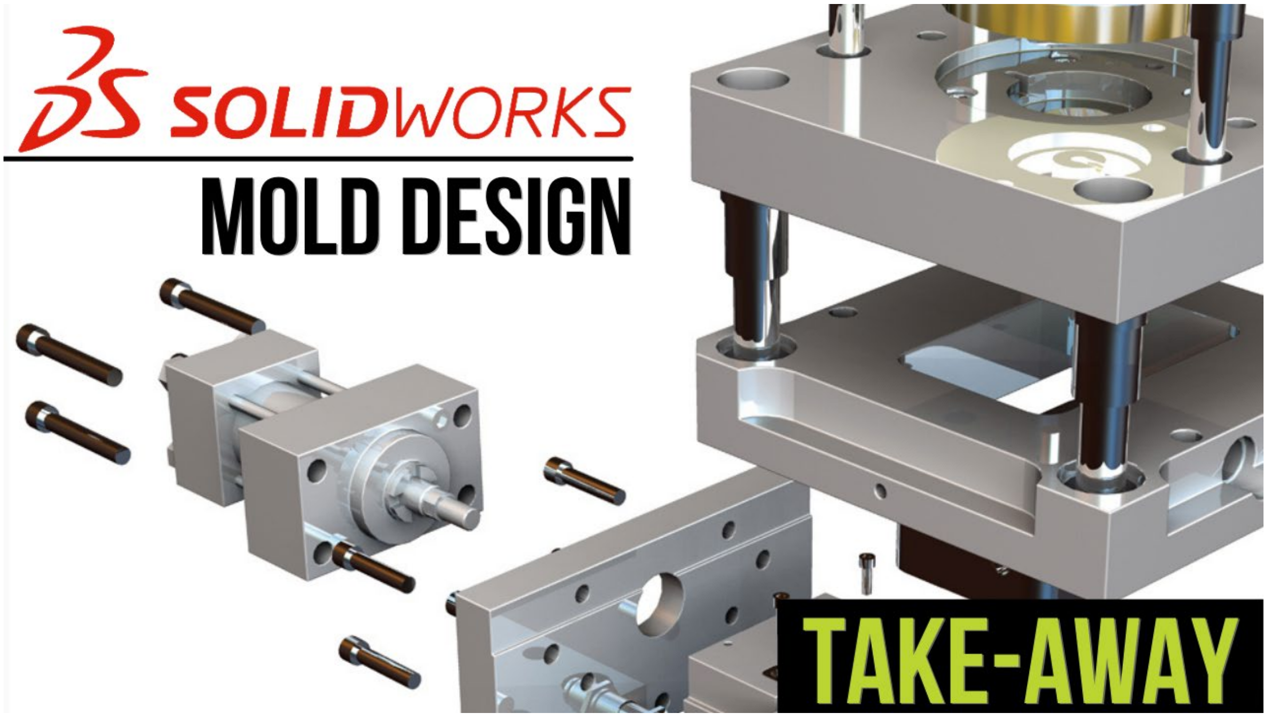


Lessons

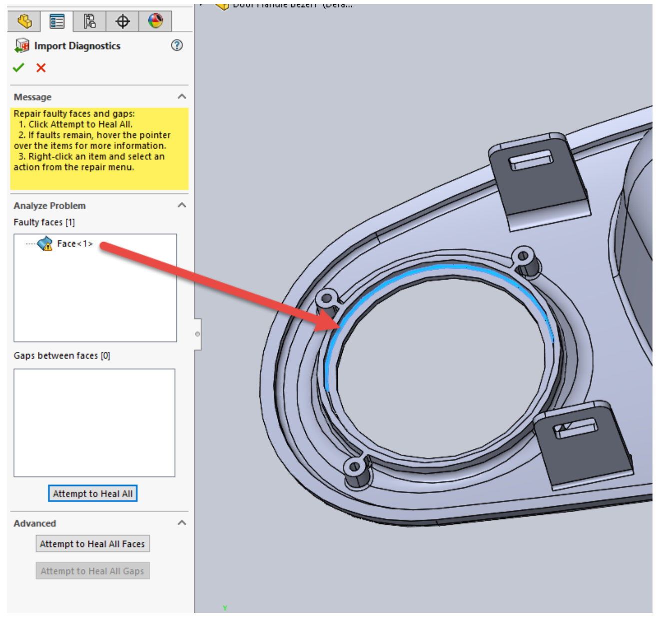
[Lesson 1: Surface Concepts and Imported Geometry]
[Lesson 2: Core and Cavity]
[Lesson 3: Side Cores and Pins]
[Lesson 4: Advanced Parting Line Options]
[Lesson 5: Custom Surfaces]
[Lesson 6: Advanced Surfacing]
[Lesson 7: Alternative Methods for Mold Design]
[Lesson 8: Reusable Data]
[Lesson 9: More Mold Design Workflows]

Disclaimer: This document is a comprehensive summary of critical key takeaways from lessons within SOLIDWORKS Mold Design offered by GoEngineer. This document should not be considered a substitute for an official SOLIDWORKS training course.

Lesson 1: Surface Concepts and Imported Geometry

Imported Geometry

It is not unusual for mold designers to import files when working on new mold designs. The imported geometry could be for the actual part they are creating or for any of the mold base components that go into the completed mold. When we import a model, SOLIDWORKS will attempt to translate all the surfaces of the model and create a solid body. On some occasions, this translation will not work as expected. Repairs will have to be made to surface bodies before a solid model can be formed.



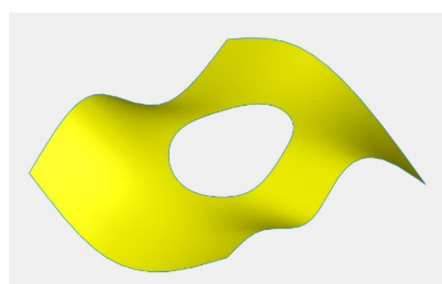
Solids vs. Surfaces

Solid models are surfaces that meet these two requirements:

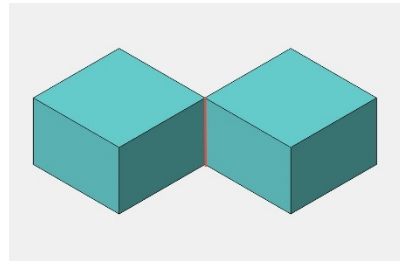
- There are no missing faces or gaps.
- They form a single closed volume.

Rule: Every edge in a solid body is the boundary between exactly two faces. This means that a surface body has edges that bound only one face.

This example shows a surface that has five edges. Each of these is a boundary of a single face.



The example on the right shows how the rule for a solid could not be applied to this model. The edge in the center would be a boundary between four faces. This could not be considered a single solid body.

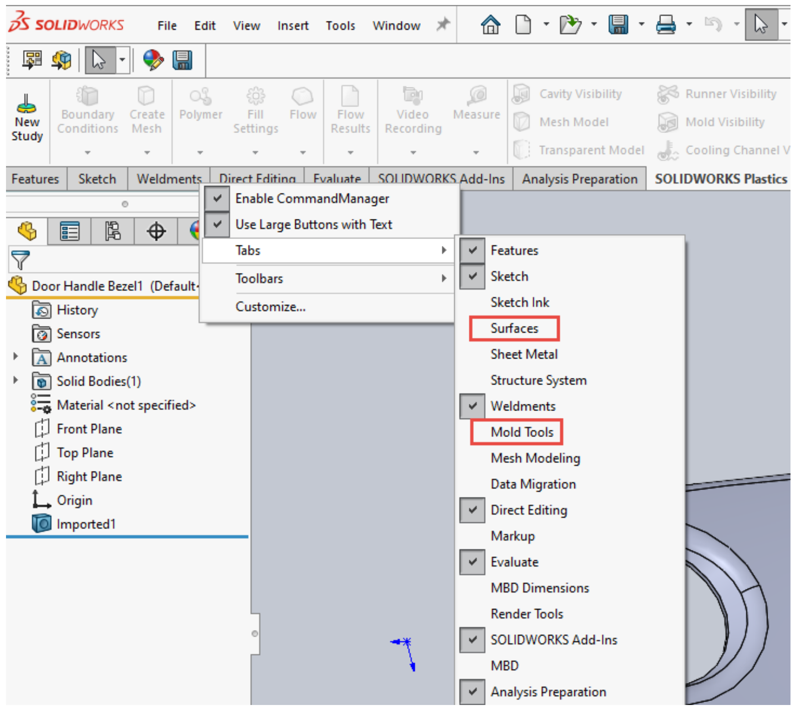


Activating CommandManager Tabs

The **Surfaces** tab and the **Mold Tools** tab should be activated when working with Surfacing and Injection Mold design.

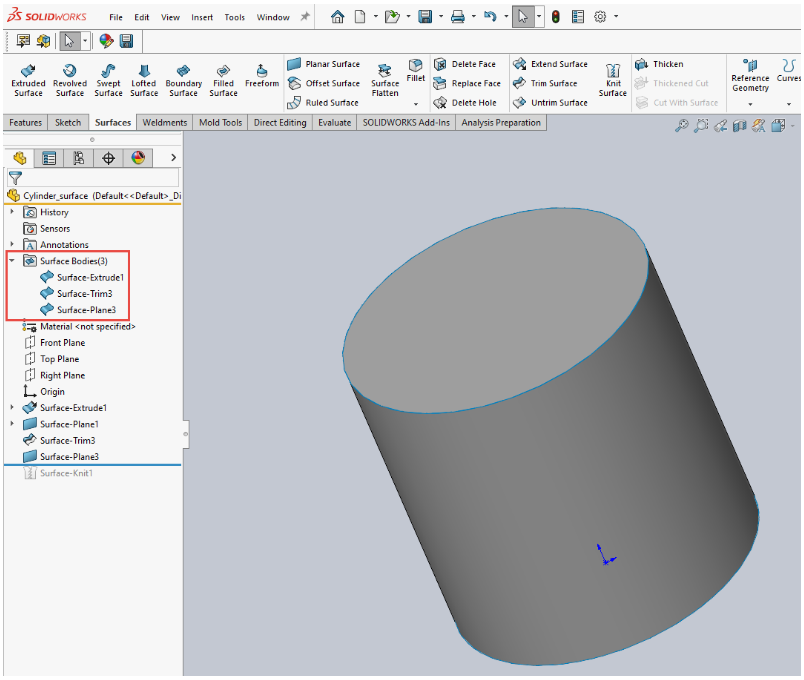
To activate a tab, right-click on any existing tab, such as **Sketch,** then click on Tabs and select the tab you want to activate.

This will need to be repeated for each tab to be activated.



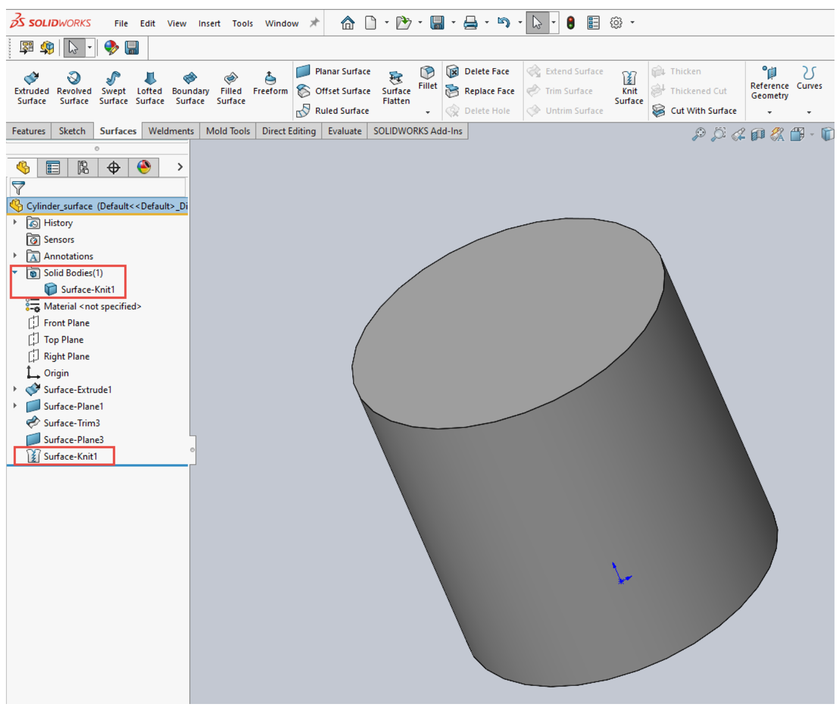
Behind the Scenes

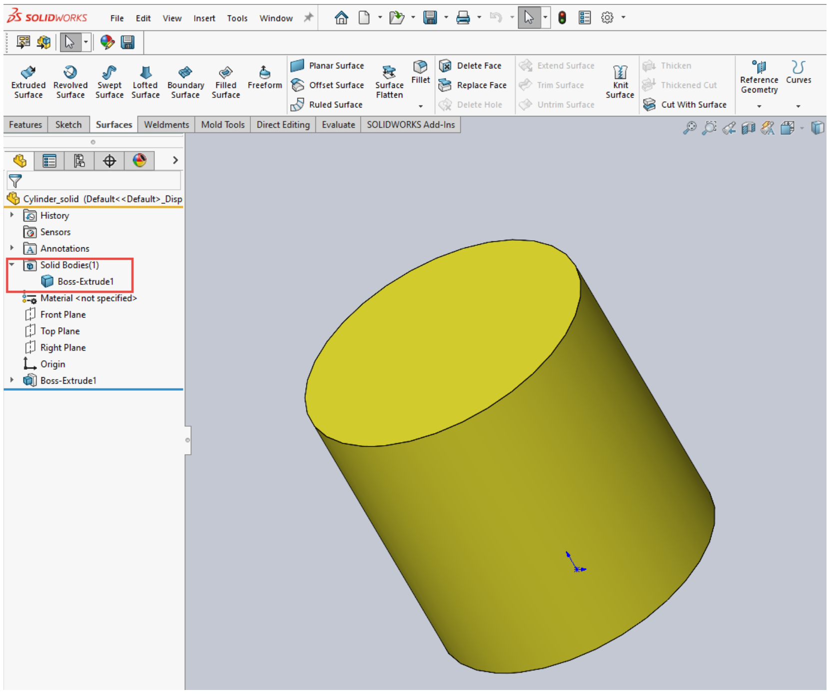
When SOLIDWORKS builds solid models, it is actually automating many surface modeling tasks behind the scenes. Each solid feature is generated by first creating surface bodies and then assembling them to form a closed volume.



The cylinder in the image on the left is shown to contain three surface bodies.

Since these surface bodies contain an enclosed volume, meaning the model is watertight, it can be formed into a solid body with the **Knit** command.



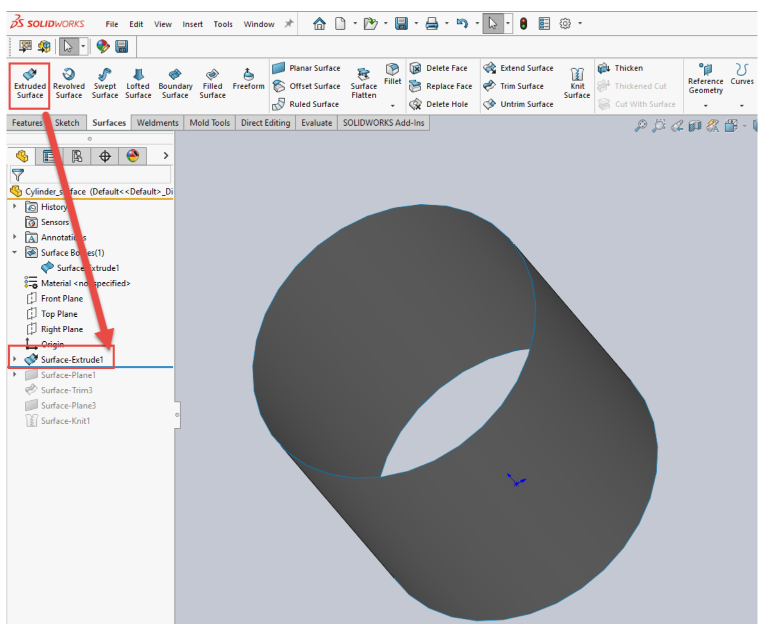


So, when a solid model is created in SOLIDWORKS, the FeatureManager design tree will show the feature and the solid body while, in the background, it creates the surfaces and knits them together automatically.

Extruded Surface

Extruded Surface works exactly like its solid counterpart except that it produces a surface instead of a solid. It does not cap the ends and it does not require a closed loop sketch.

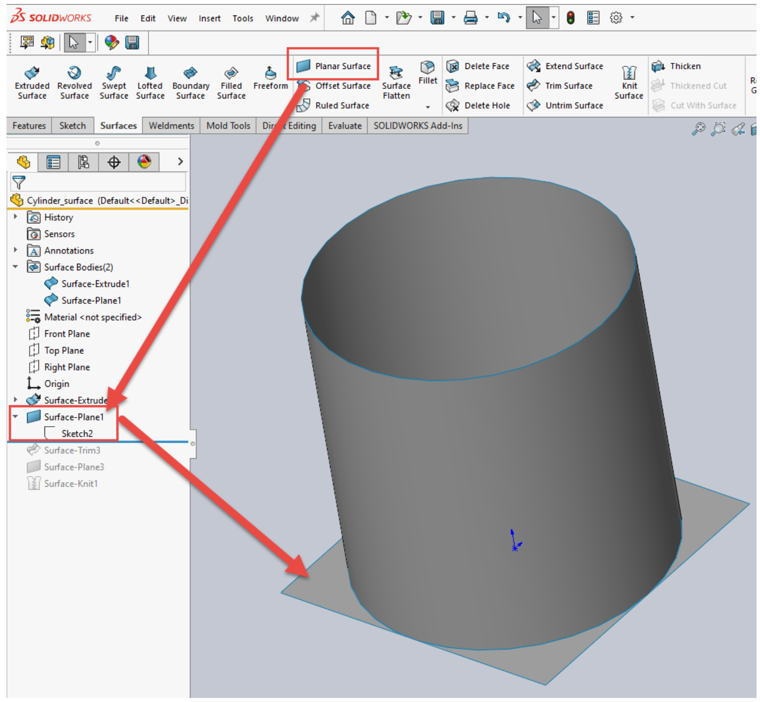
The **Extruded Surface** command can be found on the Surfaces tab.



Planar Surface

A **Planar Surface** is created from a non-intersecting closed-loop sketch, a set of closed edges, multiple coplanar parting lines, or a pair of planar entities such as curves or edges.

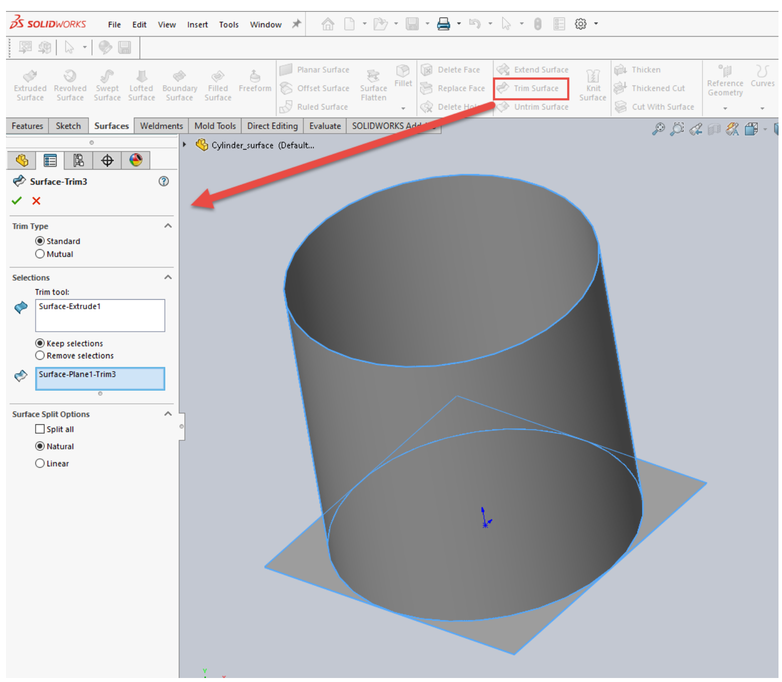
The **Planar Surface** command can be found on the **Surfaces** tab and the Mold Tools tab.



Trim Surface

The **Trim Surface** command enables you to cut back a surface using either another surface, a plane, or a sketch. There are two trim types available for this feature:

- **Standard** where a surface plane or sketch is used as the trimming tool.
- **Mutual** where multiple surfaces trim one another.

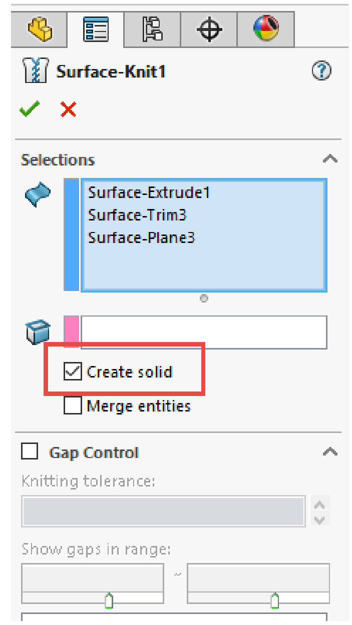


The **Mutual** trim also knits the resulting surfaces together, while **Standard** trim leaves them as separate surface bodies.
When making selections for the trim, options can be adjusted to keep the selected areas or remove selections.

The **Trim Surface** command is found in the **Surfaces** tab.

Creating Solids from Surfaces

There are two ways to create a solid body from an enclosed volume surface body.

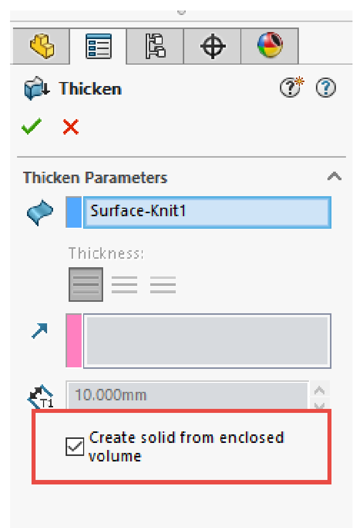


Knit Surface

The **Knit Surface** command joins together separate surface bodies into a single surface body. To knit surface bodies together, their edges must touch or be within a gap control tolerance. There is an option to choose 'Create solid' when the surface is an enclosed volume.

The **Knit Surface** command can be found on both the **Surfaces** and **Mold Tools** tabs.

Thicken



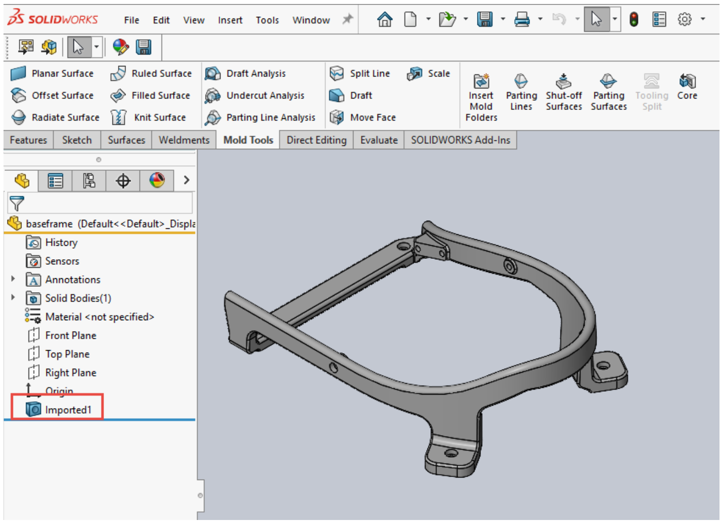
Thicken creates a solid body by thickening one or more adjacent surfaces. Surfaces must be knit together before thickening. If the surface forms a closed volume, the option to 'Create solid from enclosed volume' will be available.

Then Thicken command can be found on the **Surfaces** tab.

Importing and Mold Design

STEP File : (*.step, *.stp)
IGES: (*.igs)
PARASOLID (*.x_b, *.x_t)

Many times, when working on a part file from a customer, you will be provided with a file format such as a STEP, IGES, or Parasolid. These are called "neutral" formats that most 3D CAD packages can export and import. Most CAD packages have their own proprietary modeling engine that creates their solid model and stores the model information in a certain way. When a neutral format file is open, the CAD package will read and try to convert the file to a solid body if possible. Parasolid and STEP file formats are used most often.



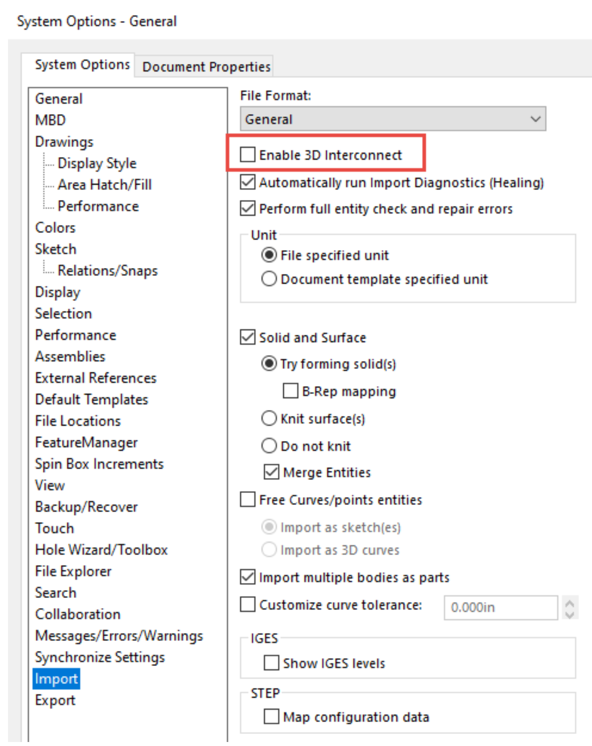
Parasolid is the native modeling kernel for SOLIDWORKS, so this would be the first choice to work with.

Even though the Parasolid file is native to SOLIDWORKS, when a Parasolid file is opened, the FeatureManager design tree will only have an imported solid body - there will be no history.

3D Interconnect

SOLIDWORKS has some import option settings that can be found in **System Options** > **Import.** A newer import option is **Enable 3D Interconnect.**

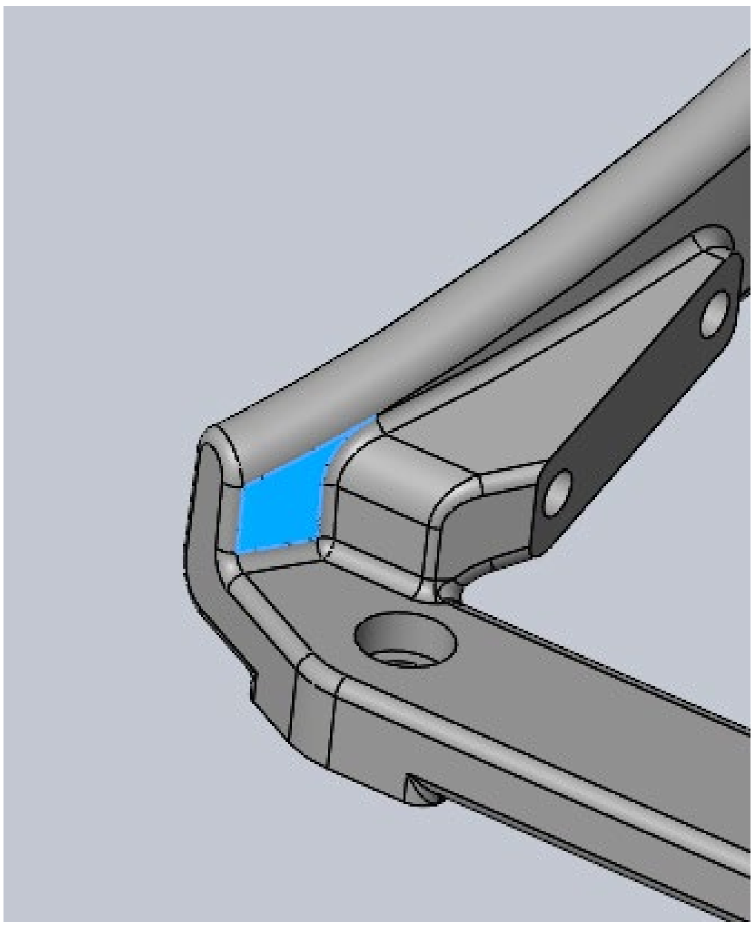
This option allows files from other CAD packages to be opened in their native format. 3D Interconnect retains a link to the native CAD file so that if it is modified in its original application, that data in SOLIDWORKS can be easily updated. This is especially beneficial when working with models that are put into an assembly.

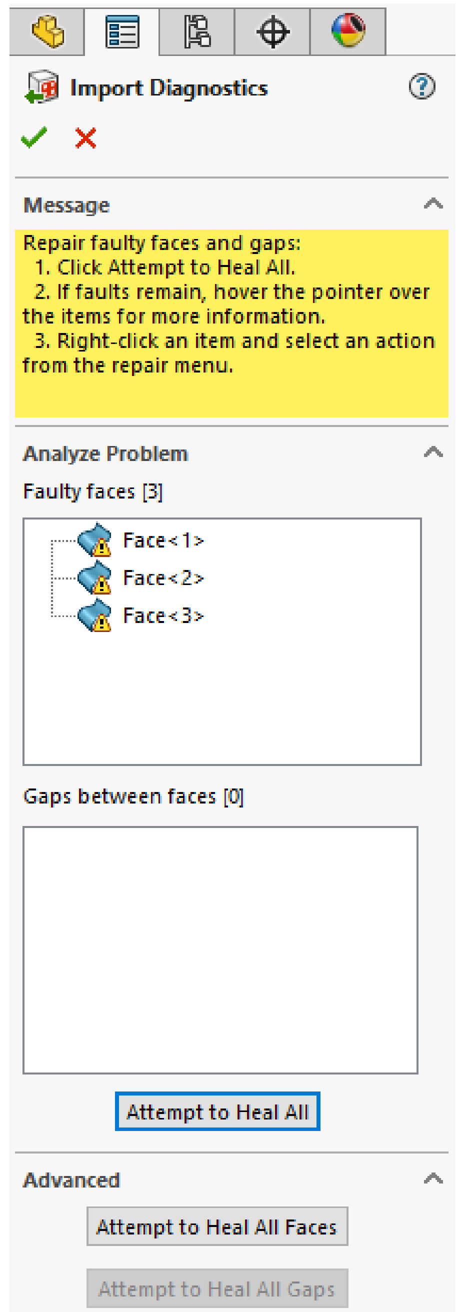


Import Diagnostics

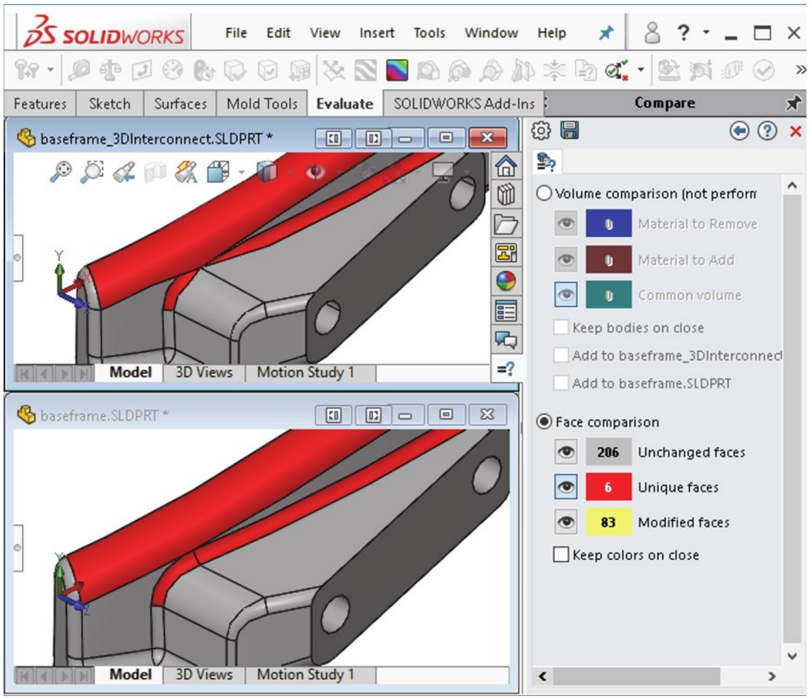
When importing a model, users are automatically prompted to run **Import Diagnostics.**

This tool can not only identify problem areas in a model but also has built-in capabilities to repair faulty faces and gaps. If gaps are present, the model will not be able to form a solid body and will result in a surface body instead.





Comparing Geometry



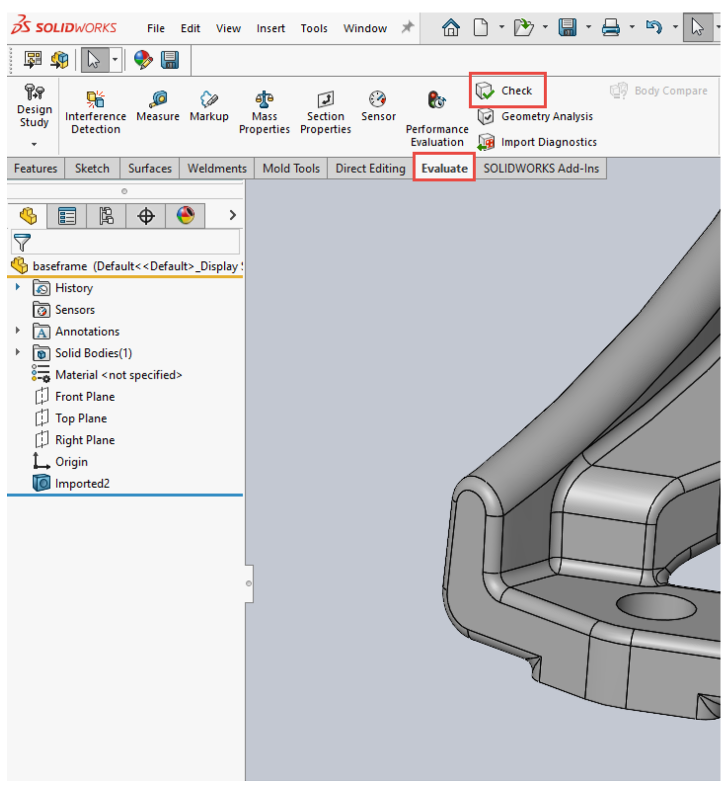
The **Compare Geometry** tool lets users compare models side by side, visually showing the differences between the models.

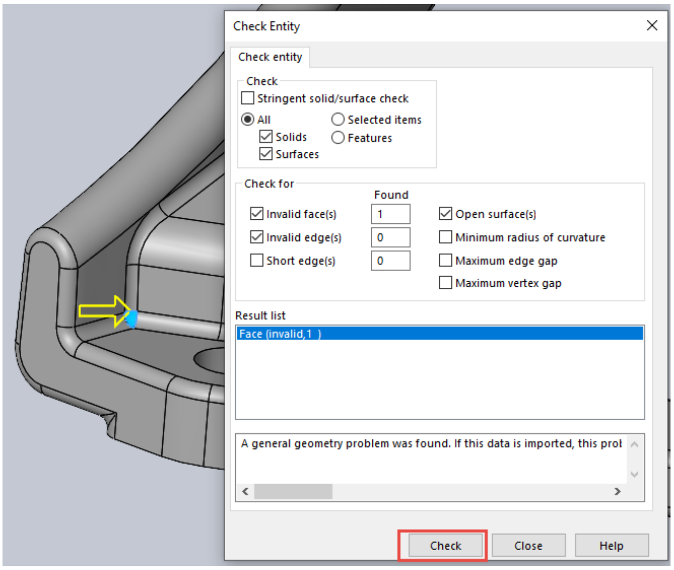
Access **Compare Geometry** in: Tools > Compare > Geometry.

Check Entity

Check Entity is a utility that identifies geometry problems and, in some cases, can provide suggestions on how to address issues. It can be used to locate invalid faces or edges that may exist in the model and can also check for the minimum radius of curvature. Additional settings can help identify open edges, short edges, and gaps. By default, the entire model is checked, but options can be adjusted to only check selected areas.

Activate **Check Entity** in the **Evaluate** Tab and choose **Check.**

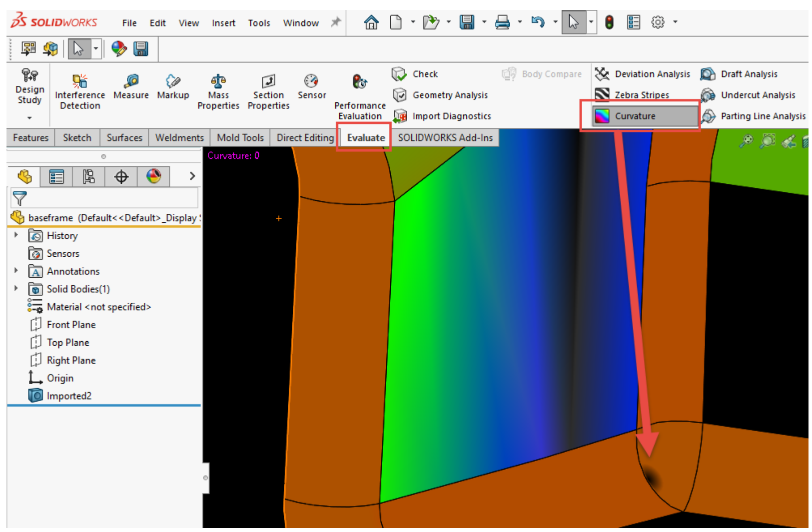




The **Check Entity** command in this case identified a geometry problem in the indicated corner of the model.

Curvature Tool

The curvature tool is another tool that can be used to evaluate geometry. Displaying curvature will render faces of a model in different colors according to the local curvature values. This tool can be useful to help analyze the quality of surface in a part.



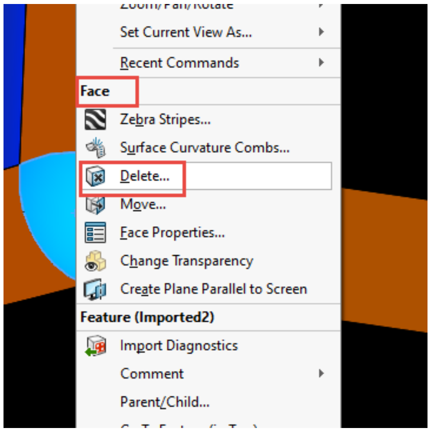
Activate the curvature tool in the **Evaluate** tab and choose **Curvature.**

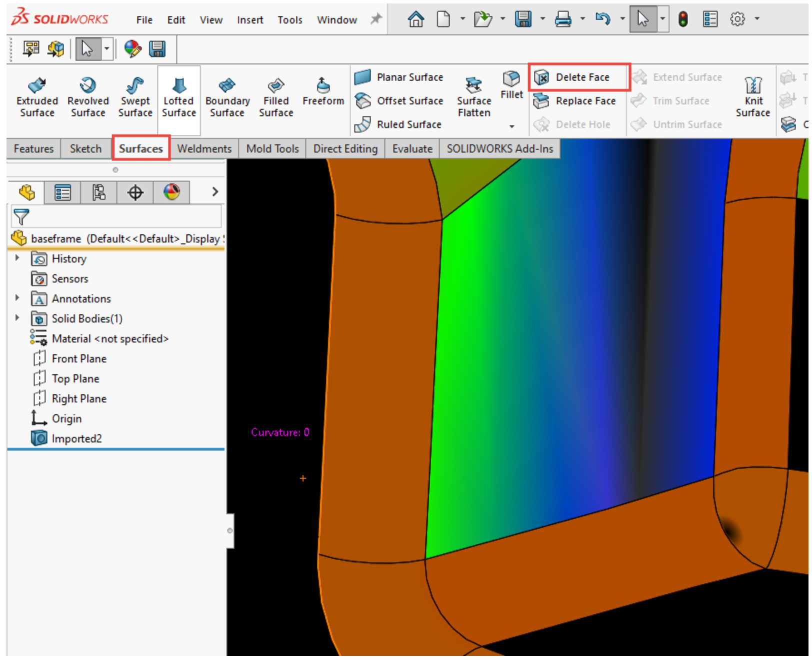
Direct Editing

There are several direct editing tools and techniques that are used when editing imported geometry. The **Delete Face** command is very useful when working with faulty faces and geometry.

Delete Face

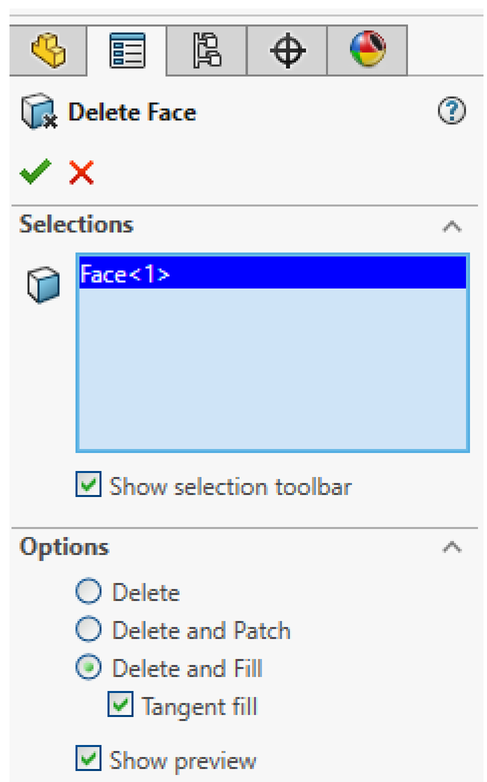
Delete Face can be found in the **Surfaces** tab. It can also be activated by right-clicking on the face and selecting Delete.





The Delete Face options are:

- **Show selection toolbar:** Shows/hides selection accelerator toolbars.
- **Delete:** Deletes a face from a surface body, or deletes one or more faces from a solid body to create surfaces.
- **Delete and Patch:** Deletes a face from a surface body or solid body and automatically patches and trims the body.
- **Delete and Fill:** Deletes faces and generates a single face to close any gap.
    - **Tangent Fill:** Will use tangent edges to form a new face.



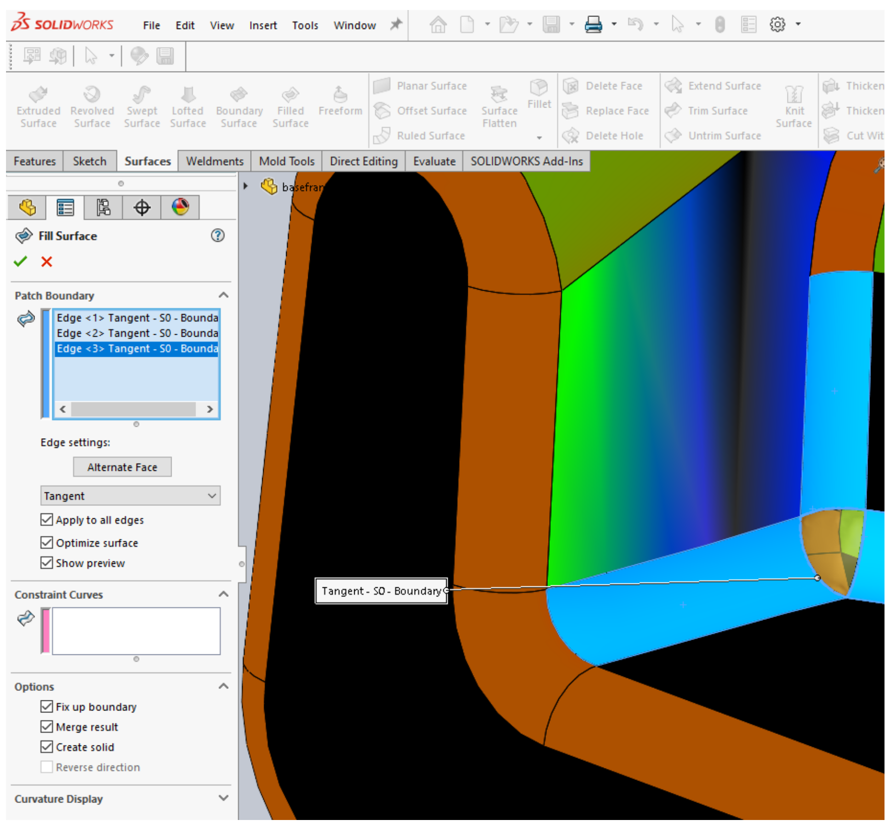
Patching Strategies

Many surface features can be useful for patching holes. Each feature contains different options, so often a trial-and-error approach is recommended to identify the best result.

Some strategies for patching holes include:

- Using a filled surface in loft between edges.
- Creating a boundary surface between edges.
- Removing surrounding geometry in rebuild the faces.

Filled Surface



The **Filled Surface** feature constructs a surface patch with any number of sides within a boundary. The boundary can be defined by existing model edges, sketches, or curves. In some instances, a filled surface can be created without a closed boundary by using the **Fixup Boundary** option.

If edges are selected for a Filled Surface boundary, boundary conditions such as **contact, tangency,** or **curvature** may be selected to relate the new surface to the adjacent faces.

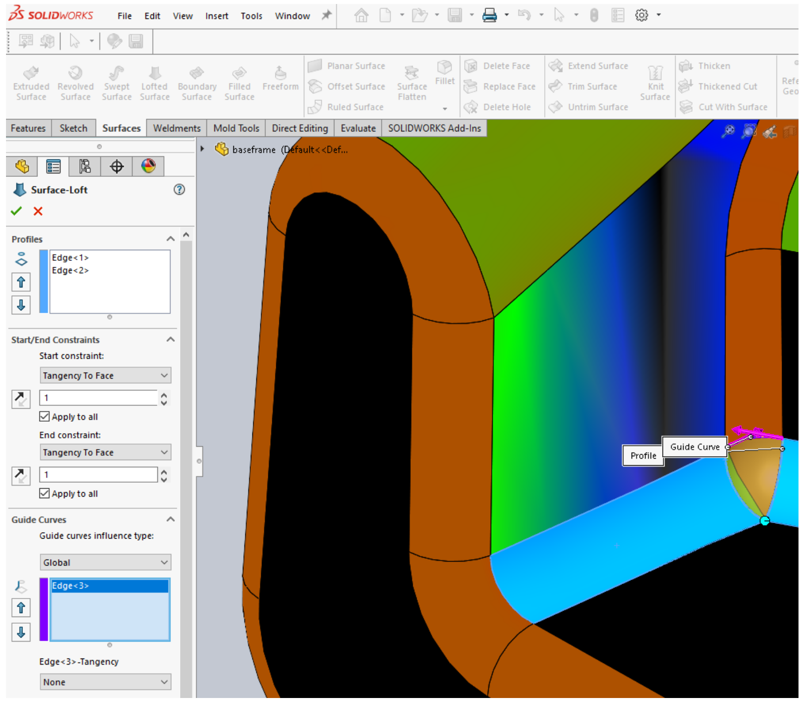
The **Filled Surface** can knit itself into the surrounding surface bodies, knit an enclosed volume into a solid, or integrate itself directly into a solid body.

The **Filled Surface** works by creating a four-sided patch and trimming it to fit the selected boundary.

Lofted Surface

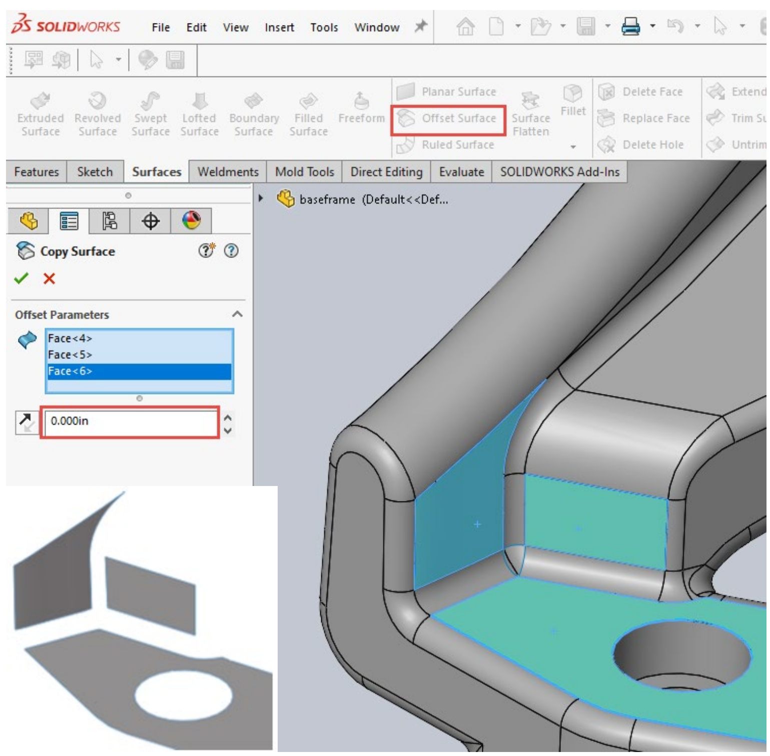
Lofted Surface creates a lofted surface between two or more profiles or sketches you specify, a start and end constraints, and add guide curves to control the path shape.

Lofted Surface can be found on the **Surfaces** tab.



Offset Surface

The **Offset Surface** command offsets surfaces using one or more contiguous faces. The tool creates an offset surface with gaps. You can repair the gaps manually or adjust the offset distance value and run the tool again.



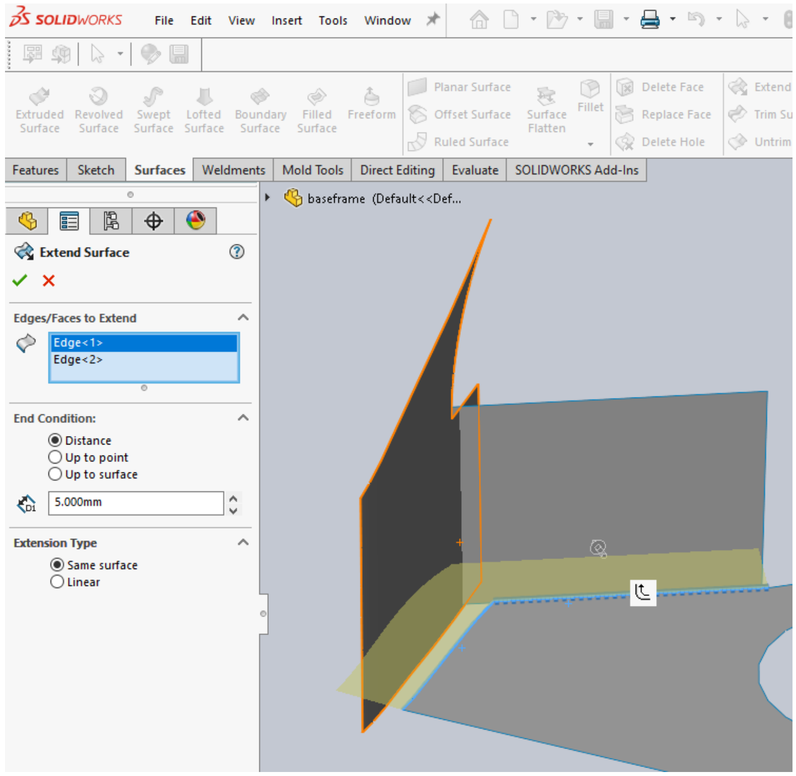
A very useful capability with the **Offset Surface** command is using a 0.000 offset distance. This creates a copy of the surface in place.

By copying the surfaces that are going to be modified, it is easier to modify them as individual surface bodies, rather than modifying them while they are knitted with the rest of the model.

Offset Surface command is found on the **Surfaces** tab.

Extend Surface

Surfaces can be made larger by extending along select edges, or all edges with the **Extend Surface** command. The extension can be an extrapolation of the existing surface or a linear surface that is tangent to the existing surface.



The **Same Surface** option attempts to extrapolate the curvature of the existing surface. The **Linear** option works on any type of surface that often creates a broken edge.

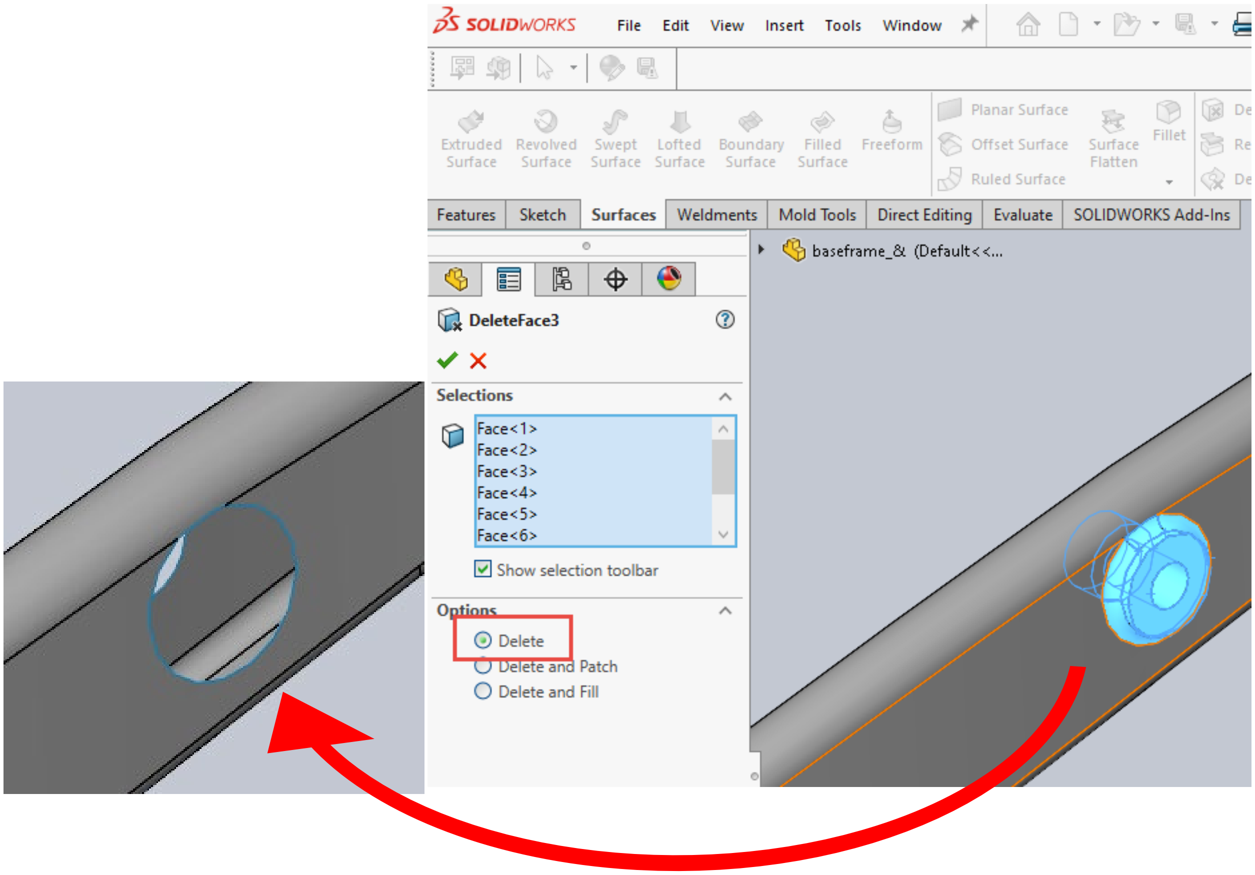
The **Extend Surface** command can be found on the **Surfaces** tab.

Editing Imported Parts

Many of the techniques used for repairing and patching imported geometry can also be used for other design tasks with imported bodies. The **Delete Face** command can be used to modify the part to remove geometry or features from a model.

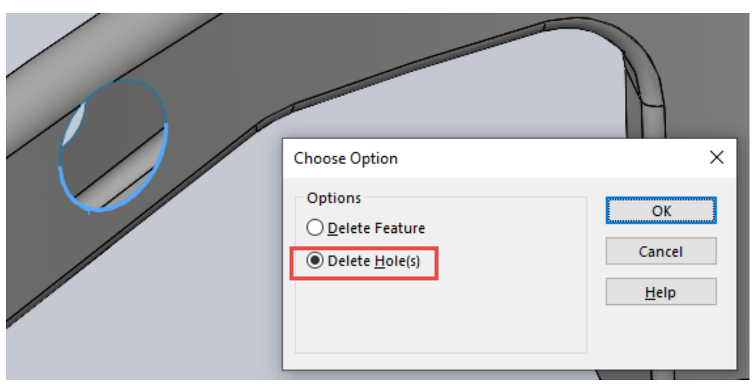
Delete Face

The **Delete Face** command deletes spaces from solid bodies to create surfaces or delete the face from a surface body.



Delete Hole

The **Delete Hole** command is like untrimmed surface except that it only works on closed interior loops. Using **Delete Hole** can be an effective technique to close holes and patch gaps in a model.

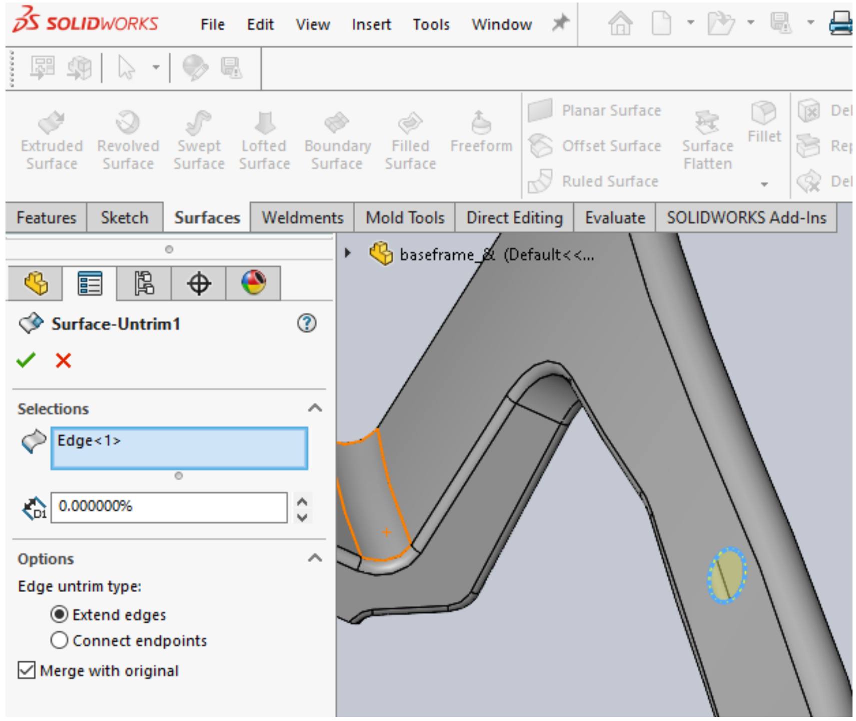


To use **Delete Hole**, select the edge of a hole and press **Delete** on the keyboard. The system will prompt you, asking if you want to delete the feature or delete the hole.

Untrim Surface

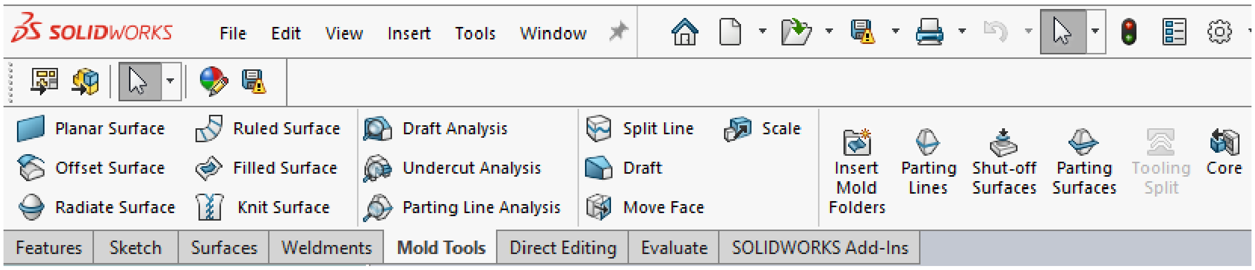
Untrim Surface patches surface holes and external edges by extending an existing surface along its boundaries. You can untrim open or closed regions and merge new surfaces with the original surface.

Untrim Surface is found on the **Surfaces** tab.



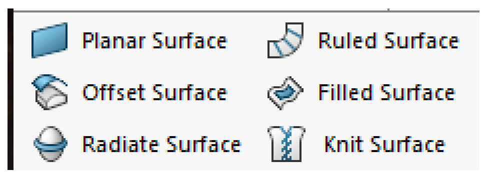
Lesson 2: Core and Cavity

Mold Tools CommandManager



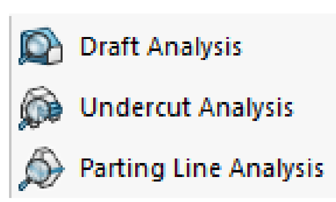
The **Mold Tools** CommandManager is laid out in an order that allows you to work from left to right to get your completed mold split.

The leftmost section has surfacing tools that are used to manually create mold shut-off and parting surfaces.



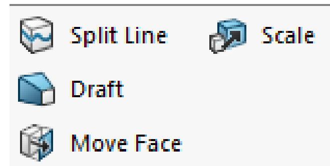
- **Planar Surface** creates a planar surface using a non-intersecting sketch or a set of closed edges or multiple coplanar parting lines.
- **Offset Surface** offsets surfaces using one or more contiguous faces.
- **Radiate Surface** radiates the surface originating from an edge parallel to a point.
- **Ruled Surface** inserts ruled surfaces in a specific direction from edges.
- **Filled Surface** patches a surface within a boundary defined by edges, sketches, or curves.
- **Knit Surface** combines two or more adjacent non intersecting surfaces into a single surface body.

The next section to the right includes the analysis tools.



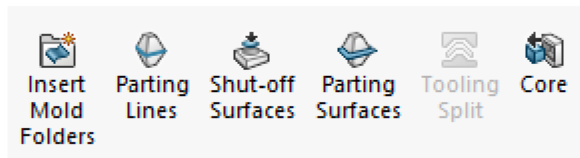
- **Draft Analysis** analyzes draft angles of faces based on a mold pull direction.
- **Undercut Analysis** identifies spaces that form undercuts on molded parts.
- **Parting Line Analysis** analyzes a mold's parting line based on a mold pull direction.

Next are some of the editing tools commonly used in mold design.



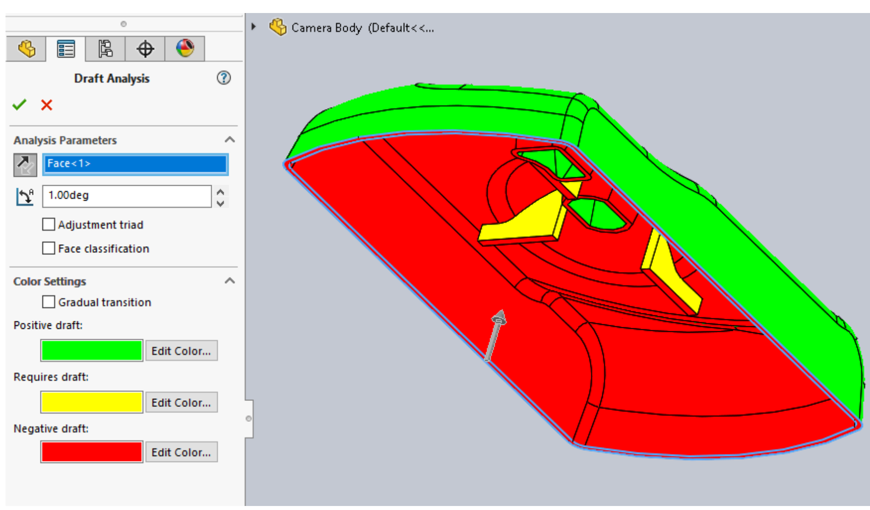
- **Split Line** projects a sketch to curved or planar faces creating multiple separate faces.
- **Draft** tapers model faces by a specified angle using a neutral plane or parting line. Draft is used to make a molded part easier to remove from the mold.
- **Move Face** offsets, translates, or rotates faces or features on a solid or surface model.
- **Scale** scales the model up by a specified factor.

The final section includes automation tools for creating the core and cavity inserts.



- **Parting Lines** define parting lines to separate core and cavity surfaces.
- **Shut-off Surfaces** creates mold shut-off surfaces by creating a surface patch along a parting line or edges that form a continuous loop.
- **Parting Surfaces** creates a parting surface that extrudes from parting lines or edges and splits the model cavity from the core.
- **Tooling Split** inserts a tooling split feature to create the core, cavity, and parting surfaces. It can also generate an interlock surface to help align the mold blocks.
- **Core** extracts geometry from a tooling solid to create a core feature. Specify the direction for extraction and options such as draft angle, end conditions, and cap ends.

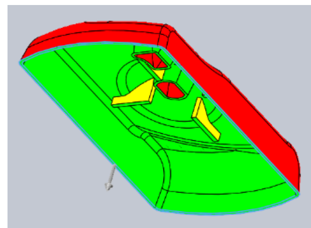
Draft Analysis

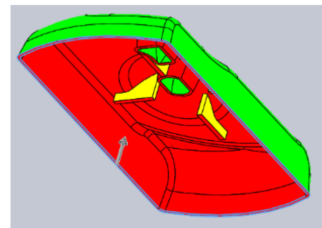


The **Draft Analysis** command is used to make sure that the faces of a part have enough draft. A Draft Analysis is run by specifying the direction of pull and the required draft angle. The direction of pull can be specified as normal to a selected plane, face, or surface **or** in the direction of a selected line, edge, or axis. Once the analysis parameters are selected, all faces of the part are assigned colors to show the amount of draft relative to the draft angle setting.

Green faces have positive draft, the side with red faces has negative draft, and the yellow faces may require more draft. The Mold Tools will use green for the cavity side of the tooling split and red for the core side.

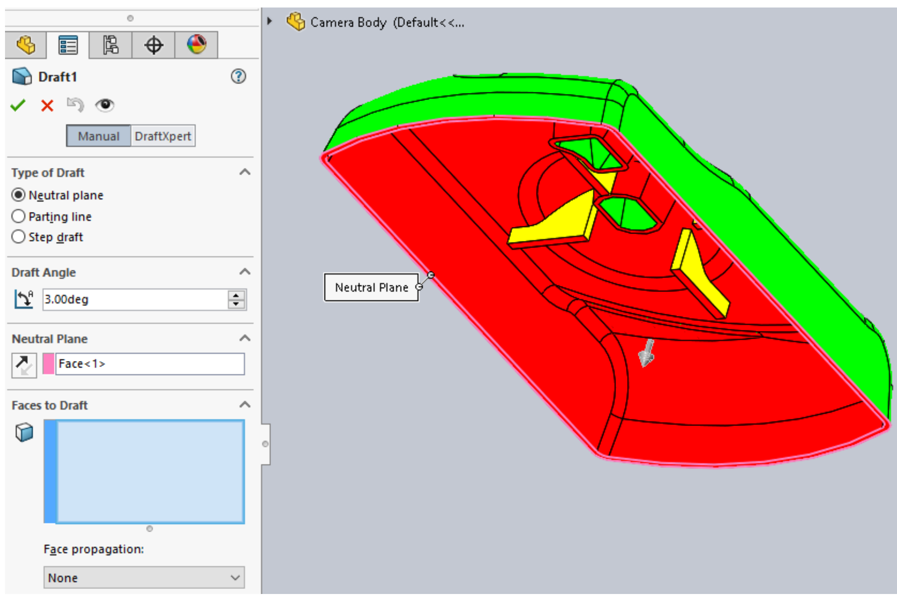
This is good to think about when running the draft analysis. If the colors do not match what you would expect, you can change the draft direction.





Adding Draft

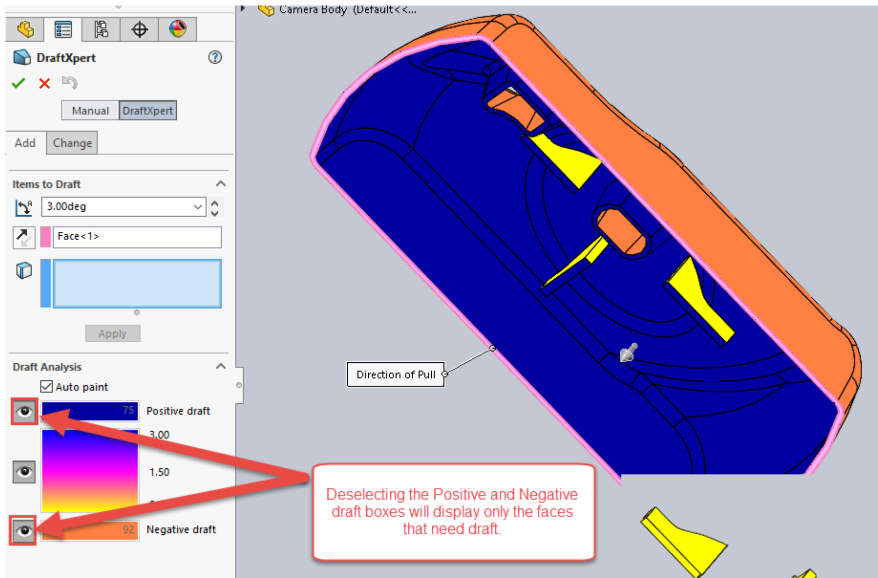
The draft tool allows for three types of draft:

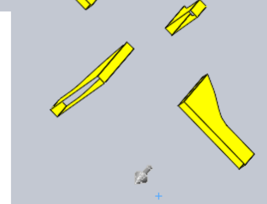


- **Neutral plane** should be used if there is a plane or face that can be selected that represents the direction of pull as well as the location that the draft angle should be applied from. This is also the only type of draft that will work with DraftXpert.
- **Parting line draft** is used when draft needs to be applied from edges that are not planar. Parting line edges can be selected that define where the draft angle should begin.
- **Step draft** is used when draft needs to be applied from edges that are not planar. Parting line edges can be selected that define where the draft angle should begin.

DraftXpert

DraftXpert mode can assist with face selections and will automatically order draft features appropriately in feature history. DraftXpert can also be used to easily change existing draft features.



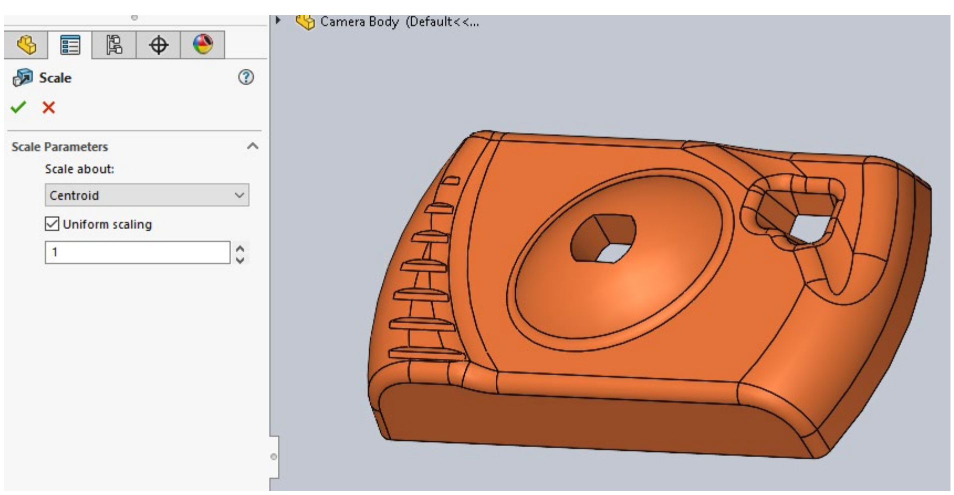


Auto Paint applies draft analysis colors according to the selections. Deselecting the eye on **Positive draft** and **Negative draft** will display only the faces that need draft added.

A quick tool to select all of these faces would be to select **Ctrl+A** on the keyboard that will select all visible faces.

Scaling the Model

Mold tooling is manufactured slightly larger than the plastic part produced from the mold. This is done to compensate for the shrinkage that results as the hot ejected plastic cools.

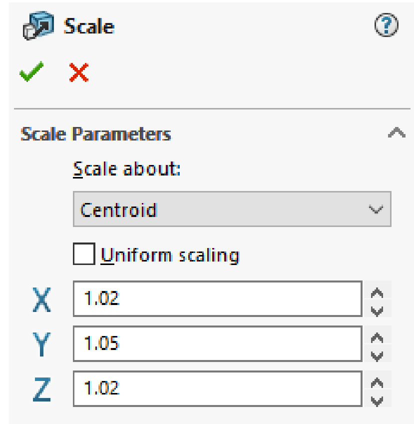


The **Scale** feature can be used to increase or decrease the size of a model.

There are three options for what to scale the model about:

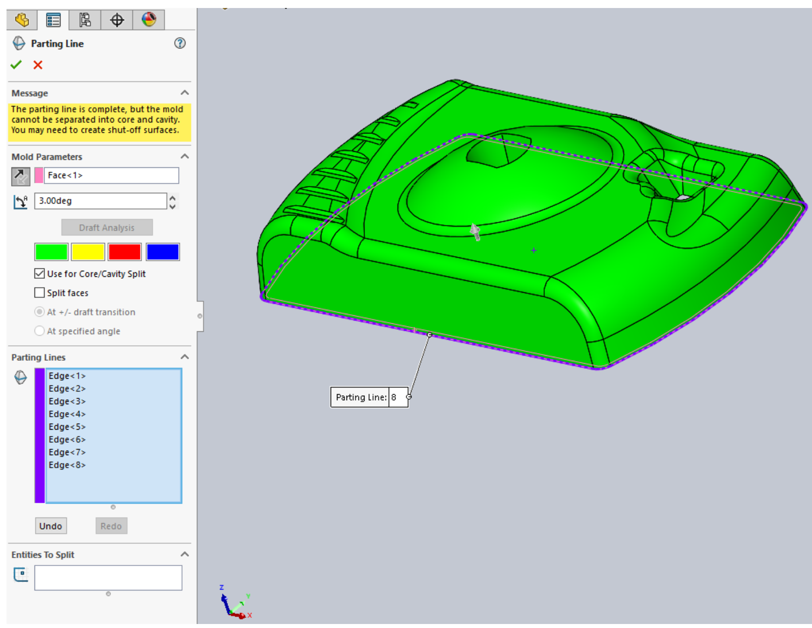
- **Centroid** scales the model about its system calculated centroid.
- **Origin** scales the model about the model origin.
- **Coordinate System** scales the model about a user-defined coordinate system.

The **Uniform scaling** option applies the same scale factor in all directions. Clearing the Uniform scaling option will allow you to specify a different scale factor for each axis.



Establishing the Parting Lines

Parting Lines are the edges of the molded plastic part that border the cavity and core surfaces. The edges of the parting lines are the edges used to separate the surfaces that belong to the core and the cavity. They are also the edges that form the inside perimeter of the parting surfaces.

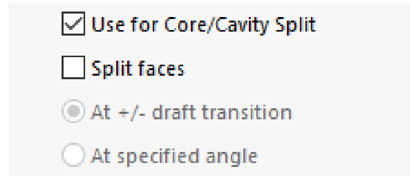


The Parting Lines command allows the designer to establish the parting edges automatically or manually. Later, this parting line feature will be used to create parting surfaces. A draft analysis is done as part of the Parting Line command.

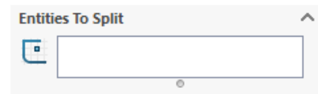
Typically, edges on the model where faces classified as positive and negative draft meet are selected as parting line edges.

Parting Line Options

- **Use for Core/Cavity Split** is used to specify that the parting line will be used for the creation of the mold tooling. When selected, a set of core/cavity surface bodies will be created automatically when the Parting Line feature is completed.



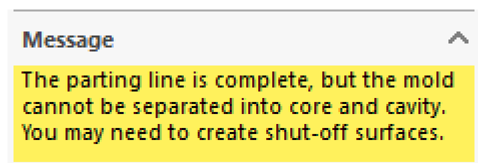
- **Split Faces** is used to split a face along the draft transition from positive to negative draft on a straddle face. A straddle face is a face that straddles the parting line plane.
- **Entities To Split** forces a parting line to cross a planar face. You can select either pairs of vertices or sketch entities.



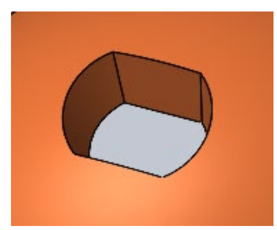
Manual Parting Lines
The software may not be able to find the full parting line with a more complex parting line automatically. In these cases, the parting line would be selected manually around the part.

Shut-off Surfaces

Once the parting line is created, the next step is to determine any open molding areas on the plastic part that need Shut-off Surfaces.
The message window at the top of the Parting Line PropertyManager will let you know if you need to add shut-off surfaces.

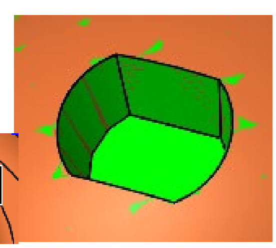


Shut-off Surfaces are used to define the boundary between the core and cavity halves of the mold when there is no physical boundary in the part itself. These open molding areas are where two pieces of tooling will touch coincidentally to form the hole.

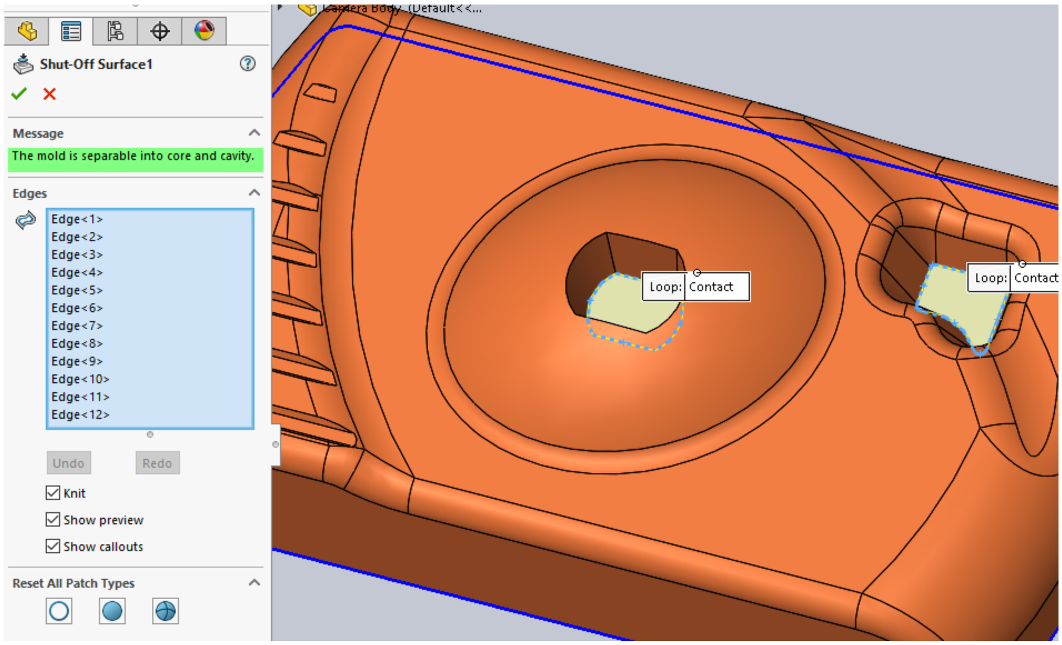


There are three patch types for **Shut-off Surface:**

- **Tangent** creates a patch that is Tangent to adjacent faces.
- **Contact** creates the simplest patch which contacts adjacent faces.
- **No Fill** creates the simplest patch which contacts adjacent faces.

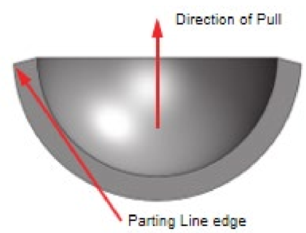


Clicking on the Loop Flag switches between the patch type.
When the shut-off surfaces are complete, the message will let you know that the mold is separable into core and cavity.



Creating the Parting Surface

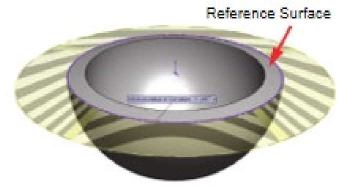
The **Parting Surface** feature is designed to automate the creation of the faces of the tooling that surrounds the part. The parting surface should extend further than the intended tooling block size unless an interlock surface will be included in the design.



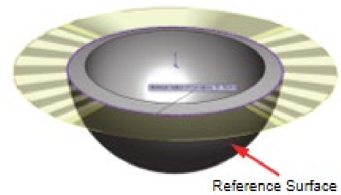
The Parting Surfaces command creates surfaces that extrude from the parting line.

These surfaces can be aligned in one of three ways:

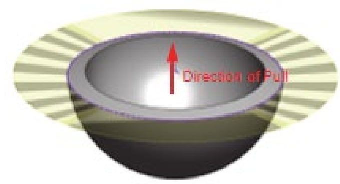
- **Tangent to Surface**: The parting surface is tangent to the surface of the model that is closest to being normal to the direction of pull.



- **Normal to Surface:** The parting surface is normal to the face which is closest to being parallel to the direction of pull.



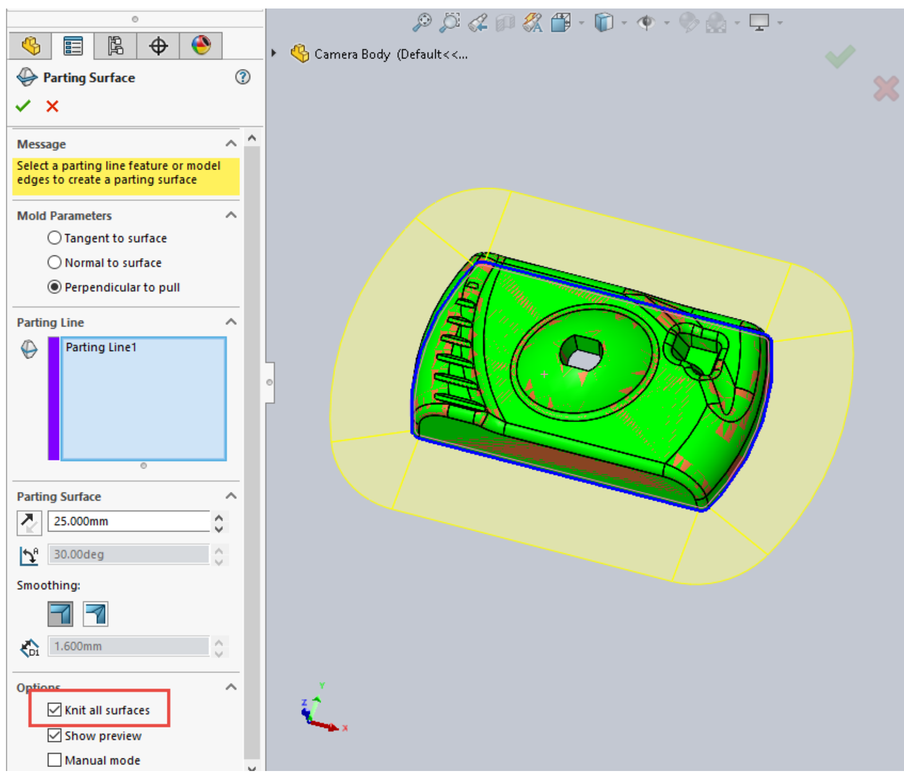
- **Perpendicular to Pull:** The parting surface is normal to the face which is closest to being parallel to the direction of pull.



The Parting Line will automatically be selected when the Parting Surface command is started.

Choose the **Knit all surfaces** option to create one parting surface body.

The Manual mode option displays handles that allow the parting surface to be manipulated manually.

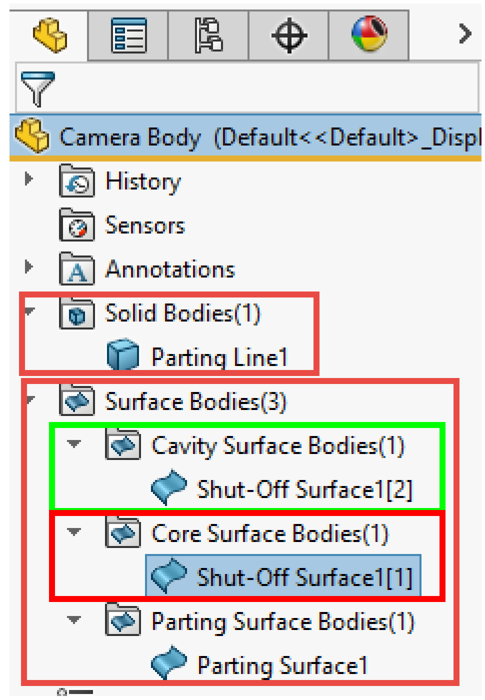


Surface Bodies

The **FeatureManager Tree** will have a **Solid Bodies** folder and a **Surface Bodie**s folder after the **Parting Surface** command is complete.

The **Solid Bodies** folder contains the original part file. There are three folders inside the surface bodies folder. These are where the surfaces that will be used during the tooling split to create the cavity, core, and parting surfaces for the mold are stored.

Having the surface in the proper folders is very important for the tooling split command to work.

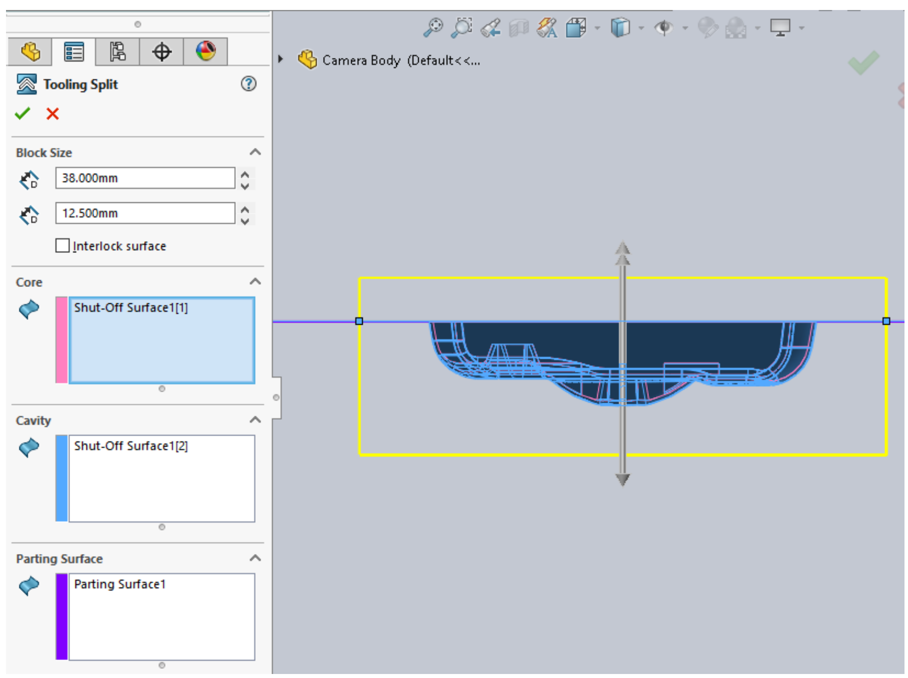


Creating the Mold Tooling

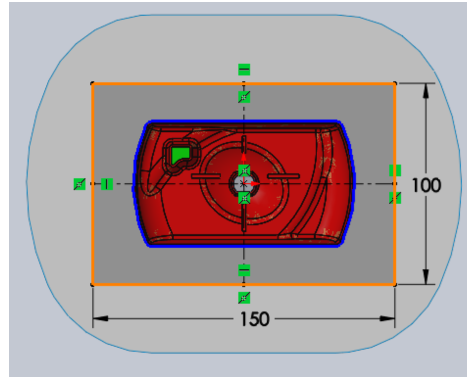
The **Tooling Split** command automates the creation of the solid bodies that represent the cavity and core of the mold tooling.

The Block Size is where the core and cavity block thicknesses are set.

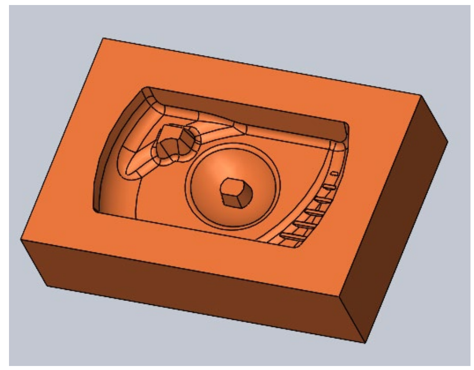
The surface bodies for the Core, Cavity, and Parting Surface are automatically in the proper selection panes.



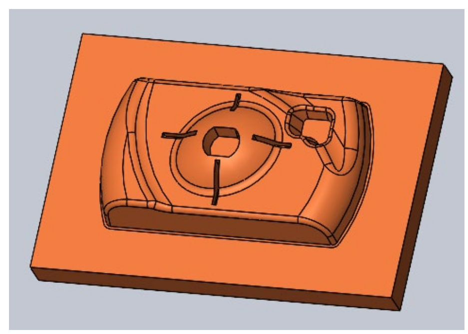
The **Tooling Split** feature requires the sketch at the proper location for the core and cavity split and at the desired size of the mold inserts. The command then uses the surfaces in the Surface Bodies folder to create the faces of the core and cavity solid bodies.



The Cavity Surface Bodies and the Parting Surface Bodies are combined and used to cut a solid block for the cavity side of the mold.

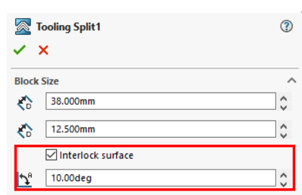


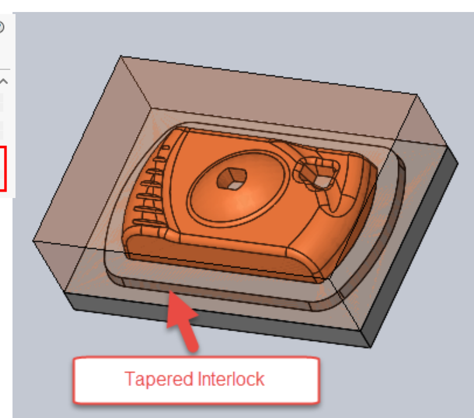
The mold core is created by combining the Core Surface Bodies with the Parting Surface Bodies. These surface bodies are cut from the same solid block.



Interlocking the Mold Tooling

The **Tooling Split** command includes an option to automatically create an interlock surface.

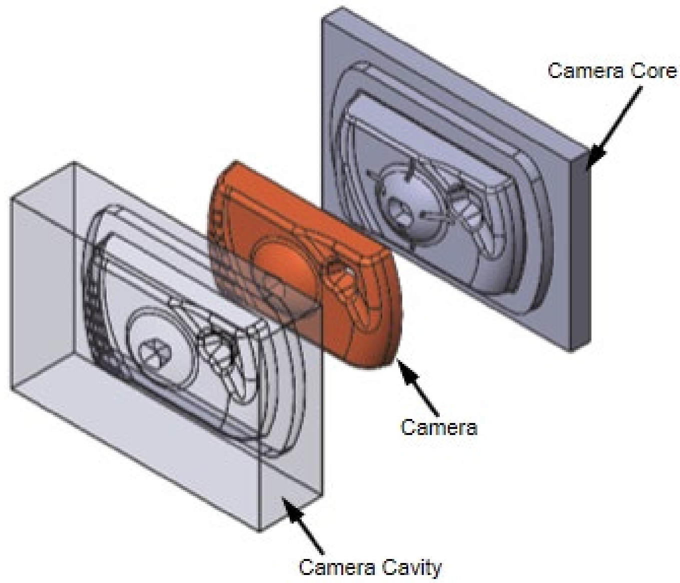




Interlock surfaces are tapered from the parting surfaces and help the mold seal properly. They also help guide the tooling into place when the mold closes and keep the tooling aligned when the mold is closed. The taper also keeps the steel that forms these surfaces from galling when the mold is open or shut.

Creating Part and Assembly Files

The final step in creating mold inserts is to save the bodies as individual parts for use in an assembly. These steps are automated using the **Save Bodies** command.

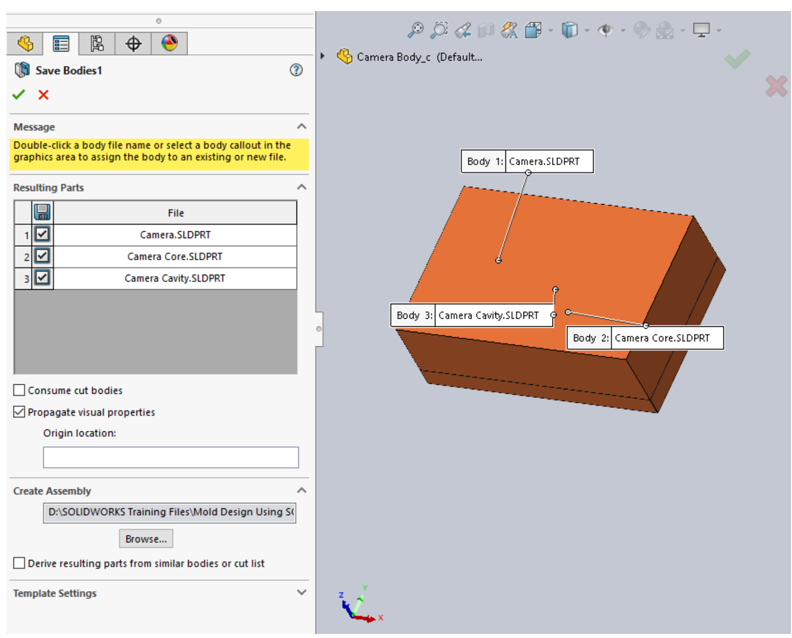


The default file name for parts created with this tool will be the names of the solid bodies. The bodies can be renamed prior to saving them to automate the correct part file names.

To activate the **Save Bodies** command, right-click on the **Solid Bodies** folder in the FeatureManager Design Tree.

The bodies to be saved are selected by clicking on the checkboxes.

The Assembly file location can be set to create an assembly containing the bodies selected.

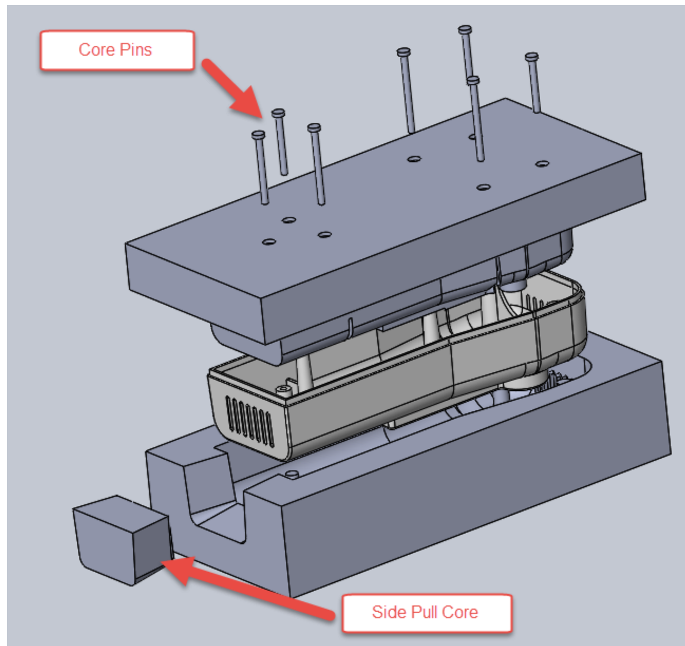


The new parts and assembly are created. Each part in the assembly has external references back to the solid bodies in the original part file.

Lesson 3: Side Cores and Pins

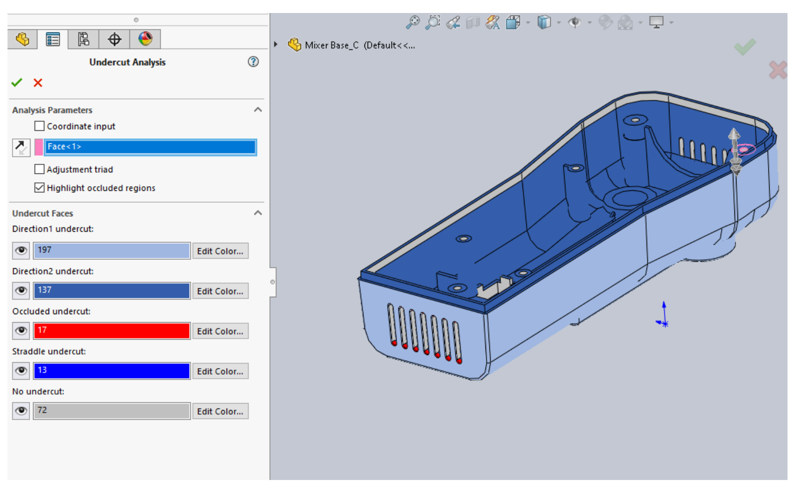
Additional Mold Tooling

Some molds may need to have components that do not travel in the same direction as the mold parting direction. These would have side pull cores that are utilized to create undercut geometry on the part. Typically, molds will have replaceable core pins that form through holes in the molded part. These are separated from the machining of the core and cavity to simplify the mold build. Core pins also tend to wear, bend, and break over time, so making them replaceable makes good sense. The **Core** feature helps automate the creation of this additional tooling.



Analyzing the Model

Undercut Analysis identifies faces that form undercuts on the molded parts by classifying and color-coding. To determine which faces are undercut, the analysis looks down the pull direction from both directions and determines which faces are not visible.

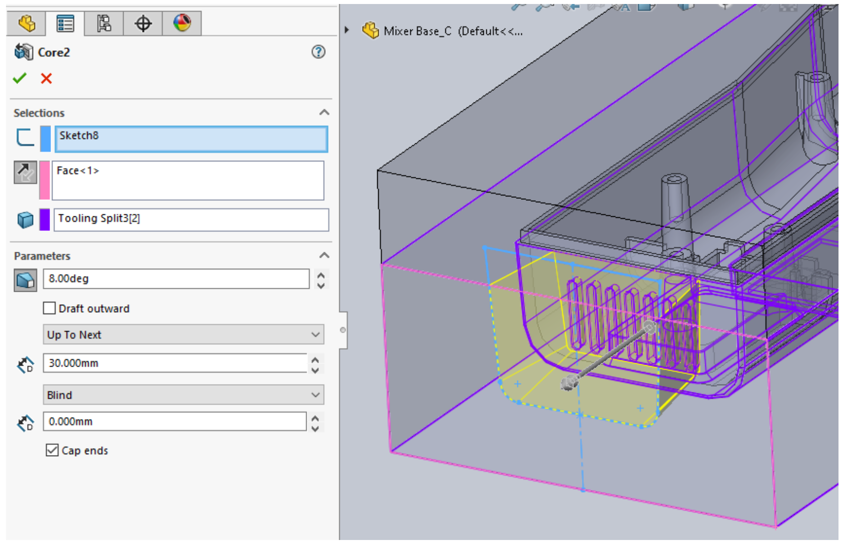


Red areas indicate trapped faces that will need to be pulled from a different direction using side pull cores.

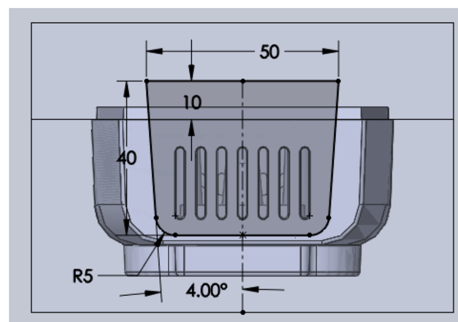
Side Cores

The side core is a piece of tooling that slides out of the mold perpendicular to the direction that the part is ejected from the mold.

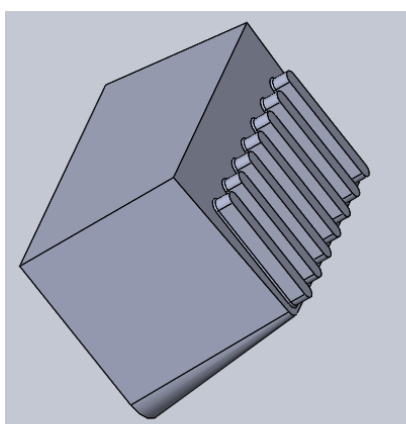
The **Core** command creates side cores or other additional tooling pieces based on an active sketch in the geometry of existing tooling blocks.



To create a **Core,** first create a sketch around the area that requires new tooling. The sketch profile can be designed parallel or perpendicular to the direction in which the core travels away from the plastic part.

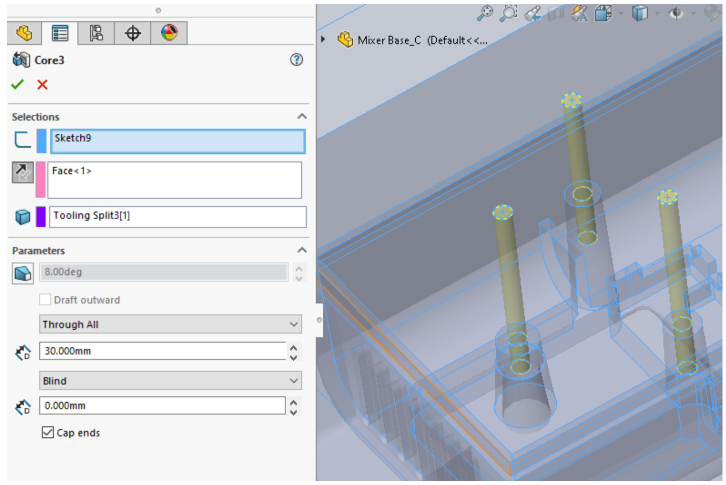


The **Core** feature then uses the sketch profile to extract a new solid body from the core or cavity tooling.



Core Pins

The **Core** command can also be used to separate the core pin molding areas from the tooling. Core pins are created to form detailed areas in plastic parts that tend to wear faster than the surrounding tooling.



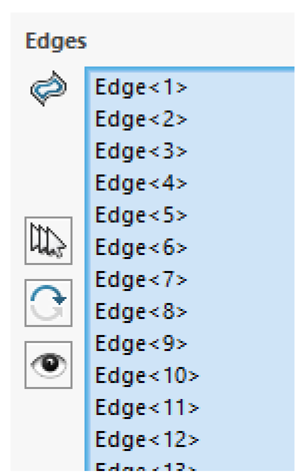
Manual Selection Techniques

With complex molded parts that require multiple directions of pull and pieces of tooling, features such as parting lines and shut-off surfaces may not be able to automatically recognize all edges for selection. Additionally, the automated selection may need to be manually modified to get the desired result. When these situations occur, manual selection techniques can be used.

Manual selection techniques include:

- Using the selection tools in the PropertyManager.
- Directly selecting using the mouse.
- Using the system selection commands such as select tangency, select loop, or propagate.

Selection Tools



Add the current edge to the selection (keyboard shortcut = "**y**").

Flip to an alternate edge (keyboard shortcut = "**n**").

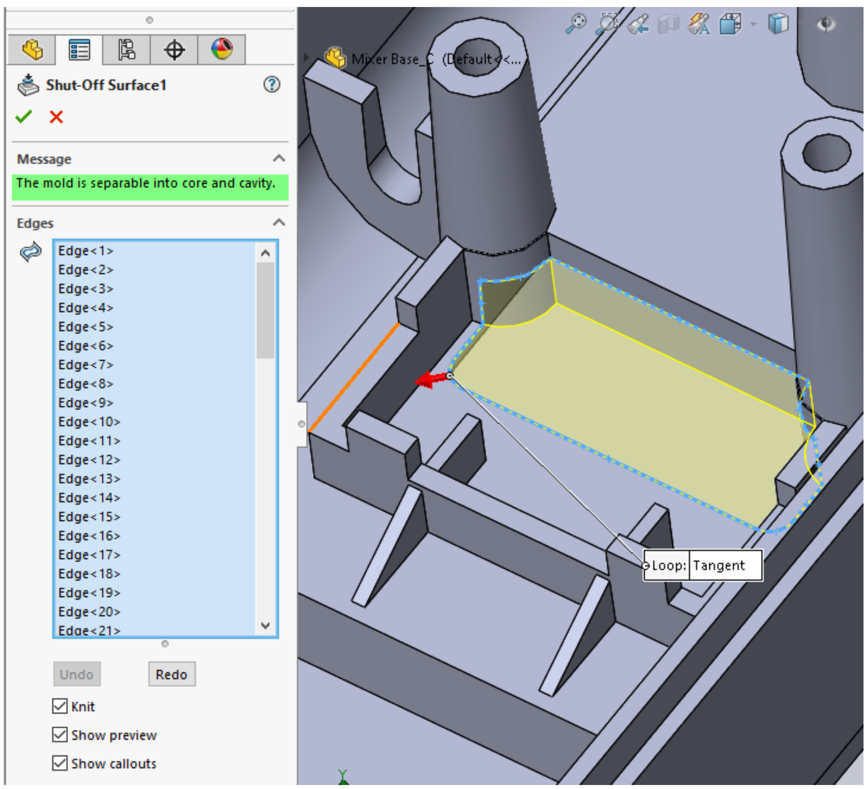
Zoom the display to the current edge.

Modifying Shut-off Surfaces

The **Shut-off Surfaces** command will attempt to automate creating surfaces to fill holes in the core and cavity based on the previously created parting line feature, the draft angle transitions, and open loops or holes recognized by the system.

Options for removing the automatic selections are as follows:

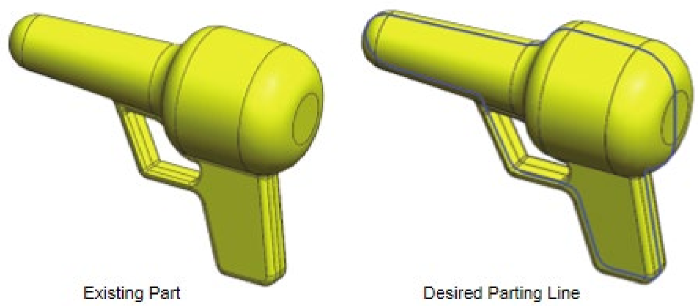
- Right-click in the selection list and click **Clear Selections** to remove all selections.
- Left-clicking a selected edge will toggle off the selection.
- Right-click individual loop flay callouts and click **Delete.**
- Right-click individual loop edges and click **Deselect Loop**.



Lesson 4: Advanced Parting Line Options

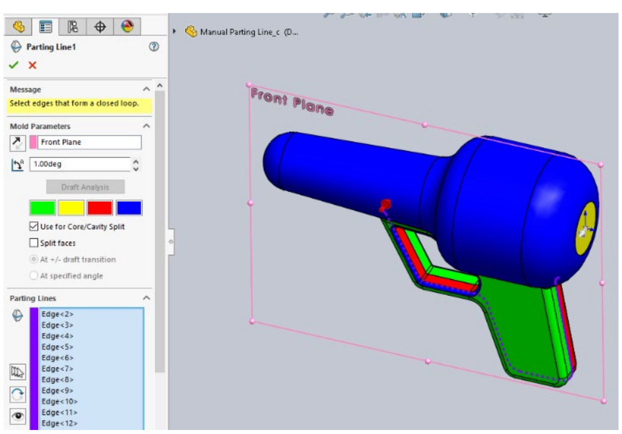
Manual Parting Line

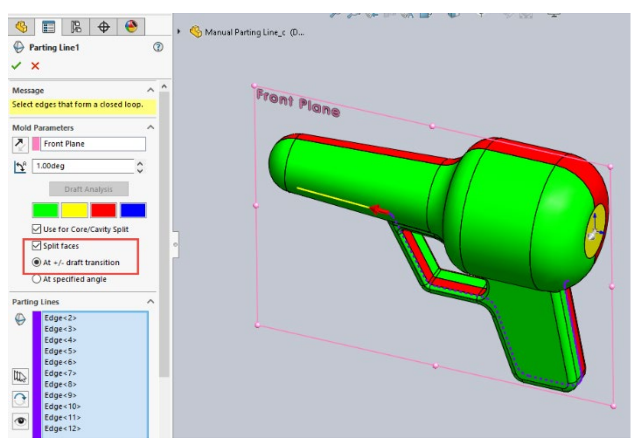
Some part designs do not have existing edges at the desired parting line location. This is common with cylindrical shapes. In these cases, the **Parting Line** command is not able to automatically make the appropriate selections to complete the parting line. Advanced options within the Parting Line PropertyManager can be used to manually make selections and define new edges.



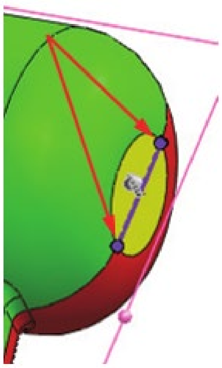
Using Split Faces

The **Split Faces** option within the **Parting Line** command can be used to automatically split straddle faces. Straddle faces are the faces that straddle the area of the desired parting line. Splitting these faces creates new edges which can be selected to establish the parting lines.





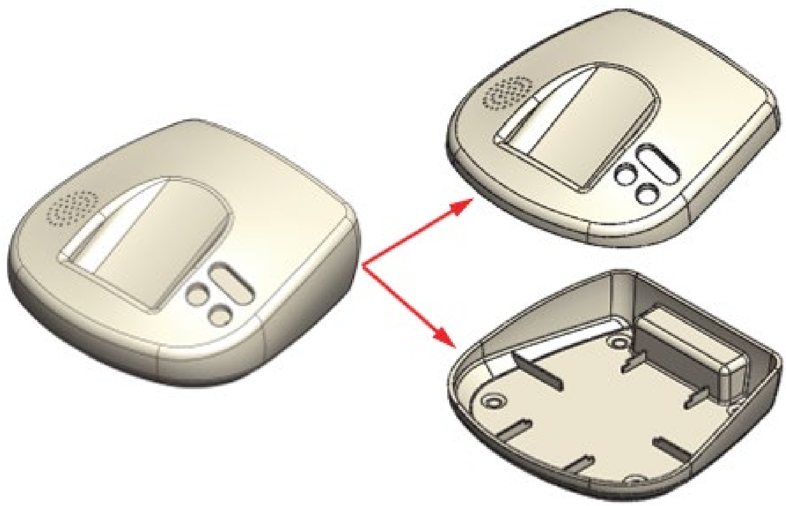
Using Entities to Split

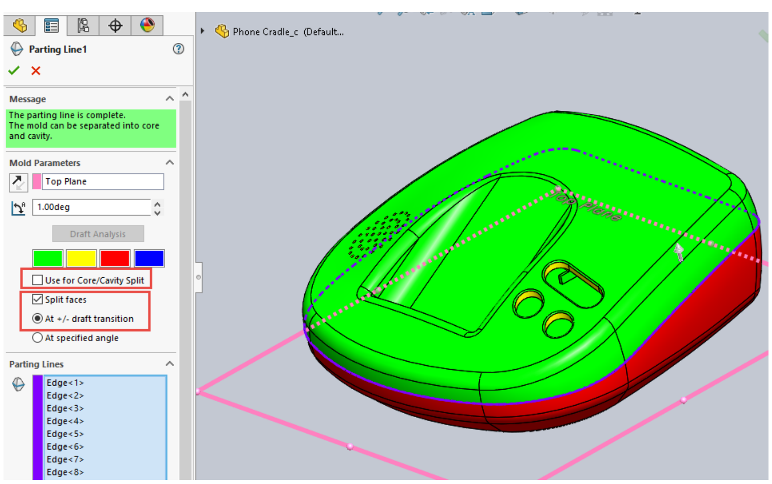


The **Entities to Split** selection pane in the **Parting Line** command is used to manually define a split line across a face. This option may need to be used for flat faces that span the parting line. Since these faces have no angle and there is no transition from positive to negative draft, the split faces option cannot automatically define the split line. To define the split line for entities to split, vertices or an existing sketch segment can be selected.

Splitting a Part

The functions in the **Parting Line** command can be useful for more than just establishing the parting line. They can also be a valuable tool for discovering the proper location to split a part into multiple bodies.



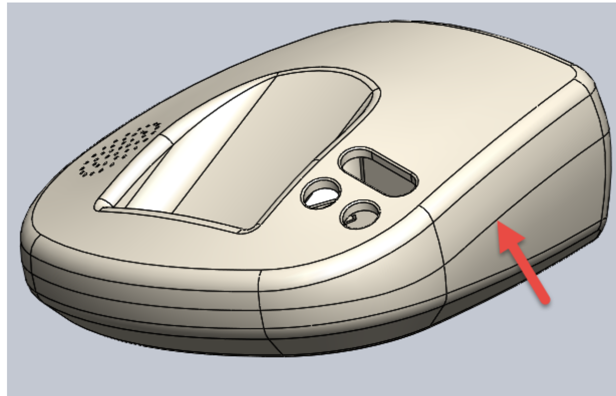


Parting Line/Split Faces: Use the **Parting Line** command with the **Split Faces** option to find the transition between positive and negative draft.

Deselecting the **Use for Core/Cavity Split** will keep the system from creating the Mold tools surface folders.

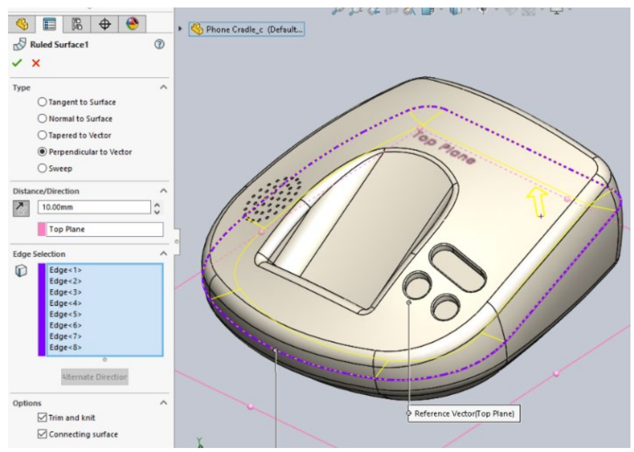
This will create a split line around the part that can be used to create a surface to use to cut the single body into two.

The **Core** command creates side cores or other additional tooling pieces based on an active sketch in geometry of existing tooling blocks.



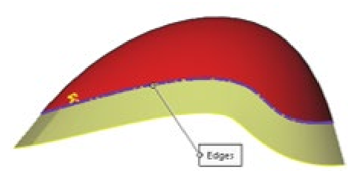
The **Ruled Surface** is the most widely used surface in mold making and is often used for model repair, interlocks, manual shut-off surfaces, and manual parting surfaces.

The **Ruled Surface** command is used to create surfaces at selected edges of a model. The ruled surface can be related to the existing geometry in several ways using the options within the command.



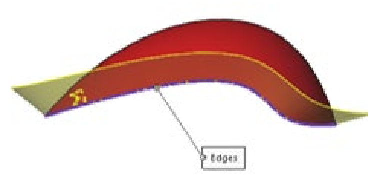
Tangent to Surface
The ruled surface is tangent to a surface at the selected edge.

The **Alternate Face** option can be selected to determine which face the surface is tangent to.

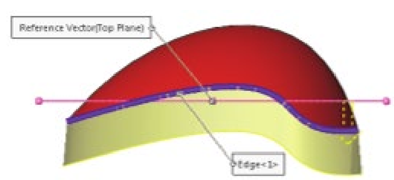


Normal to Surface
The ruled surface is normal to a surface at the selected edge.

The **Alternate Face** option can be selected to determine which face the surface is normal to.



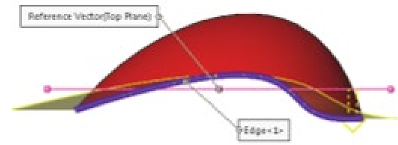
Tapered to Vector
The ruled surface is created at a specified angle to a direction vector.



The **Alternate Side** option can be selected to determine which direction the taper is applied.

Perpendicular to Vector

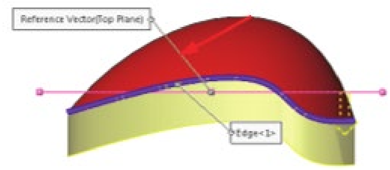
The ruled surface is perpendicular to a specified vector.



The **Alternate Direction** option can be selected to determine which direction the surface is created.

Sweep

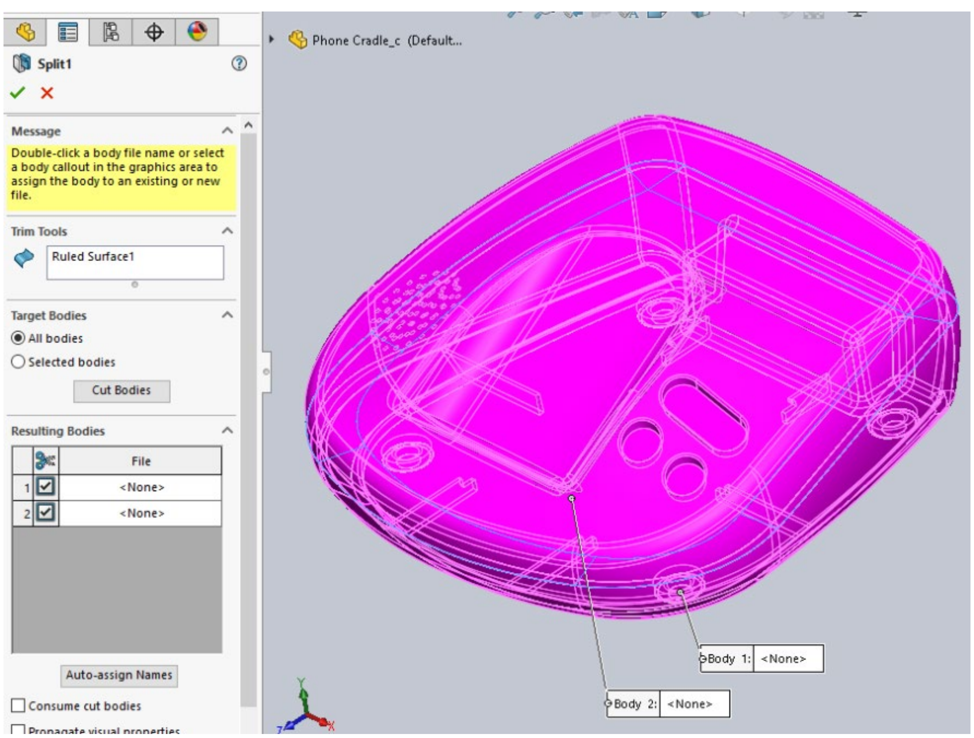
The ruled surface is built by creating a swept surface using the selected edges as a path.



Split Command

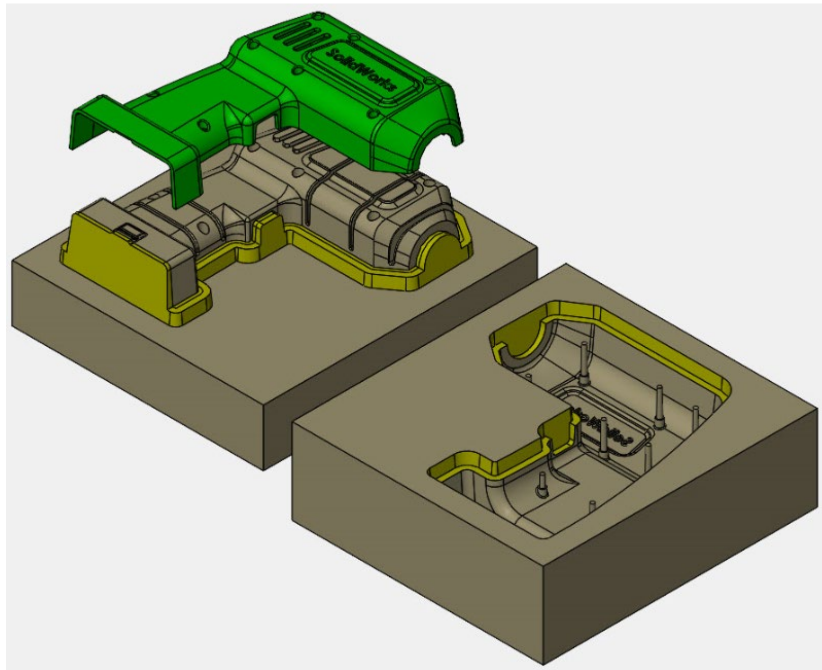
The **Split** command is used to split a part into multiple bodies.

The **Trimming Surfaces** can be Reference Planes, Planar Model Faces, Sketches, and Reference Surfaces.



Lesson 5: Custom Surfaces

Mold makers use surfaces to repair imported geometry and to manually create the shut-off surfaces of the mold tooling. SOLIDWORKS Mold Tools automates many surface modeling operations when possible. There are times when these automated tools do not create the desired surfaces and some customization is needed.

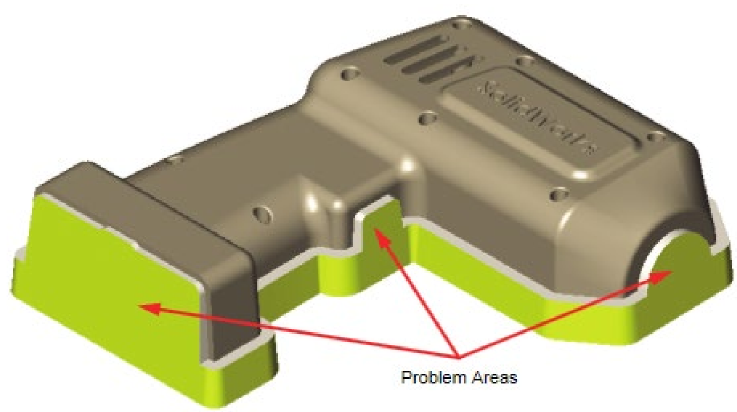


Some examples are:

- A complex parting line is not producing a satisfactory parting surface.
- A Complex parting surface is not able to generate an interlock surface.
- Shut-off surfaces are too complex.

Manual Interlock Surfaces

When a parting surface has sudden changes in direction, the automated tool to create interlock surfaces may fail. The interlock surfaces can be created manually using the **Ruled Surface** command.

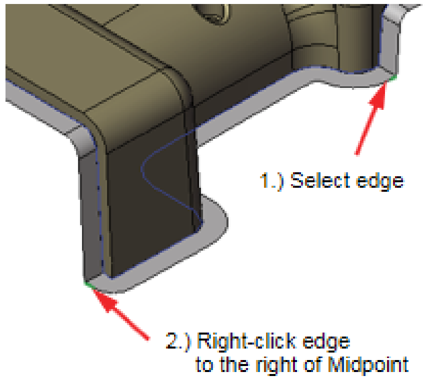


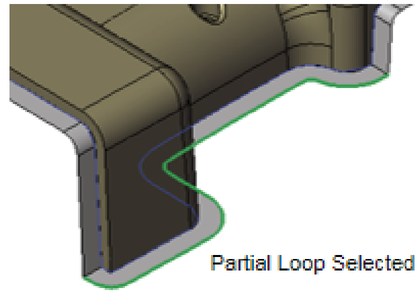
Edge Selection Tools

The **Select Partial Loop** selection tool assists in selecting a group of connected edges. This will save time by eliminating the need to individually select multiple edges.

To select a partial loop:

1. Select an edge at the end of a chain of edges.
2. Right-click the edge at the other end of the chain.
3. Click **Select Partial Loop** from the shortcut menu.



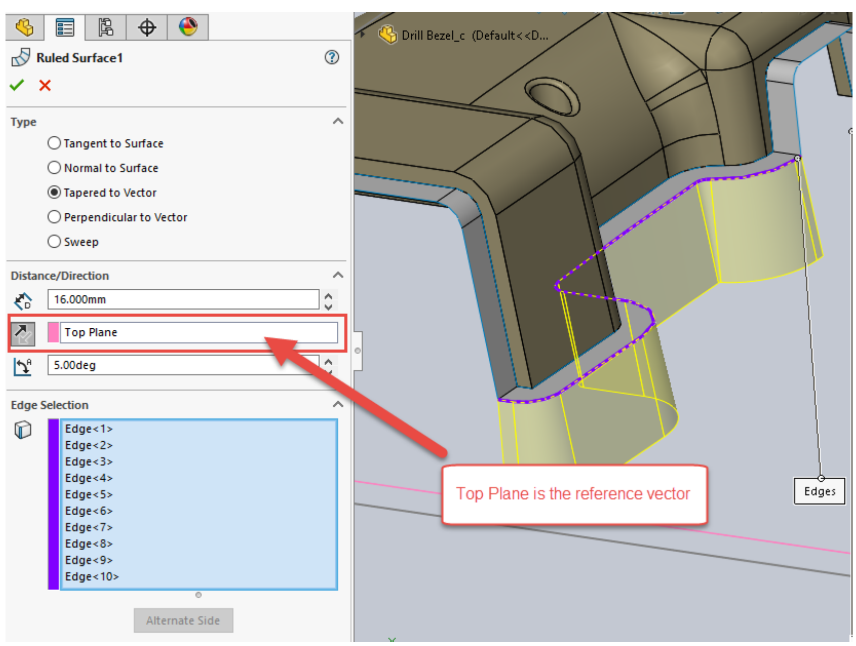


The chain direction is based on where you select the second edge:

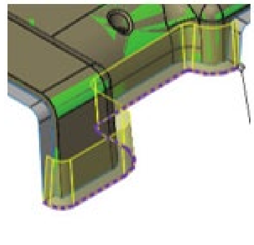
- Left of the midpoint – chain moves left.
- Right of the midpoint – chain moves right.

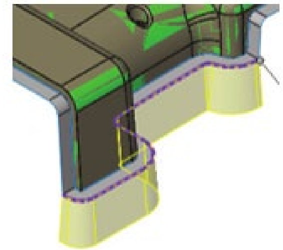
Ruled Surface Direction

The direction and taper of a ruled surface are controlled by the **Reference Vector** direction and the "side" the angle is measured from. These settings are controlled from the Ruled Surface PropertyManager.

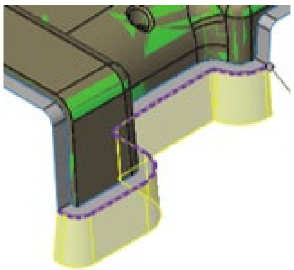


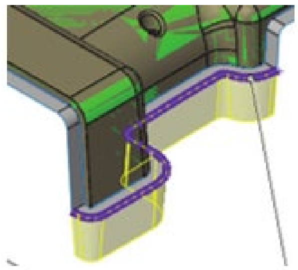
Use **Reverse Direction** to control which direction the ruled surface is extend from the selected edge.

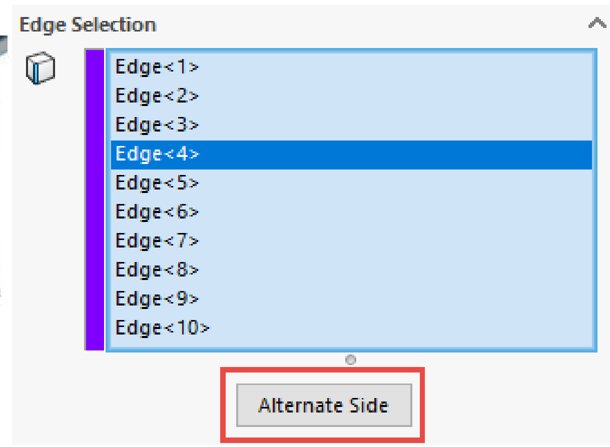




The **Alternate Side** option can control the direction of the taper or which face the ruled surface is tangent or normal to.

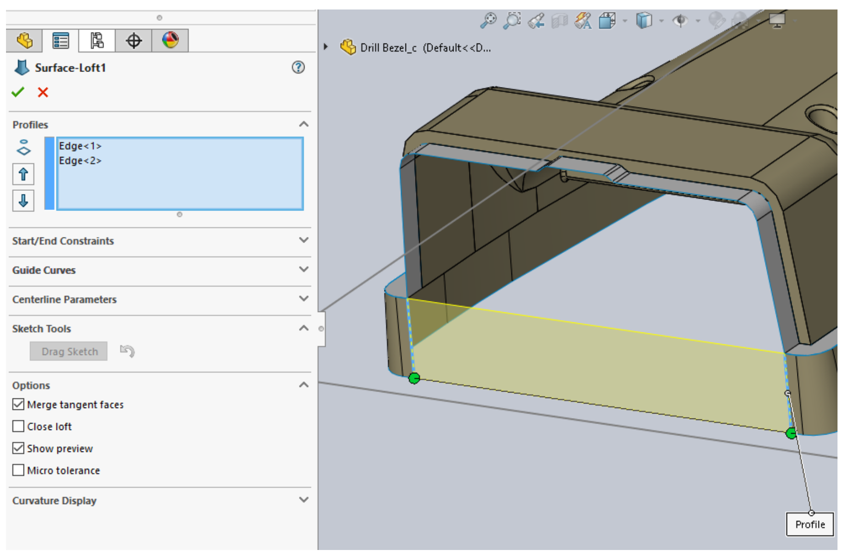






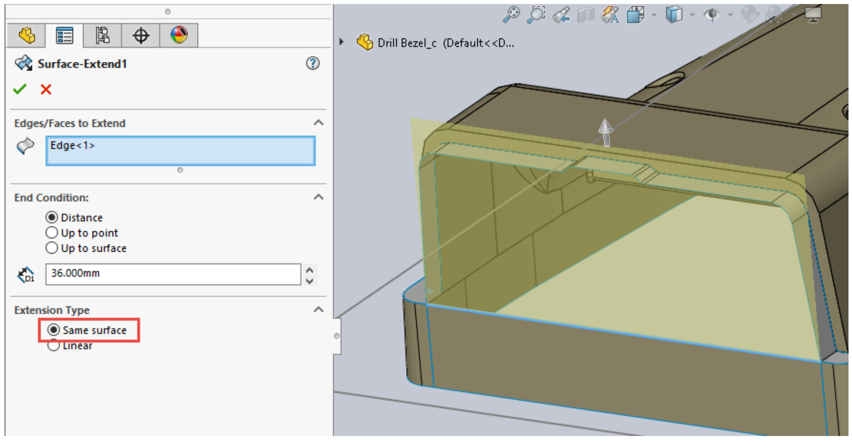
Lofted Surface

The **Lofted Surface** is used to bridge gaps between other surface edges. Selecting the edges near corresponding endpoints will prevent the loft from twisting.



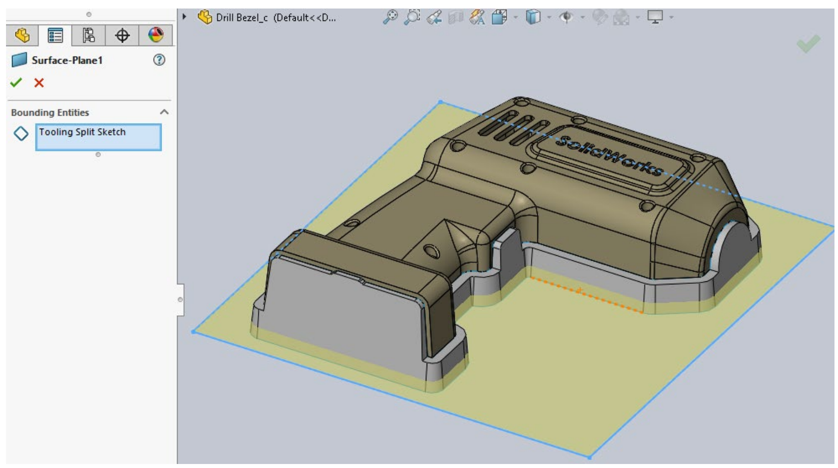
Extend Surface

The **Extend Surface** is used to extend an edge of an existing surface. Selecting the Same Surface option will extend the surface along the geometry of the existing surface.

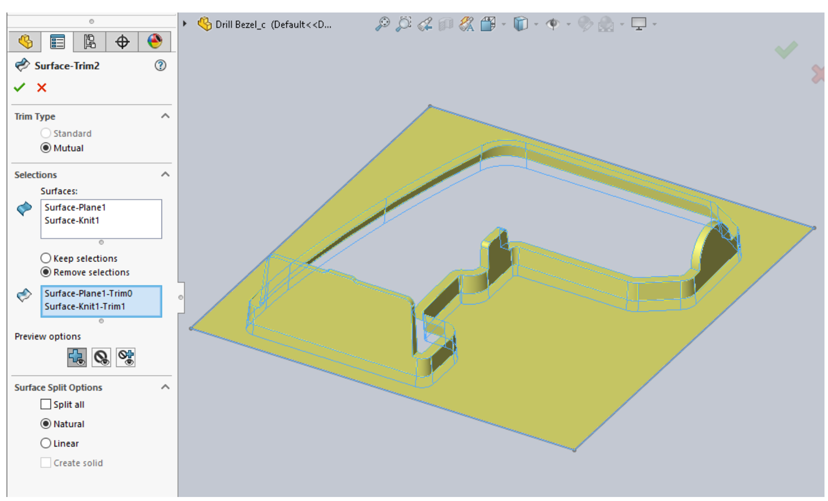


Manually Creating a Parting Surface

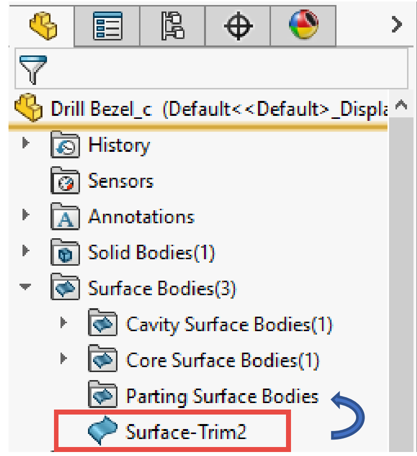
A simple parting surface is typically planar. Using the **Planar Surface** feature is an easy step in manually creating a parting surface.



Use **Trim Surface** to trim the planar surface and the ruled surface overlaps. This creates the single parting surface body.



Organizing Surfaces



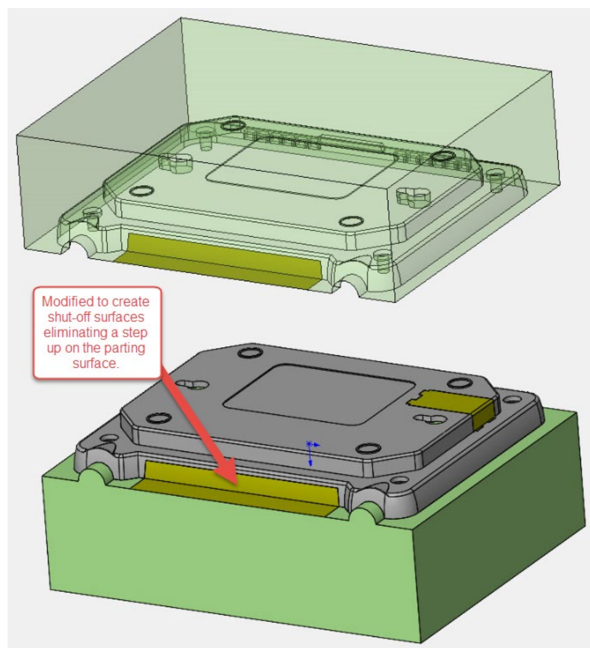
When manually creating surfaces for the Tooling Split command, the surfaces need to be organized into the appropriate mold folder.

Any surface that was manually created or modified will be in the **Surface Bodies** folder below the **Parting Surface Bodies** folder. They can be dragged and dropped into the corresponding folder as needed.

Once complete, this trimmed surface is moved into the **Parting Surface Bodies** folder.

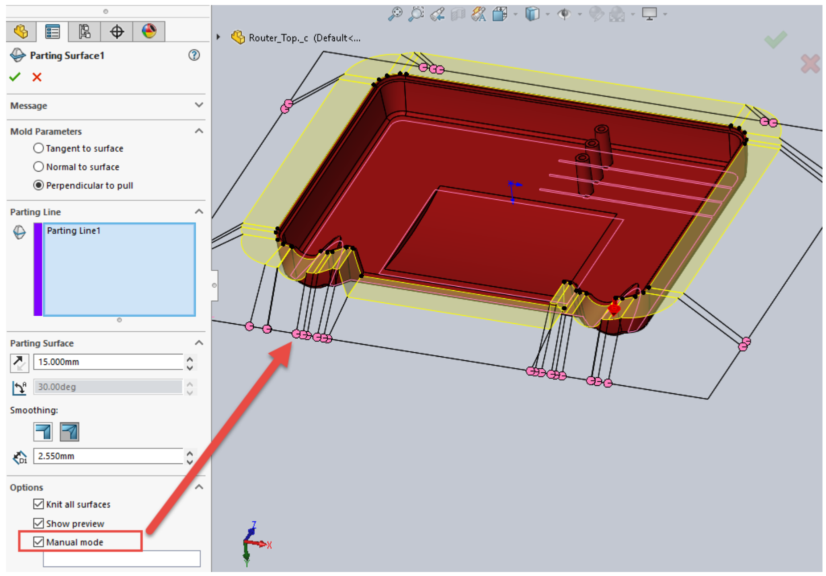
Modifying Parting Surfaces

There are times when the parting surfaces that are created can be simplified to make the mold cavities easier to manufacture. The parting surfaces can be modified using surfacing tools.



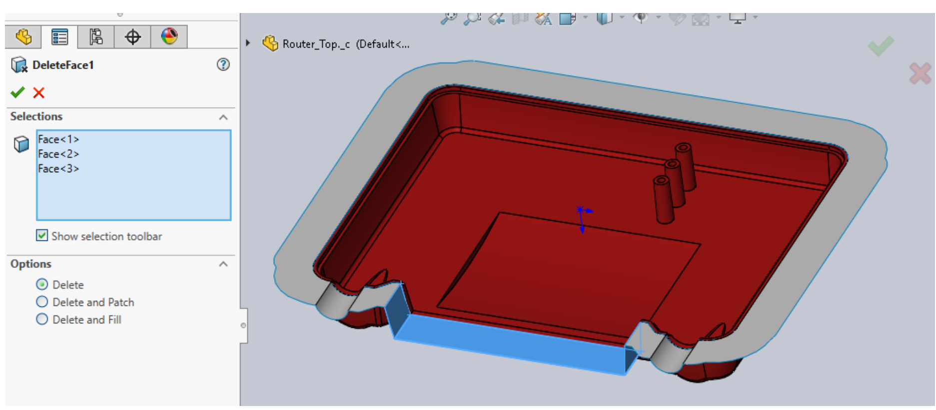
Parting Surface – Manual Mode

The **Parting Surface** command has a **Manual Mode** option that provides nodes that can be manipulated to align the faces of the parting surface.



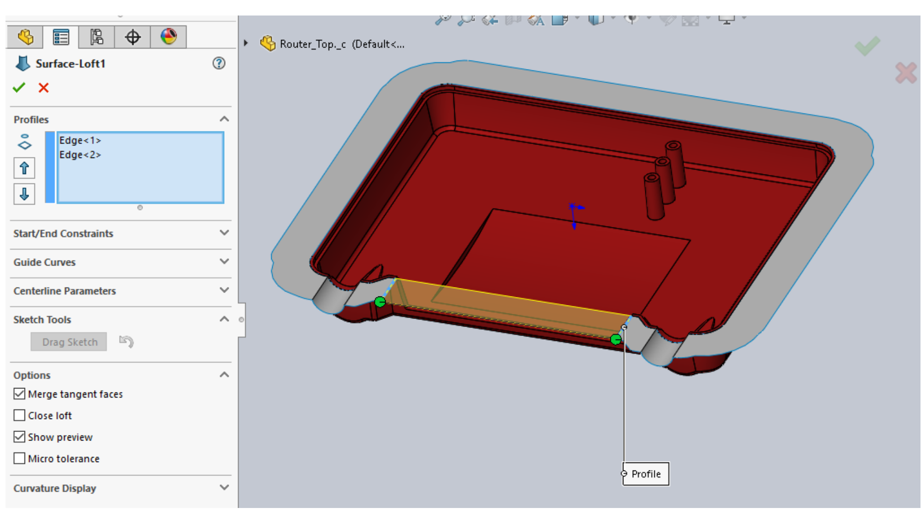
Modify a Parting Surface with Delete Face

After a **Parting Surface** is complete, it can be modified by removing the unwanted surfaces using the **Delete Face** Command.

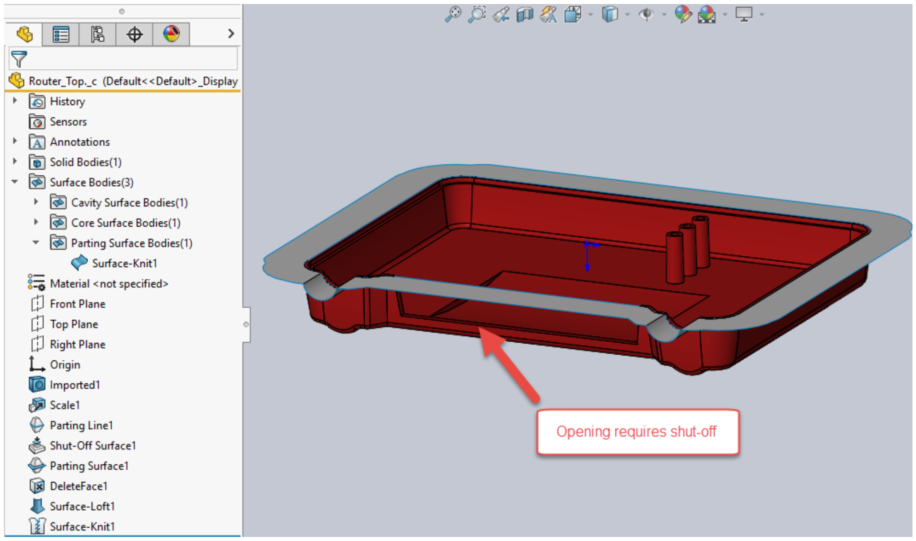


Fill in surfaces using **Lofted Surfaces.**

This leaves an easier-to-machine parting surface. The resulting surfaces are **Knit** together to form one **Parting Surface** body.

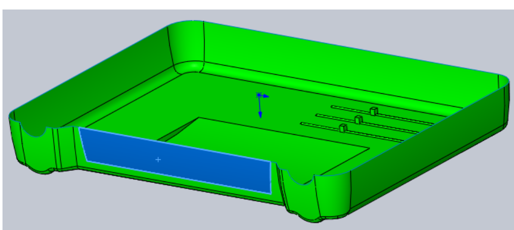


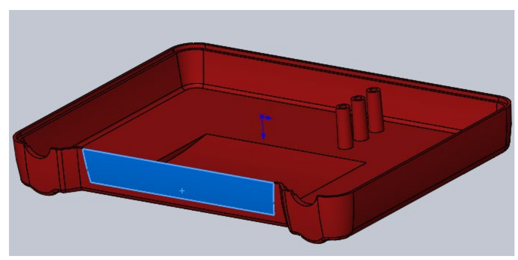
The modified **Parting Surface** left an opening that needs to be shut-off. The shut-off surfaces are created manually using the **Lofted Surface**.



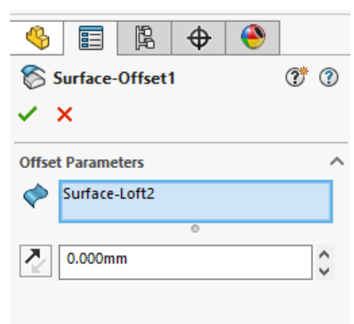
Creating Manual Shut-off Surfaces

Shut-off Surfaces define faces in both the core and cavity, so for the tooling split to use them, a copy must be available for each side of the mold tooling.





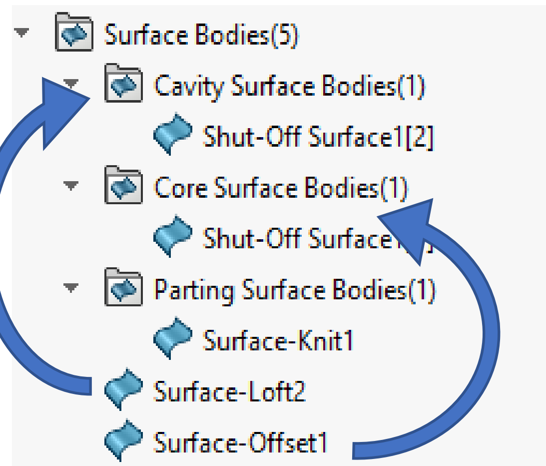
The Lofted Surface can be copied using the **Offset Surface** command with a “0” offset value.



Organizing Manual Shut-off Surfaces

With all the Shut-off Surfaces created and copied, they will need to be put into the proper folders so the **Tooling Split** will work.

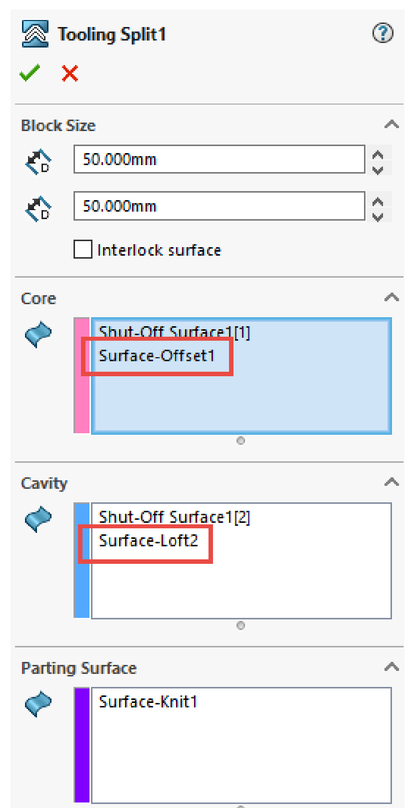
The **Surface-Loft2** body can be dragged and dropped into the **Cavity Surface Bodies** Folder.
The **Surface-Offset1** body can be dropped into the **Core Surface Bodies** folder.



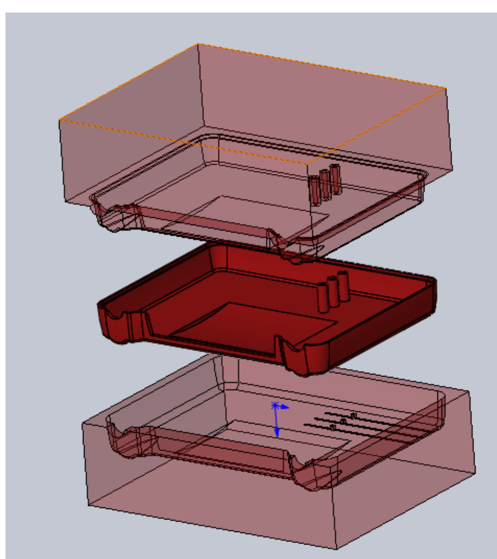
It does not matter which goes into which folder just as long as each folder has a copy inside.

Tooling Split

After moving the Shut-off Surface bodies into their respective folders, the **Tooling Split** command is used to create the mold core and cavity inserts.

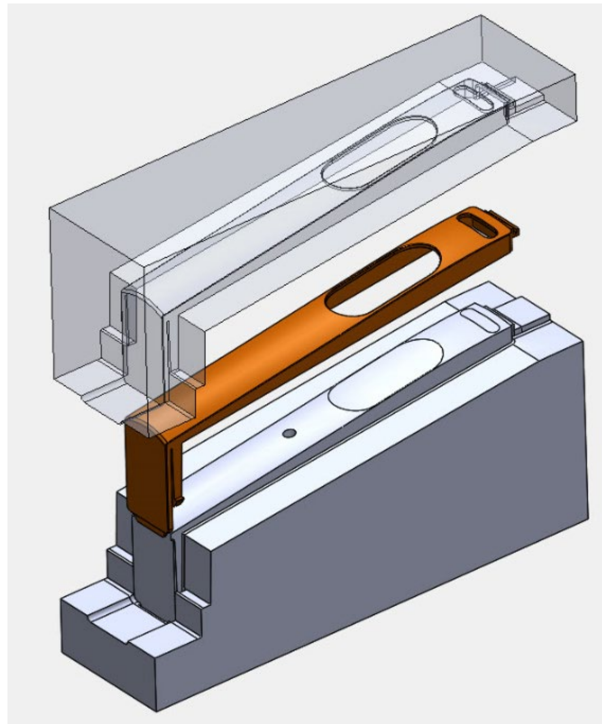


Check the Core, Cavity, and Parting Surface input panes to see that the manually created surfaces and the copy are in the folders.



Lesson 6: Advanced Surfacing

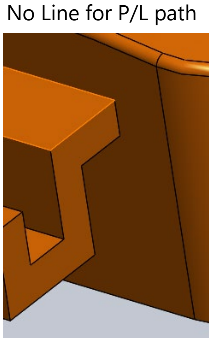
There are times when the part model and mold parting line are so complex that surface modeling techniques are required to fully create the mold design. Complex shut-off surfaces, custom side cores, and complicated parting surfaces can all be created with some advanced surfacing tools.



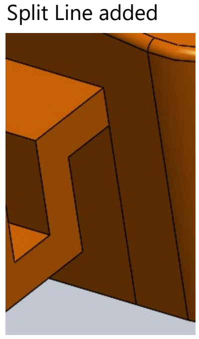
Using a Split Line in the Parting Line

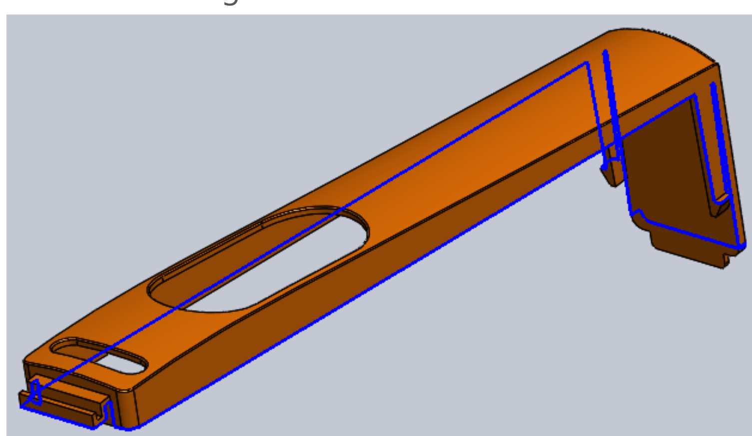
A complex parting line may need to have its path altered to be able to create a proper tooling split.
The **Split Line** command is used to place a line on a part that can be used as part of the Parting Line selection.

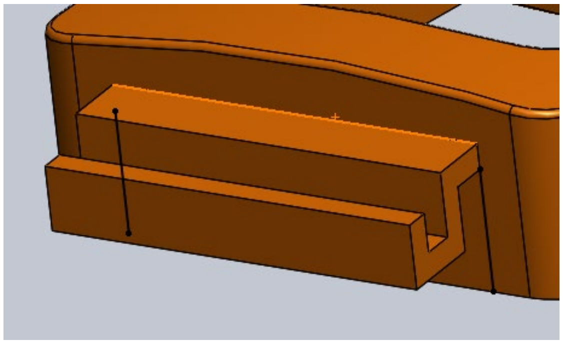
No Line for P/L path



Split Line added

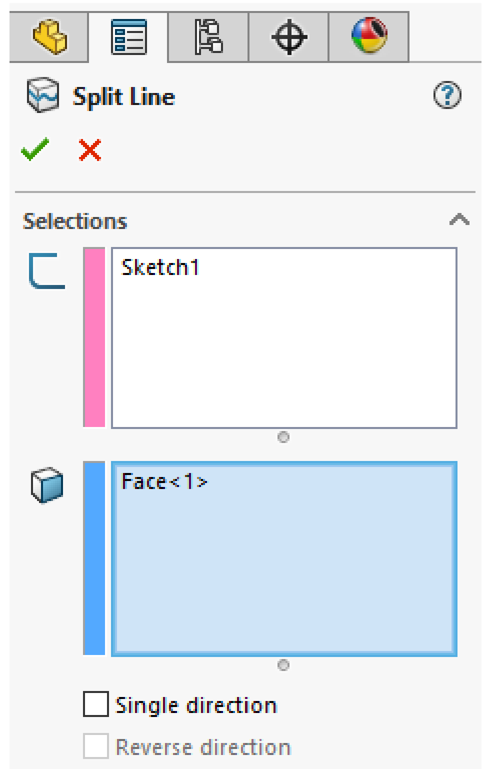






The **Split Line** command requires a sketch on the face that is being split.

A simple sketch line segment is placed where the split is needed.



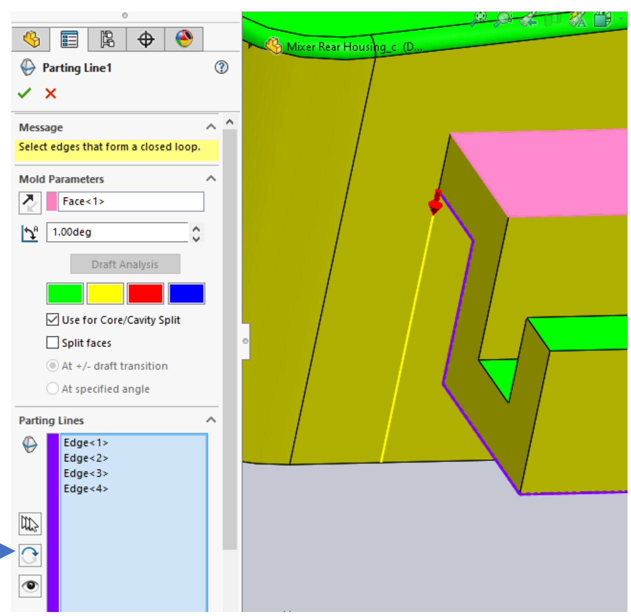
Both the Sketch and the Face that will be split are selected to create the **Split Line** feature.

The Sketch does not have to be on the Face, it can be on another plane and it will project to the selected faces.

Manual Selection of Complex Parting Line

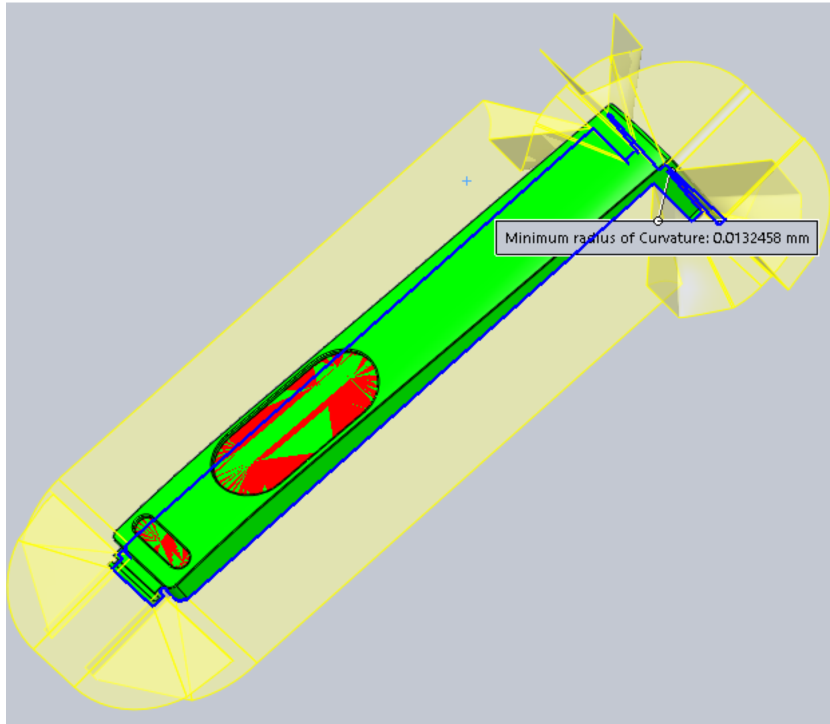
The **Parting Line** can be manually selected around the path on the part using the manual selection tools. The Split Line that was created is chosen as part of the parting line.

Remember that the "y" key can be used to accept the direction of the red arrow or the "n" key can be used to change the direction of the red arrow. This technique keeps you from having to go back and forth to the direction icons.

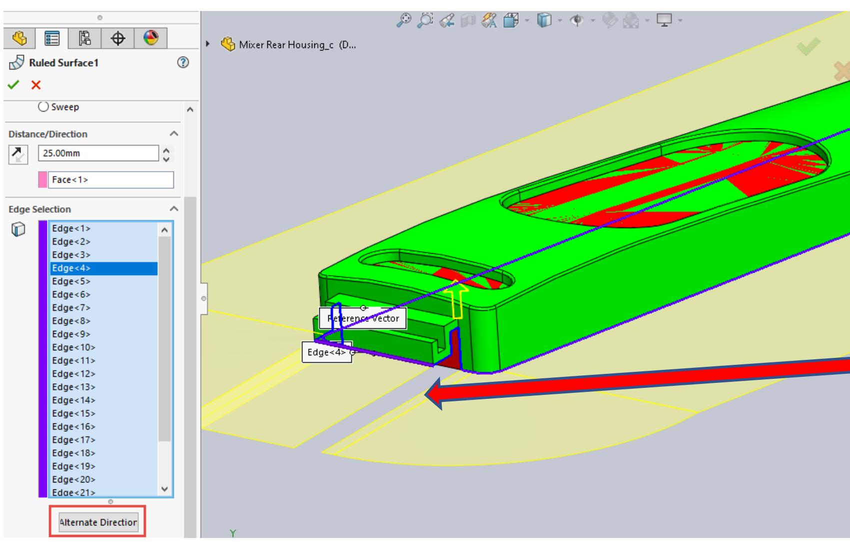


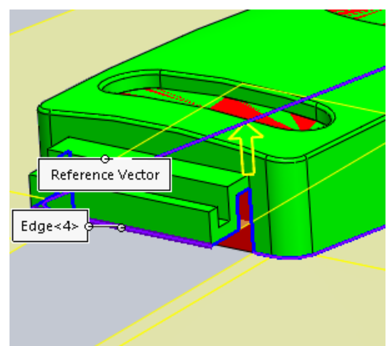
Manual Parting Surface

When the complexity of a part prevents many usable surfaces from being created, they can be made manually using surfacing tools. The **Ruled Surface** command is especially useful for building parting surfaces.



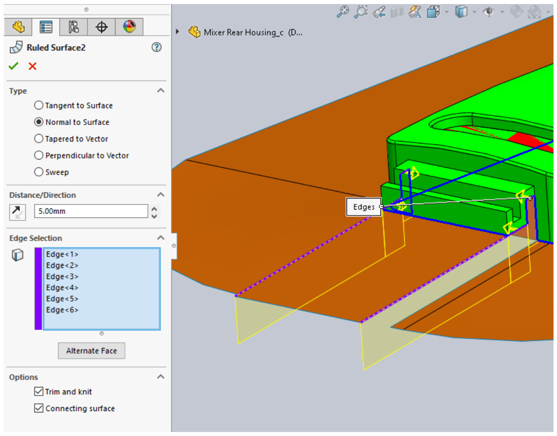
Multiple edges can be selected in one **Ruled Surface** feature. If the preview shows the surface going in the wrong direction when it is selected, use the Alternate Direction button to correct the direction.

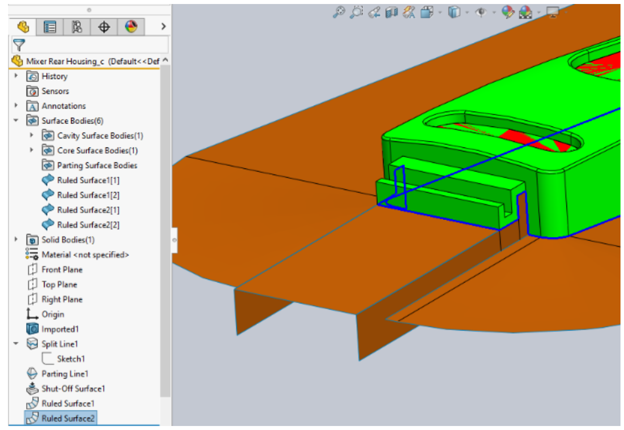




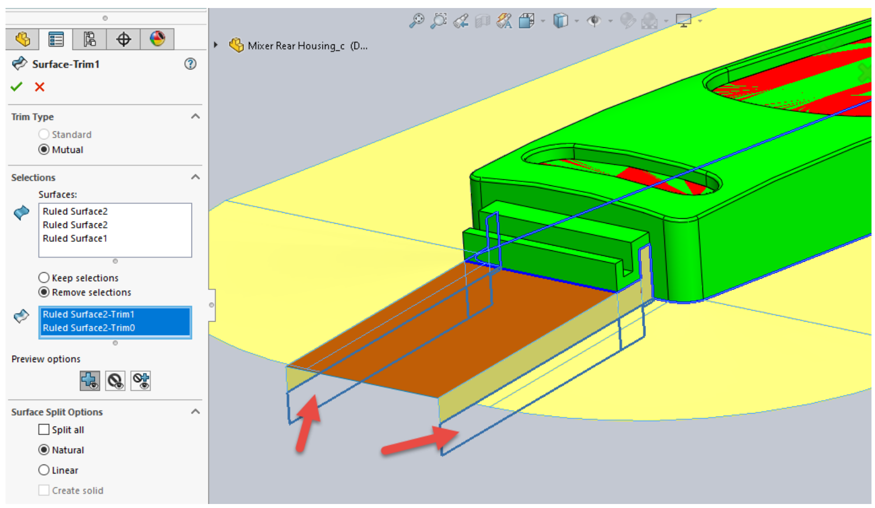
Additional Ruled Surfaces are created to close the remaining gaps.

When the manually created surfaces result in overlaps, such as these Ruled Surfaces, they can be accepted this way and modified using the **Trim Surface** tool.

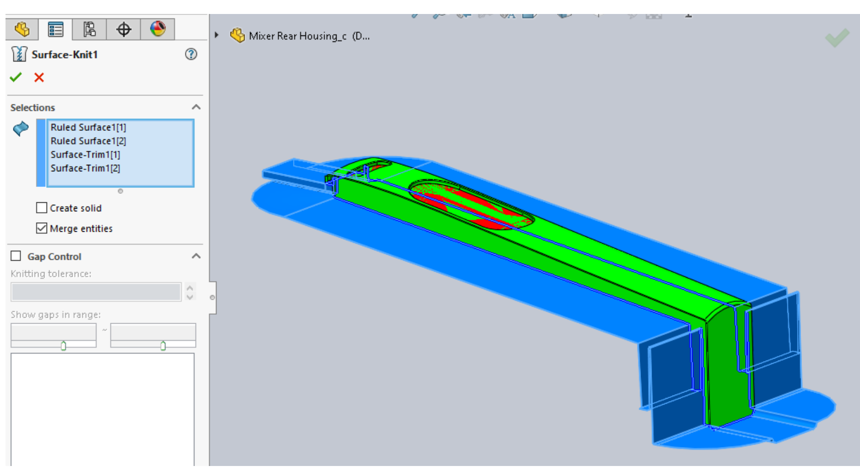


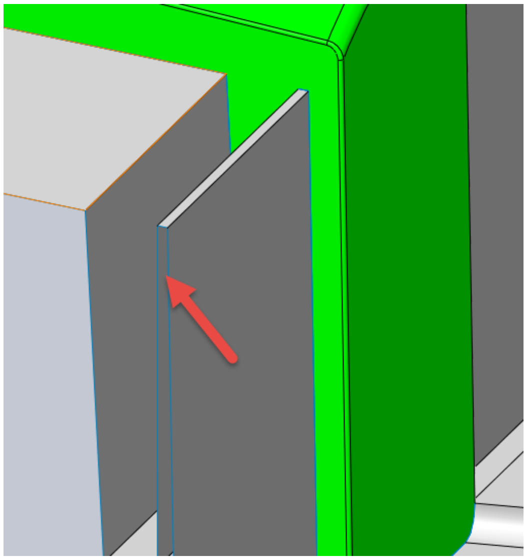


Using **Mutual Trim Surface** with the **Remove Selections** option allows the user to select all the surfaces that are intersecting and then select the portions to remove.

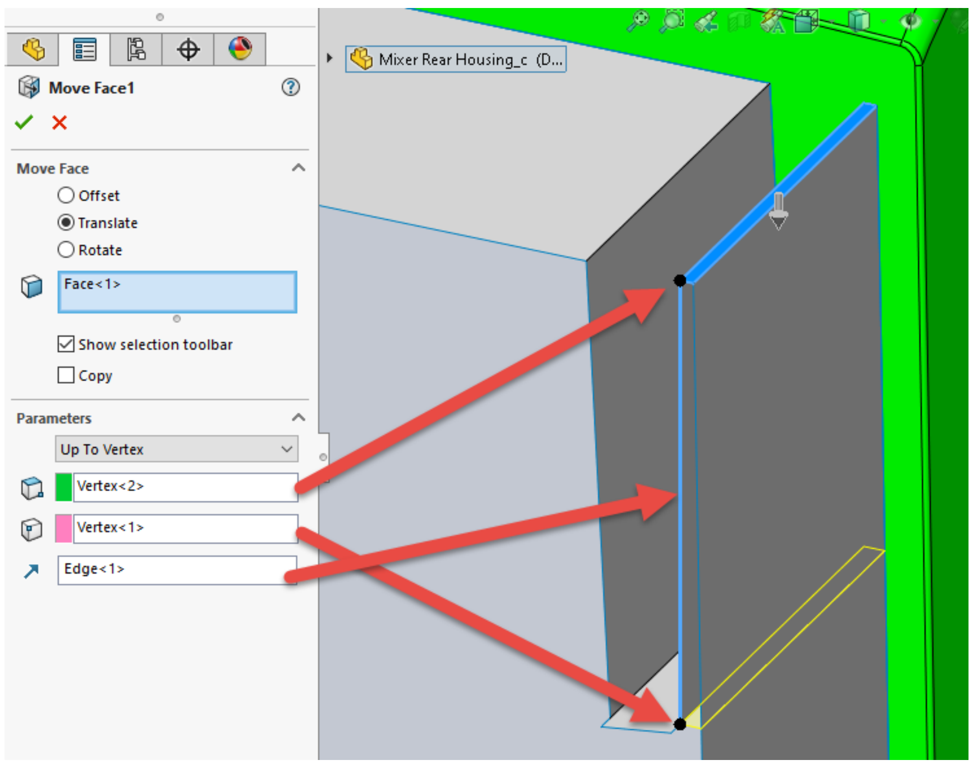


Use **Knit Surface** to combine all the resulting surfaces that represent the Parting Surface.





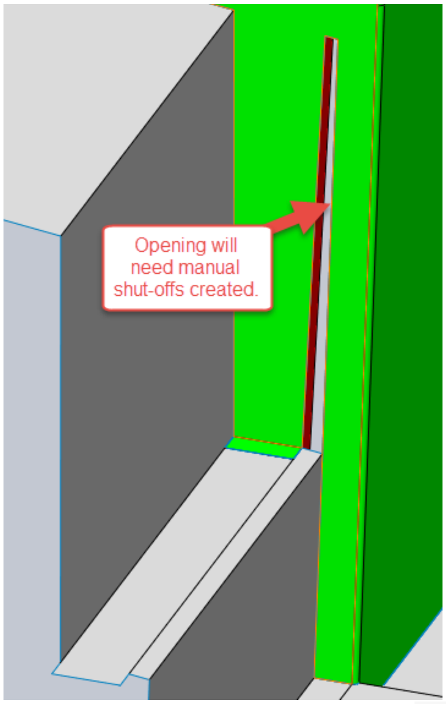
Even though the parting surface is complete, there are a couple of areas that can be simplified to make the mold easier to machine. The mold would have a very thin standing wall in this area that would be very difficult to machine and would also be susceptible to breaking.



Use **Move Face** to modify the two areas.

The **Translate** option is relocating the selected face.

Selecting the **Up to Vertex** parameter lets the user choose the vertex to move from and a vertex to move the face to. The direction selection provides a reference direction to move the face.



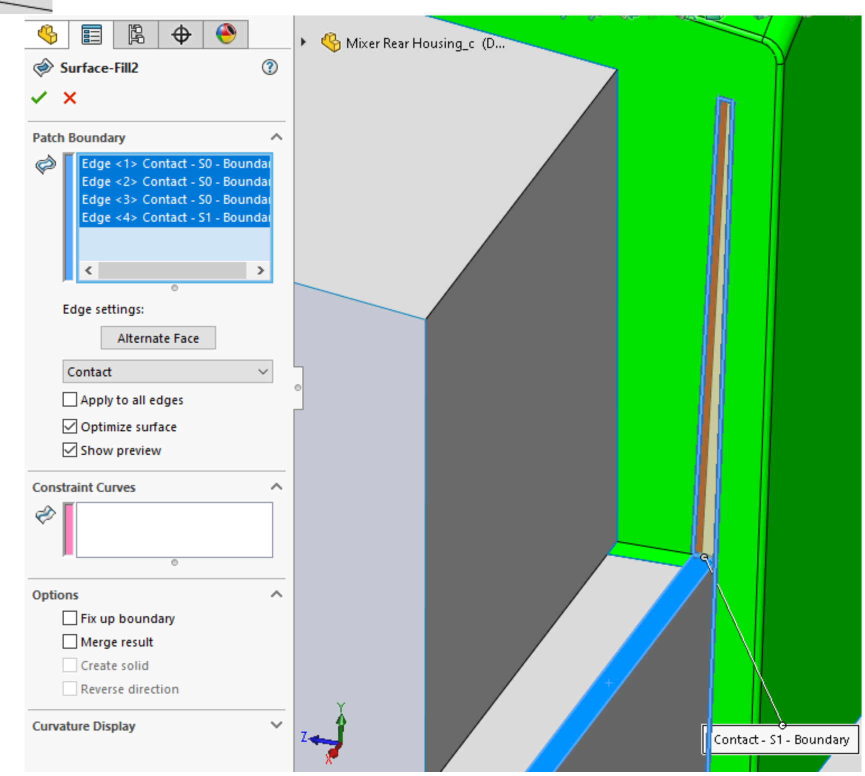
The result will move the face but there will be an opening the needs to have shut-offs created.

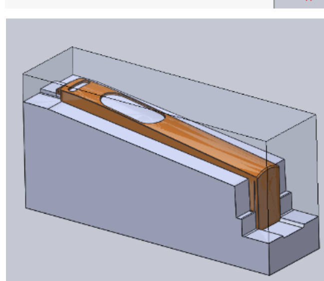
A **Filled Surface** will close in the gaps to manually create the shut-off. Selecting the four edges around the boundary of the opening will create the surface.

This was done to both sides of the part.

To complete the step, **Offset** the surfaces to make a copy and then place them in the proper mold folders.

The Tooling Split will then be able to complete.

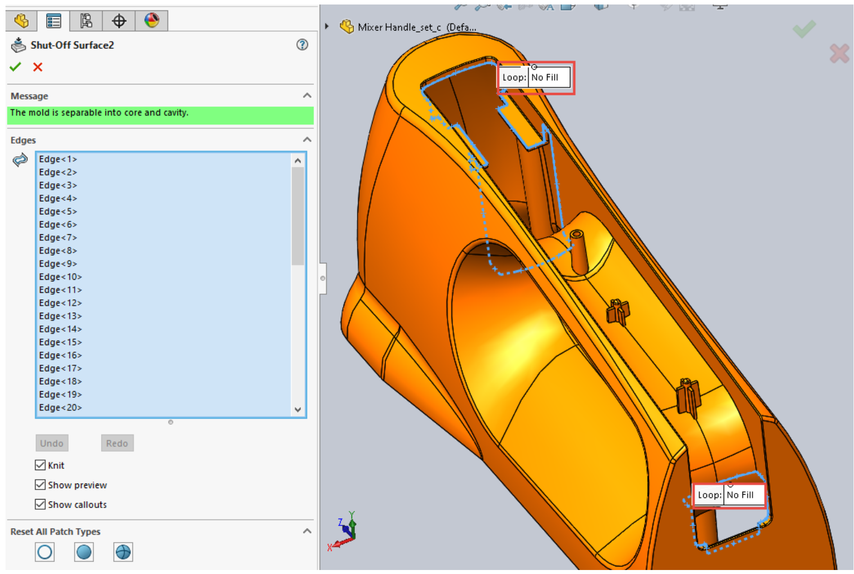




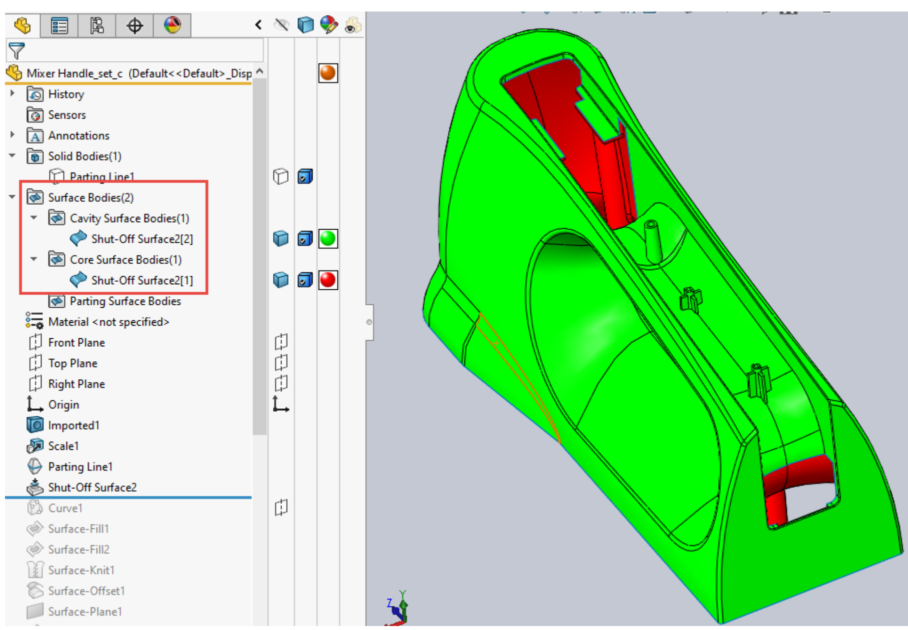
Manual Shut-off Surfaces

The **Shut-off Surfaces** command should be used to automate as many of the required surfaces as possible. Any shut-off surface the command can create automatically is one less that you must create manually. The automated selections will work well on planar openings normal to the direction of pull but may have less success with openings parallel to the direction of pull or openings with multiple edges at many angles.

The first step in creating manual shut-off surfaces is to select a loop within the **Shut-off Surface** command and set the fill type to **No Fill.**

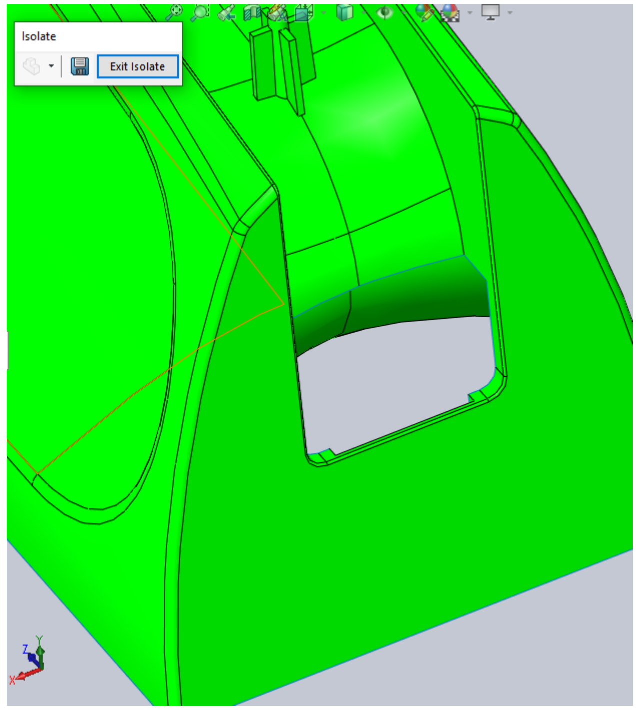


By defining the edges of a shutoff area and setting the patch type to **No Fill**, the core and cavity surface bodies can still be created. However, they will be open surfaces and will require surface features to close the no fill shut-off areas before they can be used successfully for the tooling split.



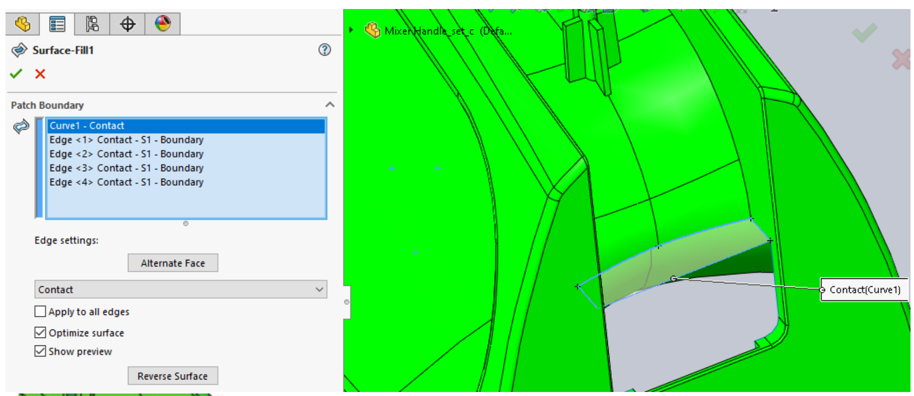
The core and cavity surface bodies are created. Manual surfaces will be created to close the openings.

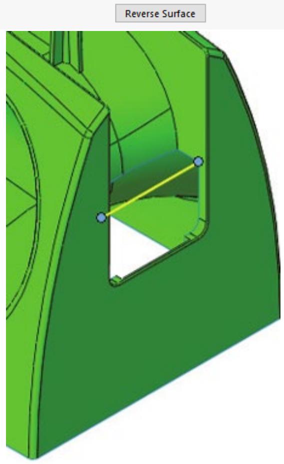
Use the **Isolate** command to make it easier to select the edges of the surfaces we need to use to fill the openings. Isolating the Cavity surface body will make edge selection easier since it hides the edges of the solid model and the Core surface body.

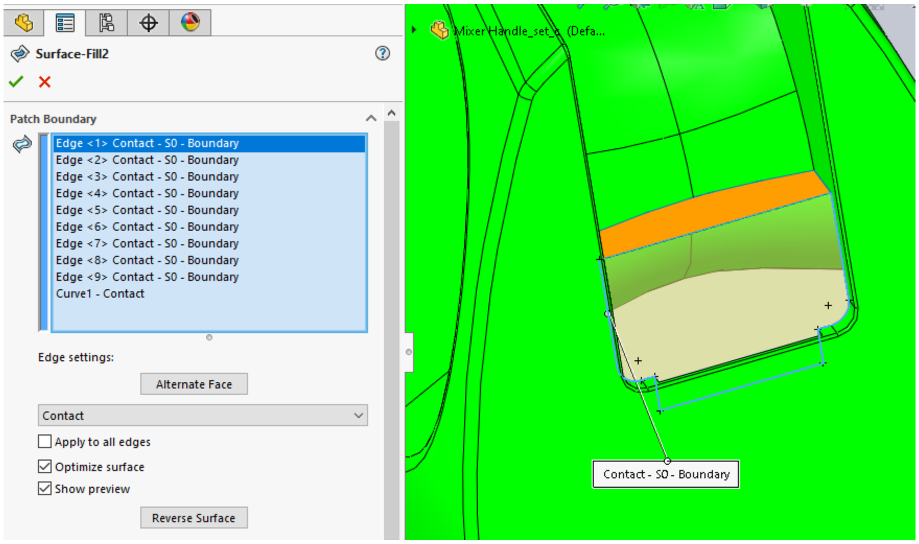


Planar Surface

A Planar Surface requires a set of close edges. Creating a **Curve Through Reference Points** is a good feature to use to close a loop of edges. A **3D Sketch** between endpoints also works well.

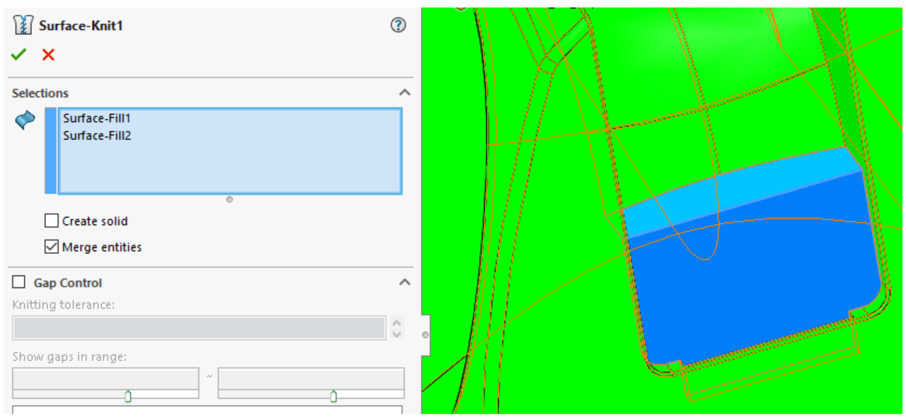






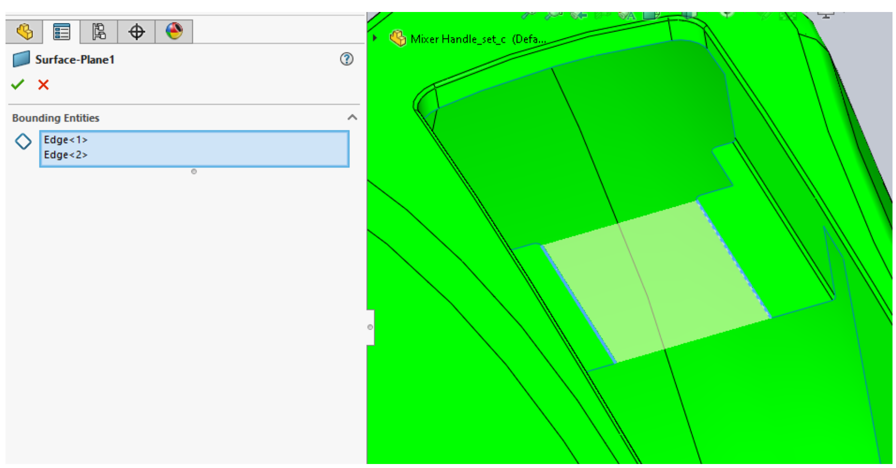
The remaining edges are used to create another Planar Surface to close the opening.

Use **Knit Surface** with the **Merge** entities option to create a single surface body.



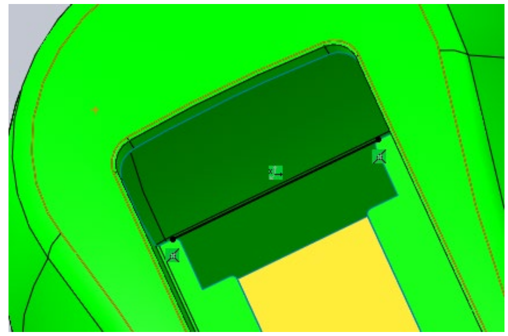
Since this is a shut-off surface, there will need to be a copy of this surface for the Core surface bodies folder. This would be a good time to use the **Offset Surface** command to copy it. Doing it now can prevent this step from being overlooked.

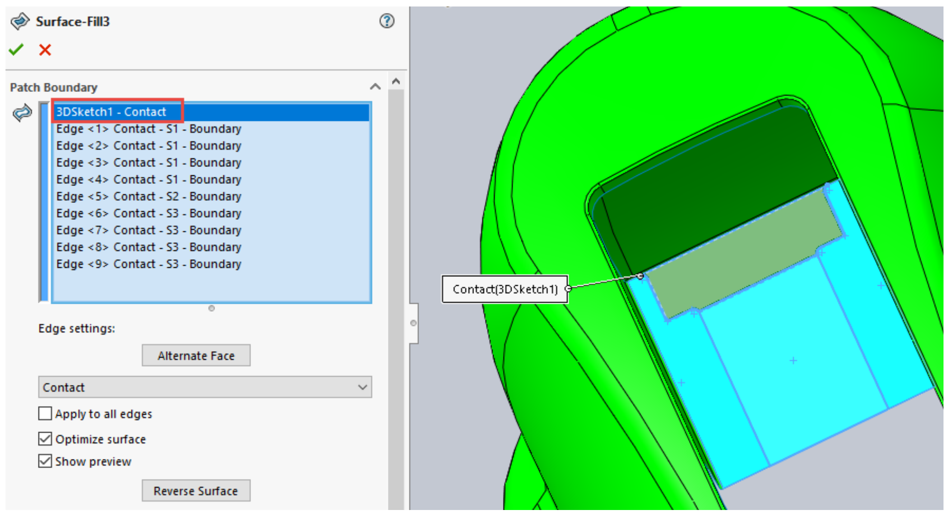
Another **Planar Surface** is used to start closing off the top opening. A **Planar Surface** can be created between two edges. A **Lofted Surface** would work in this instance as well.



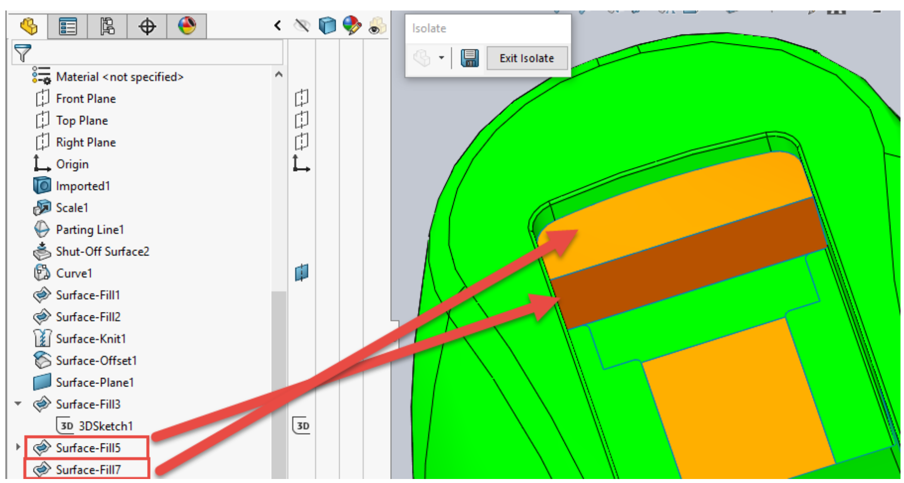
Filled Surfaces

To create a boundary for the next Filled Surface, a **3D Sketch** is used.



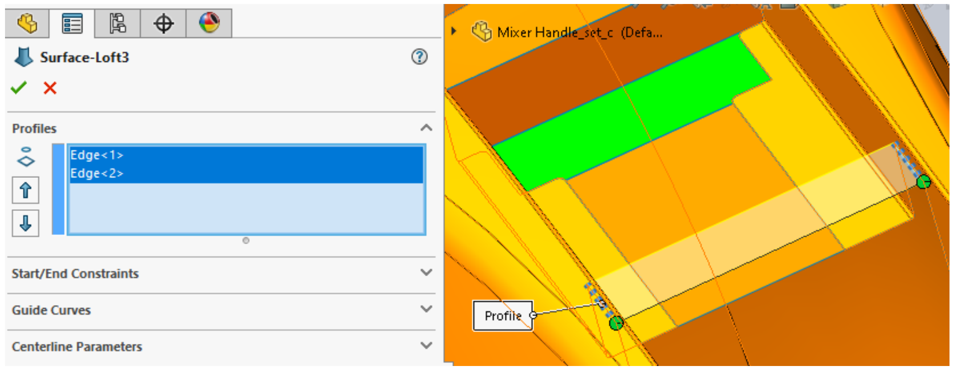


More Filled Surfaces are used.

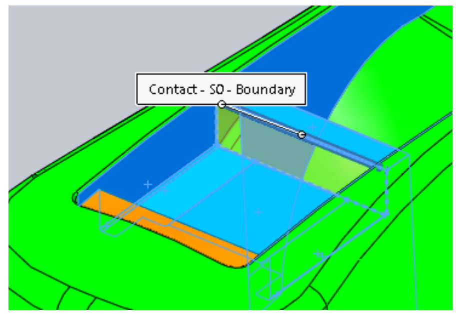


Lofted Surface

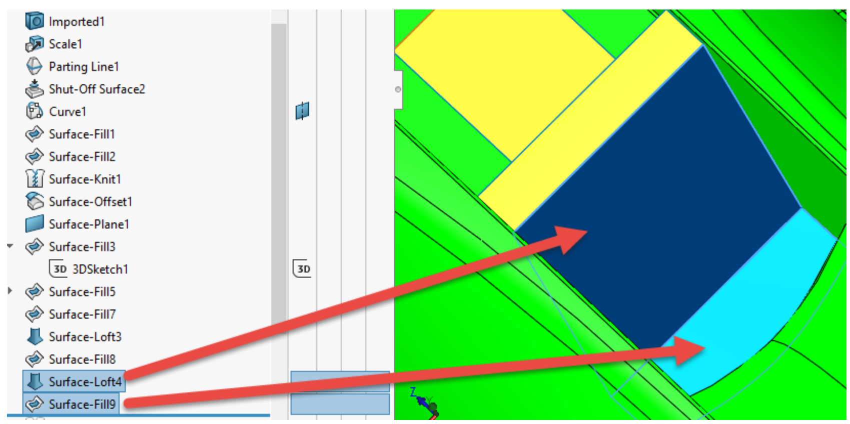
A **Lofted Surface** is used to bridge across the top of the opening.



Notice that this surface will create an edge to close off another opening with a **Filled Surface**.

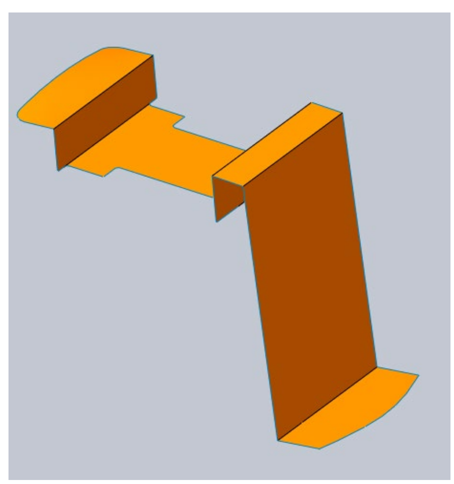


Another Lofted Surface and Filled Surface complete the shut-off for this area.

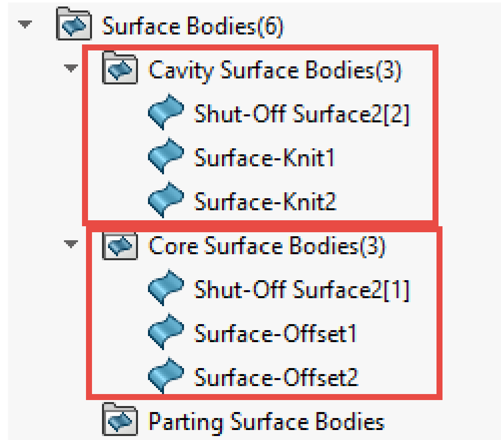


Knit the upper shut-off surface together to form one surface body.

Offset this surface body to create a copy.



Do not forget the importance of organizing the surface bodies into the proper folders.

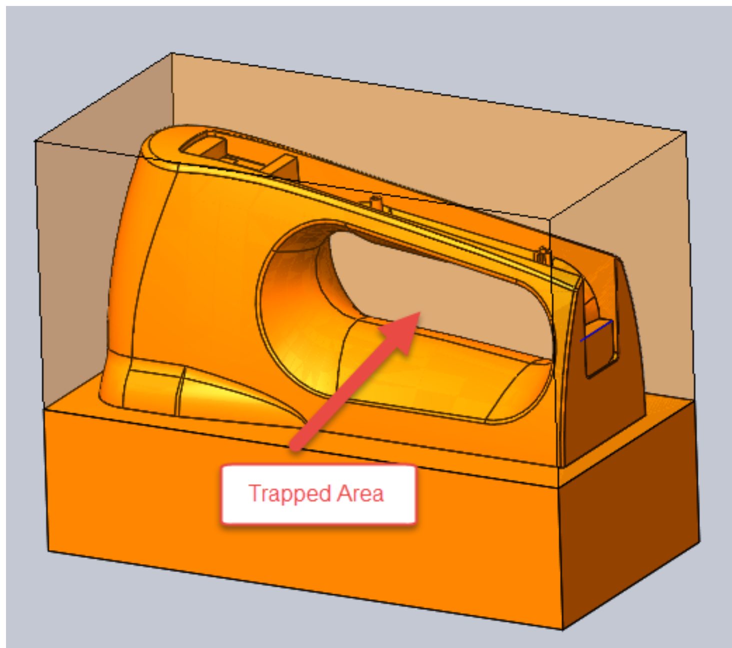


The surfaces in each folder could also be **Knit** together if the user chooses to.

The Parting Surface is created, and the Tooling Split completes. However, there is an area around the handle that is trapped.

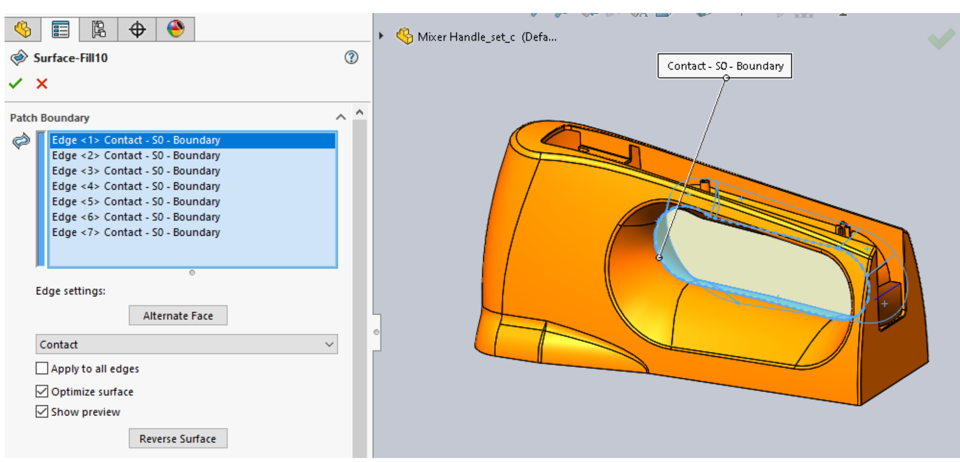
Side cores need to be created for these.

The Core command is not able to automate these side cores because there is no stopping face to define the core geometry.



To manually build the side cores, the required surfaces will be created. The face of the hole will be copied using the **Knit Surface** command.

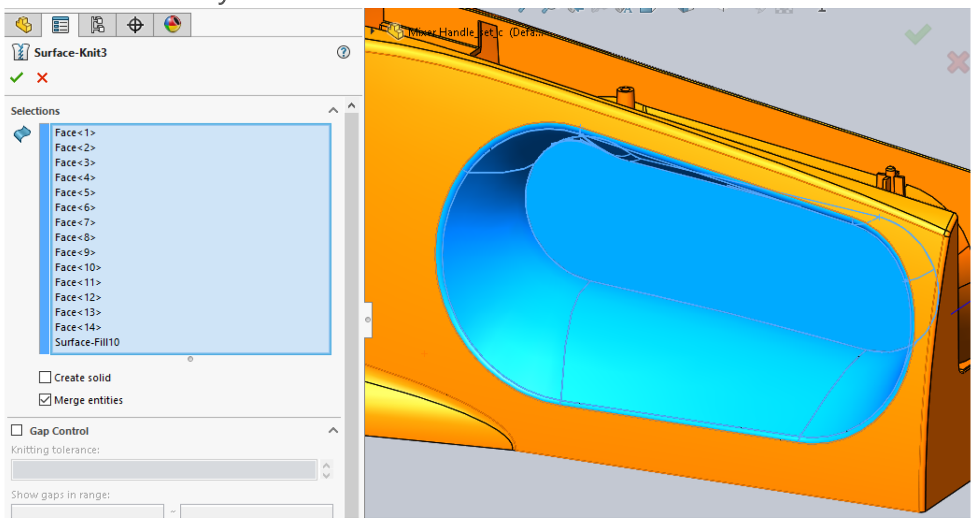
A **Filled Surface** is created between the transitions of the two side cores. This would represent where the two side cores shut-off on one another.



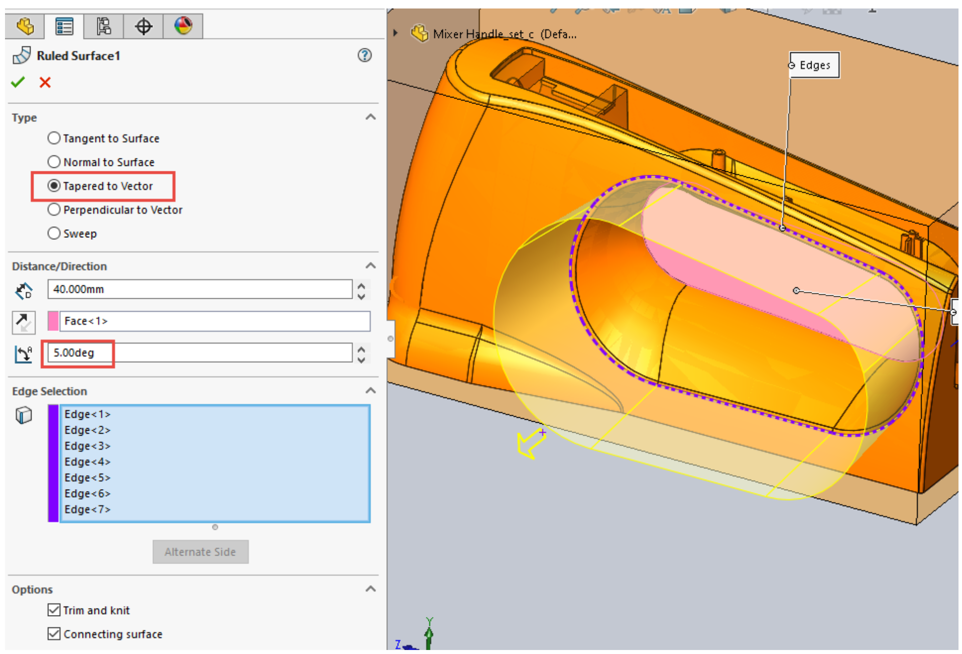
Knit Surface

The **Knit Surface** command can be used for more than just combining surface bodies. When **Faces** are selected on a model and knitted together, copies of these faces will be knitted together into a surface body.

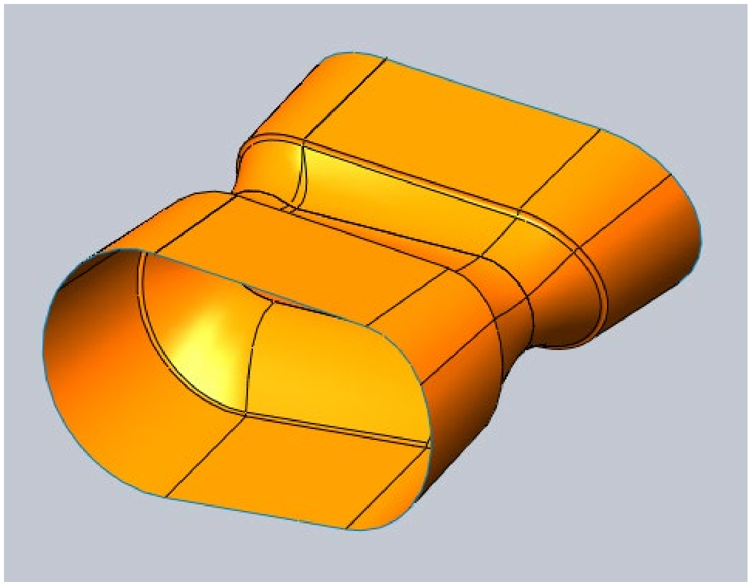
The faces on one side of the part and the Filled surface are selected. These will form the faces of the side core that form the part handle opening.



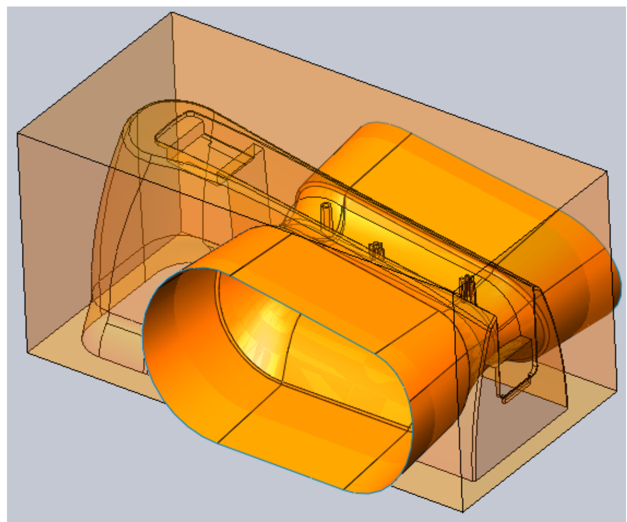
Ruled Surface is used to create the surface of the core that fits into the mold cavity half. This surface extends beyond the cavity block. It will be used to Split the cavity. The Tapered to Vector type is selected to add draft to the cores.



The surfaces are knitted together, and the surface body is mirrored about the planar face to create the symmetrical core on the other side.

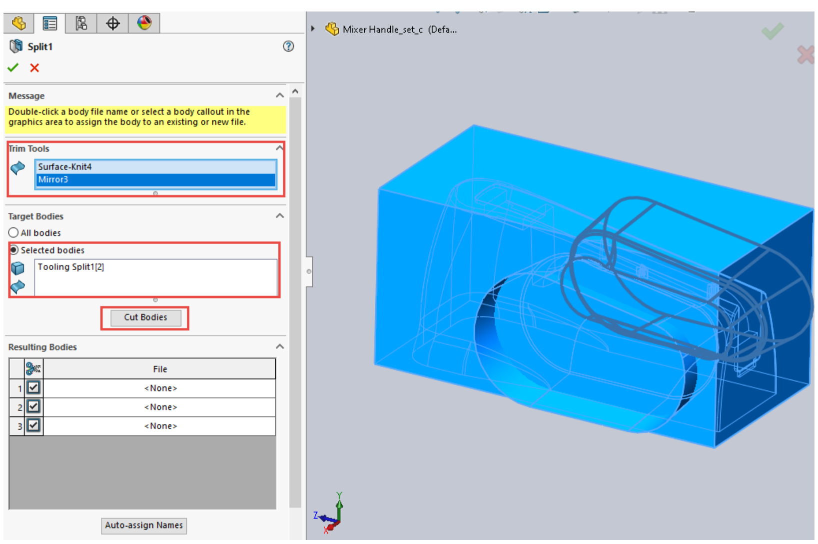


Split the Cavity Body



The **Split** command is used to split the cavity body into three separate bodies. The surfaces are used as the trim tool.

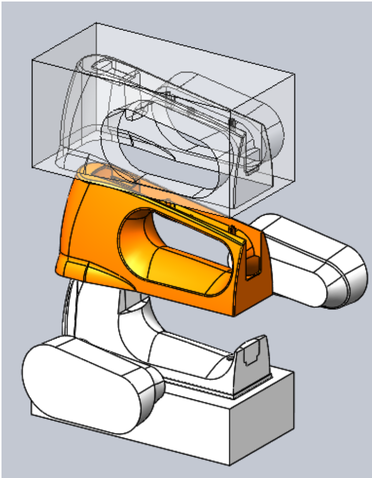
To start the Split command, click **Insert** > **Feature** > **Split.**



The surfaces created and mirrored are the **Trim Tools**.

The cavity solid body is the **Target body**.

The **Cut Bodies** button splits the cavity body.

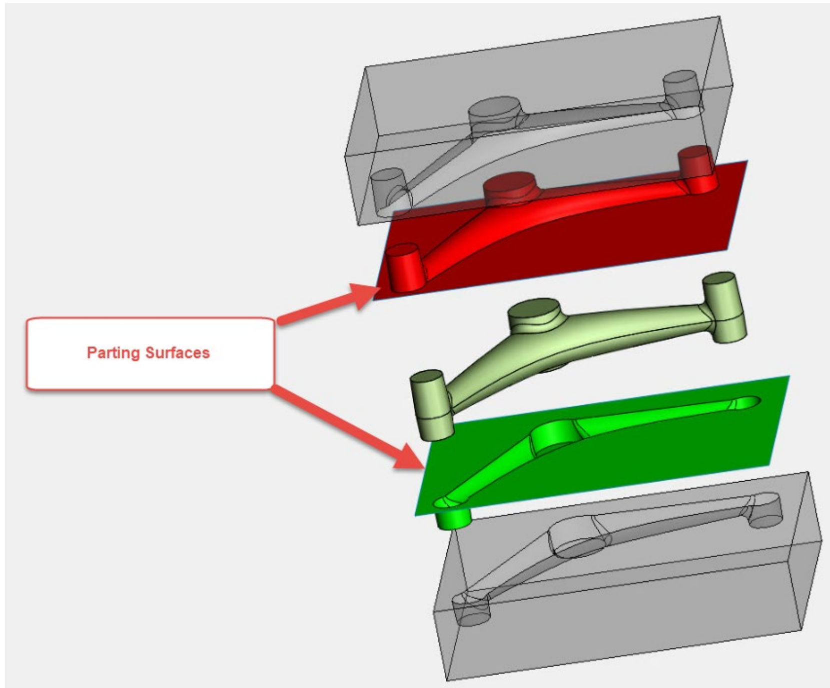


All three **Resulting bodies** are selected.

This completes the mold with Cavity, Core, and Side Core inserts.

Lesson 7: Alternative Methods for Mold Design

The main objective in mold design is to build a parting surface between the core and cavity sides of the mold. We learned how to do this using the SOLIDWORKS Mold Tools in Lesson 2. The parting surface is shared by both sides of the mold and, by combining it with the faces from the part impression, the surfaces of the core and cavity mold inserts are designed.



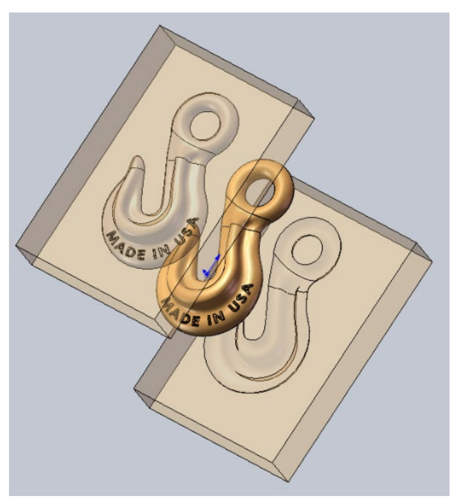
There are alternative methods for creating the necessary surface data for modeling.

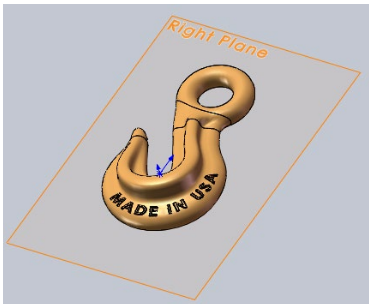
- Multibody modeling using the Combine and Split commands.
- Creating an assembly model with the part and core and cavity inserts and then using a Cavity command.
- Creating the mold blocks using surfacing alone.

These methods are typically used when the Parting Line and Parting Surface tools cannot be used effectively.

Multibody Model Technique

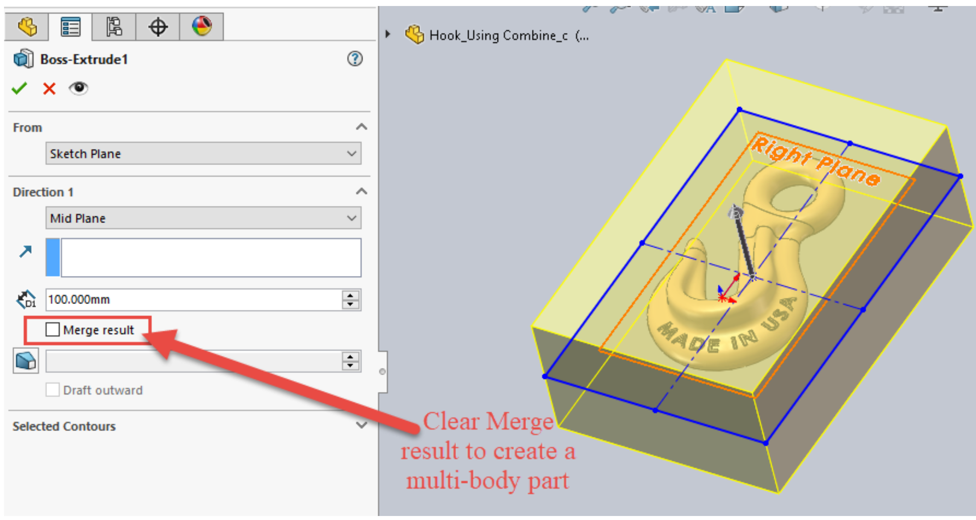
The multibody modeling technique to create mold core and cavities utilizes the **Combine** command to remove the engineered part from the tooling block and the **Split** command to separate the tooling block into two solid bodies.





The first step to using this technique is to create a block around the part that represents the mold block size. This part lies on the right plane meaning the right plane would be the parting plane.

A Sketch is created on the right plane and dimensioned to be the size of the Cavity/Core blocks.



Use a mid-plane **Boss/Extrude** to create the block. Using midplane will extrude both directions from the right plane.

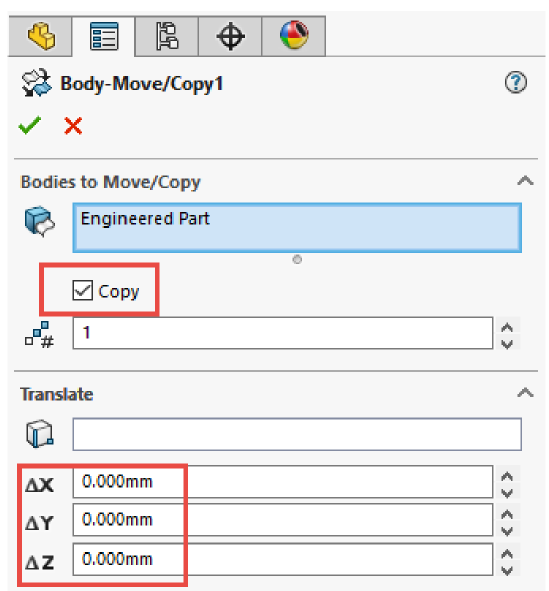
Clear the **Merge Result** box to create the mutibody part.

Move/Copy Body

The **Combine** command absorbs the "tool" body. A copy of the engineered part will need to be created to keep us from losing it.

The **Move/Copy Body** command can be used to copy surface or solid bodies in place or to move or rotate them with or without copying.

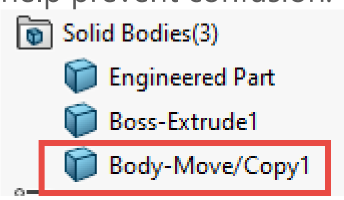
To copy a body in place, start the **Move/Copy Body** command.
Select the body to copy.
Select the **Copy** check box.



Leave the **Translate** options all set to zero.

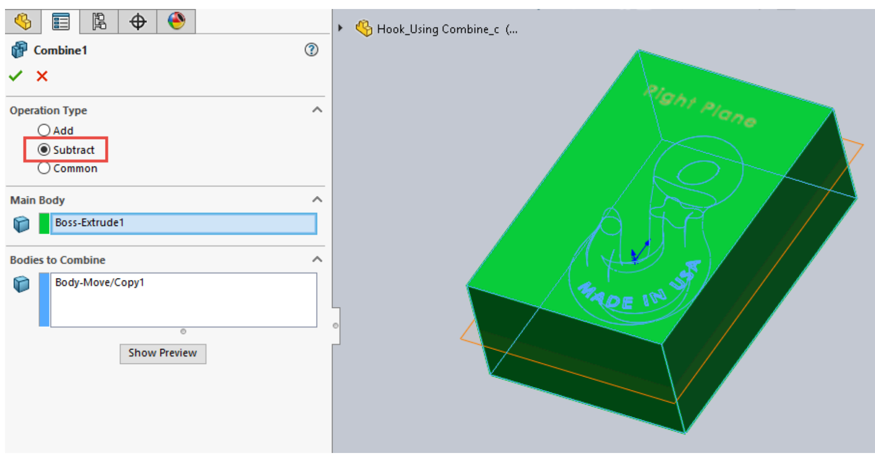
Accept the message: “Neither a translation nor a rotation is specified. Do you want to proceed?”

This creates a new solid body in the solid bodies folder. This body can be renamed to help prevent confusion.



Combine

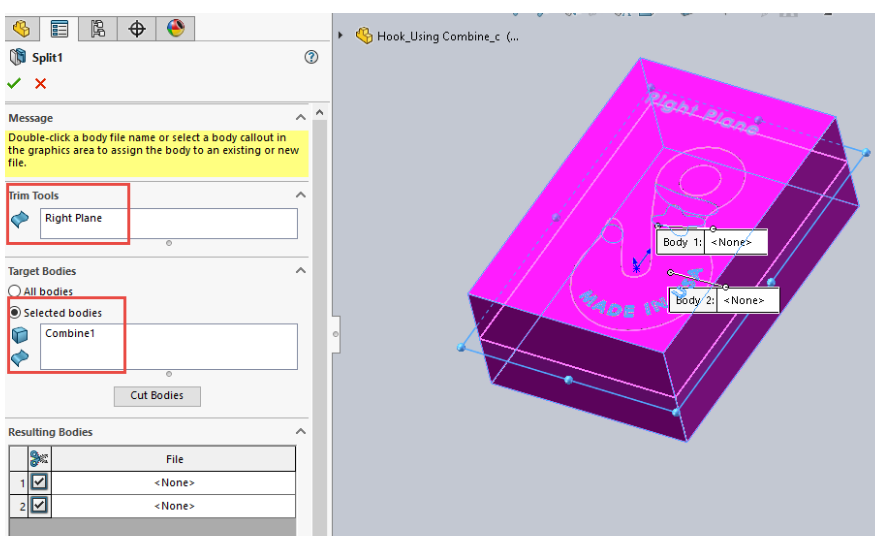
The **Combine** command gives the user the choice to **Subtract,** which will remove the **Bodies to Combine** from **the Main Body**. The subtract option leaves a cavity inside of the block.

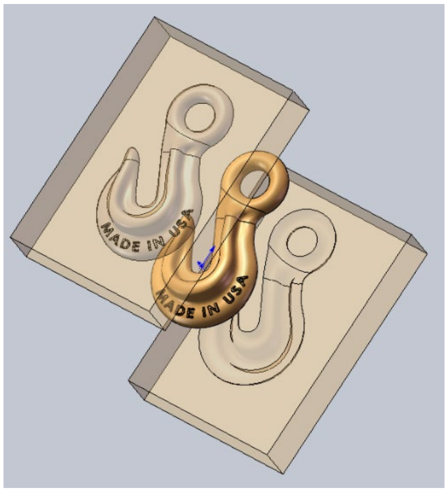


Split

The **Split** command will cut the block into two bodies using the **Right Plane** as the **Trim Tool**.

Select the **Combine Body** as the **Target** body.

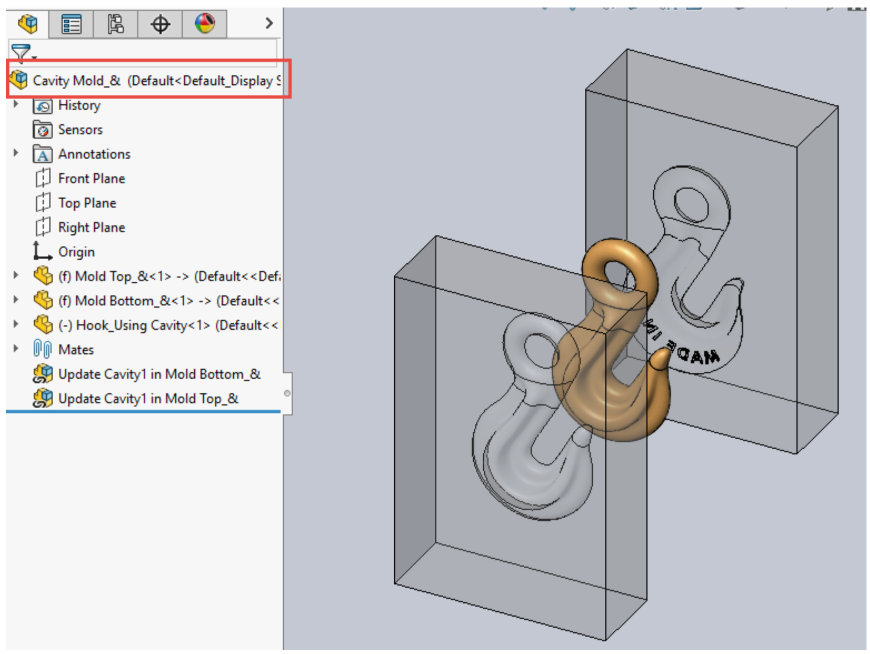




The block has been split into two tool bodies.

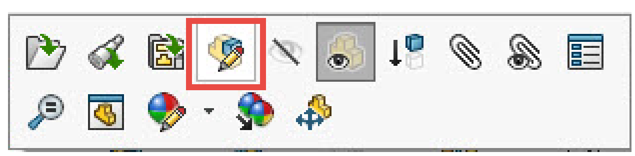
Assembly Model Technique

The **Cavity** command is very similar to subtracting one body from another using **Combine**. The **Cavity** command is different only in that it works between parts in an assembly. An advantage is that the “tool” body is not absorbed, so there is no need to create a copy.



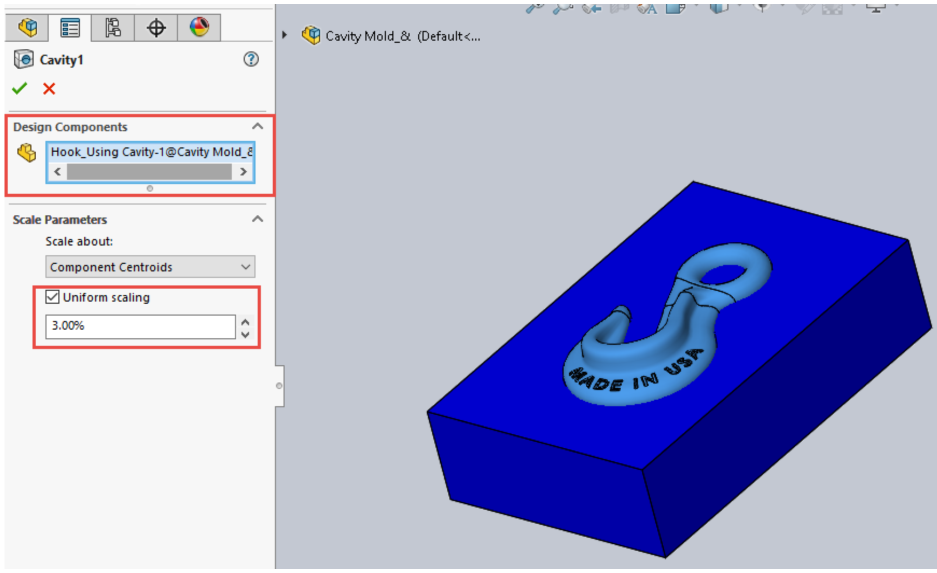
Start with an Assembly model that includes the Cavity block, Core block, and the Engineered part.

To access the Cavity command, the Part that is being cut must be editing within the assembly.



Cavity

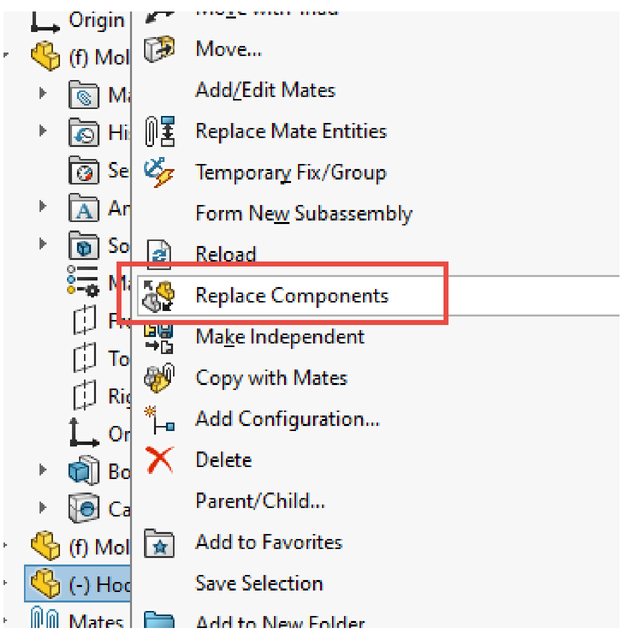
The Cavity command is located at:
Insert > **Features** > **Cavity** or **Insert** > **Molds** > **Cavity.**



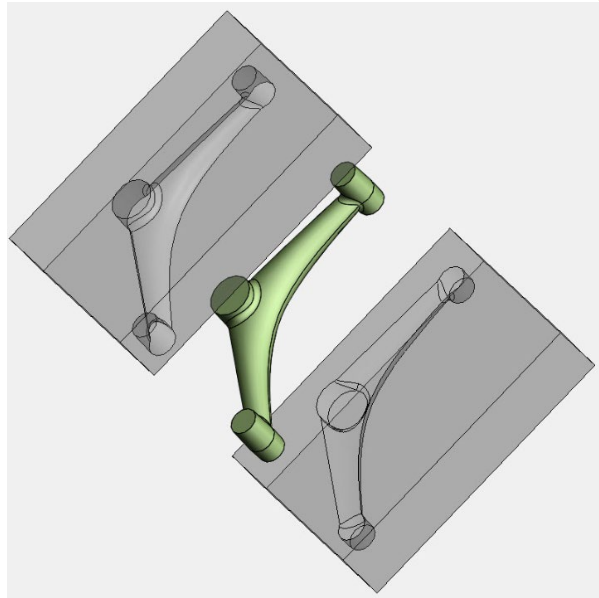
The **Design Component** is the Hook Part. The cavity can be **Scaled** up by a percentage to accommodate for shrinkage.

These steps must be repeated individually for each mold component that will receive a **Cavity**.

An advantage of using the Assembly and Cavity method is the ability to simply replace the engineered part component within the assembly. This will update the Cavity feature in the mold block components to capture the new part geometry.



Surfacing Technique



Surface features are created to develop the mold tooling parting surfaces for a simple part.

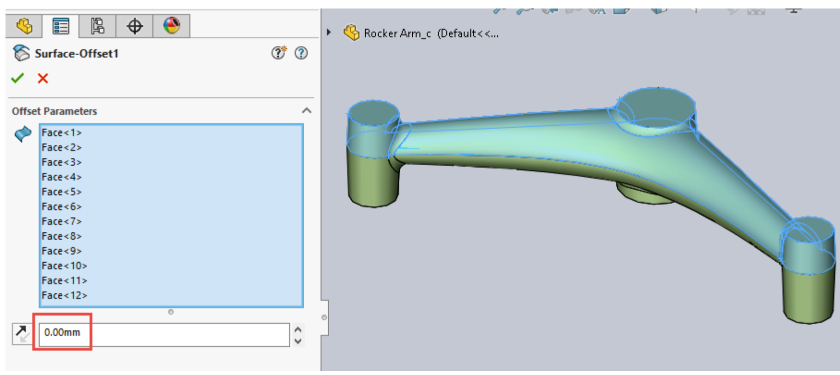
This method is similar to what happens in the background using the Mold tools commands.

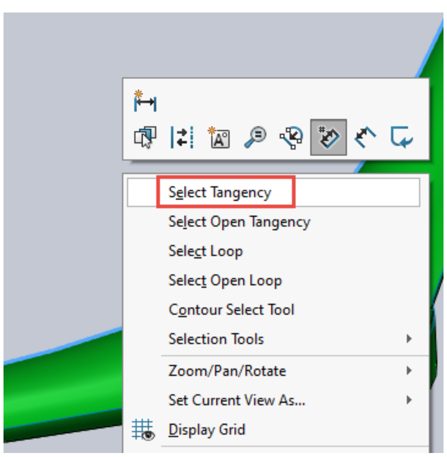
The faces of the engineered part are copied to create surface bodies. These surface bodies are then trimmed and knitted to form the Core and Cavity insert solid bodies.

Faces for Mold Tooling

To capture the surfaces for the mold cavities, use the **Offset Surface** command with a zero value.

One way to easily select all the required faces is to switch to a Front View, and then box select all visible faces.

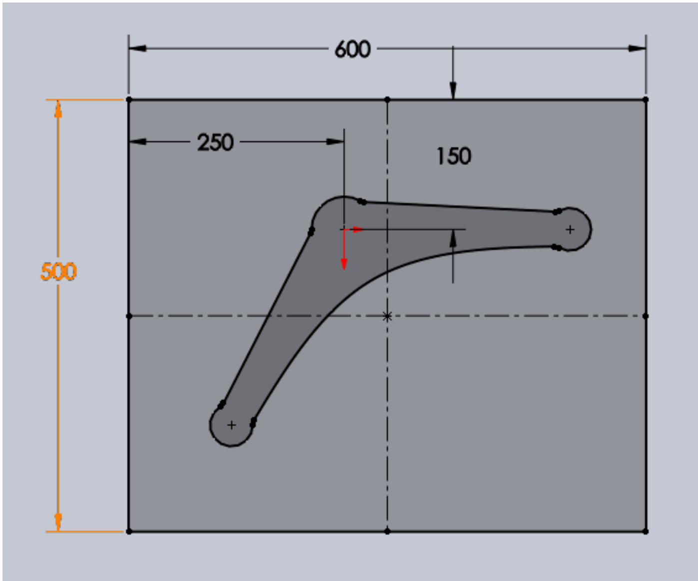


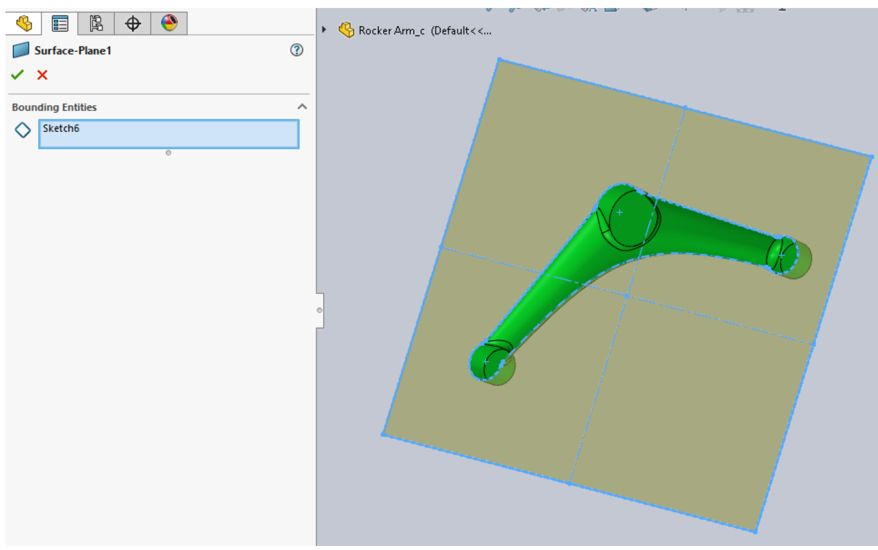


To create the Parting Surface, create a rectangle sketch on the **Front Plane** dimensioned to the intended size of the mold blocks.

Select the profile of the engineered part by **right-clicking** on an open edge of the **Surface-offset** body and click **Select Tangency**. Then click **Convert Entities.**

The Sketch result should look like this:

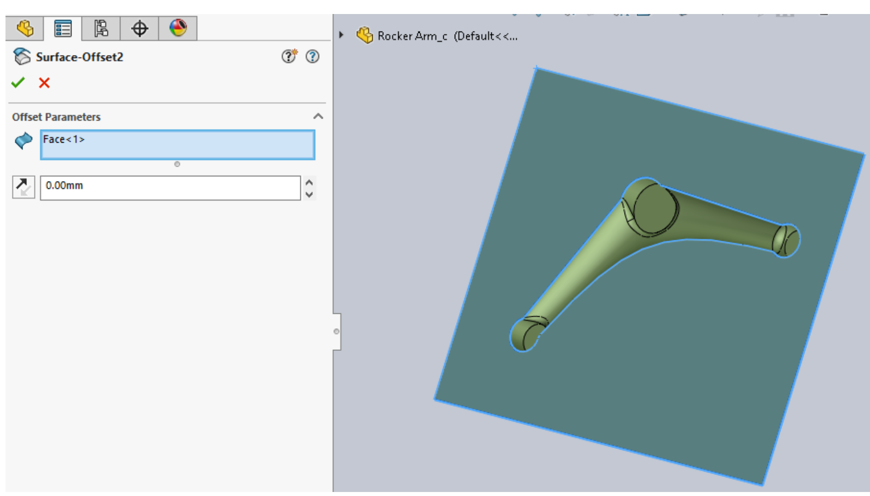




Use this sketch to create a **Planar Surface.** This Planar Surface represents the Parting Surface of the mold tooling.

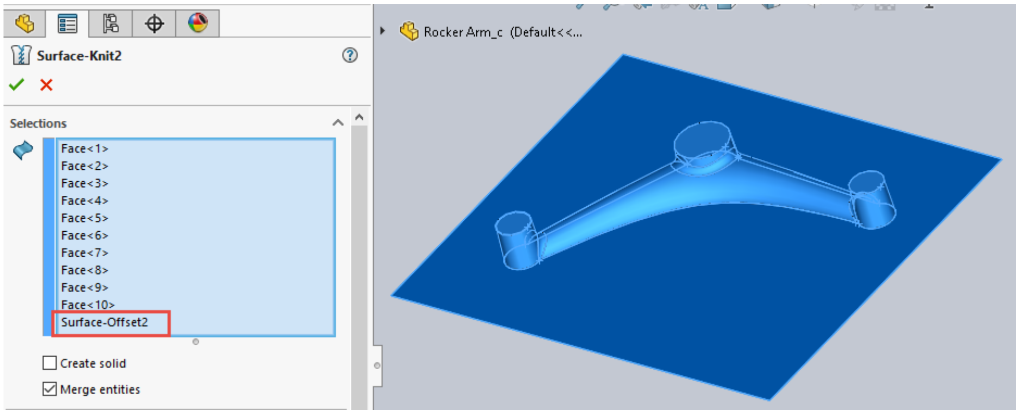
Knit the Planar Surface and the Offset Surface bodies together to create a single surface body.

This captures the cavity and face of the mold for one side.

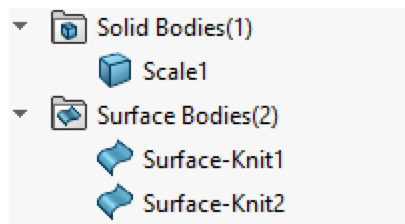


Copy the Planar face using **Offset Surface** with a distance of zero.

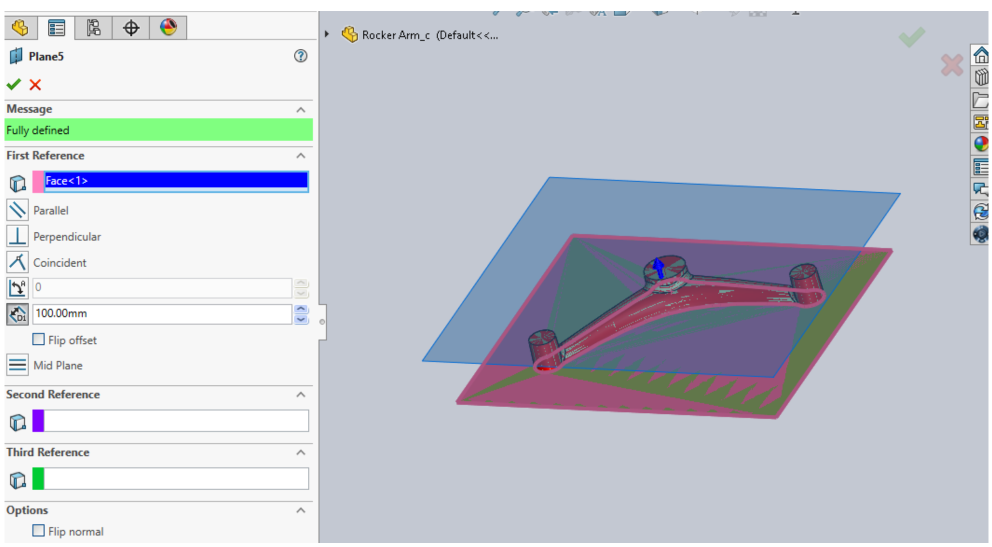
Next, the faces on the other side of the engineered part need to be copied in place. Another way to copy faces is to use **Knit Surface**. This option can copy and knit in one operation, but the faces selected must share common edges. Select the faces on the opposite side of the part from the previous and the **Surface Offset2** Planar face. Select **Merge Entities** to knit these surfaces together into one surface body.



At this point, the faces at the parting surface have been created for the two halves of the mold. The next step is to create the solid bodies.

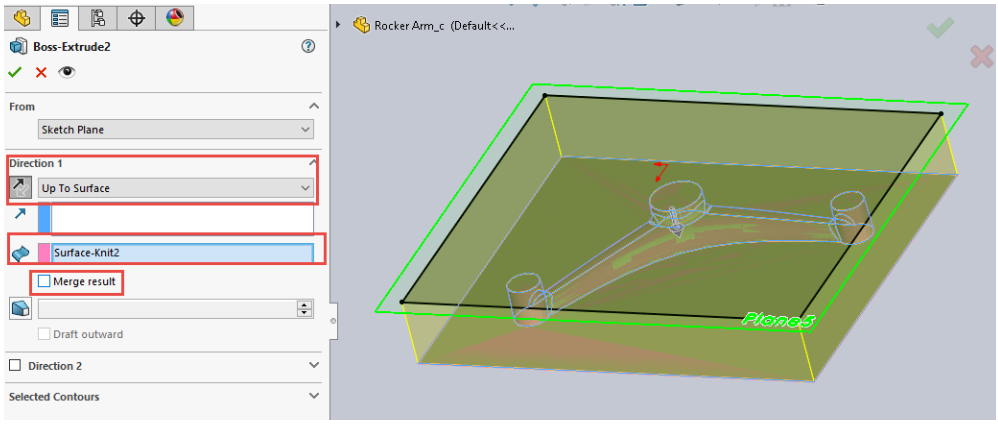


An **Extruded Boss/Base** feature is used. Start with a **Reference Plane** offset from the planar face the thickness of the mold block.



Create a Sketch on that plane. Use **Convert Entities** to copy the outer edges of the planar surface.

Extrude the sketch using the **Up to Surface** end condition. Choose the Parting Surface face as the up to reference. Clear **Merge result** to create this feature as a separate body.

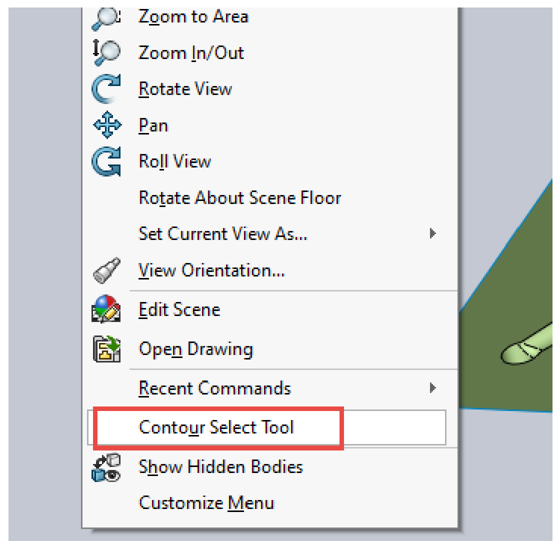


An alternative way to create the Extrude Boss sketch would be to use the **Contour Select Tool**.

Access the **Contour Select Tool** by right-clicking in an open area in the view port.

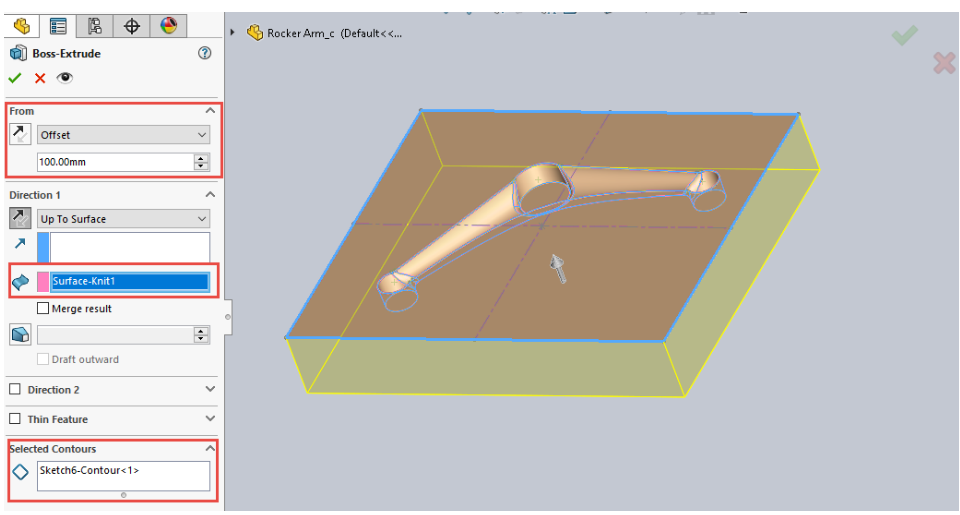
Select a line on the sketch used to create the original Planar Surface.

Start an **Extrude** feature.
In the **From** selection pane, choose **Offset.**



Use **Up to Surface.**

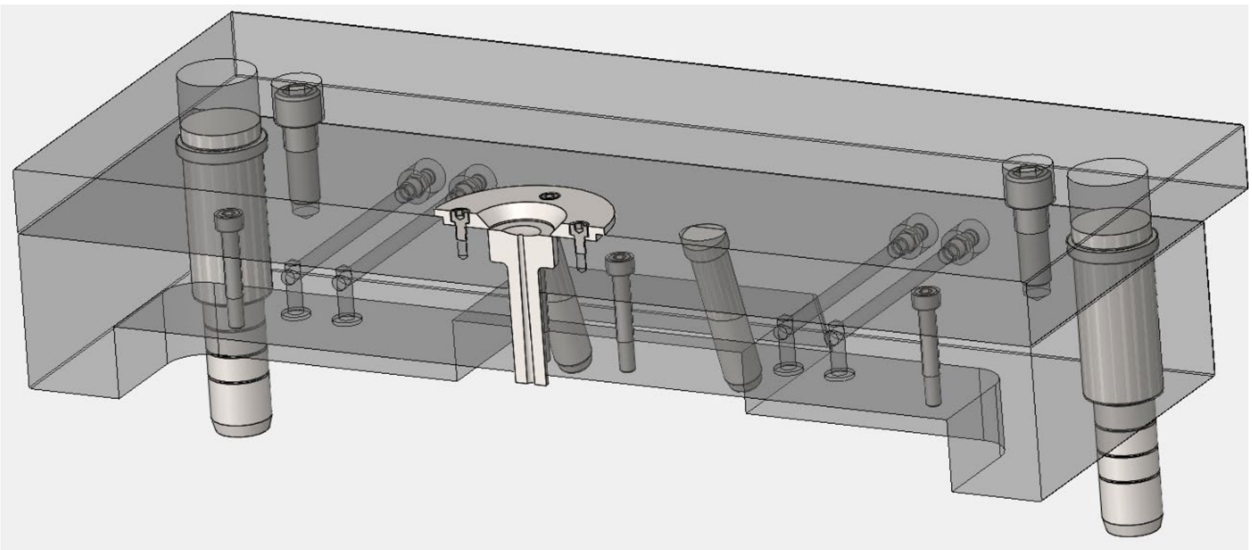
Notice that the Sketch contour is in the **Selected Contours**.



This workflow keeps the designer from having to create a reference plane and utilizes an existing sketch that saves time.

Lesson 8: Reusable Data

Even though each model is different, many tasks are repetitive. Molds have standard parts and features that may be similar or the same from mold to mold. There are ways to save time by reusing data instead of recreating items each time you design a mold. **Library features**, **Smart components**, and **downloaded files** from websites are ways to reuse design data.



Library features include one or more features that can be inserted into a part in a single operation. These can be created from scratch or saved from existing features in other parts. Library features can contain variable dimensions and configurations.

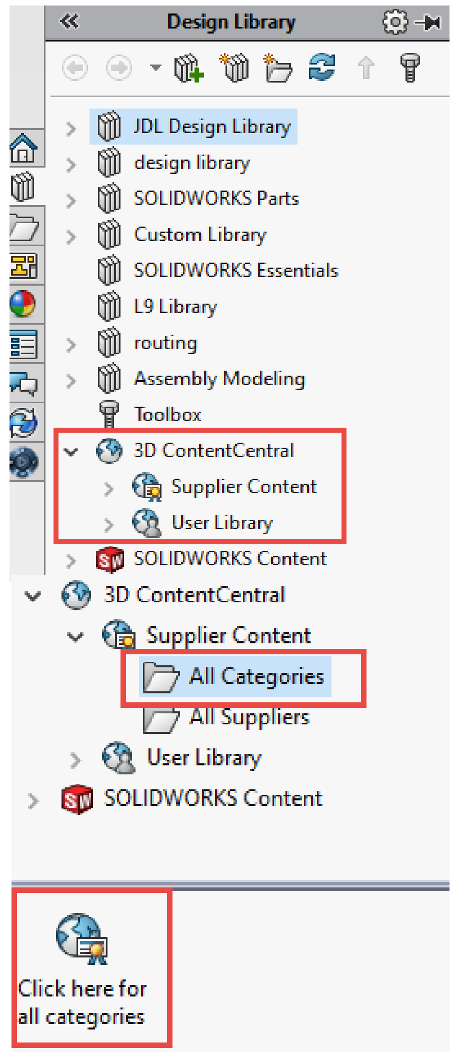
Smart components are parts that have intelligence built-in allowing features to be created in the surrounding parts within an assembly. They may also include the option to have additional components inserted with them and can include the ability to auto-size.

3D ContentCentral provides a library of models that can be downloaded. Parts and Assemblies can be directly used or inserted into a SOLIDWORKS design without having to remodel them. Many mold component suppliers have downloadable 3D models on their websites as well. These can be configured and downloaded in SOLIDWORKS format.

3D ContentCentral

Access **3D ContentCentral** from the task pane on the right side of the SOLIDWORKS viewport. Select the Design Library icon in the task pane and expand 3D ContentCentral.

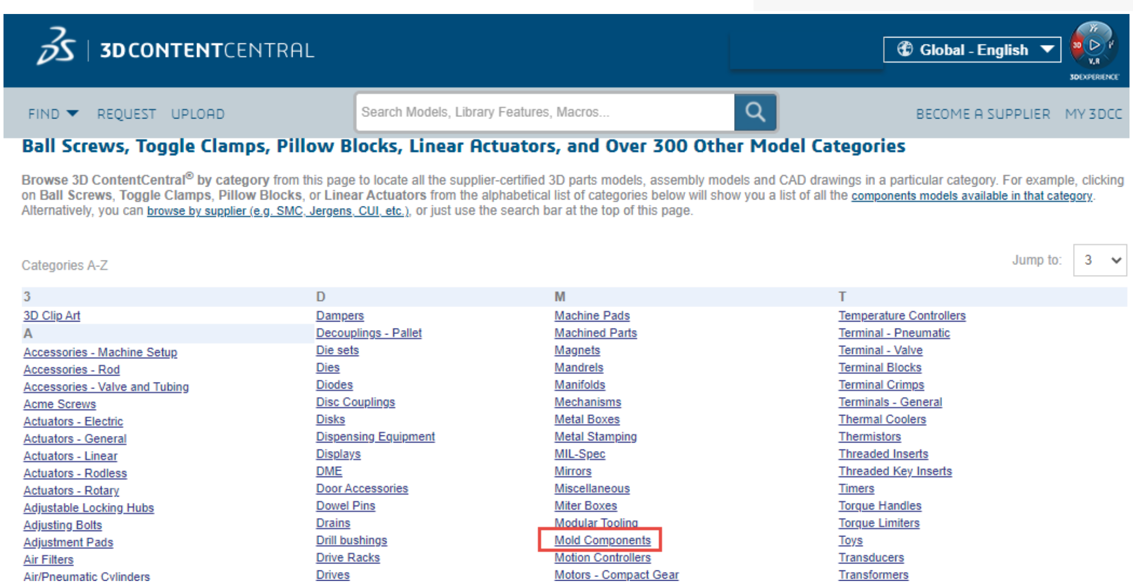
You will be prompted to set up an account to access and download from 3D ContentCentral.



In the Design Library pane, expand **Supplier Content** and click **All Categories** in the lower pane.

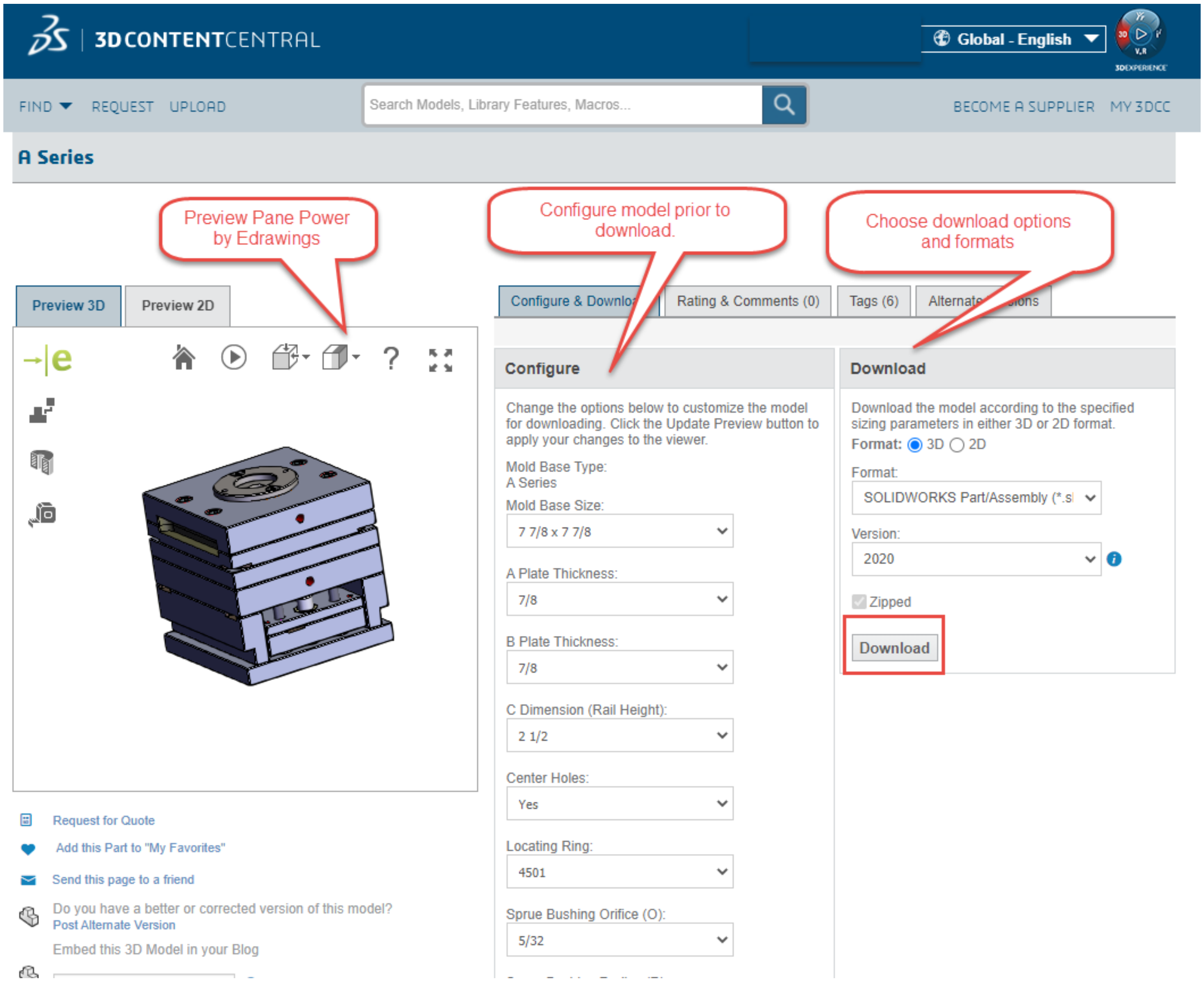
Select the **Click here for all categories** icon. This opens a web browser to the 3D ContentCentral website.

Look for the Mold Components category and select it.



The companies that have mold components available for download are listed here.

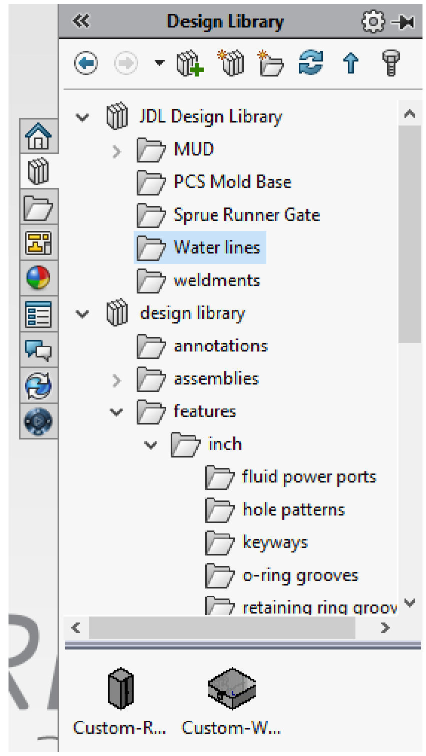
An example from PCS A series mold base.

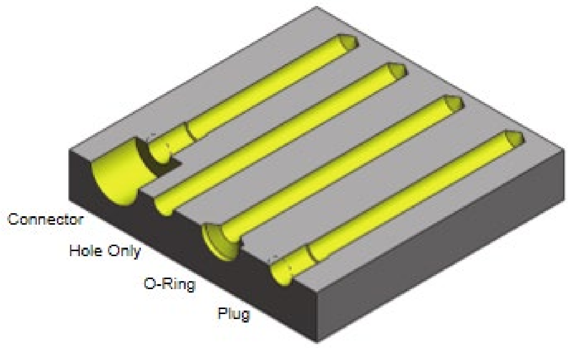


A model can be dragged and dropped from the page directly into SOLIDWORKS or it can be downloaded into a .zip file.

Library Features

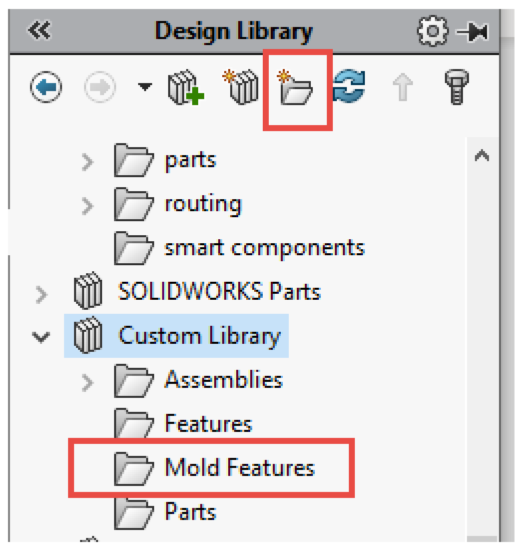
The **Design Library Features** folder contains library features that are included with SOLIDWORKS. When a feature is inserted from the library, it is copied into the active part. When inserting library features, there are two ways for locating them. The first is to reattach references on insertion. In this case, the necessary locating dimensions and references are included in the library feature and they are redefined as the feature is added from the library. The second is to use edit sketch to locate features. This approach does not include the external references in the library feature. The references are added during the edit sketch portion of the command.



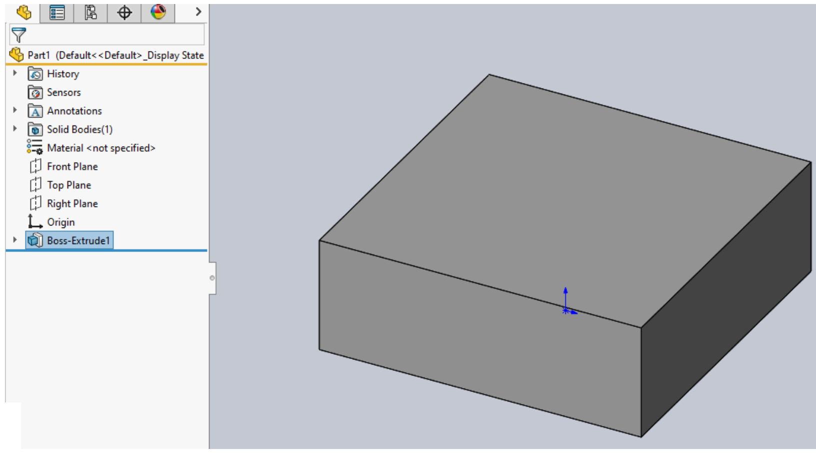


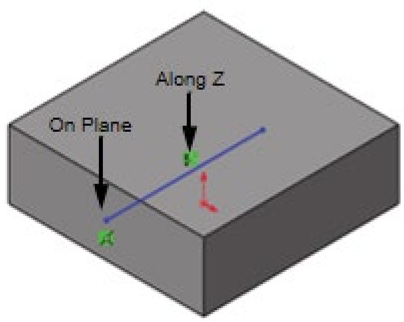
Before creating a library feature, a folder can be created in the **Design Library** to store custom mold features.

Expand the folder in the Design Library where you want the new folder to be located and then click the **new folder icon** in the upper part of the task pane. Then enter the name for your new custom design library folder.



A base feature must be created so that there is geometry present to add the feature to. Typically, just an **Extruded Boss/Base** feature will work.

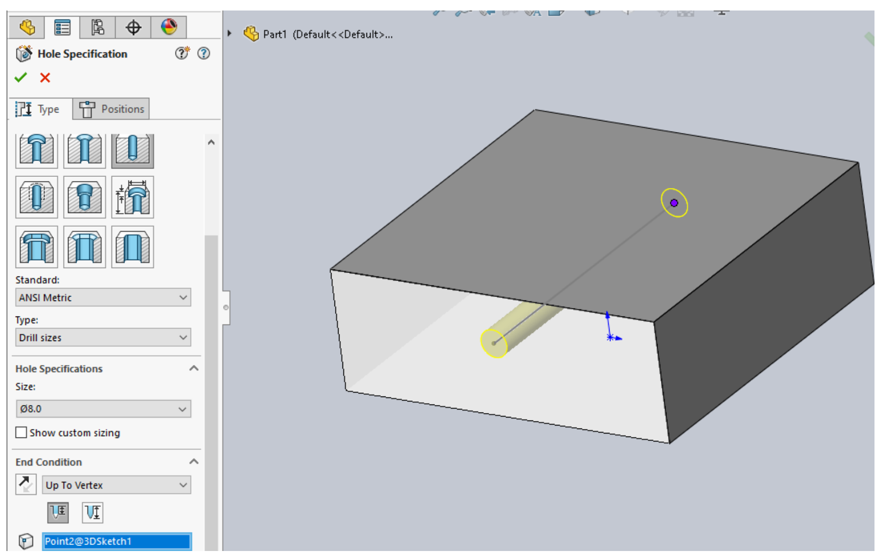




The library feature being created in this instance is for mold waterlines. A **3D Sketch** is used to define the waterline path.

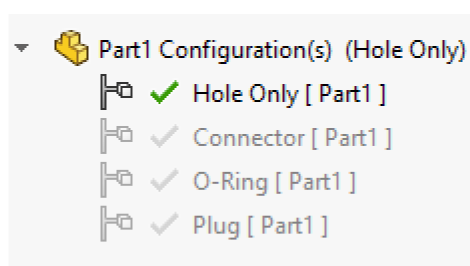
The sketch line will have an **On Plane** relation with a face on the block and an **Along Z** relation.

The next step is to create the features that are intended to be saved as library features.
An 8mm ANSI Metric drilled hole is added using **Hole Wizard**.
Place the hole center coincident with the endpoint of the 3D Sketch.



Configurations in Library Features

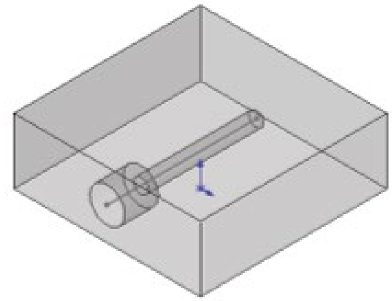
Library Features can be made more flexible by adding configurations. When you add a library feature with more than one configuration to a part, you can choose the specific configuration to insert.



Configurations are added for the Hole, Connector, O-ring, and Plug type holes.

Connector configuration

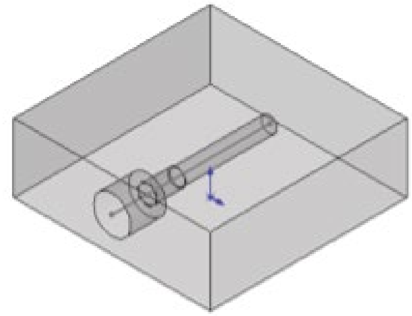
Make the Connector configuration active. Add the following hole coincident with end of the 3D sketch:



- Hole Type: Hole
- Standard: ANSI Metric
- Type: Drill Sizes
- Size: Ø19.0
- End Condition: Blind
- Depth: 18mm
- Show Custom Sizing = checked
- Angle at Bottom: 18deg

Add a second hole

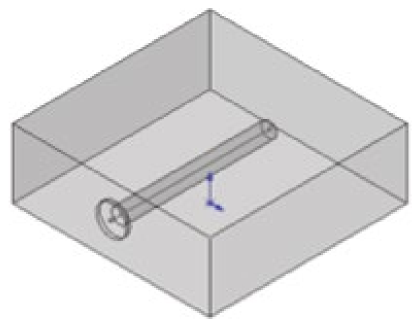
Add the following hole at the bottom of the Ø19mm hole and coincident to its center:



- Hole Type: Straight Tap
- Standard: ANSI Metric
- Type: Tapped hole
- Size: M10x1.0mm
- End Condition: Blind
- Blind Hole Depth: 15mm
- Tap Thread Depth: 10mm

O-Ring configuration

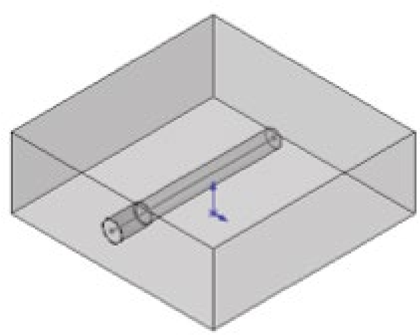
Make the O-Ring configuration active. Add the following hole coincident with end of the 3D sketch:



- Hole Type: Hole
- Standard: ANSI Metric
- Type: Drill sizes
- Size: Ø15.5
- End Condition: Blind
- Depth: 1.9mm

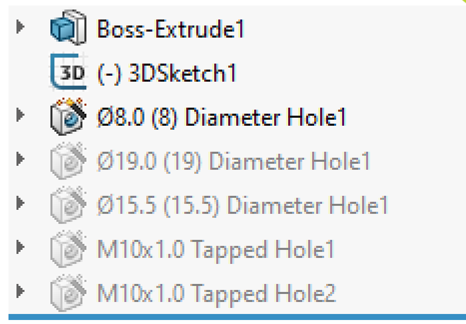
Plug configuration

Make the Plug configuration active. Add the following hole coincident with end of the 3D sketch:



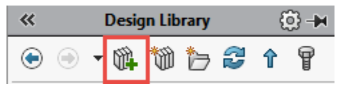
- Hole Type: Straight Tap
- Standard: ANSI Metric
- Type: Tapped Hole
- Size: M10x1.0mm
- End Condition: Blind
- Blind Hole Depth: 15mm
- Tap Thread Depth: 10mm

The Feature tree for the base feature has the holes for each of the different configurations in it.

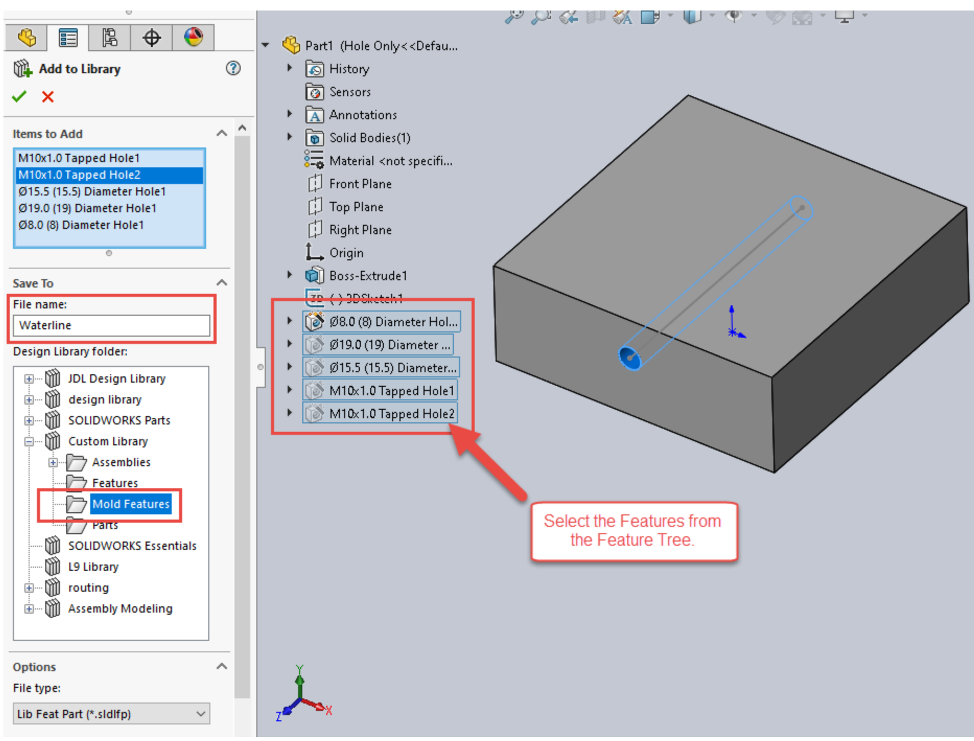


Create Library Feature

Use the **Add to Library** command to create a library feature. Include all the features except Boss-extrude1 (the base geometry) and the 3D sketch.



Name it *Waterline* and save it to the **Mold Features** folder.

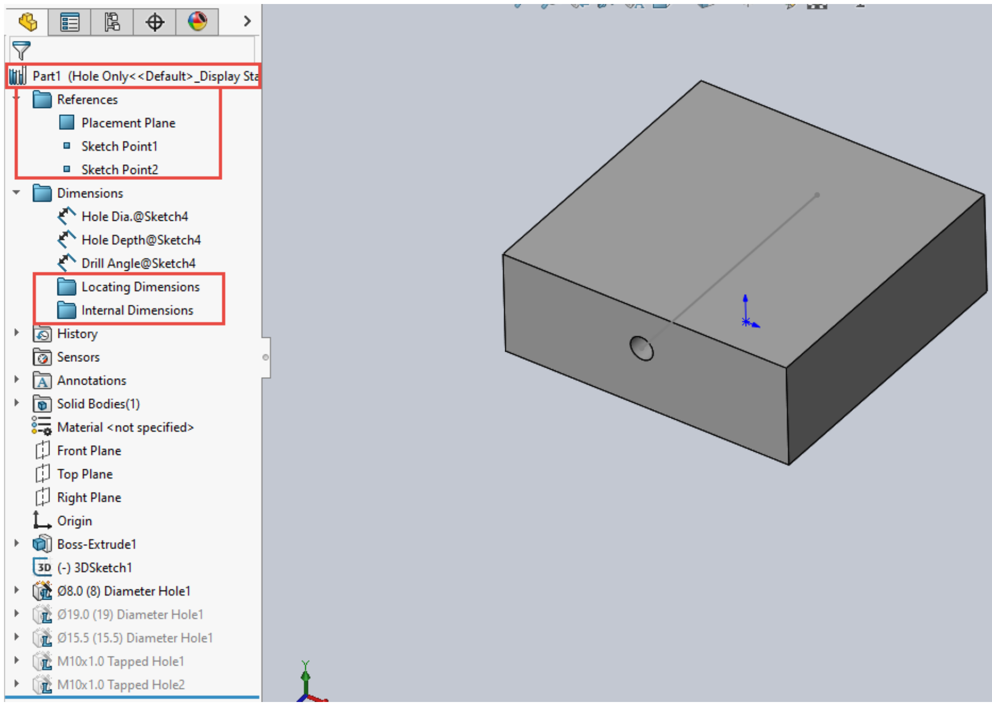


When the **Library feature** is added, the Feature tree changes.

The icon at the part level shows the library icon.

The **References** folder shows the location references for the Hole Wizard hole. Placement plane, center point, and up to vertex point.

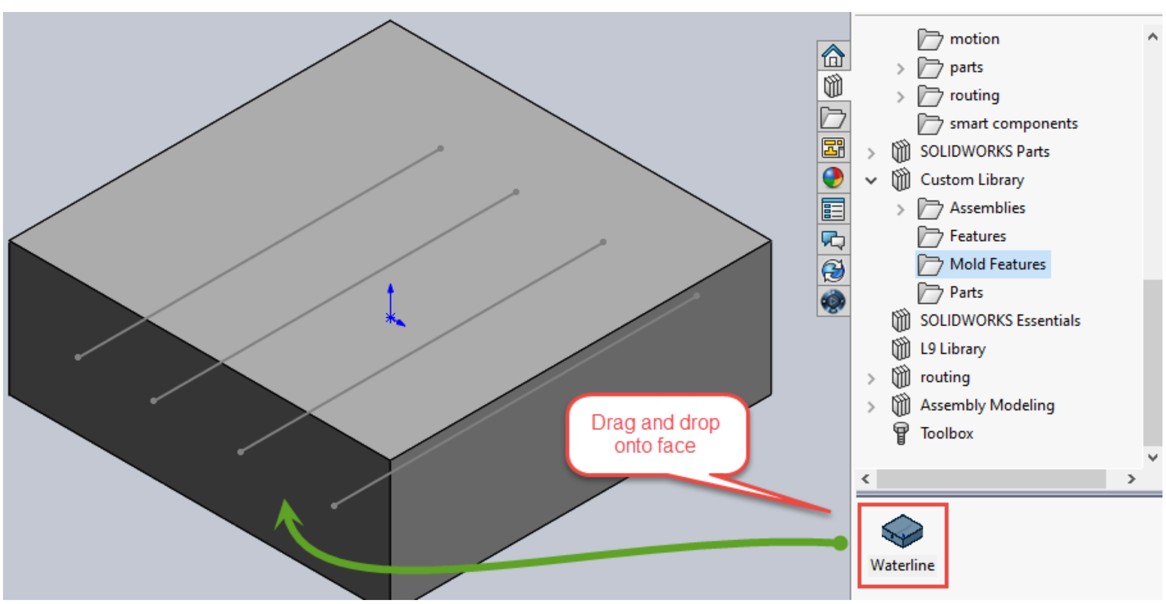
There are also **Locating Dimensions** and **Internal Dimensions** folders. There were no dimensions used to locate the hole so nothing will be put in the locating dimensions folder.



The **Internal Dimensions** folder is used to prevent these from being changed when dropping in the Library feature. The Hole Dia, Hole Depth, and Drill angle can be placed there.

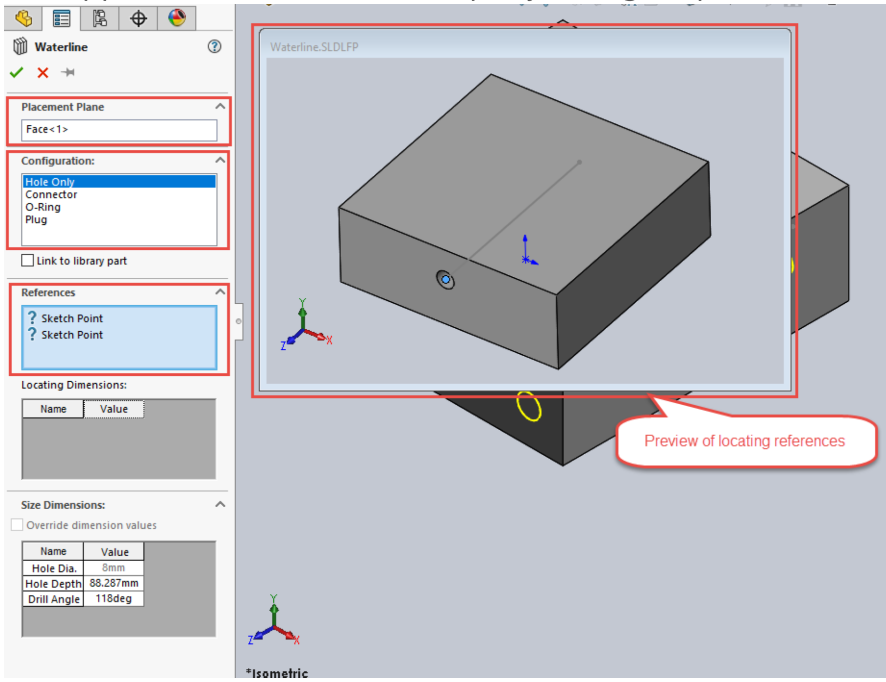
Using the Library Feature with Configurations

Drag and drop the **Library feature** from the bottom of the task pane onto the face where the feature is to be located.



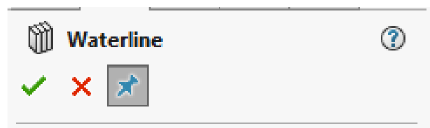
When the Library feature is dropped onto the face, the PropertyManager opens.

Placement Plane is the face where the feature was dropped. **Configuration** lets you select which configuration to place.

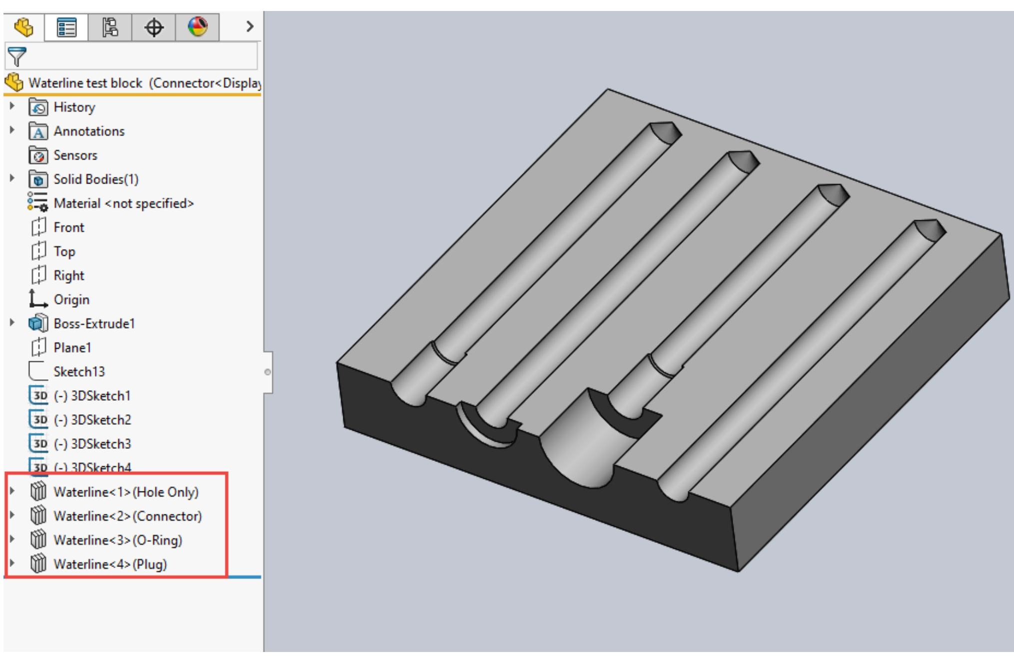


References are the locating reference to place the feature. The preview window gives an example of what to pick on the active part.

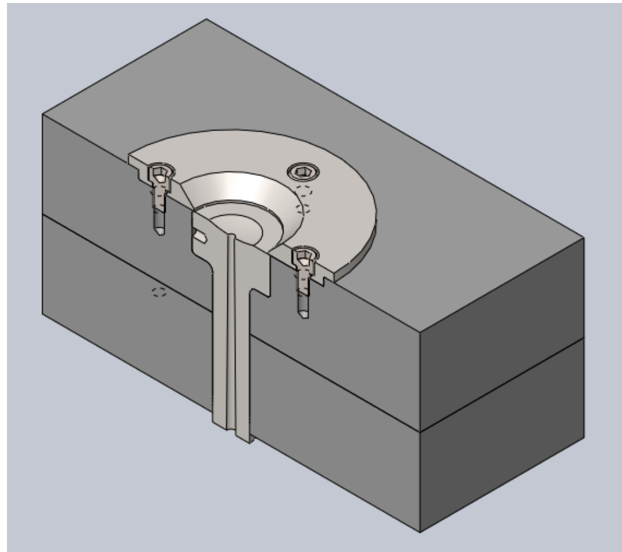
Tip: If you are placing multiples of the same Library feature, you can activate the pushpin at the top of the PropertyManager. This keeps the Library feature PropertyManager open and allows you to choose the face and other configurations without having to drag and drop again.



The feature tree for the active part has the Library features on it. These have the library icon to show the user that they were added from the Design Library.



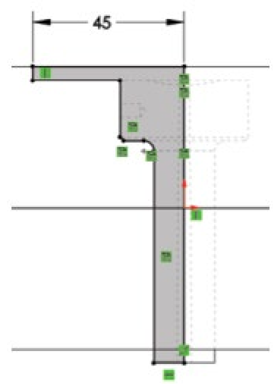
Smart Components



Smart components can be used to associate common components and features. Inserting a smart component into an assembly enables easy addition of the related components and features in one step. The smart component can then be used in any number of different assemblies and always have its associated components and features inserted with it requiring no additional steps.

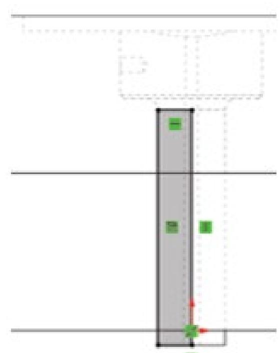
There are two steps to creating smart components. First, the component to be must be assembled in a defining assembly with the appropriate components and any in-context features. The defining assembly is like the base feature used when creating a library feature. Next, the smart component is detached from the defining assembly bringing all information about the smart feature or component references. There is no residual external reference to the defining assembly or other components.

Create Defining Assembly



The defining assembly should have all the features that will be associated with the smart component added to it along with any other components to be brought in with it.
Creating in-context features within the defining assembly is important so that these will be transferred over with the smart component as well.

To create an in-context feature, first select the component in the assembly that you are placing the feature in and select **Edit Part.**

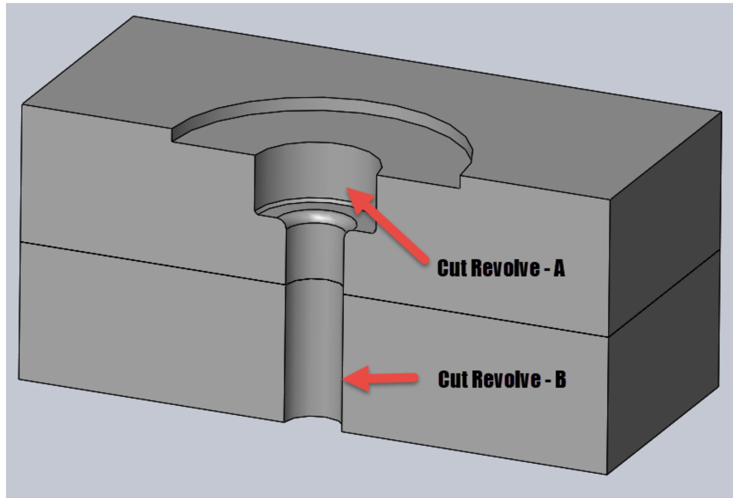


Use the geometry of the part to be the smart component as a reference when creating the sketch for the feature. **Convert Entities** is a good tool to use.

In this case, two revolve cut features are created to cut the pockets for the sprue bushing into two plates of a mold base.

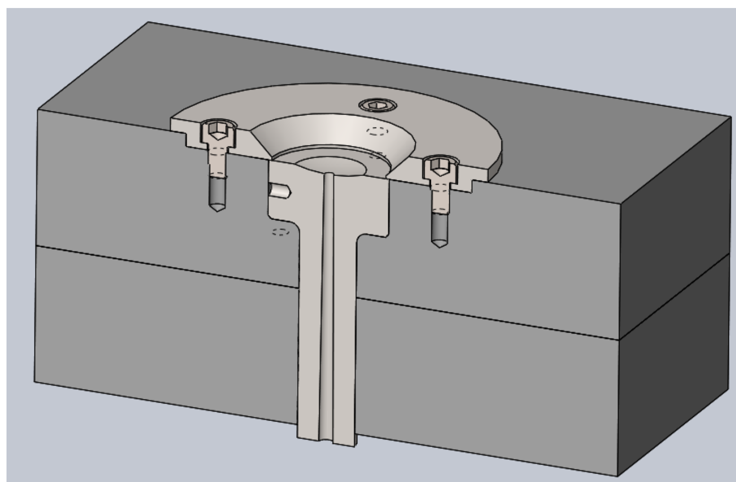
If there are multiple features to be created on different parts in the assembly, it is important to rename the features.

They must have different names for the **Smart Component** to apply each cut to the correct part.



Insert all the components to be associated with the Smart Component into the defining assembly.
In this case, there is the Locating ring and 4 cap screws with their tapped mounting holes.

This is the finished defining assembly for this model.



Make Smart Component

The **Make Smart Component** command is used to select the component, associated components, and features from the defining assembly.

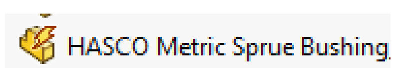
Make Smart Component can be found in the **Tools** menu.

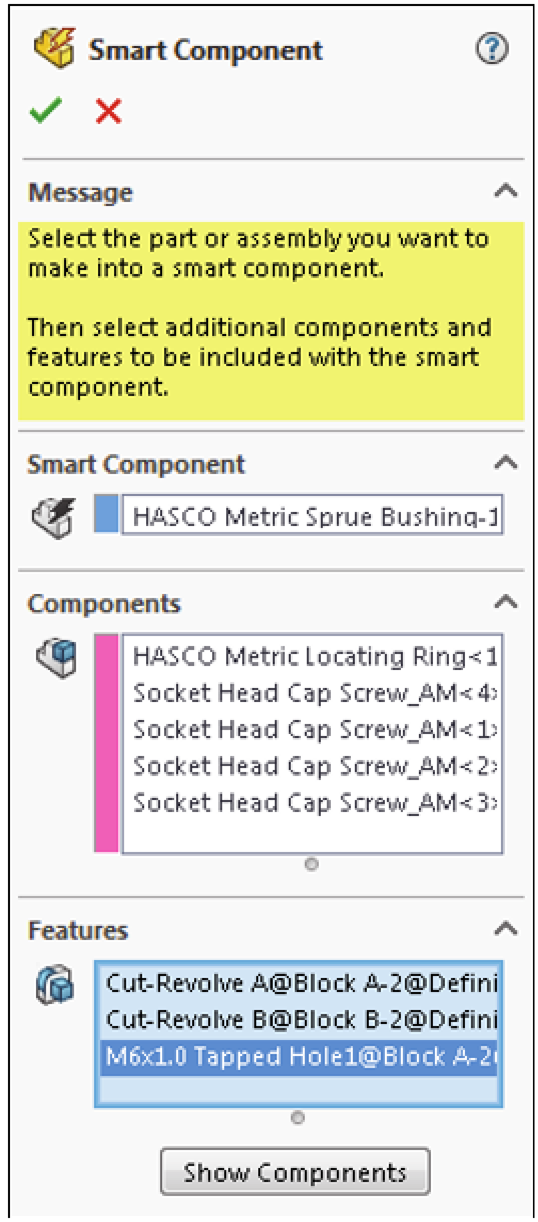
Smart Component: Select the part that is going to be the “smart part”.

Components: Select the related components that will insert with the smart component.

Features: Select all the features related to the smart component and the related components. These features will be inserted with the smart components when dropped into a new assembly.

Smart Component Icon: The lightning bolt symbol on the part icon indicates that this is a smart part.



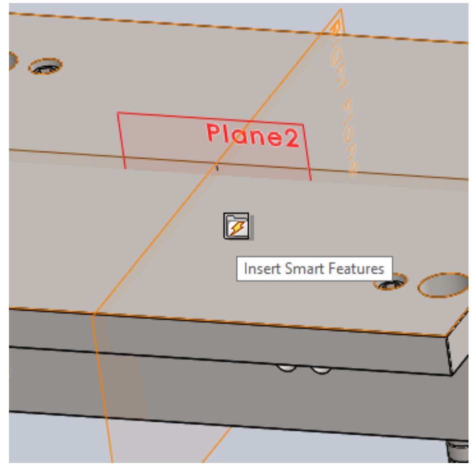


Inserting the Smart Component

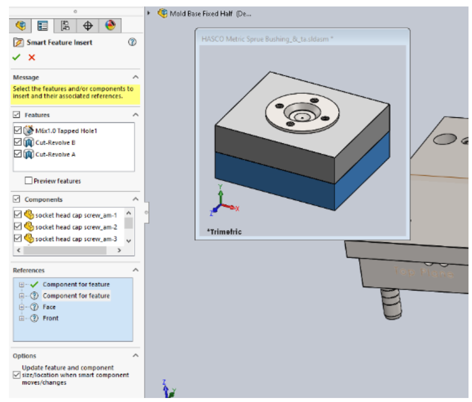
Smart Components are inserted into an assembly using the same techniques as any other component.

After the Smart Component has been added to the assembly and mated, the smart features and associated components can be added. This is accomplished using the references and selections made in the defining assembly.

Select the Component in the FeatureManager Design Tree and click **Insert Smart Features**.

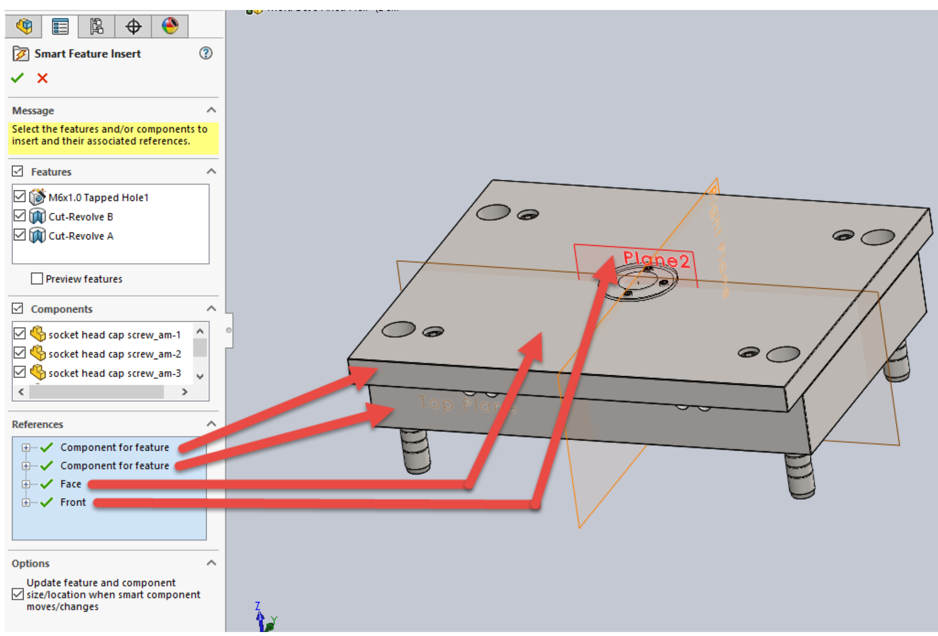


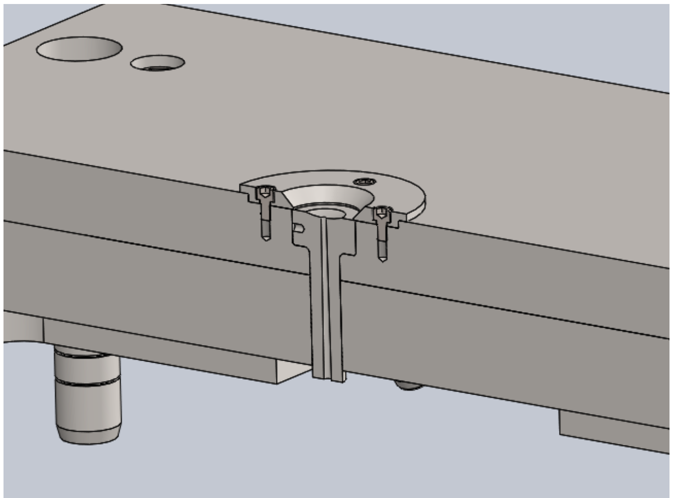
The preview pane will show the corresponding references to select on the assembly.



The **Features** & **Components** associated with the Smart Component will auto populate.

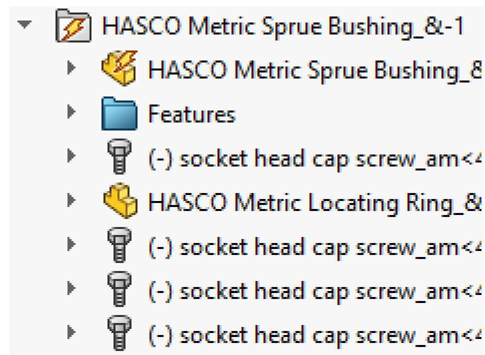
The **References** selection pane is used to define the placement.





The smart component is added with the features and components.

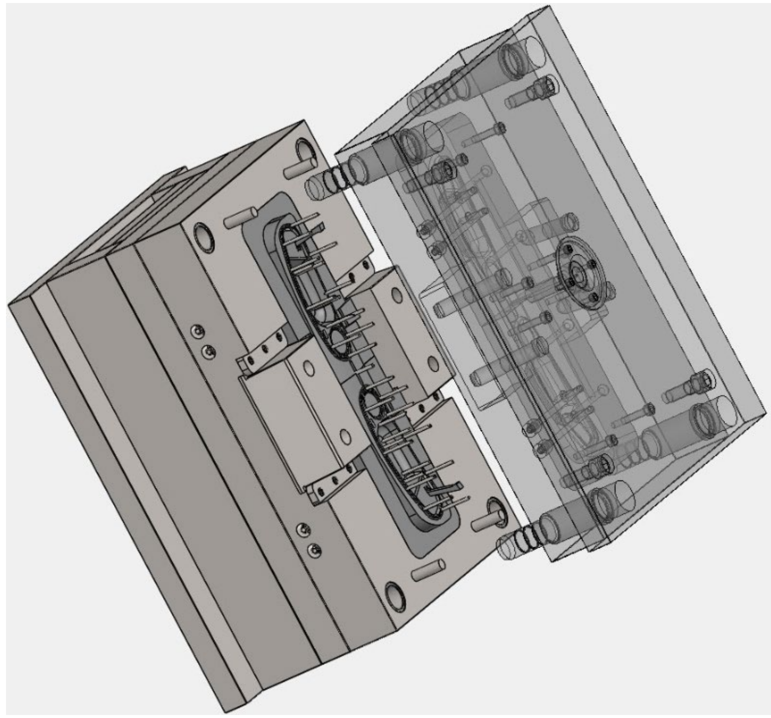
The FeatureManager Design Tree groups the smart component and related items in a new folder.



Lesson 9: More Mold Design Workflows

Injection molds typically need multiple ejector pins to push a part out of a mold. In many cases, the ejector pins are cut to many different lengths where they are flush to the molded part geometry. It would be very time consuming to modify them individually.

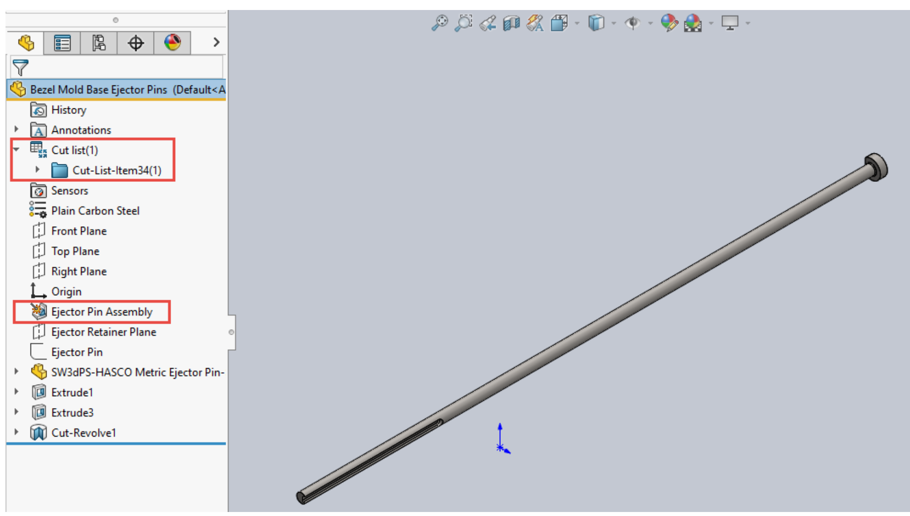
It is possible to place all the ejector pins into a single multibody part allowing them to be modified all at one time.



The ejector pin model in this case is a weldment.

The weldment feature is renamed Ejector Pin Assembly.

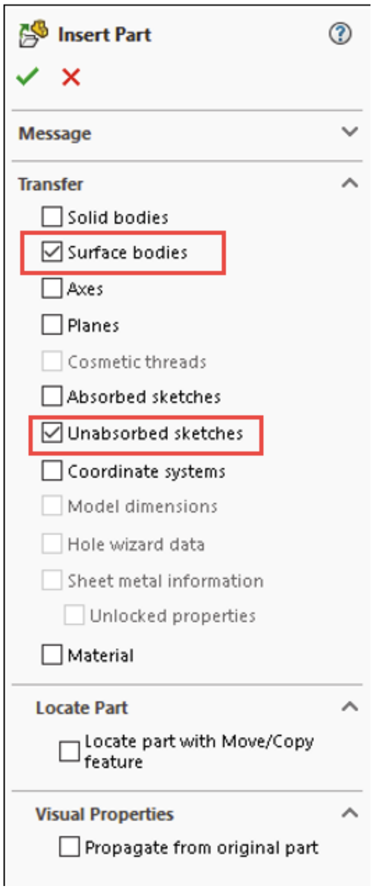
The cut list can serve as a BOM for the multibody part.



The default behavior for new features in a weldment is that they are created as independent bodies, rather than merging with surrounding geometry.

This workflow will require geometry to be used to trim the ejector pins. Inserting the molded part will provide this along with the ejector pin pattern sketch.

Insert Part



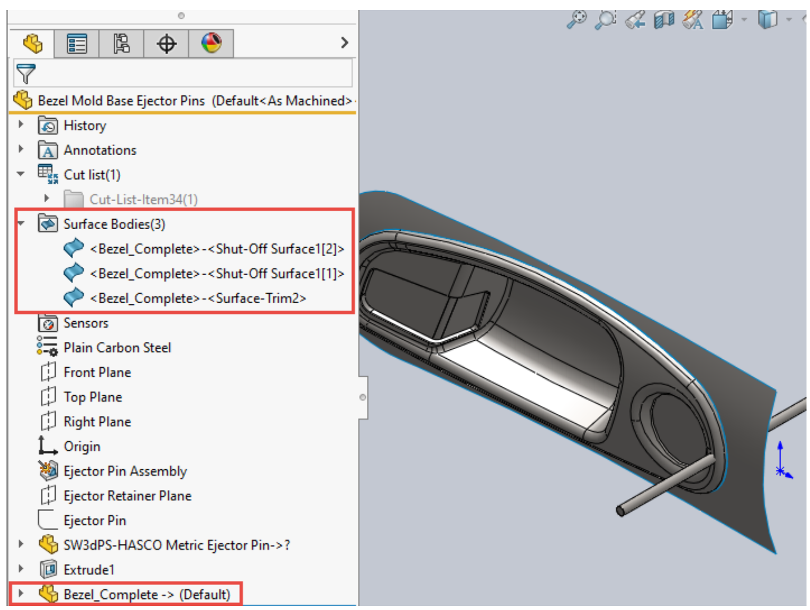
Use **Insert Part** whenever other part geometry is needed in a multibody part.

There are options to choose from in the **Insert Part** command that allows the user to only bring in the surface bodies and unabsorbed sketches.

Transfer - Inserting only the **Surface bodies** will allow the user to select the cutting surface to trim the ejector pins with.

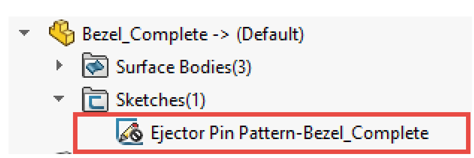
Inserting the **Unabsorbed sketches** will bring in the sketch pattern for the pin locations.

The inserted surfaces will be in the **Surface Bodies** folder. **Hide** the surface bodies that are not needed.

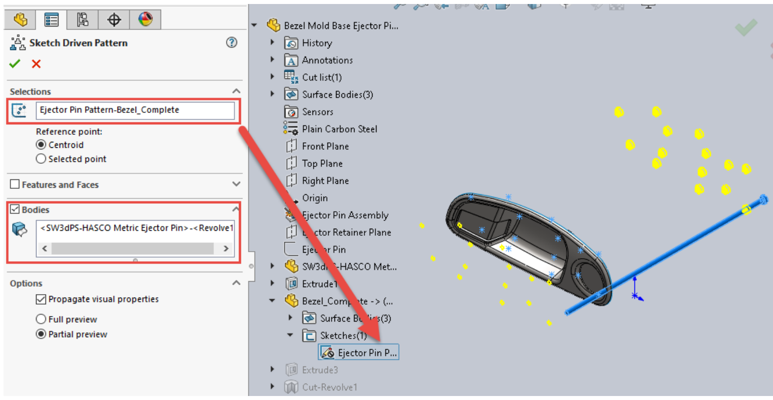


Sketch Driven Pattern

To capture all the ejector pins, use a **Sketch Driven Pattern** to copy the single pin to all the ejector pin locations.



Select the **Ejector pin pattern sketch**.



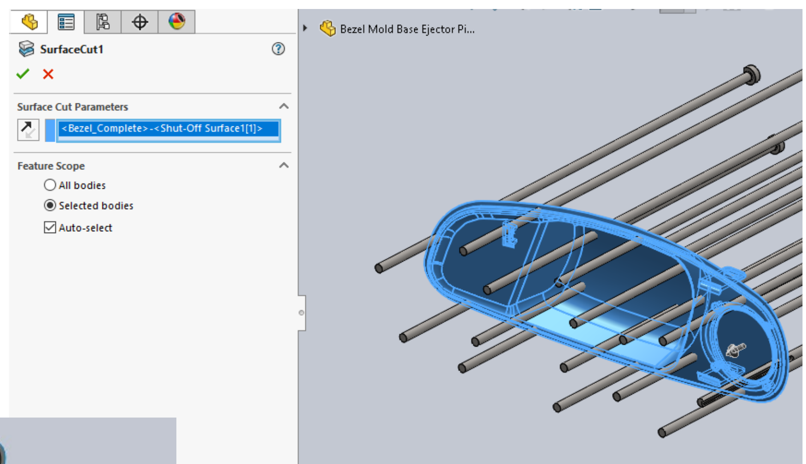
Use the **Bodies** to copy the selection pane.

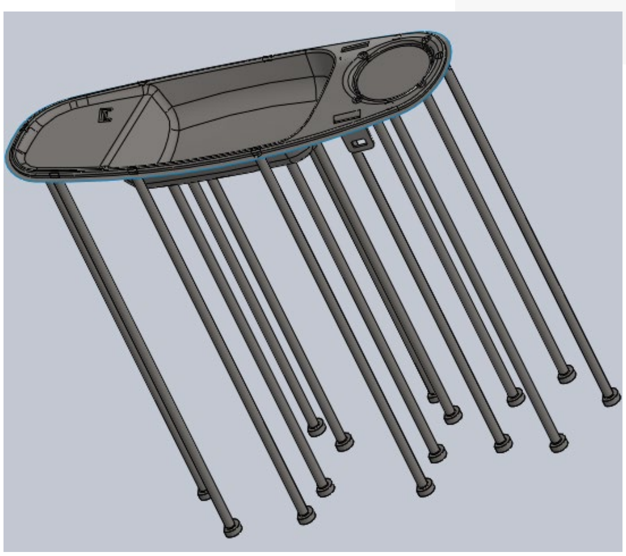
Cut with Surface

Use **Cut with Surface** to trim off the tops of the pins.

The imported surface is used as the cutting surface selection.

This will trim all the pins to this surface in one step.

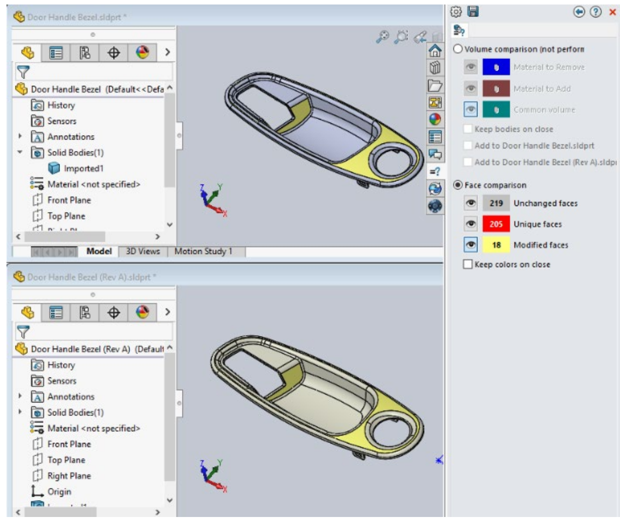




Making Changes

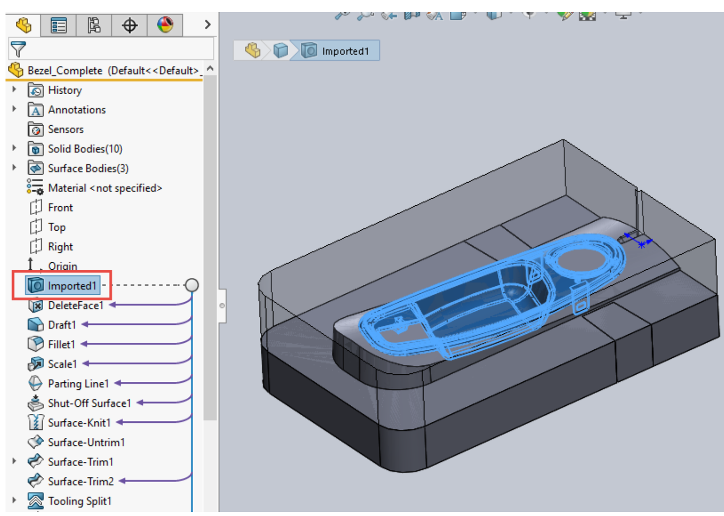
It is not uncommon to have a design change after all the mold design work is done. Incorporating the changes in a part design is straightforward if the mold design was done parametrically using SOLIDWORKS Mold tools.

The key steps are:

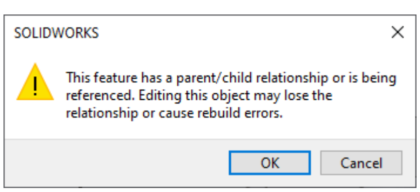


- **Determine the model changes:** Use the **Compare** tool to compare geometry between two different part models. This can give a sense of what has changed and what might need to be done to the mold.
- **Import the new model:** The new model can be imported directly into the existing mold file.
- **Fix rebuild errors:** Once the model has been replaced, SOLIDWORKS will try to rebuild the mold with the new part geometry. If SOLIDWORKS cannot repair everything, then manual methods must be used.

Importing the New Geometry



To replace the existing geometry, the original file used to create the tooling split in SOLIDWORKS Mold tools must be used.
With this file open, right-click on the **Imported part** feature and click **Edit feature**.

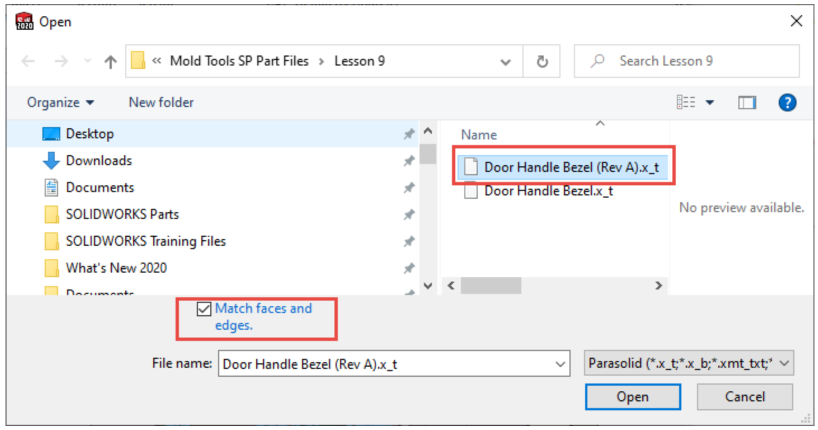


Accept the parent/child relationship warning.

Select the new part file that is replacing the original file.

Check the **Match faces and edges** option to align the part.

Select **Open** to replace the geometry and import the new file.



The process of replacing the geometry is complete. This will propagate the part revision all the way to the final tool design and drawings.