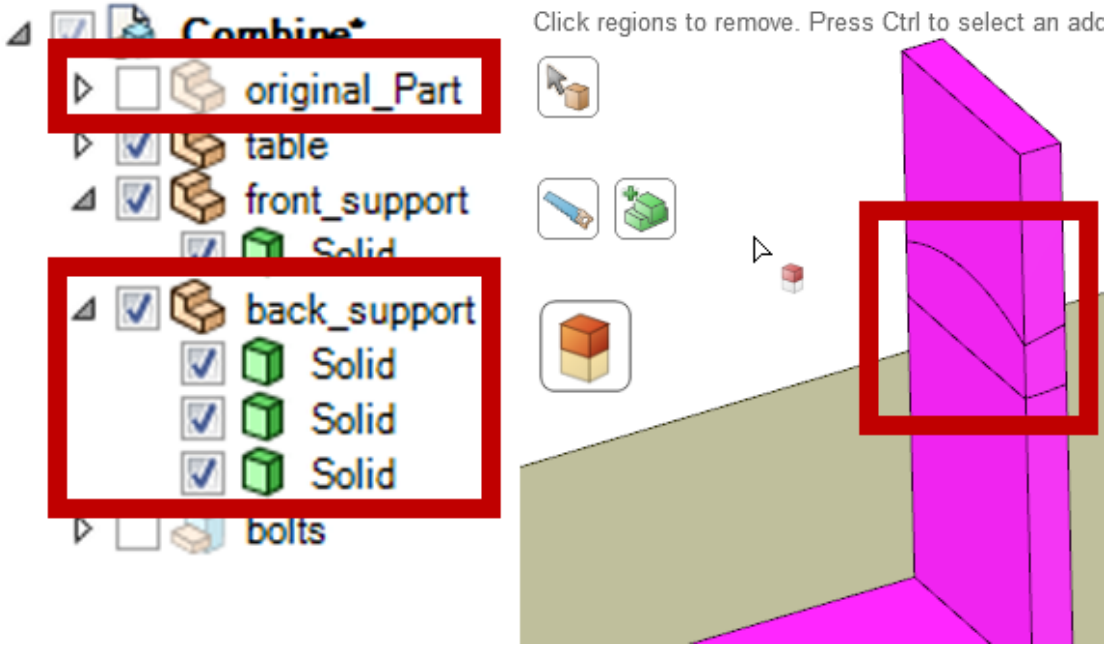
16. **Click the purple solid** (back_support) to select it as the Target Object
17. **Click the blue solid** (original_part) to select it as the cutter object

IMPORTANT: after clicking the blue solid as the cutter, don't click anywhere in the design window



NOTICE that after selecting the Target Object, the Cutter ToolGuides (looks like a saw) is large/active and there is a little saw attached to the mouse cursor. This means SC is ready for you to select a cutter

18. Immediately after clicking the blue solid as the cutter, **hide original_part** in the structure tree by unchecking it



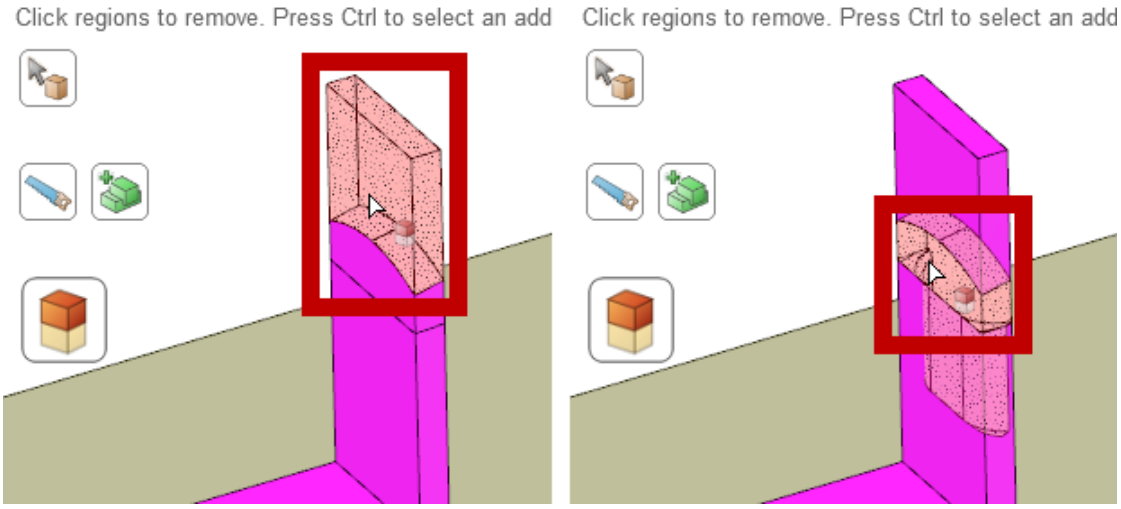
NOTICE

a. The back_support has changed from 1 solid to 3 solids in the Structure Tree

b. What appears to be new edges on the back_support where the original_part intersected

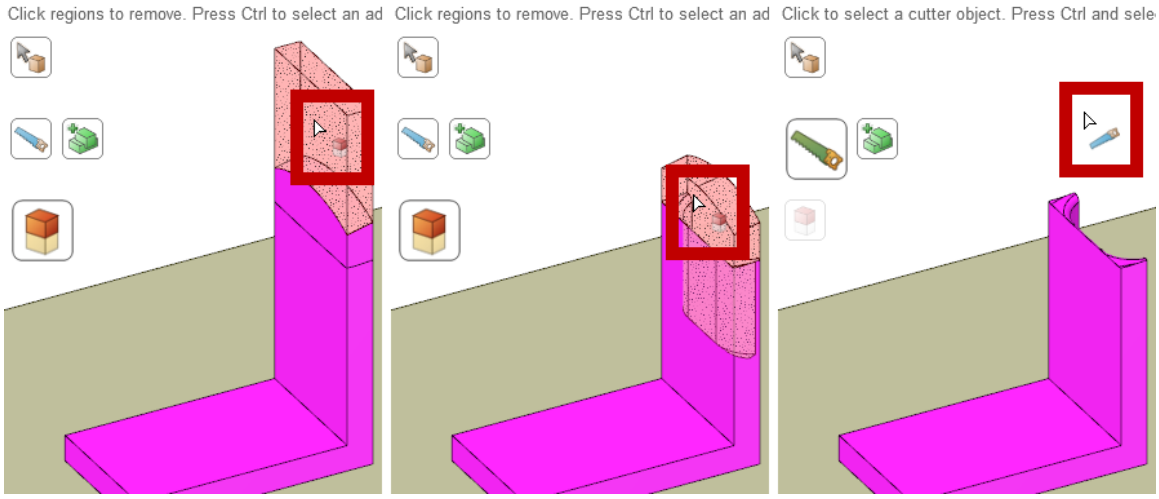
c. The Combine tool has automatically switched to the last ToolGuides, Select Regions to Remove. This ToolGuides is also attached to the mouse cursor

19. **Hover over the back_support solid**, above and below the new edges



NOTICE that the status message says Click Regions to Remove

20. **Click the Top Region of the Purple back_support**
21. **Click the Middle Region of the Purple back_support**



NOTICE how each region disappears after clicking it, both in the design window and the structure tree

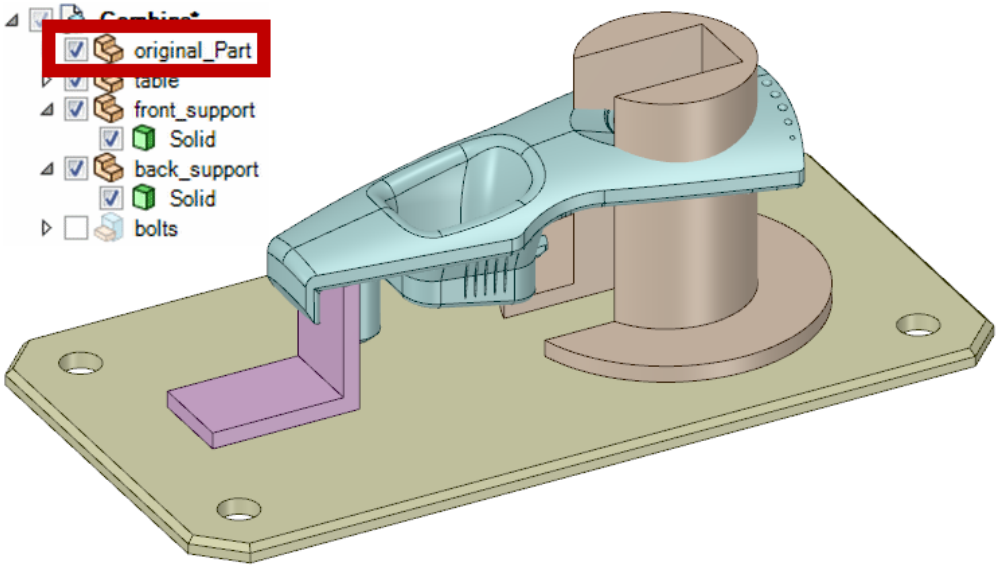
NOTICE that after clicking 2 of the 3 regions, the Combine ToolGuides automatically switches back to the Cutter ToolGuides, and the remaining piece of the back-support is still the Target Object

Combine is assuming 2 things:

a) You don't want to remove all 3 of the 3 regions created by cutting

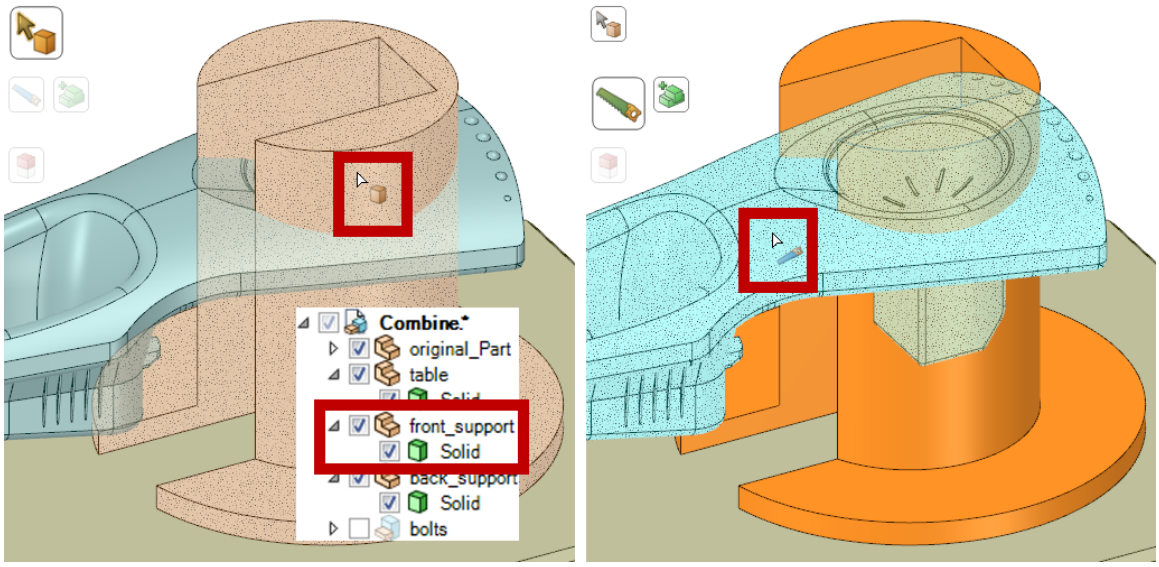
b) You may want to pick another cutter object to cut the target object again.

22. **Click in White Space** to restart Combine
23. **Show the original_part** by checking the checkbox in the structure tree



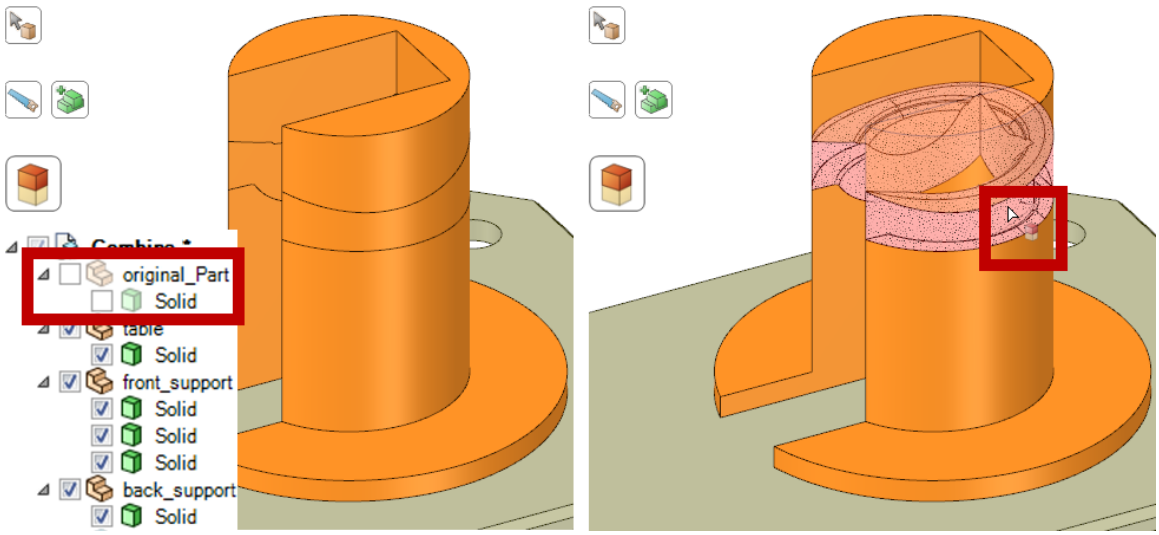
You will now repeat the same cutting on the front_support as was done the back_support

24. **Click the cylindrical front_support** while in Select Target Object mode
25. **Click the blue original_part** after combine automatically switches to Cutter mode

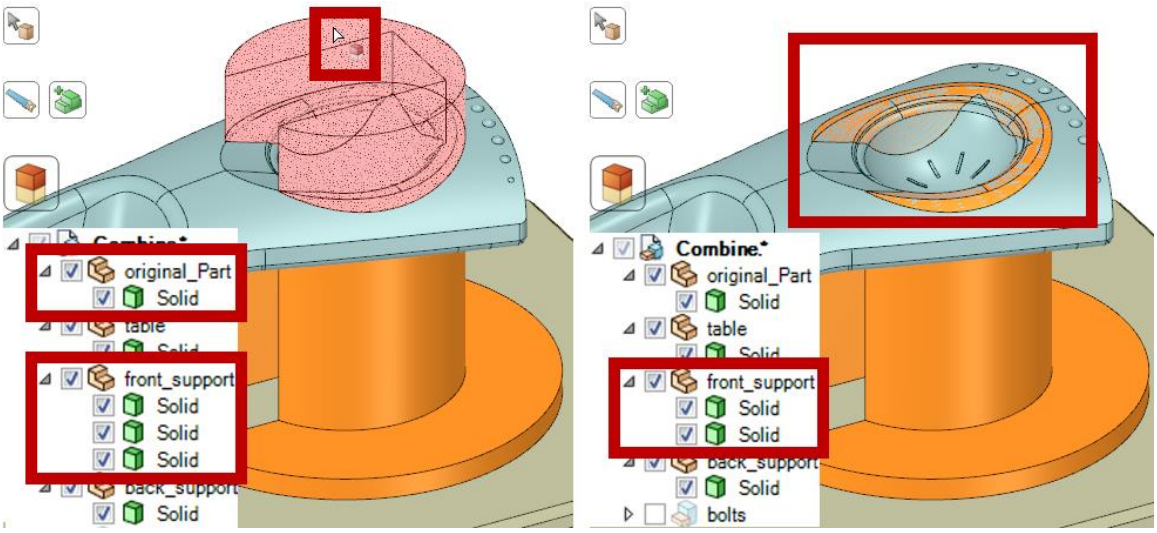


NOTICE that after cutting the front_support with the original_part, there are 3 solids under front_support in the structure tree

26. **Hide the front_support** in the structure tree by unchecking it and notice the 3 pieces the front support has been cut into
27. **Hover over the regions but Don't Click to Remove**



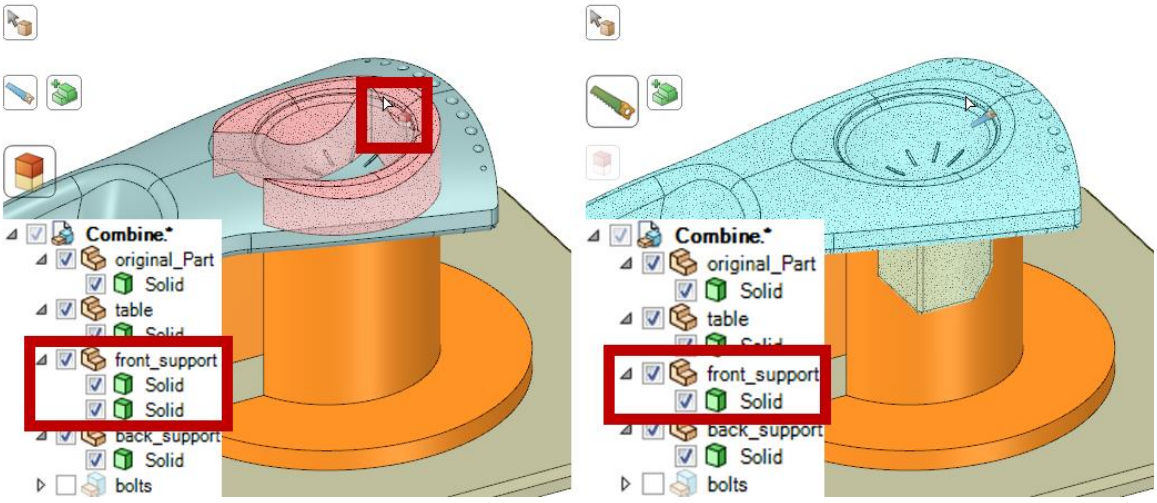
28. **Show the original_part** by checking the checkbox in the structure tree
29. **Hover over the regions of the orange cylindrical part**
30. **Click the Top Region** to Remove it



NOTICE how you can see the middle region inside of the Blue original-part

NOTICE how the front_support has changed from 3 solids to 2 solids in the structure tree

31. **Click the middle (**now Top**) region** to remove it



NOTICE how the front_support has changed from 2 solids to 1 solids in the structure tree

32. **Hide the original_part** by unchecking it in the structure tree.

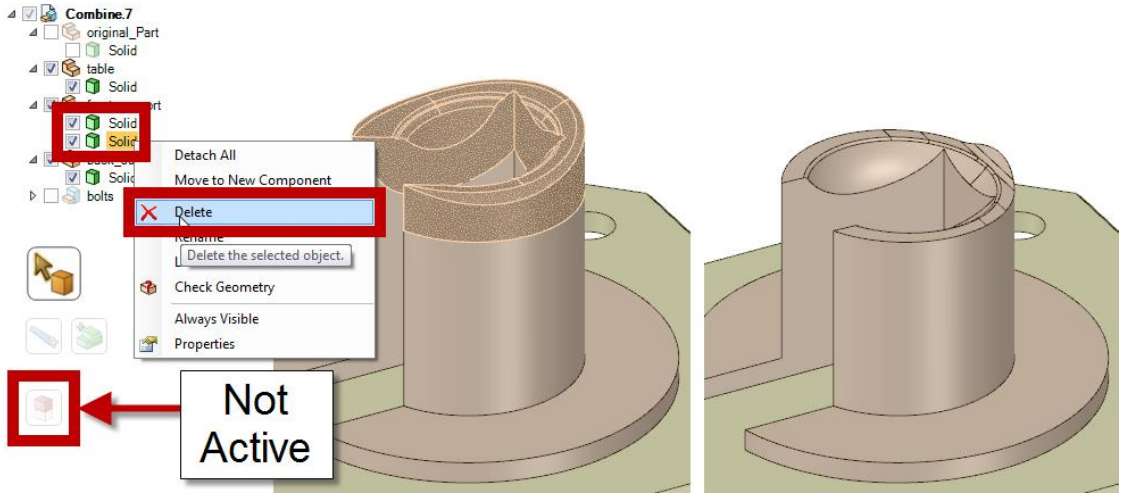
IMPORTANT: see the next page if the previous steps do no work as shown

After completing step 32, the back support should look like the right image below

If not, you can always Undo and try to repeat the steps, but you don't have to as long as the back_support was cut with the original_part

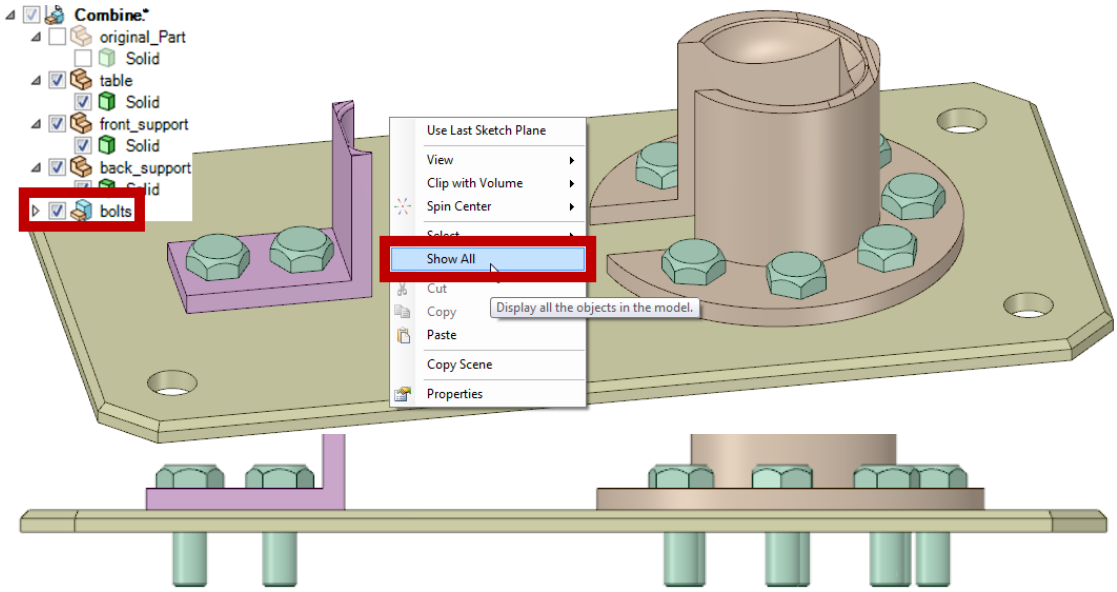
If the back_support has 2 or more solids in the tree, you can manually delete them by:

33. ESC out of the combine tool, or turn on select.
34. Hover over the solids under back_support until the upper section is highlight on the model.
35. **Right Click on that Solid in the Tree, and select Delete**.



Multiple Targets & Cutters; Sticky ToolGuides

36. Show the Bolts by Checking the box in the Structure Tree, or Right click in white space and Show All

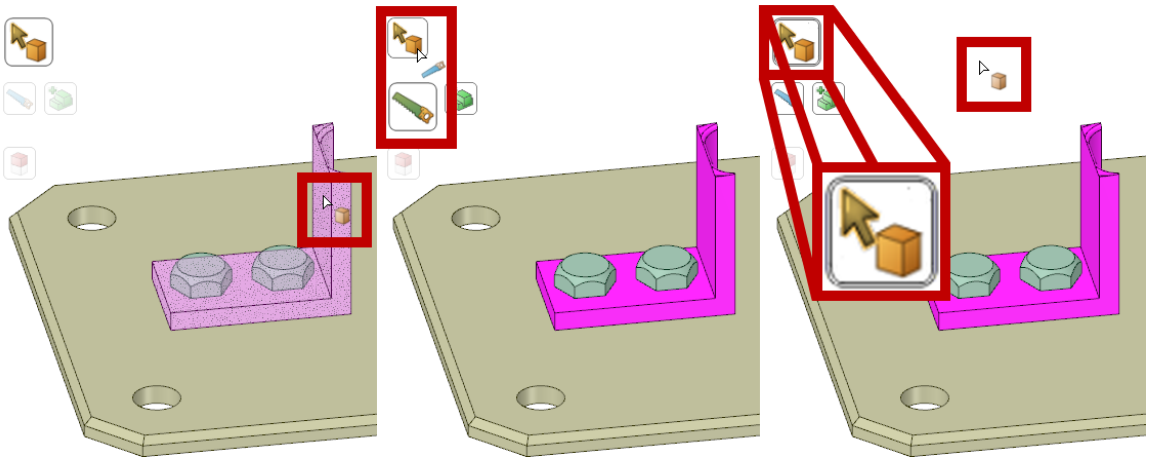


NOTICE how the bolts go all the way through the supports and the base\table

37. **Click in White Space and Click Combine**

38. **Select the Pink Solid** as the Target Object

39. **Click the Target Object ToolGuides** To select Additional Target Object,



NOTICE by clicking the Select Target Object ToolGuides it becomes Active\Large. Even after Combine Automatically switches to the next ToolGuides, you can click on the previous ToolGuides to reactive it. Also the Select Target Object icon is reattached to the mouse cursor

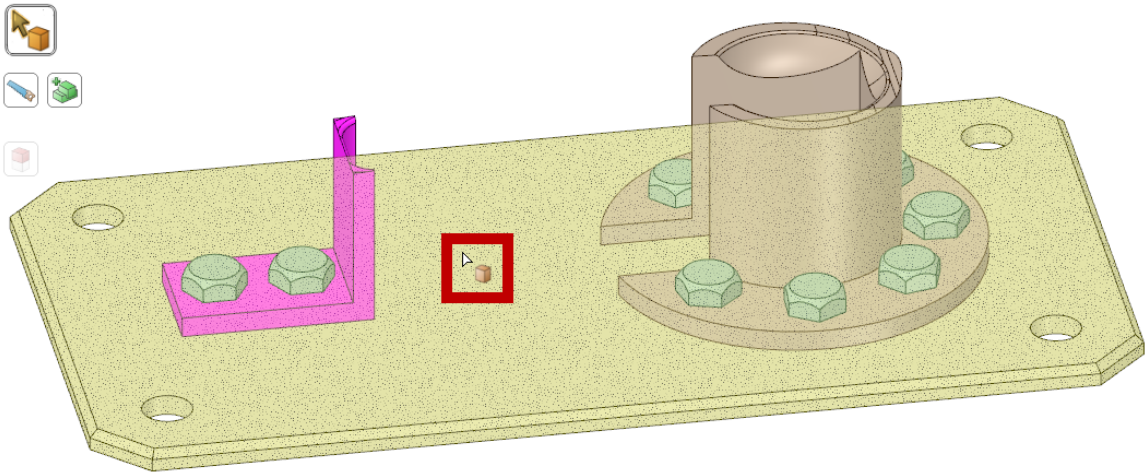
NOTICE that the Select Target Object has a Double Border around it. This means the Select Target Object ToolGuide is now **Sticky**

What is a Sticky ToolGuides?

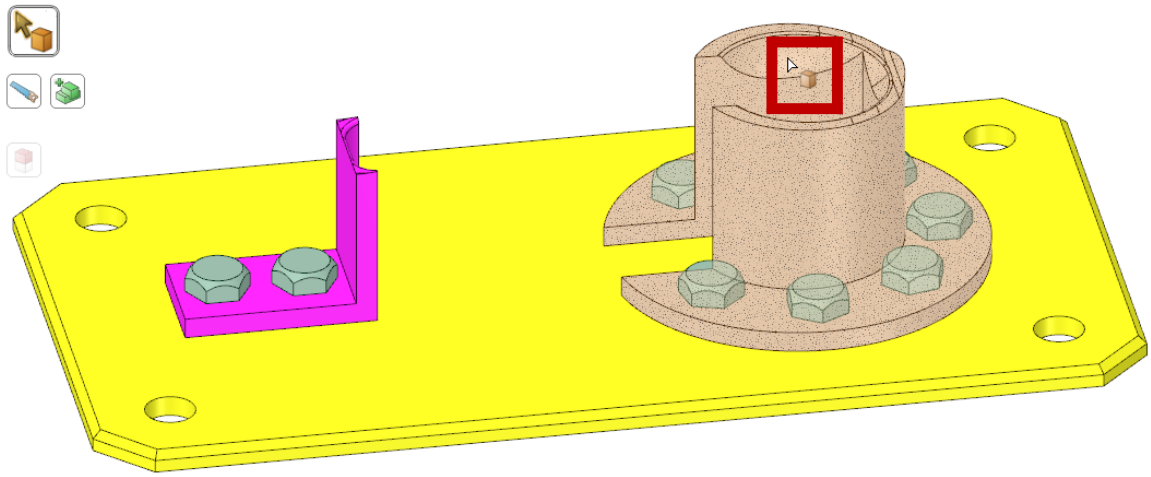
When a ToolGuides is Sticky (Double Border around it by clicking the ToolGuides) it means that ToolGuides is not locked. After clicking and object, the tool will not advance to the next ToolGuides

Not all tools have Stick ToolGuides. Typically only tools that Automatically Advance ToolGuides like the Combine tool will have ToolGuides that can be made sticky

40. **Click the Base\Table**



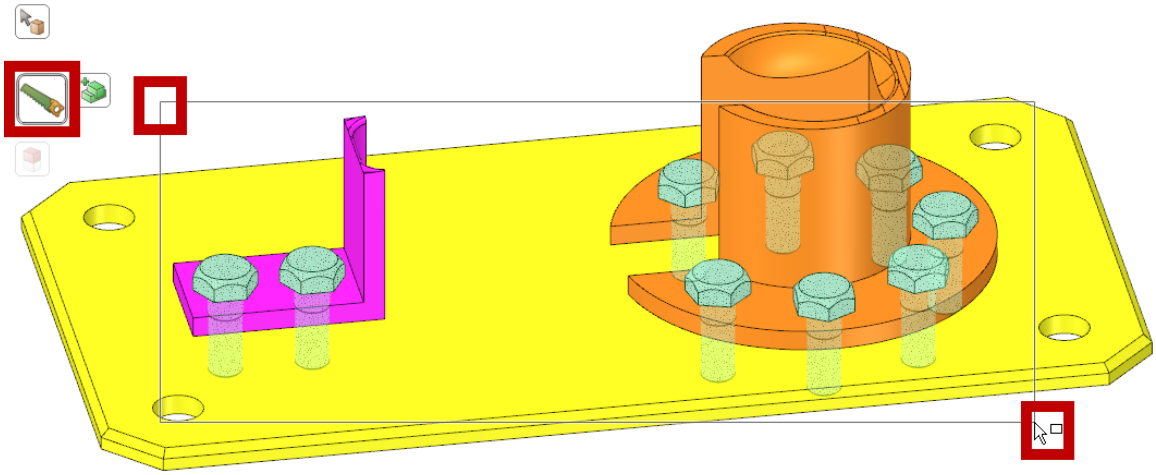
41. **Click the cylindrical front_support**



NOTICE that unlike before, the Combine Tool does not automatically switch to the next ToolGuides. This is because you clicked the Select Target Object ToolGuides, making it Sticky

42. **Click the Cutter ToolGuides** to Activate it

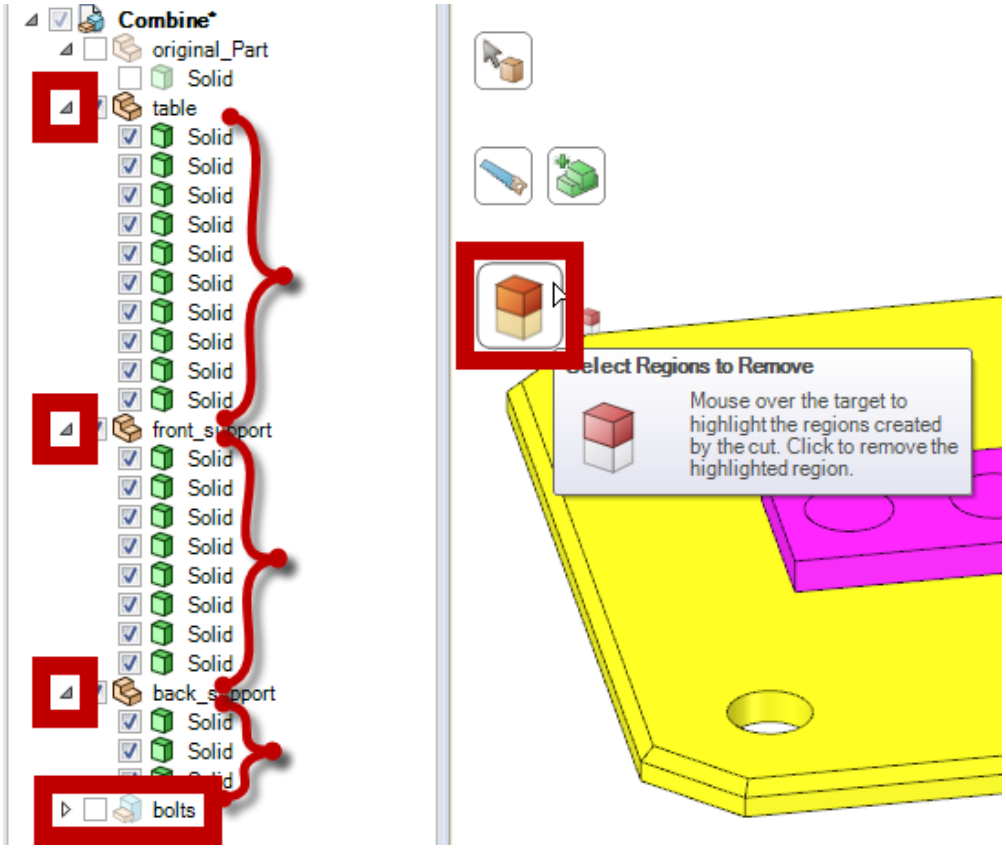
43. **Do a Left to Right Box Select** starting in the upper left, above the bolts and end below and to the right of the bolts



NOTICE it's ok if any of the Target Objects are completely inside the box. If there were objects that were neither the Target Object, nor supposed to be the Cutter, those objects should not be in the box.

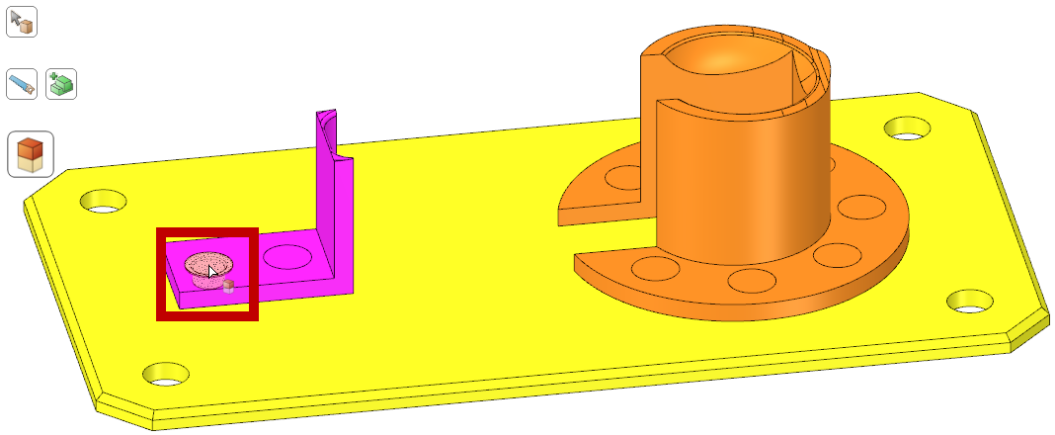
NOTICE That by Clicking the Select Cutter ToolGuides it is Sticky. After the Cutters are selected, the Select Cutter ToolGuides is still active because it is Sticky.

44. **Hide the Bolts** by Unchecking them in the Structure Tree
45. **Click the Select Regions to Remove ToolGuides** to Activate it
46. **Expand the table, front_support and back_Support** in the Structure tree by clicking the triangle next to them



NOTICE the table has changed from 1 to 10 solids, the front_support from 1 to 8 solids and the back_support has changed from 1 to 3 solids

The previous method of clicking each region can still be used to remove the bolt cut out as shown BUT…

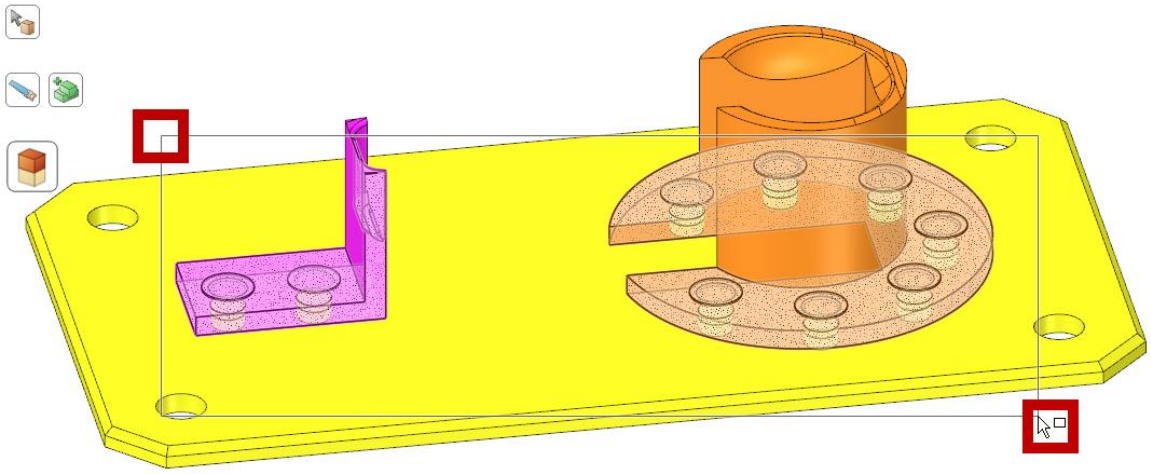


Instead of clicking each of the 18 regions to remove, let's see how to do a box select to remove them all at once

IMPORTANT: the size and position of the Box Select for the next step is very important

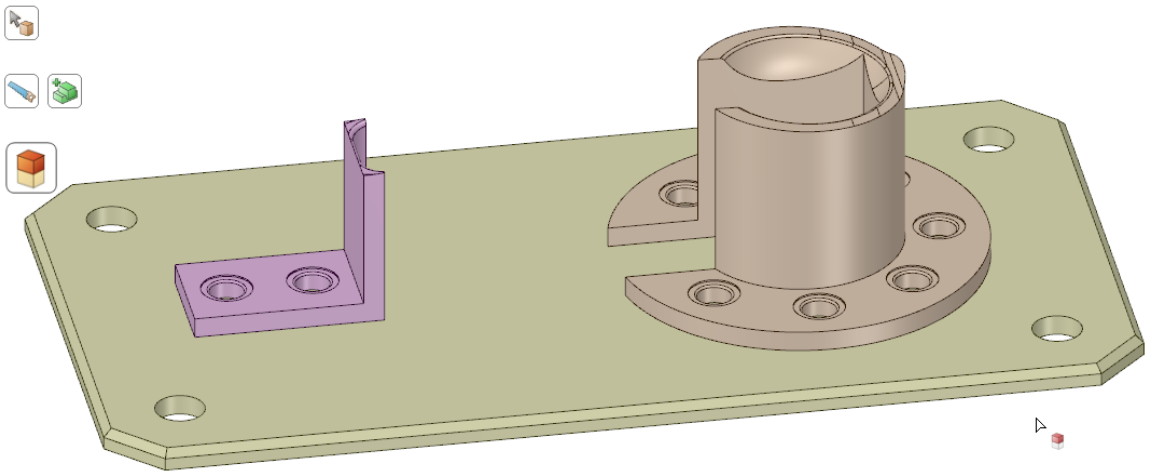
47. **Do a Left to Right Box Select** that goes around all the cutouts from the bolts, but **Does Not go entirely around the Purple, Yellow and Orange Solids** (or they will be removed too)

IMPORTANT: If you can tell the Box Select isn't correct, either press ESC before letting go, or let go and undo

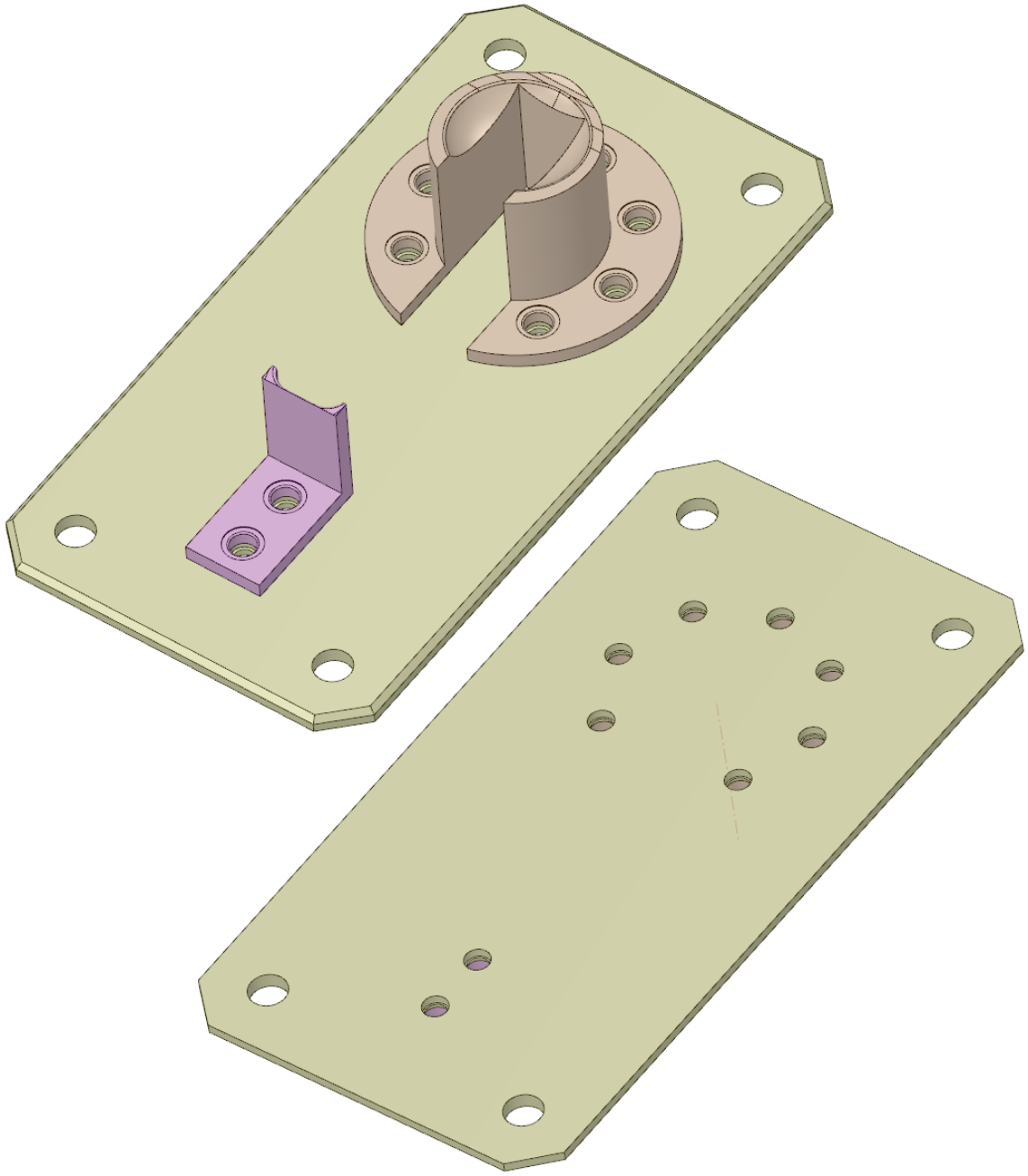


NOTICE that Purple, Yellow and Orange Solids are NOT completely inside of the Left to Right Box Select. If they were, they would be removed.

If the main part of the Yellow, Orange or Purple Solids are removed, undo and try again



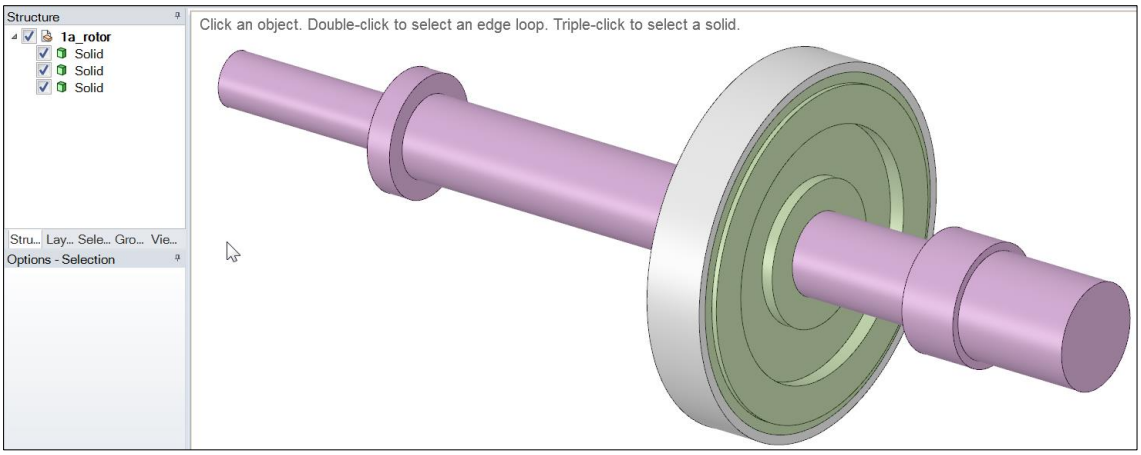
48. **Press ESC or turn on the Select Tool**. The model is all cut up and merged properly.



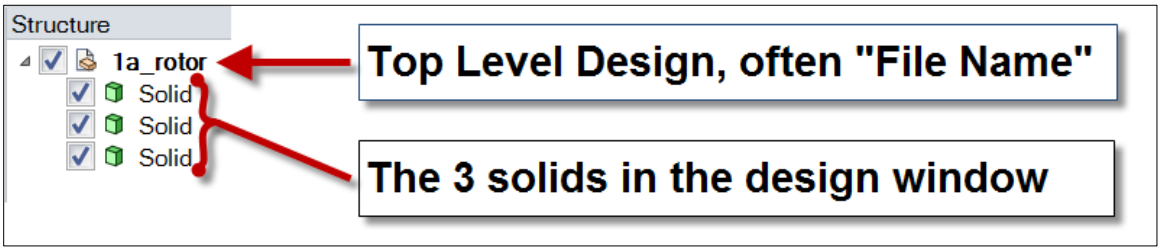
Structure tree

The Structure Tree is used to organize Objects in a design. The Structure Tree in SpaceClaim is found where typical CAD systems have a feature or assembly tree. Since SpaceClaim does not have features, you will not use this panel to change the design directly. Changes made to the geometry are done by interacting with the model itself. The Structure Tree allows you to see what objects are contained in a design, which are internal to the design or have external file references, and how they are organized. We will explore the hierarchy of the Structure Tree, what all the different icons mean, how to find objects in the structure tree, how to reorganize, and create new object in the tree.

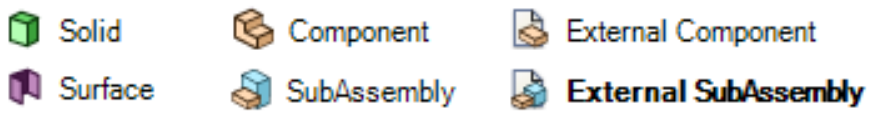
1. **File\Open,** Desktop\SpaceClaim_Basic_Training**\06_Basic_Structure_Tree_2014.0** and open **1a_rotor.scdoc**



NOTICE: There are 4 objects in the Structure Tree



The **top row** for this opened file is the Display name, which typically represents the file name, but can be different. The top row is often referred to as the **Top Level Design**. Notice the 3 Objects below the 1a_rotor are indented inward. This means the 3 solids are objects within, or part of the 1a_rotor.

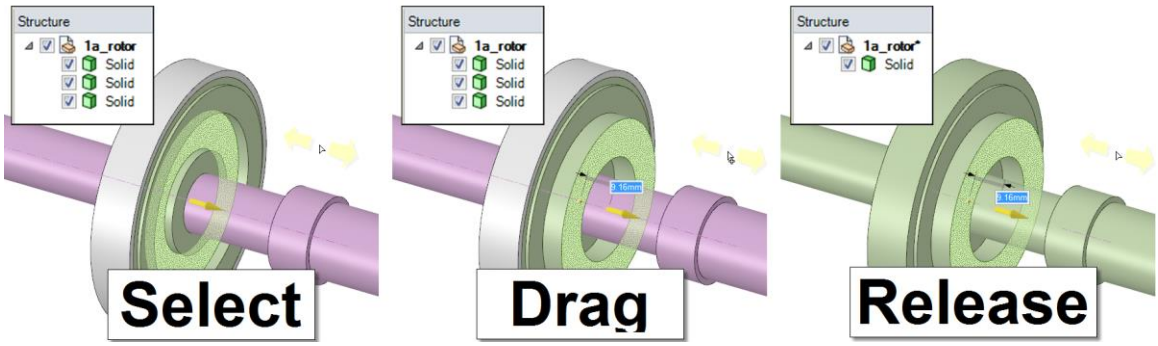


Above are the 6 main Structure Tree Icons. **The Bold text represents the Active Component**. We will discuss what all of these icons along with what the active component means in this section.

First, let's find out what happens when we edit solids that are:

A. Not in components
   AND
B. Interfere or are in contact with other solids

2. Select the face indicated below, turn on Pull and Pull it left or right



NOTICE Before Pulling, there are 3 solids. After Pulling, the 3 solids merge into each other. This is because

a. the solids are in contact with one another,

b. And they are not in separate components.

You already know 1 way for the solids to not merge together:

3. Press the **Undo** button or Ctrl-Z
4. Click on the face, and select the **No Merge** icon from the Mini-Toolbar
5. **Pull** the same face again. The solids do not merge

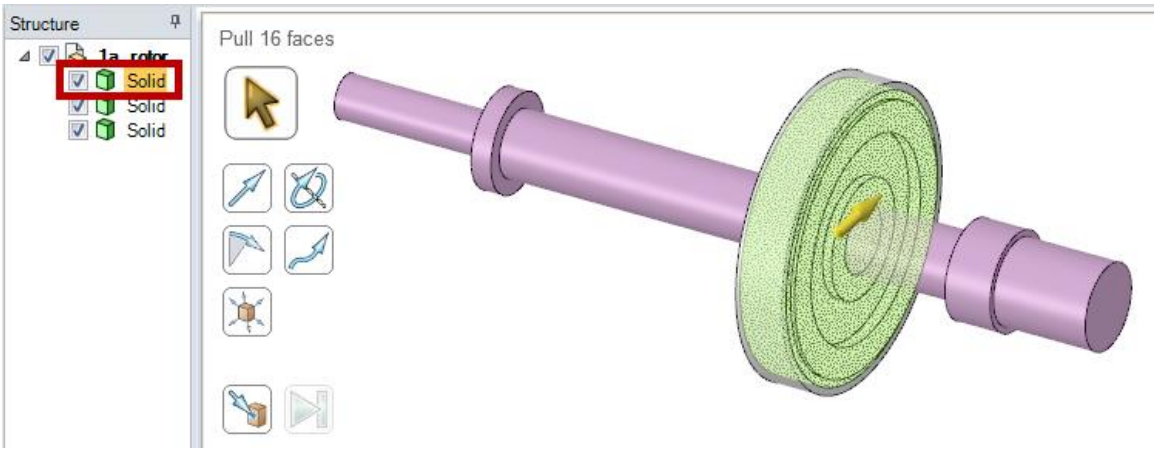
However, we don't want to have to select No Merge every time we want to edit a solid in an assembly. Let's find out the advantages of Components

Components

- **A Component is an Object in a design**, that contains any number of other objects, such as
  - Solids
  - Surfaces
  - Curves
  - Other Components
  - Datums
  - Assembly Conditions
- **Components allow an assembly** to retain and adhere to assembly relationships
  - Without Components, assembly relationships are temporary.
- **Components allow for multiple external files** to be in the design.
  - Components can be internal or external to the design
- **The Top Level Design is a Component**.

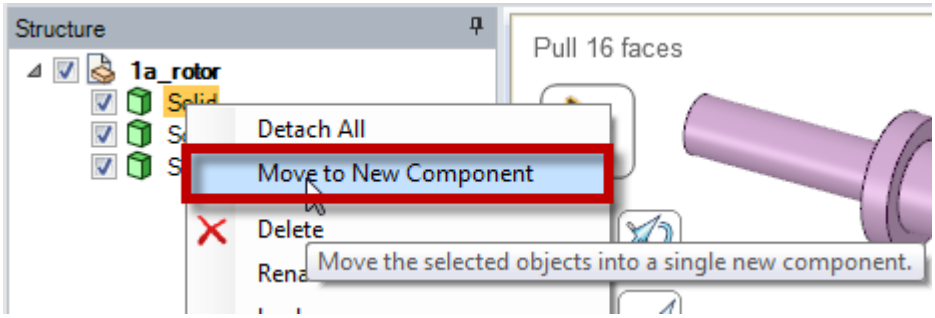
Creating Components

6. **Select the first solid** in the structure tree.

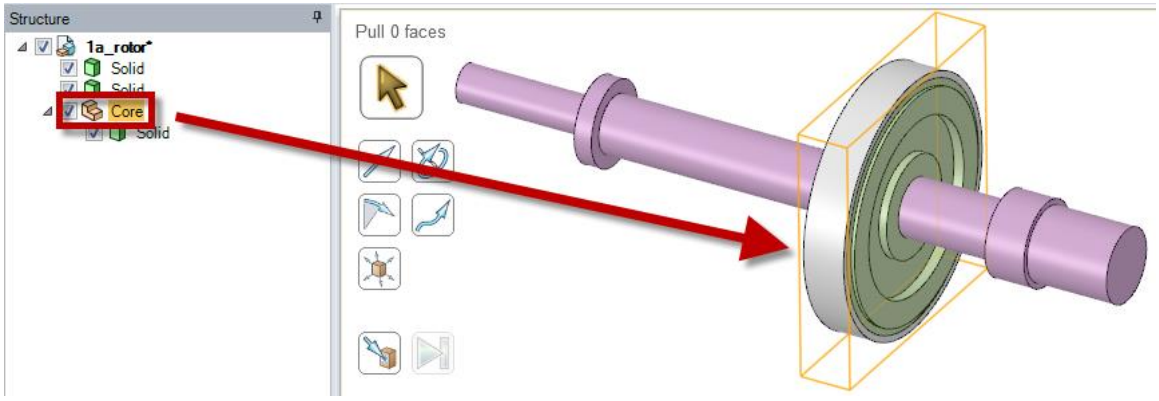


The entire green solid, all of its faces, should become selected.

7. **Right Click** on the first solid in the Structure Tree and select **Move to New Component**.



8. **Type Core** for the Component name
9. **Click the Core** Component in the tree

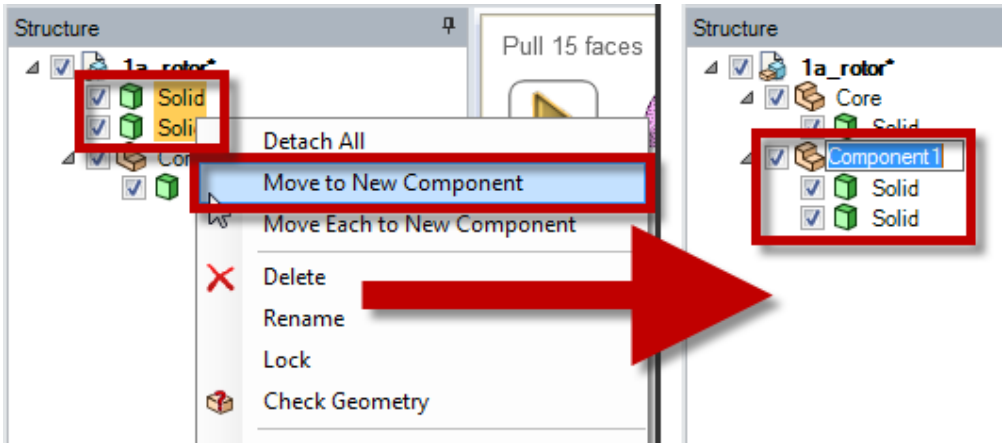


NOTICE: when a component is selected, an orange wireframe box is shown around the solids in the component, instead of the solid itself being selected.

NOTICE the solid underneath the core is indented, indicating the solid belongs to the Component.

10. **Select the 2 other solids** by clicking one, hold CTRL and click the other. **Right Click** either, and select **Move to New component**

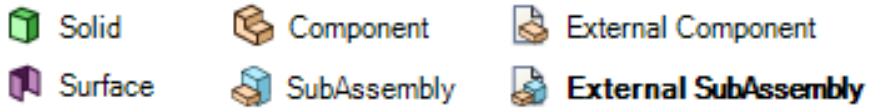
NOTICE: Both solids are now in the same component. This is not usually recommended.
It is Typical to for each solid to be in its own component, even if you plan to combine them into 1 solid.



11. **Undo** back (CTRL-Z or Undo button)

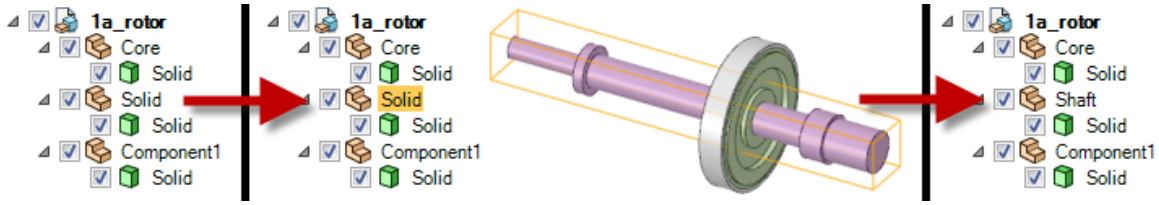
12. **Select the 2 solids** again, **Right Click**, and this time select **Move Each to New Component**

IMPORTANT: The name in the tree can be misleading. Notice that the highlighted Component 2 images below is called "Solid." It is NOT a Solid, it is a Component. Pay attention to the Icon, not the Name. Here is a reminder of the Icons

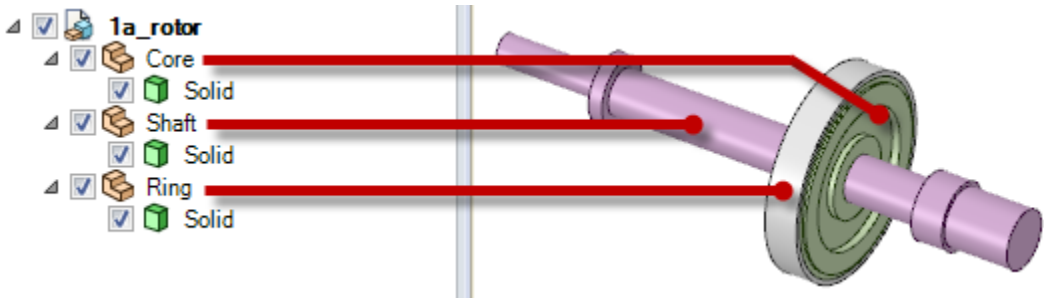


13. **Select the component** that puts a box around the long **purple solid**.

14. Either Right Click on the Component name and select **Rename** or do a very slow double click on the components name, and name this component **Shaft**

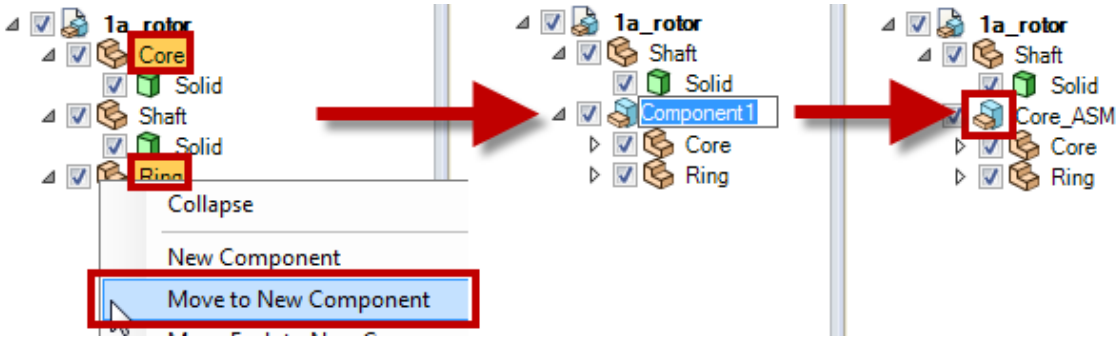


15. **Repeat** on the last unnamed component, and name it **Ring**. It is the grey solid.



While it is not typical to have more than 1 solid in a single component, it is very common to have multiple components inside of another component, automatically called a Subassembly

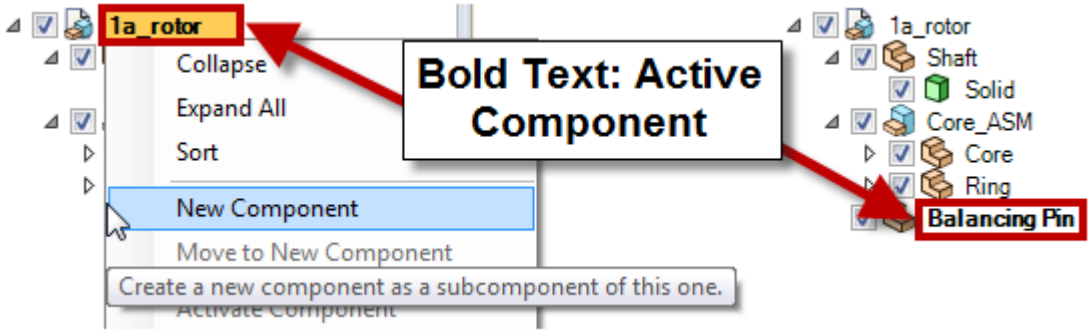
16. **Select the Core and Ring** Components (Click Core, hold CTRL & Click Ring in the structure tree).
17. Right Click and select **Move to New Component**.
18. **Rename** the Subassembly **Core_ASM**.



Any component with 1 or more components in it is a Subassembly

While working in an assembly, you might decide you want to create a new solid inside of a new component, and you don't want to edit the existing components.

19. **Right Click** on the Top Level Design, 1a_rotor in the tree, and select **New Component**, and type in **Balancing Pin**



NOTICE: The text of the New Component "Balancing Pin" is bold. In all the previous steps, the Top Level Design 1a_rotor text has been bold. Bold Text indicates the **Active Component**.

The Active Component

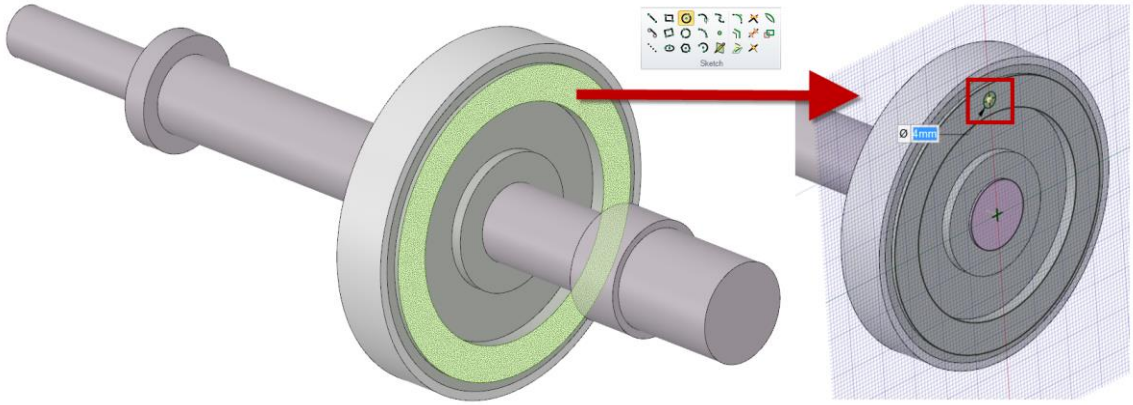
- T**he Active Component is where New Objects in the tree are created**
  - (I.e. sketches, surfaces, solids, components etc.)
- **The Active Component indicates what objects we can directly edited**.

So far, and most of the time, the Top Level Design is the Active Component. To understand the active component better, let's see what happens when the Top Level Design is not the Active Component

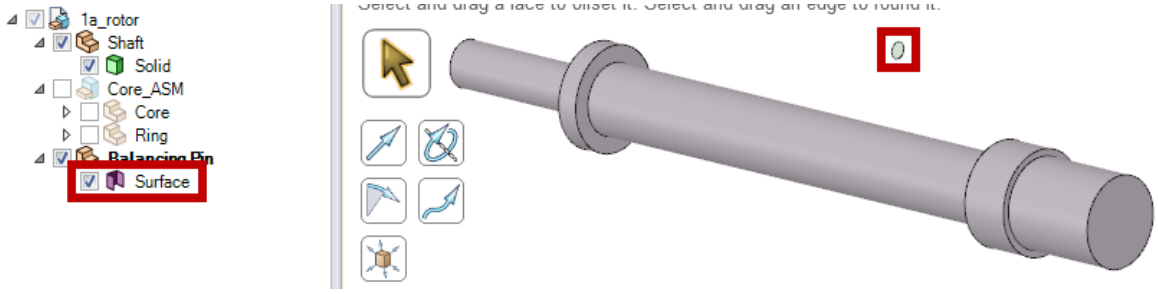
20. The new component, **Balancing Pin is the active component**, confirmed by bold text.

NOTICE that the colors of the existing solids are faded; this is because they are NOT in the active component.

21. **Select the face of the Core** indicated below, and turn on the **circle tool.** Sketch a **4mm circle** on the top portion of the face



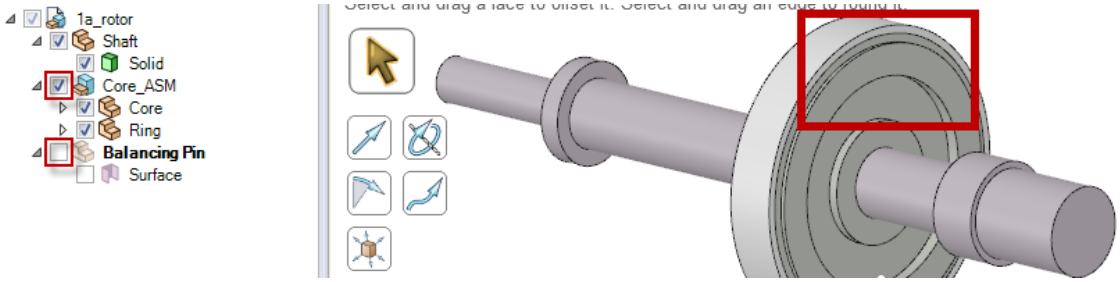
22. Turn on the **Pull Tool**. This will exit sketch and bring us back to 3D mode.
23. In the structure tree, **Expand the Balancing Pin** by clicking the arrow on the left side of it
24. In the structure tree, **Hide the Core_ASM** by unchecking the box next to it.



RECALL: In the Pull section, when we just had 1 solid and the top level design was active, sketching on the face of the solid created a new imprinted face on the solid

NOTICE that when sketching on the face of an inactive component, the sketch turns into a surface. Also, the surface is created under the current Active Component, Balancing Pin, instead of the top of the tree.

25. **Hide the Balancing Pin** and show the Core_ASM

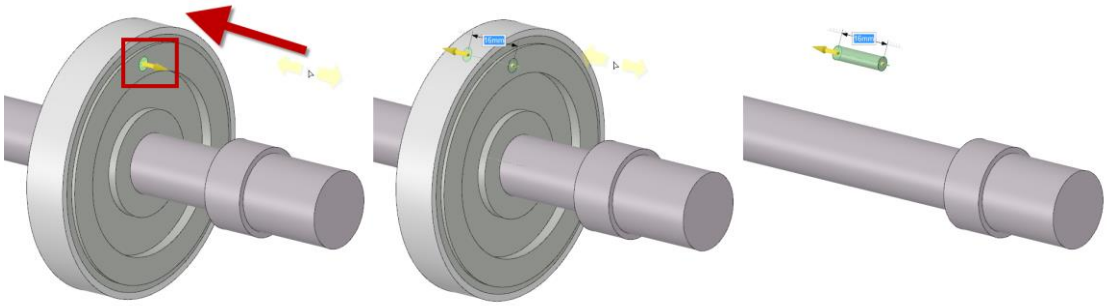


NOTICE: that no change has been made to the Core, because it is not in the active component.

26. **Show the Balancing Pin** by checking it in the Structure Tree

27. With the Pull tool still on, select the **face** of the surface**, and pull it to the left**, through the Core

28. Hide the Core_ASM

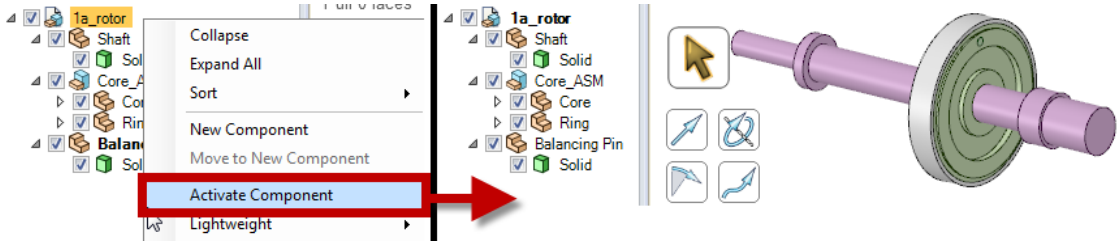


NOTICE: Normally when we pull a surface in the opposite direction of the arrow pointing from it, it will cut anything in its path. When there are only inactive components in its path, there is nothing to cut, and instead it will add. Also notice the New Solid is under the Active Component

29. **Show the Core_ASM**

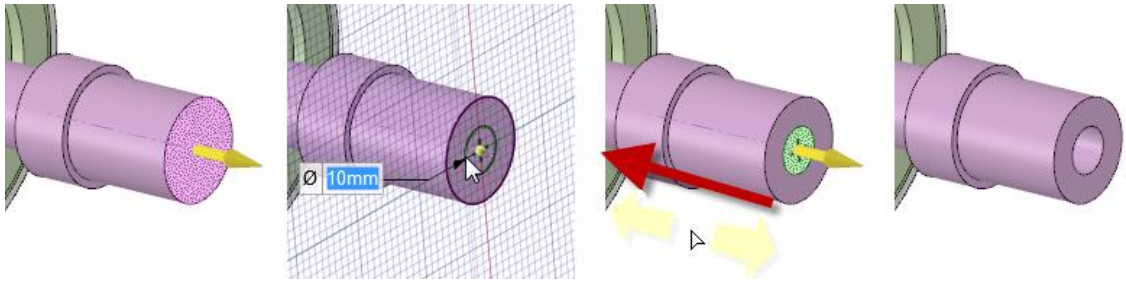
In addition to New Components automatically becoming the active component, we can choose which component is active

30. **Right Click** on the Top Level Design, **1a_rotor** and choose **Activate Component**

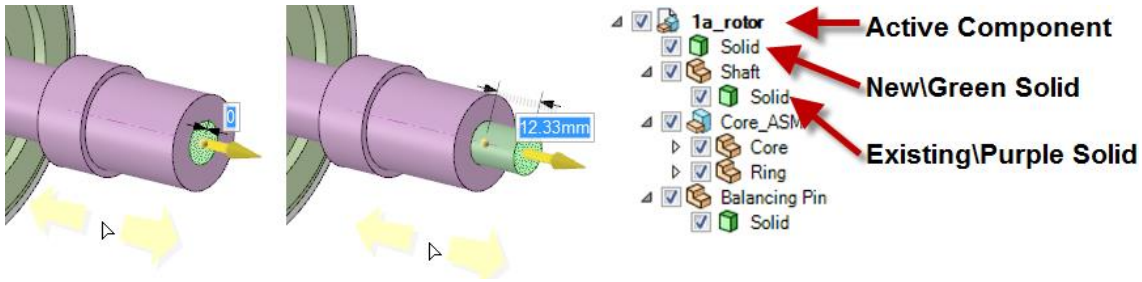


NOTICE that the Top Level Design is now bold and the colors of the solids are now vivid again.

31. **Select the face** shown on the end of the shaft,
32. Turn on the **circle tool**
33. **Sketch a 10mm circle** at the center
34. Turn on Pull, notice there is a new surface at the top of the tree, under the active component
35. select the face of the new surface and drag it to the left



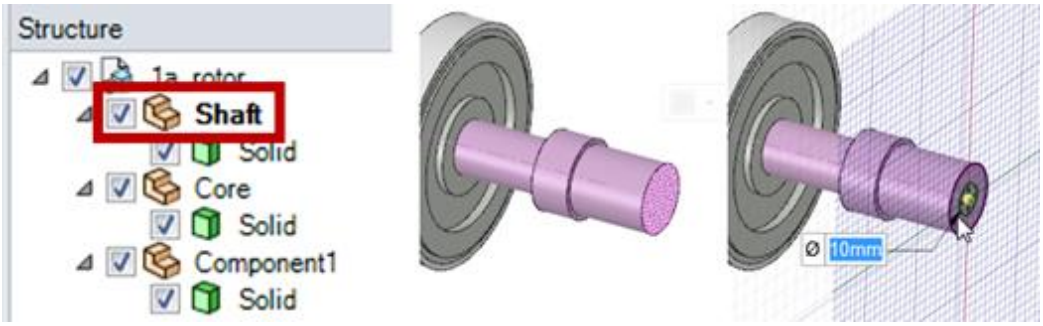
36. **Undo** back to before pulling the cut
37. **Pull** the surface to the **right**, to create a pin



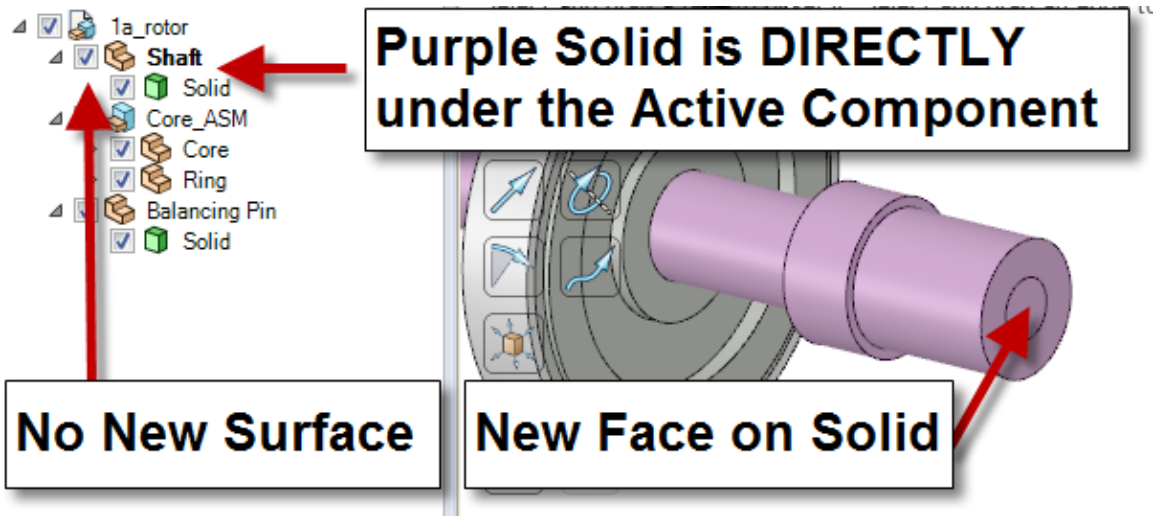
Why were we able to directly Cut the purple shaft, but when we added, it created a new solid???

NOTICE: that with the top level active, everything in the design is in the active component. Anything under the active component can be CUT when you pull in the opposite direction of the arrow. However, when a new solid is created, it's created DIRECTLY under the active component

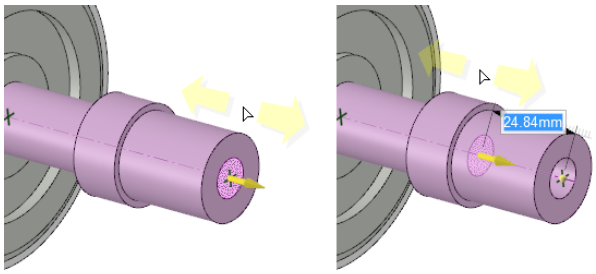
38. To further understand this, Delete the new solid at the top of the tree: Right Click & select Delete
39. Right Click on the Shaft and choose Activate Component. Only the Shaft's color should be vivid
40. Select the face shown on the end of the shaft, Turn on the circle tool, Sketch a 10mm circle at the center



41. Turn on **Pull**, notice there is NO new surface in the structure tree.



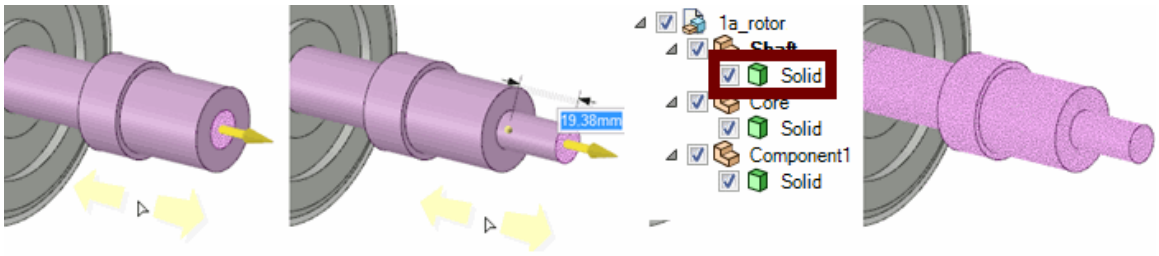
42. Select the face of the new surface and drag it to the left, it will cut just like before



43. **Undo**

44. Select the new face and pull to the right.

45. Click the Solid in the tree under Shaft



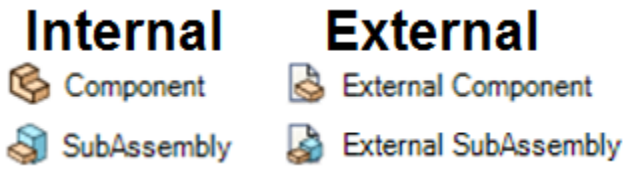
NOTICE the new pin is the same color as the shaft, and the entire shaft, including the new pin is highlighted after selecting the solid from the tree. This proves it is 1 solid

This happened because we sketched on the solid DIRECTLY under the active component.

46. Right click on the Top Level Design, 1a_rotor in the tree and select Activate Component

Internal and External components

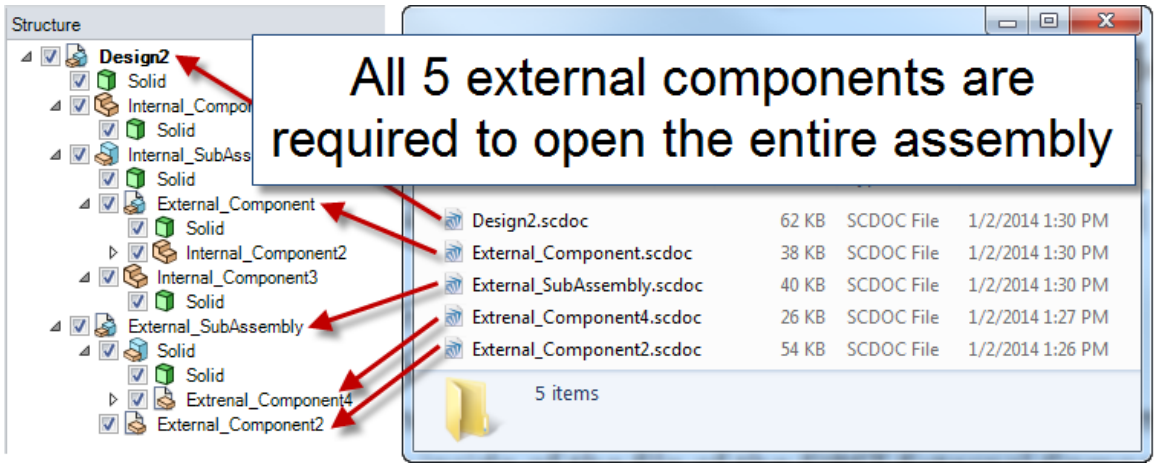
The concept of Internal and External applies only to Components and Subassemblies. It does not apply to geometry like solids, surfaces and curves. First let's take another look at what the icons look like for internal and external objects:



The difference is that the External icon has the Document symbol behind it

If you look at the Structure tree, notice that only the Top Level Design, 1a_rotor has an external icon. All the other components have an internal icon. What does this mean?

- **External Components** have a file associated with it
  - In this case there is the file 1a_rotor.scdoc
  - Editing the original file, will update all assemblies that file is an external component
  - SCDOC is the only file type in SpaceClaim, , it is for parts, assemblies and drawings
- **Internal Components** DO NOT have a file associated with them.
  - They are inside of the file of the FIRST External Component above them in the structure tree



NOTICE how many combinations of Internal and External Components and Subassemblies there are. An internal OR external Subassembly can have any combination of Internal AND external Components AND Subassemblies

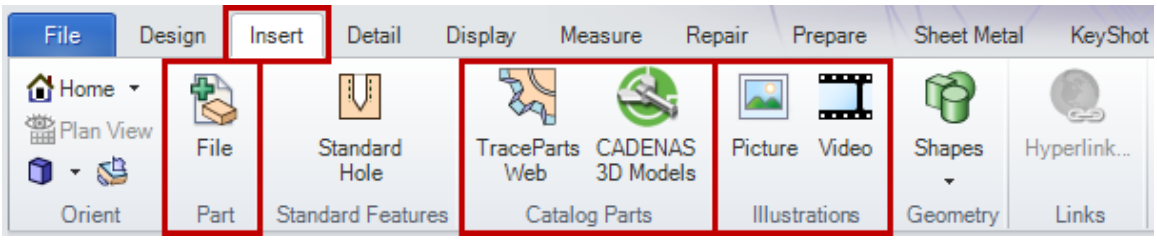
BY DEFAULT,

- Components **created** in SpaceClaim are internal, because there is no separate file associated to it.
- Components **inserted** into SpaceClaim are External since there is already an associated file.

Inserting Components

Next to the Design Tab, there is an **Insert Tab**. The Insert Tab allows you to insert objects into the design, like other 2D & 3D CAD parts and assemblies, Pictures, Images and Videos, along with some standard shapes and smart holes.

47. Click the **Insert Tab**

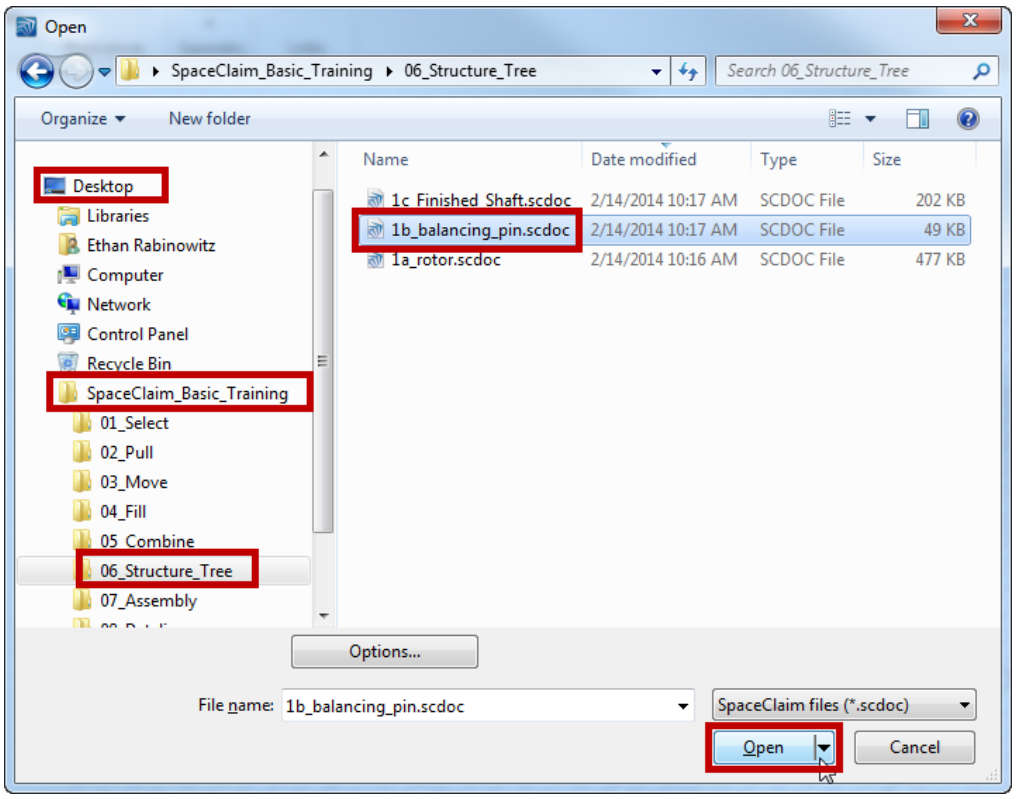


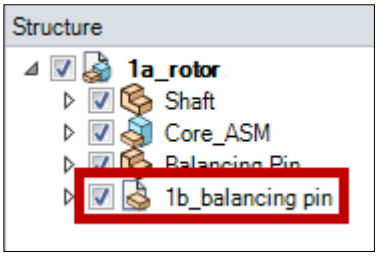
The Part, Catalog Parts and Illustrations Groups, highlighted above, all insert in an external file.

48. Click the **File button** in the Insert Tab's **Part Group** (first box above)

49. The window that opens should default to the folder the assembly is saved in, if not navigate to the 7Structure tree folder you opened the assembly from (likely on the desktop).

50. Select the **1b_balancing_pin.scdoc** and click **open**.





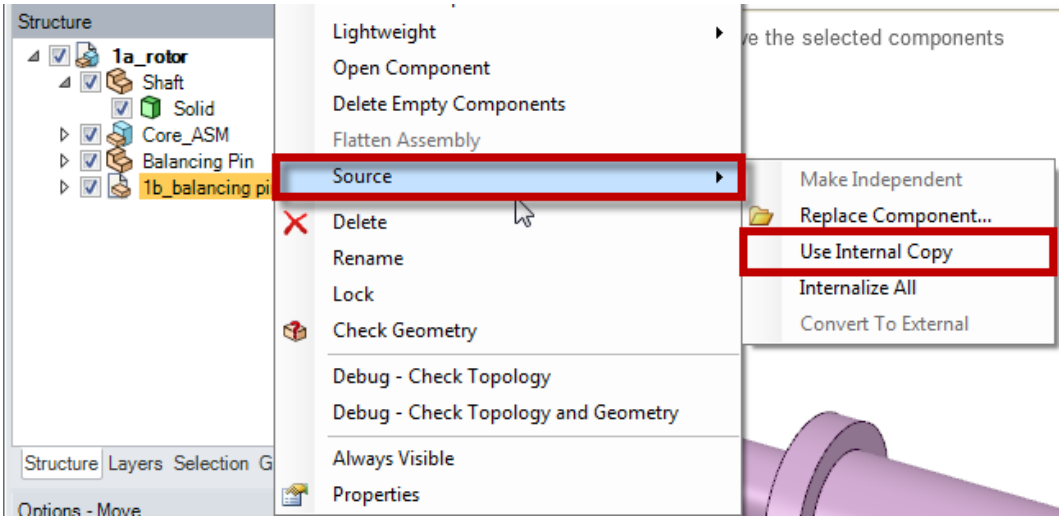
NOTICE The Inserted part, is now an external component in the tree

If you opened the file 1b_balacning_pin.scdoc, and edited it, it would change the 1b_balancing_pin in the 1a_rotor assembly also.

IMPORTANT: If you want to share the entire assembly with someone, you have to send both 1a_rotor.scdoc AND 1b_balancing_pin.scdoc

Use File\Share\As File to include all the references.

51. **Right click** on the 1b_balacning_pin in the tree and hover over the source submenu, and select **Use Internal Copy**



NOTICE: the icon for 1b_balancing pin has changed from External to Internal.

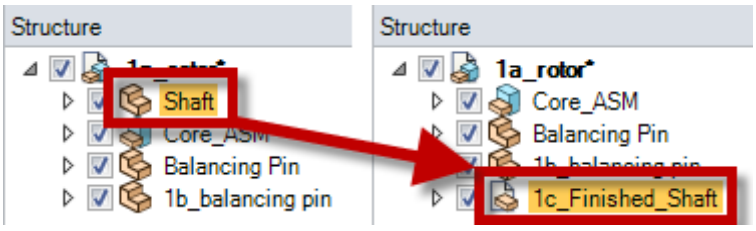
If you want to share the entire assembly with someone NOW, all you have to send is 1a_rotor.scdoc

If you opened the file 1b_balacning_pin.scdoc, and edited it, it would NOT change the 1b_balancing_pin in the 1a_rotor assembly.

52. **Right click** on the 1b_balancing_pin, hover over source and notice you now have the option to **Convert** it back **to External**. This is available for any Internal Component or Subassembly.

53. **Right click** on the Shaft in the tree, hover over **Source** and select **Replace Component**

    Select 1c_Finished_Shaft.scdoc



NOTICE: The replaced component in the tree is now external.
The replaced component is located by putting its own origin at the center of the assembly origin.

Instances

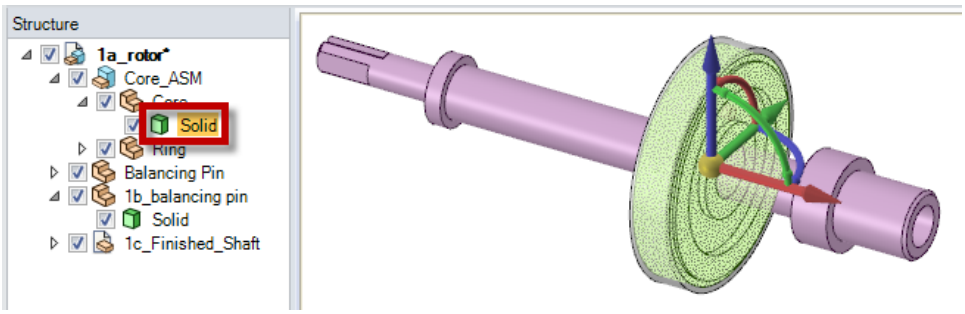
Having Instances in your design means you have more than 1 of the exact same component or subassembly. There are a few key things to remember about Instances

- Editing the geometry of any instance will update all other instances.
- Moving a solid in an instance will move all instances
- Moving a Component will only move that instance,
- Creating or deleting inside 1 instance will also occur in all other instances
- Existing instances can be made independent
- Changing independent components into instances is not possible
- Instances of 1 component or subassembly can either be all Internal or all External.

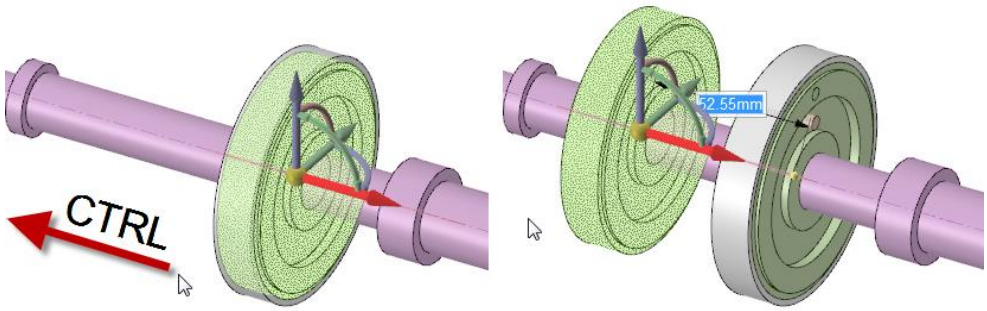
Let's explore these different characteristics interactively. Let's take a look at how to copy solids and components with the move tool

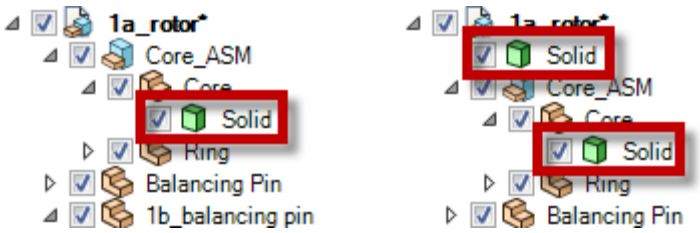
Copying a Solid

54. Turn on the **Move tool**
55. **Expand the Core_ASM** subassembly and then **Core** component
56. **Select the solid**



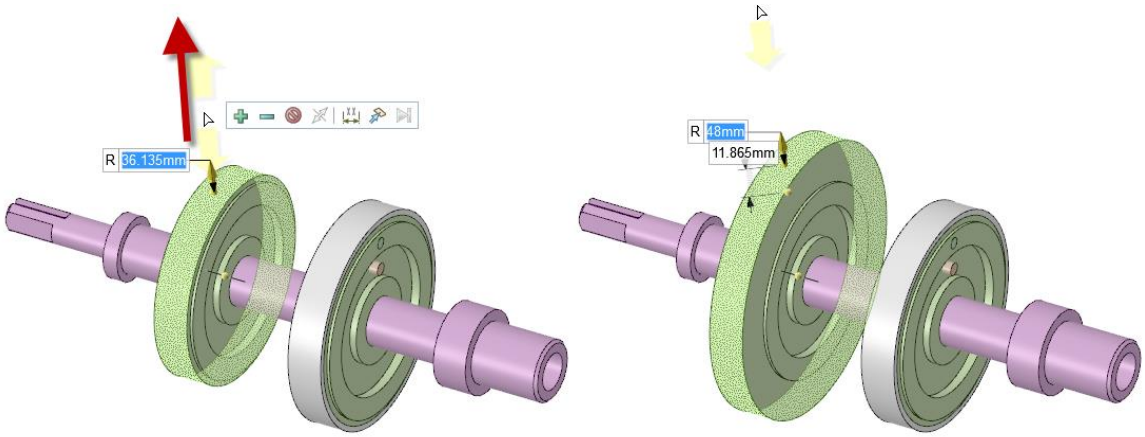
57. Click the **Red Straight Move Handle**
58. Hold **CTRL and drag** to the left, to make a **copy of the solid**





NOTICE: the new solid in the structure tree

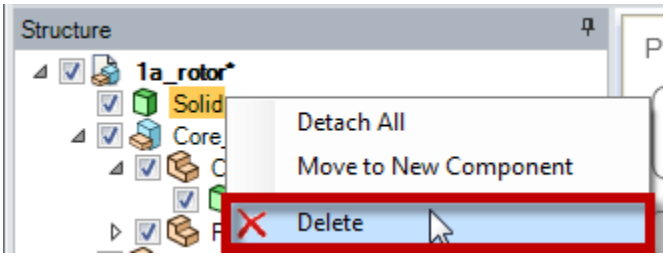
59. Turn on the **Pull tool**, select the **face** below and **pull** it bigger



Copying a Solid tips

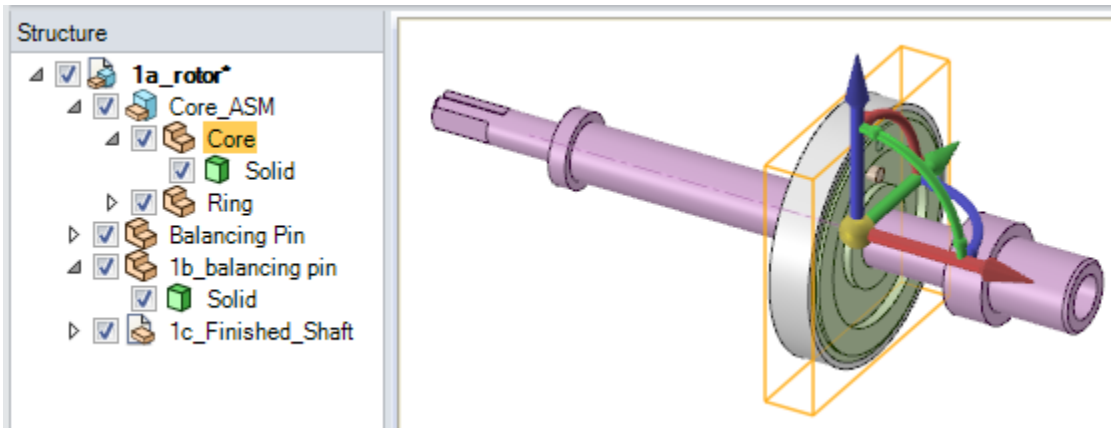
a. Editing either of the green solids will NOT change the other green solid.
b. When a solid is copied an instance is NOT created.
c. Copies of solids are Independent.

60. **Delete the new solid** at the top of the tree (right click on it, delete)

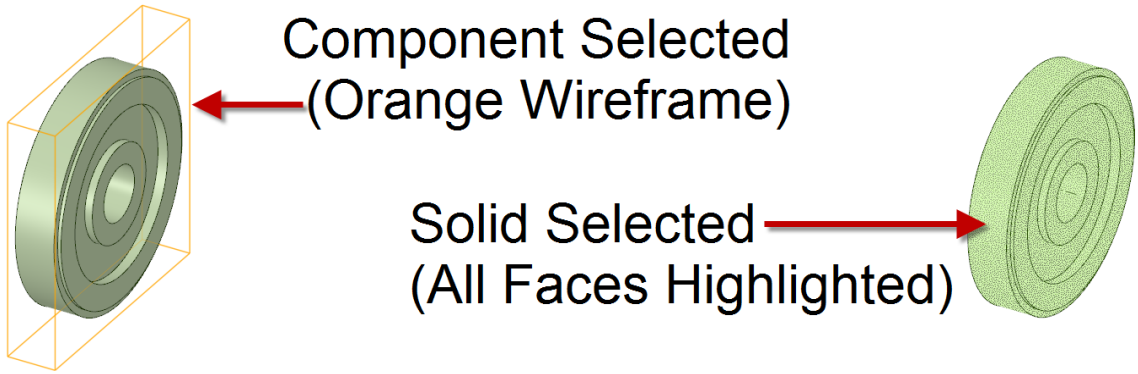


Copying a Component

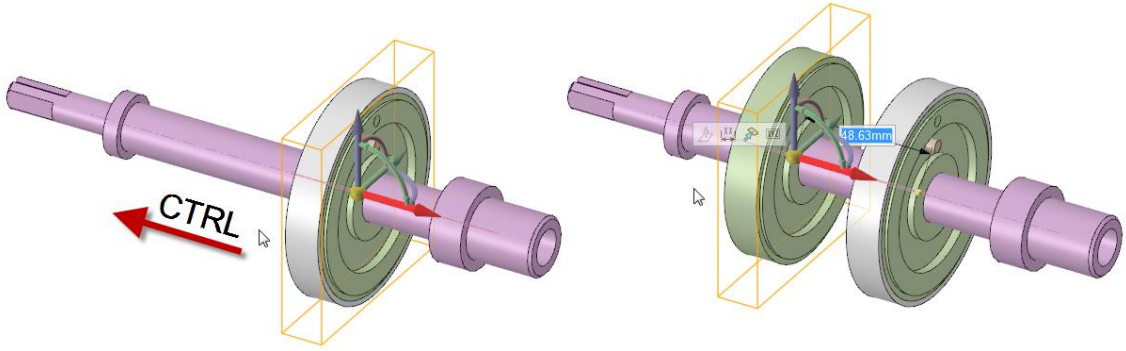
61. Turn on **Move** and **Click Core** in the tree



NOTICE: the difference between a selected Component and a Selected Solid

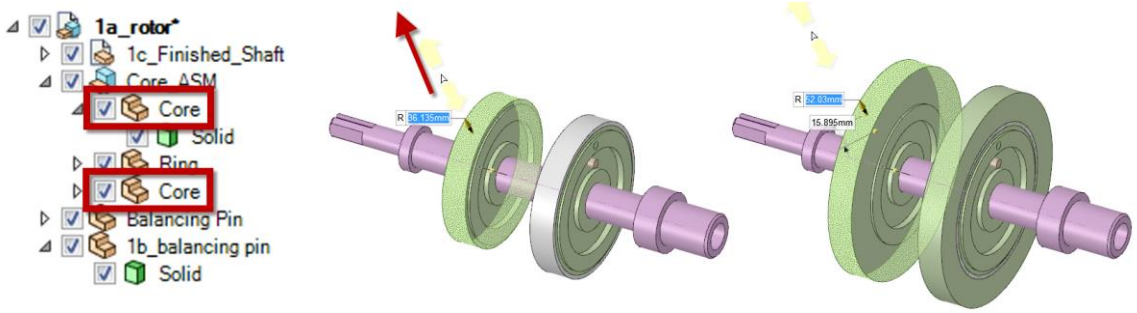


62. Click the **red move arrow**, hold **CTRL** and **drag to the left**, to copy the Component.



NOTICE there are now 2 Core components in the structure tree

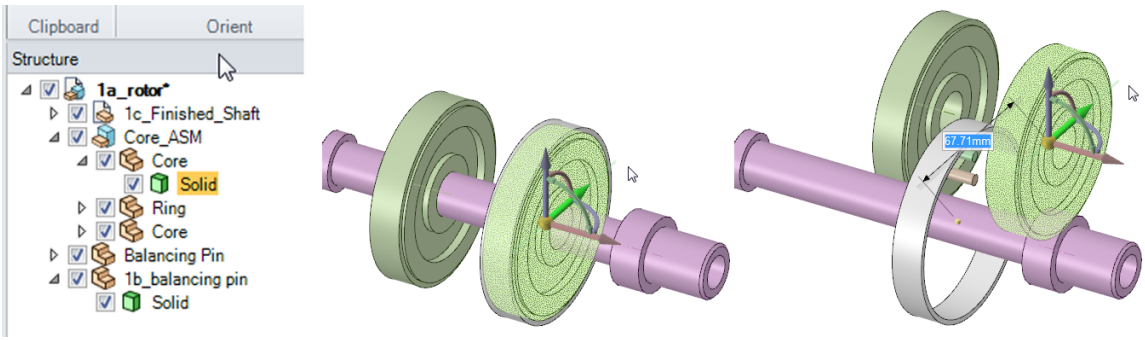
63. Turn on **Pull**, Select a face on either green solid and drag to pull the face



NOTICE: Whichever face you pull on the green solid, also gets pulled on the other green solid.

64. **UNDO**

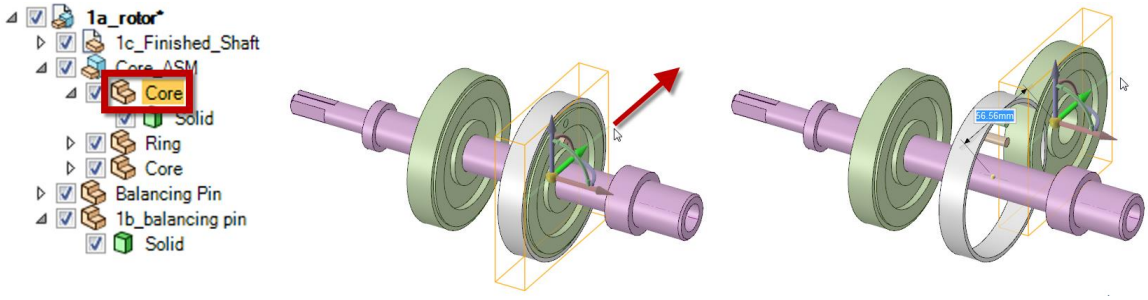
65. Turn on **Move**, select the **solid** under either Core Component, click the green straight move handle arrow and drag in that direction



NOTICE: If you move a solid of an instance, that solid will move in ALL of those instances.

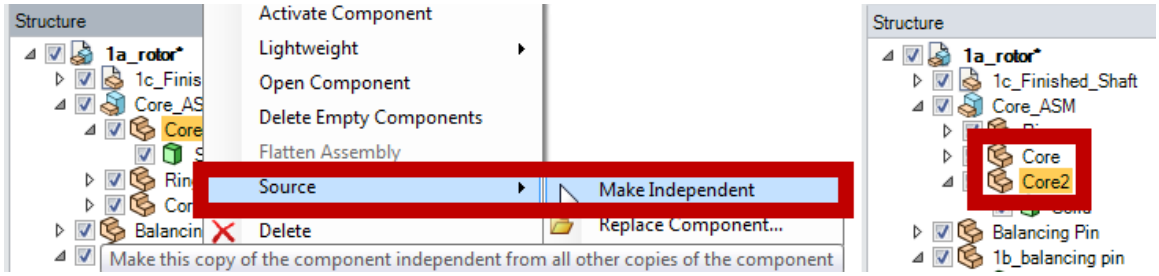
66. **Undo**

67. Select either **Core component** and **move** it in the same direction



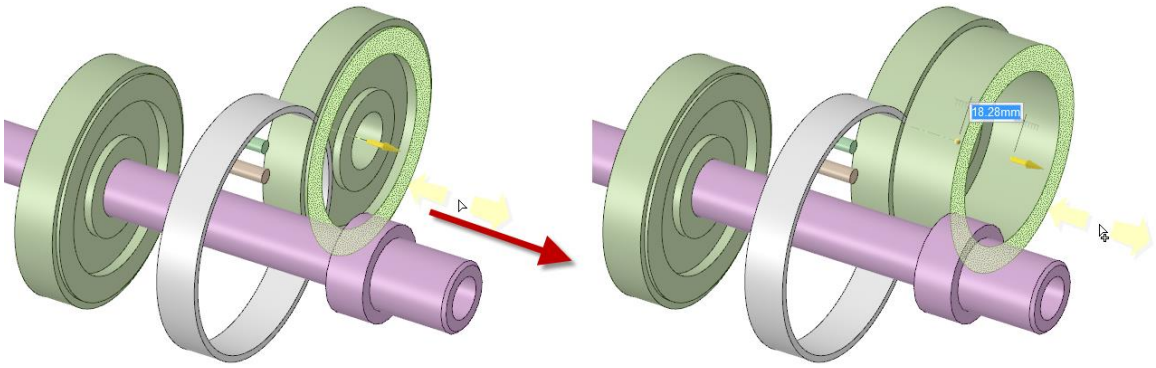
NOTICE: when you move the component of an instance, ONLY that instance moves.

68. **Right click** on either **Core** component, go to the **source** menu and select **Make Independent**.

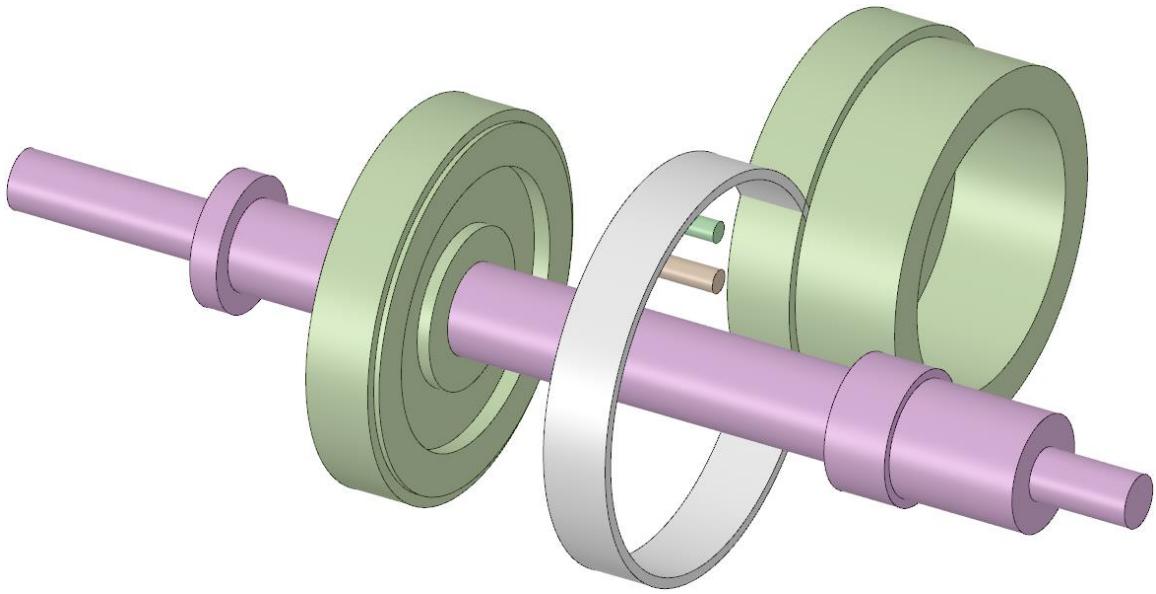


NOTICE: that the name of the component has changed from Core to Core2. It is no longer an instance of Core. Editing Core or Core2 in any way will not change the other.

69. Turn on pull, select any face on either green solid and pull.



NOTICE: the other green solids does not change



Assemblies

Arranging parts in space and in relation to other parts is what SpaceClaim's Assembly tools do. We will explore tools like align, tangent and orient, along with local and world origins. SpaceClaim is designed to work seamlessly between individual parts, entire assemblies and documentational drawings. The reason for this is SpaceClaim's 1 file format, SCDOC, which can be a part or assembly. An entire assembly can be contained in a single SCDOC file, or it could require multiple SCDOC files.

1. **File\Open** Desktop\SpaceClaim_Basic_Training**07_Basic_Assembly_2014.0** and open **Basic_Housing_2014.0.scdoc**

The **Assembly Tools** can be found in the **Design Tab**, in the **Assembly Group**, on the right side of the Ribbon bar

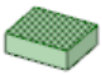
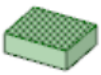
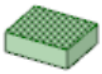
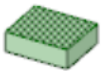
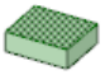
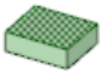
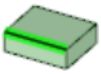
The 3 main Assembly Tools are **Tangent, Align and Orient**. We will also look at **Rigid** and **Anchor** in this section.

Position a Solid to the World Origin

2. Go to the **Display Tab**, on the right click **Show**, and check **World Origin**.

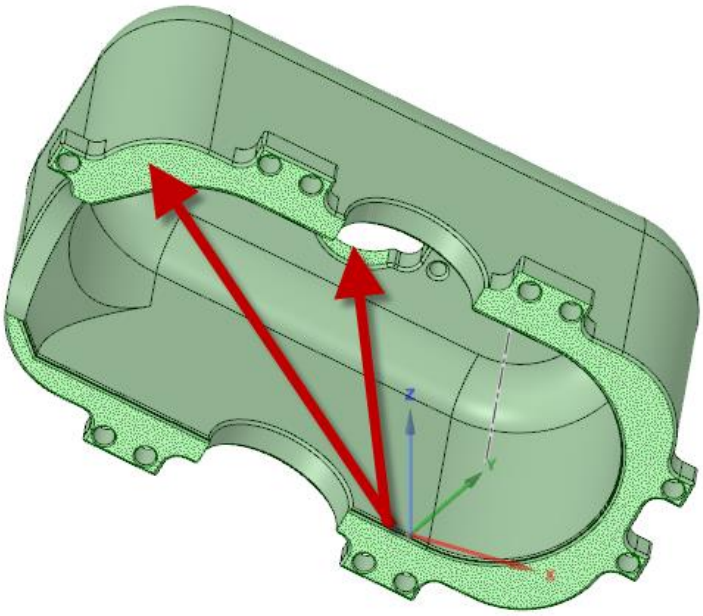
3. Go back to the **design Tab**, **Hover** over **Align** and press **F1**
4. In the help file, click the link for **Assembly Constraints Reference Chart.**

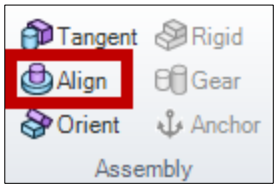
Assembly constraints reference chart

Geometry combination		Tangent	Align	Orient
Planar face  +	Planar face 	Planar faces are aligned in the same plane.	Planar faces are aligned in the same plane.	Planar faces are oriented so they are parallel.
Planar face  +	Cylinder face 	Planar face is aligned tangent to the cylinder.	Planar face and the axis of the cylinder are aligned.	Planar face and axis of cylinder are oriented in the same direction.
Planar face  +	Conical face 	n/a	n/a	Planar face and axis of conical face are oriented in the same direction.
Planar face  +	Spherical face 	Planar face is aligned to the center point of sphere.	Planar face and center of sphere are aligned in the same plane.	n/a
Planar face  +	Edge 	Planar face is aligned tangent to edge.	Planar face and edge are aligned in the same plane.	Planar face and edge are oriented so the edge is perpendicular to the face.

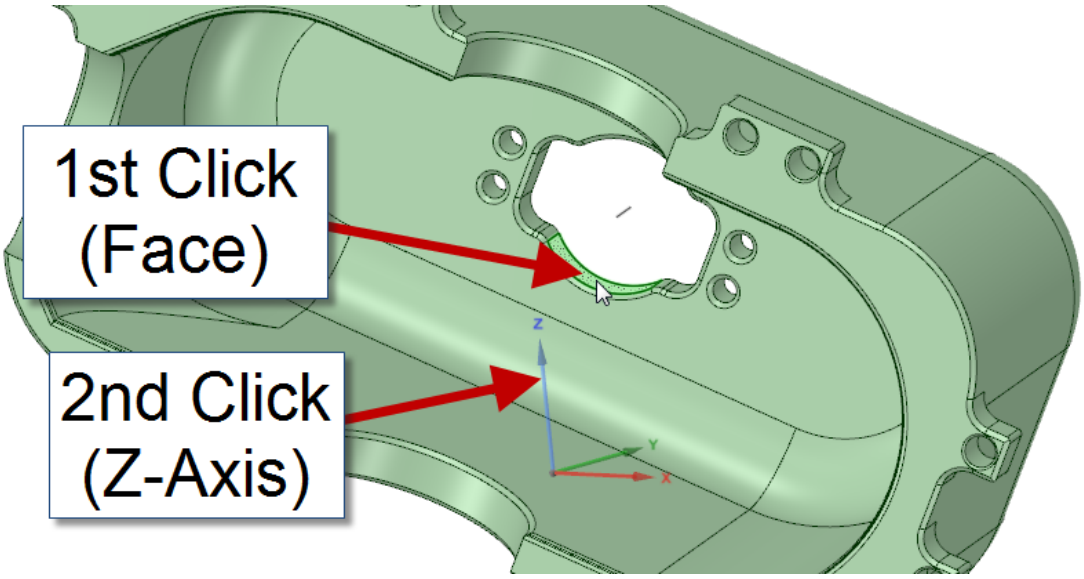
The Assembly Constraints Reference Chart is 5 pages longs, and explains how each of the 6 Assembly tools applies to different combination of Geometry.

The goal is to assemble the solid in the housing.scdoc file to the world origin. The hole in the center will be coaxial to the Z axis of the world origin. You will also assemble the face on the bottom to the X-Y plane of the World Origin



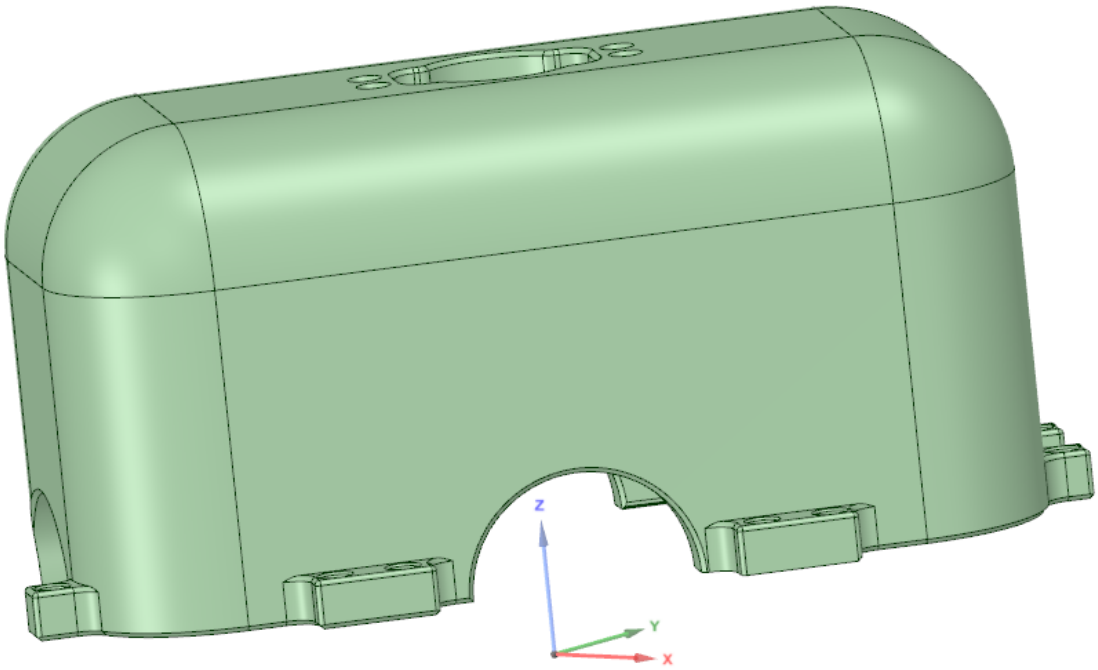


5. **Click in white space** to make sure nothing is selected
6. Turn on the **Align Tool** in the Assembly Group of the Design tab
7. Spin, Pan or Zoom to have a good view of the inside of the Housing
8. Click the **cylindrical face of the hole**.
9. Click the **Z-Axis**.

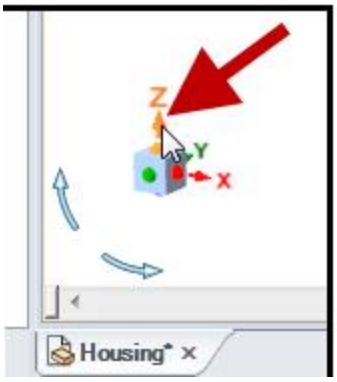


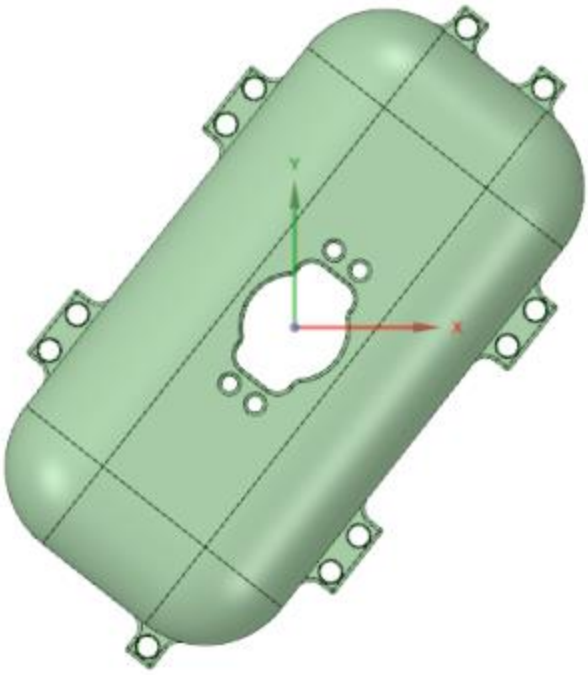
NOTICE the solid has translated and rotated to align the hole to the Z-Axis

Ignore the message “Some conditions were not created”



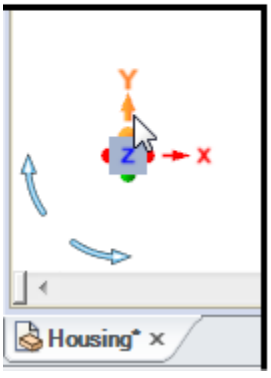
10. Click the **Z-Axis in the orientation widget** in the bottom left Corner

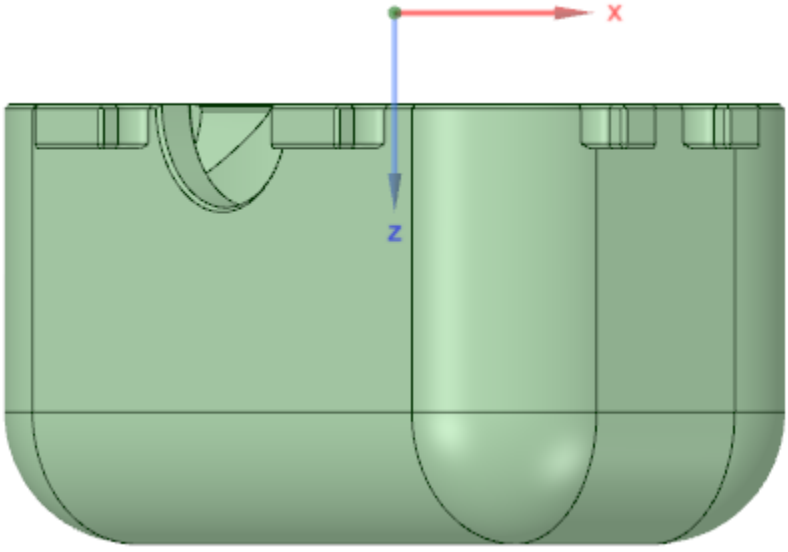




NOTICE how the World Origin in in the middle of the hole

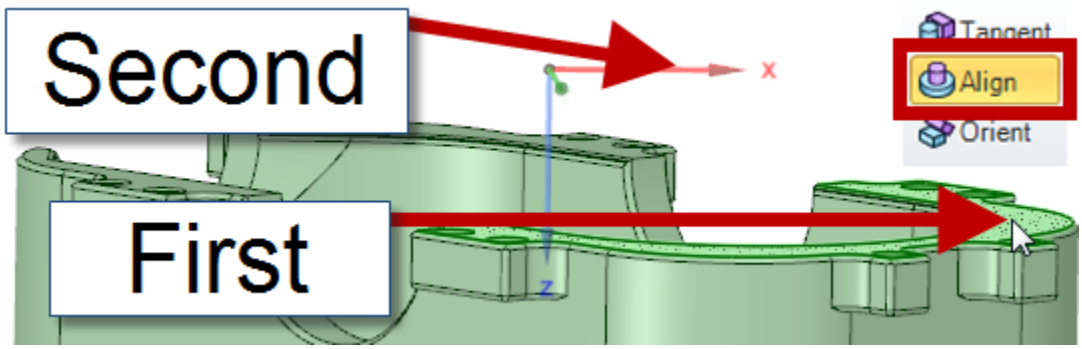
11. Click the **Y-Axis in the orientation widget** in the bottom left



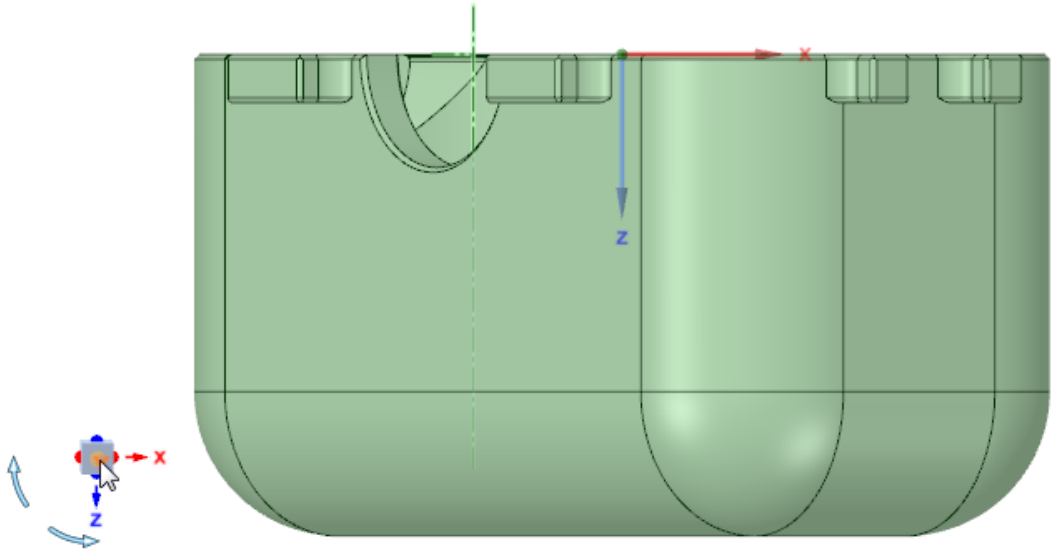


NOITCE how the World Origin is not on the bottom face of the part.

12. Spin the model like the view below.
13. With the **Align tool** still on, click the **face** below, then the red **X-Axis**.

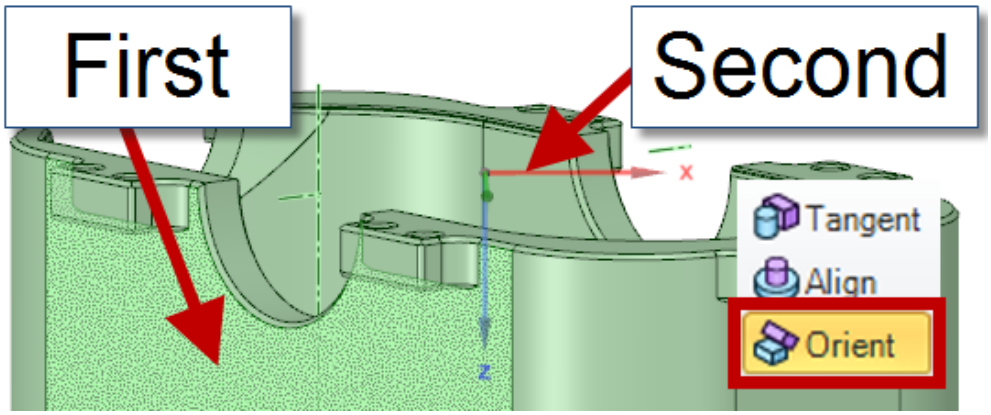


14. Click the **Y-axis of the orientation widget** in the bottom left again.

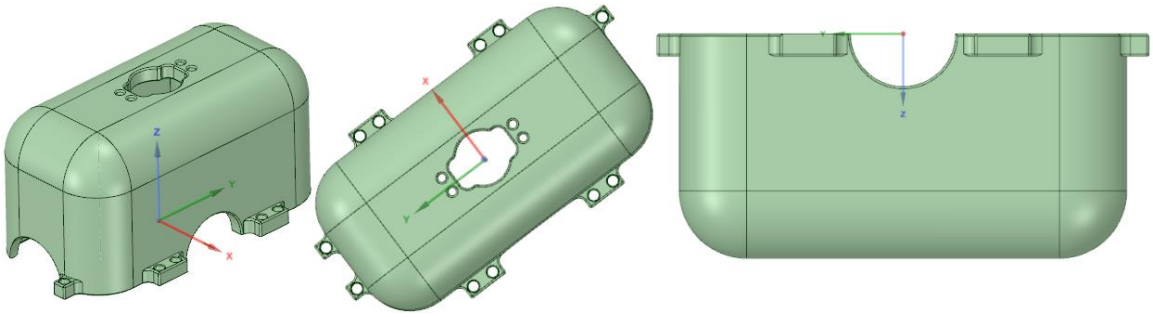


The Last step is to rotate the part so that one of the faces on the side is Oriented to the Y or X axis. Align would in Incorrect to use here, because Align puts the 2 selected object on the same plane.

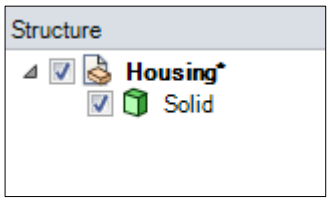
15. Orient does not make them coplanar. See Assembly Constraints Reference chart



16. Spin the Model to see how it is now Assembled to the World Origin



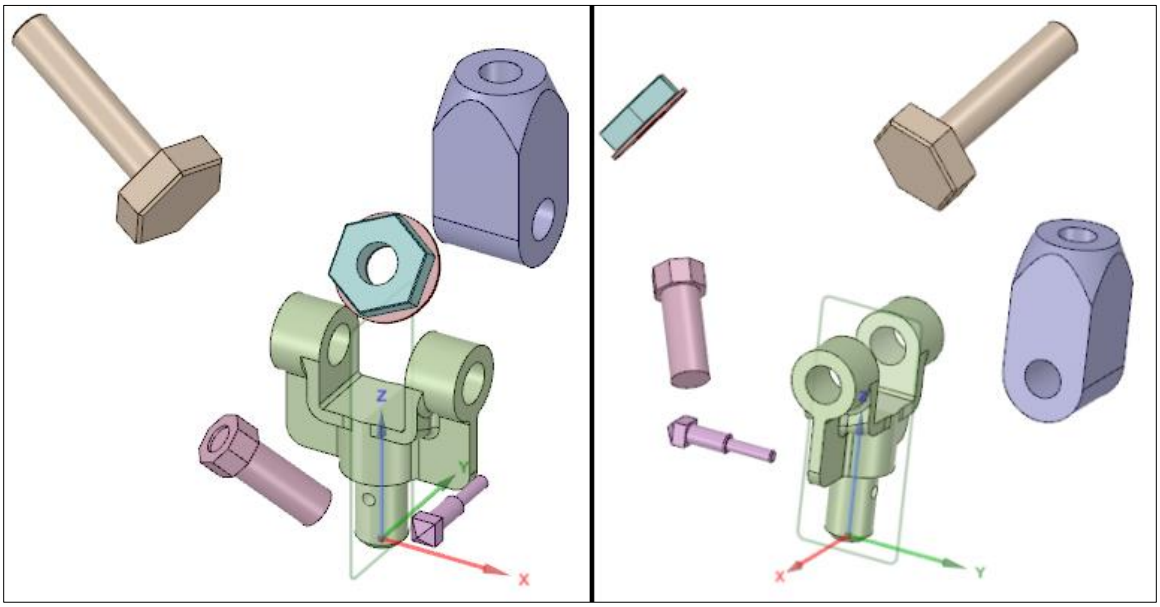
NOTICE that nothing has been added to the Structure Tree. This happens when geometry like a solid is not in a component. Next you will explore using the assembly Tools with multiple parts in an Assembly.



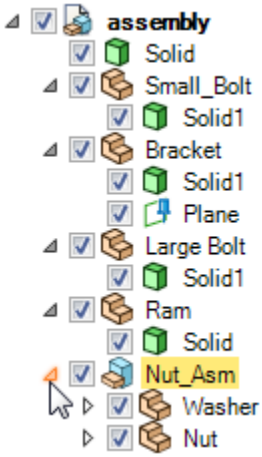
Assembling Multiple Parts Together

17. Open **Basic_Assembly_2014.0.scdoc**

18. Spin the model around to see how the components are scattered in space



19. Take a look at the Structure Tree, and **Click the Triangles** next to each Component and Subassembly to expand the Tree



NOTICE:

a. there is a solid at the top

b. 5 Components below the solid

c. 1 Subassembly with 2 Components at the bottom

20. Make sure the World Origin is still shown (Display Tab). The World Origin should be at the bottom center of the green Bracket. (see image below)

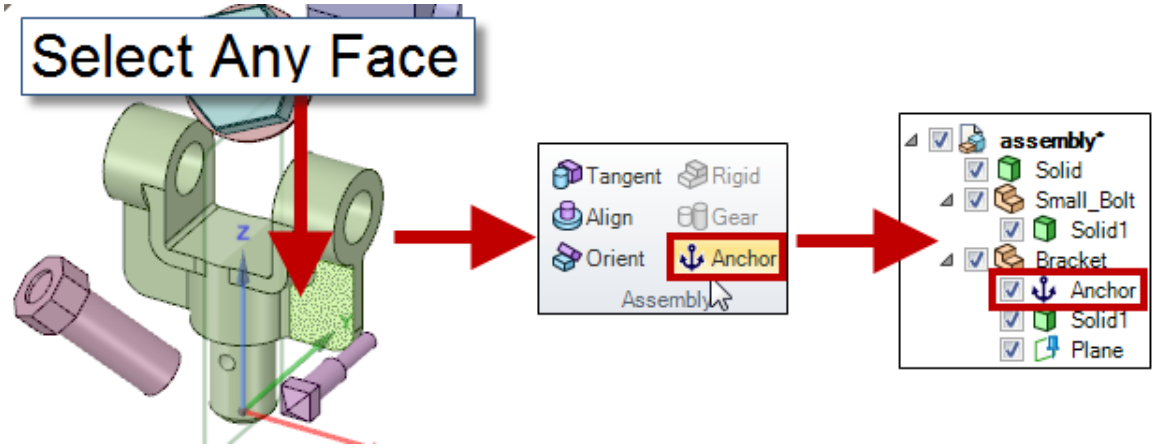
Very Often Assemblies import with some Components in the correct Position.

It is best practice to **Anchor** the one component in place that you will assemble the rest of the components to.

Imagine a Mr. Potato head toy: Anchor the feet on the table, then assemble the body to the feet, and all the parts to the body.

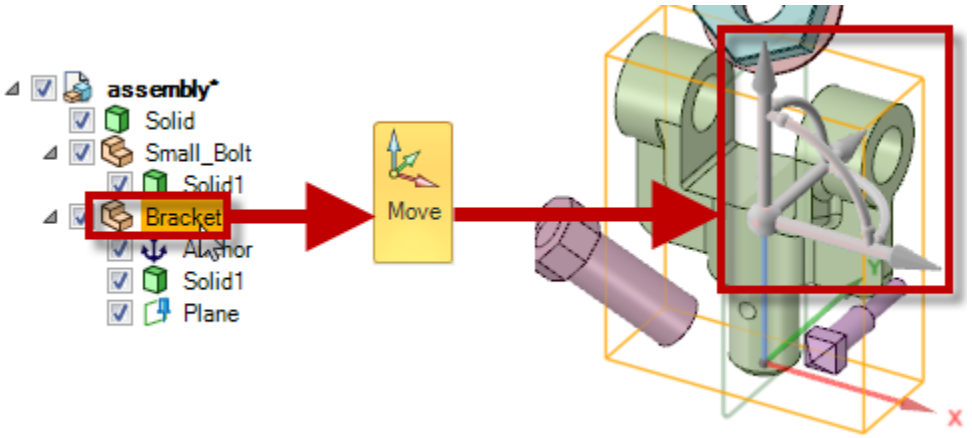
---

21. Select any **Face** on the Green solid (Bracket) and click the **Anchor Tool** in the Assembly Group



NOTICE in the structure tree, an Anchor condition has been added to the Bracket Component.

22. Select the **Bracket Component**, turn on **Move**, and notice the Move Handle has been greyed-out. You Cannot Move an Anchored Component.

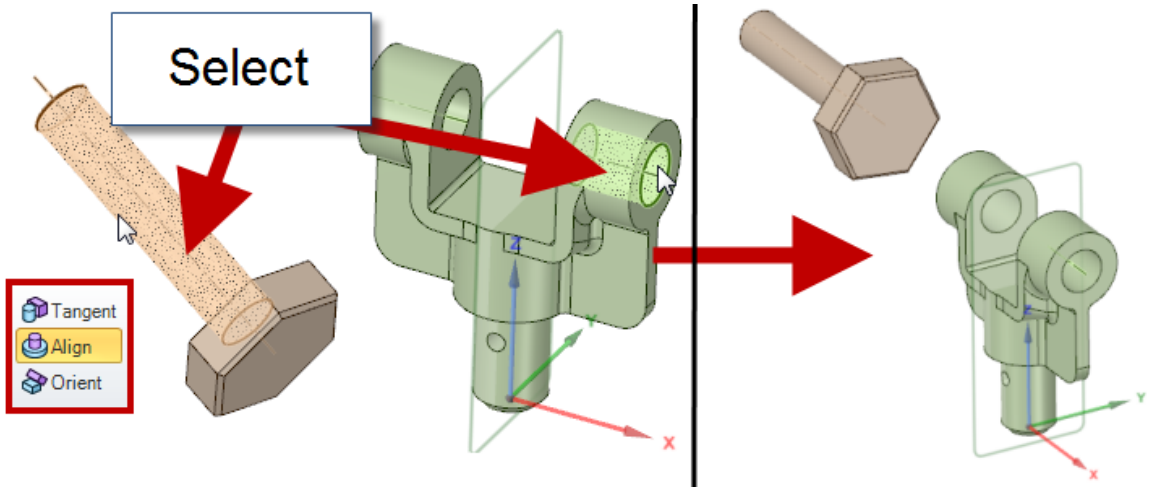


NOTICE: When the Move Handle is Greyed-out, you cannot move the component in the greyed directions.

23. **Click in white space** to clear your selection

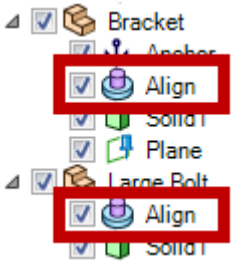
24. Turn on the **Align tool**

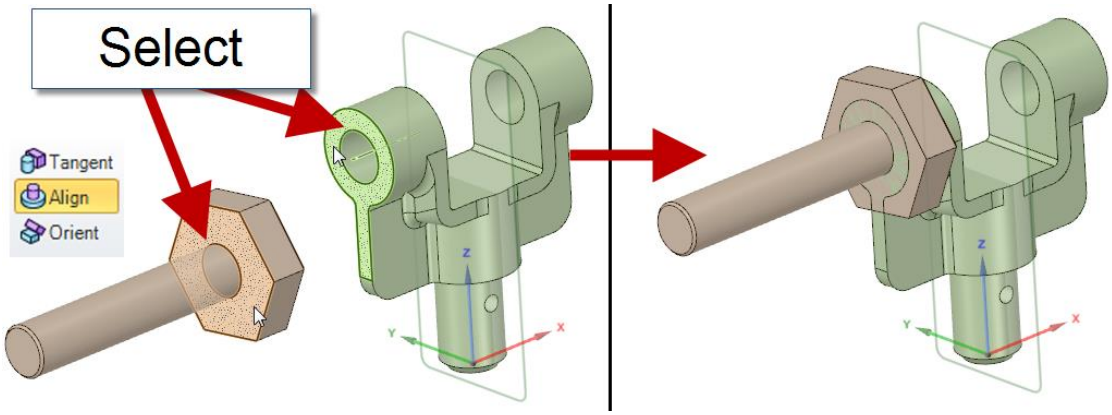
25. Click the cylindrical shaft of the large orange bolt, then the cylindrical hole of the Green Bracket



26. Spin the model around to notice the shaft of the Large Bolt is aligned to the hole in the Bracket

NOTICE an Align Condition has been added in the tree to BOTH the Bracket AND the Large Bolt, for the single alignment.

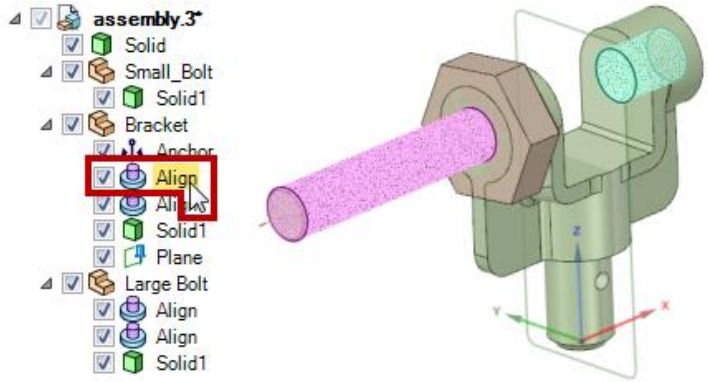




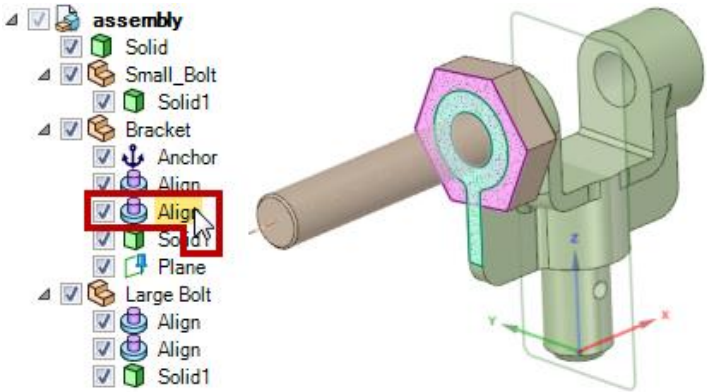
NOTICE that you could have aligned the planar face of the bolt to the right side of the bracket instead of the left, only because the bracket is symmetric.

Reverse or Flip an Assembly Condition.

27. Expand the Bracket and Large Bolt in the structure tree by clicking the triangle next to them.
28. Notice the Bracket has an Anchor and 2 Align conditions, and the Large_Bolt has 2 Align conditions

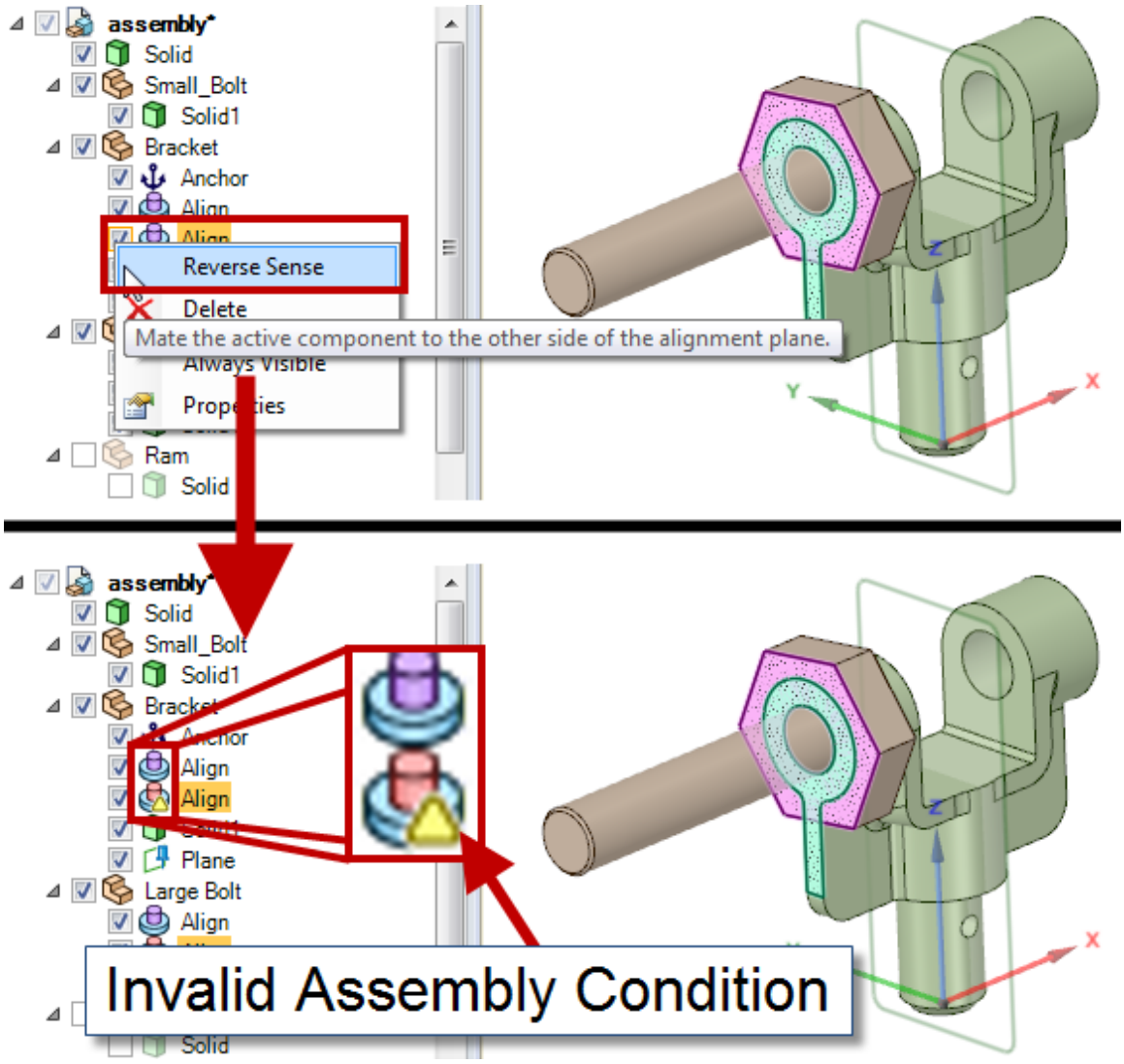


29. Hover the mouse cursor over the first Align for the Bracket. Notice the highlighted faces
30. Hover the mouse cursor over the second Align for the Bracket. Notice the highlights faces.



NOTICE when you hover over an assembly condition, it highlights the assembled faces on the model.

31. Right click on the second Align in the tree under the Bracket, and click **Reverse Sense**



NOTICE that nothing changed to the model.

NOTICE the icon for the Align condition now has a yellow triangular flag on it. This means there is an issue or conflict with the Align Condition.

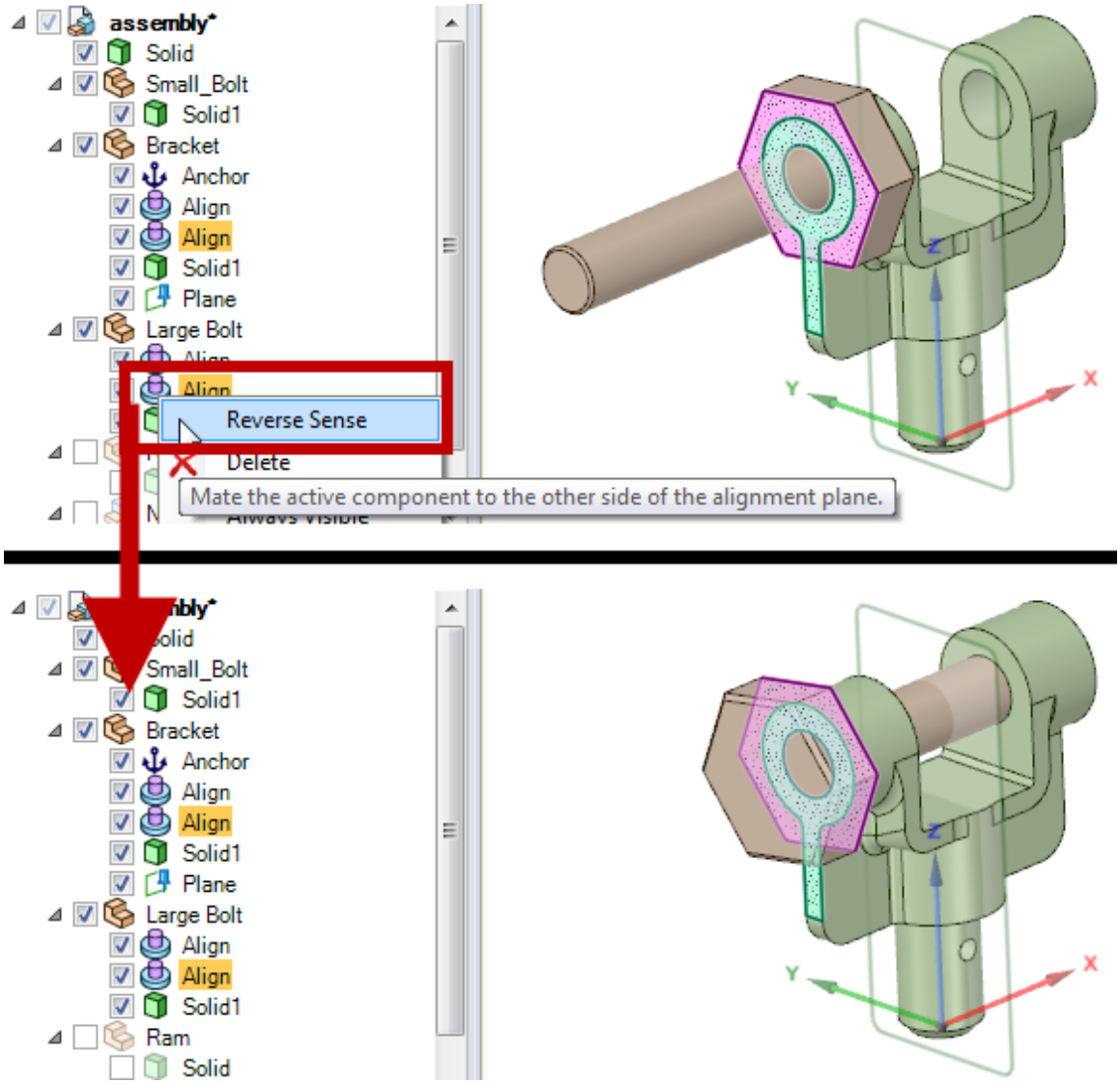
Recall that the Bracket was anchored in place. You cannot reverse the sense on a component that is anchored.

IMPORTANT: While creating or changing Assembly Conditions, keep an eye on the structure tree.

- Watch out for Invalid Assembly Condition with the Yellow Triangle.
- Hover over existing conditions to check what is being assembled.

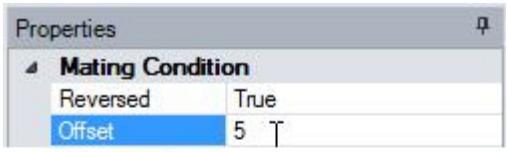
32. **Undo**.

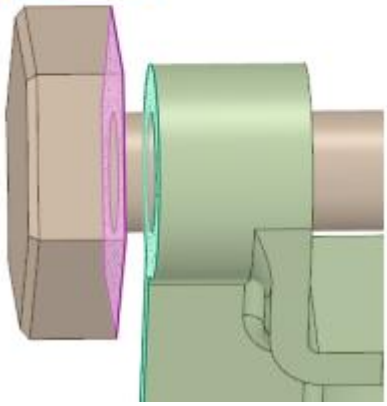
33. Right Click on the second Align Condition in the Large Bolt, and choose **Reverse Sense**



IMPORTANT: When you reverse the sense of an assembly Condition, you must right click on the Align condition under the component you want to flip.

34. Click the 2nd Align condition under either the Bracket or Large Bolt
35. Look to the properties panel in the lower left corner of the screen. Change the Offset value to 5mm

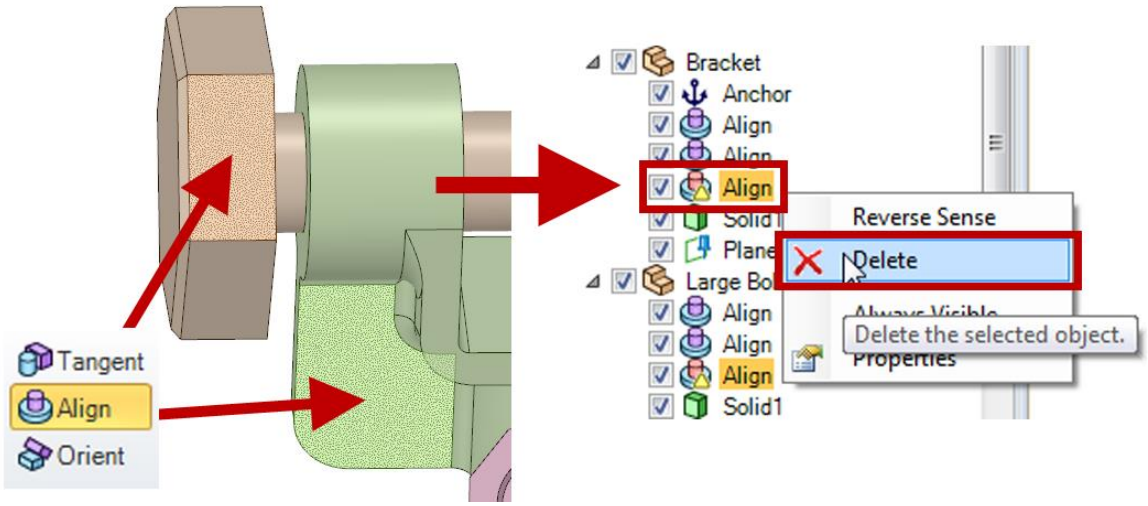




NOTICE that since the Bracket is anchored, the Large_Bolt will offset 5mm. To Offset the Large Bolt in the opposite direction, enter a negative value

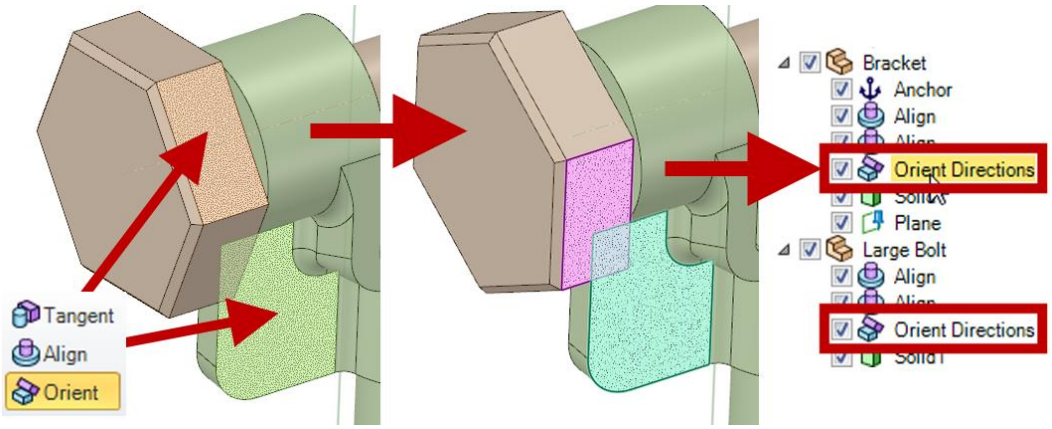
The Large Bolt is almost fully assembled. The rotation of the head is the only thing not defined.

36. With the Align tool still on, click the 2 faces indicated below. The model will not change, but you will have an invalid Align Condition under Bracket and Large Bolt
37. Instead of Undo, Right click on either invalid Align and select **Delete**.



NOTICE: Align did not work because Align tries to put 2 planar faces on the same plane. Putting these 2 faces on the same plane would violate the first Align condition of the Pin and the Hole.

38. Turn on the Orient tool, and click the same 2 faces

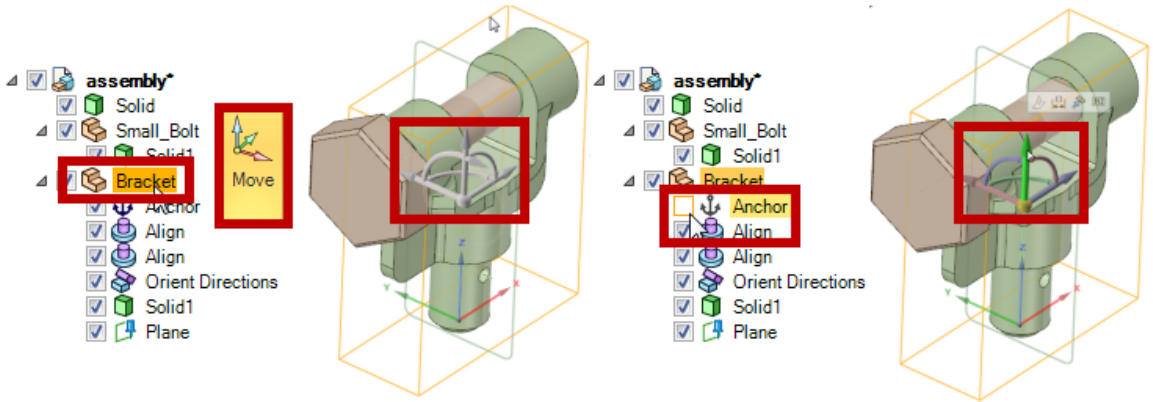


The Bolt is now fully assembled to the Bracket

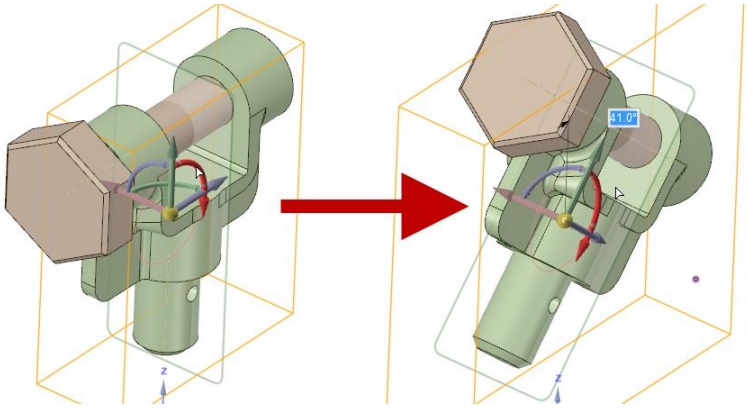
39. Click Bracket In the Structure Tree

40. **Turn on Move**. Like before, the move handle will be greyed out

41. **Uncheck the Anchor** condition in the Tree to turn it off. The move handle will un-grey.

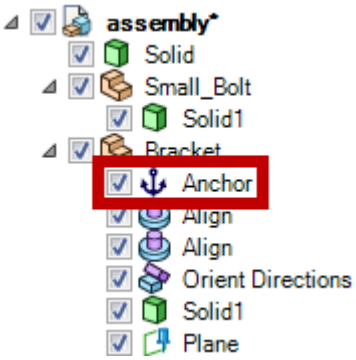


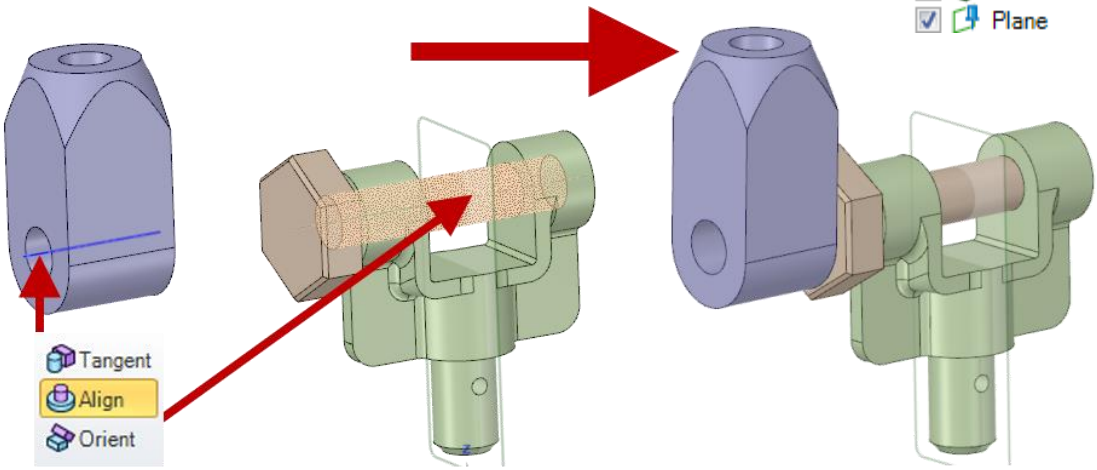
42. Click any of the move handles, then drag in that direction



NOTICE: When moving the Bracket Component, the Assembled Bolt moves too.

43. **Undo** to before moving the Bracket. That was just to show that the Assembly conditions are held during move and other operations.
44. **Recheck the Anchor** Condition in the Tree
45. Turn on **Align**
46. Select the **Axis through the hole** of the purple Ram
47. Select the **cylindrical shaft** of the Bolt



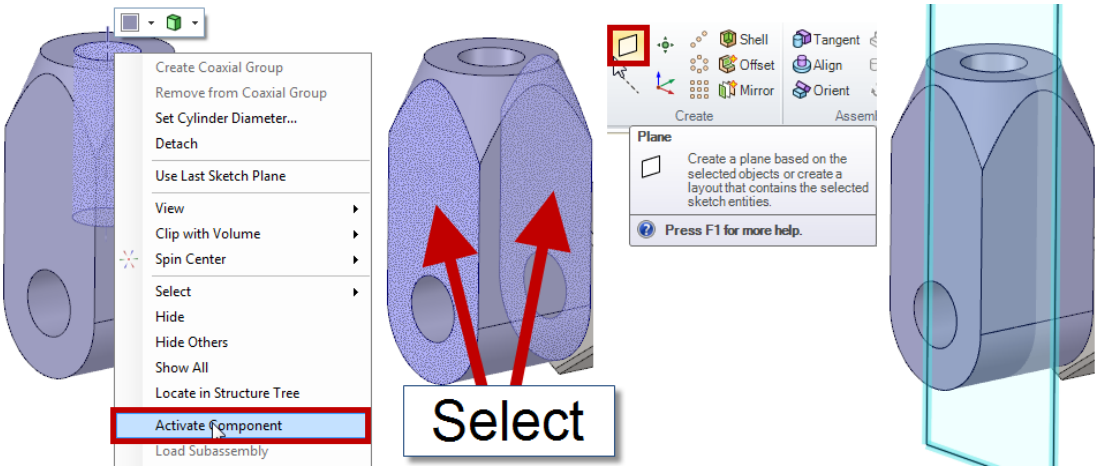


NOTICE when aligning things like holes and pins, you can use any combination of Axes and Cylindrical faces. You do not have to use 2 axis, or 2 faces

To center the Ram to the Bracket, you could align the vertical hole and vertical pin. If there was no hole and pin, there would be no geometry to align the centers.

Assembling Centerlines Using Planes.

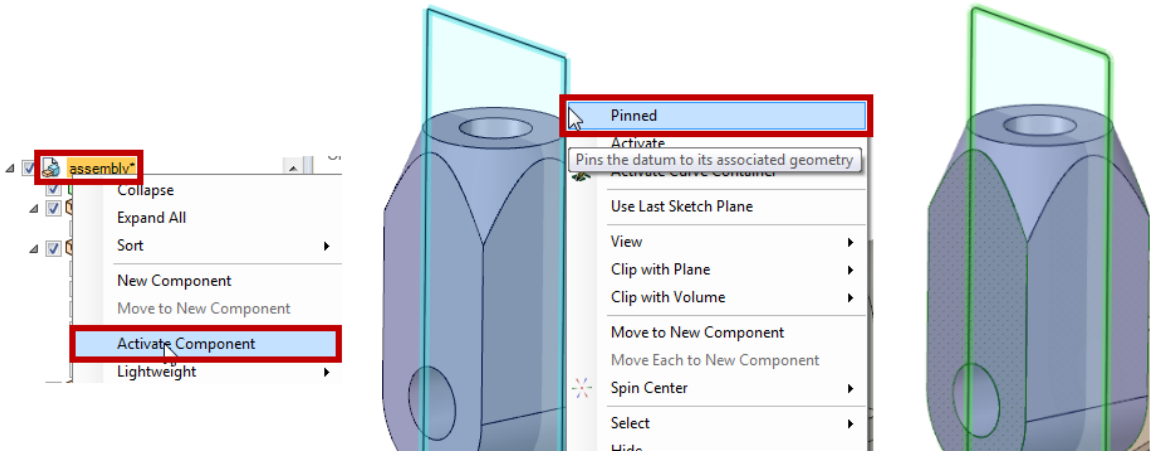
48. Right Click on the Purple Ram in the Tree or on the Solid and click **Activate Component**.
49. Select the 2 faces on the right and left side of the Ram
50. Click the **Plane Tool** in the **Design tab**, under the **Create group**.



NOTICE with the 2 faces selected, the Plane tool inserts a Plane between the 2 faces. Because you activated the Ram, the Plane is created in the Ram Component

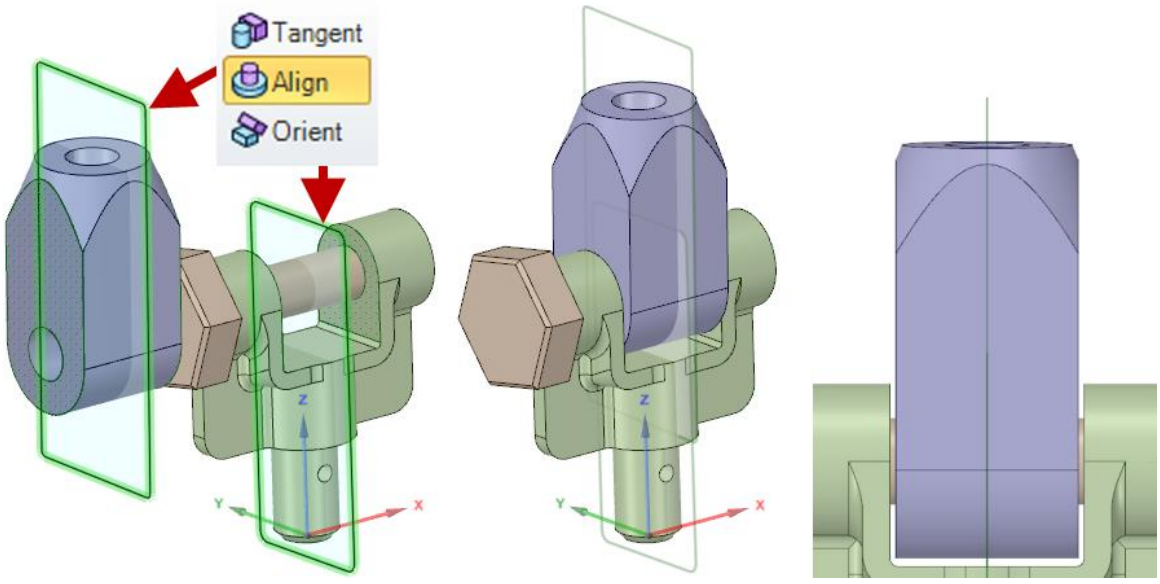
51. Right Click on the **Top level assembly** and Choose **Activate**

52. Right click on the border of the new Plane and select **Pinned**



NOTICE the original unpinned Plane is blue with sharp corners. The Pinned plane is green, with rounded corners. When the Pinned plane is selected, the faces it is pinned to will be highlighted with a speckled pattern.

53. Turn on Align, select the 2 planes



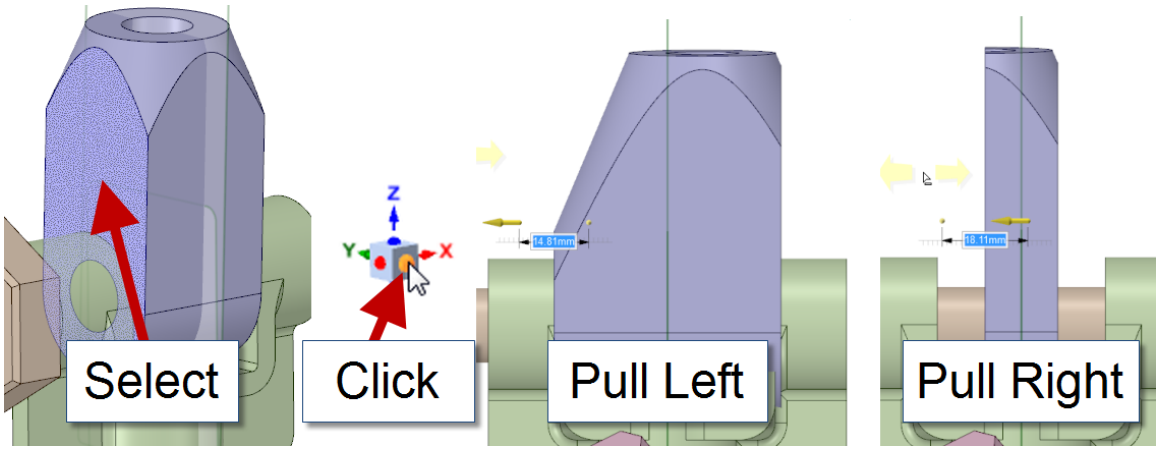
NOTICE the Planes are aligned, centering the Ram to the Bracket

Changing Something that's Pinned:

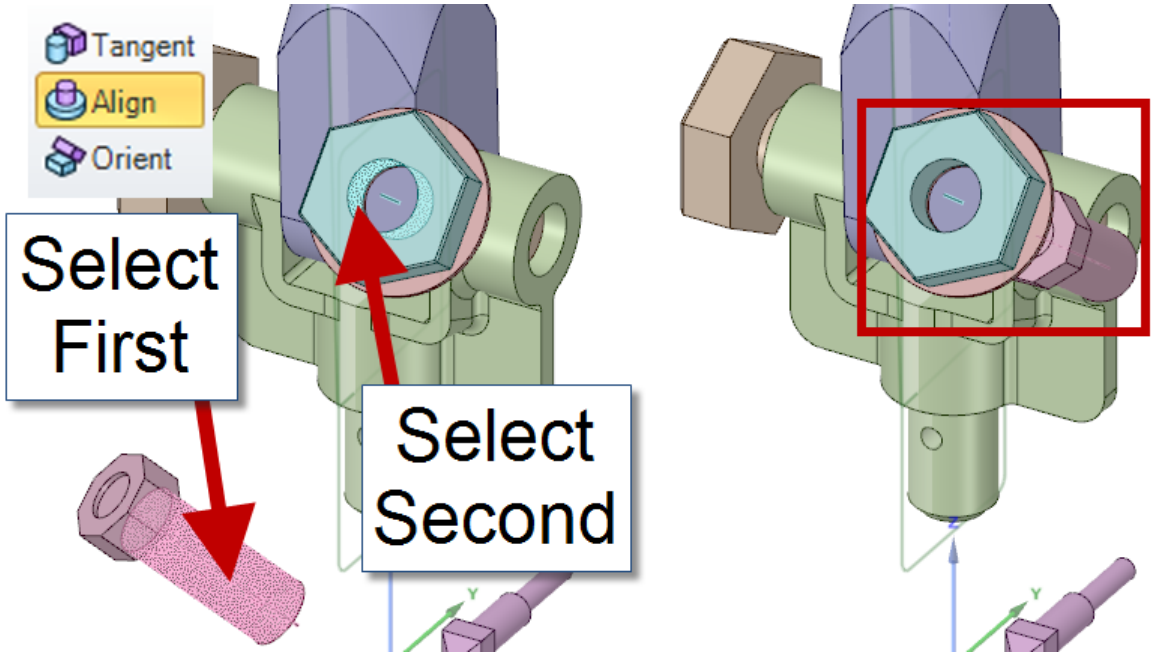
54. Select the face below
55. Click the green Y-Axis dot in the widget in the bottom left corner.
56. Turn on Pull, drag left and drag right. DO NOT LET GO OF THE MOUSE BEFORE NEXT STEP

NOTICE: how even though the Ram is no longer symmetric, it stays centered to the Bracket

57. Click **ESC** while pulling to cancel the Pull action

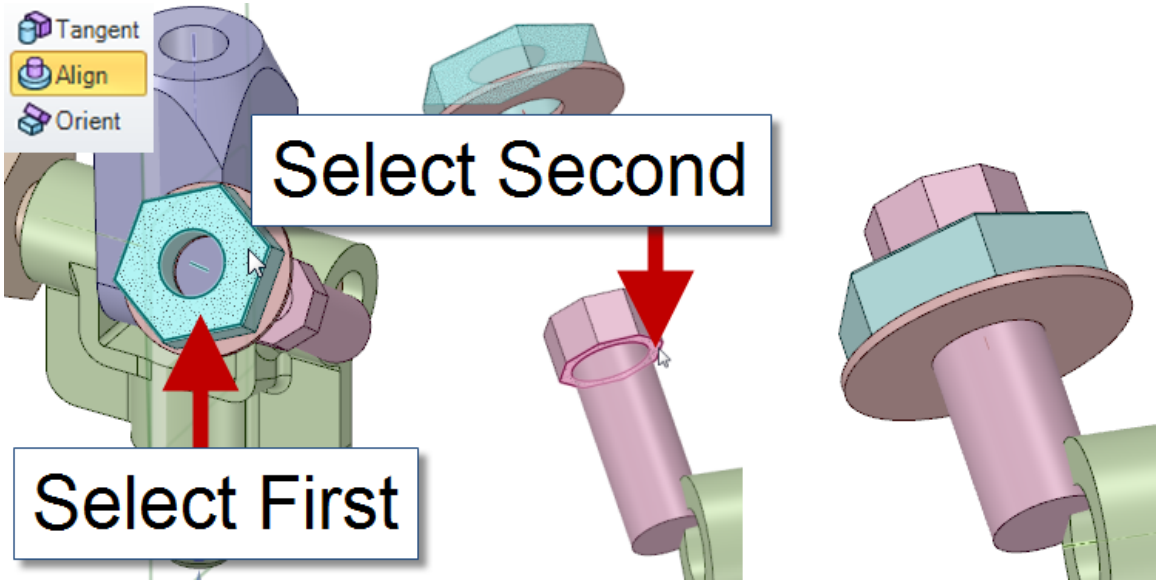


58. If you let go of the mouse while pulling left and right, Undo to before pulling the face of the Ram.
59. Return to a **Home View**
60. Turn on the Align Tool, Select the Shaft of the Small Bolt, and Select the hole of the Nut.

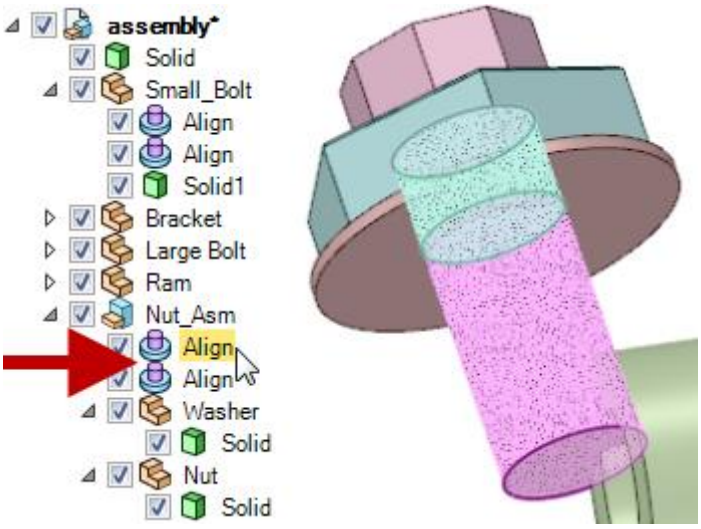


NOTICE that the Small Bolt Moved to the nut

61. Click the top face of the Nut, Click the face of the Small Bolt shown.



NOTICE The nut moves to the small bolt. If neither components are anchored, **whichever you click 1st, moves to whichever you click 2nd**.



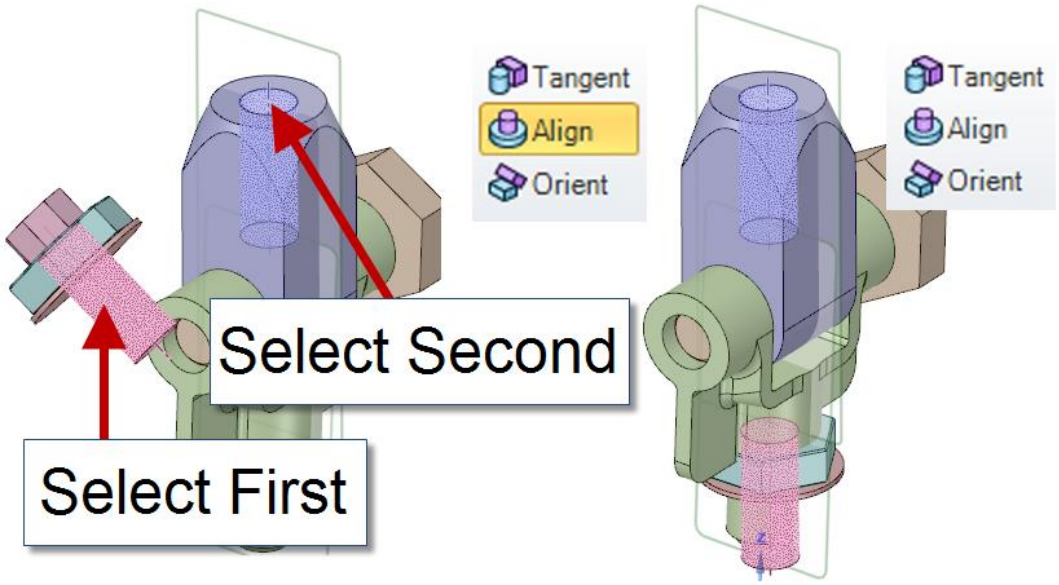
NOTICE The Nut and Washer moved together. This is because they are in the same Subassembly.

NOTICE that even though you clicked the face of the Nut, there are no Assembly Conditions in the Nut component. The Align Condition is in the Subassembly.

Assembly Conditions are added to the highest Subassembly

To add assembly conditions between the nut and washer, you would right click on the Nut_asm subassembly, click Open Component, and apply the assembly conditions there.

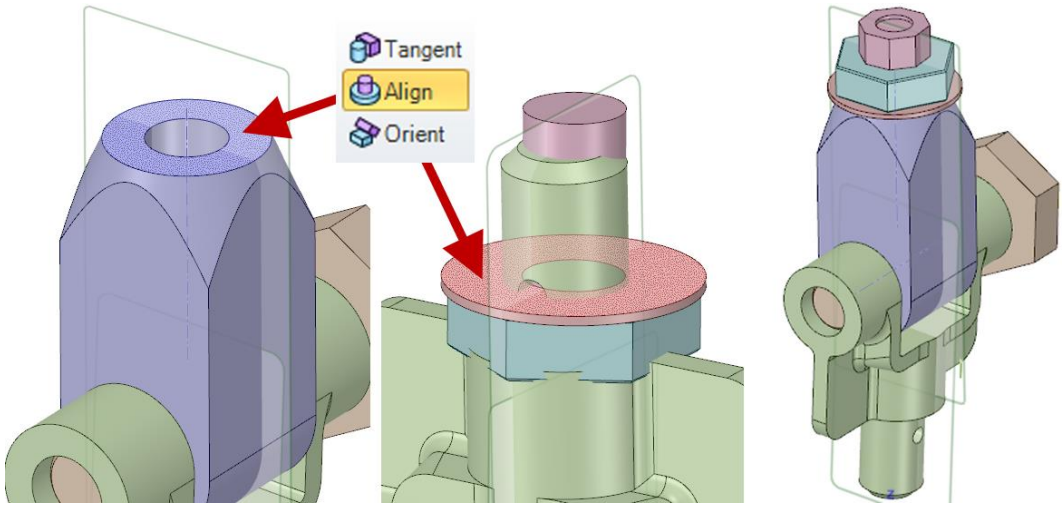
62. Click the Select tool, or press ESC to exit out of Align.
63. Select the faces in order below
64. With the faces selected, click the Align Tool



NOTICE

a. You can either turn on the tool and select the objects to align.
b. You can select the objects, and then turn on the tool to apply the assembly condition\
c. Whether you turn on the tool and select, or preselect and turn on the tool, the object selected first gets moved to the object selected second, assuming neither is anchored.
d. When you turn on the tool then select, the tool stays on. If you select then turn on the tool, the tool turns off

65. Finish assembling the Nut_Asm by using the align tool along with the 2 faces shown below

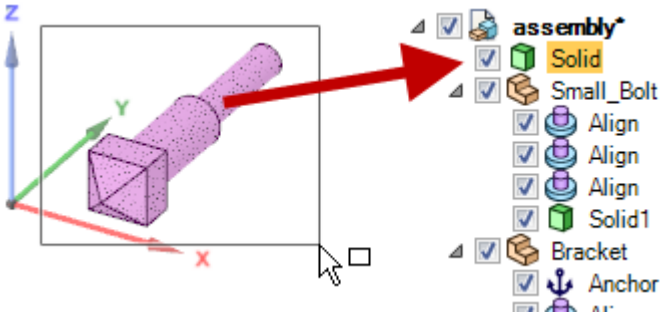


66. Return to a Home view (H)

The last part to assemble is the Square screw below the bracket. Before using the assembly tools, let's find out more about this part.

67. Escape out of the Align tool if it's on

68. Box Select the screw and look to the top of the structure tree

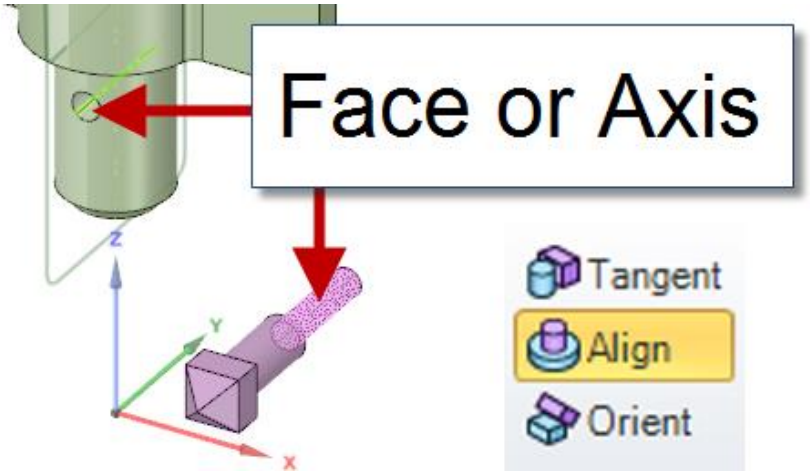


NOTICE that this screw is not in a component.

RECALL that if a solid is not in a component, assembly conditions are not saved.

69. Turn on Align

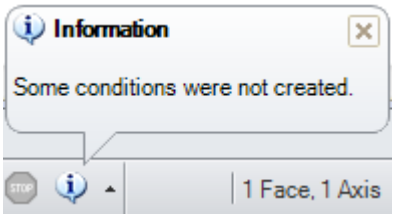
70. Select the geometry indicated



NOTICE a message at the bottom

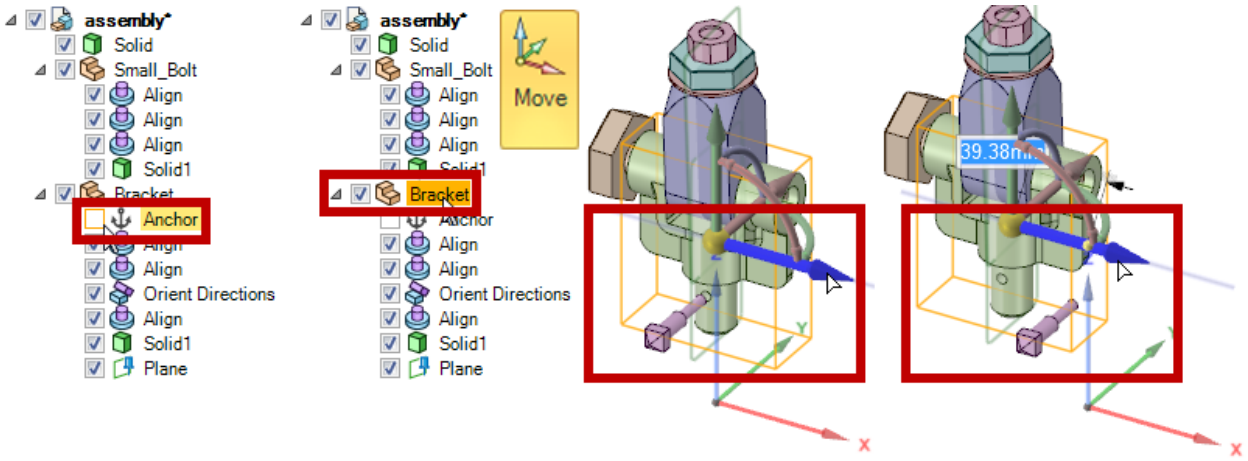
"Some conditions were not created"

Since 1 of the 2 solids was not in a component, the assembly condition could not be saved.



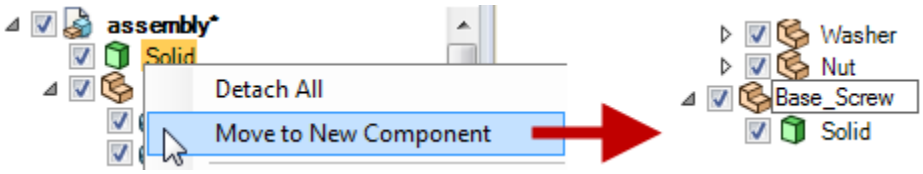
What does it mean if an assembly condition is not saved?

71. **Turn off the Anchor** in Bracket by unchecking it
72. Select the **Bracket** component
73. Turn on **Move**, select the arrow shown and move the Bracket left or right

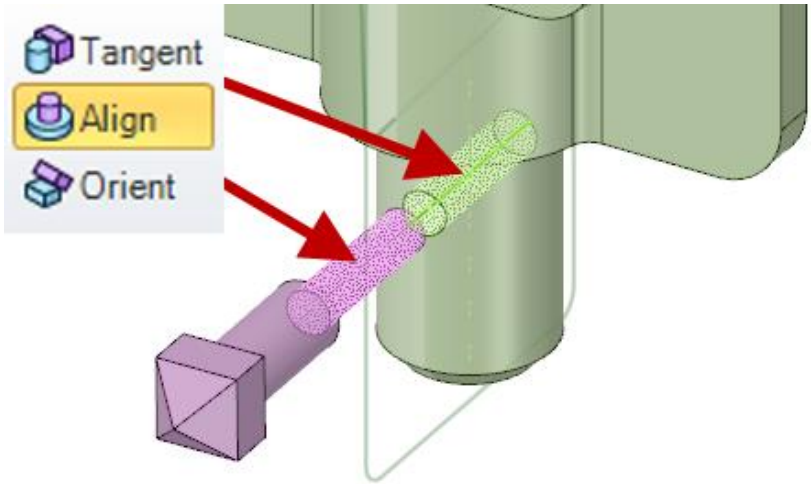


NOTICE the Screw does not move with it. This is because no assembly condition was saved in the tree, because the screw was not in a component. The component is where the condition is saved.

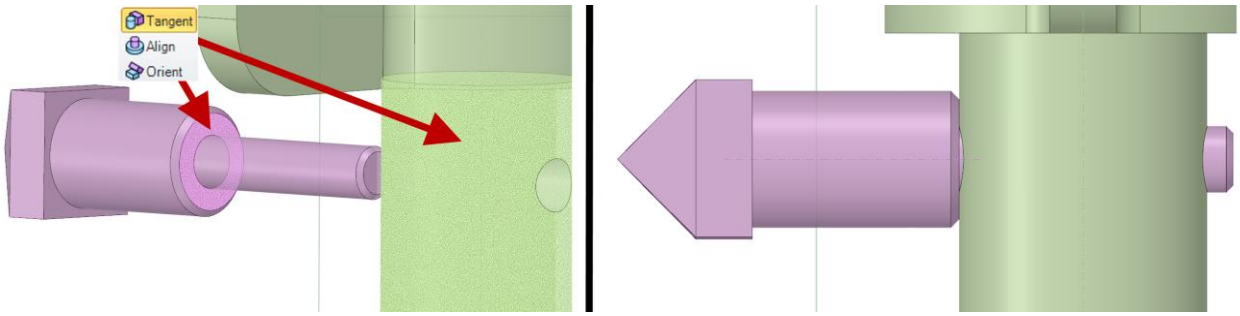
74. **Undo** to before moving the Bracket
75. Right Click on the solid at the top of the tree, **Move to New Component**, name it Base Screw



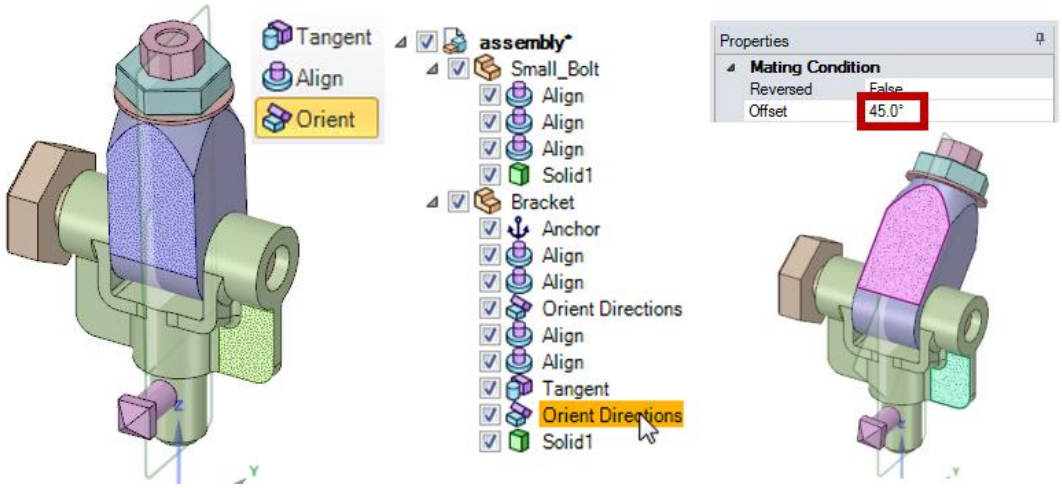
76. **Align** the shaft of the base screw to the hole in the Bracket



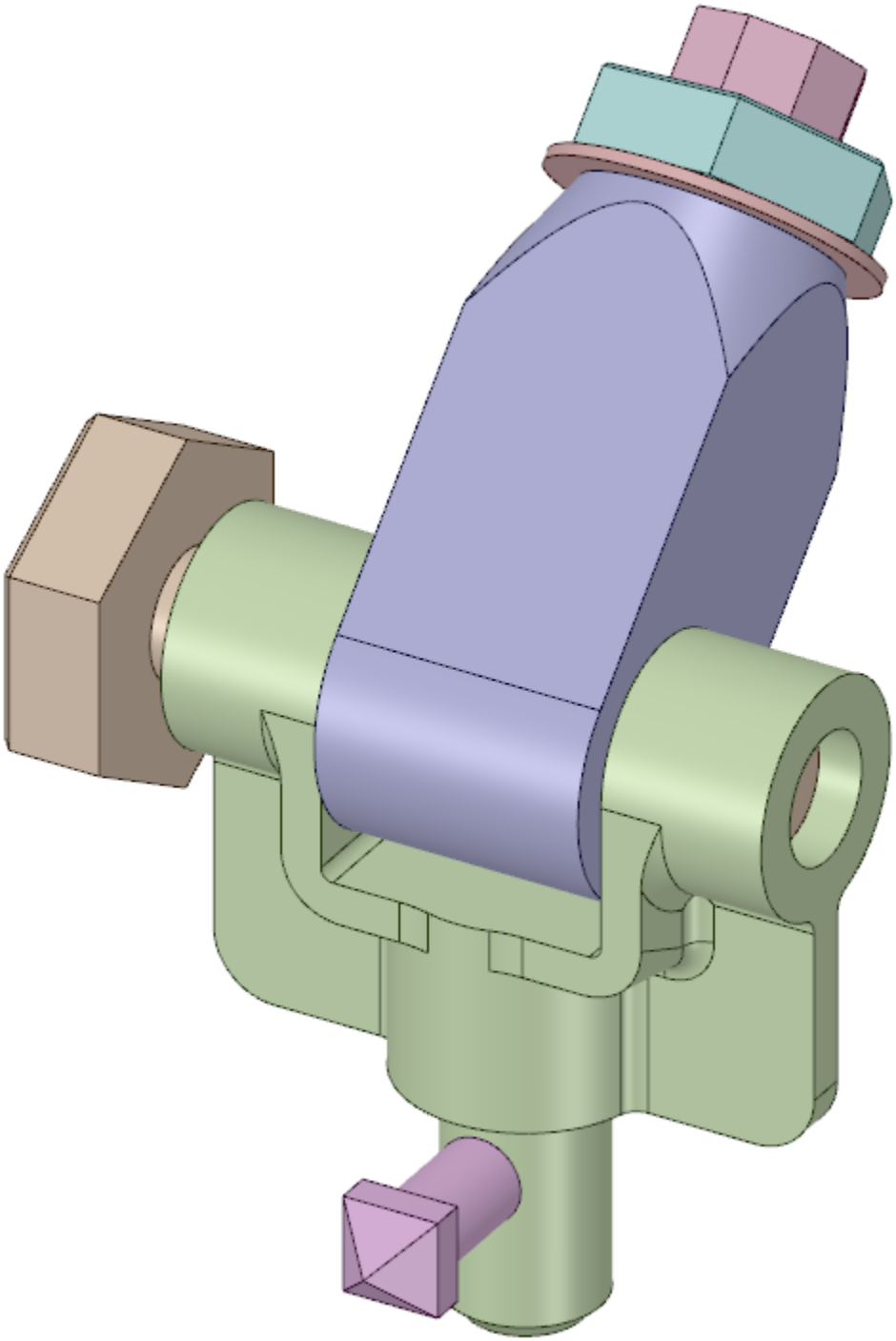
77. Zoom in on the view shown below
78. Turn on Tangent and select the 2 faces below



79. Return to a Home view
80. Turn on **Orient** and select the 2 faces below.
81. Select the last Orient Direction condition in the tree under Bracket, and enter a value of 45 for offset in the properties panel in the lower left



82.

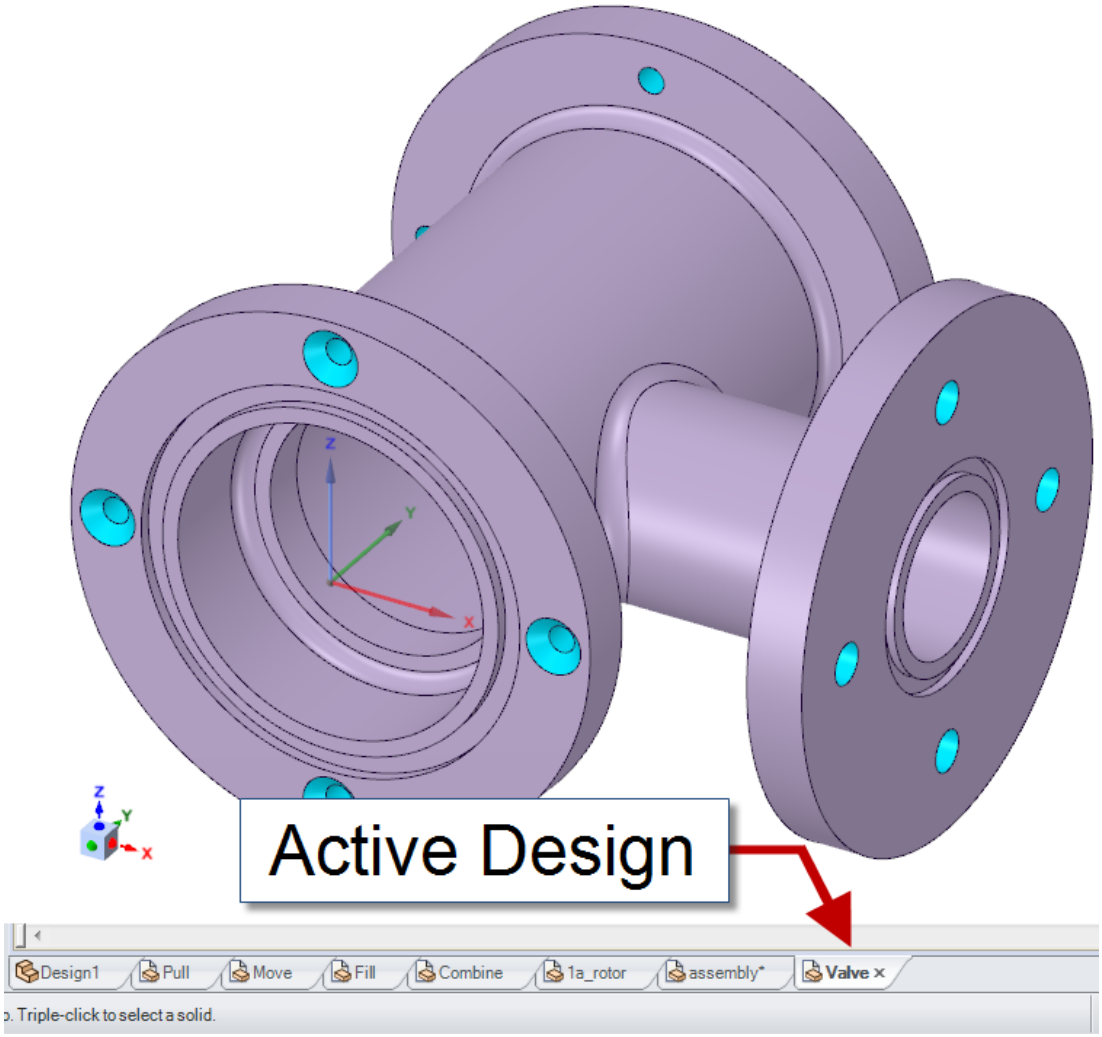


Drawings and Detailing

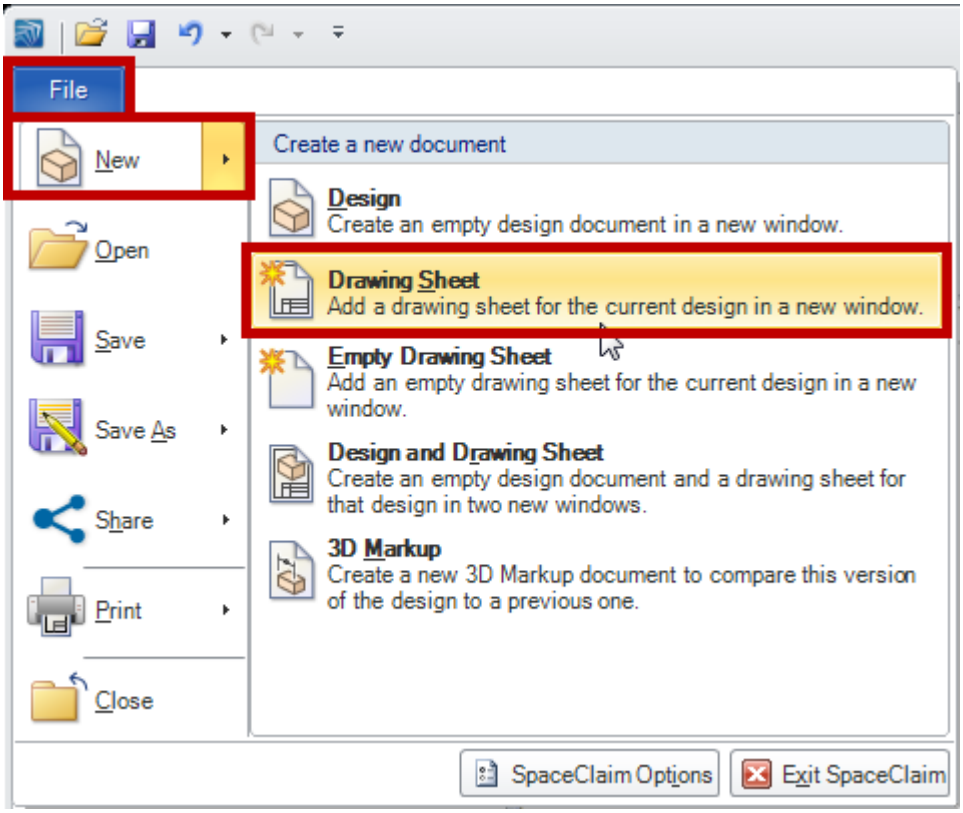
A SpaceClaim drawing sheet is a great way to communicate the details of your part or assembly design with customers, clients, suppliers and other team members. The drawing sheet allows you to highlight and convey what is important, critical and not obvious about a design when just looking at the 3D model.

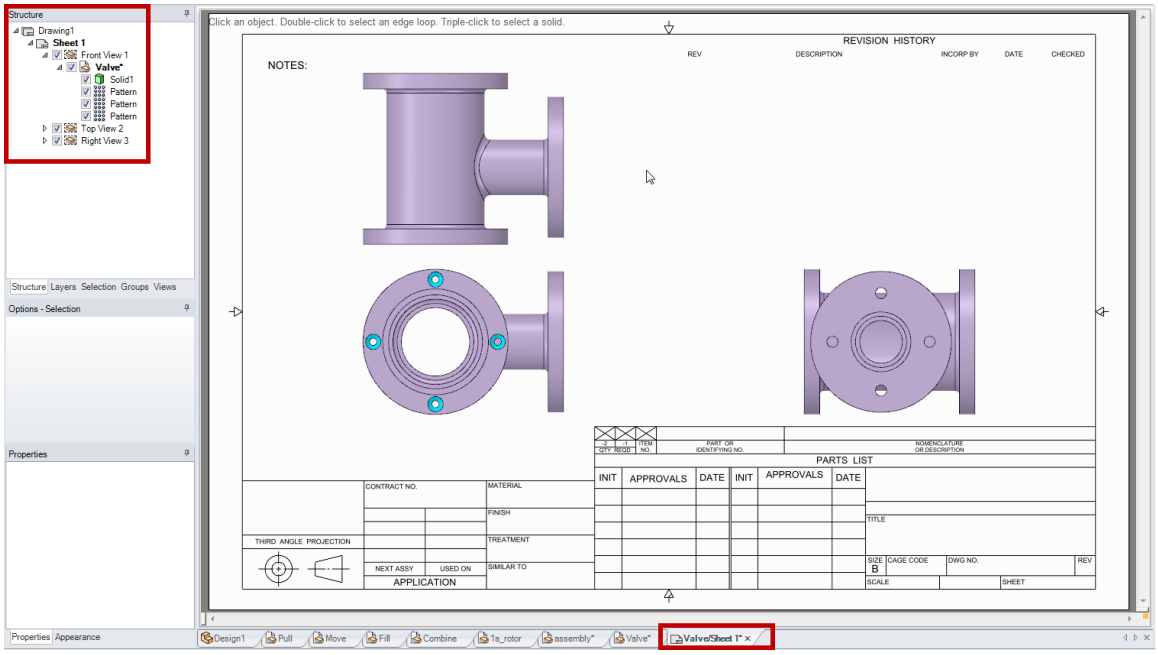
In a drawing sheet, you can add a variety of views, including projection, cross sections and details views. You can choose from a vast library of annotations like dimensions, notes, tables, geometric tolerances, datums and even weld symbols. The best part about SpaceClaim's drawings sheets, aside from general ease of use, is the ability to edit the model right in the drawing sheet during a design review while using the on screen dimensions to drive the model.

1. **File\Open** Desktop\SpaceClaim_Basic_Training**08_Basic_Detaling_2014.0** and open **valve.scdoc**
2. To create a drawing sheet, first make sure the design you want a drawing for is the active design. Clicking the tabs at the bottom of the window switches which design active.



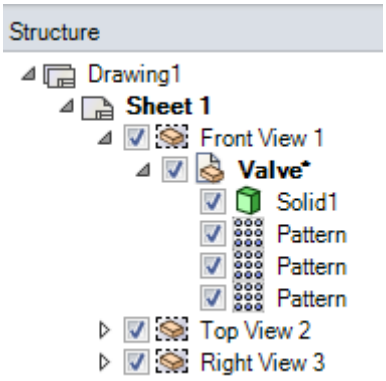
3. Click **File** in the upper left section of SpaceClaim.
4. Hover over **New**
5. Click on **Drawing Sheet**





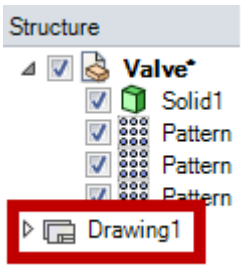
NOTICE a Drawing is started with a default Template (settable in SpaceClaim Options\detail\size and format) and 3 standard Views (Front, Top & Right)

NOTICE a new tab has been created at the bottom



NOTICE the Structure Tree has changed, showing we have a Drawing with Sheet 1 and 3 views, and what is in each view

6. Click the Valve Tab at the bottom



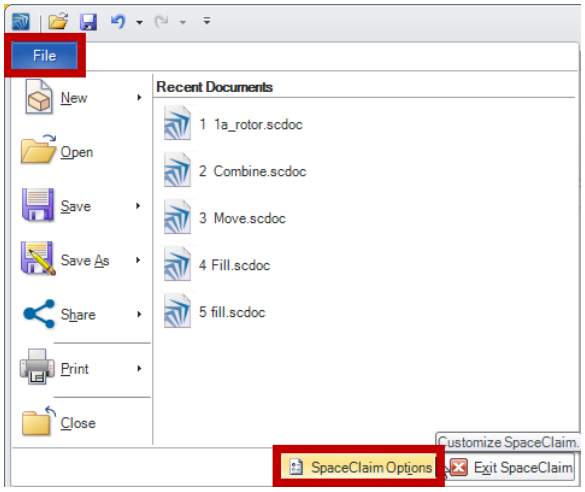
NOTICE in the Structure Tree there is now Drawing 1.

7. In SpaceClaim, there is no separate Drawing File. The Drawing is tied to the Design and File it was created from. Saving the Valve part now would save the Drawing with the file Valve.scdoc

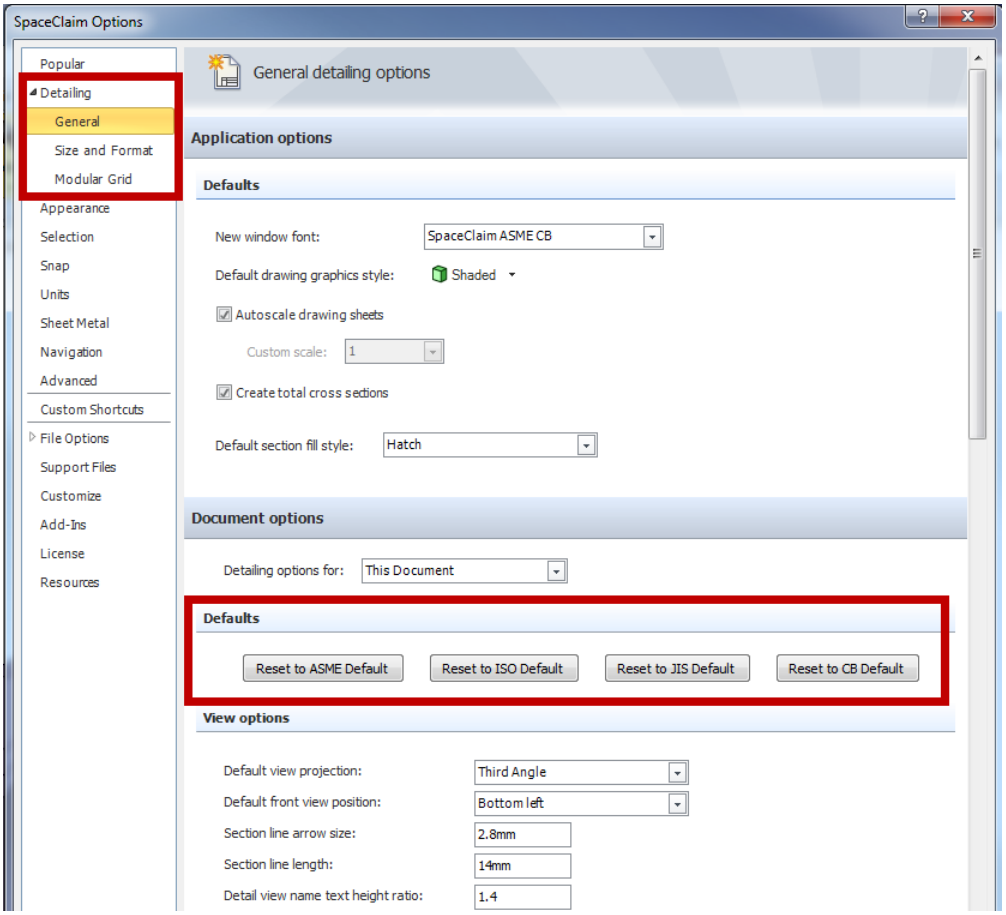
Drawing Settings

8. Click the **Valve/Sheet 1 tab** at the bottom to switch back to the drawing sheet.

9. Click **File\SpaceClaim Options**



10. Click on the **Detailing** Tab



NOTICE there are 3 subsections to the Detailing tab. In General, you can apply a universal default like ASME or ISO. Below that are individual settings for views, annotations, line style and more.

Sheet Format

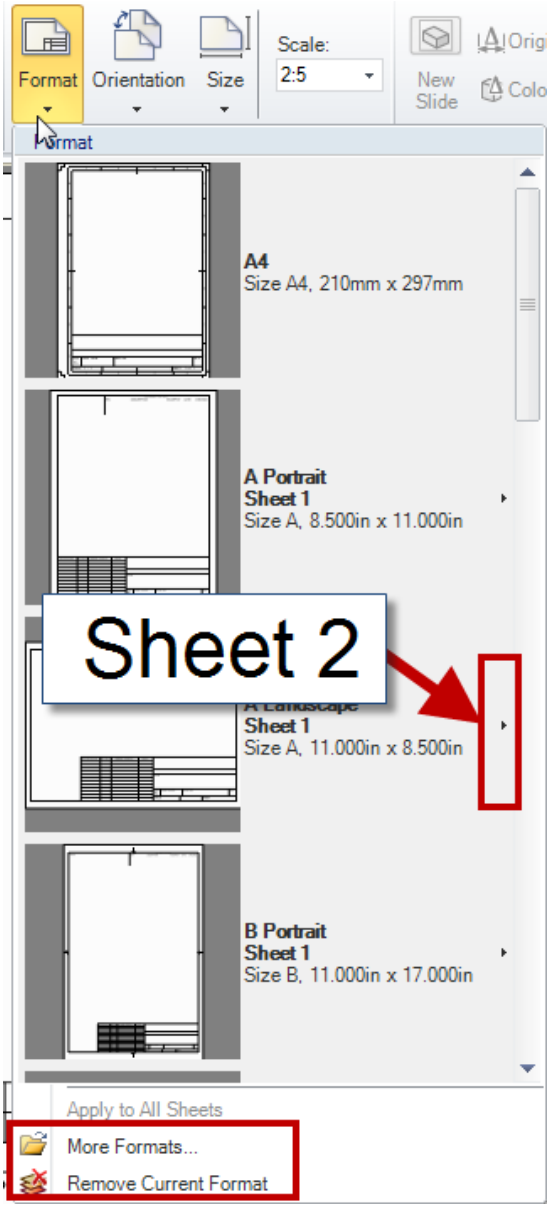
11. Close the SpaceClaim Options window

NOTICE that SpaceClaim has automatically switched tabs from Design to Detail.

The Detail tab is where you will find all the tools to create and edit your drawing sheet.

The **Sheet Setup group** on the right is where you can change **format, orientation, size and scale**.

12. Click the **Format** tool dropdown in the Sheet Setup Group



NOTICE

a. All the default preloaded sheets.

b. If you use the Format drop down, you will not need to use the Orientation or Size tool next to the Format tool. The format sets both size and orientation.

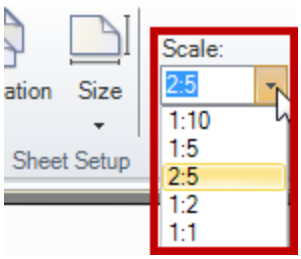
c. An arrow next to each sheet, click this to change from Sheet 1 to Sheet 2

d. At the bottom the option to browse to More Formats or Remove the Current Format altogether to have a blank sheet. If you Remove the Current Format, you would use the Orientation and Size tools in the ribbon bar to change the sheet.

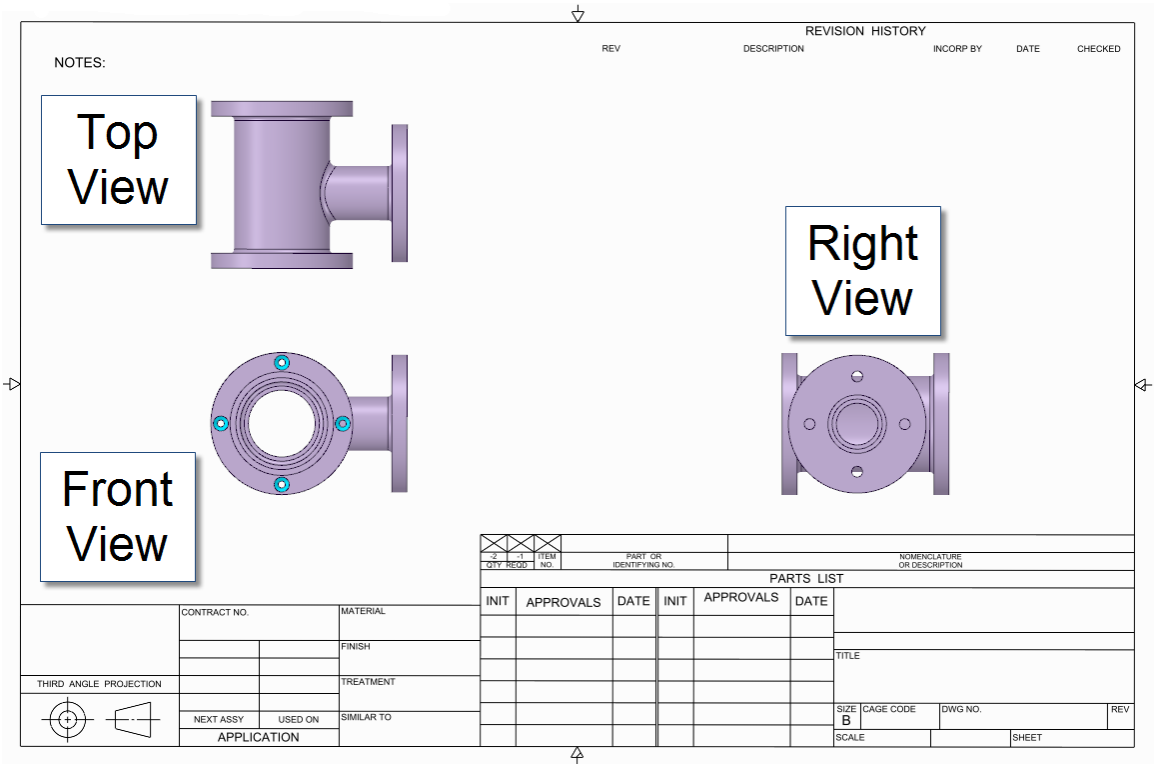
The default format is B-Size Landscape, which we'll leave as it's typically the largest workable size for many companies. However there isn't much room on the sheet to annotate the part.

13. Click the Scale Drop down in the Sheet setup group, and notice there are some default scales.

14. Click inside the Scale box, type in a value of .3.



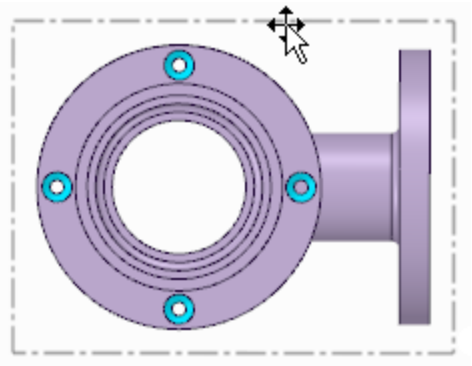
NOTICE you can type in any value. You can type in a decimal (.3) a fraction (1/3) or even a ratio (1:3).



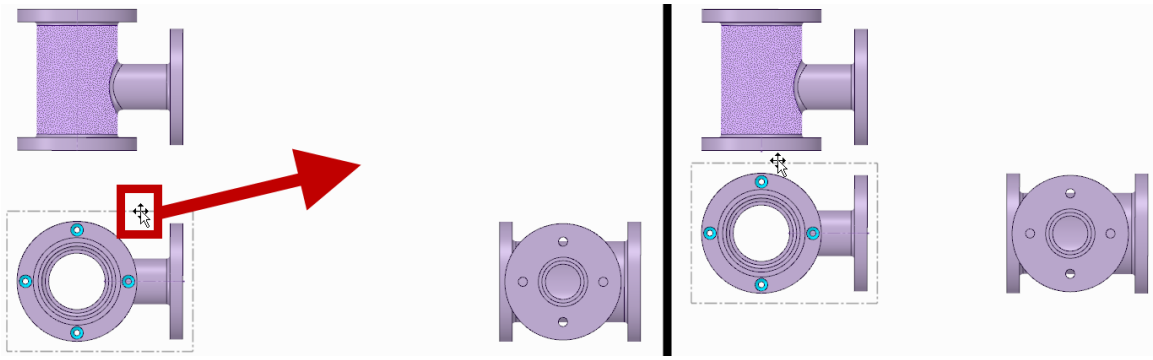
NOTICE all the views changed size. Views by default are tied to the sheet Scale you just changed.

Each model you see in the drawing sheet is a different view.

15. Hover over any of the views and notice a dashed box around the view. This dashed box is the boundary of the view, and allows us to do a few things to the view

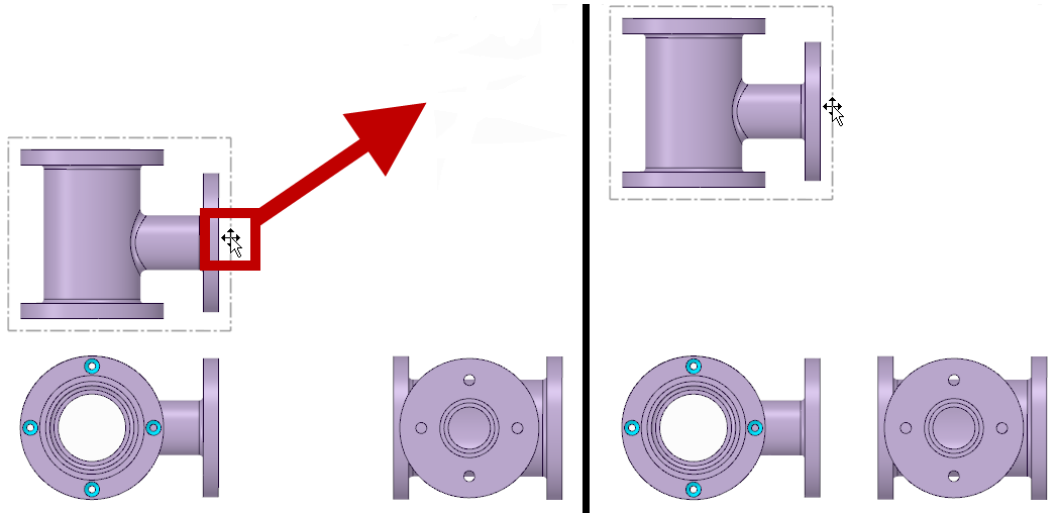


16. Drag the Front View (bottom left) around.



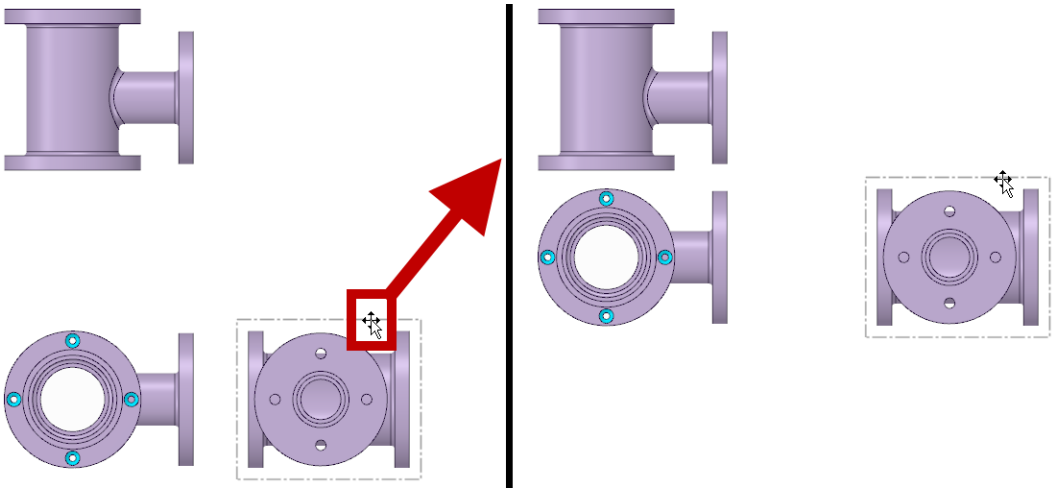
NOTICE all the views move when you drag the Front View around

17. Drag the Top view around

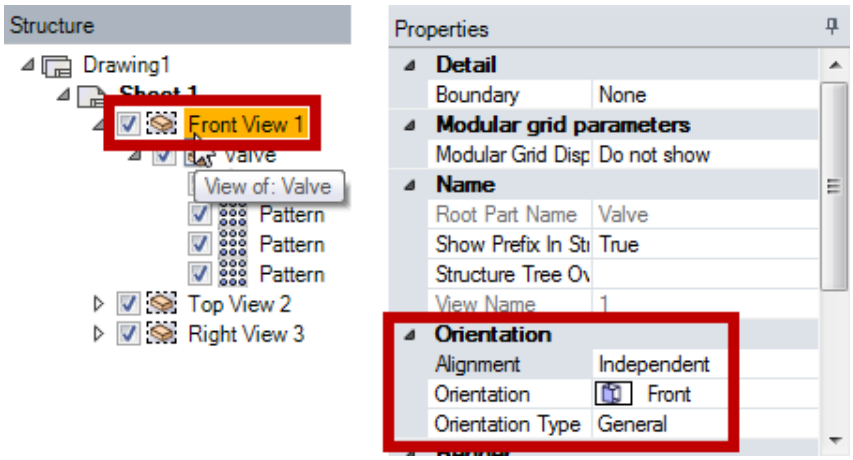


NOTICE that only the Top and Front view move when you drag the Top view around.

18. Drag the Right View around and notice only the Right and Front View move

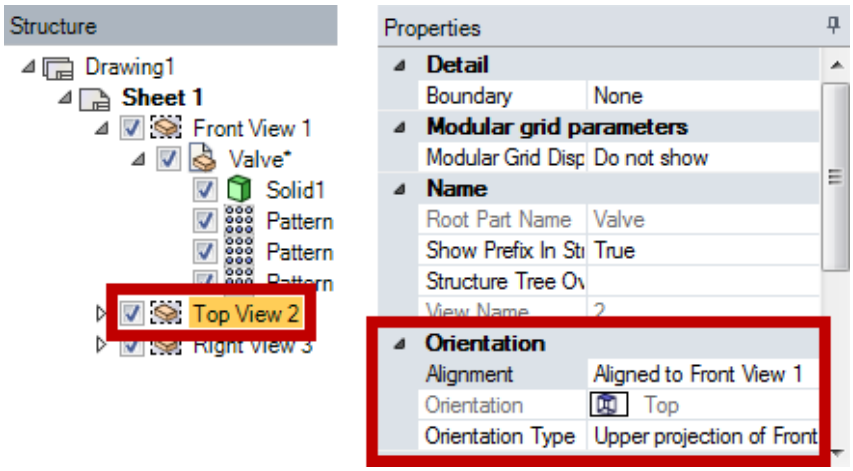


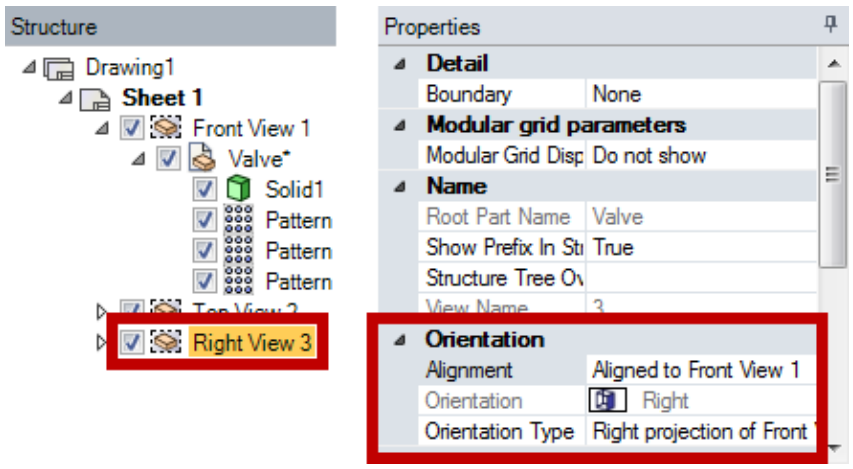
19. Click on the Front view in the Structure Tree and look at the properties panel in the bottom left.



NOTICE the properties for the Front view state it is an Independent General view.

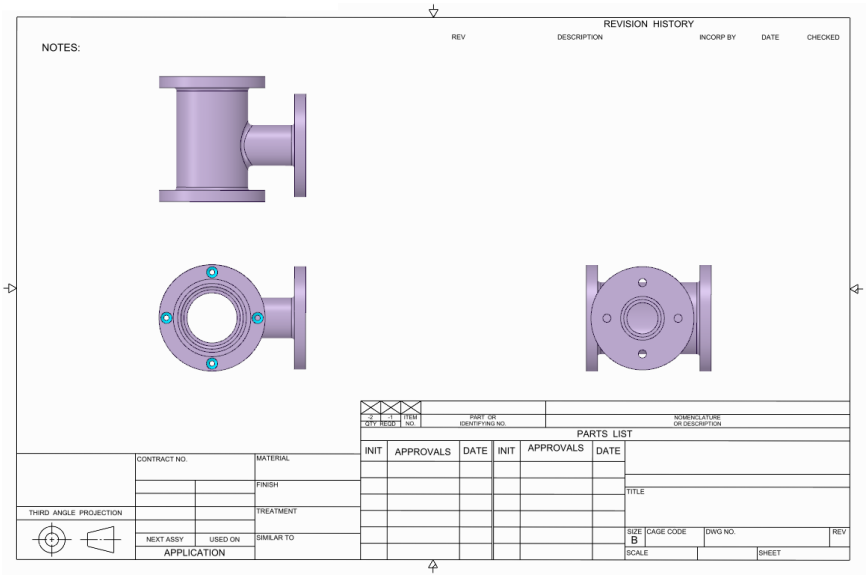
20. Select the Top and Right views and look at the Properties Panel.





NOTICE both the Top and Right view are aligned to and projections of the Front view.

21. Drag the views back to their approximate original locations, or Undo the dragging of the views

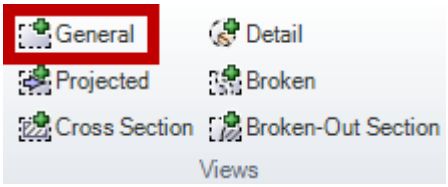


Adding and Modifying Views

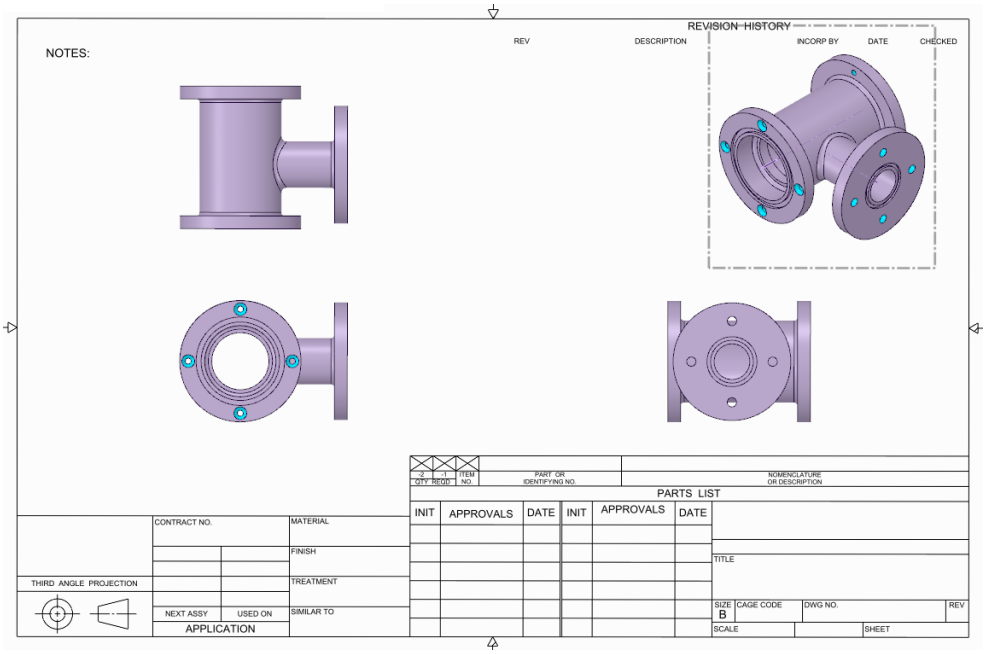
In the Detail tab there is a Views Group which allows you to Create or Modify views.

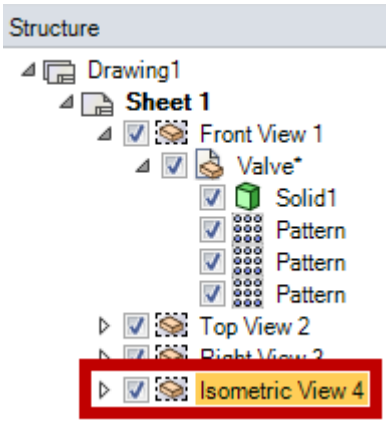
22. Click the **General** view tool

23. Move your cursor onto the drawing sheet and notice the new general view attached to the cursor



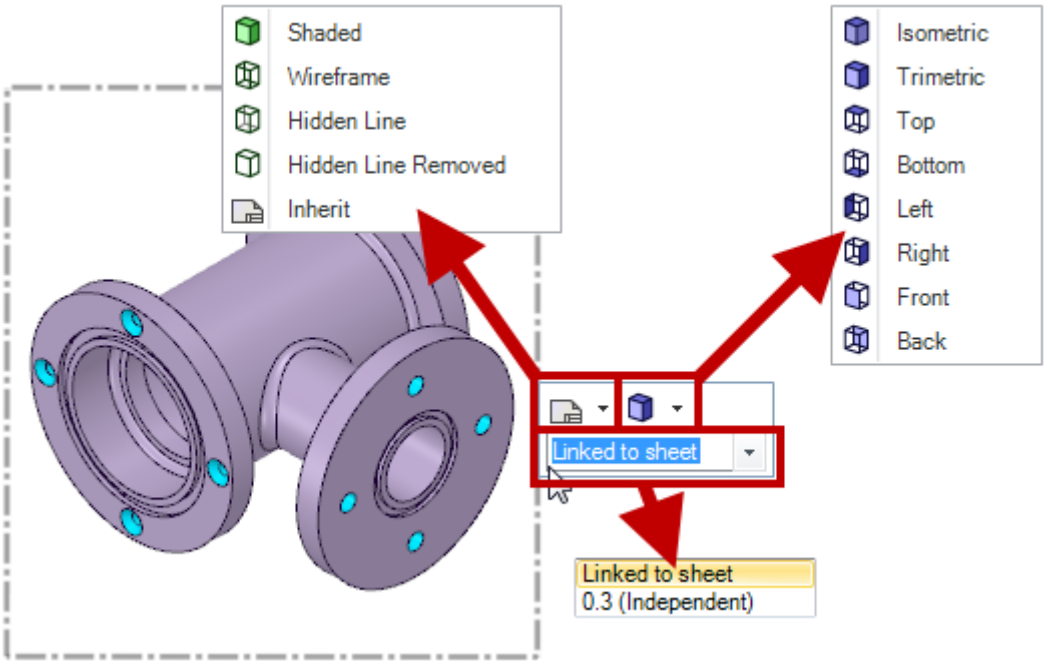
24. Click in the upper right section of the drawing to add the default Generic View, the Isometric view.





Notice like other views, the new General Isometric view can be dragged around by its border, and it has been added to the Structure Tree.

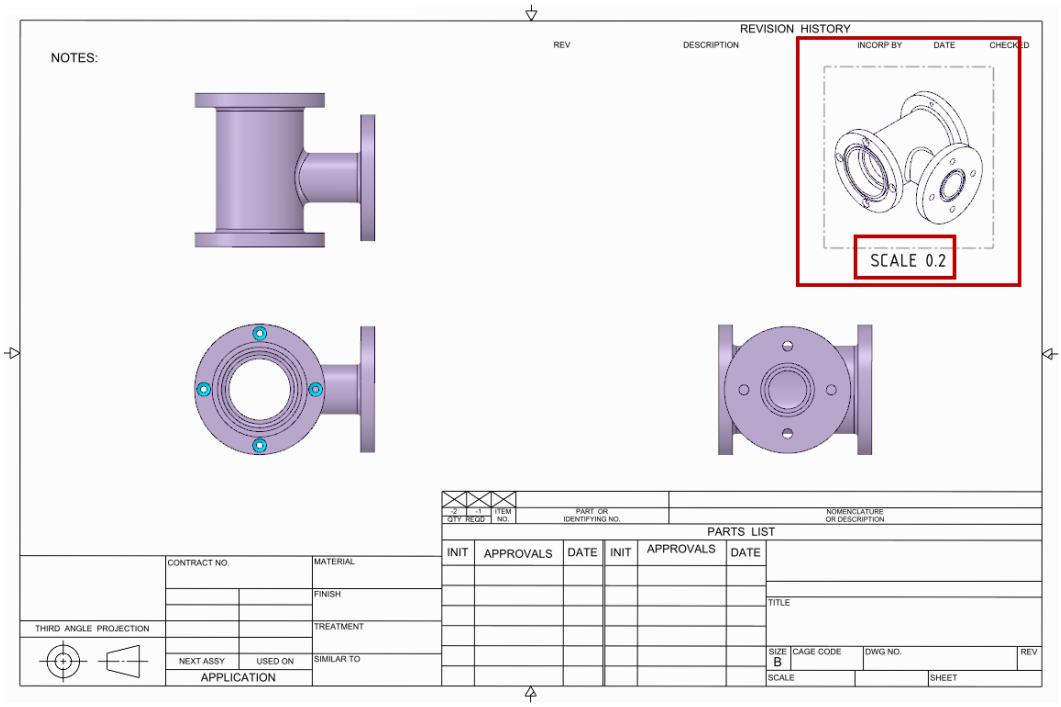
25. Click on the border of the new Isometric view, and notice a Mini-Toolbar start to appear near the cursor. Move the cursor to the Mini-Toolbar to illuminate it.



NOTICE there are 3 different buttons in the **Mini-Toolbar**

- The upper left button is **Graphics Rendering**
  - I.e. shaded, wireframe, hiddenline removed etc.
- The upper right button is the **View Orientation**
  - I.e. isometric, front, top right
- The bottom button is the **Scale**
  - I.e. Linked to sheet scale, or independent.

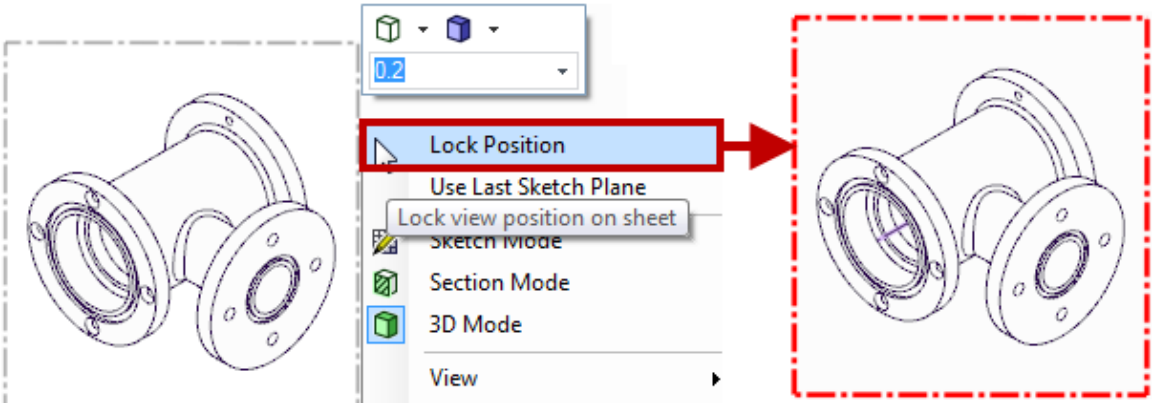
26. Using the Graphics Rendering button in the Mini-Toolbar set the graphics to Hidden Line Removed
27. Using the View Orientation button in the Mini-Toolbar set the view to Trimetric
28. Click where it says Linked to Sheet in the Mini-Toolbar and type in .2



NOTICE when the scale is not linked to the sheet, the scale is displayed below the view

IMPORTANT: the native display in SpaceClaim is a Shaded Solid. Displaying any of the views as anything other than Shaded (i.e. Wireframe, Hidden Line Removed, etc.) takes extra processing power. It is recommended, especially on lower performing computers, to change the Graphics Rendering Last, before saving or printing.

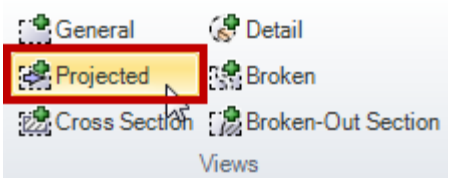
29. After placing a view were you want it, Right Click on the view, and choose Lock Position to prevent the view from being dragged around, accidentally or intentionally



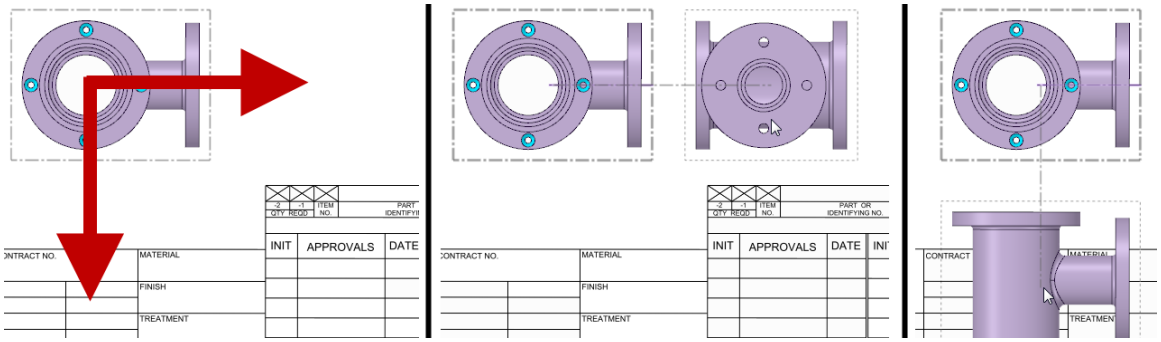
NOTICE after locking the view, the outline is now red, indicating it is lock. You can right click to unlock

The Front view (bottom left) in the drawing is a general view with the View Orientation set to Front.

The Top and Right view are projection views.



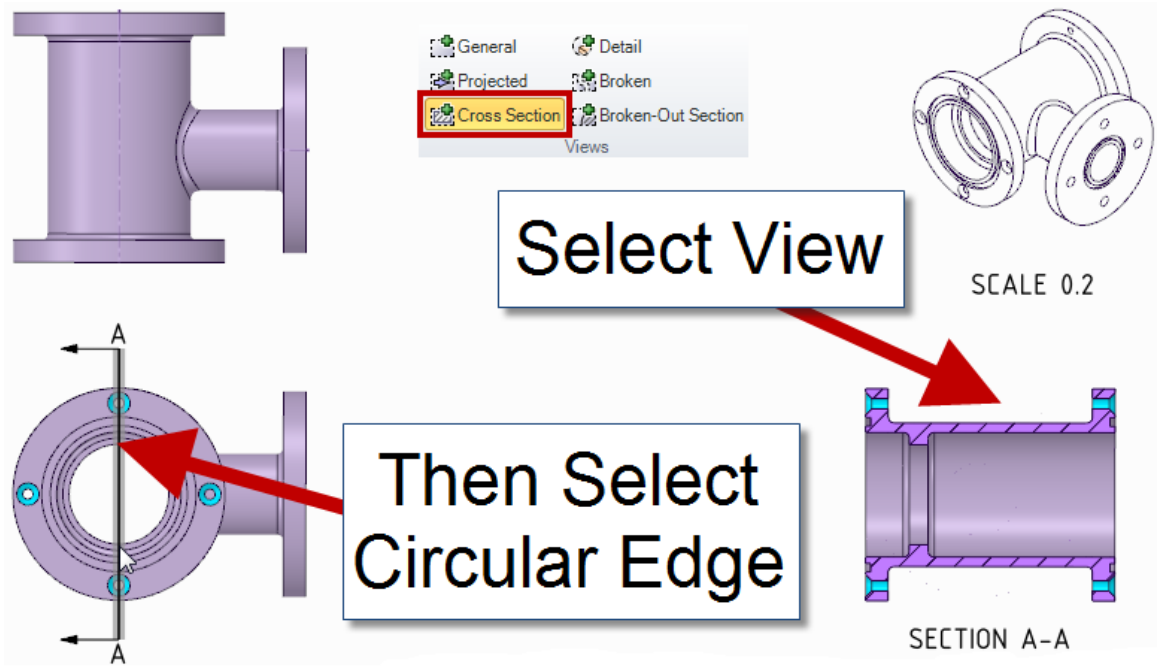
30. To create a new **Projected** view, click the Projected tool in the Views group.
31. Click in the white space in an existing view - the Front View - to project from. (Do not click a Face or Edge)
32. Press Escape to cancel out of creating the projected view, since this drawing has enough views.

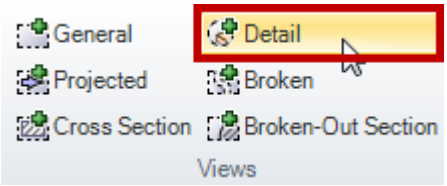


33. Click the **Cross Section** tool.

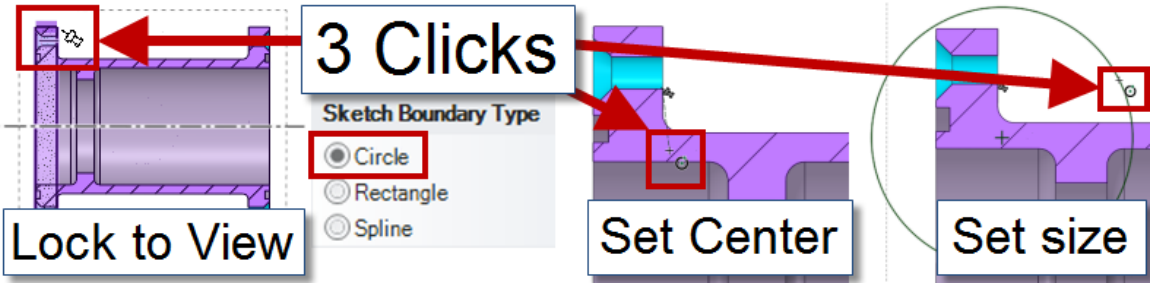
NOTICE that the Cross Section tool is one of the view tools that by default converts an existing view instead of creating a new view (look at the status message in the upper left)

34. Click the Right view in the bottom right
35. Click the top of one of the circular Edges in the Front view to snap the Section arrow to the center

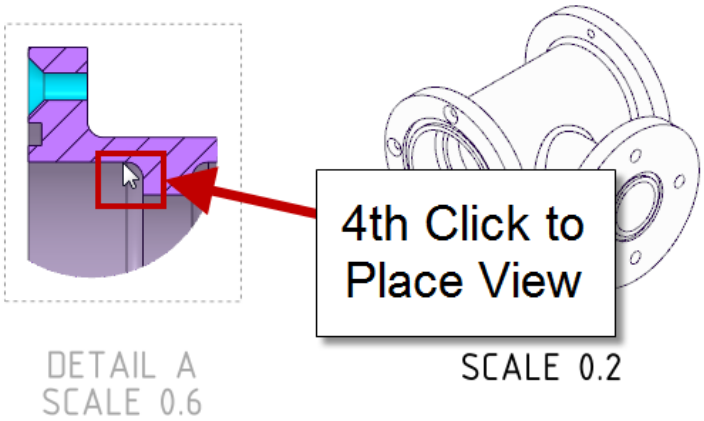


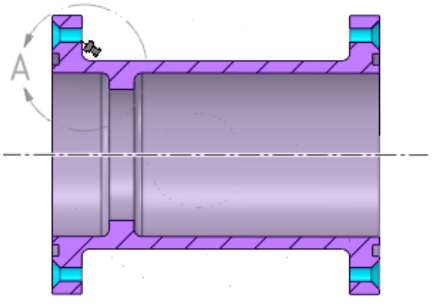


36. Click the Detail tool in the Views group
37. The Detail tool requires 3 clicks to setup the view, and a 4th click to place the view
    a. .1st Click locks the new Detail view to an existent view. The detail view with have a scale of 2X of the references view. Click any face or edge in the sectioned right view
    b. 2nd click starts the center of the Circle. NOTICE in the panel on the left, there are different boundary types, including Rectangle and Spline
    c. 3rd click finishes the circle and sets the boundary/size of the detail view.



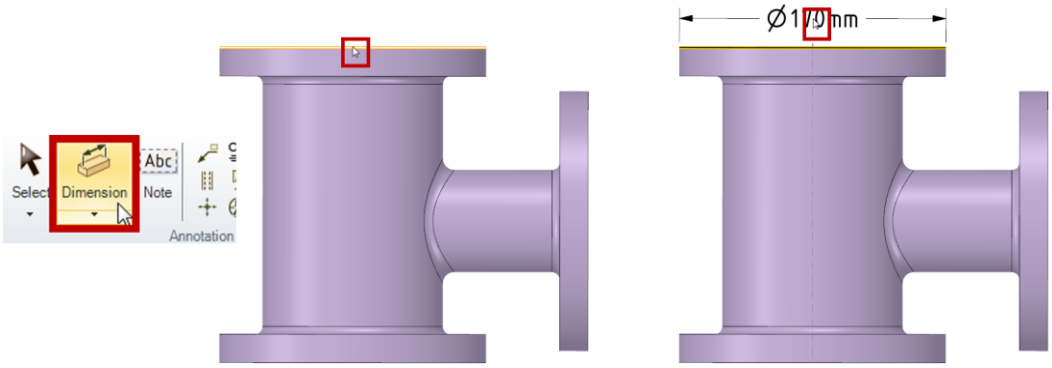
    d. 4th click will place the new Detail view after moving your cursor to the desired location





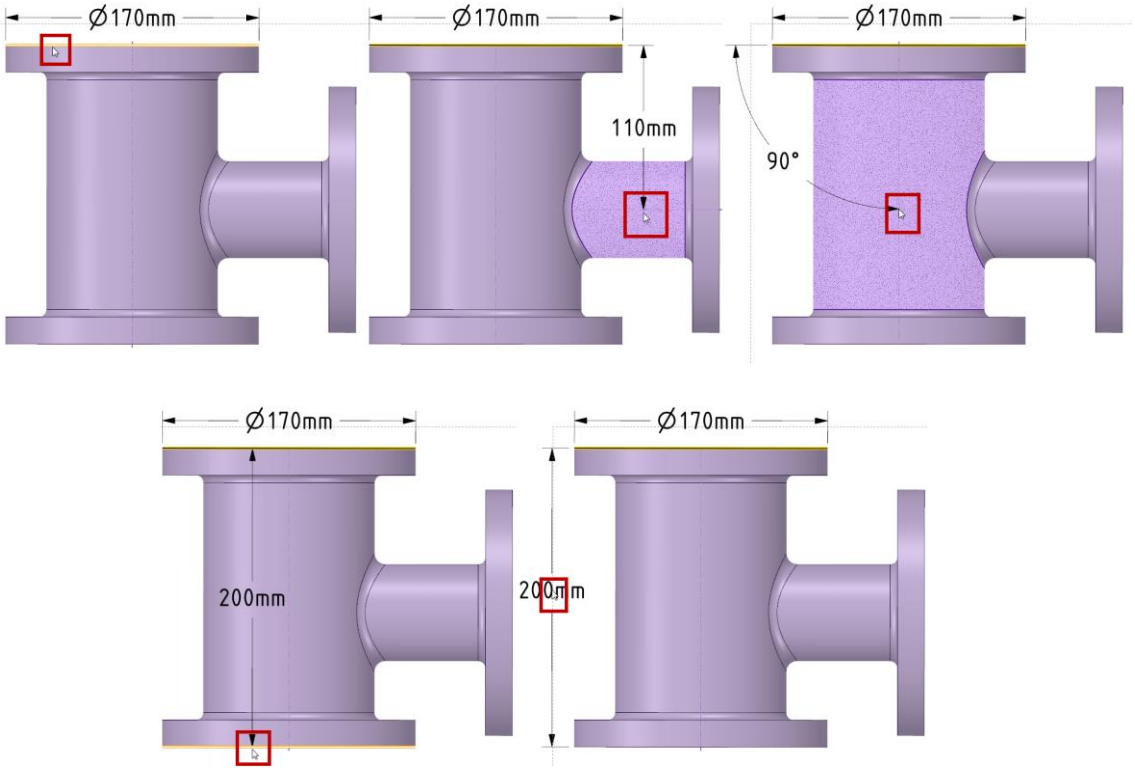
Dimensions & Callouts

38. Zoom in on the Top view
39. Turn on the Dimension tool in the Detail tab
40. Select the top edge of the Valve.
41. Move the above the edge into white space and click to place the dimension

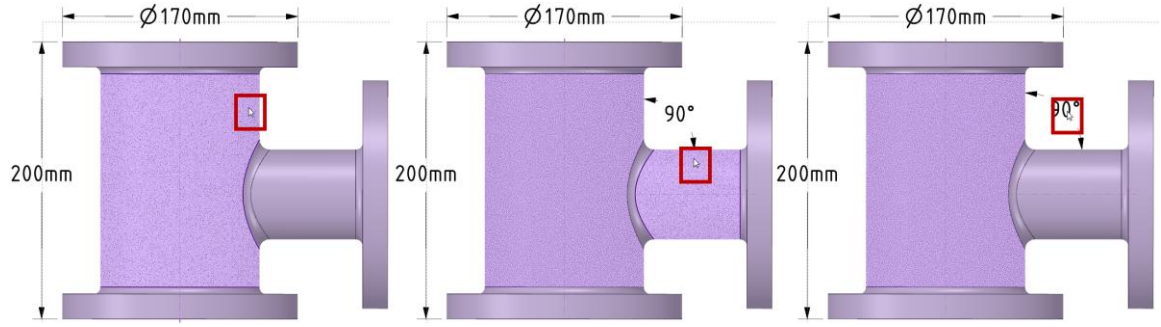


NOTICE a diameter symbol has automatically been added to the dimension

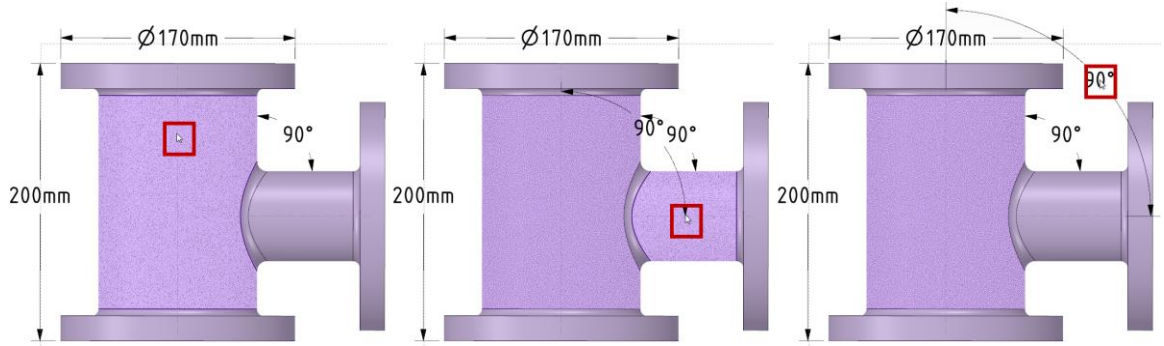
42. Click on the top edge again
43. Instead of clicking in whitespace, hover over different faces and edges in the model
44. Select the bottom edge and Click in white space



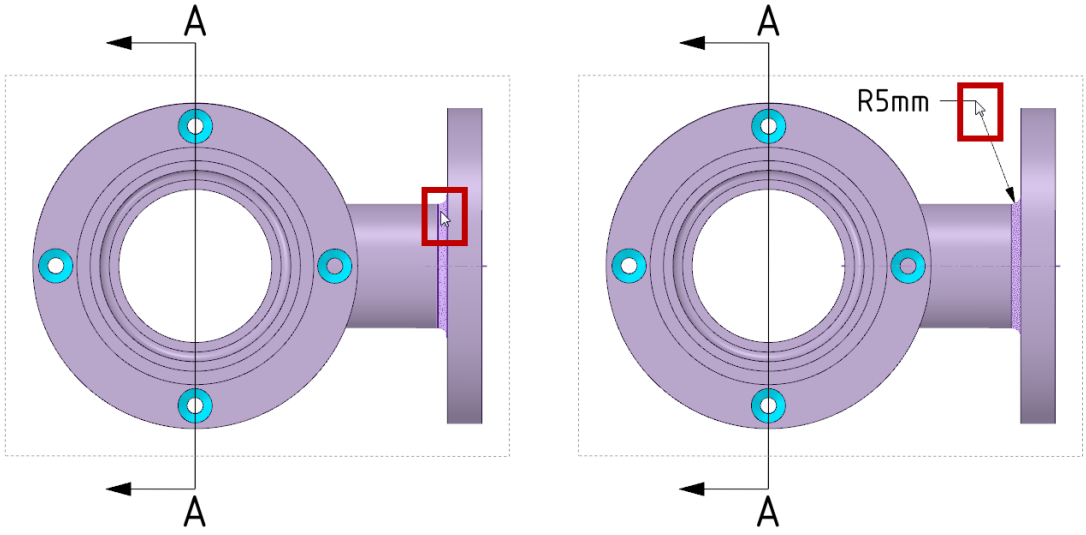
45. Select the large cylindrical face
46. Select the small cylindrical face
47. Click in white space to place the angular dimension



NOTICE clicking toward the sides of the cylindrical; face snaps the dimension arrow to that side. Clicking in the middle of the cylindrical face along the centerline snaps the arrow to the centerline.

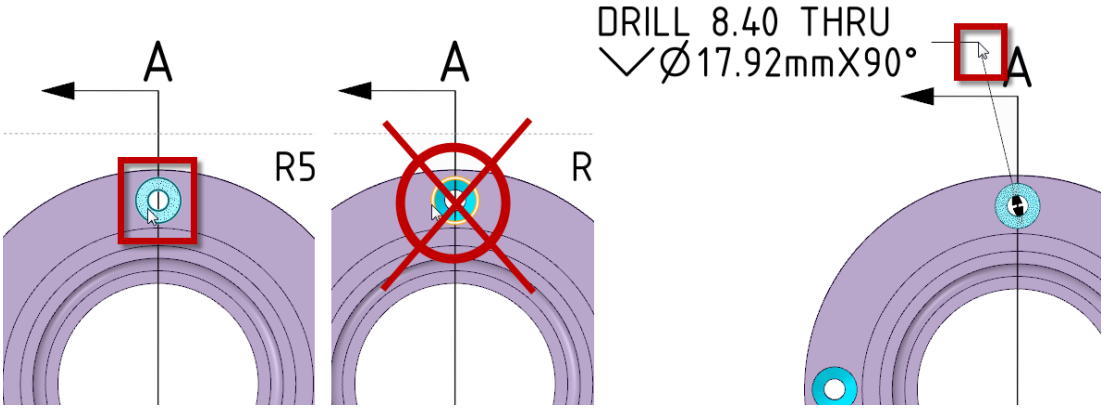


48. Pan down to the Front view (lower right)
49. Click on the Round and click in white space to place the dimension

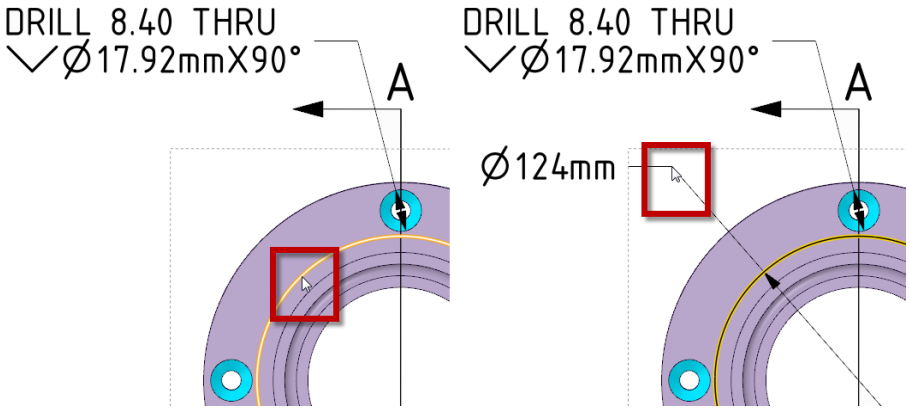


50. Select the blue face of the hole on the top of the Front view. This is a Standard hole and carries manufacturing information about the hole.

IMPORTANT: Do NOT Click the edge of the hole, shown below, highlighted in orange. This will not display the manufacturing information. This will Dimension the diameter of the Edge.

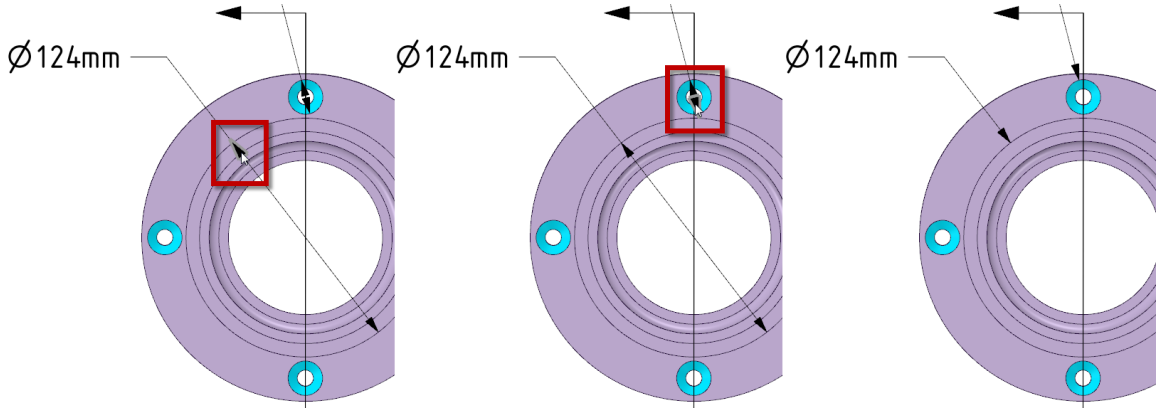


51. Click on the 2md outermost circular edge, click in white space to place the dimension



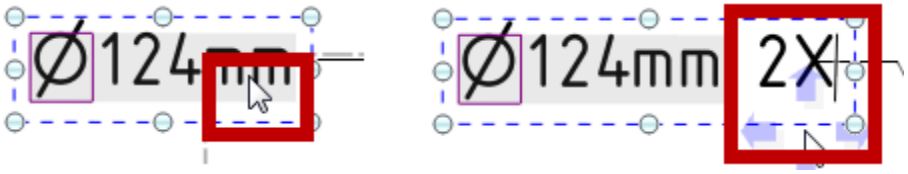
You may want to change the Leader type; a double arrowed line across the edge is not desired

52. Click on either arrow on the new dimension to change the type until it gets to the type below

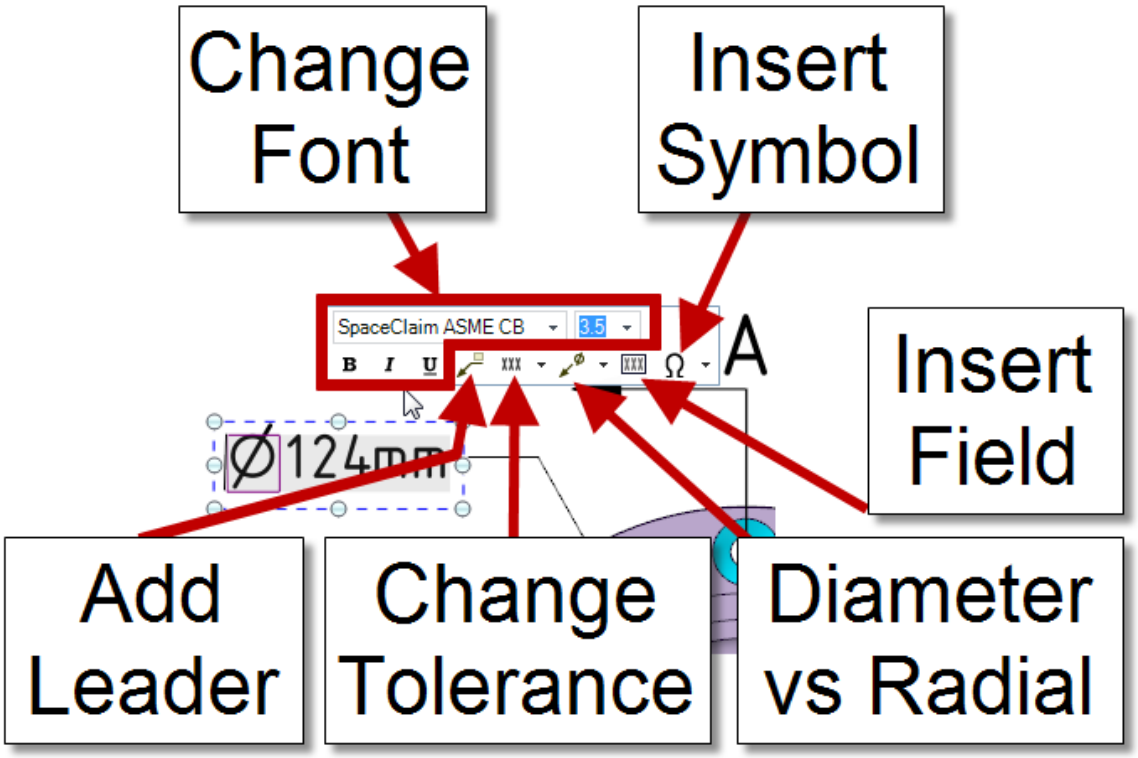


Editing Dimensions: There are a few different ways to edit a dimension.

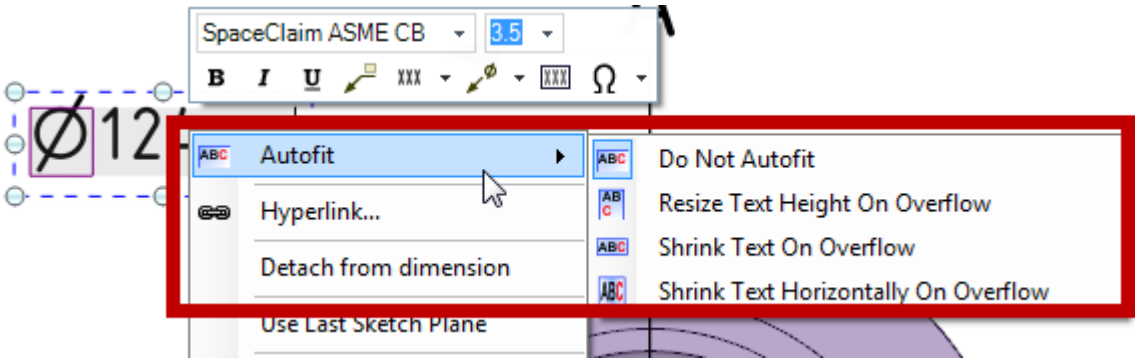
53. Left click on the dimension to edit the text, i.e. to add “2X” or “All around”



54. Left or Right Clicking on the dimension will bring up a mini toolbar, with buttons to edit the font, edit the tolerance and insert symbol

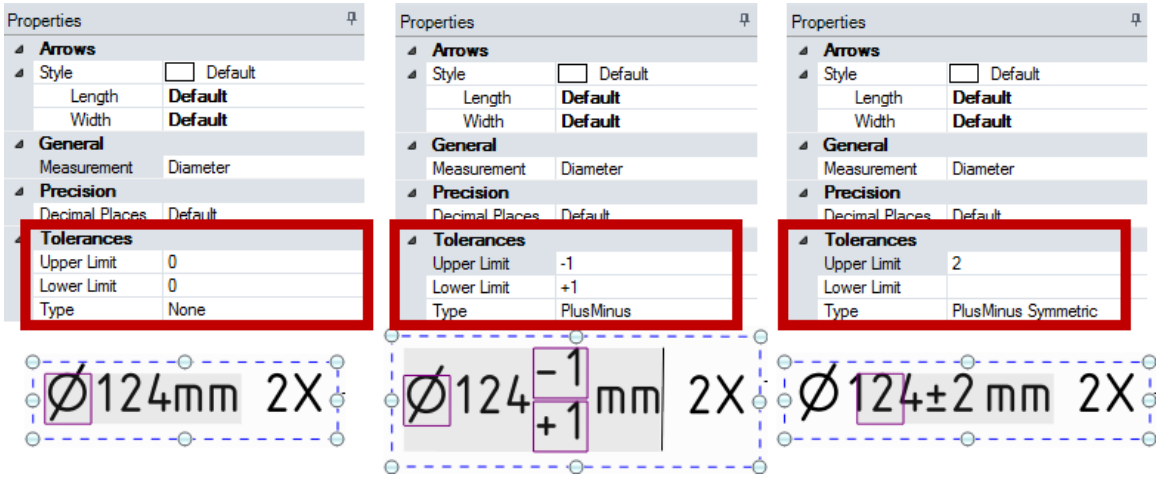


55. Right clicking also brings up a context menu with a few extra options



56. Select the dimension and look to the **Properties** panel in the lower left.

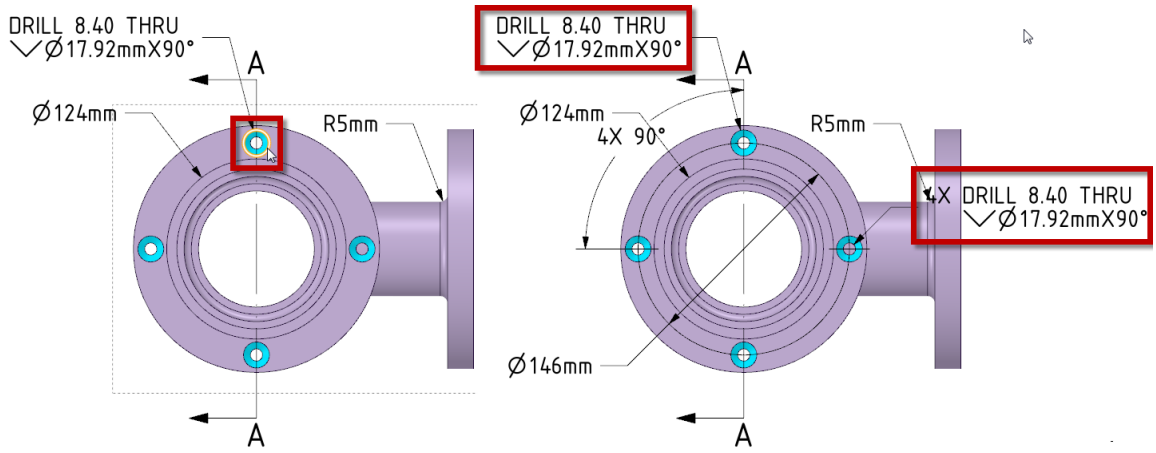
57. Change the Tolerance to **PlusMinus** and enter values, and repeat for **PlusMinus Symmetric**



NOTICE the Properties panel has many of the same Dimension setting as right or left clicking on the dimension. Style and Decimal places, or precision are the only options only found only in the Properties pane

58. Click the Bolt Circle tool

59. Click anywhere on one of the blue holes, even on an edge of the hole

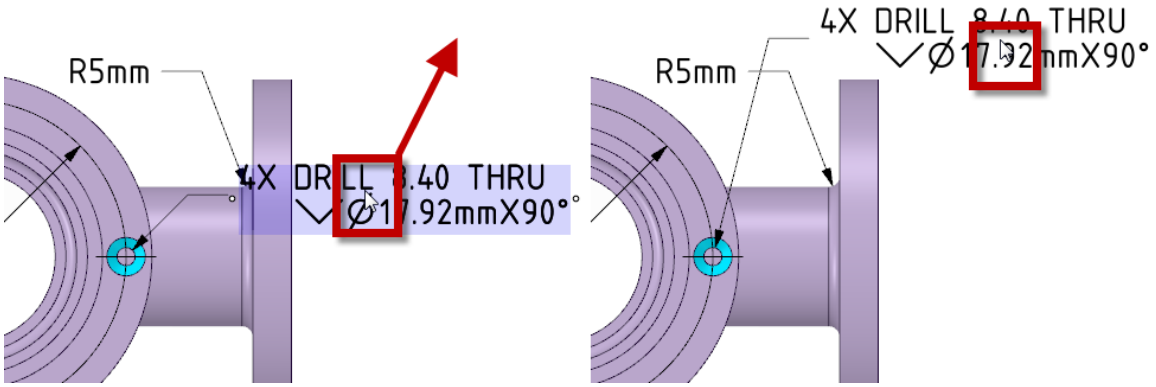


NOTICE the bolt circle tool includes a dimension of the Standard hole like we just placed, except it has a “4X” which is more accurate. However it is in a bad location.

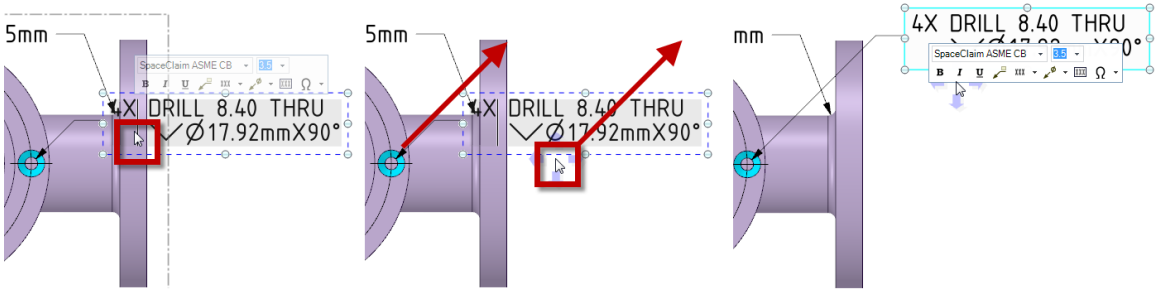
IMPORTANT: the bolt circle was automatically created because the holes are part of a Pattern.

Any Dimension can be moved by dragging it around the drawing sheet.

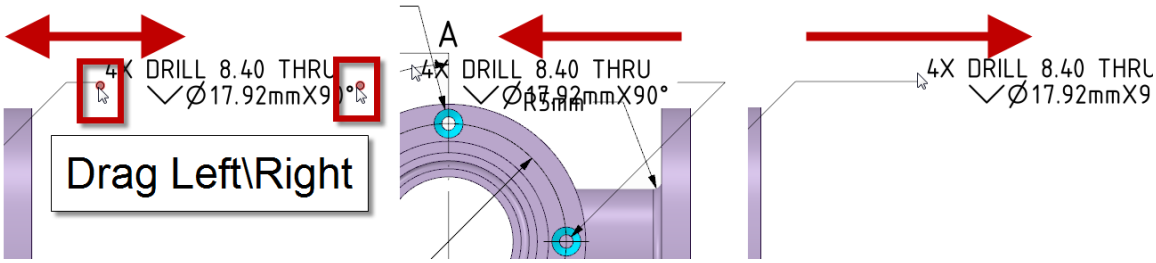
60. To drag around in any direction, select and drag in 1 action as below



61. To drag in the direction of the leader with the arrow only, Click once to select and drag the border around the text as shown below



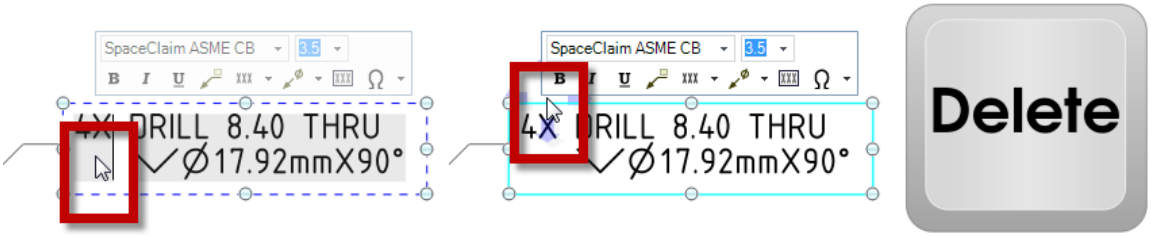
62. To drag the note left and right only, drag the red dot shown by hovering over the sides of the annotation.



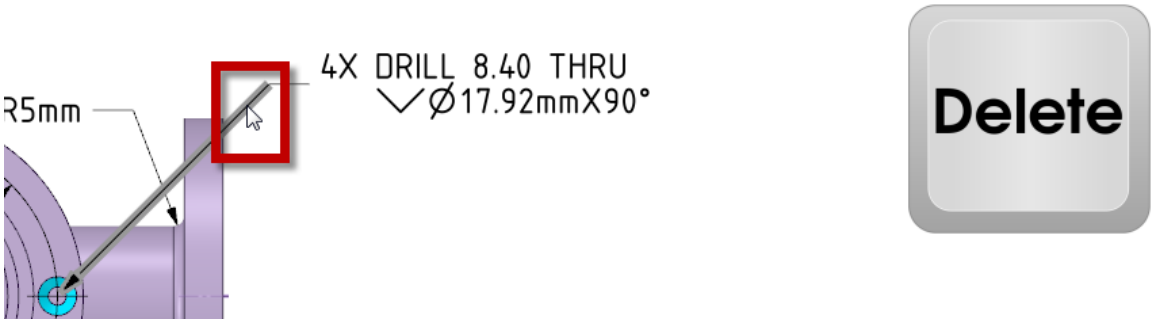
63. To Delete a dimension, either

a. Click once on the dimension

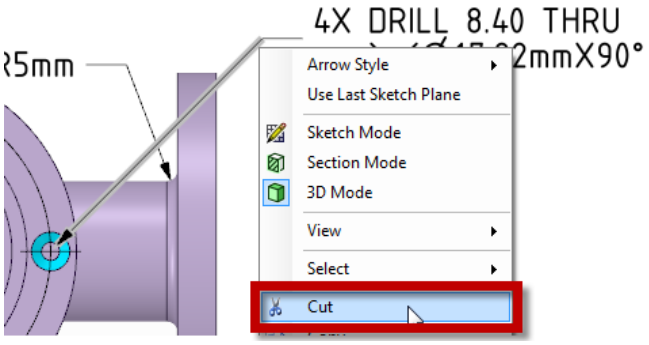
b. Click a second time on the border of the dimension, and press Delete on the keyboard



c. Or Select the leader with the arrow and Press Delete on the Keyboard

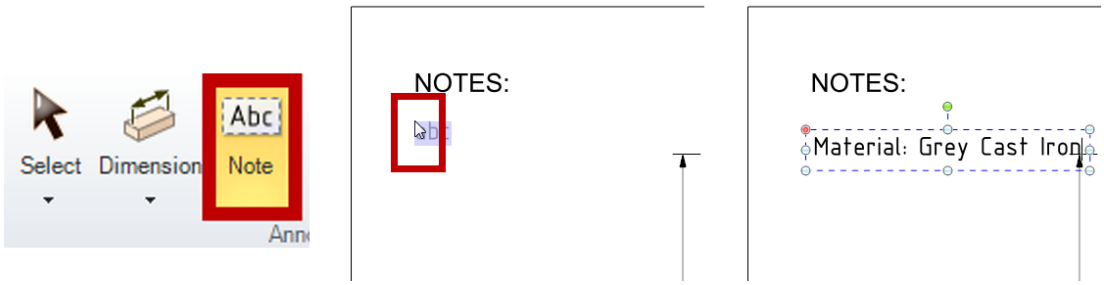


d. Or Right Click on the leader with the arrow and select Cut



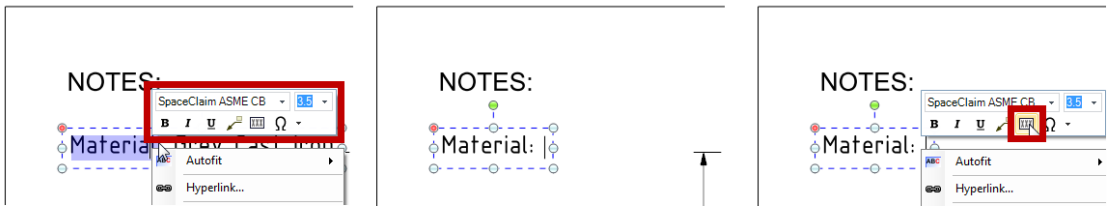
Notes

64. Click the Notes tool in the Annotation group of the Detail tab
65. Click below the word “NOTES” in the upper left of the drawing sheet to place the note.
66. Type “Material: Grey Cast Iron



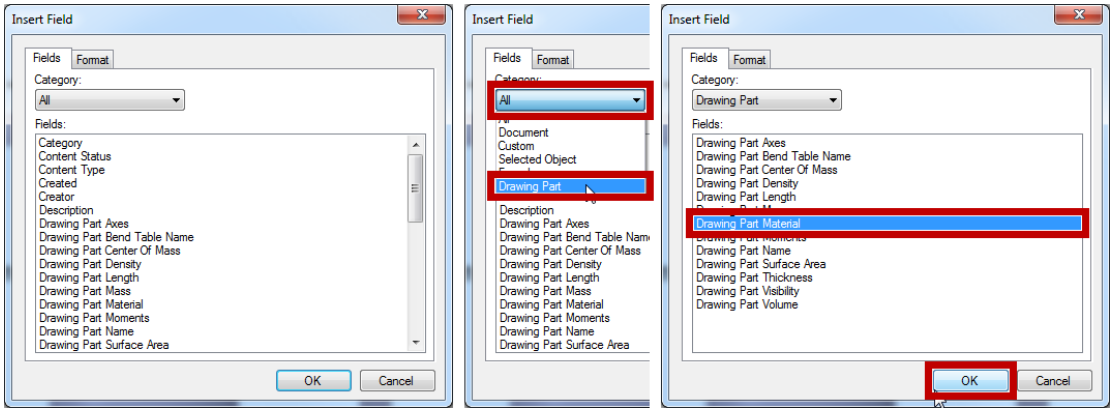
NOTICE: anything can be typed into the note field

67. Notes have the same left and right click menus as dimensions for editing the text
68. Delete “Grey Cast Iron”
69. Right or left click on the note and select the Insert Field button from the Mini-Toolbar

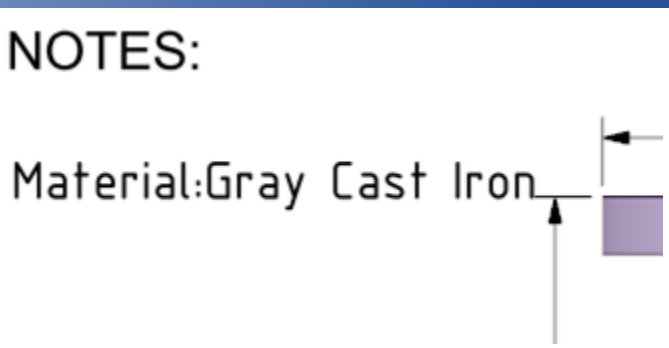


There are many properties that populate note field that can be extracted from a part file, such as mass, volume, revision, file name or material to name a few. The insert field tool allows you to insert any of these. Some will need to be entered or assign to the design, some are automatic.

70. In the Insert Field window, select Design part from the drop down, Design Part Material and OK

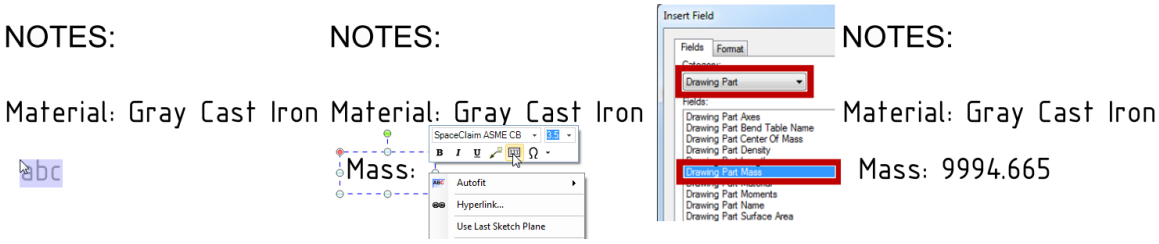


NOTICE all the different properties in the insert field drop down to choose from



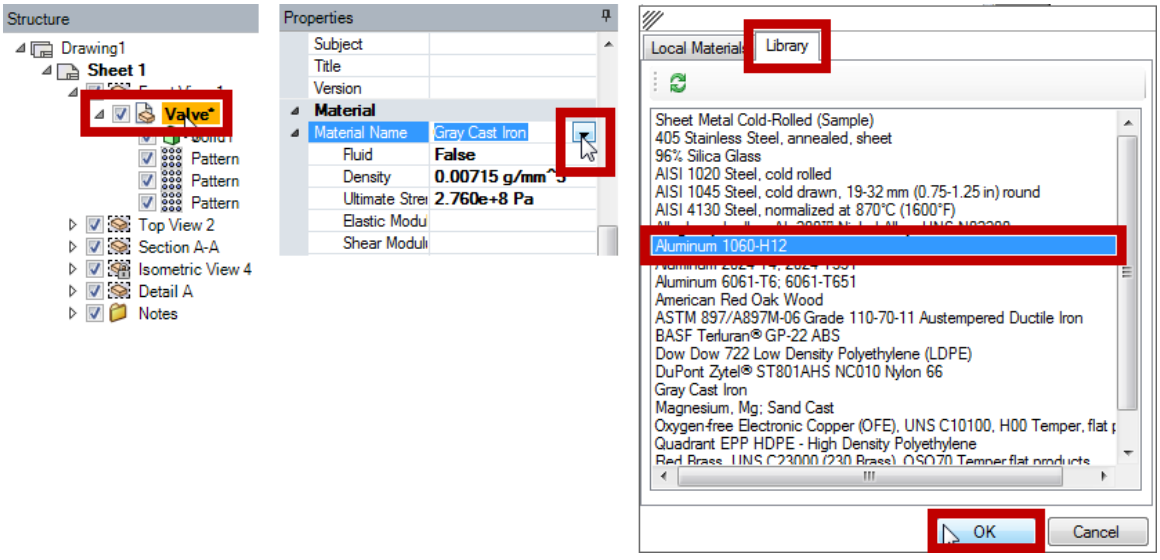
NOTICE the note says again that the material is Gray Cast Iron

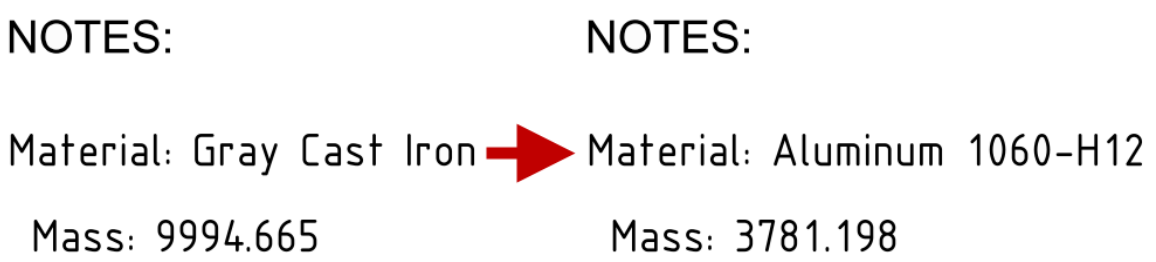
71. With the Notes tool on, add another note below the first,
72. type in “Mass”
73. Right or Left click on the note and click the Insert Field tool
74. From the Drawing Part menu, select Drawing Part Mass



75. Select the Valve component in the Drawing’s Structure Tree.
76. In the Properties panel in the lower left, click the down arrow in the Material Name row

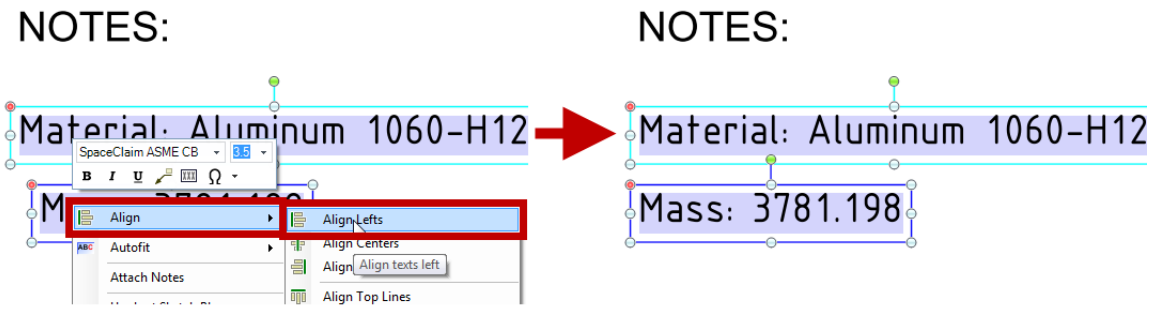
    Click the Library tab in the Material window and choose the first Aluminum.



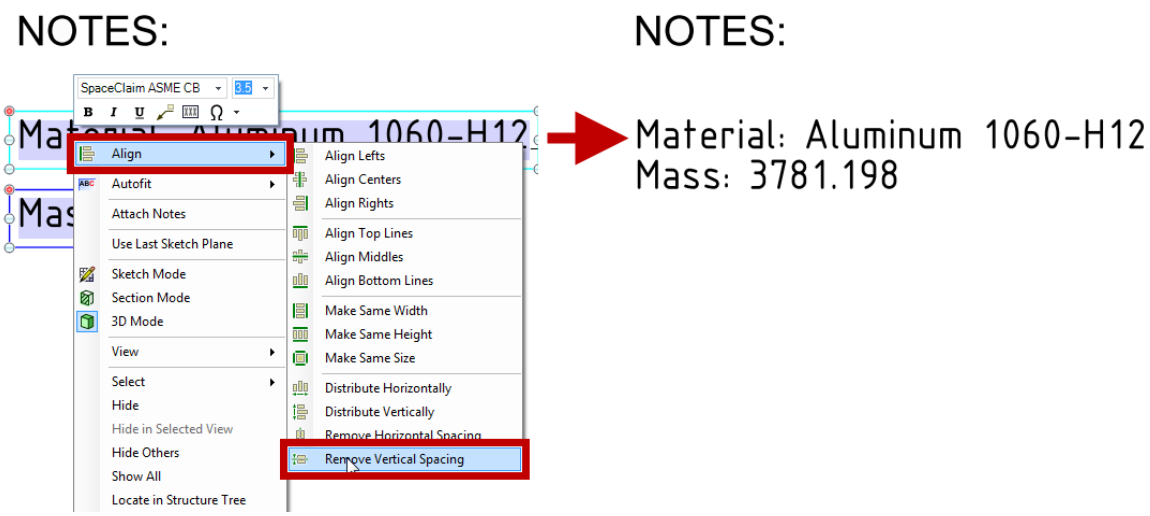


NOTICE how the Material name automatically updates.

77. CTRL select the 2 notes, Right Click and from the Align menu select Align Lefts



78. Right Click again on both notes, and in the Align menu select Remove Vertical Spacing.



Some note require a leader, and some dimension which automatically get 1 leader need an additional leader added.

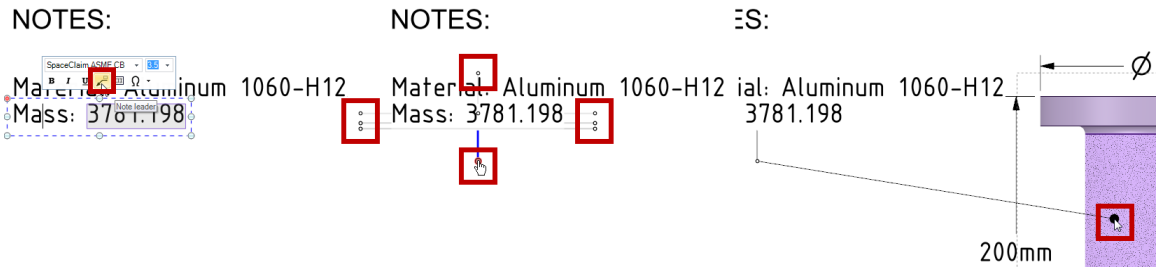
79. To add a leader,

    a. Right or Left Click on the note,

    b. Click the Leader button-

    c. Select one of the attachment points for the leader

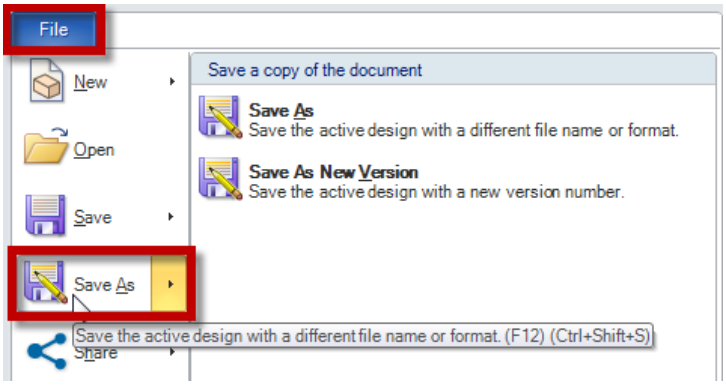
    d. Click on the model in the view to lock to a face with a dot, or an edge with an arrow

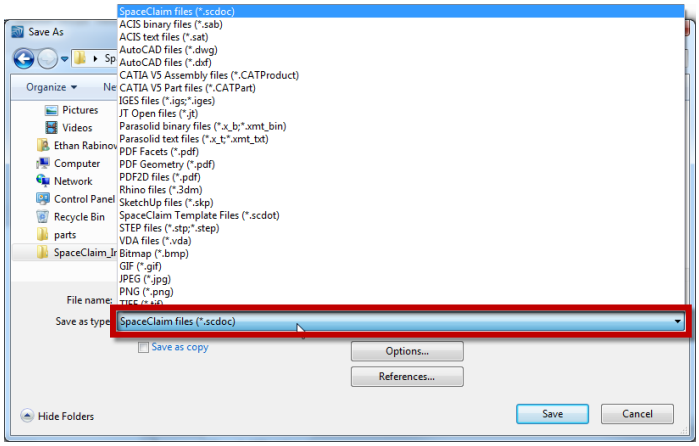


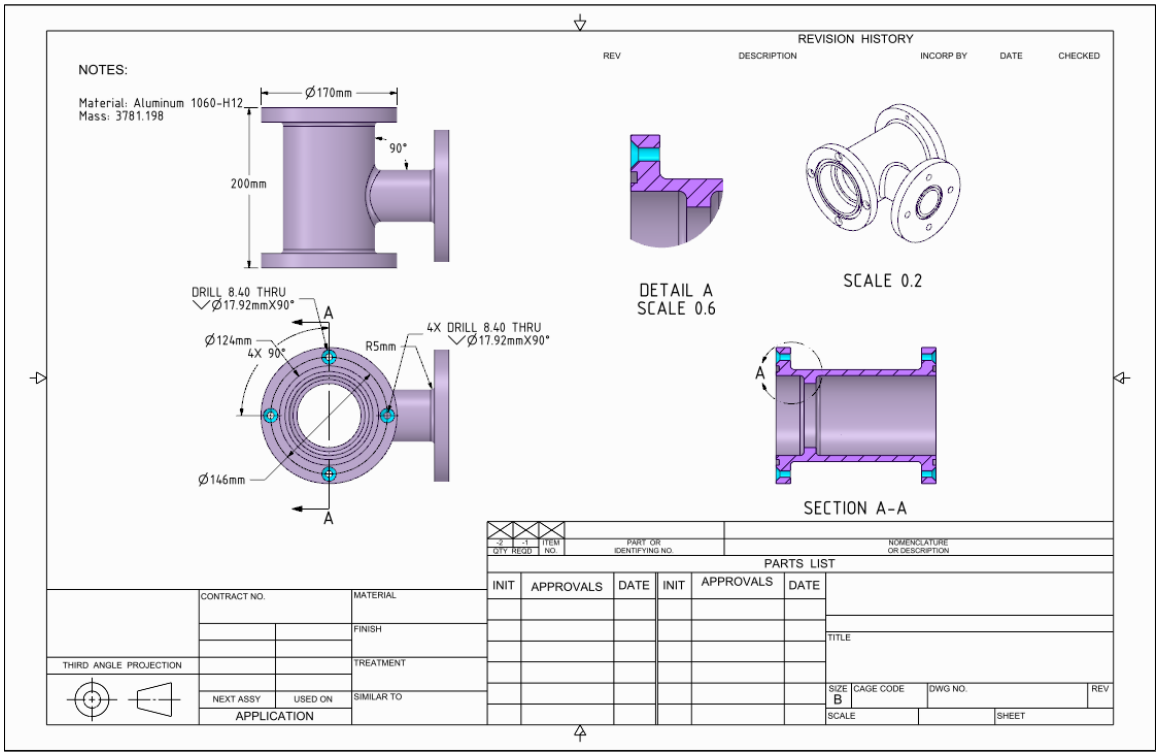
Exporting

Saving either the Valve part or Drawing will save both to Valve.scdoc.

80. To Export to another format, Click File\Save as and choose from the list of formats.





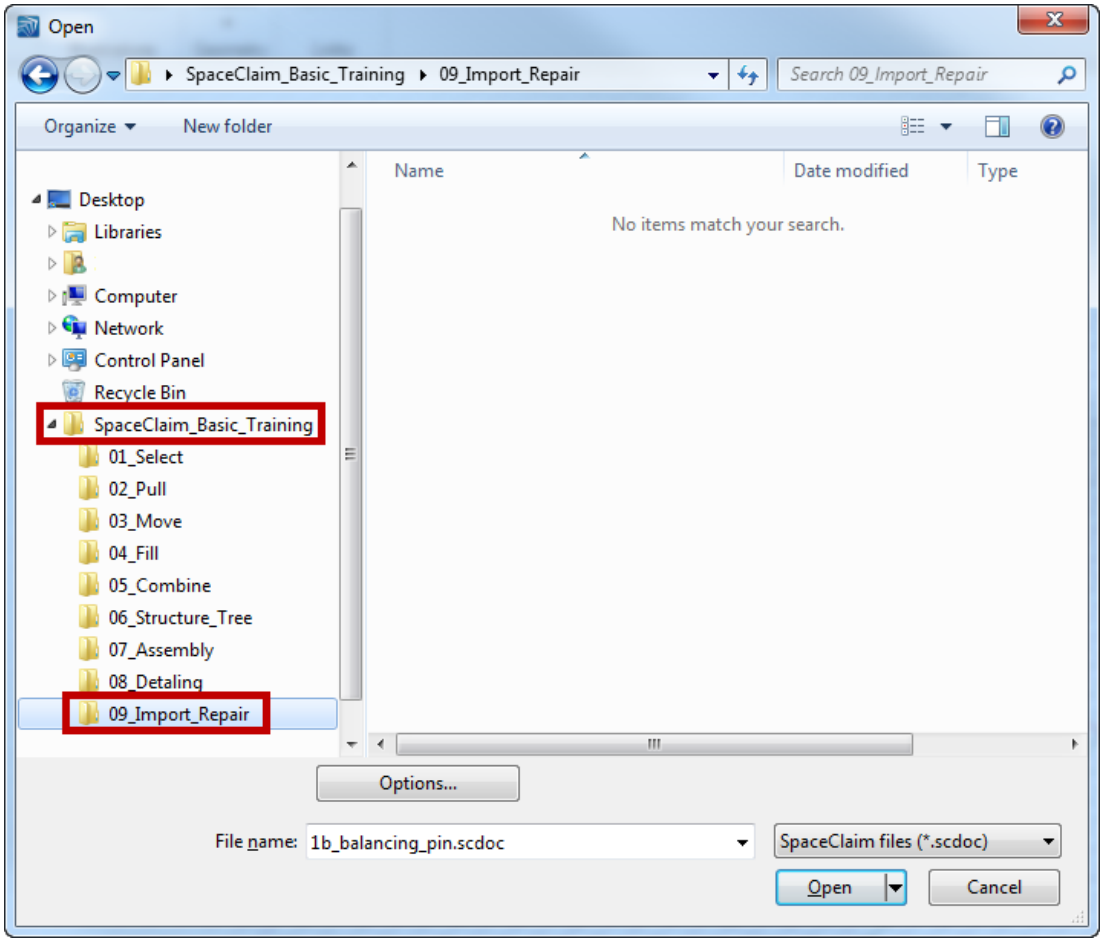

REVISION HISTORY
REV
DESCRIPTION
INCORP BY
DATE
CHECKED
NOTES:
Material: Aluminum 1060-H12
Mass: 3781.198
Ø170mm
90°
200mm
DRILL 8.40 THRU
Ø17.92mmX90°
A
Ø124mm
4X 90°
R5mm
4X DRILL 8.40 THRU
Ø17.92mmX90°
Ø146mm
A
DETAIL A
SCALE 0.6
SCALE 0.2
A
SECTION A-A
QTY REQD
ITEM NO.
PART OR IDENTIFYING NO.
NOMENCLATURE OR DESCRIPTION
PARTS LIST
INIT
APPROVALS
DATE
INIT
APPROVALS
DATE
CONTRACT NO.
MATERIAL
FINISH
TREATMENT
THIRD ANGLE PROJECTION
NEXT ASSY
USED ON
APPLICATION
SIMILAR TO
TITLE
SIZE
B
CAGE CODE
DWG NO.
REV
SCALE
SHEET

Import and Repair

SpaceClaim can Import a large list of Native and Generic CAD file types. Most imported parts can be immediately edited as if they were modeled directly in SpaceClaim from scratch. Some models have geometry issues when imported, which SpaceClaim's repair tools are able to fix so they can be edited. In this section you will import in different file types, edit and repair them.

IMPORTANT: There are many tools in the Repair Tab, and each serves a very specific purpose. It is NOT recommended to import a CAD file and use every tool in the Repair Tab without understanding what issues the model has, and what each tool does to fix them.

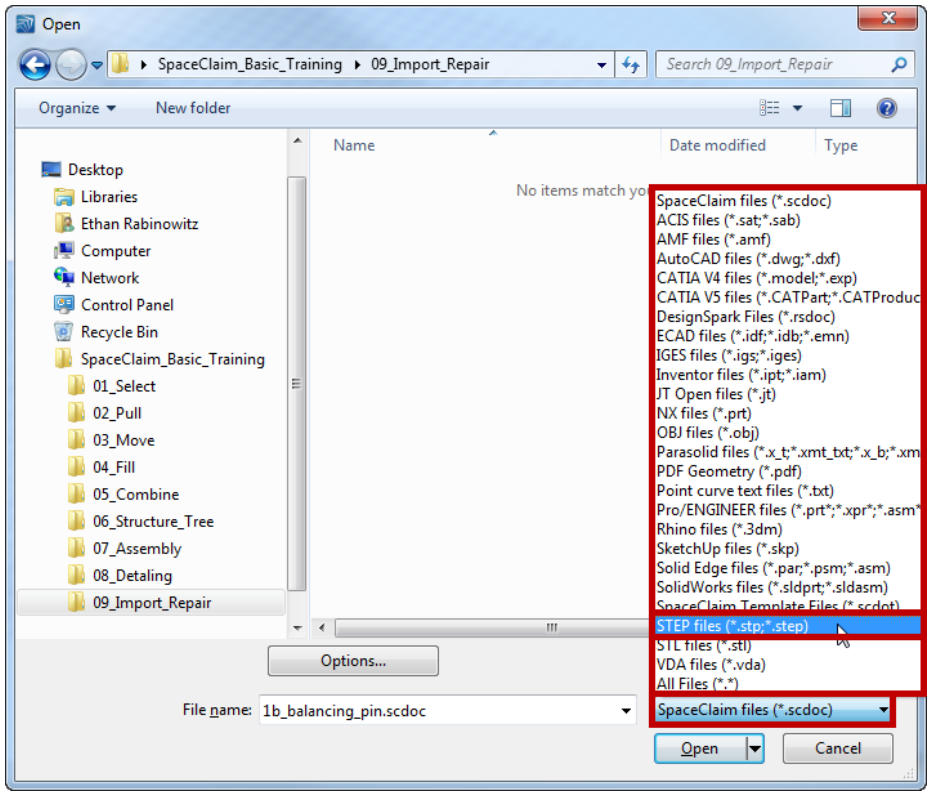
1. **File\Open** Desktop\SpaceClaim_Basic_Training**09_Basic_Import_Repair_2014.0**



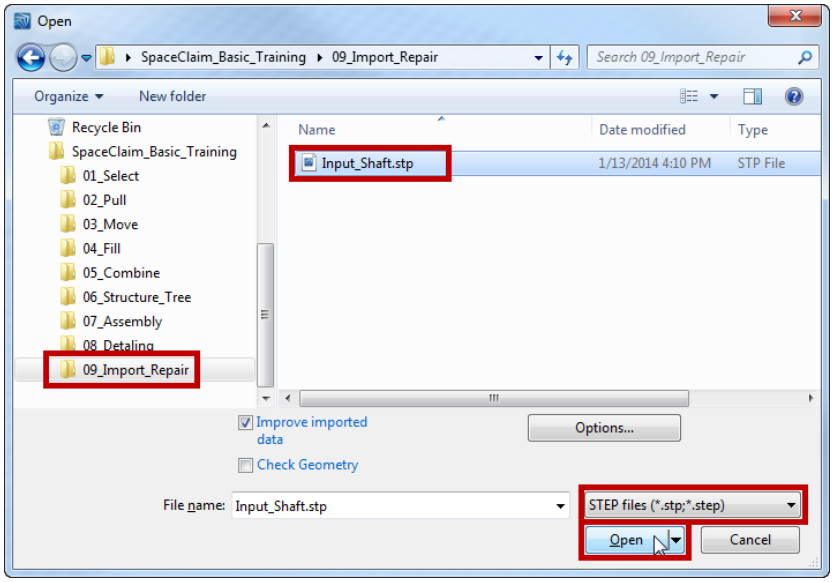
NOTICE that no files are shown. All the files opened so far have been SpaceClaim documents: *.scdoc.

2. Click the File Type Drop down menu in the bottom right that says “SpaceClaim Files (*.scdoc)”
3. Select **STEP** from the list

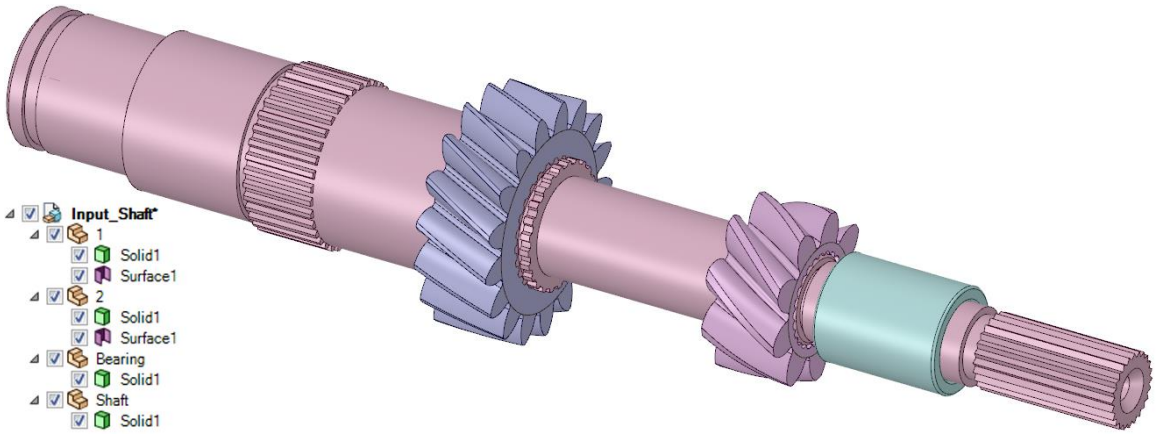
NOTICE all the different file types that SpaceClaim can open. Your may not be able to open all these file types due to which translator bundles you have purchased. For a full list, go to http://files.spaceclaim.com/SupportedFormats.pdf



4. Open Input_Shaft.stp



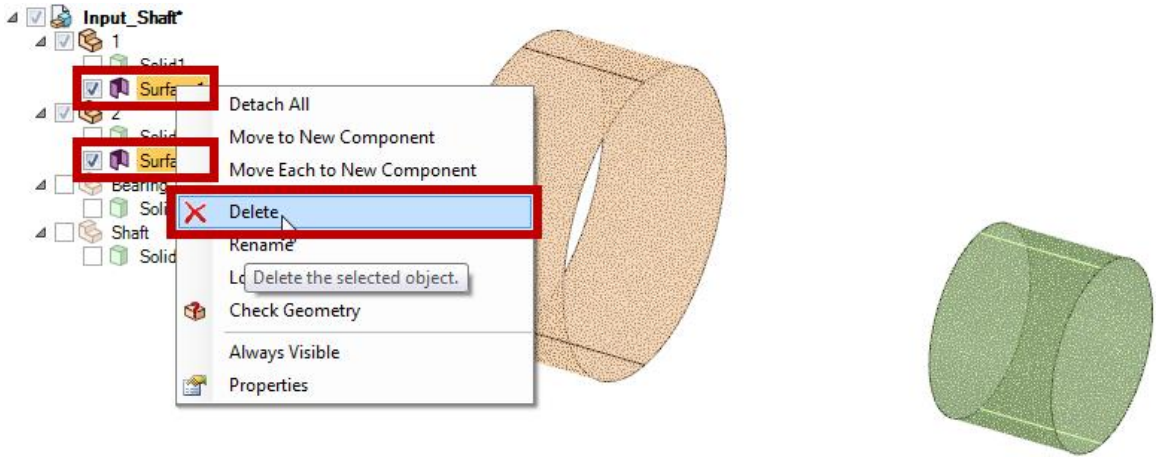
Editing an Imported File



NOTICE there are 4 components in the Structure Tree, each with a solid and 2 with a surface. Some models may come in with extra or duplicate geometry that is not needed. I.e. the surfaces.

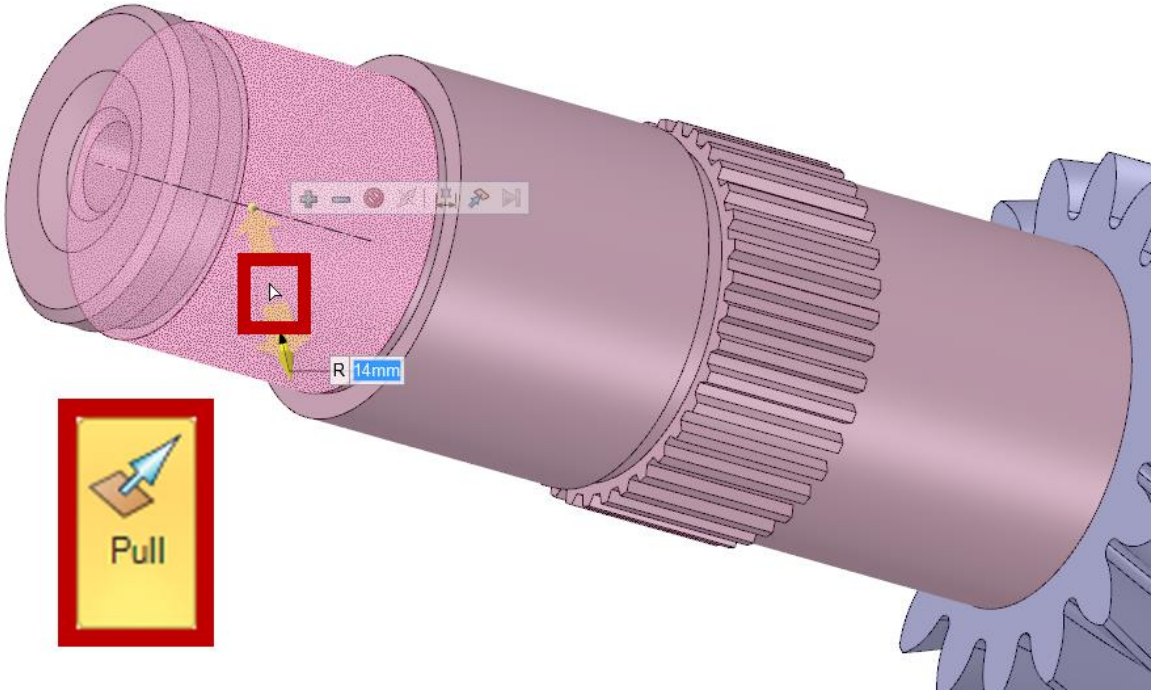
5. Hide the solids to see the surfaces. These surfaces defined a connection point in another CAD program but are not needed in SC.

6. Delete the Surfaces

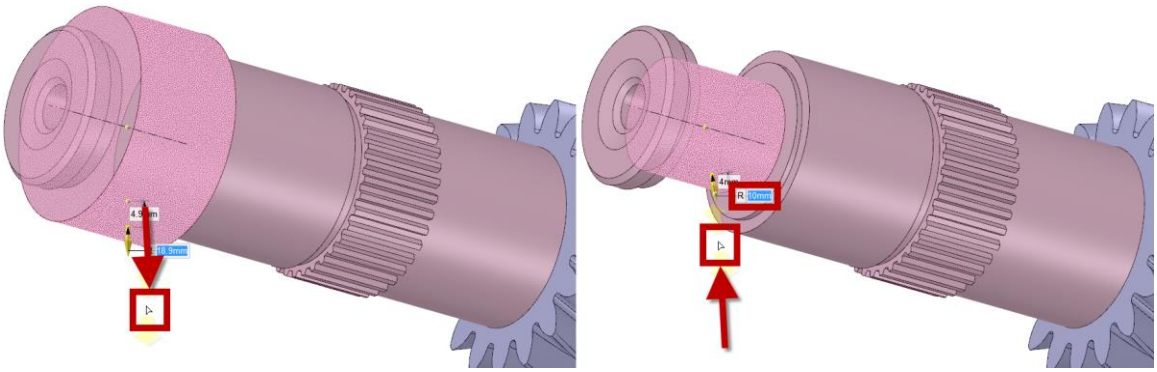


IMPORTANT: Any Imported part that is Solid can be edited using all the tools you've learned without doing any repair

7. Show All the Solids
8. Spin, Pan and Zoom this imported model just like a SpaceClaim file
9. Turn on Pull and Select the Face below

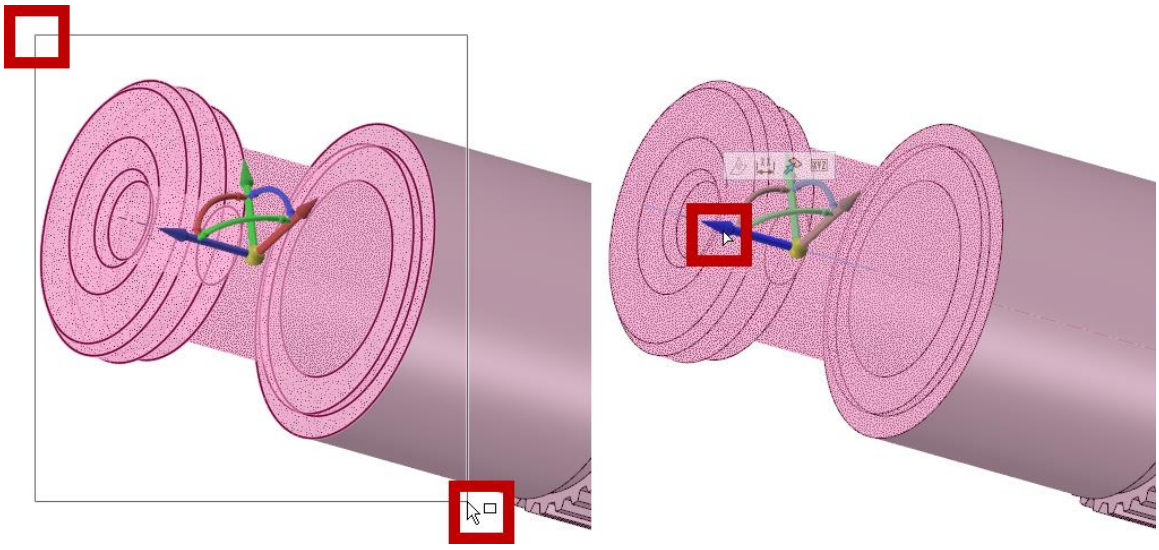


10. Pull out and in, to a radius of 10mm

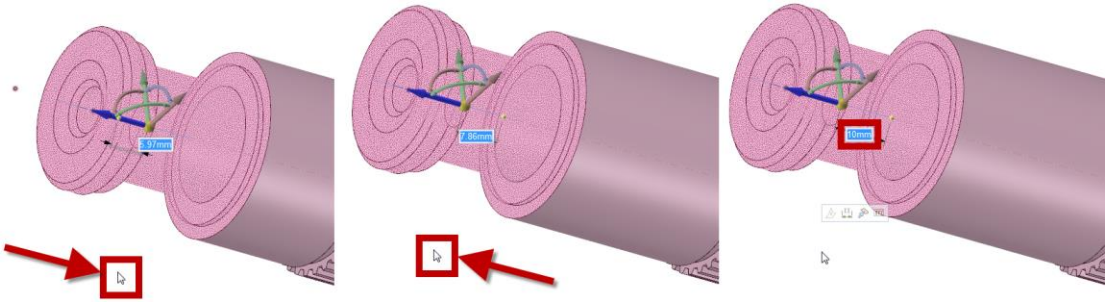


NOTICE that this Imported Solid can be directly edited without an extra steps. Even if the surfaces weren't deleted, you could still pull the faces of the solid

11. Turn on the Move tool and Box Select the Face below from left to right
12. Click the Straight Move Arrow along the axis, in blue below

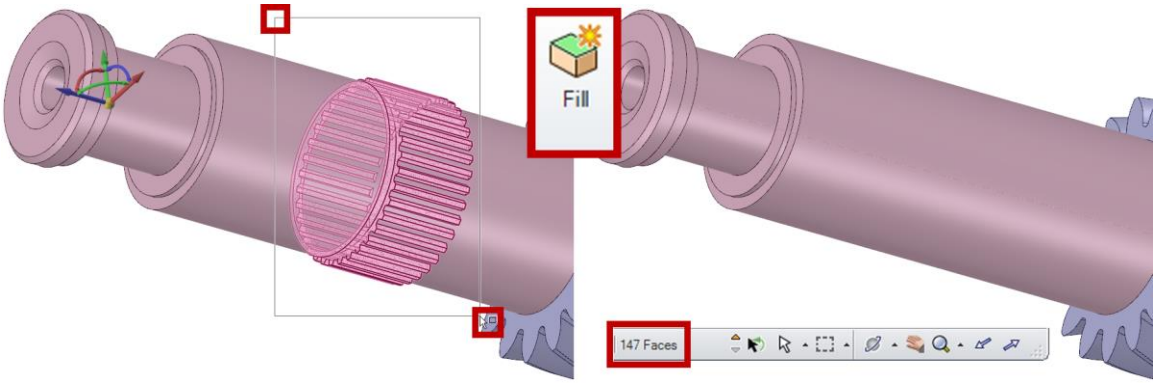


13. Drag to right, to the left, or just enter a value like 10

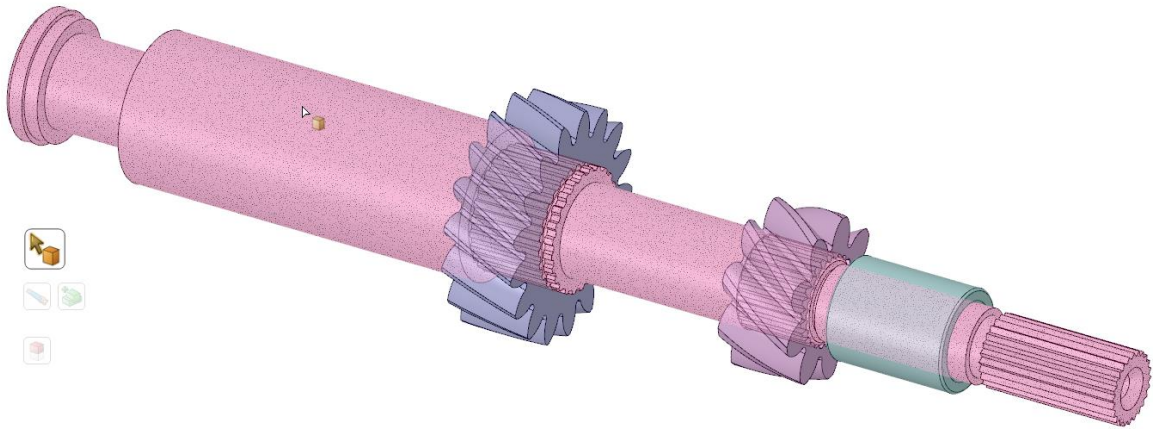


NOTICE that moving these faces is just like in the move section with the .scdoc.

14. Box Select the Teeth on the shaft. There should be 147 Faces selected as shown in the bottom right. Remember to pause during box select to see a preview of what will be selected.

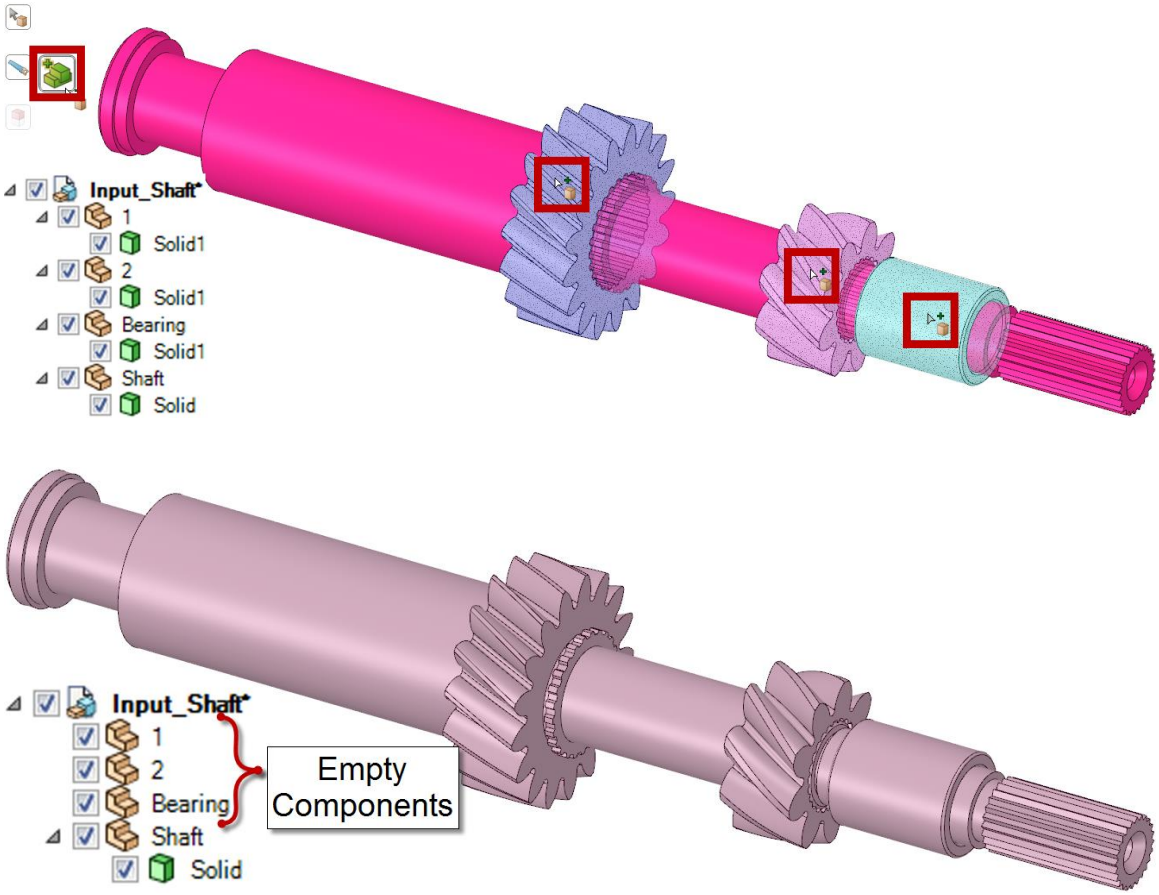


15. Return to a Home View (H)
16. Turn on the Combine tool and select the Shaft as the Target Object (the new object will be the same color as the Shaft, and the new Solid will be in the Shaft Component



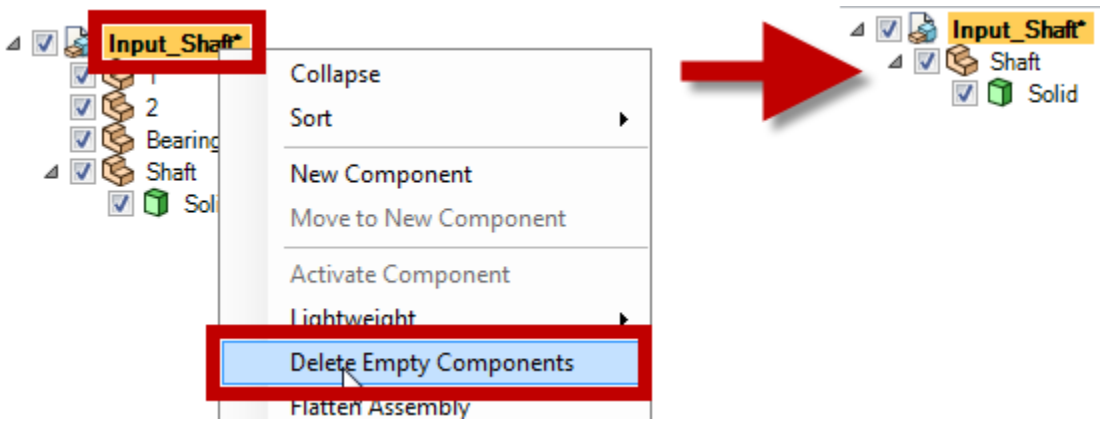
NOTICE that the Combine tool switches to the cutter ToolGuides, although there are no intersecting bodies to cut with

17. Click the Select Bodies to Merge ToolGuides and select the 3 other solids



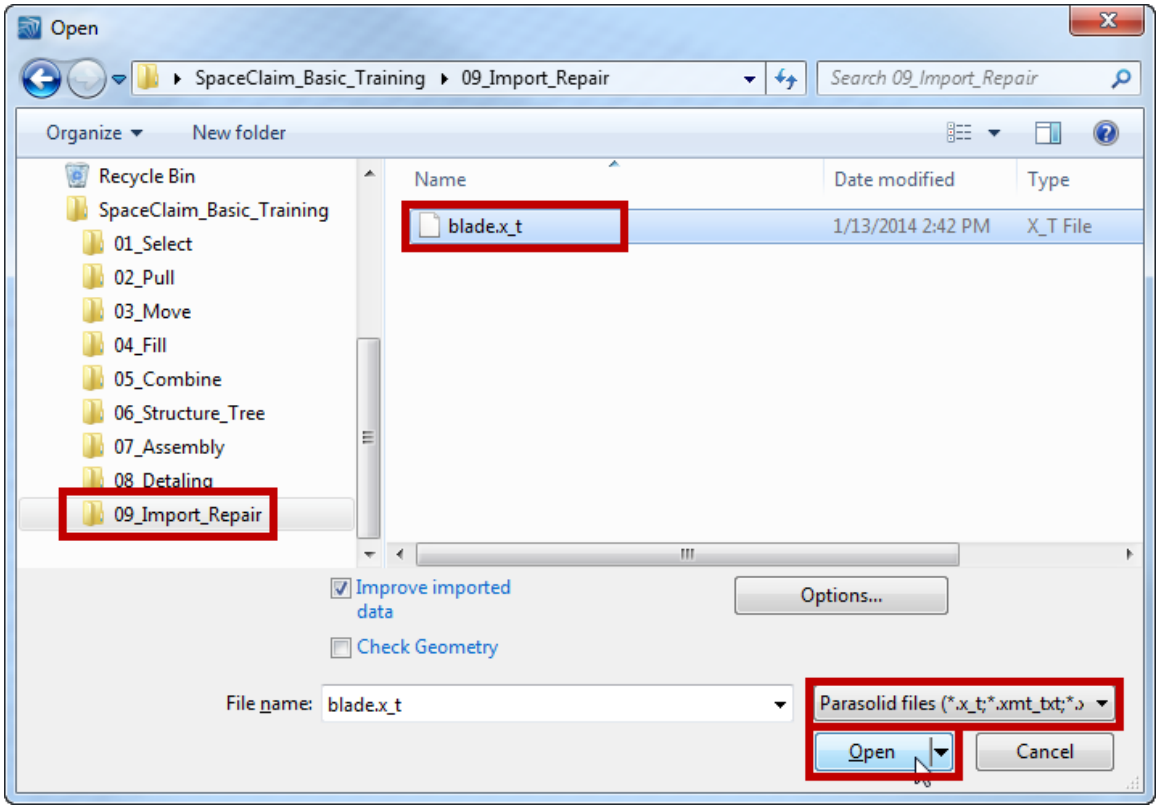
NOTICE that you've gone from 4 Components with 1 Solid each, to 1 Component with the newly merged solid, and 3 Empty Components. One last piece of housekeeping is to delete the empty Components

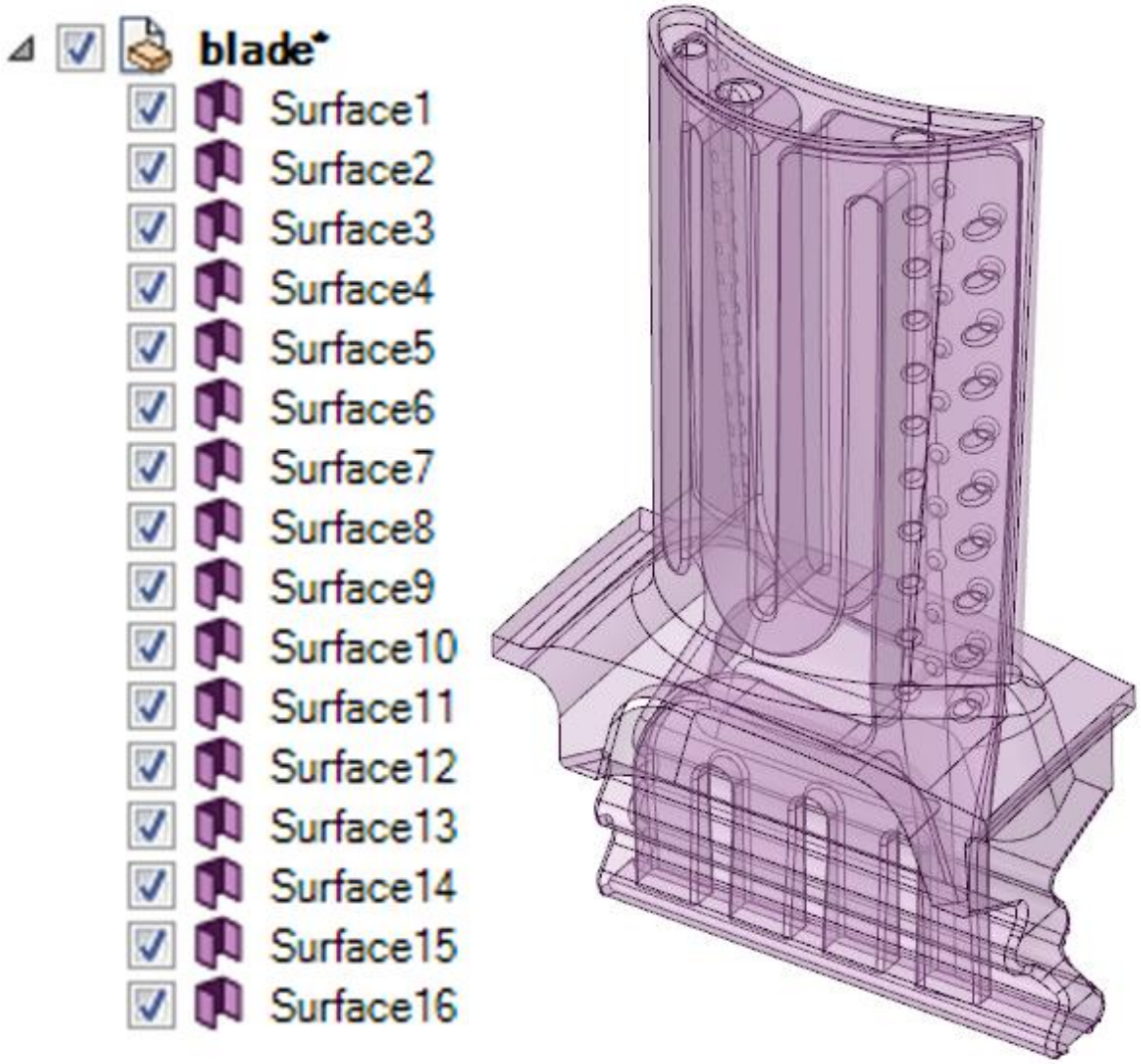
18. Right Click on the Input_Shaft in the Structure Tree and select Delete Empty Components



Repairing Imported Files

19. Go to File\Open and change the File Type to Parasolid Files and Open blade.x_t.





NOTICE

a. that there are 16 Surface Bodies in the Structure Tree

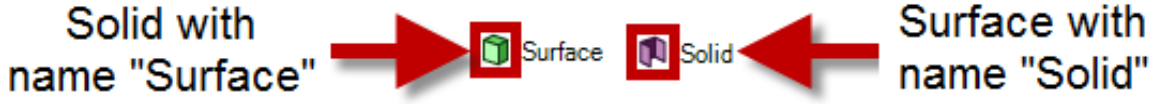
b. that the model in the Design Window is Transparent

IMPORTANT

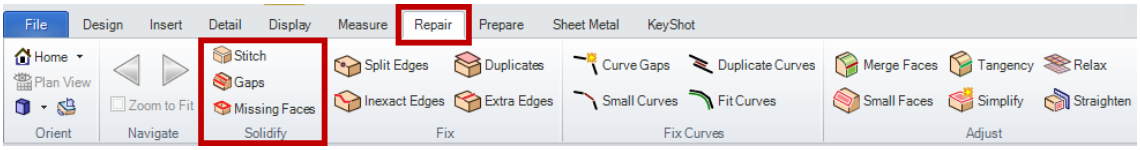
a. There are 2 Red Flags that the Imported Model is Damaged and needs to be Prepared. The majority of imported models are supposed to be Solids.

b. Files can get damaged when being converted out of the Original CAD system to another file type, such as STEP or IGES. Even Native files direct from other CAD systems could be damaged due to bad modeling techniques or bugs in the CAD system.

c. When identifying a Solid or Surface, DO NOT TRUST THE NAME. The Icon will always tell you with 100% accuracy if the body is a solid or surface



20. When you Import a file that should be a Solid but it's 1 or more surfaces in the Structure Tree and the model is Transparent and you can see through it, you should **go to the Repair Tab** to fix it



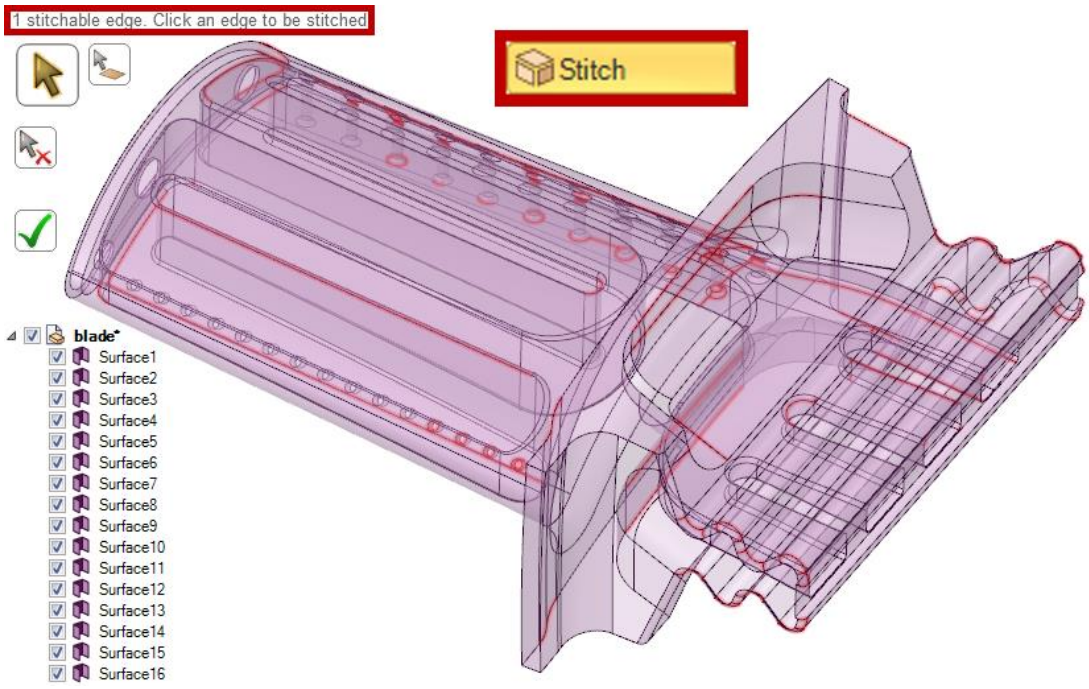
NOTICE there are many tools and groups in the Repair Tab. You will be focusing on the Solidify Group for this section.

Stitching Surfaces

A Solid in SpaceClaim is a Single, Water-Tight Body.

The first step in repairing a damaged part is to make sure you have only 1 surface body per part. If an Assembly is damaged, it may be OK to have more than 1 surface body.

21. Turn on **Stitch**. Since this file is 1 part, we want 1 surface in the Structure Tree

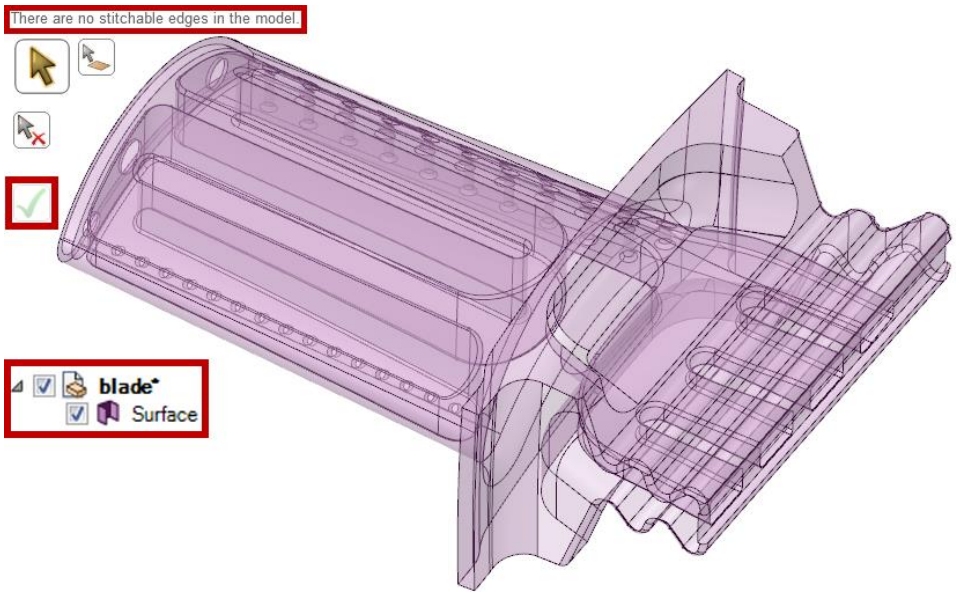


NOTICE the Edges highlighted in Red. The Stitch tool looks for edges that are very close to each other, but are not officially joined.

Imagine a Jacket where some of the stitching has come out. The edges of the pieces of fabric that were sewn together are still touching each other, but they are not actually joined together. The Stitch tool puts back the thread/stitching or connection between the edges of multiple surfaces that are touching.

NOTICE the status message says 1 Stitchable Edge, but it looks like there are many edges. The message really means there is 1 web of edges that after fixing will result in 1 surface body

22. Click the **Check Mark** or click an edge to Stich as The Status Message says. Both do the same because there is only 1 web of edges to stitch.

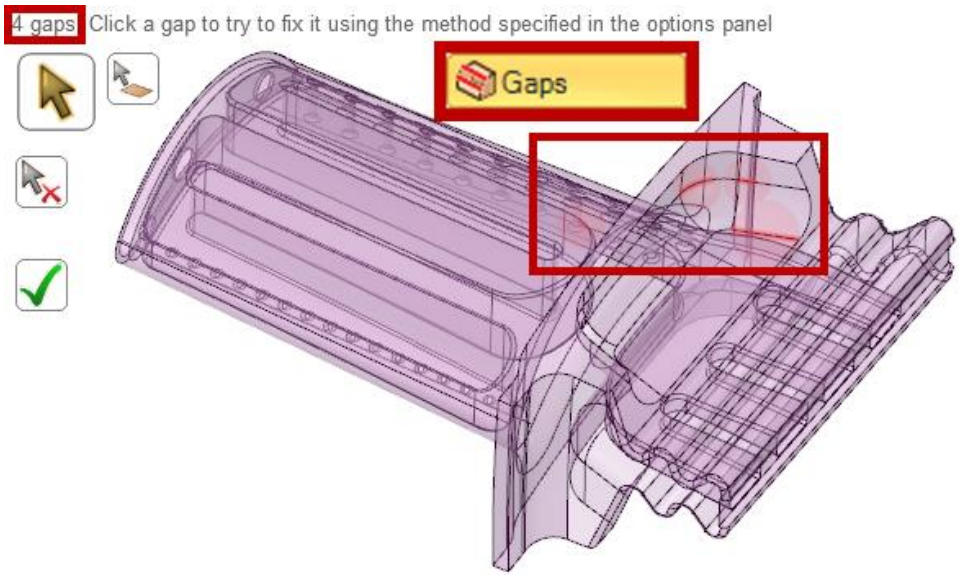


NOTICE that the model doesn't look any different besides no red edges, but the Structure Tree shows just 1 surface body. With many models, all that needs to be done is Stitch to repair the file into a solid.

If after running Stitch, there is a solid, you are finished with the repair tools. If there is 1 surface, proceed to the Gaps and Missing Faces Tool

Fixing Gaps

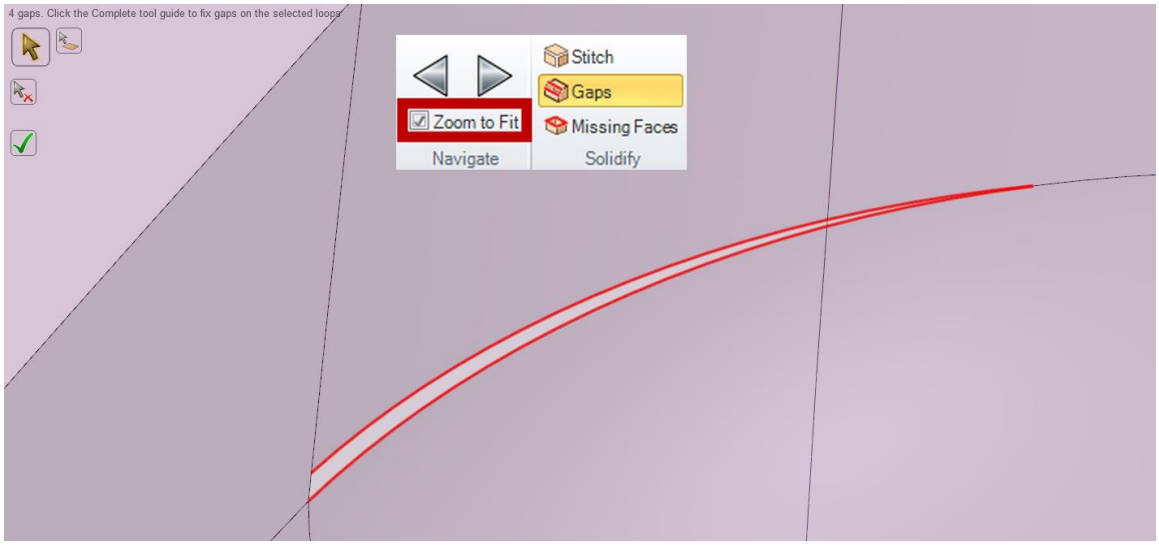
23. Turn on the **Gaps** Tool



NOTICE the Gaps tool found 4 gaps and highlighted them on the model

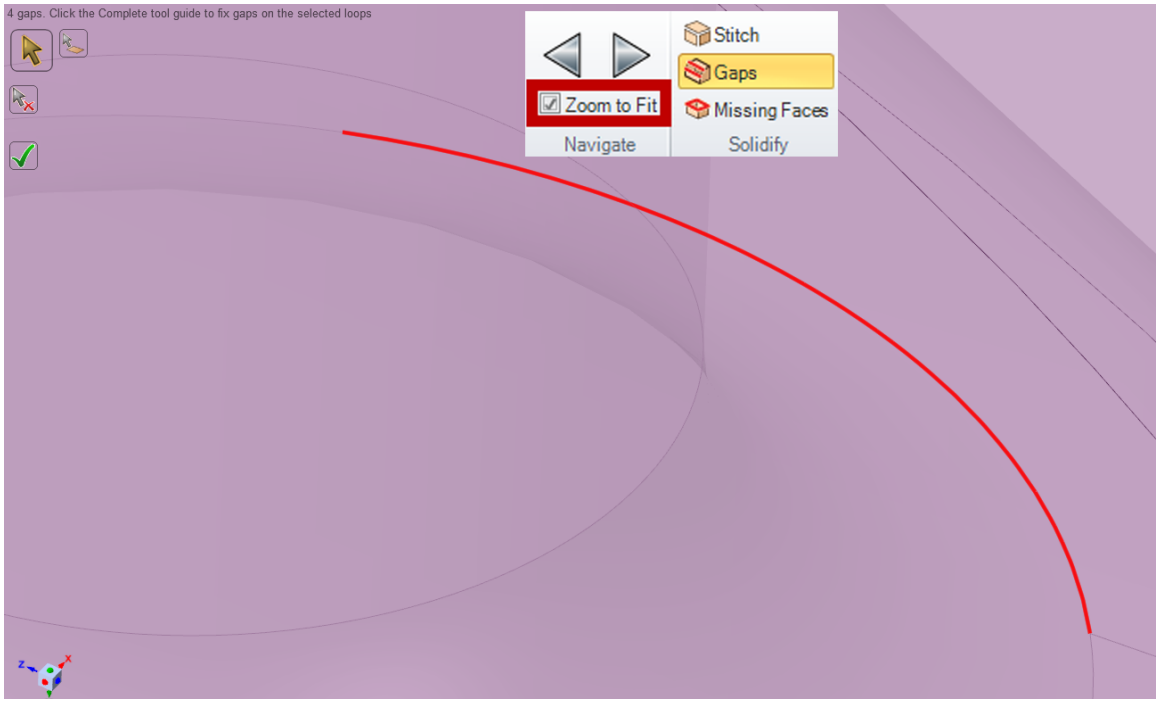
It is recommended to zoom in and take a look at the Gaps the Gaps tool Finds, either manually, or

24. Click the Zoom to Fit check box to the left of the Gaps Tool



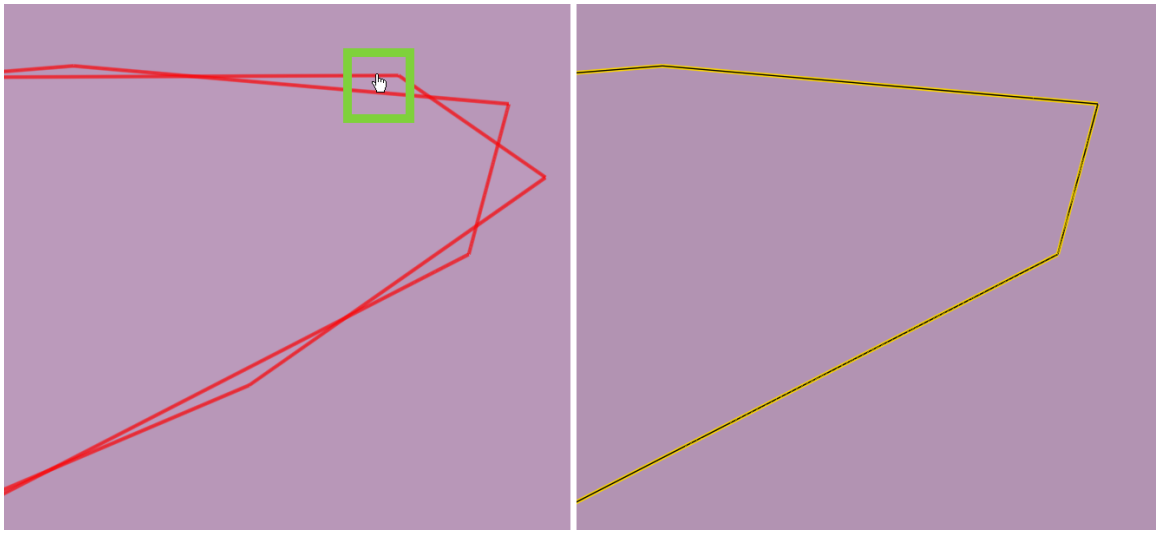
NOTICE how SpaceClaim Zooms in right on one of the Gaps. This Gap is very obviously showing 2 edges where there should be 1 edge.

25. Click the Right Arrow above Zoom to Fit



NOTICE that Zoom to Fit Zooms and Pans but does not Spin. You may have to manually Spin the Model to see the Gap more clearly.

26. Zoom very far in on this Gap
27. Click Directly on the Red Edges (may have to move mouse around till cursor shows a hand)

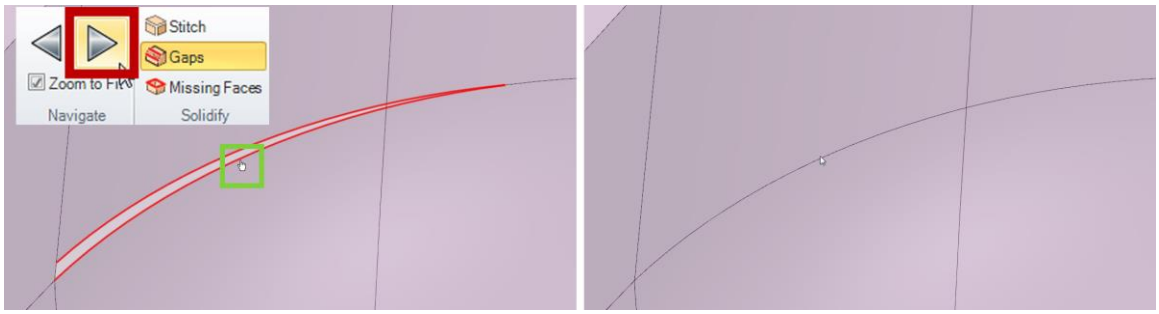


NOTICE after clicking the Double Red Edge, it turns into 1 edge (2nd image has Stitch off and the new edge selected to show it better)

The Gaps Tool works by adjusting the Edges to meet using the contours of the surface as a guide.

28. Click the Right Arrow again above Zoom to Fit to advance to the next Gap
29. Click on the Red Gap to Fix it

30. Click the Right Arrow again above Zoom to Fit to advance to the next Gap
31. Click on the Red Gap to Fix it



32. Click the Right Arrow again above Zoom to Fit to advance to the next Gap
33. Click on the Red Gap to Fix it

NOTICE after fixing this last Gap, the Status Message says there are no more gaps using the current search criteria

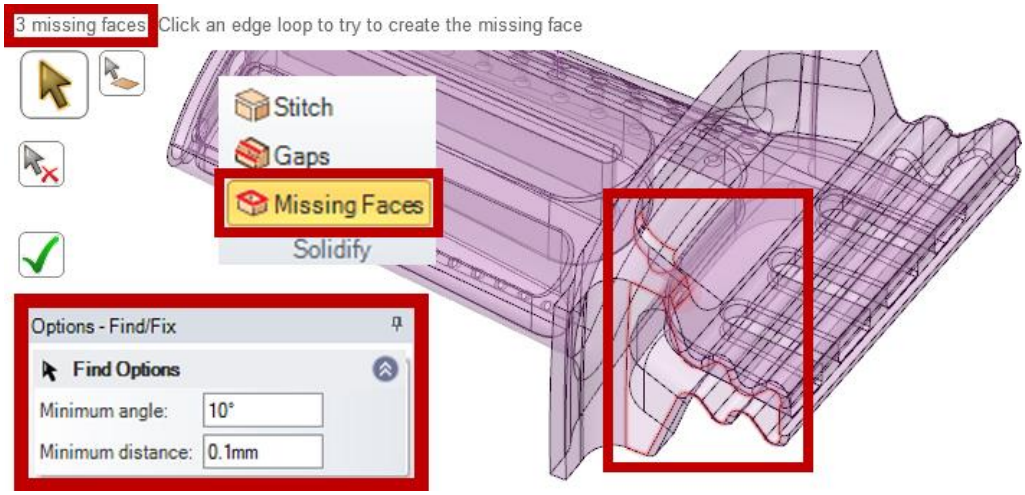
NOTICE in the Options Panel on the left that the Gaps tool has 2 **Minimum** Search Parameters

- **Maximum Angle:** 10 degrees
- **Maximum Distance:** 0.1mm

The Gaps tool looks for any openings in a Single Body under 10 degrees and 0.1mm

Missing Faces

34. Turn on the **Missing Faces** tool



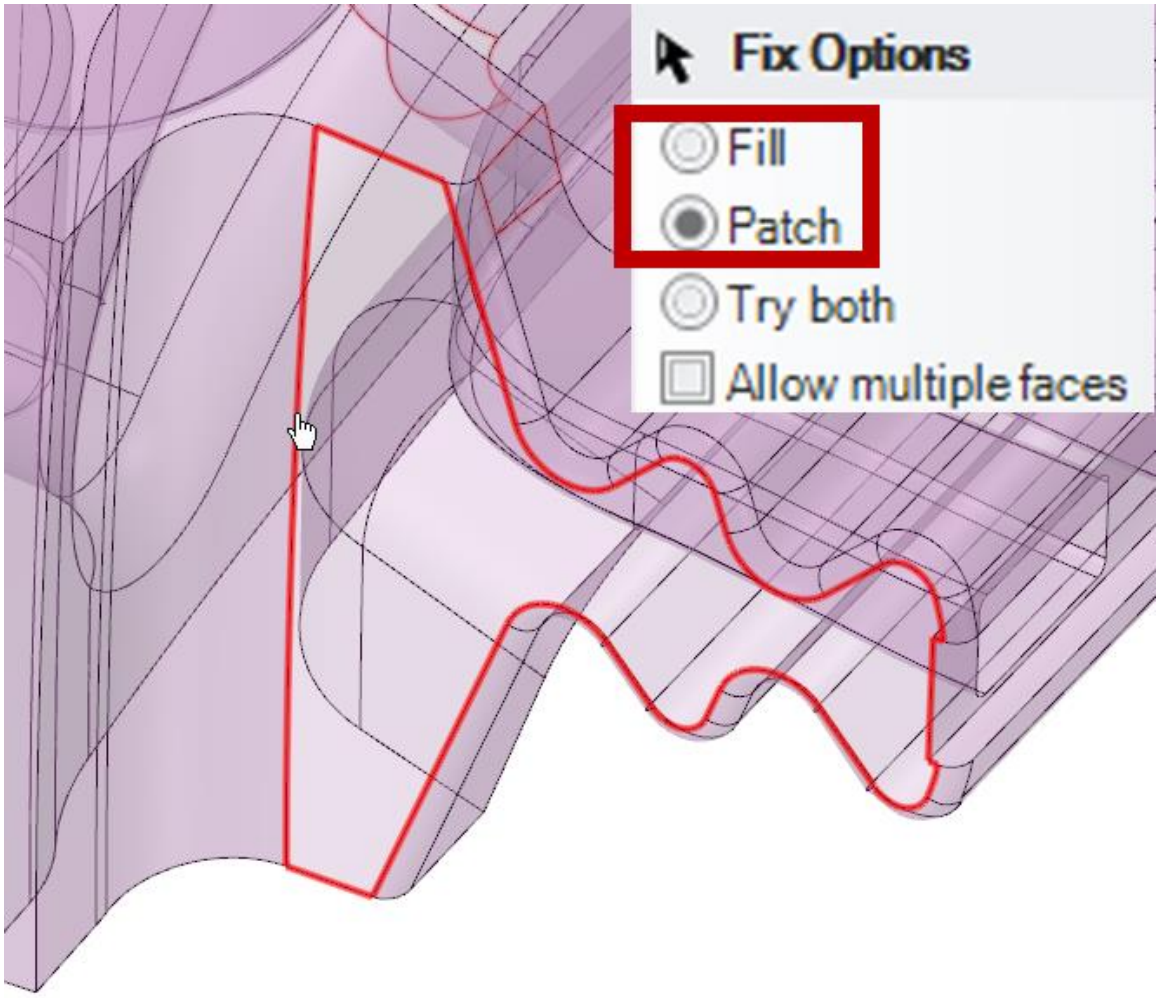
NOTICE that the Options on the Left for Missing Faces has a **Minimum Angle and Distance**.

Minimum Angle: 10 Degrees

Minimum Distance: 0.1mm

Notice the Values are the same for Gaps and Missing Faces. While the **Gaps tool** finds any opening in a single Surface **smaller than 10 degrees and 01.mm, the Missing Faces Tool** finds any opening **Larger than 10 degrees and 0.1mm** (by default). These values can be changed

Let's take a look at what the Missing Faces tool finds and what Options there are to fix it



The Missing Faces Tool has 2 main fixing methods:

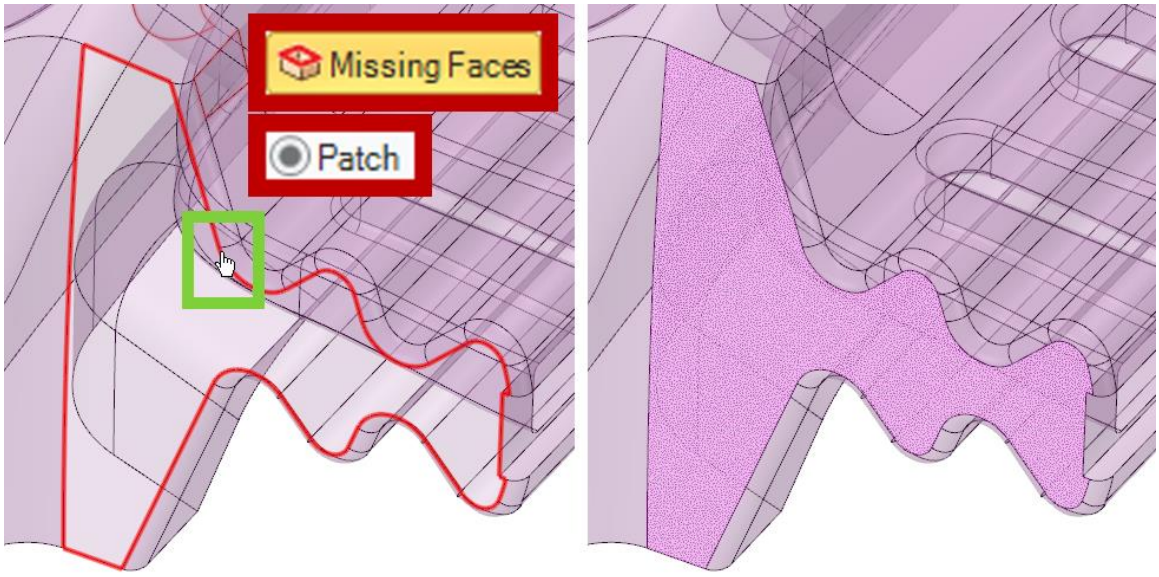
- **Fill:** Tries to extend the surrounding faces to remove the missing face. Works like the Fill Tool
  - Typically used for Smaller Faces or when Part of a Face is Missing
- **Patch**: Creates a Brand New Face within the Edges\Boundaries of the Missing Face
  - Typically used for Larger Faces and when an Entire Face is missing

LOOK at the highlighted face above. Is Part of the Face missing? Or is the Entire Face missing?

The Entire Face is missing, so the Patch Repair option should be used.

Patch is a safer option than Fill, which is why Patch is the default.

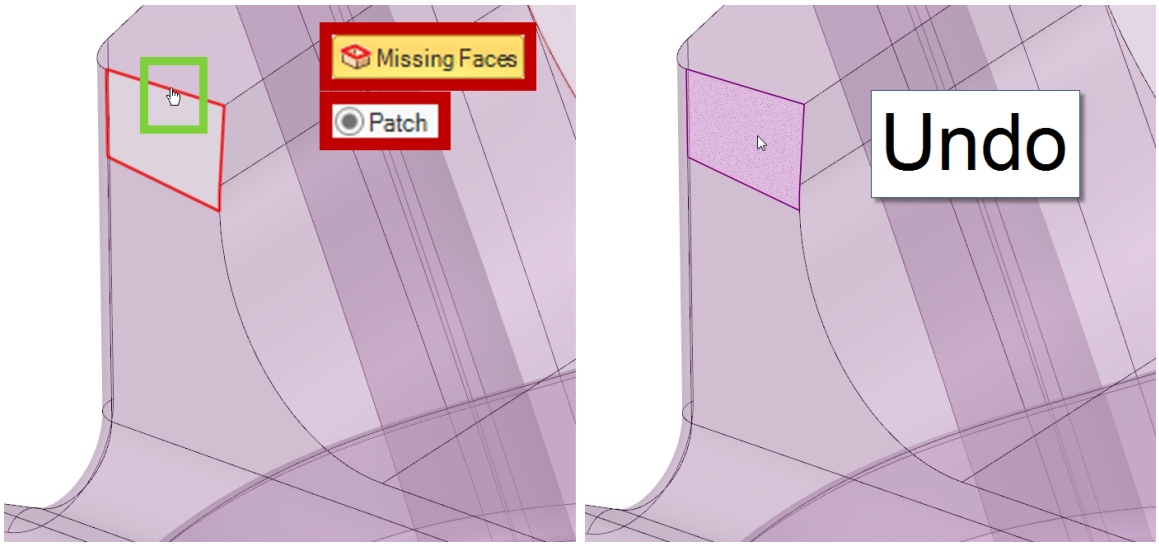
35. Click the Red Edge of the Large Missing Face



NOTICE a brand new Face Patched in within the edges (2nd image is in Select Tool with face selected)

36. With Missing face still on, spin, pan and zoom to the missing face below, or use "zoom to fit"

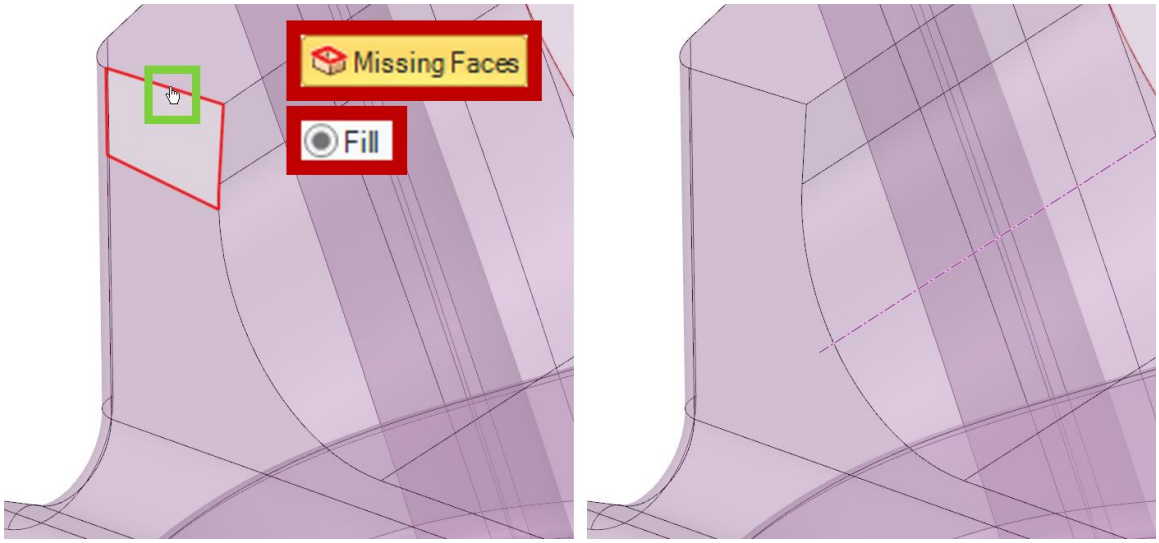
37. Click the Red Edge below with the Patch option still on



NOTICE that it does not look like a new face should have been patched in. It looks like the face below the new Face can be extended to remove the missing face

38. Undo to before the last Patch

39. Click Fill in the Options panel on the left
40. Click the Missing Face a 2nd time to Fill the Missing Face

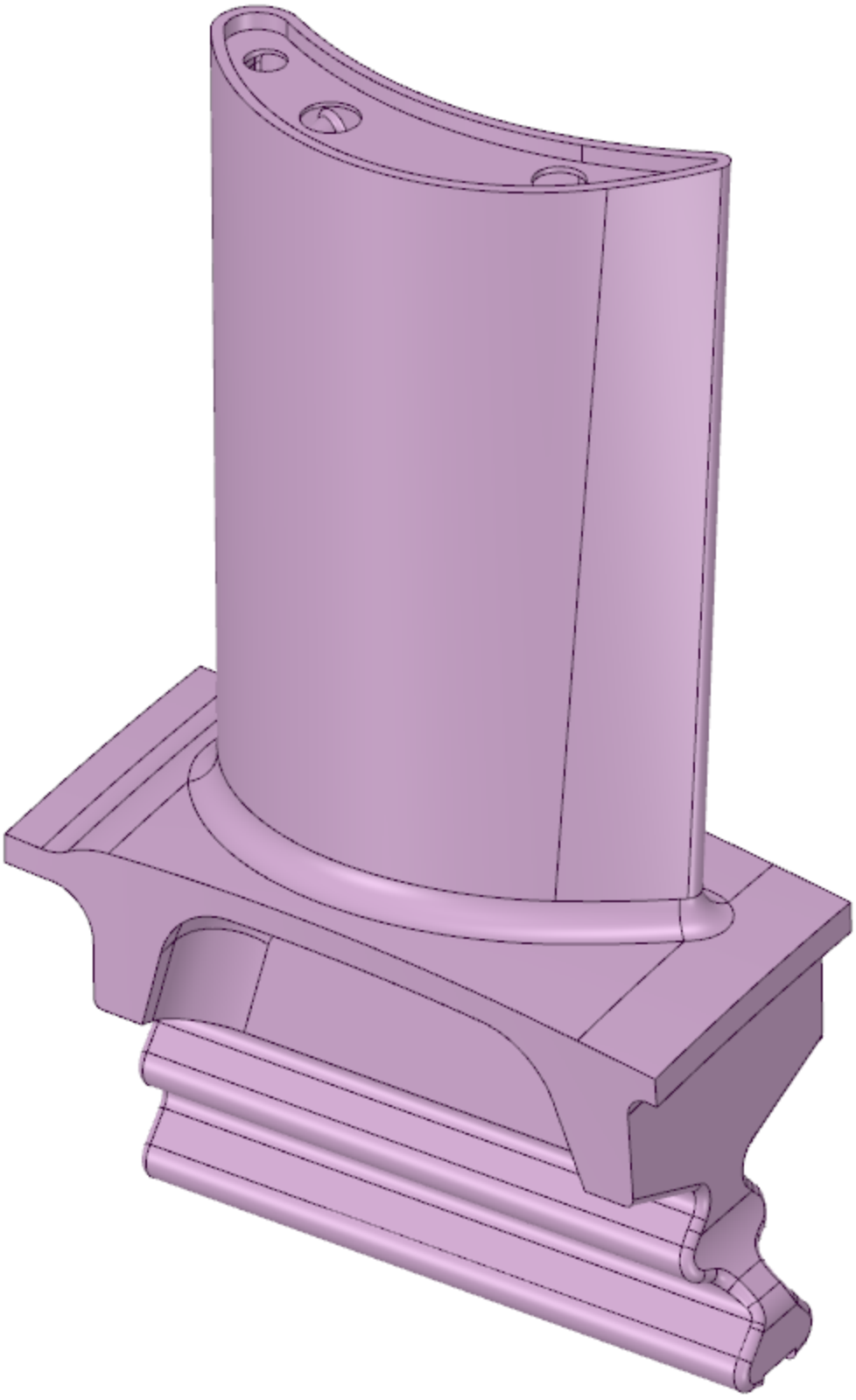


NOTICE how the Missing Faces tool Filled in the Missing Face by extending the neighboring Face

41. The last missing face is missing in its entirety, so turn on the Patch option
42. Click the Last Face

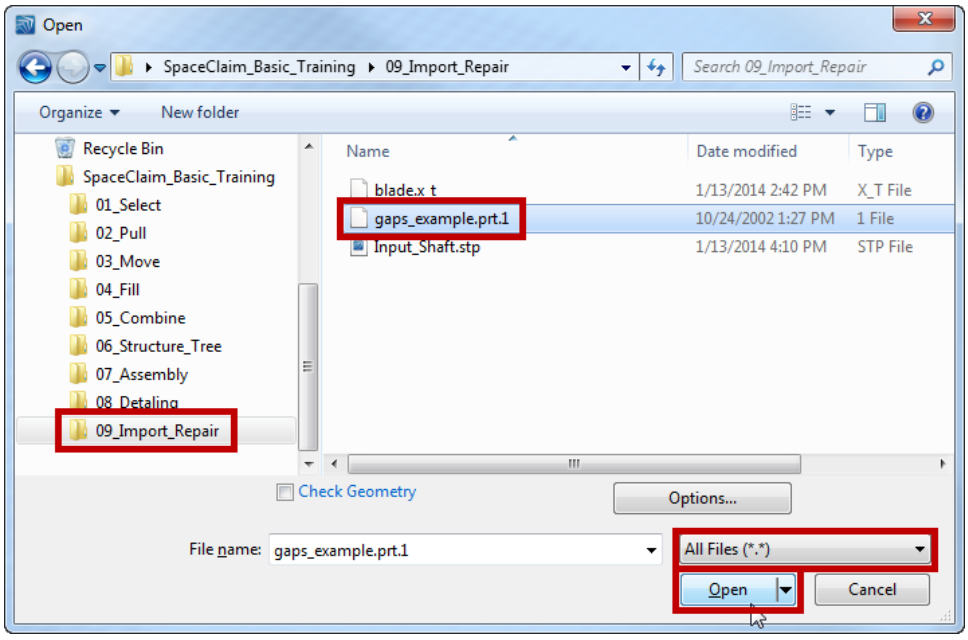
NOTICE that as you patch in the last missing face, the model automatically turns opaque because it is now a solid, as confirmed in the Structure Tree

This part can now be edited as a solid as if created in SpaceClaim

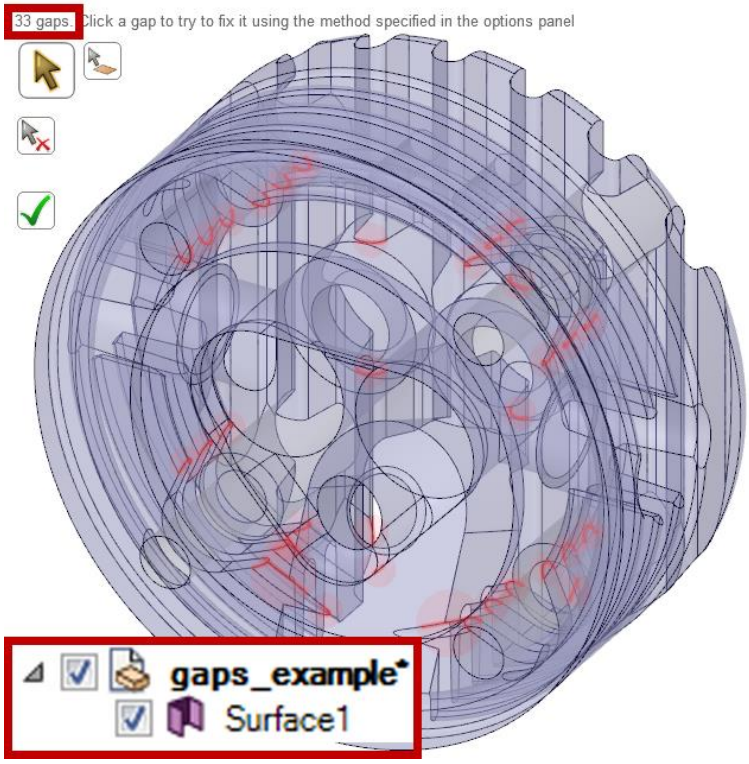


Repairing Many Gaps

43. Go to File Open, select All Files and open gaps_example.prt.1

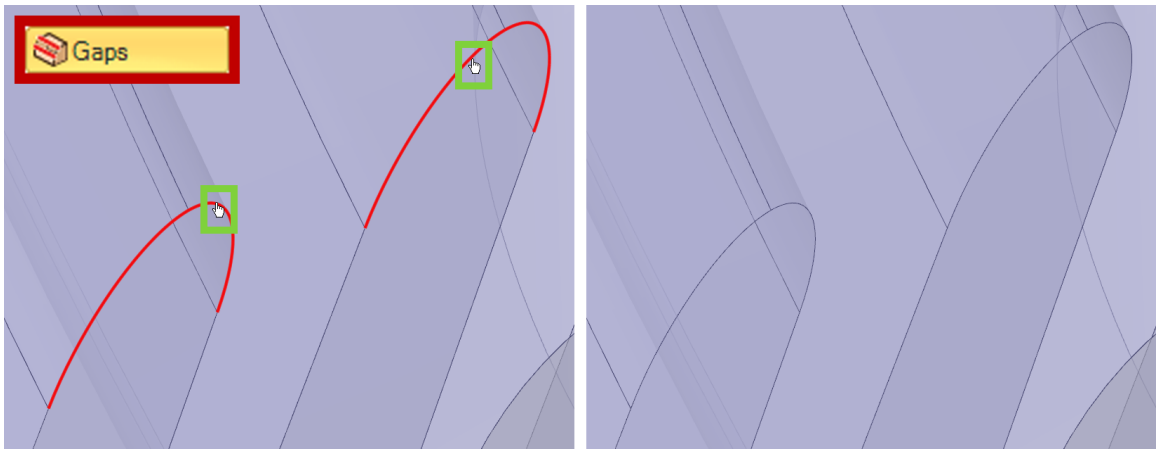


44. This model is 1 Surface, so you can skip Stitch and Turn on Gaps

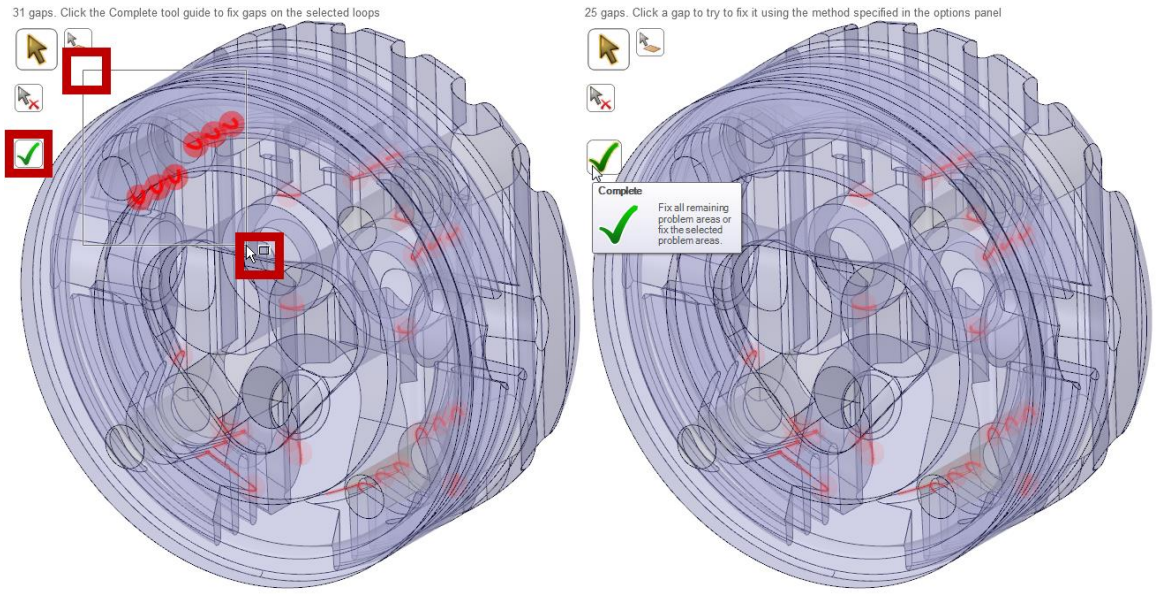


NOTICE that Gaps finds 33 Gaps that look very similar.

45. Zoom in on an area where there are a few Gaps
46. Click on a couple Red Edges to have the Gaps tool fix the Gap.

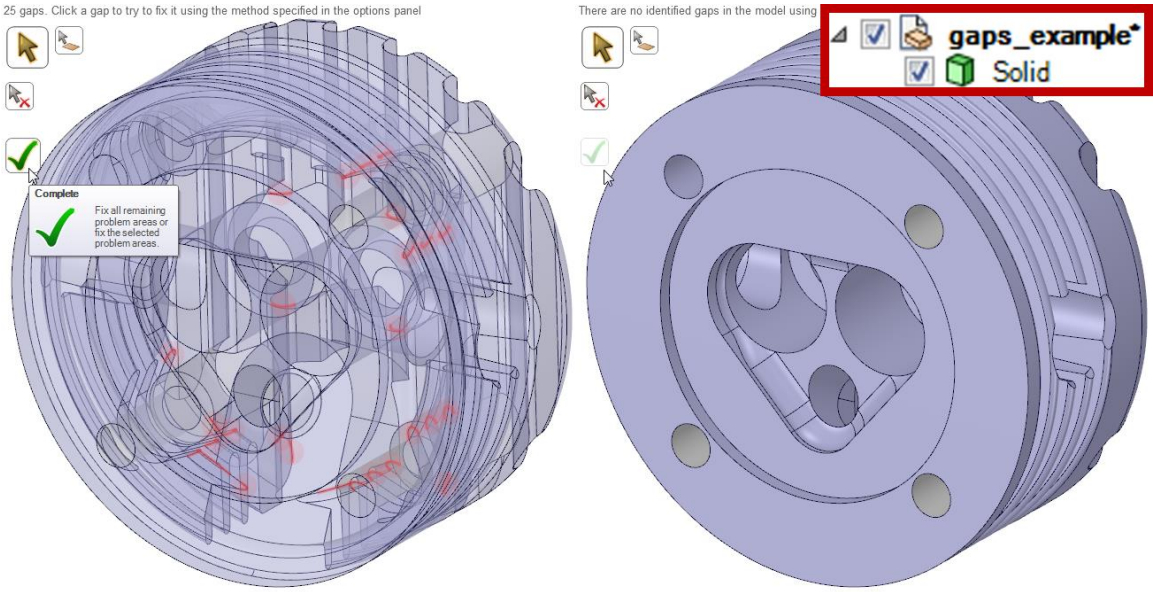


47. If the results look good, zoom out and box select as shown while still in the Gaps tool

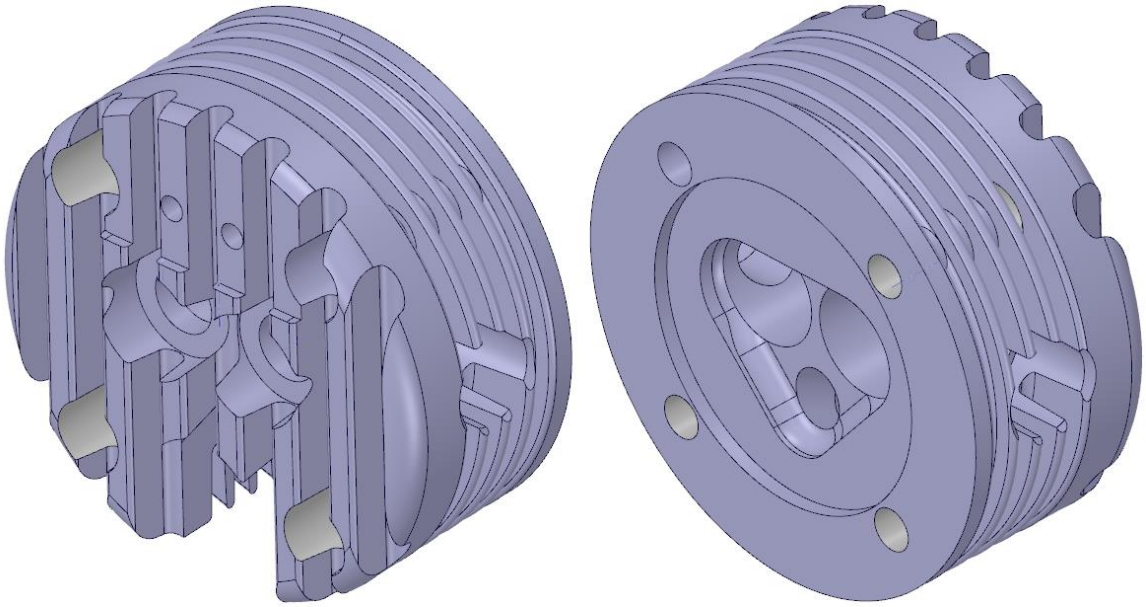


NOTICE the Gaps tool fixed all the gaps that were selected.

48. Click the Checkmark with nothing box select to fix the rest of the Gaps



This Solid can now be edited with all the tools you've learned as if created in SpaceClaim



Next Steps

Practice, Practice, Practice

Practice makes perfect with any new skill or learned software. Repeat the sections that you will use the most, and try out the skills you've learned in the manual on models that you have in your own library or even from online libraries like www.grabcad.com

Additional Live Training

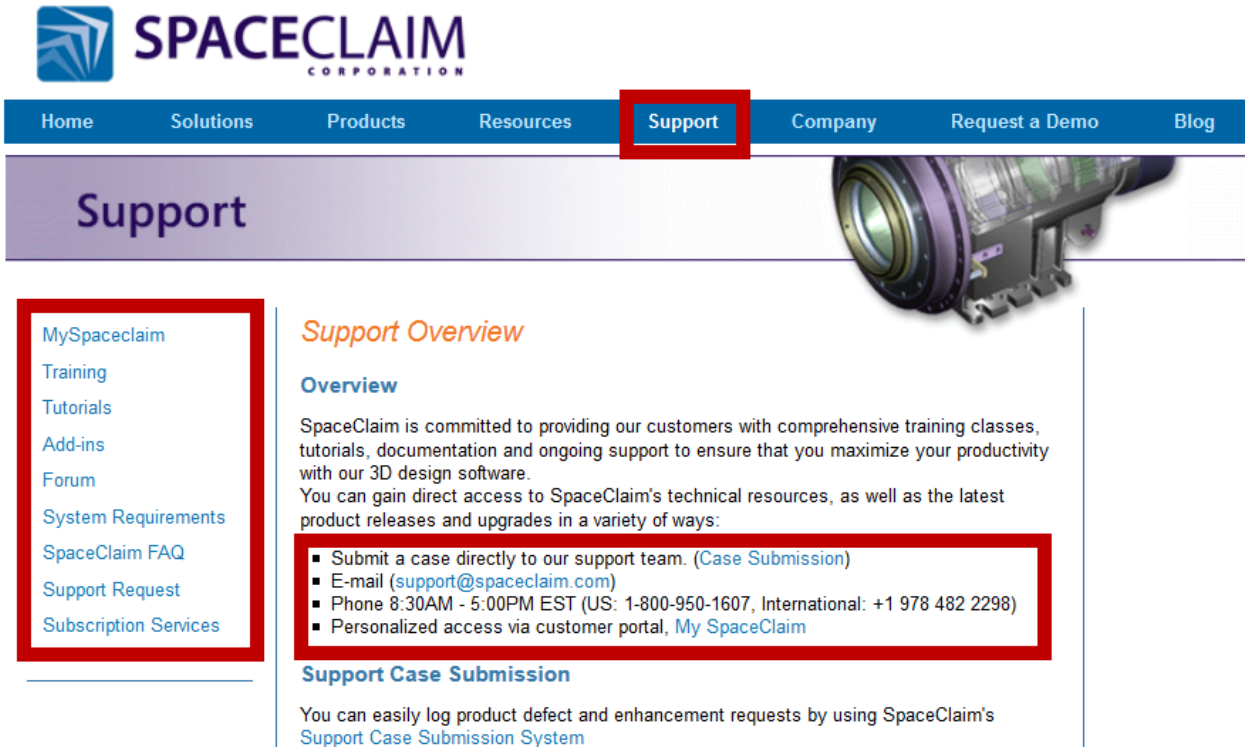
SpaceClaim is continually expanding its live training offerings. Contact your local SpaceClaim Reseller or Sales Manager to inquire about setting up additional live training sessions.

Additional Support

The SpaceClaim www.spaceclaim.com webpage has lots of additional help

Click on the **Support Tab** to see a list of additional help, like more **Tutorials**, **Add-ins**, a **Forum** and how to **contact SpaceClaim** directly for **free support** that is included with your subscription service.

Please contact support with any questions using SpaceClaim or to request any enhancements



SPACECLAIM
CORPORATION

150 Baker Ave. Ext., Concord, MA 01742 USA
Tel: +1 978.482.2100 Fax: +1 978.369.5864

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