

SolidWorks® 2006

Advanced Part Modeling

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Introduction

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About This Course

The goal of this course is to teach you how to build freeform shapes using SolidWorks mechanical design automation software.

The tools for modeling advanced, freeform shapes in SolidWorks 2006 are quite robust and feature rich. During this course, we will cover many of the commands and options in great detail. However, it is impractical to cover every minute detail and still have the course be a reasonable length. Therefore, the focus of this course is on the fundamental skills, tools, and concepts central to successfully building multibody and freeform shapes. You should view the training course manual as a supplement to, not a replacement for, the system documentation and on-line help. Once you have developed a good foundation in the skills covered in this course, you can refer to the on-line help for information on less frequently used command options.

Prerequisites

Students attending this course are expected to have the following:

- Mechanical design experience.
- Completed the course *SolidWorks Essentials: Parts and Assemblies*.
- Experience with the Windows™ operating system.

Course Design Philosophy

This course is designed around a process-based (or task-based) approach to training. Rather than focus on individual features and functions, a process-based training course emphasizes the processes and procedures you follow to complete a particular task. By utilizing case studies to illustrate these processes, you learn the necessary commands, options and menus in the context of completing a design task.

Using this Book

This training manual is intended to be used in a classroom environment under the guidance of an experienced SolidWorks instructor. It is not intended to be a self-paced tutorial. The examples and case studies are designed to be demonstrated “live” by the instructor.

Laboratory Exercises

Laboratory exercises give you the opportunity to apply and practice the material covered during the lecture/demonstration portion of the course. They are designed to represent typical design and modeling situations while being modest enough to be completed during class time. You should note that many students work at different paces. Therefore, we have included more lab exercises than you can reasonably expect to complete during the course. This ensures that even the fastest student will not run out of exercises.

A Note About Dimensions

The drawings and dimensions given in the lab exercises are not intended to reflect any particular drafting standard. In fact, sometimes dimensions are given in a fashion that would never be considered acceptable in industry. The reason for this is the labs are designed to encourage you to apply the information covered in class and to employ and reinforce certain techniques in modeling. As a result, the drawings and dimensions in the exercises are done in a way that compliments this objective.

About the CD

Bound inside the rear cover is a CD containing copies of the various files that are used throughout this course. They are organized by lesson number. The *Case Study* folder within each lesson contains the files your instructor uses while presenting the lessons. The *Exercises* folder contains any files that are required for doing the laboratory exercises.

Windows® 2000 and Windows® XP

The screen shots in this manual were made using SolidWorks 2006 running on Windows® 2000 and Windows® XP. You may notice differences in the appearance of the menus and windows. These differences do not affect the performance of the software.

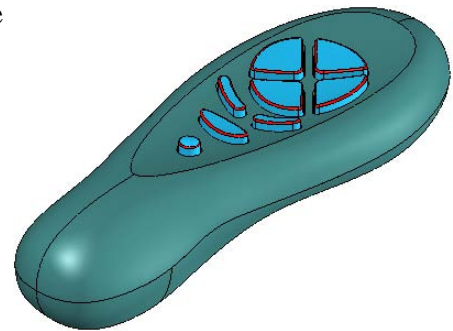
Conventions Used in this Book

This manual uses the following typographic conventions:

Convention	Meaning
Bold Sans Serif	SolidWorks commands and options appear in this style. For example, Insert, Boss means choose the Boss option from the Insert menu.
Typewriter	Feature names and file names appear in this style. For example, <i>Sketch1</i> .
17 Do this step	Double lines precede and follow sections of the procedures. This provides separation between the steps of the procedure and large blocks of explanatory text. The steps themselves are numbered in sans serif bold.

Use of Color

The SolidWorks 2006 user interface makes extensive use of color to highlight selected geometry and to provide you with visual feedback. This greatly increases the intuitiveness and ease of use of SolidWorks 2006. To take maximum advantage of this, the training manuals are printed in full color.



Also, in many cases, we have used additional color in the illustrations to communicate concepts, identify features, and otherwise convey important information. For example, we might show the result of an operation in a different color, even though by default, the SolidWorks software would not display the results in that way.

Lesson 1 Multibody Solids

Upon successful completion of this lesson, you will be able to:

- Create various multibody solids.
- Identify the different uses of a multibody solid.
- Combine solid bodies with add, subtract and common.
- Create an assembly from a multibody part.
- Modify a multibody cut using feature scope.

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Multibody Solids

Multibody solids occur when there is more than one continuous solid in the same part file. Often times, multibody techniques are useful for designing parts that require specific distance separation of features. These bodies can be accessed and modified separately and later merged into a single solid.

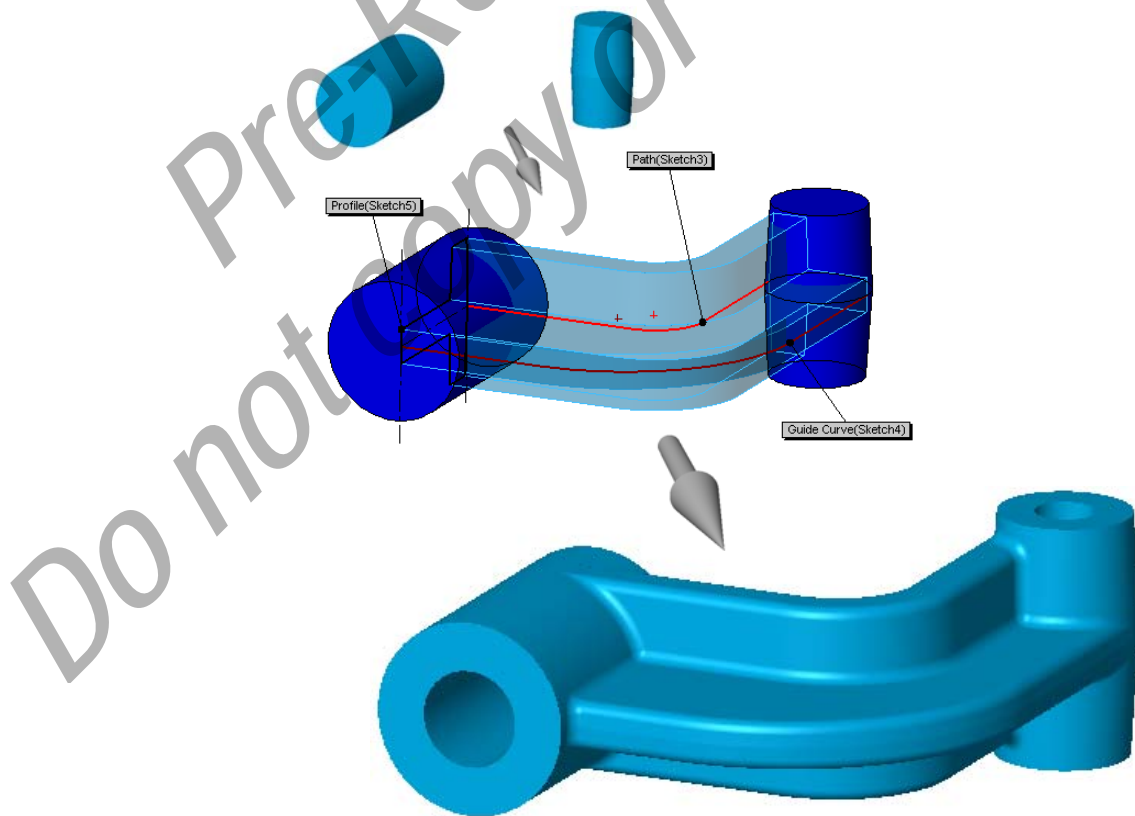
Creating a Multibody

Multibody solids are created in several ways. The following commands have the option of creating multiple solid bodies from a single feature:

- Extruded bosses and cuts (including thin features).
- Revolved bosses and cuts (including thin features).
- Swept bosses and cuts (including thin features).
- Lofted cuts.
- Thickened cuts.
- Cavities.

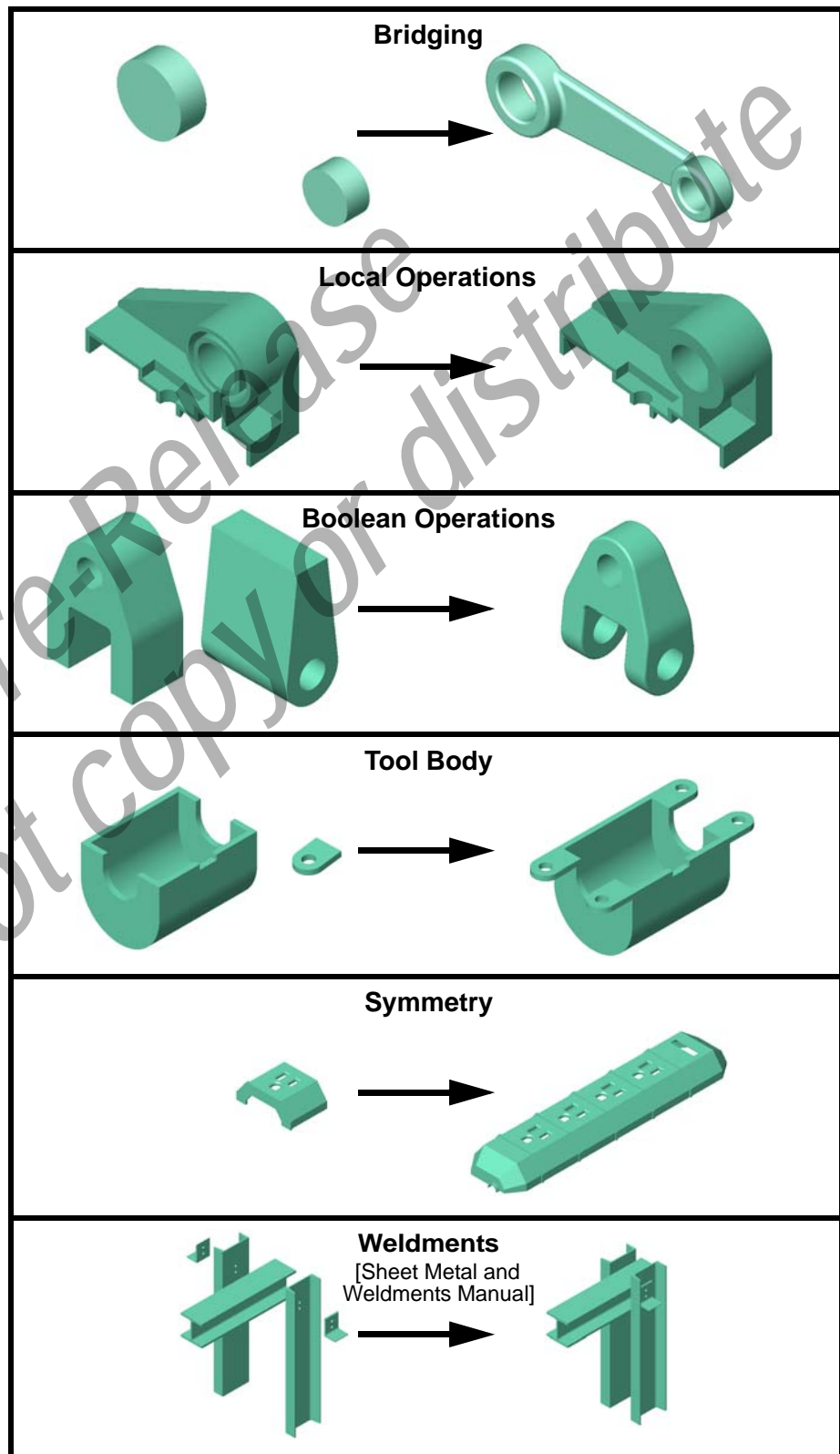
The most direct way to create a multibody solid is by *clearing* the **Merge result** check box for specific boss and cut features.

However, this option does *not* appear for the first feature.



**Multibody
Techniques**

There are several classes of parts that are well suited for working in the multibody solid environment. To accomplish successful multibody design, we will explore the following techniques:



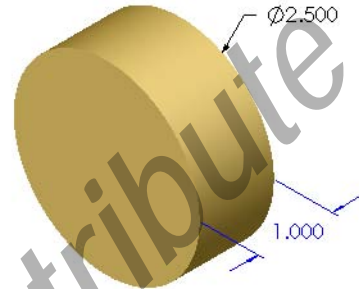
Bridging

The **Bridging** technique is used to build connecting geometry between multiple bodies. This example creates a multibody solid where multiple bodies are connected and merged by a new boss feature.

1 New part.

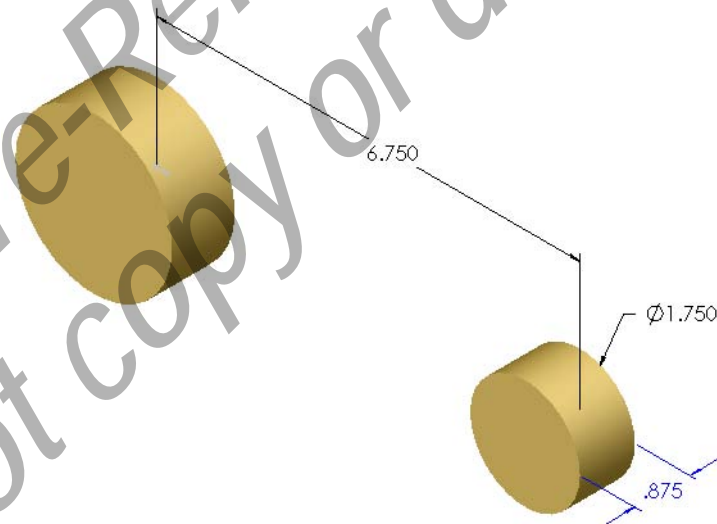
Create a new part with units set to inches.

Create a cylinder as the first feature using the **Front** reference plane as the sketch plane.



2 Create a multibody.

Create a second cylinder as shown.



Note

When boss features are created without intersecting the first feature, they are saved as multiple bodies. The **Merge result** check box remains checked by default, and the bodies will merge if they intersect through a later change.

Introducing: Solid Bodies Folder

The **Solid Bodies** folder holds all solid bodies in the part. Each solid body may be hidden from the folder. The names are taken from the last feature added to that body.

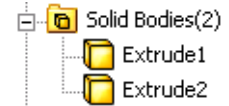
Where to Find It

- From the FeatureManager expand the **Solid Bodies** folder.



3 Explore the Solid Bodies folder.

The second cylinder causes the creation of another solid body. In the FeatureManager, expand the Solid Bodies folder to view these features.

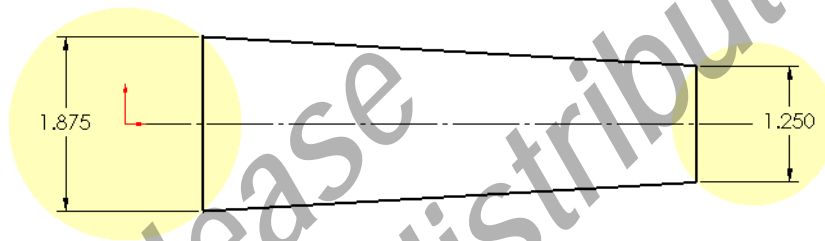


Note

If the part contains one solid, the folder will contain a single feature.

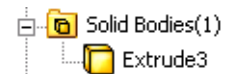
4 Create a bridge.

Create a boss using the edges of each cylinder.



Extrude the sketch **0.375"** and check **Merge result**.

The Solid Bodies folder now displays only one solid, Extrude3.



5 Finish the part.

Complete the part by adding the following features:

- Fillets = **0.125"**
- Cuts = **1.5"** and **1"** diameter
- Chamfers = **0.0625" x 45°**



Extrude From

The **Extrude From** option can be used with *extrusions* to move the starting position of a sketch by moving its “plane”. Options include:

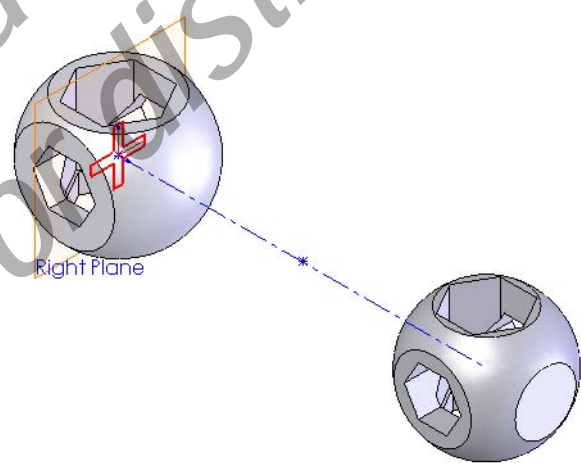
- **Sketch Plane**
The default sketch plane is used.
- **Surface/Plane/Face**
The sketch plane is moved to the selected surface, plane or (planar) face.
- **Vertex**
The sketch plane is positioned at the vertex or point.
- **Offset**
The sketch plane is offset parallel a specified distance.

1 Open the part **Extrude From.**

The part contains two ends of a wrench.

The Centerline sketch runs between bodies with a point at the midpoint of the line.

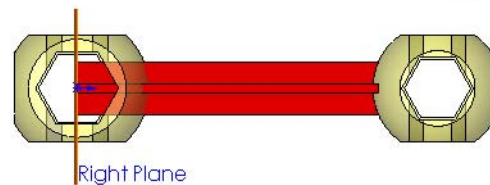
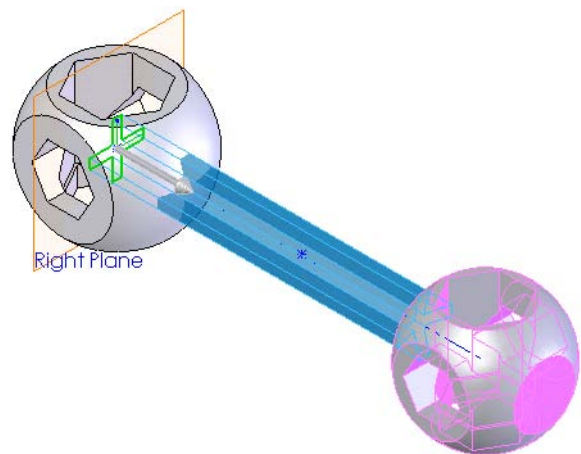
The Bridge Profile sketch is on the Right Plane.



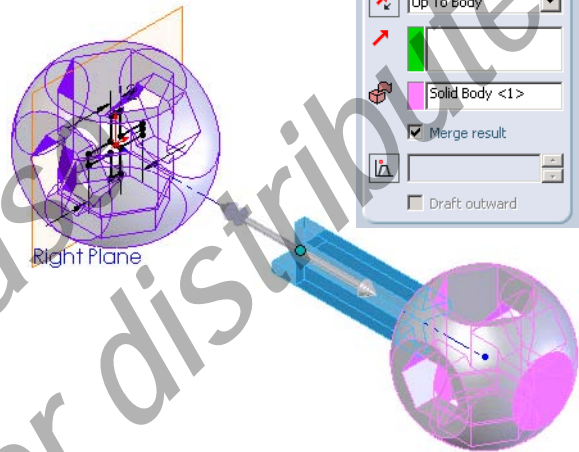
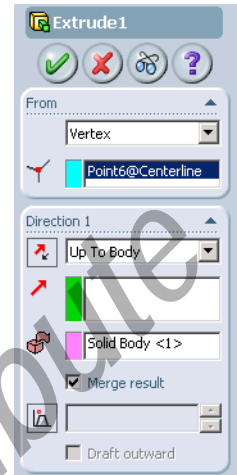
2 Up to body.

Extrude the sketch **Up To Body**, selecting the Right Sphere body as the end of the extrusion.

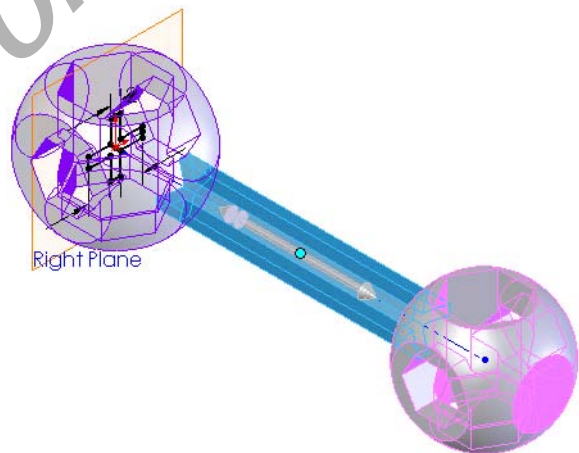
A problem occurs as the extrusion fills in some of the open volume of the Left Sphere.



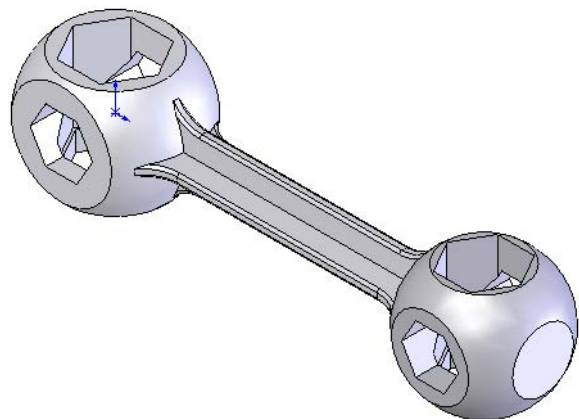
- 3 Extrude From.**
Use **Edit Feature** and expand the **From** group box. Choose **Start Condition: Vertex** and select the point in the Centerline sketch.



- 4 Direction 2.**
Select **Direction 2** and **Up To Body**, using the Left Sphere body.



- 5 Fillets.**
Adding **7mm** and **0.5mm** fillets completes the part.

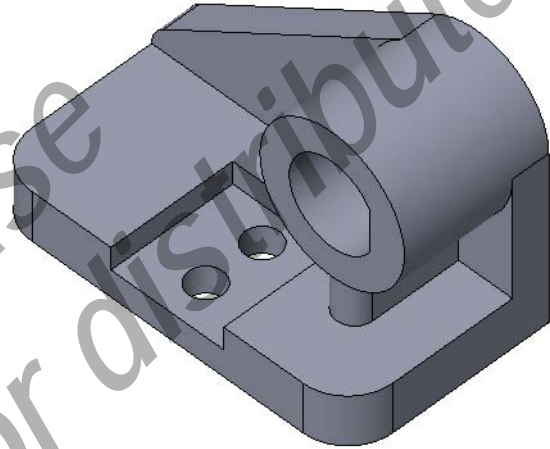


- 6 Save and close the part.**

Local Operations

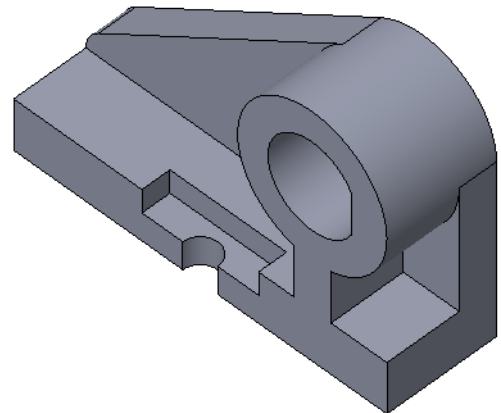
The **Local Operations** technique is used to make specific modifications on one body without affecting another body. A common example of this technique is a variation on shelling. The shelling operation, by default, affects all features of the solid body that precede it. In this example, a shelling problem will be solved using **Merge result** and **Combine**.

- 1 Open part.**
Open the part
Local Operations.

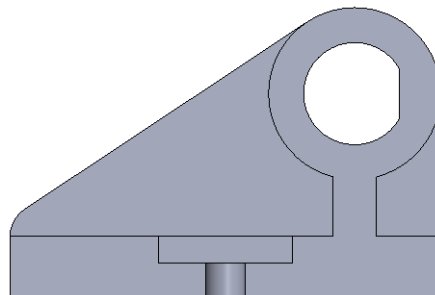


- 2 Create a section view.**
Use section view and this new plane to display how the shell command affects the entire part.

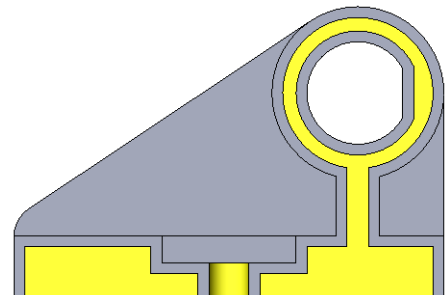
Create a section plane using
Offset Distance -42mm
from the Front Plane.



- 3 Shell part.**
Add a **4mm** shell that removes the bottom face.



Without shell feature



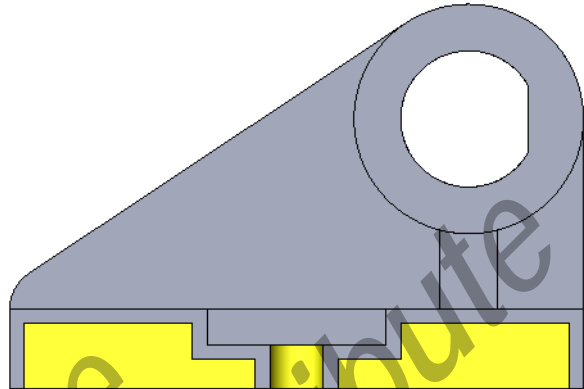
With shell feature

4 Modify each boss feature.

Use **Edit Definition** on these three bosses:

Vertical_Plate,
Circular_Boss,
and Rib_Under.

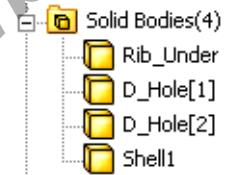
Clear the **Merge result** check box for each boss and click **OK**.



5 Explore the solid bodies.

After clearing the **Merge result** check box for each boss, the model breaks into four solid bodies.

Expand the **Solid Bodies** folder to view.

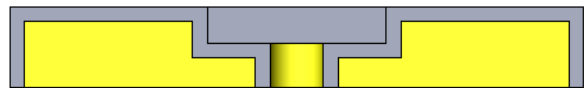
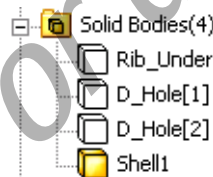


6 Hide solid bodies.

Hold down **Shift** and select:

Rib_Under,
D_Hole [1] and
D_Hole [2].

Right-click and select **Hide Bodies**.



The **Shell** body remains visible.

Combined Bodies

The **Combined Bodies** technique is used to create a single solid by adding, subtracting or intersecting the volumes of solid bodies.

Combine Tool

The **Combine** tool is used to combine the volumes of multibody solids into a single solid body. The bodies can be combined in different ways using different operations. The combine tool has three options:

■ **Add.**

The **Add** operation uses the **Bodies to Combine** list to merge the bodies into a single solid by adding all volumes. This operation is also known as a *union* in other systems.

■ **Subtract.**

The **Subtract** operation uses the **Main Body** and **Bodies to Combine** list to merge the bodies into a single solid by subtracting the bodies to combine from the main body.


- **Common.**

The **Common** operation uses the **Bodies to Combine** list to merge the bodies into a single solid by finding the volume that is common to all. This operation is also known as a *intersection* in other systems.


Where to Find It

- Click **Combine**  on the Features toolbar.
- Or, click **Insert, Features, Combine**.
- Or, select solid bodies and right-click **Combine**.

Note

Another method of selection is to filter solid bodies with the **Solid Bodies** filter .

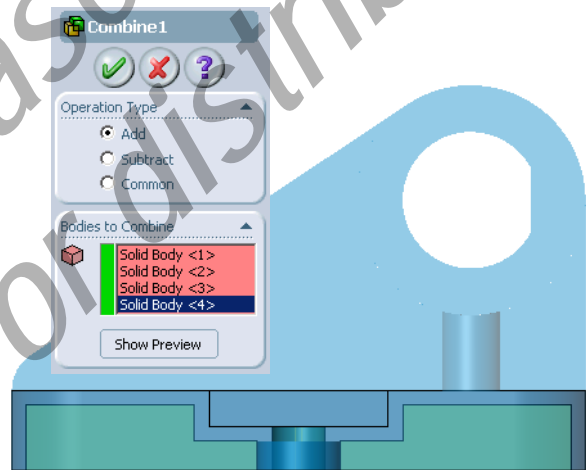
7 Combine the solid bodies.

Click **Combine**  on the Features toolbar.

Use the **Add** option for **Operation Type**.

Select all four bodies from the Solid Bodies folder for **Bodies to Combine**.

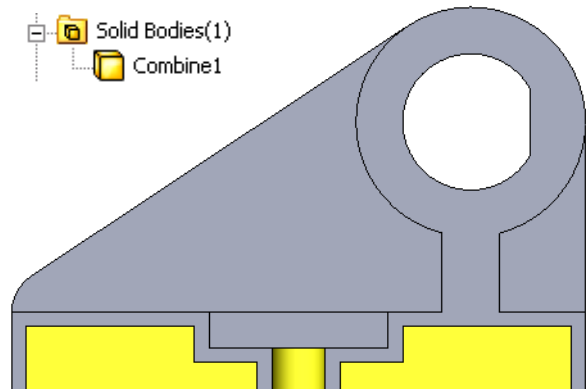
Click **Show Preview** and **OK**.



8 Explore the single solid.

The part now exists as a single solid body **Combine1**.

The name is assumed from the last feature added to the body.



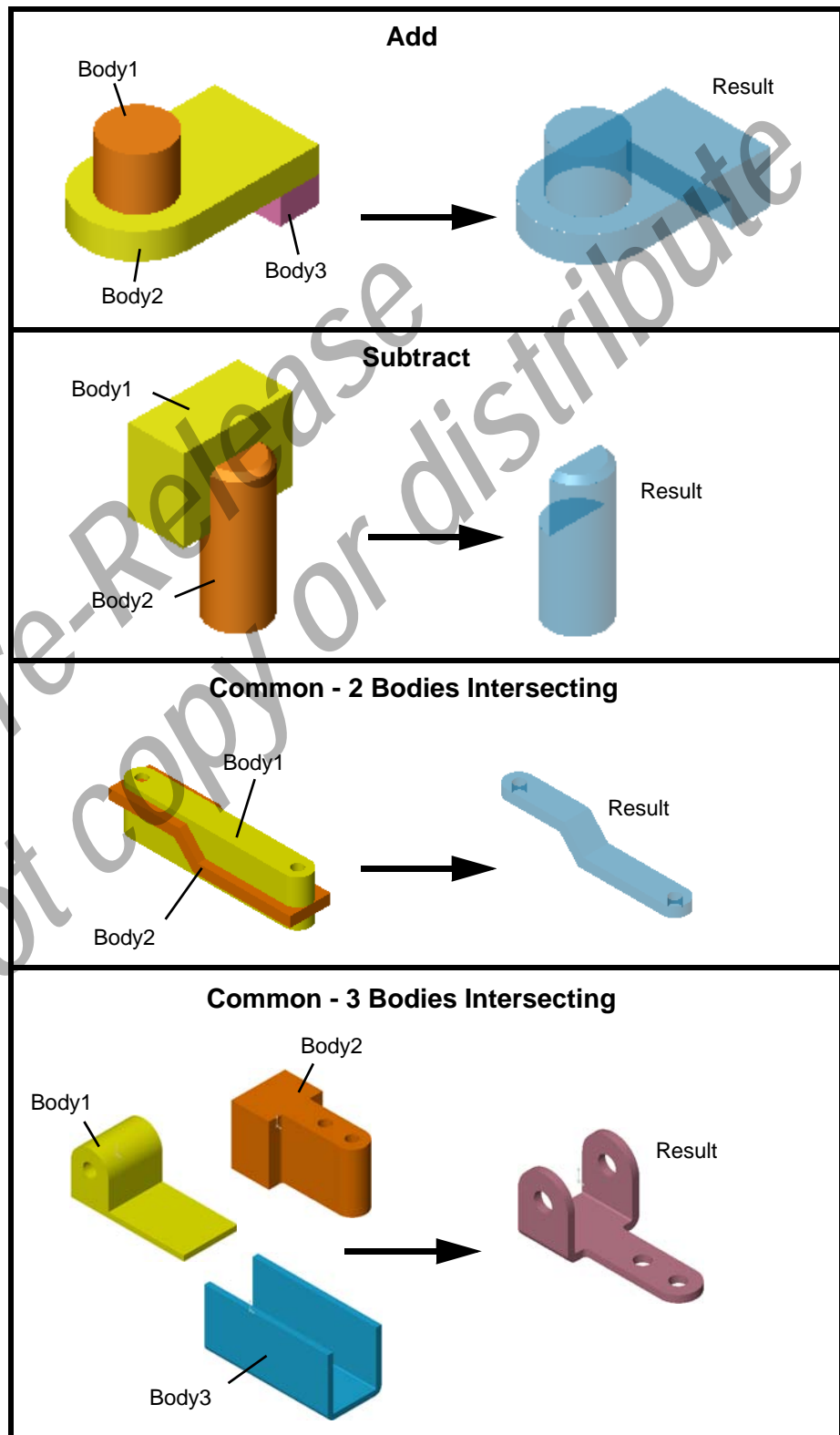
Tip

Features, such as fillets, using the edges formed by merged solid bodies, will fail if **Merge result** is unchecked in a later operation. The following rebuild error will appear:

Fillet1: Multiple bodies not supported for this feature.

Examples of Combined Solids

The following table displays the results from various combining techniques available.



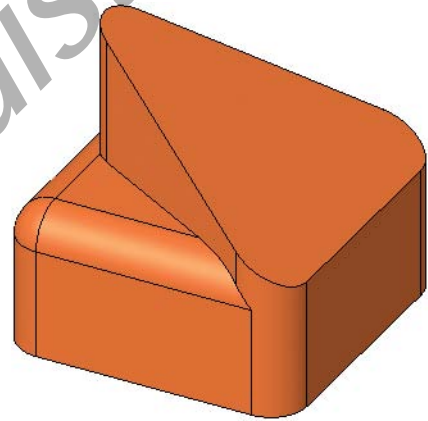
Using Local Operations to Solve Filleting Problems

Many times success in filleting depends on the order in which you apply the fillets. Multibody solids and local operations give you the ability to alter the sequence in which fillets are applied. This can be very helpful with particularly difficult filleting problems.



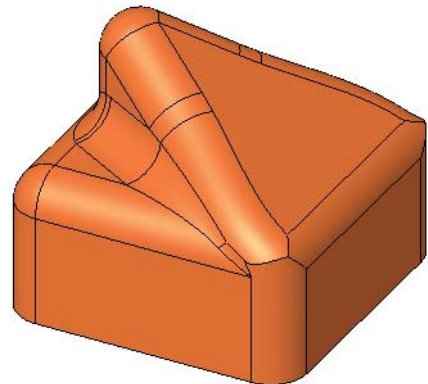
Thanks to Keith Pedersen at Computer-Aided Products, Inc. for submitting this example.

-
- 1 Open the part named **Fillet Problem**.



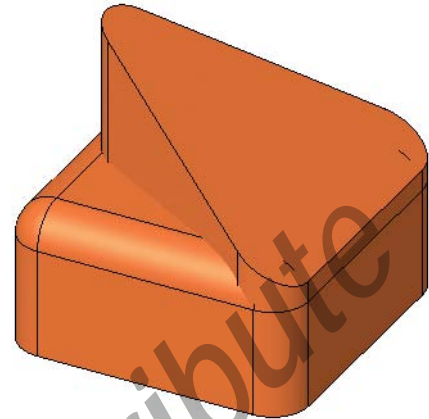
- 2 **Attempts at filleting.**

Various attempts to apply a 0.25" fillet do not yield satisfactory results. This is because the fillets are affected by adjacent faces. The solution is to fillet the bodies separately.

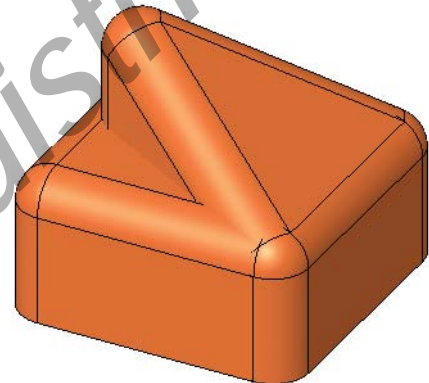



- 3 Unmerge the solids.**
Right-click the Angled Piece feature and select **Edit Feature**.

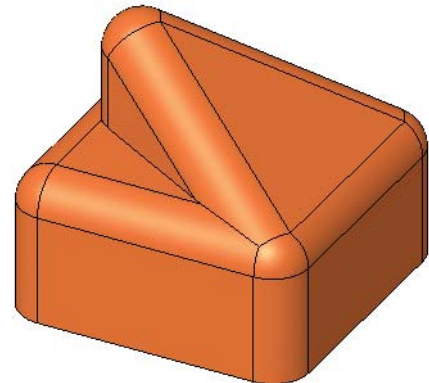
Clear the **Merge results** check box and click **OK**.



- 4 Fillet the Angled Piece feature.**
Apply a 0.25" fillet to the uppermost face of the Angled Piece.

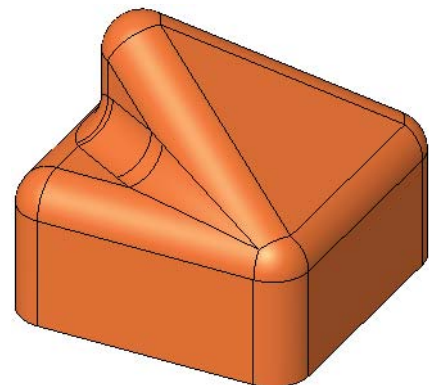


- 5 Combine the solids.**
Click **Combine**  on the Features toolbar.
Merge the two solids using the **Add** option.
Click **OK**.



- 6 Fillet.**
Apply the remaining 0.25" fillet as shown.

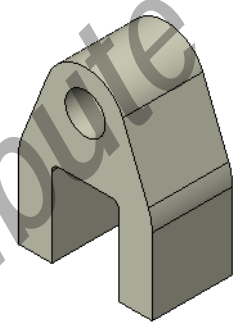
- 7 Save and close the part.**



Common Bodies

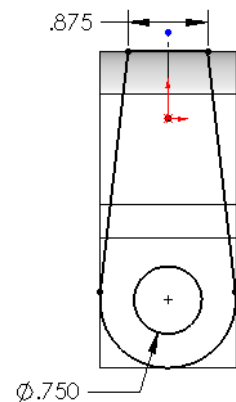
There are a variety of ways to combine multiple bodies into a single solid body. This example uses one of the more appealing options, **Common**, or intersecting volumes.

- 1 **Open the part Combine1.**



- 2 **Create a sketch.**

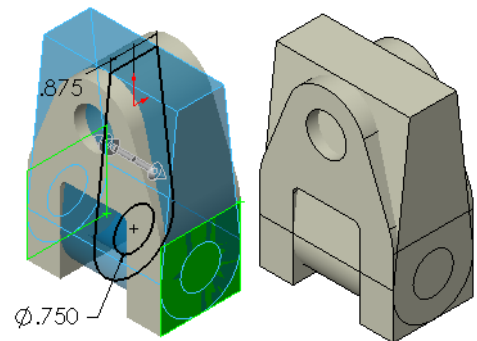
Using the **Right** reference plane, sketch the profile shown.




- 3 **Create boss extrusion.**

Extrude the sketch using the **Up To Surface** end condition for each direction.

Make sure that **Merge result** is cleared.

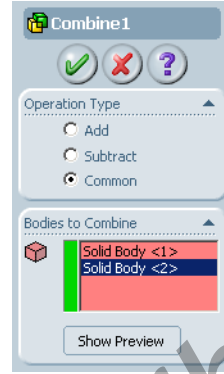


4 Combine the solid bodies.

Click **Combine**  on the Features toolbar.

Use the **Common** option for **Operation Type** and select both bodies.

Click **Show Preview** and **OK**.



5 Complete the part.

Add **1/16"** fillets to complete the part.

6 Save and close the part.

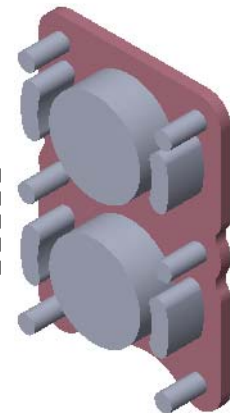
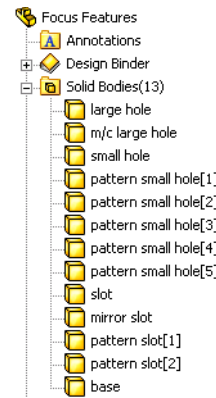


Focus on Features

Consider a part where the placement of holes, relative to each other and the origin, are of primary importance. In this example, the “plate” that the holes remove volume from is based on *their* location.

1 Open the part Focus Features.

The part contains multiple solid bodies.

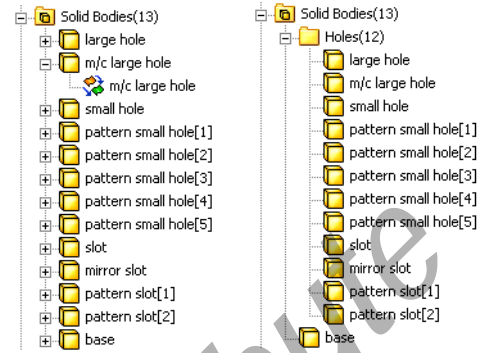


**Solid Bodies
Folder Options**

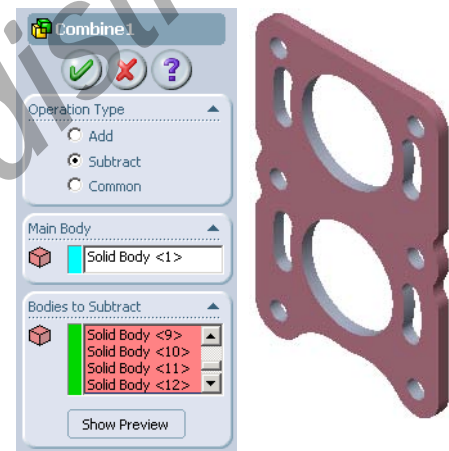
The appearance of the **Solid Bodies** folder can be modified for ease of use.

Right-click the **Solid Bodies** folder (left) and choose **Show Feature History** to see the features used to create the body.

Select one or more features (right) and right-click **Add to New Folder** to place them in a user defined folder. The folder includes a count of the solid bodies.

**2 Combine the solid bodies.**

Using the base as the **Main Body** and the remaining solid bodies as the **Bodies to Subtract**, combine with a **Subtract** operation.

3 Save and close the part.**Tool Body**

The **Tool Body** technique is used to add or remove model volume using specialized “tool” parts.

**Introducing:
Insert Part**

You can use the **Insert Part** tool to add one or more solid bodies into the active part, placing the origin of the inserted part on that of the active part. The inserted parts are then oriented using the **Locate Part** dialog.

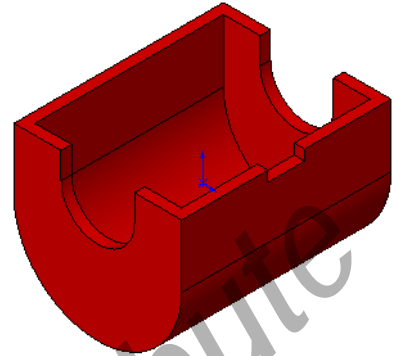
Where to Find It

- Click **Insert, Part**.
- Or, click **Insert Part**  on the Features toolbar.

1 Open part.

Open the part Cover without Tabs.

Rollback to just *before* the Fillet1 feature.

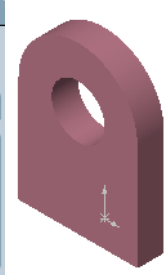
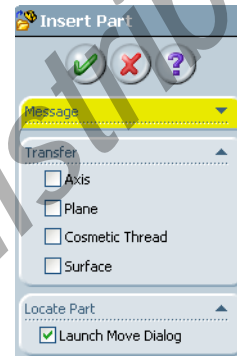


2 Insert a part.

Click **Insert, Part** and select the part Tool Body Tab.

Make sure **Launch Move Dialog** is checked and click **OK**.

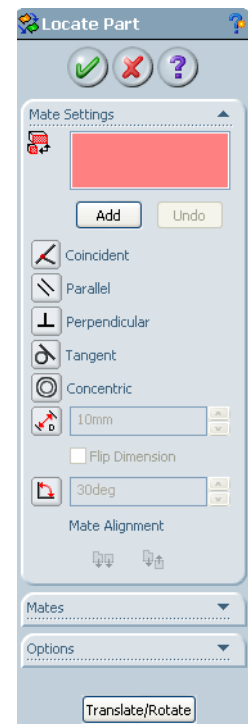
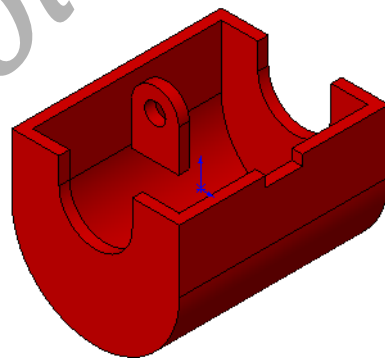
The part being inserted is simply a standard part file.



3 Results.

The **Locate Part** menu appears and an instance of the Tool Body Tab is added to the active part.

The **Insert Part** command inserts an instance of a part into another part. Multiple parts and/or multiple instances of the same part can be inserted into the active part.



**Introducing:
Move/Copy Bodies**

Use **Move/Copy Bodies** to orient solid bodies within a part. Bodies can be moved using two different methods:

1. Mates, similar to the way components are mated in an assembly
2. Specifying translation and/or rotation with respect to the X, Y, and Z axes.

The **Locate Part** dialog is the same as the **Move/Copy Bodies** dialog.

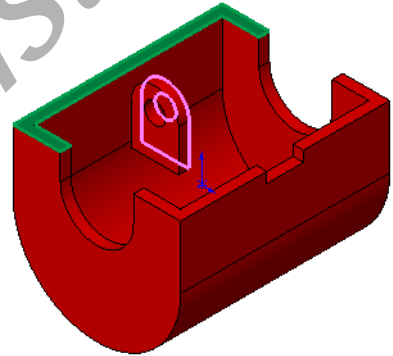
Where to Find It

- Click **Insert, Features, Move/Copy**.
- Or, click **Move/Copy Bodies**  on the Features toolbar.

This example illustrates using mates to locate the solid body. For an example using explicit translation and rotation, see *Exercise 6: Using Indent* on page 59.

4 Select the faces.

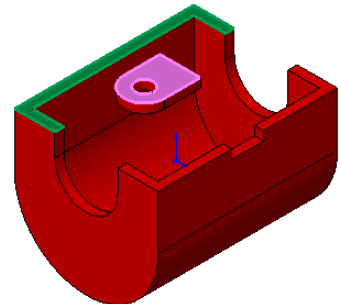
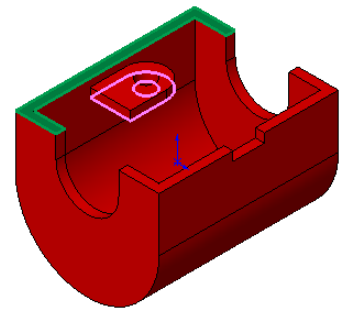
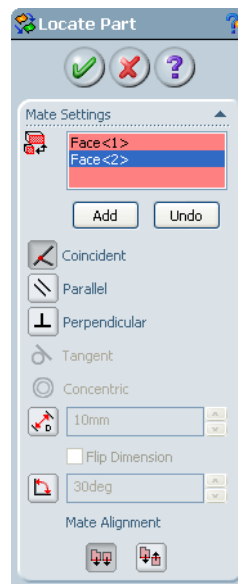
Select the rear face of the tab and the upper face of the cover as shown.

**5 Mate the body.**

The system selects **Coincident** as the default mate type. In cases when this is not what you want, you can select a different type.

Verify the orientation of the **Tool Body Tab**. If it is upside down, as in the upper picture, change the **Mate Alignment**.

Coincident mates can be either **Aligned** or **Anti-Aligned**.

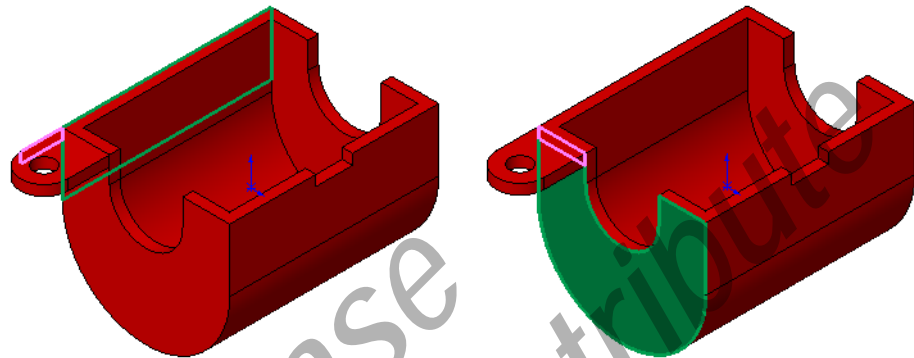


Click **Add** to apply the mate.

For more information about mates, see the *Essentials: Parts and Assemblies* manual.

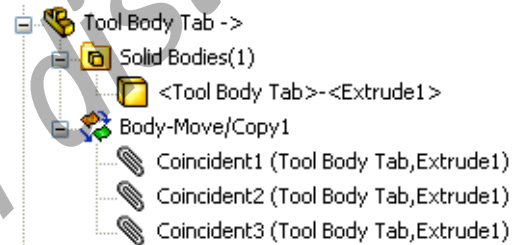
6 Additional mates.

Add two more **Coincident** mates, selecting the faces as shown in the illustration below. This completes positioning the tab.



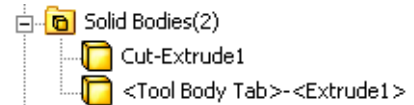
7 Examine the feature.

Expand the Tool Body Tab feature listing. The feature representing the **Locate Body** command is listed as a child of the Tool Body Tab.



8 Explore the solid bodies.

A second solid body is listed in the folder.



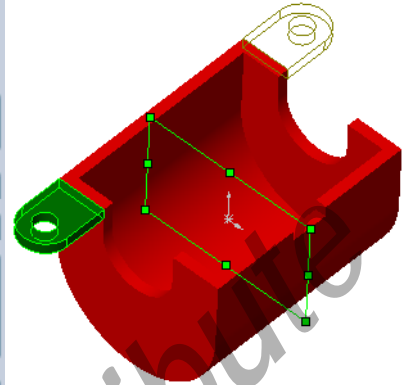
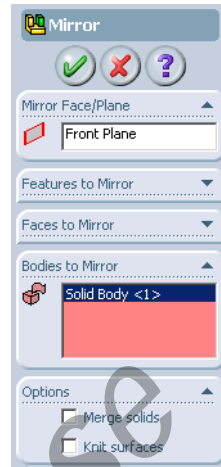
Patterning Bodies

Each pattern feature can be used to create instances of solid bodies. The **Bodies to Pattern** field is used to set which body or bodies will be patterned.

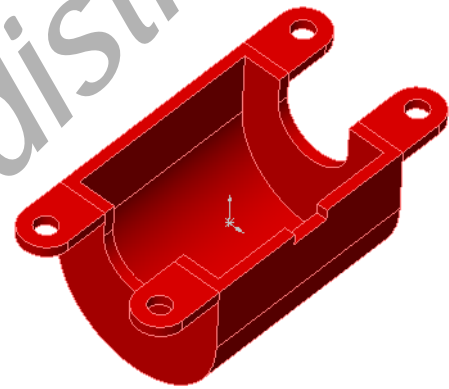
The **Bodies to Pattern** field exists in the following pattern tools:

- Linear.
- Circular.
- Mirror.
- Table Driven.
- Sketch Driven.
- Curve Driven.

- 9 Mirror body.**
Insert a **Mirror** pattern using the Front reference plane and the Tool Body Tab as the body to mirror.
- Keep **Merge solids** cleared.
- Click **OK**.



- 10 Mirror again.**
Mirror both tabs, this time using the Right reference plane as shown.



- 11 Combine the solid bodies.**
Combine the solid bodies into one using **Add**.

- 12 Save and close the part.**



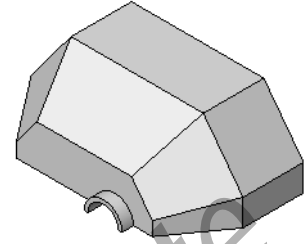
Symmetry

The **Symmetry** technique is used to help create parts faster by using patterns. In this example, solid bodies, rather than features, are patterned and combined.

1 Open part.

Open the part Symmetry.

It contains the part PowerCordEnd as a feature.

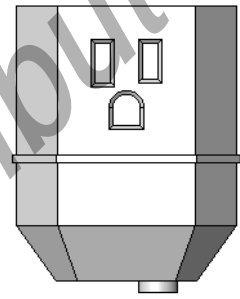


2 Insert a part.

Insert the part PowerBlock.

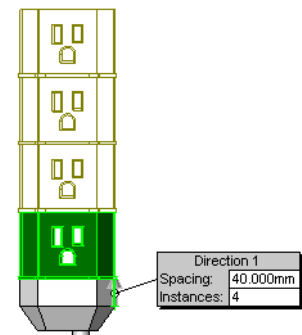
Clear the **Launch Move Dialog** check box, and click **OK**.

It drops in the proper position, the Origin, by default.



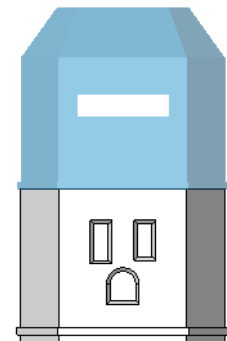
3 Create a linear pattern.

Using the **Linear Pattern** tool, create **4** instances of the solid body PowerBlock **40mm** apart.

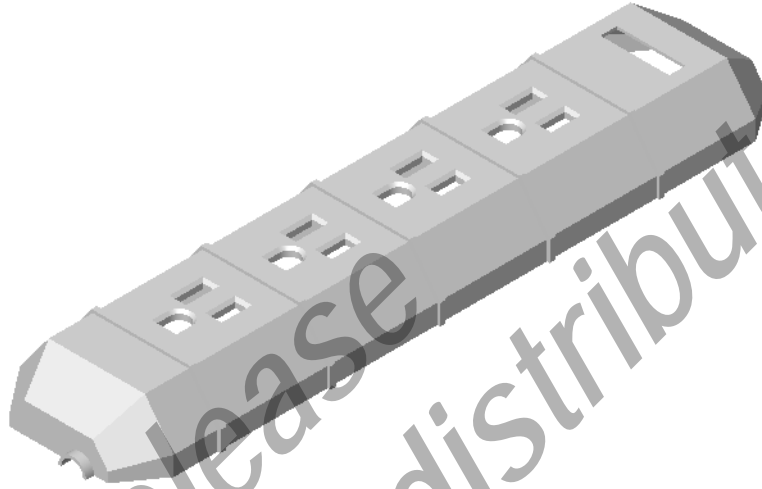


4 Insert and locate part.

Insert the part PowerSwitchEnd and mate it to the last instance in the linear pattern.



- 5 **Combine the solid bodies.**
Combine the solid bodies into one using **Add**.



- 6 **Save and close the part.**

Indent Feature

The **Indent** feature is used to reshape thin walls of the **Target Body** to the shape of one or more intersecting **Tool Bodies**. The indentation thickness and optional clearance can be controlled by numeric values.

- **Target Body**
The **Target Body** is the body being indented.
- **Tool Body Region**
The **Tool Body Region** is a selection of both a solid body (tool) and a region as the tool body is divided by the target body.

Where to Find It

- Click **Indent**  on the Features toolbar.
- Or, click **Insert, Features, Indent**.

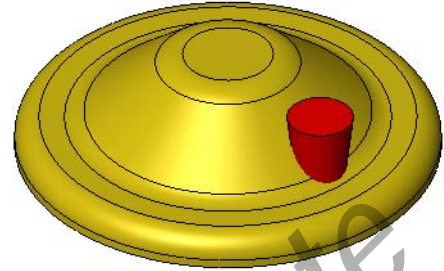
Using Indent

In this example, **Indent** is used to reshape an existing thin walled feature for a hole, fastener and clearance for tools. The selection of the tool body region determines to which side of the target body the indent feature is applied.




1 Open Indent1.

Open the part Indent1. It includes two intersecting solid bodies.

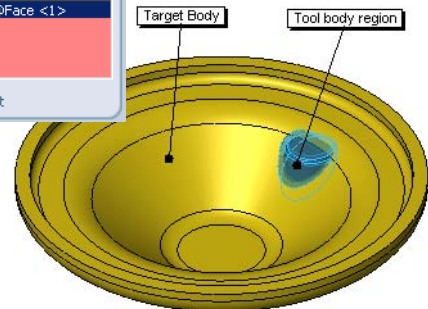
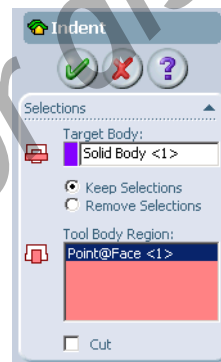


2 Target and tool.

Click **Indent**  and select the large solid as the **Target body**.

Select the revolved solid body as the **Tool body region**.

The blue preview shows the indent.

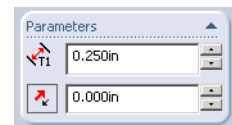


Important!

Make the tool body region selection from the *underside* of the target body as shown.

3 Parameters.

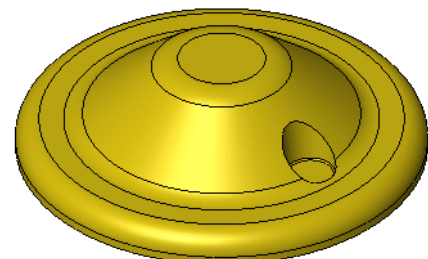
Under **Parameters**, set the **Thickness** to **0.25"** and **Clearance** to **0"** as shown.



Click **OK**.

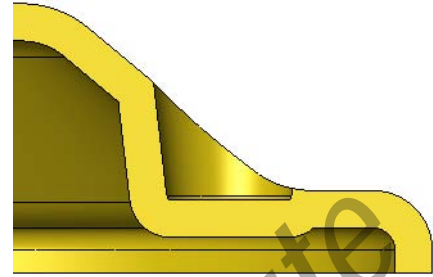
4 Hide body.

Hide the tool body to see the results.



5 Section View.

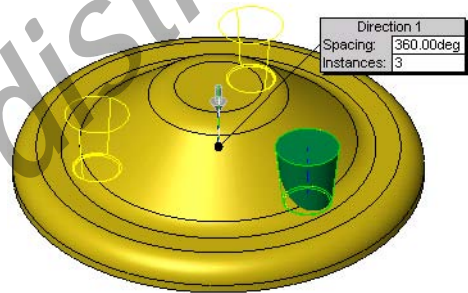
Use the **Section View** tool with the Front Plane to cut the display.

**Using Multiple Tool Bodies**

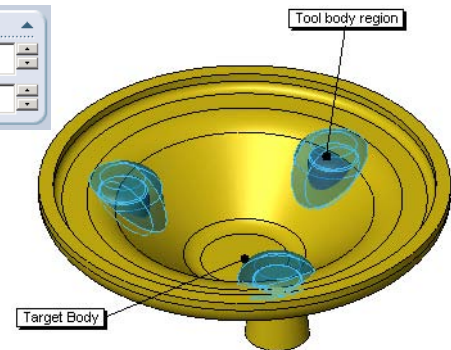
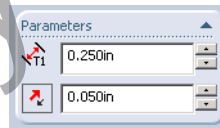
Multiple tool bodies can be used with **Indent**. Since the **Indent** feature cannot be patterned, the tool bodies themselves must be patterned.

6 Rollback and pattern.


Use **Roll to Previous** to position the rollback bar between the Fillet1 and Indent1 features. Add a **Circular Pattern** of the tool body as shown.

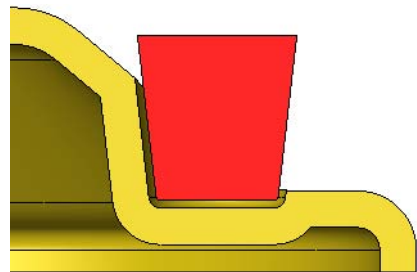
**7 Edit Feature.**

Use **Roll to End** and edit the Indent1 feature. Click in **Tool Body Region** and select the additional bodies as shown. Change the **Clearance** to **0.050"** and click **OK**.

**8 Section View.**

Use the **Section View** tool with the Front Plane to cut the display.

Note how the **Clearance** is applied. It can be reversed  if necessary.



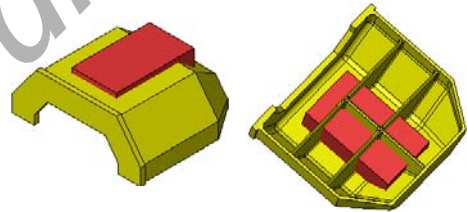
Indent with Multiple Target Regions

When the Target Body contains features (such as ribs) that subdivide the Tool Body, multiple target regions are created. In this example, the base of a blender is used as the target. A solid body representing the cup and a drain is used as the tool.

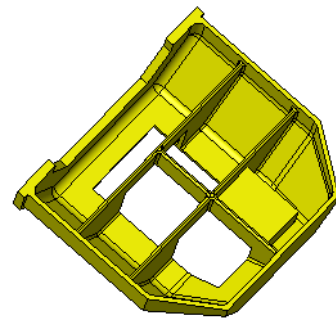
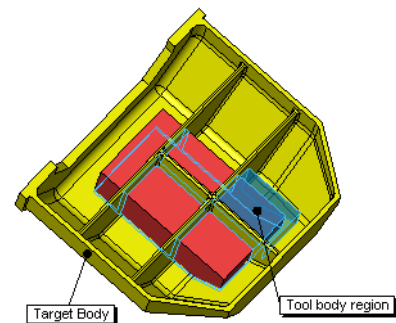
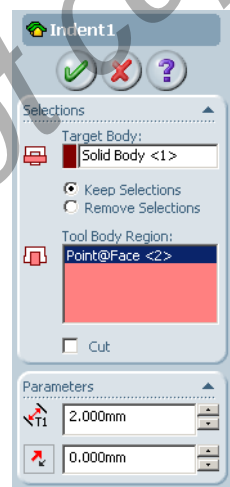


Target Selections and Options

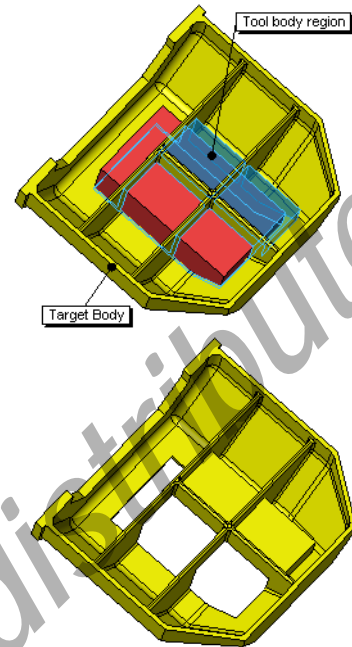
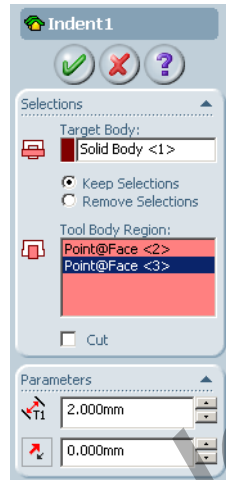
In the chart that follows, the yellow body represents the **Target Body** and the red block represents the **Tool Body**. In this example there are six possible **Tool Body Regions** to select.



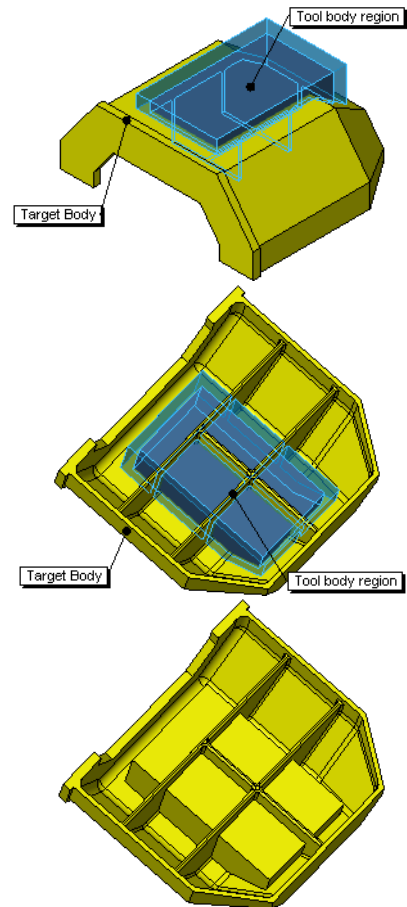
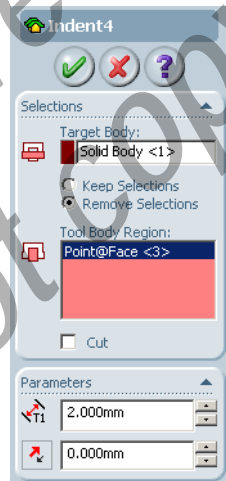
Single Tool Body Region with Keep Selections:

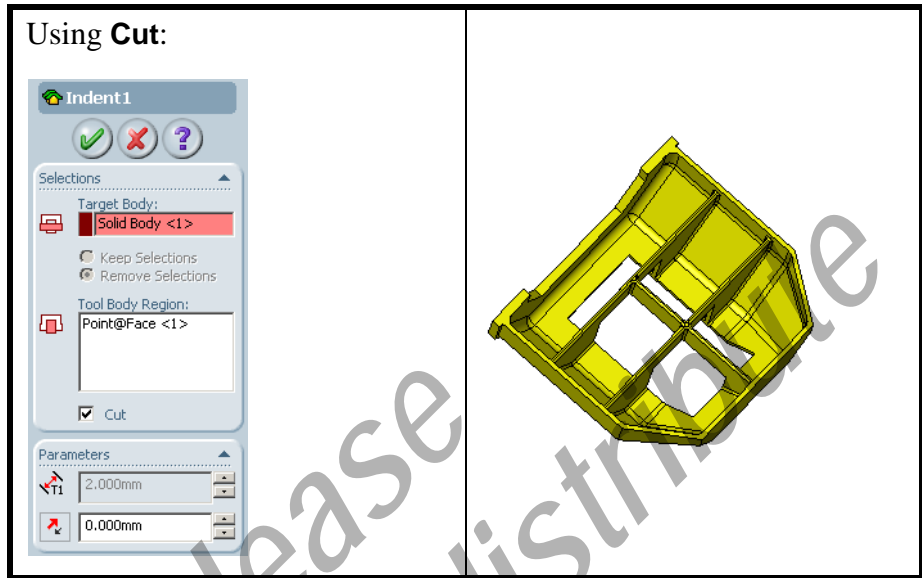


Multiple Tool Body Regions with Keep Selections:



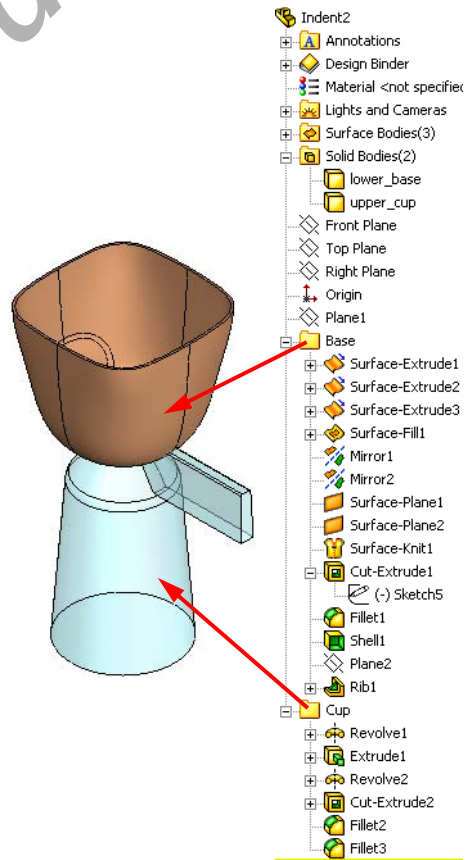
Single Tool Body Region with Remove Selections:






1 Open part Indent2.

The Indent2 part is a multi-body part with solid bodies (lower_base and upper_cup) representing a blender base and mixing glass. The features used to make the base are collected in the folder base. The cup folder contains features for the other body.




2 Interference between bodies.

Select the solid bodies and use **Combine**  with the **Common** option to view the interfering volume of geometry. Use the **Show Preview** button and click **Cancel** to avoid adding the feature.

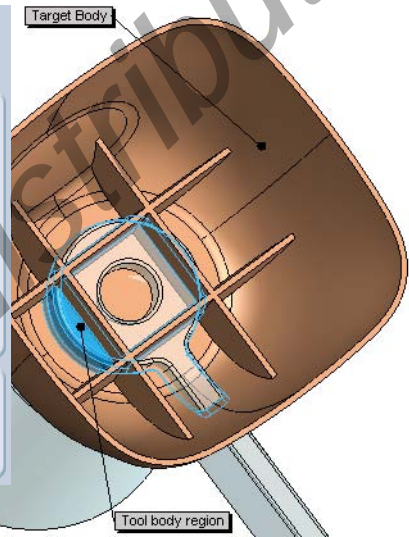
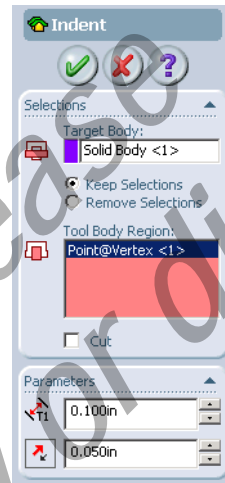
**Note**

The **Interference Detection** tool can only be used in an assembly.

3 Select Tool Body Region.

Click **Indent**  and select the **Target Body** as shown. Select the **Tool Body Region** by clicking the face indicated by the callout. The preview identifies the face and region to indent.

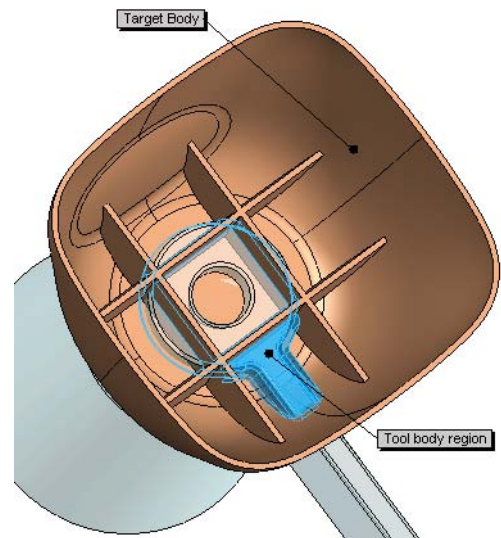
Set the **Thickness** and **Clearance** values as shown.

**4 Additional selections.**

Additional regions can be selected provided that the selections are unique faces.

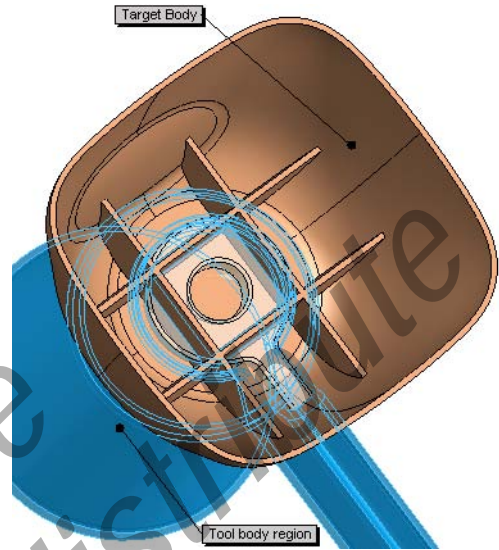
Selecting a face that has already been selected, even in a different region, deselects it.

In a case such as this, with multiple regions dividing the same faces, another approach is more efficient.



- 5 **Selections.**
Right-click in the **Tool Body Region** and select **Clear Selections**.

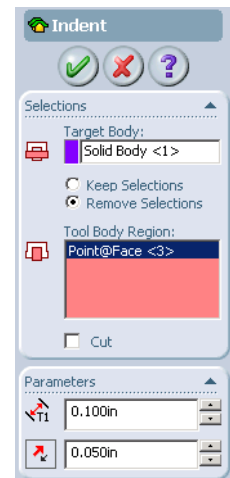
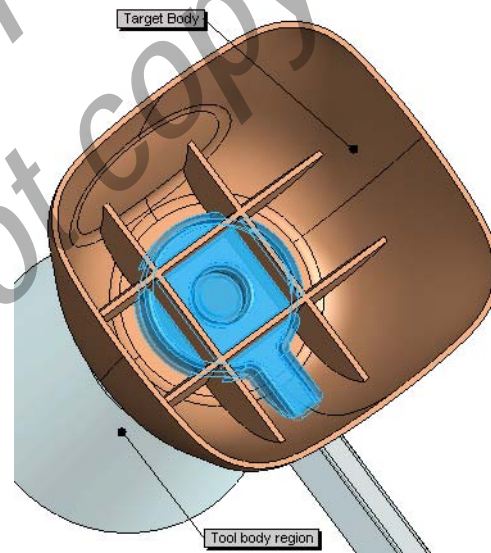
Select the region of the tool body *outside* the target.



- 6 **Using Remove Selections.**
Click **Remove Selections** to reverse the entity selection.

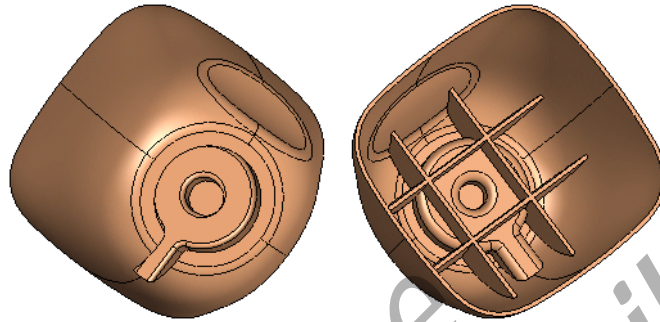
All regions of the tool body *inside* the target are now selected.

Click **OK**.



7 Results.

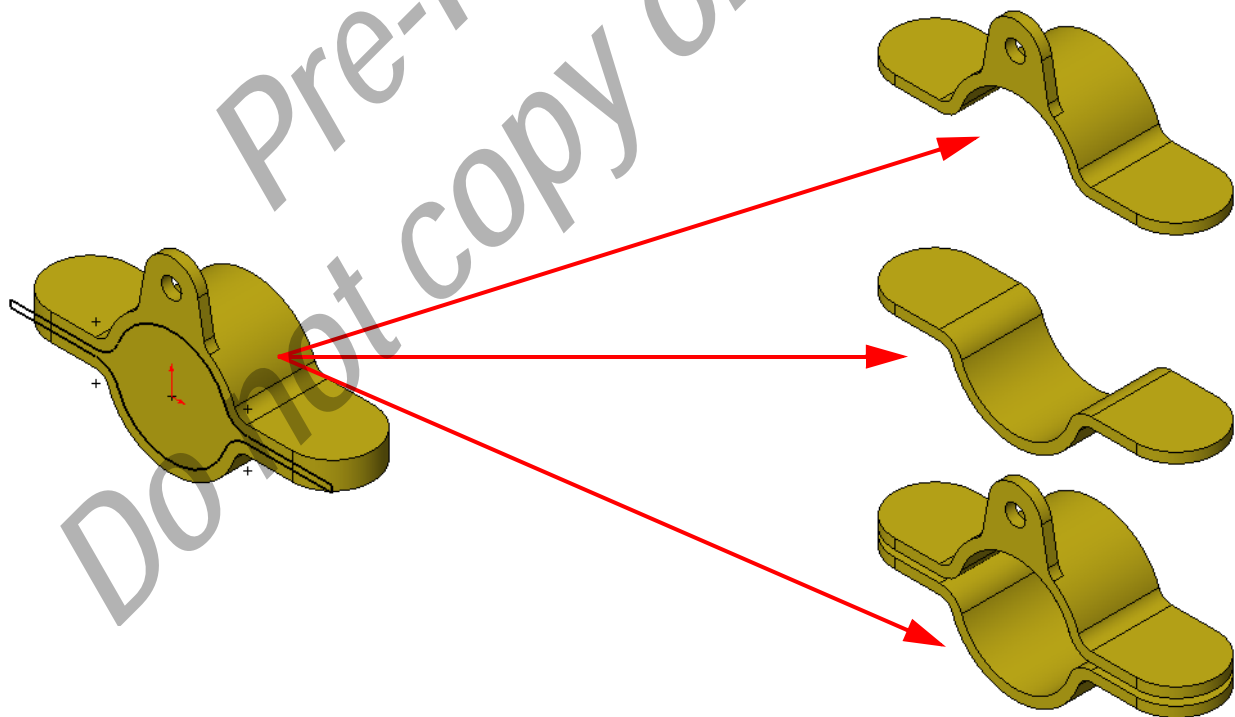
After hiding the tool body solid, the results can be seen.

**Note**

The tool body can be removed using **Delete Body** . It creates a Body-Delete1 feature.

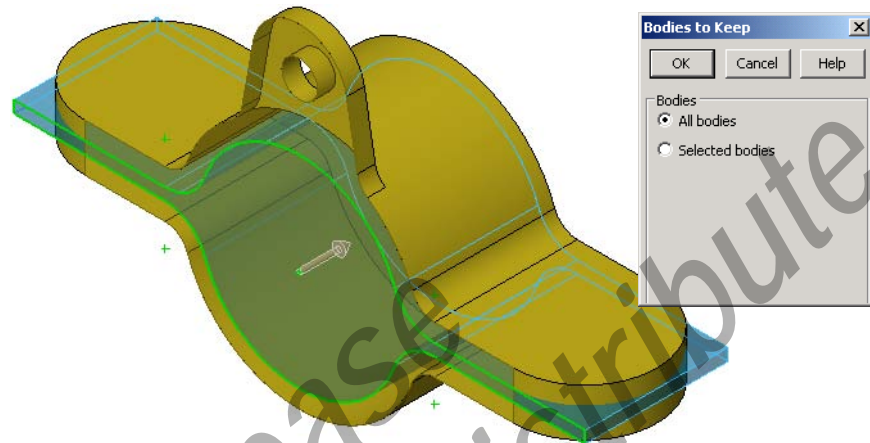
Using Cut to Create Multibodies

Certain cut features will split a part into multiple solid bodies. If this happens the **Bodies to Keep** dialog box appears. You can control how to split the part.

**1 Open part Cut into Bodies.**

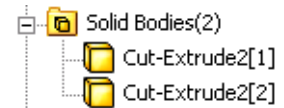
2 Create multibodies.

Using Sketch3, create a **Through All** cut with the **All bodies** option.



3 Explore the Solid Bodies folder.

The cut feature creates two solid bodies.



Saving Solid Bodies as Parts and Assemblies

You can save one or more of the solid bodies in a multibody part as separate part files. There are several commands to do this, each with different characteristics. Some commands give you the option to also generate an assembly from the saved parts.

Default Templates

The commands in this section create new SolidWorks documents – either a part or an assembly or both as appropriate. You have the option of specifying a document template or allowing the system to use the default template. This choice is determined by the settings in **Tools, Options, System Options, Default Templates**.

Introducing: Insert into New Part

Insert into New Part allows you to save individual solid bodies as part files. Each resulting part file is linked by an external reference back to the source part. A `Stock-<source part name>` feature appears in the saved part. This feature carries the external reference. For more information about external references, refer to the *Advanced Assembly Modeling* training course.

Note

If you select multiple bodies or the `Solid Bodies` folder, the saved part will be a multibody part with a `Stock` feature for each body.

Insert into New Part does not create a feature in the source part. The solid bodies are saved as they are after the last part feature is rebuilt. Any changes you make to the source part *will* propagate to the saved parts.

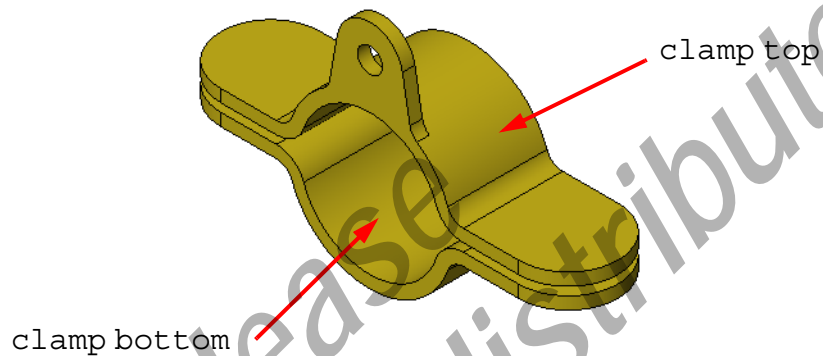
Where to Find It

- Expand the `Solid Bodies` folder and right-click the body you want to save. Select **Insert into New Part**.

4 Insert the solid bodies into new parts.

Expand the **Solid Bodies** folder. Use **Insert into New Part** to create the parts as shown below, one part for each body.

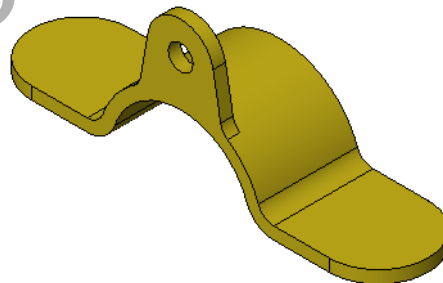
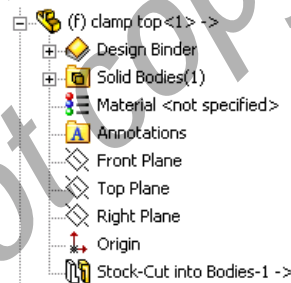
The new parts are opened automatically.

**5 Create an assembly.**

Open a new assembly. Add the saved parts. Name the assembly `clamp_assy`.

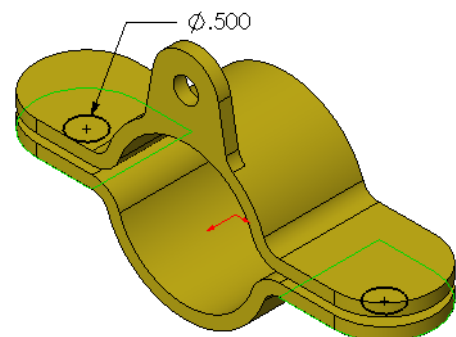
6 Newly created part.

Switch to one of the newly created parts. Examine the FeatureManager. Note the **Stock** feature. This carries the external reference.

**7 Make changes to the source part.**

Switch back to the source part.

Insert a sketch on the planar face on the underside of the bottom half of the clamp as shown.



8 Through All cut.

Click **Extruded Cut** . Set the end condition to **Through All**.

9 Click Detailed Preview .

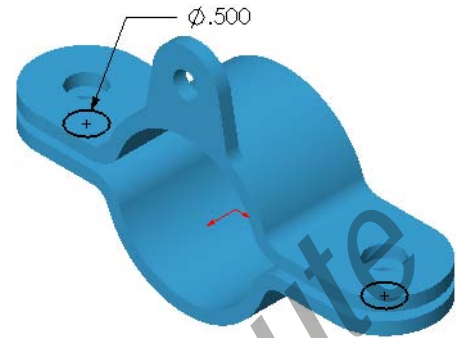
Under **Options**, select **Show only new or modified bodies**.

Clear the **Highlight new or modified faces** check box.

Examine the preview. It shows that the feature will cut through both bodies.

Do not click OK yet.

10 Turn off Detailed Preview.



Feature Scope

The **Feature Scope** allows you to select which bodies are affected by a feature. The **Feature Scope** option exists in the following tools:

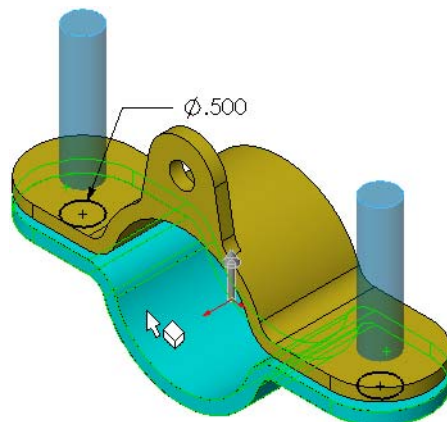
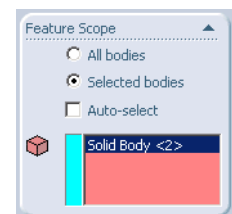
- Extrude
- Revolve
- Sweep
- Loft
- Cut with Surface
- Thicken

11 Set the feature scope.

Expand the **Feature Scope** group box.

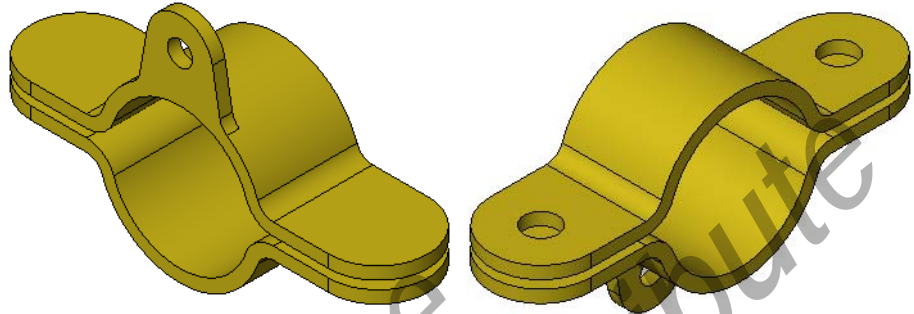
Clear the **Auto-select** check box.

Select the bottom half of the clamp and click **OK**.

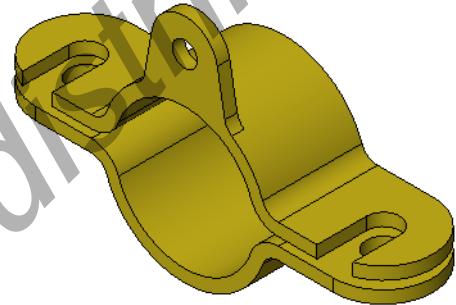


12 Results.

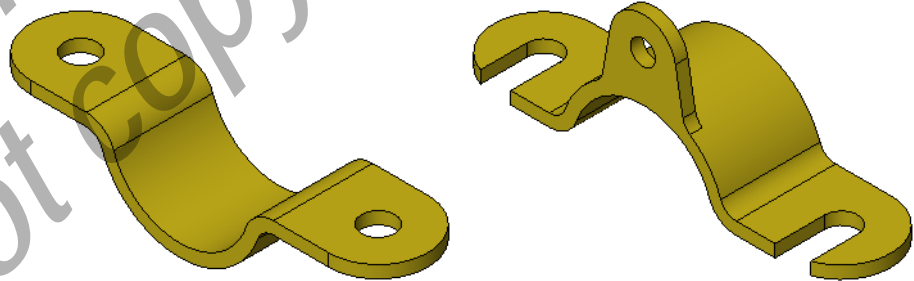
The cut feature only affects the selected body.

**13 Second Through All cut feature.**

Create another **Through All** cut feature as shown. Use the **Feature Scope** to limit its effect to just the upper half of the clamp.

**14 Examine the individual parts.**

The changes made to the source part propagated to the saved files.

**15 Save and close the files.****Introducing: Save Bodies**

Save Bodies also allows you to save individual solid bodies as part files. You can indicate which bodies you want to save. Optionally you can generate an assembly from the saved parts.

Save Bodies adds a **Save Bodies** feature in the FeatureManager of the source part.

The bodies are saved at the point in the part history where the **Save Bodies** feature appears. Any subsequent features added to the source part will *not* propagate to the saved files.

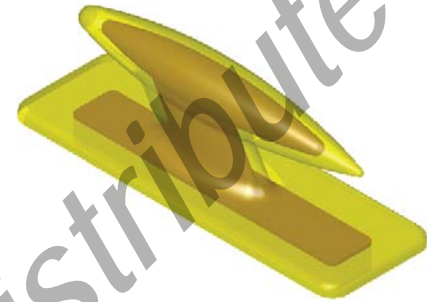
Each resulting part file is linked by an external reference back to the source part. A Stock-<source part name> feature appears in each saved part. This features carries the external reference.

Where to Find It

- Click **Insert, Features, Save Bodies**.
- Right-click the **Solid Bodies** folder and select **Save Bodies**.

1 Open part.

Open the part Boat Cleat. There are two solid bodies representing the core and the pattern. One body is shown semi-transparent for illustration purposes.



2 Edit color.

Select the topmost feature in the FeatureManager.

Click **Edit Color** . Remove the transparency.

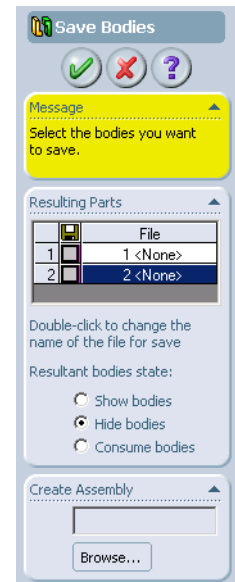
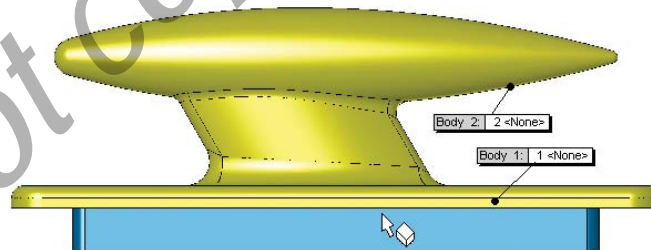
3 Saving the bodies.

Click **Insert, Features, Save Bodies**. The PropertyManager appears.

As you move the cursor over the model, the individual bodies highlight.

Tip

Sometimes it is hard to tell which callout points to which body. Changing the view usually helps.

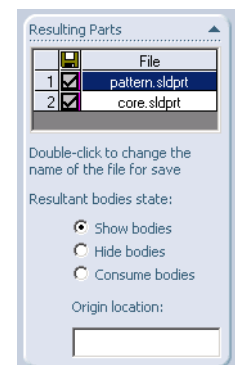


4 Saving the bodies.

There are two ways to save the bodies as separate part files:

- In the PropertyManager, under **Resulting Parts**, double-click the name field. The **Save As** dialog appears.
- In the graphics area, click the name field of the callout **Body 1: 1 <None>**. The **Save As** dialog appears.

Save the bodies as Core and Pattern.



Note When you save the bodies you can specify an origin location. If you do not, the saved parts have the same origin as the source part.

5 Resultant bodies state.

Click **Show bodies**. This will keep the solid bodies in the source part visible. The default is **Hide bodies**.

Creating an Assembly

If you want to create an assembly do the following:

1. In the **Create Assembly** group box, click **Browse**. The **Save As** dialog opens.
2. Browse to where you want to save the assembly.
3. Give the assembly a name and click **Save**.

In this example it is not necessary to save the assembly. If later you decide you need an assembly you can always create one from the saved parts using traditional bottom-up assembly modeling techniques.

6 Click OK.


The saved parts open.

7 FeatureManager.

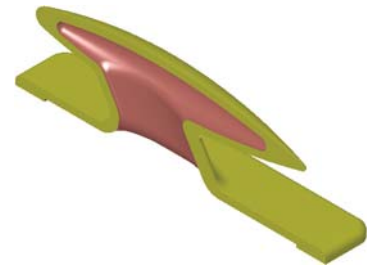
Examine the FeatureManager design tree of the *source* part. A **Save Bodies** feature has been added. This records the point in the part's history when the bodies were saved. Changes made to the source part after this feature will *not* propagate to the saved parts.

8 Make changes to the source part.

Make sure the source part, **Boat Cleat**, is active.

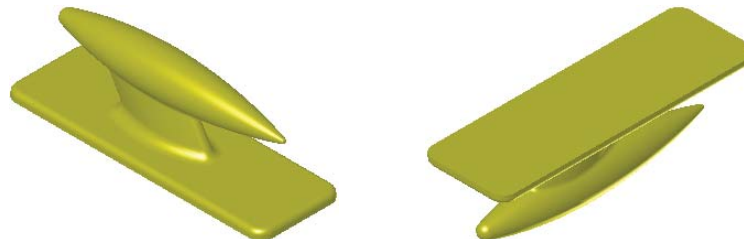
Click **Combine** . Subtract the core from the pattern.

The results are shown in a section view for clarity.



9 Examine the Pattern part.

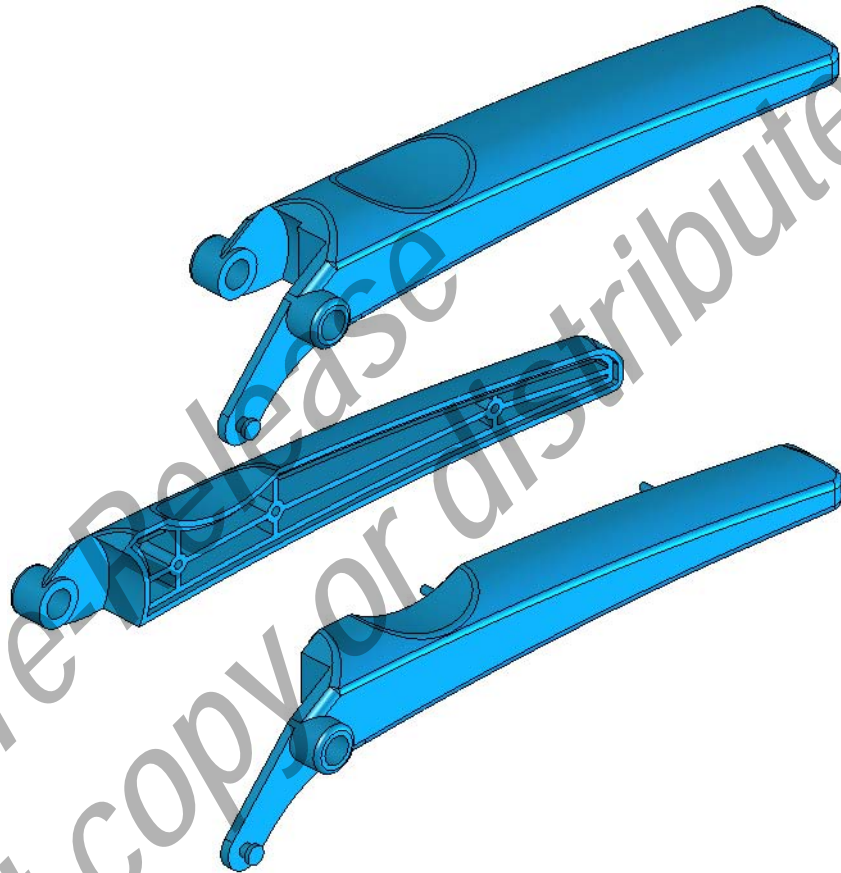
The change made to the source part did *not* propagate to the saved files.



10 Save and close the files.

Splitting a Part into Multibodies

Sometimes it is easier to start a design as a single part. Then, after form, fit, and function are defined, the part is split into its individual components. This is particularly handy when aesthetics are important.




Introducing: Split


Split allows you to break a part into multiple solid bodies using splitting tools such as faces, planes, or surfaces. In the **Split** command you have the option to save the resulting bodies as individual part files.

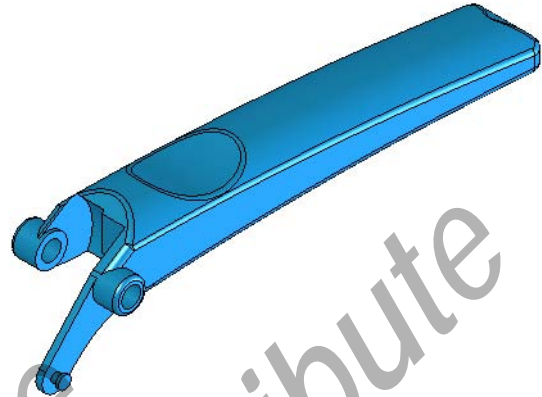
The **Split** command creates a `Split` feature in the FeatureManager of the source part. This means the bodies are saved at the point in the part history where the `Split` feature appears. Any subsequent features added to the source part will not appear in the saved files.

If you delete the `Split` feature in the original part, the new parts still exist, but the status of the external reference in the new parts is dangling.

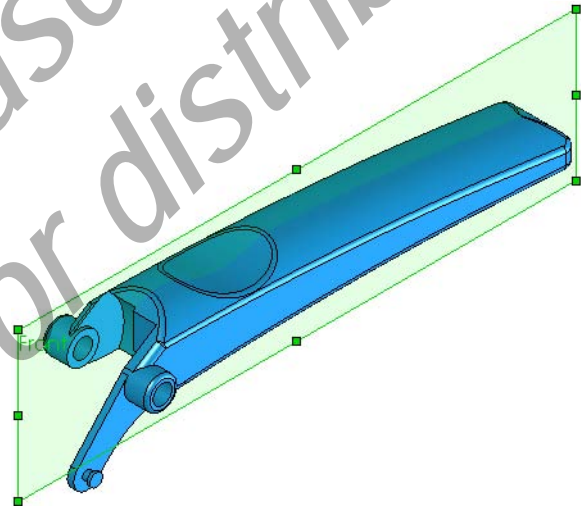
Where to Find It

- Click **Split**  on the Features toolbar.
- Or, click **Insert, Features, Split**.

- 1 **Open part named Handle.**
- 2 **Split the part.**
Click **Split**  or click **Insert, Features, Split**.

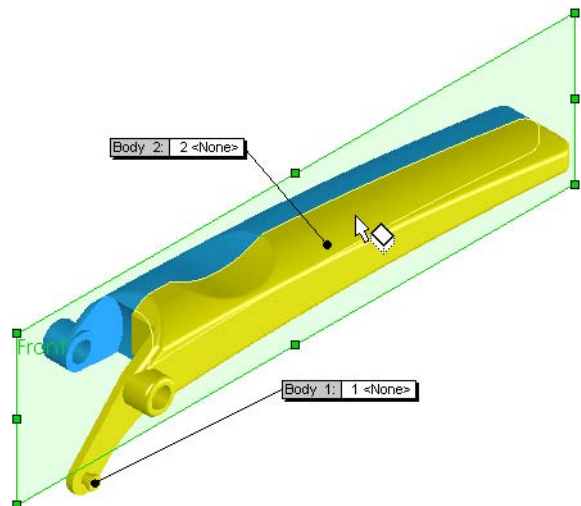


- 3 **Trim tools.**
Select the **Front** reference plane as the trim tool.



- 4 **Cut the part.**
Click **Cut Part**. The system computes the intersection of the trim tools with the part and calculates the results.

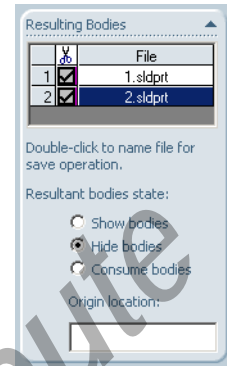
As you move the cursor over the model, the individual bodies highlight. Click on the bodies you want to create. In this case, click both resulting bodies.



5 Saving the bodies.

There are two ways to save the resulting bodies as separate part files:

- In the PropertyManager, under **Resulting bodies state**, double-click the name field. The **Save As** dialog appears.
- In the graphics area, click the name field of the callout **Body 1: 1 <None>**. The **Save As** dialog appears.



Note

When you save the bodies you can specify an origin location. If you do not, the saved parts have the same origin as the source part.

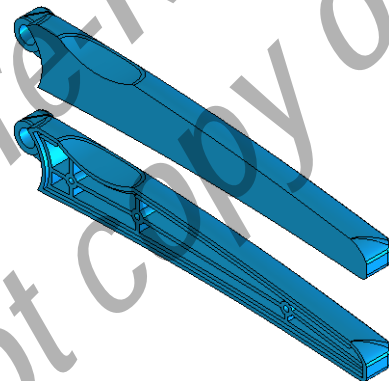
6 Resultant bodies state.

Click **Show bodies**. This will keep the solid bodies in the source part visible. The default is **Hide bodies**.

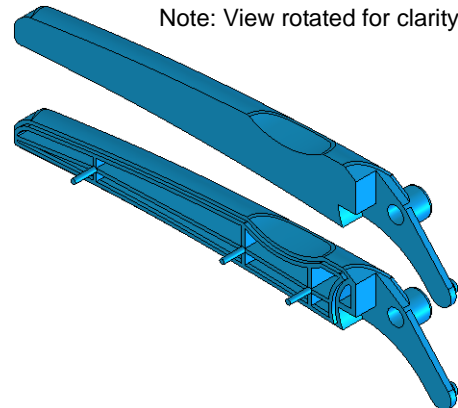
7 Click OK.

The new part files are created. Open them in their own windows.

You would now finish modeling the details of each part.



Note: View rotated for clarity.



Creating an Assembly

Once the solid bodies have been saved as part files, you can use them to create an assembly just as you would with any other parts. You can create an assembly manually using traditional bottom-up assembly modeling techniques, or you can automate the process.

Introducing: Create Assembly

Create Assembly collects the part files saved by one or more split features and creates a new assembly from them.

Where to Find It

- Right-click the **Split** feature in the FeatureManager design tree and select **Create Assembly**.
- Or, click **Insert, Features, Create Assembly**.

8 Create Assembly.

Right-click the **Split** feature and select **Create Assembly**. The PropertyManager opens. If you want you can select more than one **Split** feature.

9 Click Browse.

The **Save As** dialog box appears.

Browse to the folder where you want to save the assembly, and type a name for the assembly in the **File name** box.

10 Click Save.

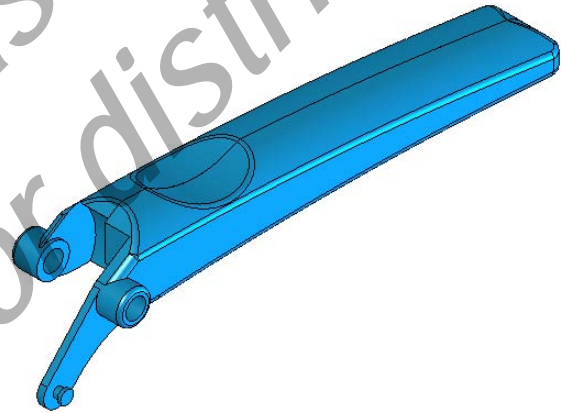
The **Save As** dialog box closes and the file name appears under **Assembly file** in the PropertyManager.

11 Click OK.

The new assembly document opens.

Note

There are no mates in this assembly. Both components are fixed with their origins at the assembly origin.

**12 Save and close the files.****Summary**

There is quite an assortment of tools and techniques for saving individual solid bodies as part files and for creating assemblies from multibody parts. All of the techniques create an external reference between the saved part file and the original source part.

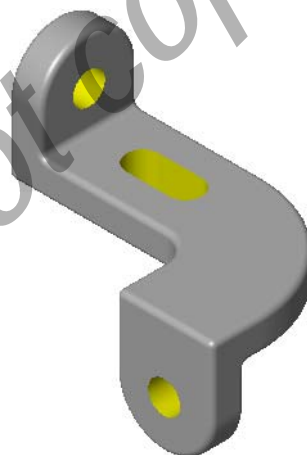
The various commands and techniques are summarized in the table below.

Technique	Results
<p>Insert into New Part Allows you to create new parts from the bodies in the Solid Bodies folder.</p>	<p>If you use Insert into New Part on the Solid Bodies folder instead of an individual body, you will create a multibody part that is linked back to the original part. Each body will be represented by its own Stock feature.</p> <p>Insert into New Part does <i>not</i> add a feature in the FeatureManager of the source part. Therefore, any features you add to the bodies in the source part will propagate to the saved files.</p>

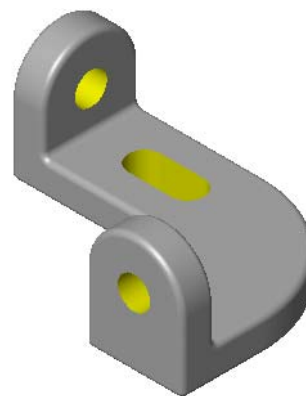
<p>Split Part Allows you to split a single solid body into multiple bodies.</p>	<p>Adds a <code>Split</code> feature in the FeatureManager of the source part. The bodies are saved at the point in the part history where the <code>Split</code> feature appears. Any subsequent features added to the source part will <i>not</i> appear in the saved files. Any features added <i>before</i> the <code>Split</code> feature <i>will</i> propagate to the saved files.</p>
<p>Save Bodies Like Split Part without the splitting tools. It takes existing bodies in the part and lets you write them out as parts.</p>	<p>Adds a <code>Save Bodies</code> feature in the FeatureManager of the source part. The bodies are saved at the point in the part history where the <code>Save Bodies</code> feature appears. Any subsequent features added to the source part will <i>not</i> appear in the saved files. Optionally you can generate an assembly from the saved parts.</p>
<p>Create Assembly Collects the part files saved by one or more <code>Split</code> features and creates a new assembly from them.</p>	<p>This is a convenience tool that automates generating an assembly from a <code>Split</code> feature. You could do exactly the same thing by manually opening a new assembly and adding all the saved parts. Create Assembly does <i>not</i> add a feature in the FeatureManager of the source part. Therefore, it is not parametric in the sense that if you create more solid bodies later, they do not automatically appear in the assembly.</p>

Using Split Part with Legacy Data

You can use **Split Part** to modify imported geometry or legacy parts that would otherwise be difficult to change.




Before



After

1 Import an IGES file.

Click **Open**  or click **File, Open**.

For **Files of type**, select IGES (*.igs, *.iges).

Select the files Legacy Data.igs and click **Open**.

2 Cutting plane.

Define a reference plane that is parallel to the Front plane and that passes through the vertex shown.

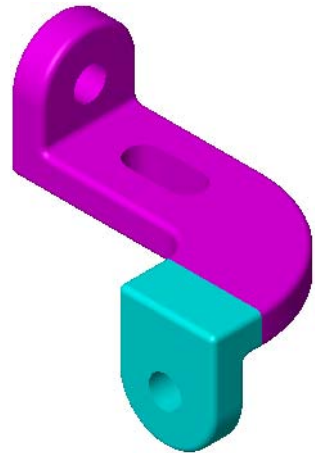
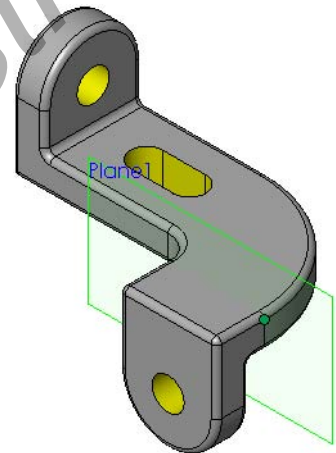
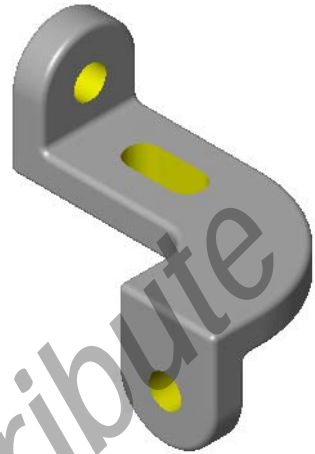
This will be used as the cutting plane in the **Split Part** command.

3 Split Part.


Using the plane created in the previous step, split the part into two separate bodies.

Note

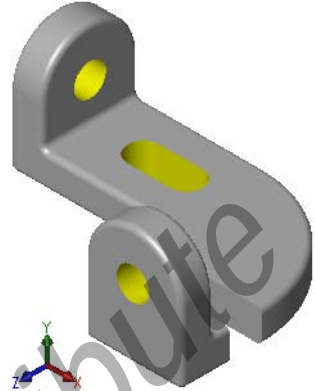
The bodies are shown here in different colors for illustration purposes.



4 Move/Copy Body.

Click **Move/Copy Bodies**  or click **Insert, Features, Move/Copy**.

Use **Coincident** and **Distance** mates to rotate the body **180°** and move the body **0.75"** with respect to the Z axis as indicated by the reference triad.




Filling the Gap

How you fill in the gap depends on the shape on the part geometry. In this example a simple extrude feature will work. For an example of how a loft feature can be used to fill in a gap, see *Merging a Multibody with Loft* on page 144.



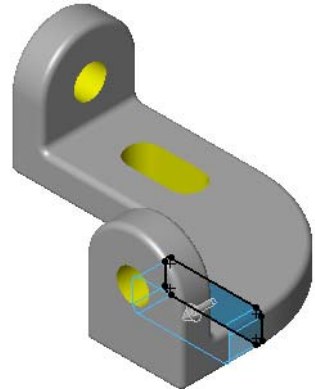
5 Bridge the gap with an extruded boss.

Create a sketch on the flat face of the rear body. Use **Convert Entities**  to copy the edges of the face.

Extrude the sketch using the end condition **Up To Next**.

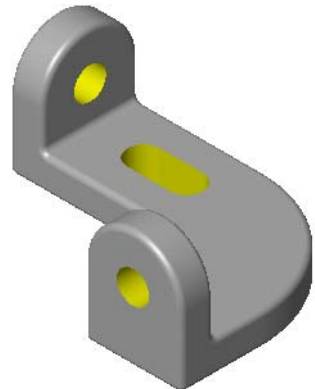
Be sure the **Merge results** check box is selected.

This is a variation of the bridging technique shown on page 9.



6 Results.

7 Save and close the part.



Exercise 1: Combining a Multibody Part

Create this part by following the steps as shown.

This lab reinforces the following techniques:

- Multibody solids
- Combining
- Hole Wizard

Units: **inches**

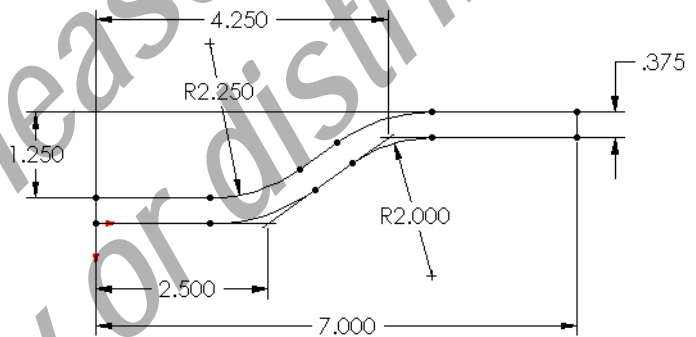
Procedure

Open a new part using the Part_IN template and name it Mbody1.

1 Sketch first profile.

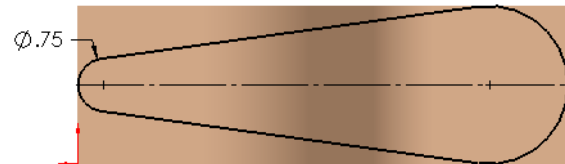
Use lines, fillets and offsets.

Extrude the profile **2.25"**.



2 Sketch second profile.

Extrude as required.



3 Combine bodies.

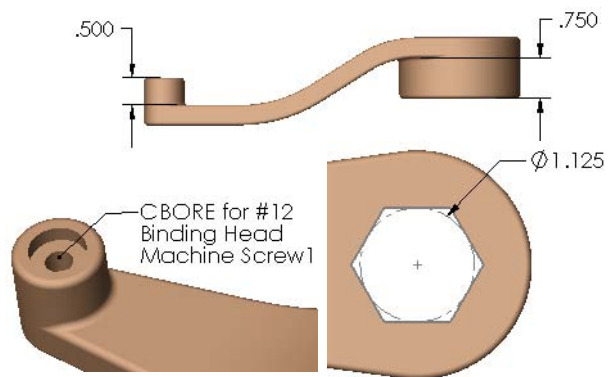
Combine the two solid bodies into one.



4 Add features.

Add boss, cut, hole wizard and fillet features.

Finish part with **0.0625"** radius fillets and rounds.



5 Save and close part.



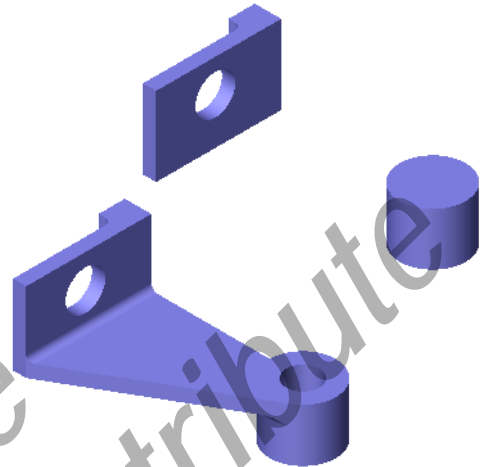
Exercise 2: Bridging a Multibody Part

Create this part by following the steps as shown.

This lab reinforces the following techniques:

- Multibody solids
- Bridging

Units: **millimeters**



Design Intent

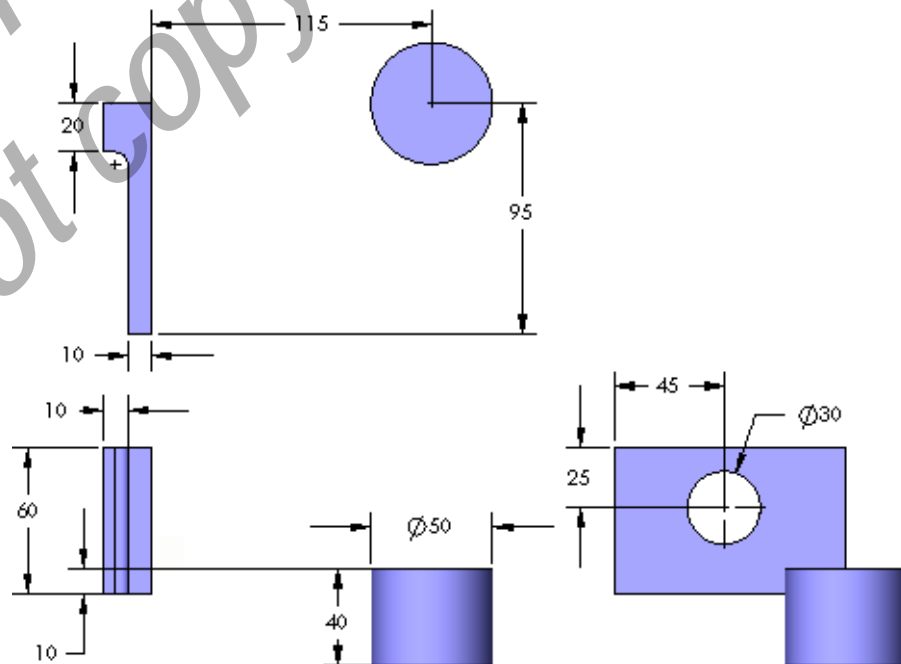
The design intent for this part is as follows:

1. Part is *not* symmetrical.
2. Holes are through all.
3. All fillets and rounds are **5mm** radius.

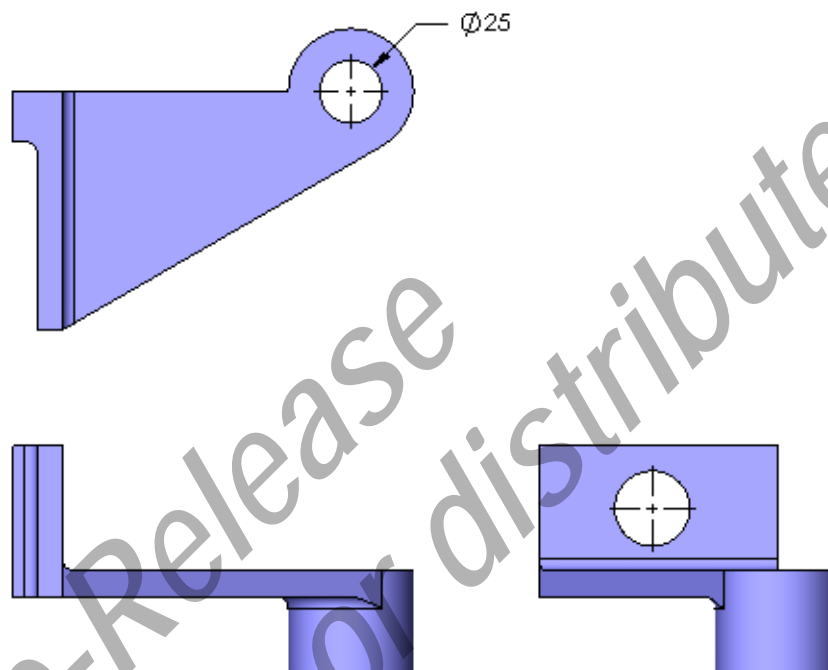
Procedure

Open a new part using the Part_MM template and name it Mbody2.

1 Create a multibody part.



2 Finish part with bridge technique.



3 Save and close part.

**Exercise 3:
Creating a
Multibody with
Mirror Pattern**

Create this part by following the steps as shown.

This lab reinforces the following techniques:

- Multibody solids
- Patterning
- Combining

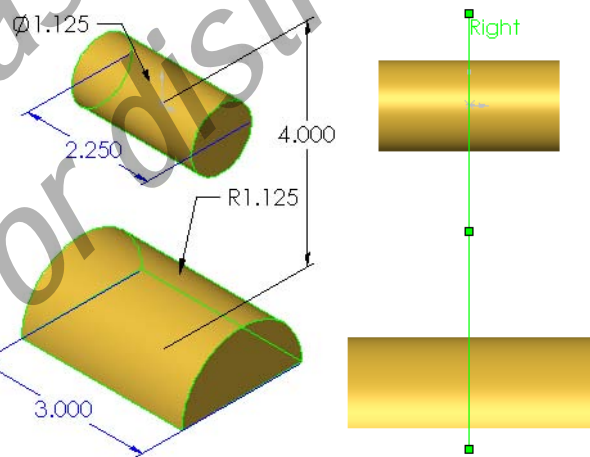
Units: **inches**



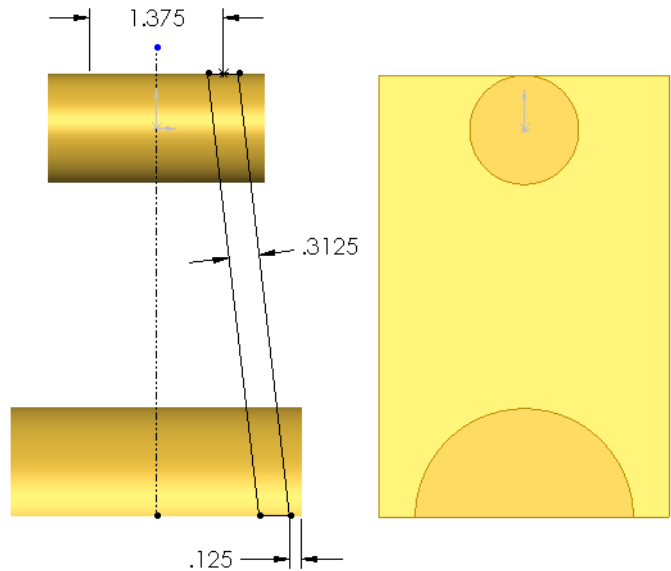
Procedure

Open a new part using the Part_IN template and name it Mbody3.

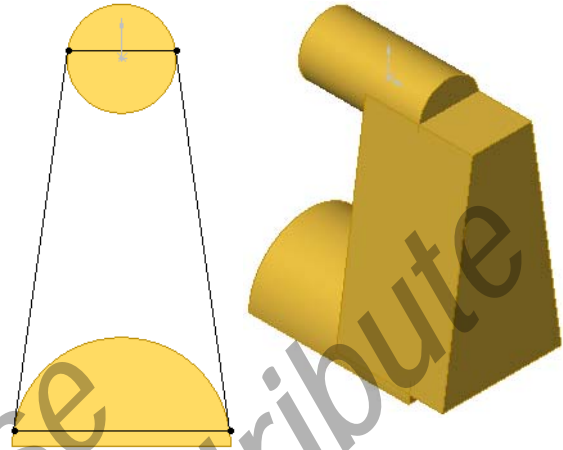
- 1 Create a multibody part.**



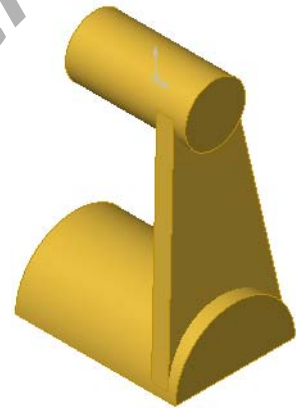
- 2 Create feature.**
Sketch and extrude the feature shown as a third body.



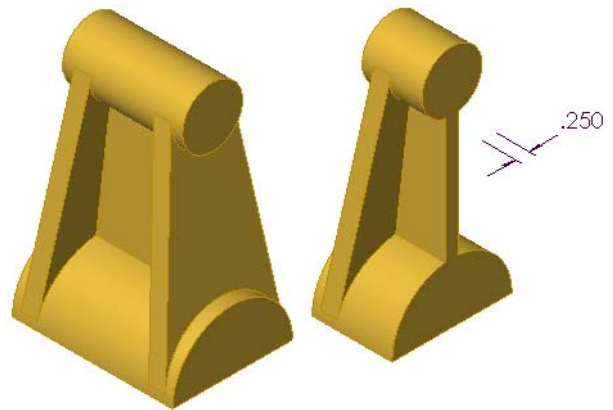
- 3 Create feature.**
Extrude sketch as shown.



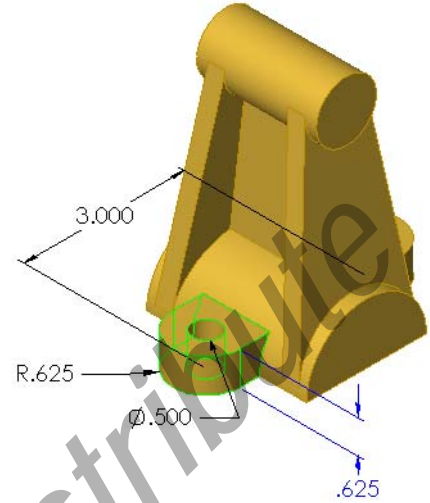
- 4 Combine bodies.**
Combine the last two solid bodies.



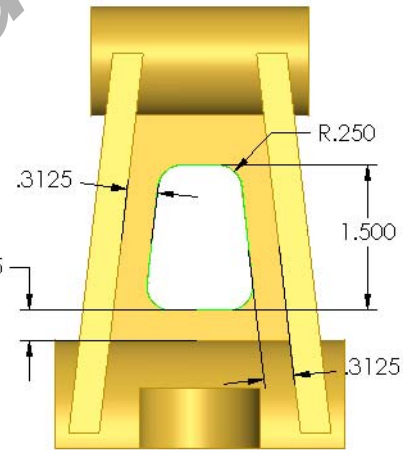
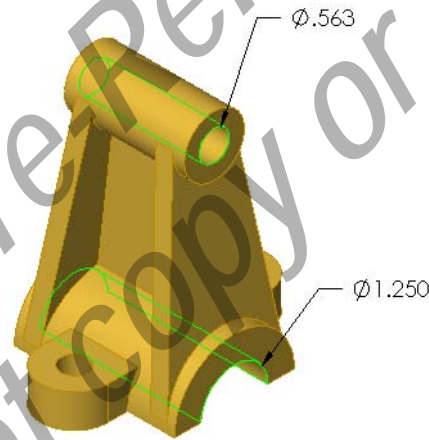
- 5 Insert mirror.**
Mirror the combined body and add a centered boss that merges the bodies into one.



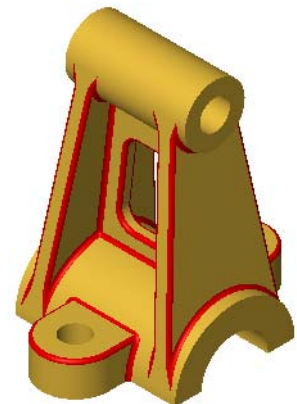
- 6 Add mounting lugs.**
 Add a boss and a cut for the mounting lug. Mirror the body across the model. Merge the lugs.



- 7 Add features.**
 Add holes and cuts.



- 8 Add fillets and rounds.**
 Finish part with **0.0625"** radius fillets and rounds.



- 9 Save and close part.**

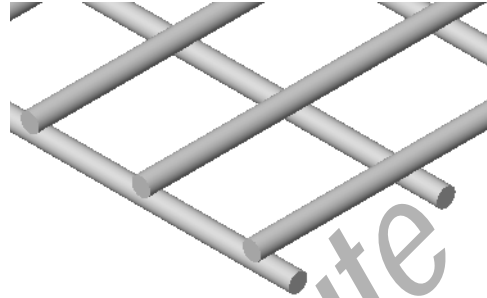
**Exercise 4:
Creating a
Multibody with
Linear Pattern**

Create this part by following the steps as shown.

This lab reinforces the following techniques:

- Multibody solids
- Patterning
- Combining

Units: **inches**



Design Intent

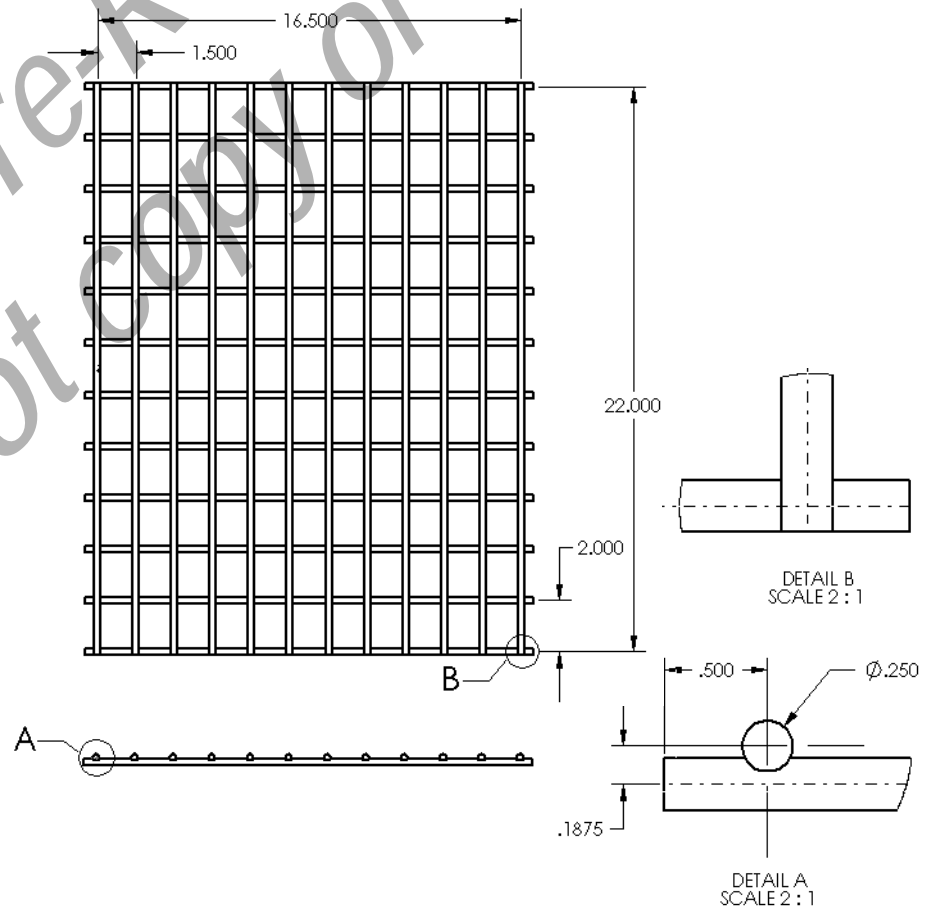
The design intent for this part is as follows:

1. Part is symmetrical.
2. All bars circular section are same radius.
3. Equal spacing of bars.

Procedure

Open a new part using the Part_IN template and name it Mbody4.

1 Create a multibody part.



2 Save and close part.

Exercise 5: Positioning Inserted Parts

Create this part by following the steps as shown.

This lab reinforces the following techniques:

- Inserting parts
- Move/Copy bodies
- In context editing



Procedure

Open an existing part named Base.

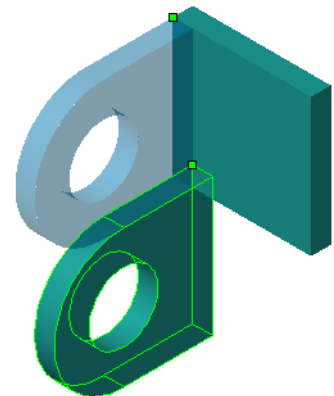
1 Insert part.

Insert the part Lug and rotate it as shown.



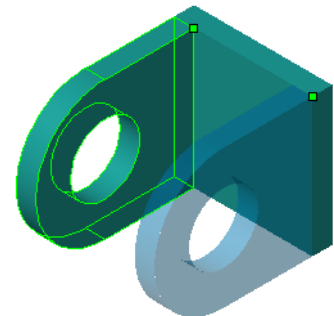
2 Move body.

Position the part Lug on the part Base as shown.



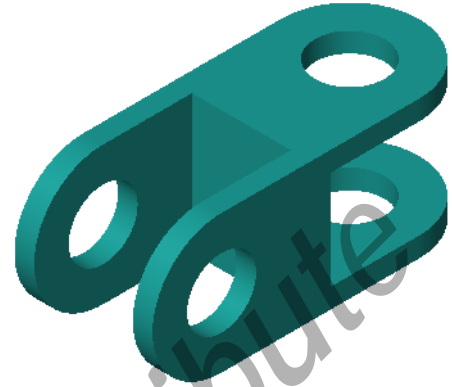
3 Copy body.

Add another instance of the Lug.



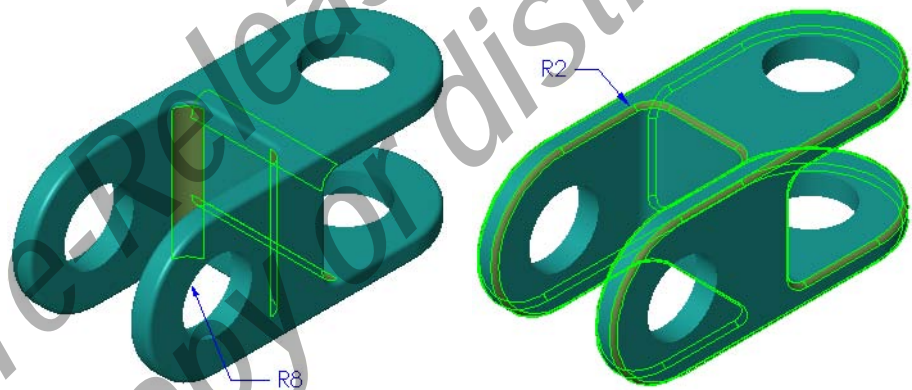
4 Continue.

Add two more copies of the Lug, placed as shown.



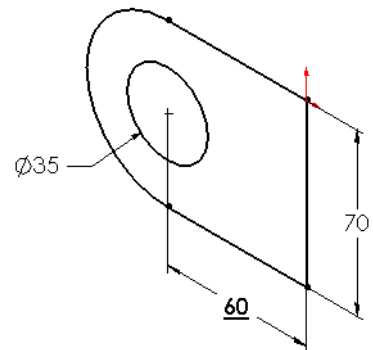
5 Combine bodies and add fillets.

Combine all the solid bodies into one. Add fillets as shown.



6 Modify sketch.

Open the Lug part and change a dimension.



- 7 Propagate change.**
Return to the main part.



- 8 Save and close part.**

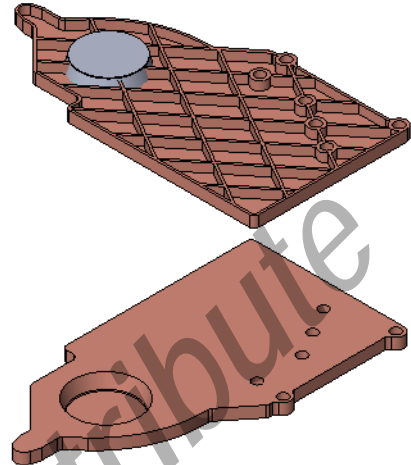
Pre-Release
Do not copy or distribute

Exercise 6: Using Indent

Create this part by following the steps as shown.

This lab reinforces the following techniques:

- Inserting parts
- Move/Copy bodies
- Combine feature
- Indent feature

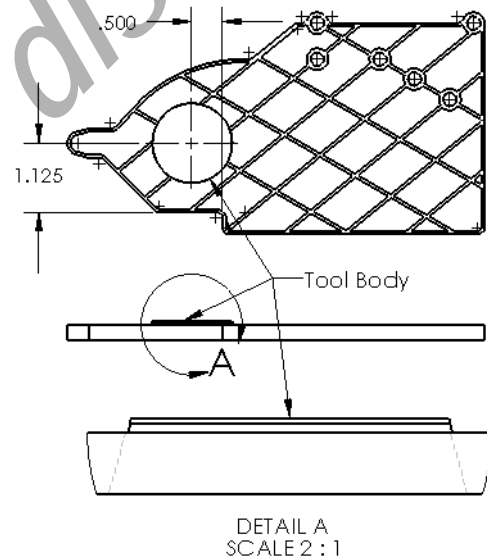


Procedure

Open an existing part named Target Body.

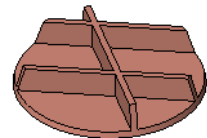
1 Insert part.

Insert the part Tool Body and position it as shown in the Front and Top views.



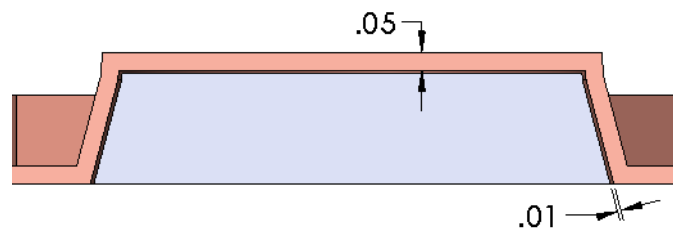
2 Interference.

Check to see that there is an interference between the solid bodies. The volume of interference should look like this.



3 Indent.

Indent the Tool Body into the Target Body using the settings shown below for **Thickness** and **Clearance**.



4 Save and close parts.

Exercise 7: Copying Bodies

Create this part by following the steps as shown.

This lab reinforces the following techniques:

- Inserting parts
- Move/Copy bodies
- Patterning
- In context editing

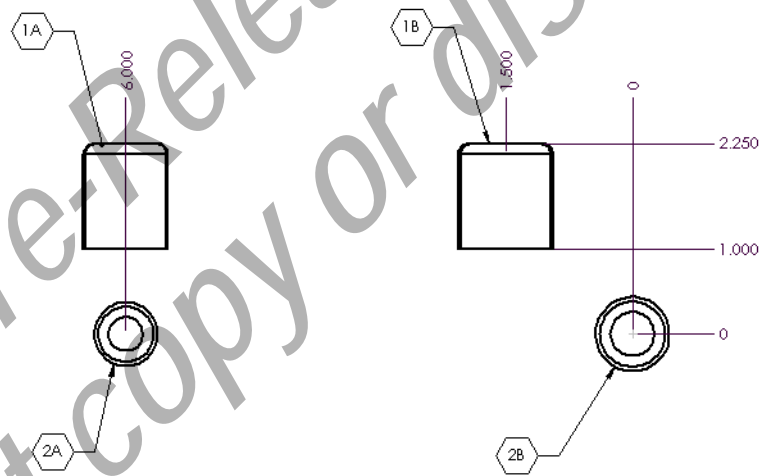


Procedure

Open a new part using the Part_IN template and name it Mbody6.

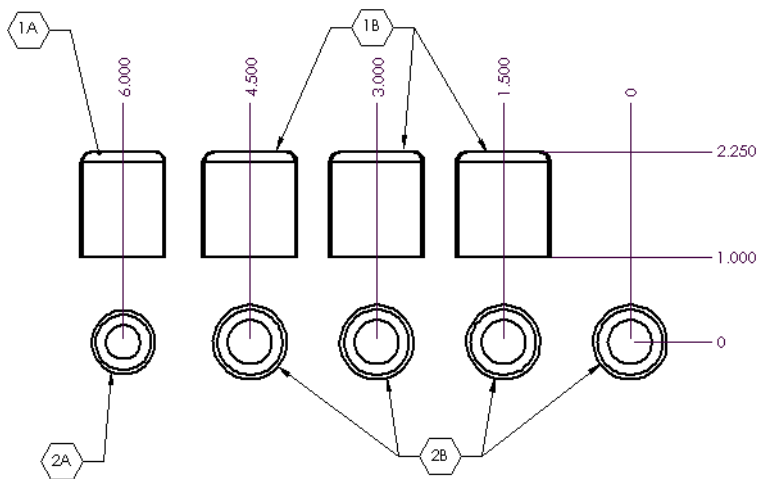
1 Insert parts.

Insert and position 1A, 1B, 2A and 2B parts as shown.



2 Add patterns.

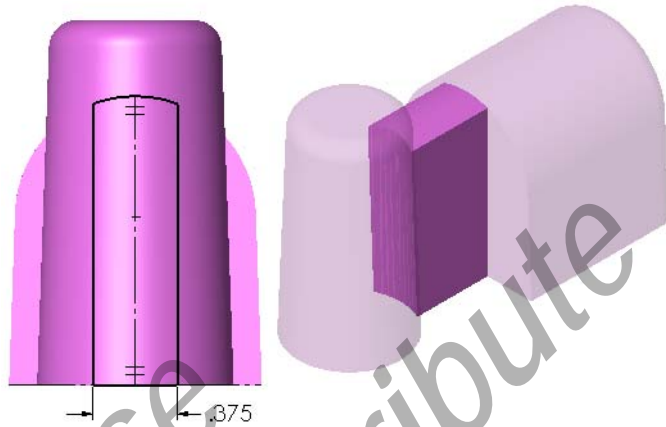
Pattern the solid bodies as shown.



3 Connect bodies.

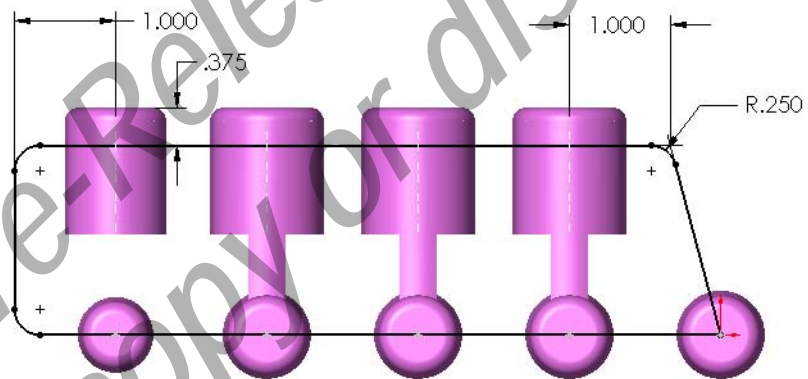
Create a bridge that connects the bodies *without* merging.

Pattern bridge with 3 instances.



4 Create plate.

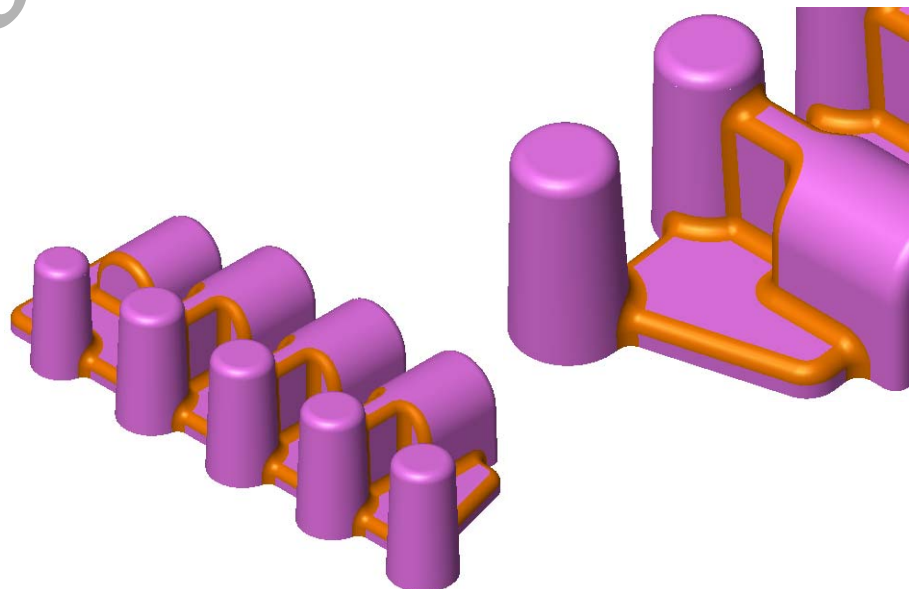
Sketch on the top reference plane to create the plate feature.



Extrude the feature **0.25"** and **Merge result.**

5 Add fillets and rounds.

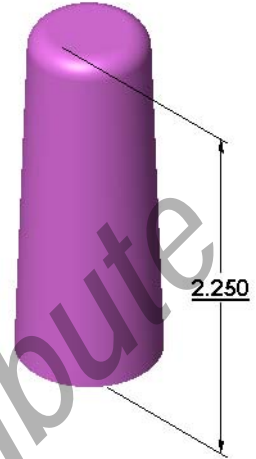
Finish part with **0.125"** radius fillets and rounds.



6 Modify referenced part.

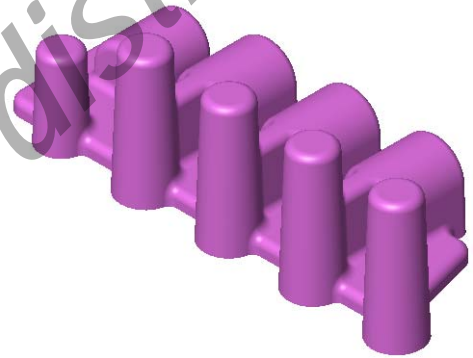
Right-click the feature 2B and choose **Edit in Context**.

Change the depth of the extrusion to **2.250"**.



7 Propagate change.

Return to the main part.



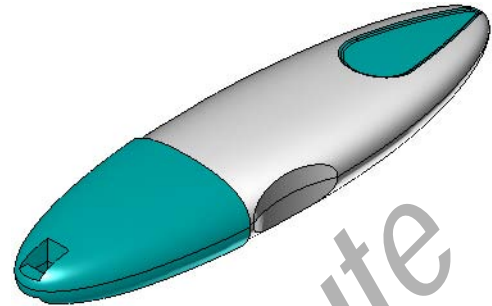
8 Save and close part.

Exercise 8: Split Part

Using the part provided, create multiple parts that are related to the original.

This lab uses the following skills:

- Split Part



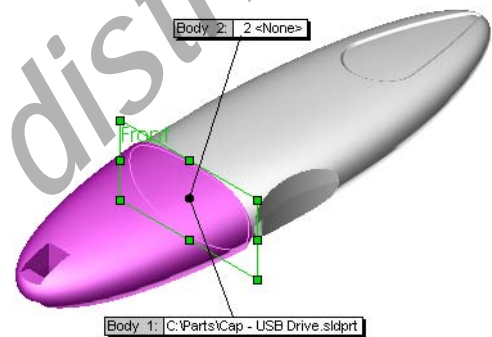
Procedure

Use the following procedure:

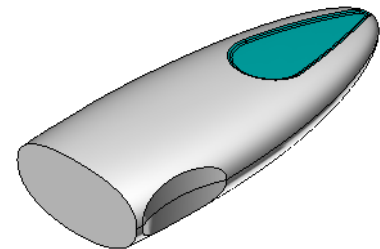
- 1 **Open the part named USB Flash Drive.**
This part represents the conceptual design of the product.

- 2 **Split the part.**
Split the part to separate the cap from the remainder of the body.

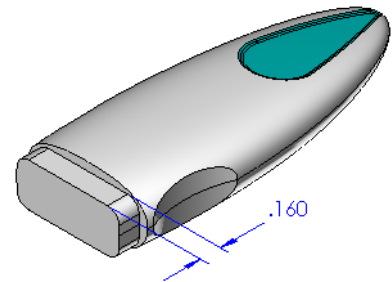
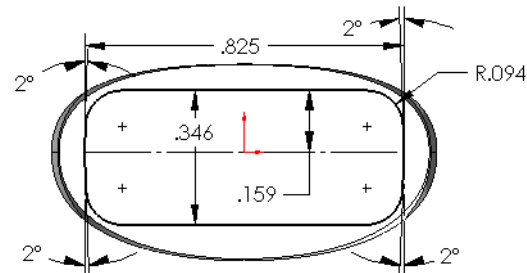
Name the saved part
Cap - USB Drive.sldprt.



- 3 **Resultant bodies state.**
Click **Hide bodies**. This will hide the cap making it easier to split the remaining body along the parting line.
Click **OK**.



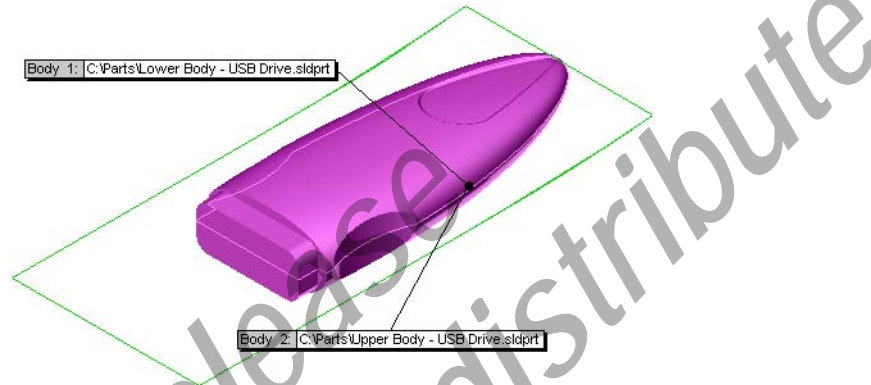
- 4 **Add a boss.**
Create the sketch shown below and extrude a boss a distance of **0.160"**.



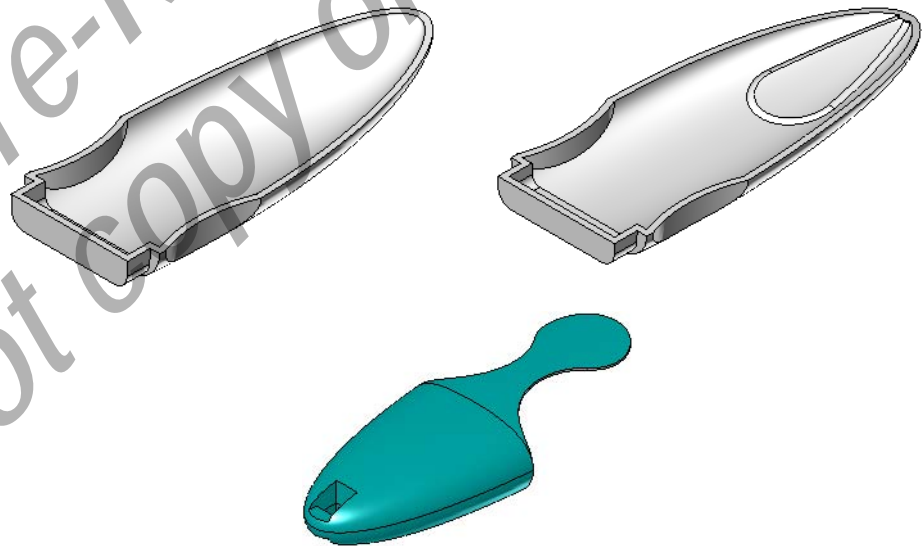
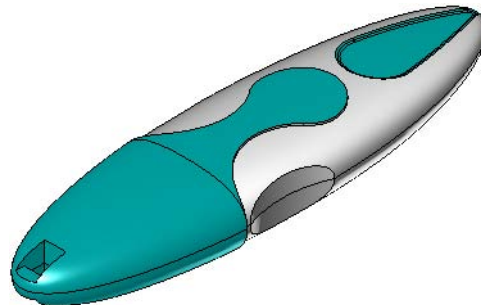
5 Split the part.

Split the part to create the upper and lower halves of the body. Use the Parting Surface as the trim tool. The surface is hidden. It does not have to be visible in order to use it as a trim tool.

Name the parts as shown in the illustration below.

**6 Open the individual parts.**

Add any additional design details needed.

**7 Create an assembly.****8 Save and close all files.**

Lesson 2 Sweeps

Upon successful completion of this lesson, you will be able to:

- Explain the difference between sweeping and lofting.
- Create a curve through a set of data points.
- Create a multi-thickness shell.
- Create a non-planar curve by projecting a sketch onto a surface.
- Create a variable radius fillet and a face fillet.
- Create boss and cut features by sweeping.
- Analyze sketches for curvature, minimum radius and inflection points.
- Analyze surfaces with zebra stripes.
- Model threads.
- Create an axis.
- Create a 3D sketch.
- Create a hole using Hole Wizard on a non-planar face.

Pre-Release
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Introduction

This lesson contains case studies that explore different modeling techniques that can be applied to modeling advanced, free-form shapes. Some of the commands and techniques that will be explored are:

- Sweeping
- Variable-radius filleting capabilities
- Analyzing sketches and surfaces
- 3D sketching



Case Study: Bottle

Modeling free-form shapes requires some techniques for creating features that are quite unlike the extruded or revolved shapes built in the basic course. This example will go through the steps of creating the molded plastic bottle shown at the top of the page.

Stages in the Process

Some of the key stages in the modeling process of this part are given in the following list:

- **Create the basic shape of the bottle.**
This will be done by sweeping an ellipse in such a way that the major and minor axes will be controlled by two guide curves.
- **Create a raised outline for the label.**
We will sketch the outline of the label area and then project it onto the surface of the bottle. This projected curve will be used as the path for sweeping the raised outline.
- **Add the neck.**
This is a simple boss extruded upwards from the top of the swept body.
- **Fillet the bottom.**
The radius fillet on the bottom of the bottle varies from 0.375" at the two sides to 0.25" at the center of the front and back.
- **Shell the bottle.**
The bottle has two different wall thicknesses. The neck has to be thicker (.060") because of the threads. The body is thinner (.020").
- **Model the threads.**
This is another sweeping operation. However, this time a different sort of path is used: a helix.

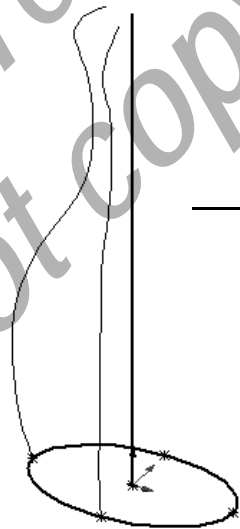
Sweeping and Lofting: What's the Difference?

Both sweeping and lofting are capable of creating many complex shapes. Which tool you use to build a particular part depends primarily on what design information you have to work with. There are also some general differences between sweeping and lofting that will influence which method to use. In essence:

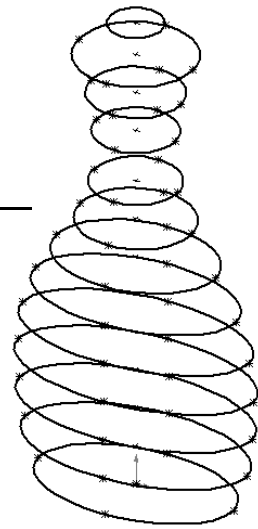
- Sweeping uses a single profile sketch.
- Lofting uses multiple profile sketches.

Consider the first feature of a plastic bottle such as the one shown in the illustration at the right. If the design data you are working with consists of the two curves that describe the outline of the bottle as seen from the front and side, and the cross section is similar throughout the shape, you can create the feature using sweep, with guide curves controlling the major and minor axes of the elliptical section.

If the design data you are working with consists of a set of cross sections, you can use loft to build the part. This is especially useful when the cross sections are dissimilar although that is not the case in this example.



Starting with this?
Use **Sweep**.

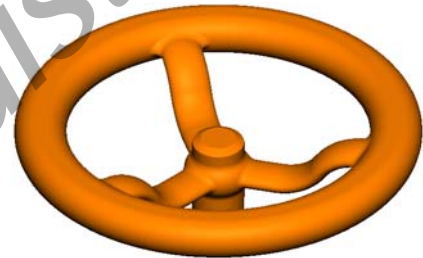
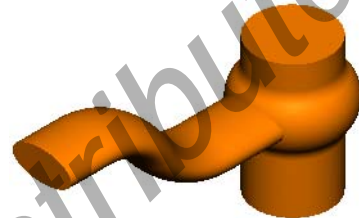
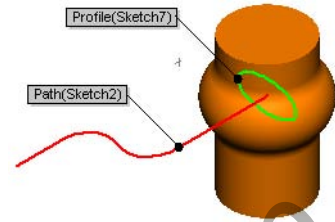


Starting with this?
Use **Loft**.

Sweeping

Sweeping can be simple or complex. For example, the spoke of the handwheel in the illustration at the right is swept using a 2D sketch for the path and an ellipse for the sweep section. The sweep section does not vary along the length of the path.

Sweeping can be much more complex than this simple example. Swept features can also incorporate 3-dimensional curves or model edges as paths, and the sweep section can be made to vary as it moves along a set of other curves called guide curves.

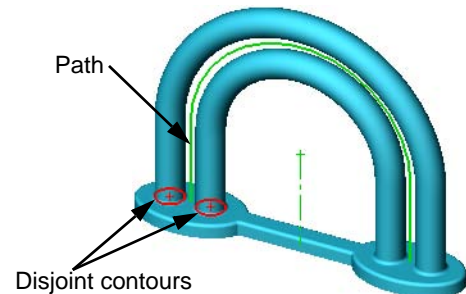
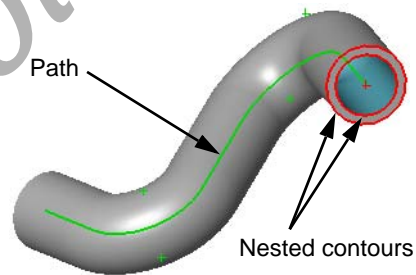


Sweep Components

Below is a list of the major components used in sweeping, including descriptions of their functions.

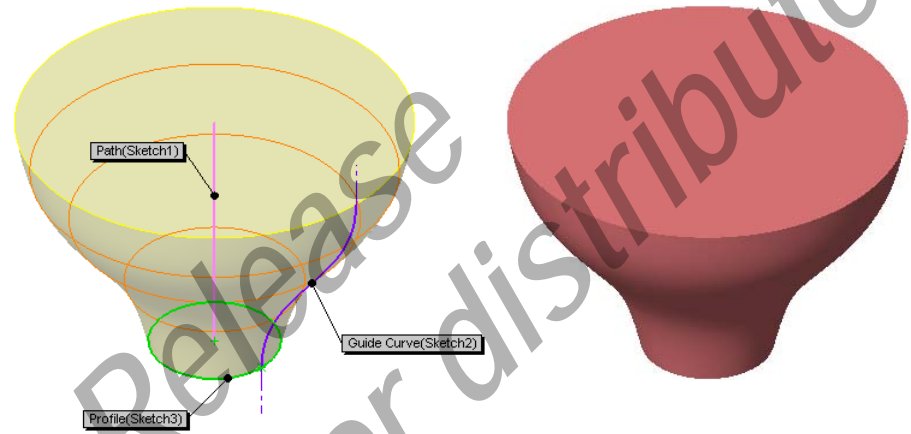
- **Profile.**

Sweeping only supports a single profile sketch. It must be a closed, non-self-intersecting boundary. However, the sketch can contain multiple contours – either nested or disjoint.



■ **Guide Curves.**

Sweeps can contain multiple guide curves which are used to shape the solid. As the profile is swept, the guide curves control its shape. One way to think of guide curves is to visualize them driving a parameter such as a radius. In this illustration, the profile is attached to the guide curve. As the profile is swept along the path, the radius of the circle changes, following shape of the guide.



■ **Sweep Path.**

The **Sweep Path** helps determine the length of the sweep by its endpoints. This means that if the path is shorter than the guides, the sweep will terminate at the end of the path.

The system also uses the path to position the intermediate sections along the sweep. Assuming the profile plane is normal to the path:

- The **Orientation/Twist Type** option **Follow Path** means that the intermediate sections will always stay normal to the path.
- If the **Keep Normal Constant** option is used, the intermediate sections will stay parallel to the plane of the profile sketch.

Creating a Curve Through a Set of Points

Curve Through XYZ Points enables you to create a 3-dimensional curve through a series of X, Y, Z locations. You can enter these locations directly into a spreadsheet-like dialog or you can read them from an ASCII text file. The file should have the file extension * .SLDCRV or * .txt. The curve will pass through the points in the same order as they are entered or listed in the file.

Where to Find It

- Click **Insert, Curve, Curve Through XYZ Points**.
- Or, click **Curve Through XYZ Points**  on the Curves toolbar.

Entering Points “On the Fly”

If you haven’t created a text file containing the locations beforehand, you can enter the X, Y, Z coordinates directly into the **Curve File** dialog. In addition, once you have done that, you can save the point list as a file for reuse. To do this, follow this procedure:

Note

The curve is created *outside* of a sketch. Therefore, the X, Y, and Z are

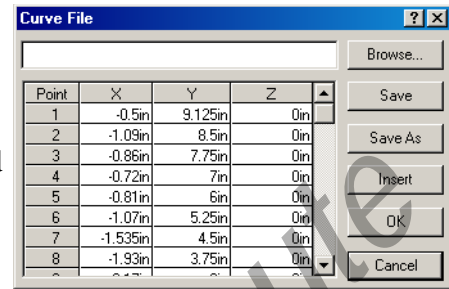
interpreted with respect to the Front (XY) coordinate system.

Double-click in the upper-left cell (top row, under the heading **Point**) and the system will open a row for the first coordinate point using the default values of X=0.0, Y=0.0, and Z=0.0.

Type in the appropriate values. Use the **Tab** key on the keyboard to move from one cell to another or just double-click each cell in turn.

Double-click in the next cell *below* Point #1 to add more rows. If you need to, you can insert a row in the middle of the list. Highlight the row by single-clicking the number in the point column and click the **Insert** button.

If you anticipate using this data set again, you can save it to a file using the **Save** button. If you are editing an existing file, **Save** will overwrite the original file; **Save As** will save a copy of it.

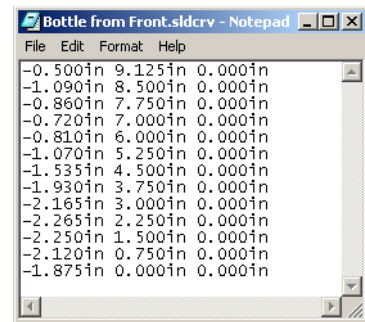


Reading Data From a File

Instead of entering the point data directly, we will browse for a file and read the data from it.

The files used here must be ASCII text files. You can use spaces or tabs between the columns of X, Y and Z coordinates. One easy method of creating the file is to use the Notepad accessory that comes with Windows.

Remember: the curve is created *outside* of a sketch. Therefore, the X, Y, and Z are interpreted with respect to the Front coordinate system.



Editing the Curve

If you need to modify the data points associated with a curve created through a data point set, use **Edit Feature**, the same as you would for any feature. When editing the definition of the curve, you have several options:

- Browse for and substitute a replacement file.
- Edit the existing point list.
- Edit the original file and read it in again.

Procedure

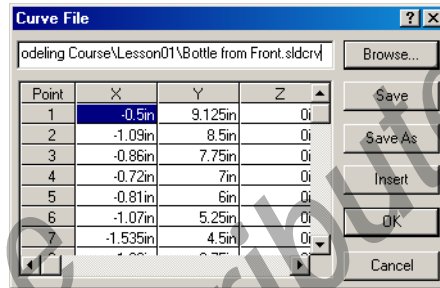
Begin by opening a new part using the Part_IN template.

1 Insert curve.

On the Curves toolbar, click **Curve Through XYZ Points** .

2 Select the file.

Click on **Browse** and select the file **Bottle** from **Front.sldcrv** from the directory.



The file contents are read into the dialog and separated into columns.

Note

The browser can be set to search for Curves (*.SLDCRV) or Text Files (*.txt).

3 Add the curve.

Click **OK** to add the curve to the part. A smooth spline curve is created using the points contained in the file as shown at the right in an **Front** view. A feature named **Curve1** appears in the **FeatureManager** design tree.

4 Create the second guide curve.

Click **Curve Through XYZ Points**  again.

From the browser, select the file **Bottle** from **Side.sldcrv**.

Click **OK** to create the second guide curve. This curve represents the shape of the bottle when viewed from the side.

The illustration at the right shows both guide curves in a **Trimetric** view orientation.

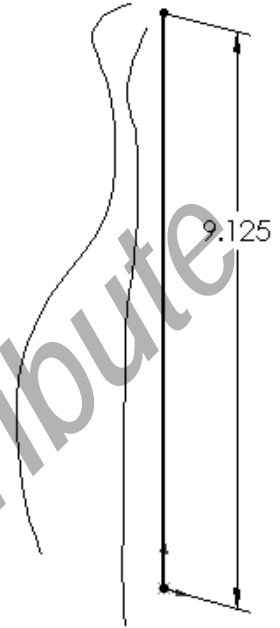


5 Sweep path.

Select the **Front** reference plane and open a sketch.

Sketch a vertical line, starting at the **Origin**. Dimension this line to a length of **9.125"**.

This will be used as the sweep path.

**Introducing:
Insert Ellipse**

Sketching an ellipse is similar to sketching a circle. Position the cursor where you want the center and drag the mouse to establish the length of the major axis. Then release the mouse button. Next, drag the outline of the ellipse to establish the length of the minor axis.

Important!


To fully define an ellipse you must dimension or otherwise constrain the lengths of the major and minor axes. You must *also* constrain the orientation of one of the two axes. One way to do this is with a **Horizontal** relation between the ellipse center and the end of the major axis.

Where to Find It

- Click **Tools, Sketch Entity, Ellipse**.
- Or, click **Ellipse**  on the Sketch toolbar.

6 Sweep section.

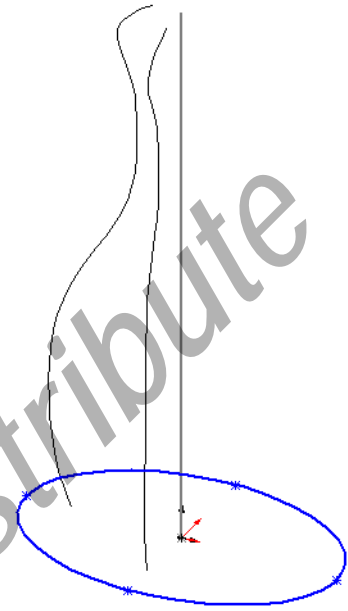
Select the Top reference plane and open a sketch.

On the Sketch toolbar, click the **Ellipse** tool  and sketch an ellipse with its center at the Origin.

7 Relating the sweep section to the guide curves.

We want the profile of the sweep section to be related to the guide curves. This way the guide curves will control the size of the ellipse. We can do this using a **Pierce** or a **Coincident** relation. This is why we created the guide curves *before* the profile.

Press the **Ctrl** key, and select the point at the end of the major axis and the first guide curve. Right-click, and select **Pierce**. Repeat this procedure for the minor axis and the second guide curve.



8 Fully defined.

Since the **Pierce** relation on the major axis defines its size *and* orientation, we do not need to further constrain it. If we had used a dimension to control the size of the major axis, we would need to control the orientation of the major axis in some way.

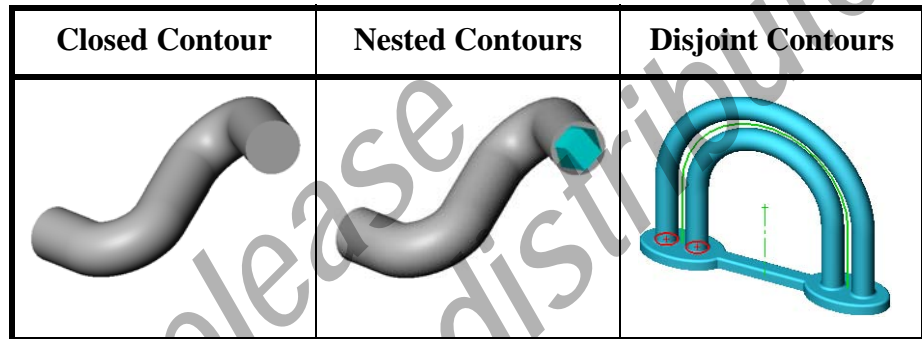
9 Exit the sketch.

The sweep section is now fully defined so you can exit the sketch. We are now ready to sweep the first feature.

Unlike extruded or revolved features, swept features cannot be created while active in a sketch. You must exit the sketch first. This is because swept features require multiple sketches which you identify individually.

Sweeping


The **Sweep** option creates a cut or boss that uses at least two pieces of geometry, a **Profile** and a **Path**. The profile (usually a closed sketch) is the cross sectional shape that is pushed along the path. The path (usually an open contour sketch or curve) is used to orient the profile in space. Other components can be added to further define the sweep. One or more **Guide** curves can be used to shape the profile as it moves along the path. There are several options for the profile sketch.



Introducing:
Insert, Boss, Sweep

Insert, Boss, Sweep creates a feature from two sketches: a sweep section and sweep path. The section is moved along the path, creating the feature.

Where to Find It

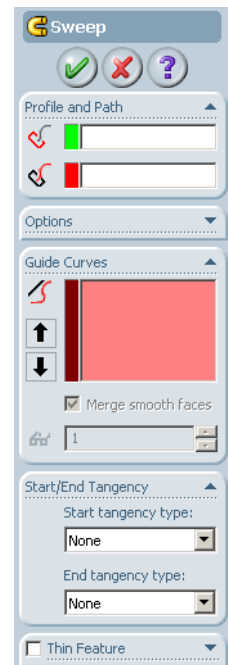
- Click **Sweep Boss/Base**  on the Features toolbar.
- Or, click **Insert, Base/Boss, Sweep**.

Sweep Dialog

The **Sweep** dialog contains selection lists for several types of objects: **Profile**, **Path** and **Guide Curves**. It also has options to determine how the system orients the sections while sweeping.

The dialog is divided into five sections or group boxes:

- **Profile and Path**
- **Options**
- **Guide Curves**
- **Start/End Tangency**
- **Thin Feature**



Options

The **Options** group box contains one or more of the following controls depending whether the sweep is a boss or a cut, a base feature, or a multibody:

■ Orientation/twist type

With a simple sweep, the orientation of the profile is controlled by choosing either **Follow path**, **Keep normal constant**, **Twist along path** or **Twist along path with normal constant**.

If the sweep includes guide curves, the orientation of the profile can be controlled by choosing either: **Follow path and 1st guide curve**, **Follow 1st and 2nd guide curves**. This is optional.

■ Merge tangent faces

With this option *on*, it merges tangent faces together, creating an approximation. Planar, cylindrical and conic faces are *not* merged.

■ Show preview

With this option *on*, it displays a shaded preview of the sweep, changing as each component is added. The more complex the sweep, the longer a preview takes.

■ Merge result

With this option *off*, the sweep generates an additional solid body.

■ Align with end faces

With this option *on*, it will continue the sweep beyond the geometric end. For more information, see *Align with End Faces* on page 104.

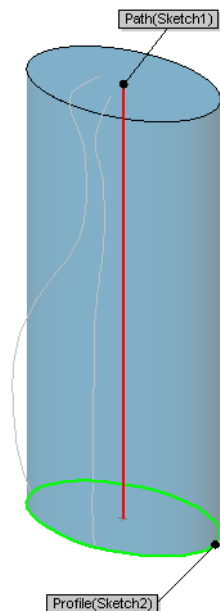
10 Sweep PropertyManager.

Click **Sweep Boss/Base** , or click **Insert, Base, Sweep** to access the **Sweep** PropertyManager.

11 Select profile and path.

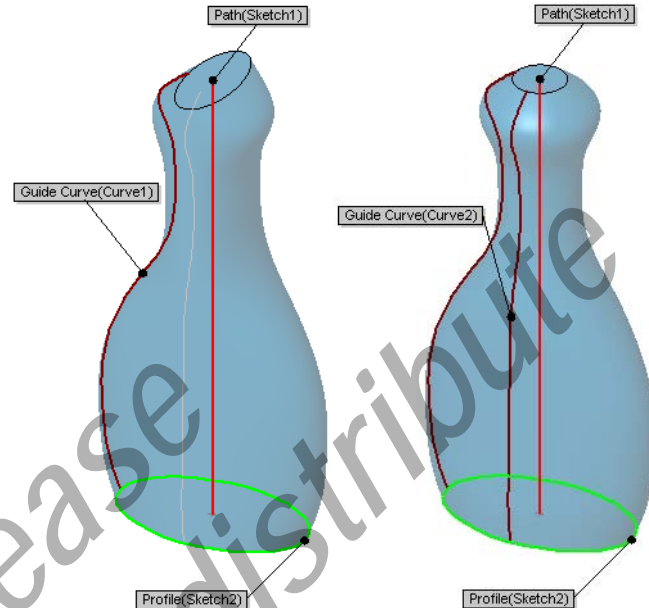
Make sure the **Profile** box is active, and select the ellipse. When you select the profile, the **Path** box automatically becomes active. Select the vertical line for the path. Callouts appear on each selection.

The preview displays the result without the effect of any guide curves.




- 12 Guide curves.**
Expand the **Guide Curves** group box. Click in the selection list, and select the two curves indicated.

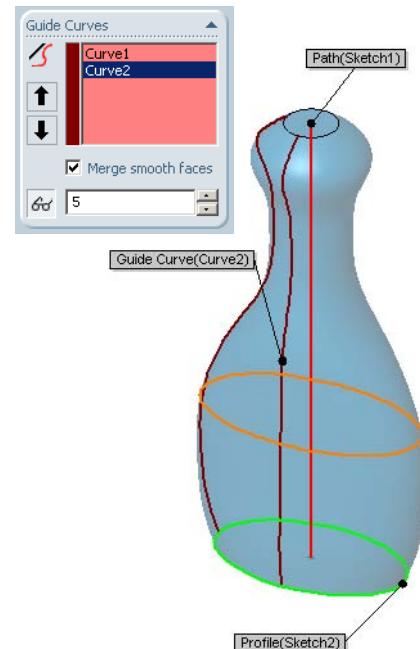
A callout appears only on the last guide you select.



Showing Intermediate Sections

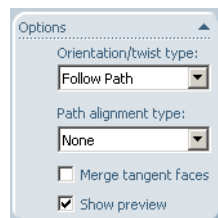
When sweeping a complex shape, you can see how the intermediate sections will be generated by clicking the **Show Sections** option. When the system computes the sections, it displays a spin box listing the number of the intermediate section. You can click the up and down arrows to display any of them.

- 13 Showing Sections.**
Click the **Show Sections** button , and use the spin box to display the intermediate sections. Notice how the shape of the ellipse is driven by its relationship with the guide curves.



- 14 Options.**
Expand the **Options** group box, and make sure that the default **Follow Path** is selected.

Click **OK**.



15 Finished sweep.

The swept feature is shown at the right in a Trimetric view.



The Label Shape

The shape of the label is created using a sketch that is projected onto the face of the bottle. The curve that is generated will be used as the **Sweep Path** of another swept feature. The sketch is already built and has been stored as a library feature.

Library Features

Library Features are generally applied using the Design Library (see the *Essentials: Parts and Assemblies* manual) but can also be dragged and dropped from the **File Explorer** or Windows Explorer.

File Explorer

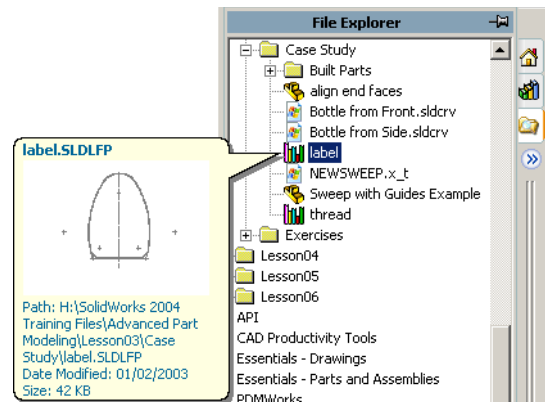
The **File Explorer** is used to search drives and folders for SolidWorks file types. The files can be dragged and dropped into SolidWorks.

16 File Explorer.

Click the **File Explorer** tab

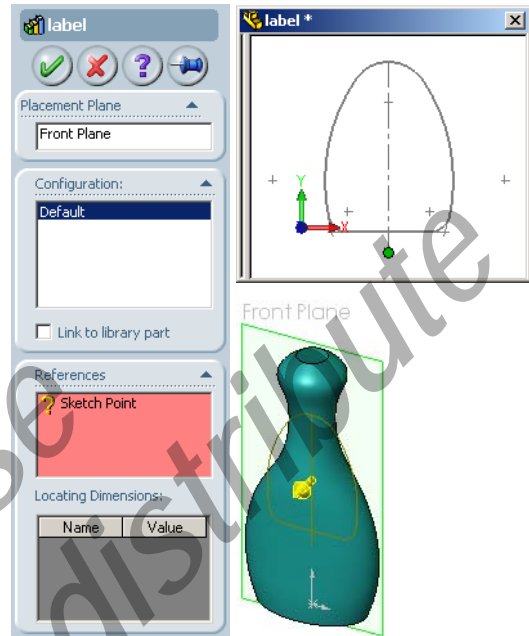
 of the Task Pane.

Double-click the folders Lesson 2 and Case Study to find the library feature label.



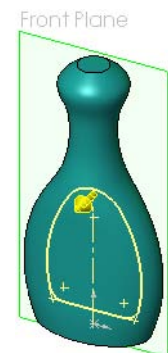
17 Drag and drop.

Show the Front Plane of the part. Drag the label from the File Explorer and drop it on the Front Plane.

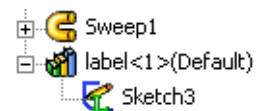
**18 References.**

Select the **Sketch Point** reference and click the part's Origin. Although this reference is not required, selecting it avoids having to repair the dangling relation.

Click **OK**.

**The Library Feature Folder**

The sketch appears in the FeatureManager design tree in a folder named label<1>. The actual sketch cannot be used in this form, it must be removed from the library feature folder.

**Tip**

Dissolve Library Feature can be used to break down the LibFeat folder. This removes the library feature icon and causes each of the features it contained to be listed individually in the FeatureManager design tree.

Working with a Non-planar Path

There are several techniques for creating non-planar paths. During the remainder of this example we will examine the two techniques:

- Projecting a sketch onto a surface.
- Creating a helix.


Projecting a Sketch onto a Surface

In the next part of this example, we will create a projected curve to use as the sweep path for the label outline on the bottle. We will do this by projecting a 2D sketch onto the curved surface of the bottle. The sketch was created using a **Library Feature**.

Introducing: Insert Projected Curve

Projected Curve projects a sketch onto a face or faces of the model. When these faces are curved, the result is a 3-dimensional curve. This command can also merge two orthogonal sketches into one 3D curve.

Where to Find It

- Click **Project Curve**  on the Curves toolbar.
- Or, click **Insert, Curve, Projected**.

19 Projected Curve dialog and preview.

Click **Project Curve** , or click **Insert, Curve, Projected**.

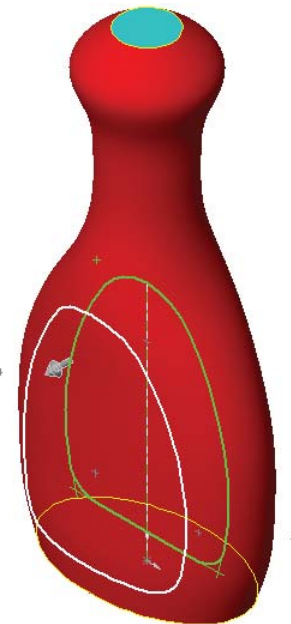
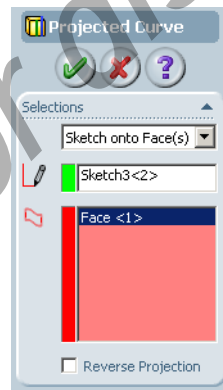
Select the **Sketch onto Face(s)** option from the list.

20 Selections.

Click in the **Sketch to Project** list and select the sketch. Click in the **Projection Faces** list and select the model face.

By default, the system projects the sketch normal to the sketch plane (along the positive Z axis). If you want to project the curve onto the back of the bottle, click **Reverse Projection**.

Click **OK**.



21 Projected curve.

The system projects the sketch onto the front surface of the bottle. This curve will be used as the sweep path to create a boss to outline the area the label area on the bottle.

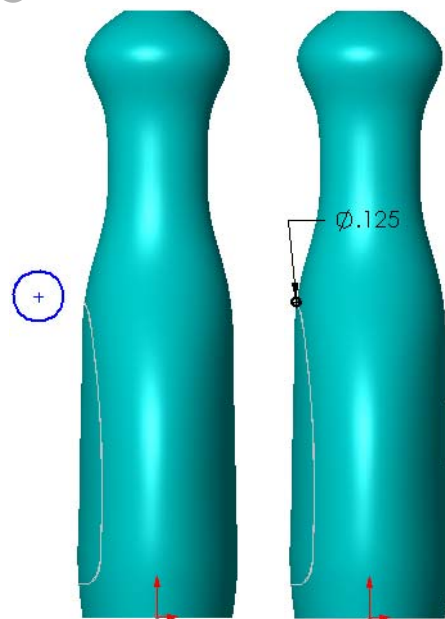
**22 Sketch the profile.**

Change to a Right view and select the Right reference plane. Open a sketch and draw a circle in any convenient location.

23 Pierce relation.


Add a **Pierce** relation between the center of the circle and the projected curve to define its location. Dimension the circle to **0.125"** diameter.

The projected curve pierces the sketch plane in two places: at the top and the bottom. The system chooses the pierce point closest to where you select the curve. If you want the circle located at the top, select the projected curve near the top. It's that simple.



24 Sweep the boss for the label outline.

Exit the sketch.


Click **Sweep Boss/Base** . Select the circle as the **Profile** and the projected curve as the **Path**.

Click **OK**.

Notice the system has no difficulty sweeping a feature with the profile located at the middle of a closed path.



25 Add the neck.

Select the top face of the bottle and open a sketch. Use **Convert Entities**  to copy this edge into the active sketch. Extrude the sketch upward a distance of **0.625"**.



Variable Radius Filleting

A variable radius fillet runs around the bottom of the bottle. Variable radius fillets are defined by specifying a radius value for each vertex along the filleted edges and optionally, at additional control points along the edges. Variable radius control points operate as follows:

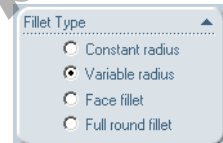
- The system defaults to three control points, located at equidistant increments of 25%, 50%, and 75% along the edge between the vertices. You can increase or decrease the number of control points.

- You can change the position of any control point by changing the percentage assigned to that control point. You can also drag any control point, and its assigned percentage will update accordingly.
- Although there is a visual display of the control points, they are only active if you select them and assign a radius value.
- Inactive control points are red. Active control points are black, and have a callout attached to them indicating the assigned radius and percentage values.

In this case there only a single vertex on the bottom edge of the bottle. Therefore, we will use control points.

26 Fillet the bottom.

Click **Fillet**  on the Features toolbar. For **Fillet Type**, choose **Variable radius**.

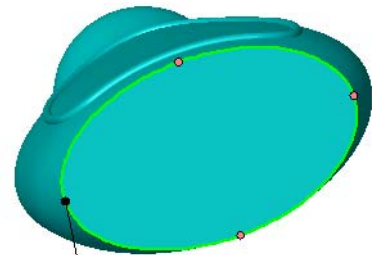


27 Select the edge.

Select the bottom edge of the bottle. A callout appears at the vertex, and three control points appear along the edge.

Note

For variable radius filleting, you must select an edge. You cannot select a face.



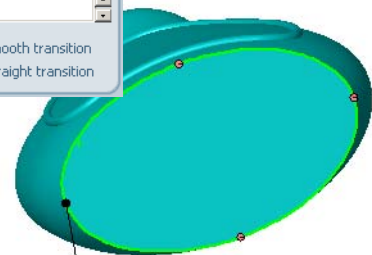
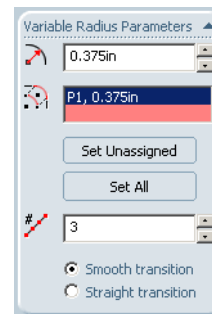
28 Assign radius value to the vertex.

Click the callout and enter a radius value of **0.375"**.

The assigned radius also appears in the vertex list in the PropertyManager.

Tip

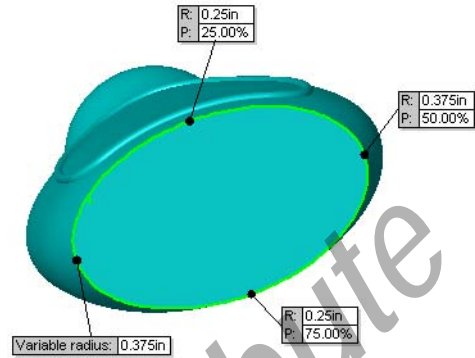
The buttons **Set Unassigned** and **Set All** are used to assign one radius value to many vertices (not control points) at once. If most, but not all, vertices have the same radius, it is faster to assign the same value to all of them, and then change only those that require a different value.



29 Radius values.

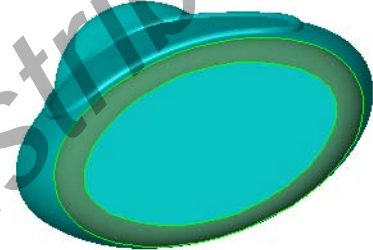
Click the control points and use the callouts to set the radius **R** to **0.25"** and **0.375"** as shown. Leave the positions **P** at their default values of **25%**, **50%**, and **75%** as shown in the illustration at the right.

Click **OK** to create the fillet.



30 Result.

The result of the variable radius fillet is shown at the right. The fillet forms a closed loop varying smoothly from 0.375" to 0.25" to 0.375" to 0.25" and back to 0.375" at the start.



Another Approach to Filleting

This portion of the example was based on the assumption that the design intent called for exact radius values at specific locations around the base of the bottle. Let's consider a different approach based on a different design requirement.

Look at the bottle from the front. The edge of the fillet, also called the rail, is not straight across the front of the bottle. Let's examine how we would fillet the edge if the design requirement specified this edge must be straight and located 0.375" from the bottom face. In other words, rather than have the fillet define the rails, we will define where the rails should be, and let the system compute the fillet radius.



Adding a Split Line

A split line is used to divide model faces into two. Split lines are created like any other sketched feature. They can be one or more connected sketch entities. They must be oriented so that they will pass through model faces when projected normal to the sketch plane.

Introducing: Split Lines

Insert, Curve, Split Lines uses one or more curves to split one model face into two. The curves are sketched on a plane and projected onto the faces to be split.

Where to Find It

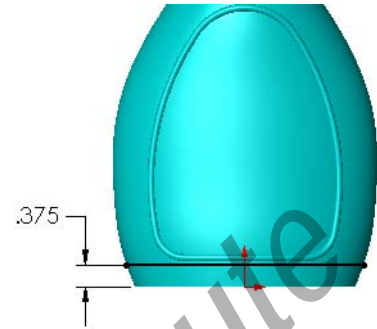
- Click **Insert, Curve, Split Line**.
- Or on the Curves toolbar, click **Split Line** .


31 Delete the fillet.

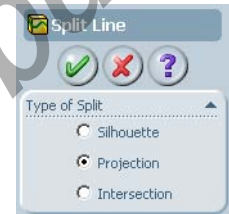
Right-click the variable radius fillet, and select **Delete Feature**.

32 Sketch the split line.

Select the **Front** reference plane, and open a sketch. Sketch a horizontal line making its ends coincident to the silhouette edges of the bottle. Dimension it as shown in the illustration.

**33 Projection split line.**

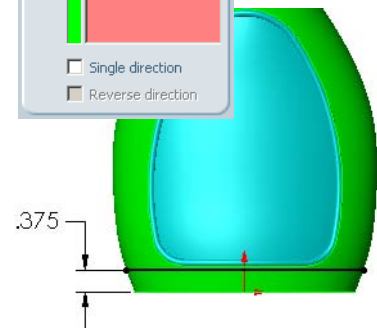
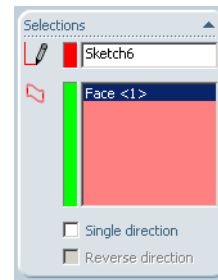
Click **Split Line** , or click **Insert, Curves, Split Line**. Since we are still active in the sketch, the **Projection** option is automatically chosen. This option projects the curve through the model onto the selected faces.

**34 Select faces.**

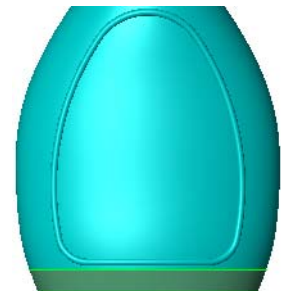
Click in the **Faces to Split** list to activate it, and select the face that forms the main body of the bottle.

Make sure the **Single direction** check box is cleared. Since the sketch is on the **Front** plane, it is “inside” the bottle. The sketch must be projected in both directions to completely split the face.

Click **OK** to complete the command.

**35 Results.**

The horizontal sketch line breaks the single face into two faces.



Face Fillets

A face fillet differs from an edge fillet in that instead of selecting a edge, you select two sets of faces. The advanced options enable you to use geometry to define the radius of the fillet instead of specifying a numeric radius value. This is very powerful.

Introducing: Face Fillet

The **Fillet** command has an additional group box, **Fillet Options**, where a **Hold Line** can be assigned to define the fillet's tangent edge or rail. Defining the rail of the fillet defines the fillet's radius. In this example, the edge created by the split line will be used.

Where to Find It

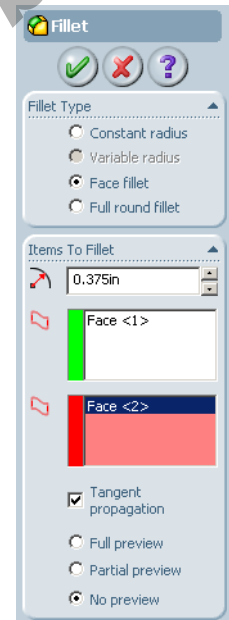
- **Face Fillet** is located on the **Fillet PropertyManager**.

Note

Since the **Hold line** will define the radius, you do not need to enter a radius value. Also, when you expand the **Fillet Options** group box and select the hold lines, the radius field disappears.

36 Insert Fillet.

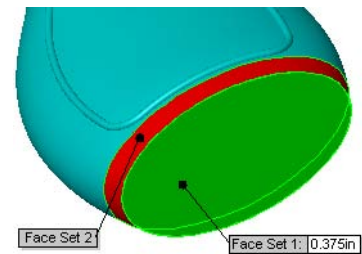
Click **Fillet** . In the **Fillet Type** group box, choose the **Face Fillet** option.



37 Select the faces.

Verify that the **Face Set 1** selection list is active and select the bottom face of the bottle.

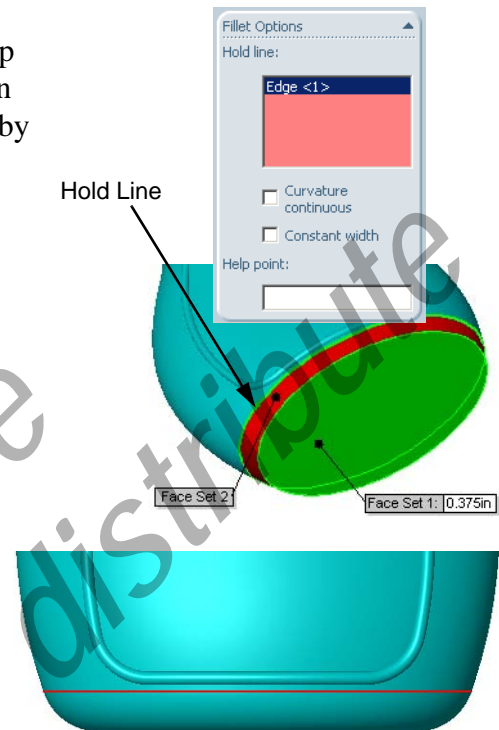
Activate the selection list for **Face Set 2** and select the face created by the split line.



38 Fillet Options.

Expand the **Fillet Options** group box. Click in **Hold line** selection list, and select the edge created by the split line.

Click **OK** to create the fillet.

**39 Results.**

The face created by the split line (Face Set 2) is completely removed. The fillet is created with a variable radius defined such that the fillet ends exactly on the hold line.

Analyzing Geometry

SolidWorks has several tools that are used to obtain information and to assess the quality of curves and surfaces. They include:

- **Display Curvature**
- **Show Curvature Combs**
- **Show Minimum Radius**
- **Show Inflection Points**
- **Zebra Stripes**

What is Curvature?


To avoid getting too deep into mathematics, we will use this working definition: Curvature is the reciprocal of the radius.

If a surface has a local radius of 0.25, it has a curvature of 4. The smaller the curvature value, the flatter the surface.

Introducing: Display Curvature

Displays the faces of the model rendered in different colors according to their local curvature values. You can assign different curvature values to the scale of colors. Red represents the largest curvature (smallest radius) and black represents the smallest curvature (largest radius).

Where to Find It

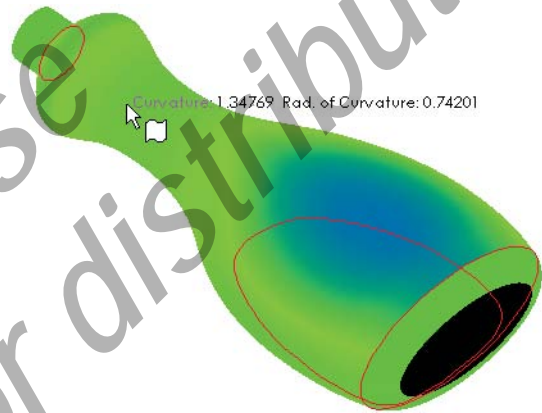
- Click **Curvature**  on the View toolbar.
- Or, click **View, Display, Curvature**.
- You can display the curvature for selected faces by right-clicking the face, and selecting **Curvature**.

Tip

Displaying the curvature can be system resource intensive. In many cases you can improve performance by displaying the curvature only on the face or faces that you want to evaluate.

40 Display Curvature.

Click **View, Display, Curvature**. The part is rendered in colors according to the curvature of the faces. As you move the cursor over a face, a print out appears giving both the curvature and radius of curvature values.



41 Look at the fillet.

Notice the dramatic change in color from the body of the bottle to the fillet around the bottom. This indicates that although the fillet is tangent to the body, it is not curvature continuous. This means the faces do not have the same curvature at the edge where they meet.

42 Turn off curvature display.

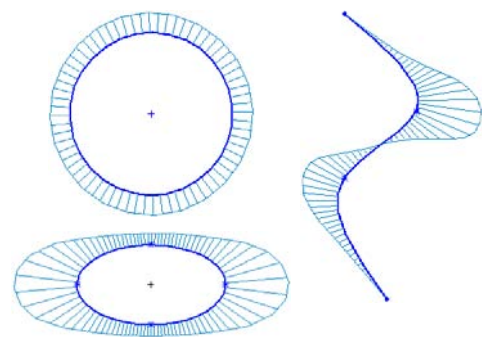
Click **View, Display, Curvature** to turn off the curvature display.

Show Curvature Combs

Provides visual representation of the slope and curvature of most sketch entities. You can use **Show Curvature Combs** to evaluate splines before they are used to sweep or loft solid features. You can also indirectly evaluate curved faces by generating intersection curves and then evaluating the curves.

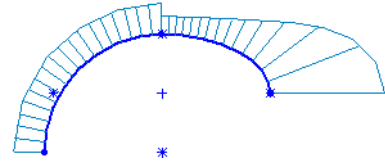
Introducing: Show Curvature Combs

Show Curvature Combs gives a graphic representation of the curvature in the form of a series of lines called a *comb*. The length of the lines represents the curvature. The longer the line, the greater the curvature (and smaller the radius).



When the comb crosses the curve, it indicates an inflection point. An inflection point is where the curve changes direction. This only applies to splines.

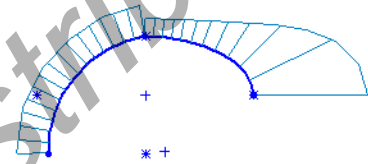
You can use **Show Curvature Combs** to learn other things about how curves are connected. Look at the illustration at the right. The two sketch entities are a circular arc and a quarter of an ellipse.



The two curves are tangent but not matched in curvature. This is indicated by the fact that the curvature lines are the common endpoint are:


- Collinear (indicates tangency).
- Not the same length (different curvature values).

In the illustration at the right, the two entities are *not* tangent as indicated by the fact that the curvature lines at the common endpoint are *not* collinear.



The curvature comb remains visible when you close the sketch (unless the sketch has been made into a feature). To remove the display, right-click the sketch entity, and select **Show Curvature Combs** again from the shortcut menu to remove the check mark.

Where to Find It

- Click **Show Curvature Combs**  on the Spline Tools toolbar.
- Or, right-click the sketch entity, and select **Show Curvature Combs**.

Intersection Curves

Show Curvature Combs only works on sketch entities. In situations where you do not have a sketch entity, you will have to apply other techniques. For example, to evaluate a face or surface, one technique is to generate an intersection curve.

Introducing: Intersection Curve

Intersection Curve opens a sketch and creates a sketched curve at the following kinds of intersections:

- A plane and a surface or a model face.
- Two surfaces.
- A surface and a model face.
- A plane and the entire part.
- A surface and the entire part.

Where to Find It

- Click **Intersection Curve**  on the Sketch toolbar.
- Or, click **Tools, Sketch Tools, Intersection Curve**.

43 Intersection curve.

Select the Front Plane reference plane and open a sketch.

Click **Intersection Curve**  on the Sketch toolbar.

Select the face of the fillet and the main body of the bottle.

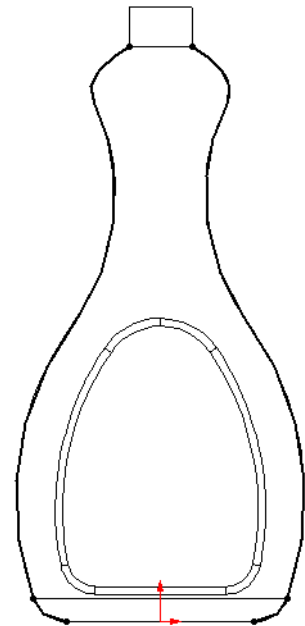


44 Results.

The system generates intersection curves between the sketch plane and the selected faces. Two sets of intersection curves are created because the reference plane intersects the faces in two locations. Only one set is needed for this example.

45 Turn off the intersection curve tool.

Click **Intersection Curve**  again to turn off the tool.

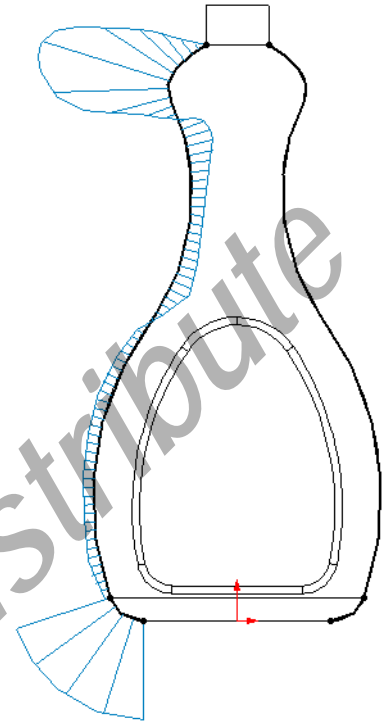


46 Show Curvature Combs.

Right-click one set of the intersection curves and select **Show Curvature Combs**.

Note the following:

- The fillet has a circular cross section as indicated by the curvature comb.
- The fillet and the side of the bottle are matched in tangency.
- The fillet and the side of the bottle are *not* matched in curvature as indicated by the different lengths of the curvature combs.

**Color**

The color of the curvature comb is controlled by **Temporary Graphics, Shaded** which is listed under **Tools, Options, System Properties, Color**. Depending on the color of the viewport background, you may want to change the temporary graphics color for maximum visibility.


47 Modify Curvature Scale.

Right-click the intersection curve and choose **Modify Curvature Scale**. Slide the bar right (decrease) or left (increase) to change the scale of the curvature combs.

**Show Minimum Radius**

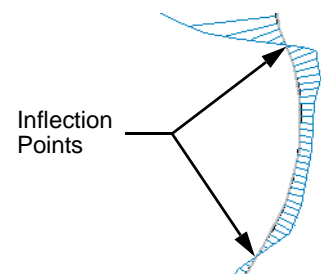
Show Minimum Radius (of curvature) can be used to graphically display the position and value of the minimum radius of curvature on the curve. This is important information for shelling and offset geometry.

Where to Find It


- Click **Show Minimum Radius**  on the Spline Tools toolbar.
- Or, right-click the sketch entity, and select **Show Minimum Radius**.

Show Inflection Points

Inflection Points are those points on a curve where the curvature changes direction, shown in the curvature comb display as a crossover. These points can be shown on the curve.

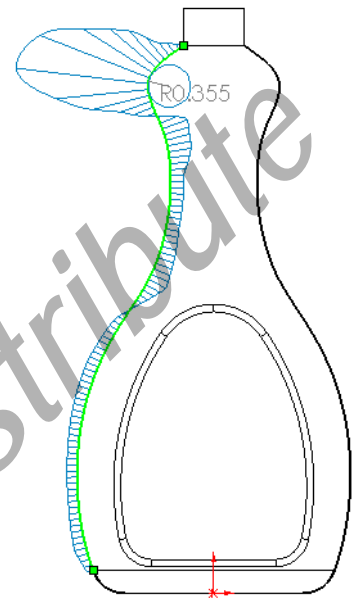


Where to Find It

- Click **Show Inflection Points**  on the Spline Tools toolbar.
- Or, right-click the sketch entity, and select **Show Inflection Points**.

48 Minimum Radius.

Right-click the curve and select **Show Minimum Radius**. A graphic circle, tangent to the curve, appears on the screen. A *radius* value is attached to the circle.



49 Inflection Points.

Right-click again turn off **Show Curvature Combs**.

Turn on the **Show Inflection Points** option.

A small double facing arrow symbol appears at each inflection point in the curve.

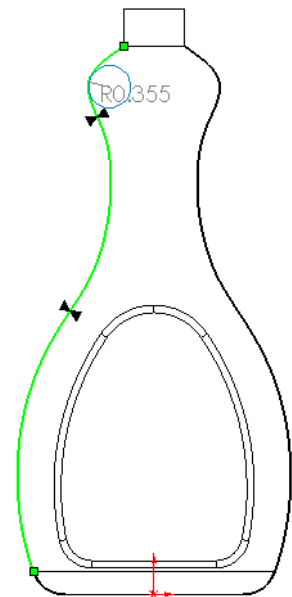
50 Turn off the displays.

Right-click the intersection curves, and select **Show Inflection Points** and **Show Minimum Radius**.

51 Exit the sketch.

52 Rollback.

Right-click the sketch, and select **Rollback**.



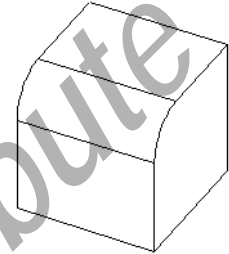
Zebra Stripes

Zebra Stripes simulate the reflection of long strips of light on a very shiny surface. Using zebra stripes you can see wrinkles or defects in a surface that may be hard to see with a standard shaded display. Also, you can verify that two adjacent faces are in contact, are tangent, or have continuous curvature.

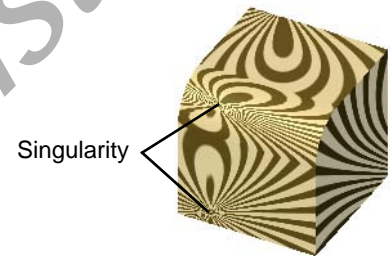
**Introducing:
Zebra Stripes**

Properly interpreting the zebra stripe display requires some explanation. To illustrate, we will look at some examples using a box with a fillet.

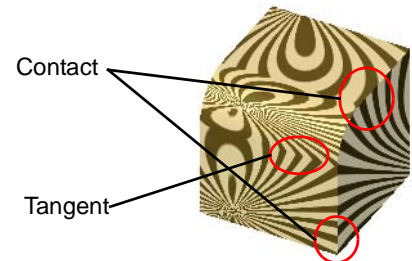
The first point to consider is the pattern of the stripes. By default, the part appears to be inside a large sphere that is covered on the inside with strips of light. The zebra stripes are always curved (even on flat faces) and display singularities.

**What is a
Singularity?**

A singularity is where the zebra stripes appear to converge to a point.

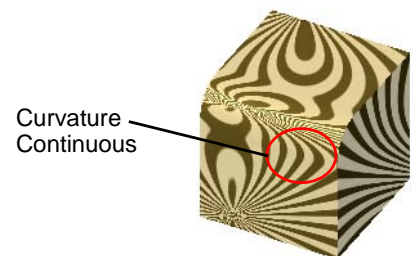
**Boundary
Conditions**

The next point to consider is how the zebra stripes are displayed where they cross the boundaries of faces. Evaluating the zebra stripe display will give you information about how the faces within a part are blended one into the other.



There are three boundary conditions:

- Contact – the stripes do not match at the boundary.
- Tangent – the stripes match, but there is an abrupt change in direction or a sharp corner.
- Curvature continuous – the stripes continue smoothly across the boundary. Curvature continuity is an option for face fillets.

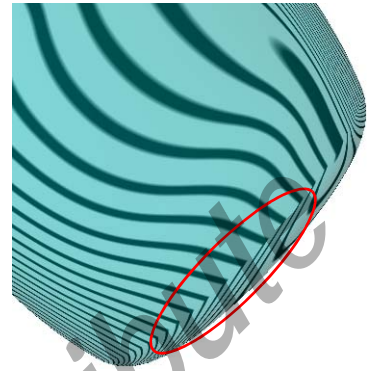
**Where to Find It**

- Click **Zebra Stripes**  on the View toolbar.
- Or, click **View, Display, Zebra Stripes**.

53 Zebra stripes.

Click **View, Display, Zebra Stripes**.

Rotate the view and watch how the pattern of stripes changes. Pay particular attention to how the stripes blend from the face of the bottle to the fillet. The fillet is matched in tangency, but not curvature.



Tip

Save this view display state so you can return to it later.

**Curvature
Continuous Fillets**

The **Curvature continuous** option for face fillets can create a smoother transition between adjacent surfaces. Only face fillets can be curvature continuous. There are two ways to specify the radius of a curvature continuous, face fillet:

1. Specify a **Radius** value.
2. Use the **Hold line** option. This requires *two* hold lines, one for each set of faces.

Where to Find It

- On the **Fillet PropertyManager**, select **Face fillet**, expand the **Fillet Options** group box, and click **Curvature continuous**.

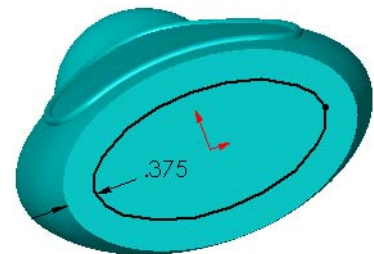
54 Turn off zebra stripes.

55 Rollback.

Right-click the fillet, and select **Rollback**.

56 Second split line.

Open a sketch on the bottom face and create an offset of **0.375"**. Use this sketch to split the bottom face.



Note

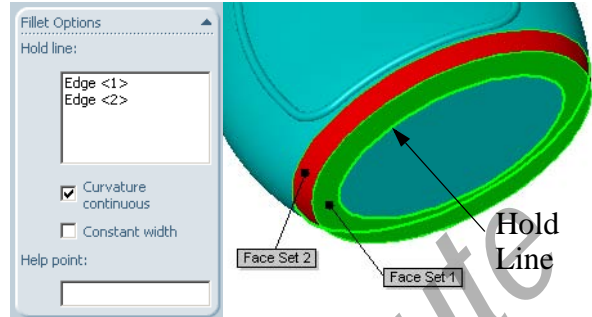
This will cause an error in the next step because the split line eliminates one of the faces that was selected for the face fillet.

57 Roll forward and Edit Feature.

One of the face set lists will be empty. Click in that list, and select the face created by the split line.

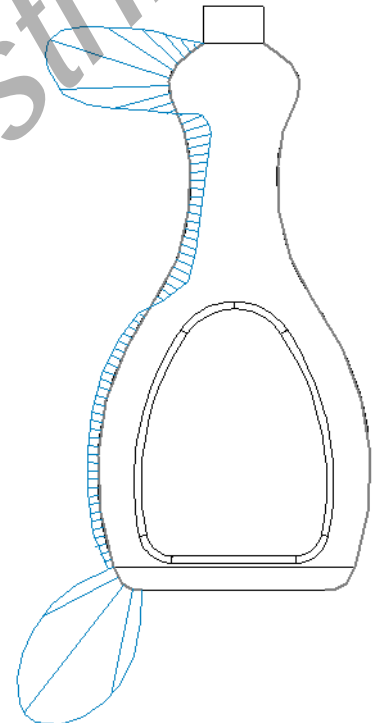
Click in the **Hold line** list, and select the edge of the face for the second hold line.

Click **Curvature continuous**, and **OK**.

**58 Inspect the curvature.**

Roll forward and examine the curvature of the intersection curves. Notice particularly how the curvature display for the fillet has changed. The unequal lengths of the curvature comb indicate that the fillet is not circular in cross section. This is understandable.

Curvature continuous fillets are not circular. Also, the last comb element on the body and the first element on the fillet are the same length. This indicates that the fillet is curvature continuous with the body of the bottle.

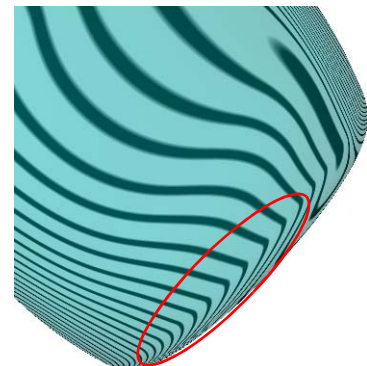
**59 Delete the sketch.**

Delete the sketch which contains the intersection curves. We do not need it any more.

60 Zebra stripes.

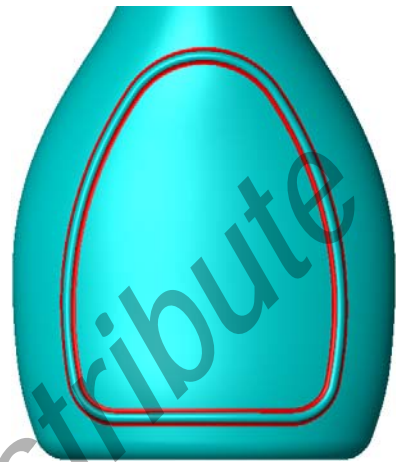
Click **View, Display, Zebra Stripes**.

Examine how the stripes blend from the body of the bottle to the fillet.

61 Turn off zebra stripes display.

Filleting the Label Outline

The next step is to create a fillet around the inside and outside edges of the label outline, shown here in red.

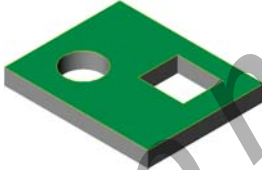
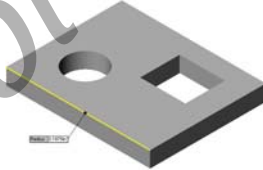
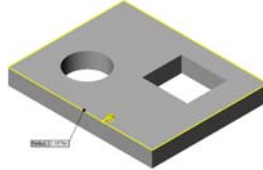
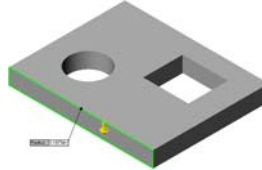
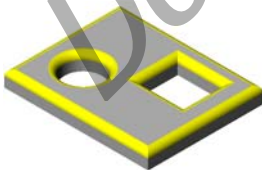
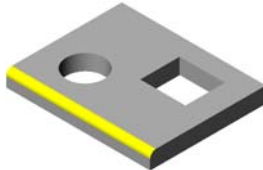
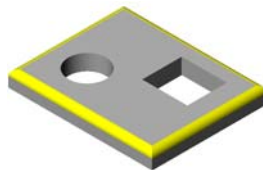
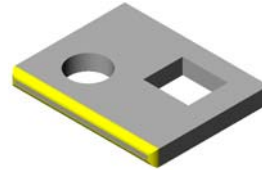


Selecting Edges

Filleting (other than a face blend fillet) depends on selecting edges. There are a number of different ways to select edges. You can:

- Select individual edges. If **Tangent Propagation** is enabled, selecting one edge will select other edges that form a tangent chain.
- Select a face. Selecting a face will fillet all the edges of that face.
- Select a loop.

Consider the examples below:

Select Face	Select Edge	Select Loop	Select Loop Click the handle to select the edges of the adjacent face.
			
			

What is a Loop?

A loop is a set of connected edges in a face. In a solid, an edge is always the boundary between two faces. Therefore, when you use loop selection on an edge, there are always two possible results. A handle points to the face whose edges are being selected. Clicking the handle selects the edges of the adjacent face.

Introducing: Select Loop

Select Loop can be used to select multiple, connected edges that constitute one loop of a face.

Where to Find It

- Right-click the edge, and select **Select Loop**.

62 Fillet the label outline.

Run a **0.060"** radius fillet around the inside *and* outside edges of the swept label outline. This fillet, shown here in **red**, has to be added *before* the bottle is shelled.

Experiment with different ways of selecting the edges to be filleted:

- Selecting tangent edges
- Selecting a face
- Selecting a loop


**Multi-thickness Shell**

The **Shell Feature** command gives you the option of creating a multi-thickness shell, in which some walls are thicker (or thinner) than others. You should decide what thickness represents the usual case, which is applied to most faces. Then, you should determine what thickness represents the exceptions, applied to fewer faces. In the case of the bottle, all faces are 0.020" thick *except* the neck, which is 0.060".

Shell the Bottle

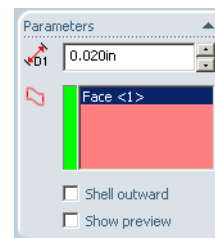
Create a multi-thickness shell, removing the top of the bottle neck. Use a wall thickness of **0.060"** for the neck and **0.020"** for all the other faces.

63 Shell command.

Click **Shell**  on the Features toolbar, or click **Insert, Features, Shell**.

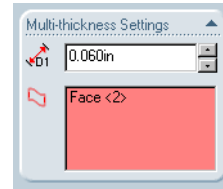
Set the **Thickness** to **0.020"** as the default.

For the **Faces To Remove**, select the top face of the bottle neck.

**64 Multiple thickness.**

Expand the **Multi-thickness Settings** section. Face selections here will not be the default thickness.

- 65 Select thicker faces.**
Click in the **Multi-thickness Faces** field and select the outside face of the bottle neck. Set the thickness to **0.060"**.



Click **OK** to create the shell.

- 66 Results shown in section view.**
The illustration at the right shows a section view, viewed from the back.

- 67 Save your work.**
We have invested a lot of time into this case study. Now would be a good time to save the file.

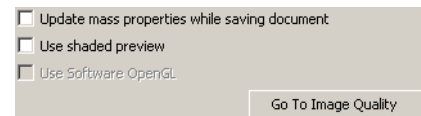


Performance Considerations

When working on a part like this one, performance tends to slow as the geometry gets more complex. Sweeps, lofts, variable radius fillets, and multi-thickness shells in particular have an impact on system resources and performance. There are, however, some steps you can take to minimize this impact and optimize system performance.

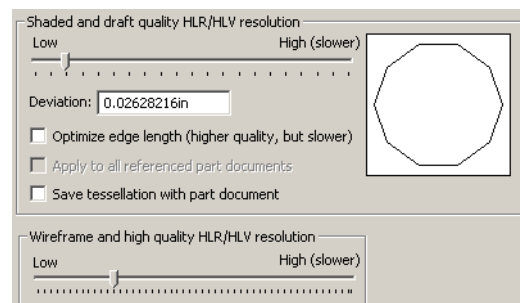
Performance Settings

The **Performance** tab for **Tools, Options, System Options** contains settings which affect all documents.



Turning off shaded/dynamic previewing and limiting updates to affected faces can speed up the process.

Image Quality settings for **Shaded** and **Wireframe** also have an impact on system performance. Use the lowest possible settings that still give acceptable image quality.



Suppressing Features

Suppressing a feature causes the system to ignore it during any calculations. Not only is it removed from the graphic display, the system treats suppressed features as if they aren't even there. This will significantly improve system response and performance when working with complex parts.


Parent/Child Relationships

Parent/child relationships affect suppressing features. If you suppress a feature, its children will automatically be suppressed also. When you unsuppress a feature (turn it back on again) you have the option of leaving its children suppressed or unsuppressing them as well.

The second implication of parent/child relations and suppressed features is that you cannot access or reference any of the geometry of a suppressed feature. Therefore, you need to give careful consideration to modeling technique when you suppress something. Don't suppress a feature if you will need to reference its geometry later.

Accessing the Suppress Command

There are several ways to access the **Suppress** command:

- On the Features toolbar click **Suppress** .
- On the pull-down menu click **Edit, Suppress**.
- On the right-mouse menu click **Feature Properties**.
- On the right-mouse menu click **Suppress**.


Interrupt Regeneration

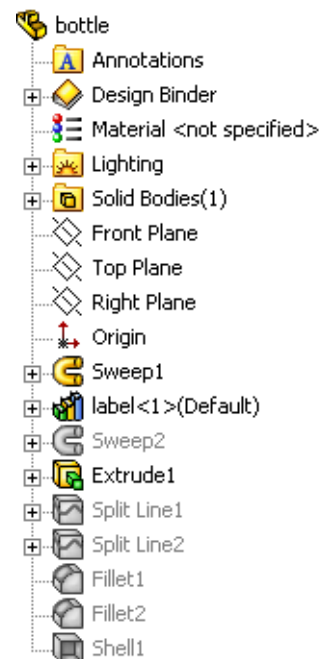
Press **Esc** to interrupt the regeneration of a part. This also works when opening parts, during rollback, and so on.

When you interrupt the regeneration of a part, the system completes regeneration of the current feature and then places the rollback bar after that feature.

68 Suppress features.

In the FeatureManager design tree, select the features for the label outline (Sweep2), the split line features (Split Line1 and Split Line2), the face blend fillet (Fillet1), the fillet around the label outline (Fillet2) and the multi-thickness shell (Shell1).

Click **Suppress**  on the Features toolbar, or click **Edit, Suppress**. The features are removed from the graphics window and grayed out in the FeatureManager tree.



Modeling Threads

Models can contain two types of threads: standard or cosmetic threads, and nonstandard threads. Standard threads are *not* modeled in the part. Instead, they are represented in the model and on the drawing using thread symbols, drawing annotations, and notes.

Nonstandard threads *should* be modeled. These threads, like the threads on the neck of this bottle, cannot simply be specified by a note on a drawing. Model geometry is needed because downstream applications such as NC machining, rapid prototyping, and FEA require it.

Creating a Helix

A thread is modeled by sweeping a profile along a helical path. The helix can also be used to sweep springs and worm gears.

The major steps in modeling threads are:

- **Create the helix.**
The helix is based on a sketched circle tied to the diameter of the neck.
- **Create the sketch for the cross section of the feature.**
The sketch is oriented with respect to the helix and penetrates the neck.
- **Sweep the sketch along the path (helix) either as a boss or a cut feature.**

In this example, the threads are a swept boss.

Introducing: Helix and Spiral

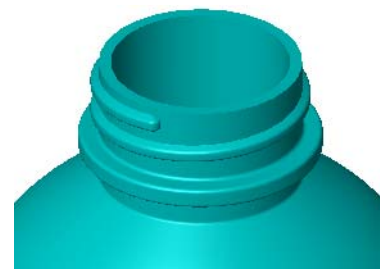
Insert, Curve, Helix/Spiral creates a helical 3D curve based on a circle and definition values such as pitch and number of revolutions. The curve can then be used as a sweep path.

Where to Find It

- Click **Helix and Spiral**  on the Curves toolbar.
- Or, click **Insert, Curve, Helix/Spiral**.

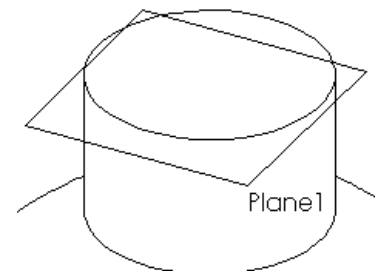
Procedure

In the remainder of this example, we will build the threads on the neck of the bottle as shown at the right.



69 Offset plane.


Create a reference plane offset **0.10"** *below* the top of the bottle neck. This is where the threads will start.

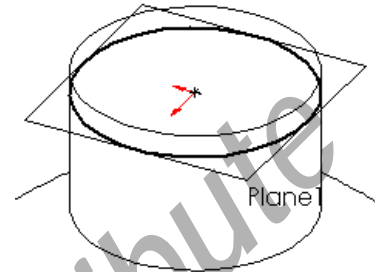



70 Insert sketch.

With this plane selected, open a new sketch.

71 Copy the edge.

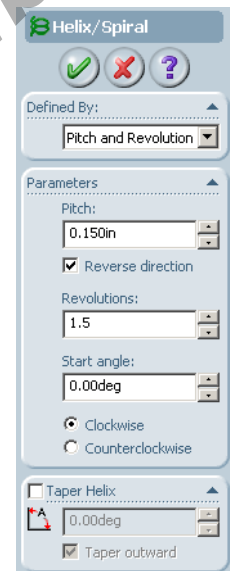
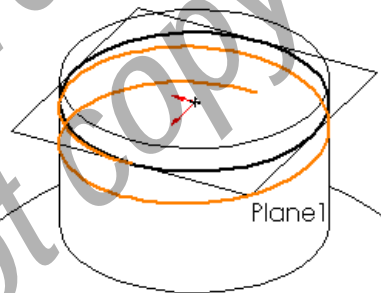
Copy the edge of the bottle neck into the active sketch using **Convert Entities** . This circle will determine the diameter of the helix.

**72 Create the helix.**

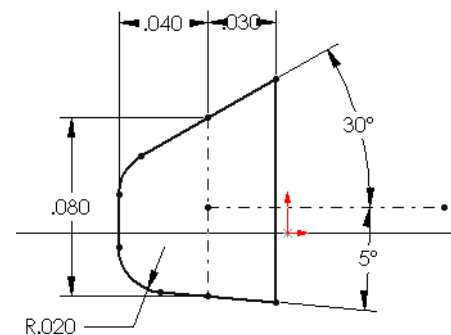
Click **Helix and Spiral** . The **Helix Curve** dialog is used to specify the definition of the helix. The threads have a **Pitch** of **0.15"** for **1.5 Revolutions**. The threads are **Clockwise** and go down the neck from a **Starting Angle** of **0°**.

As you change the parameters of the helix, the preview graphics update to show the result.

Click **OK** to create the helix.

**73 Insert a sketch.**

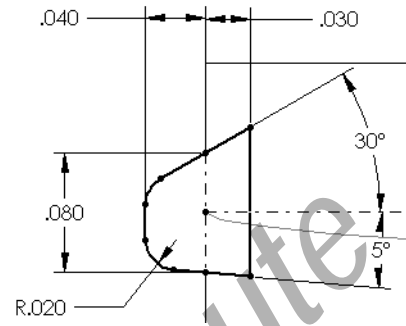
Using another library feature, insert the sketch used for the thread profile. Insert the library feature `thread.sldlfp` onto the Right reference plane.




74 Relations.

Edit the sketch of the library feature. Create a relation of **Collinear** between the horizontal centerline of the sketch and the plane Plane1.

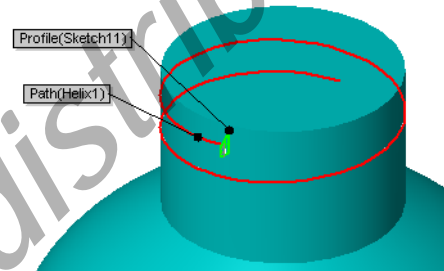
Use a silhouette edge to add a **Collinear** relation between the vertical centerline and the outer edge of the model. The sketch is now fully defined. Exit the sketch.



75 Sweep the threads.

Click **Sweep Boss/Base** . Select the sketch as the sweep section, and the helix as the sweep path.

Click **OK**.

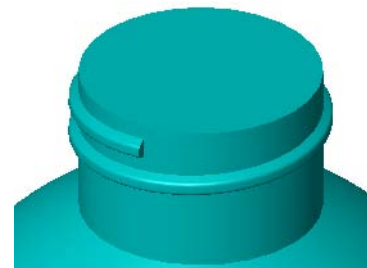


Note

If you are wondering what the option **Align with End Faces** is used for, we will cover a simple example explaining its purpose after we finish with the bottle. See *Align with End Faces* on page 104.

76 Results.

The results of sweeping the thread are shown at the right.

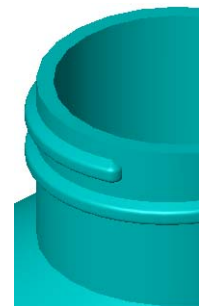


77 Add the finishing details.

An easy way to round off and finish the ends of the thread is to create a revolved feature. Do this for both ends of the thread.

Tip

An easy way to create the centerline that is needed for the revolved feature is to use **Convert Entities** to copy the vertical edge where the thread meets the body of the neck. Then change the line's properties to **Construction Line** and you have your centerline.



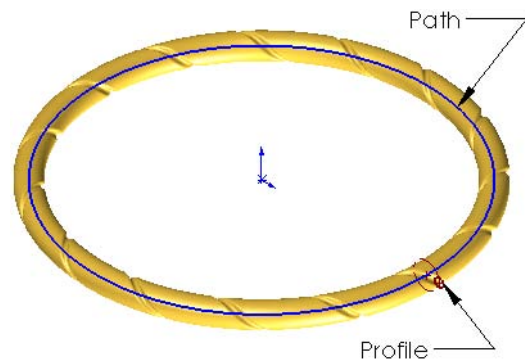
78 The finished bottle.

The bottle in this illustration has an added lip around the base of the neck. This is a simple extruded boss. Many bottles have this lip to provide a secure grip for those shrink-wrapped, tamper-evident seals that are so common.

**Using Twist**

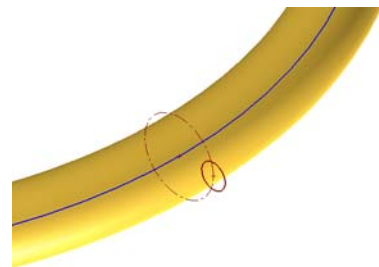
The **Twist Along Path** option can be used with **Sweep** to twist the **Profile** around the **Path** and move along it.

The twist can be defined by a value of **Degrees**, **Radians** or **Turns** along the entire length of the path.

**1 Open part.**

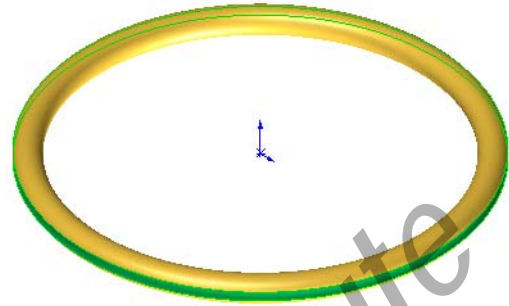
Open the part Twisted Ring. It contains two sketches:

- Sketch2 is the **Profile**
- Sketch3 is the **Path**



2 Sweep.

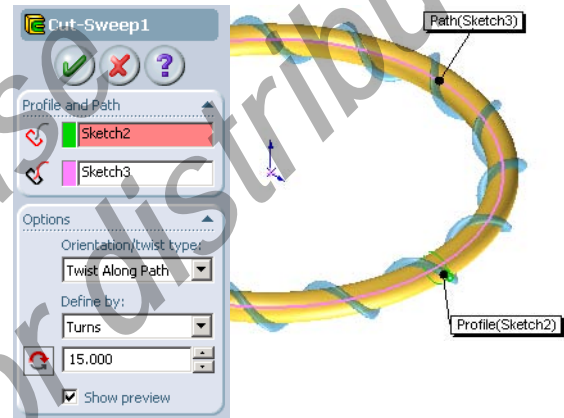
Click **Cut Sweep**  and create a simple cut sweep using the default **Follow Path** option.



3 Edit Cut - Sweep1 feature.

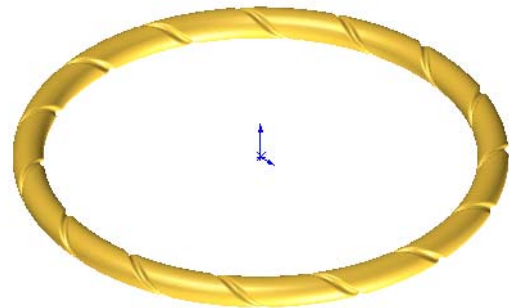
Edit the Cut - Sweep1 feature and set the **Orientation/twist type** to **Twist Along Path**. Use **Define by: Turns** and **15 turns**.

Click **OK**.



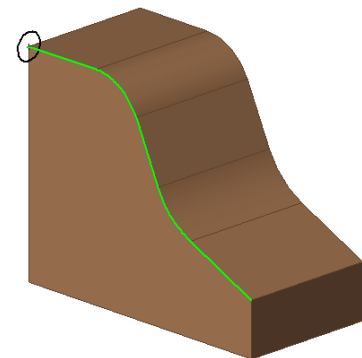
4 Completed.

Complete the model by adding a **R0.013"** fillet to the edges of the cut feature.

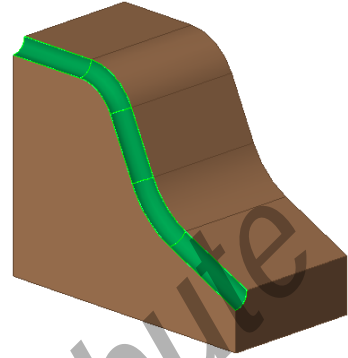


Align with End Faces

You are probably wondering what the option **Align with End Faces** is used for. Consider this simple example. Suppose you wanted to create a cut by sweeping a profile along the edge of a model as illustrated at the right.

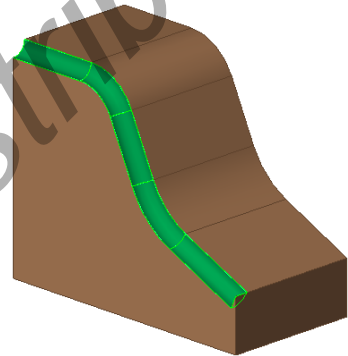


If you use **Align with End Faces**, the cut continues all the way through to the end face of the model. This is similar to the **Through All** end condition used in extruded features. This is usually desirable and is why this option is selected by default – when you are sweeping a cut.



If you *do not* use **Align with End Faces**, the cut terminates when the profile reaches the end of the path, leaving a small lip of uncut material.

The reason we did not use **Align with End Faces** when sweeping the threads is because there were no end faces for the boss to align with. Using it in that case could have forced the system to give an incorrect result. Fortunately, **Align with End Faces** is deselected by default when sweeping a boss.



Sweeping Along Model Edges

There is something else this example shows: model edges are valid entities for a sweep path. They can be selected directly, without copying them into a sketch.

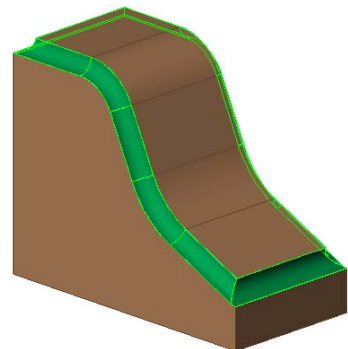
Propagate Along Tangent Edges

When you select a model edge as a sweep path, an additional option becomes available in the **Sweep** dialog. This option is **Tangent propagation** and it serves the same function as the similar option in filleting. If you select a single segment of the edge, this option causes the sweep to continue along the adjacent, tangent edges.

The sweep command only allows you to select a single entity for the path. Therefore, you *cannot* use the right-mouse menu option **Select Tangency**.

What if the Edges Aren't Tangent?

Consider a situation where you want to run a swept feature around a number of edges, not all of which are tangent. The **Sweep Path** selection list only accepts one selection. There is no way to select multiple edges. And since some of the edges are not tangent, they won't propagate.



**Introducing:
Composite Curve**

A **Composite Curve** enables you to combine reference curves, sketch geometry, and model edges into a single curve. This curve can then be used as a guide or path when sweeping or lofting.

Where to Find It

- On the **Insert** menu, click **Curve, Composite**.
- Or, click **Composite Curve**  on the Curves toolbar.

1 Composite Curve dialog.

Click **Composite Curve**  on the Curves toolbar.



**Introducing:
Select Tangency**

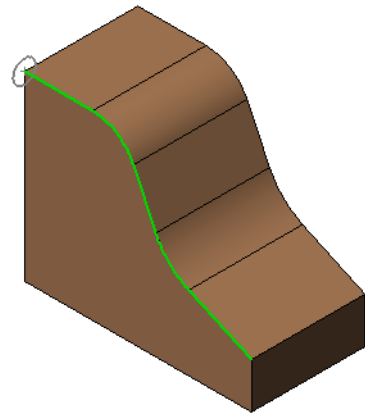
Select Tangency is used to select a tangent-continuous chain of edges.

Where to Find It

- Right-click an edge and select **Select Tangency** from the shortcut menu.

2 Select the edges.


Right-click one of the side edges, and choose **Select Tangency**. All the tangent edges are chosen.

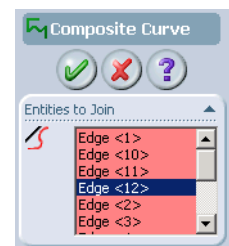


3 Select remaining edges.


Do the same for the other side and add the single edges.

4 Create curve.

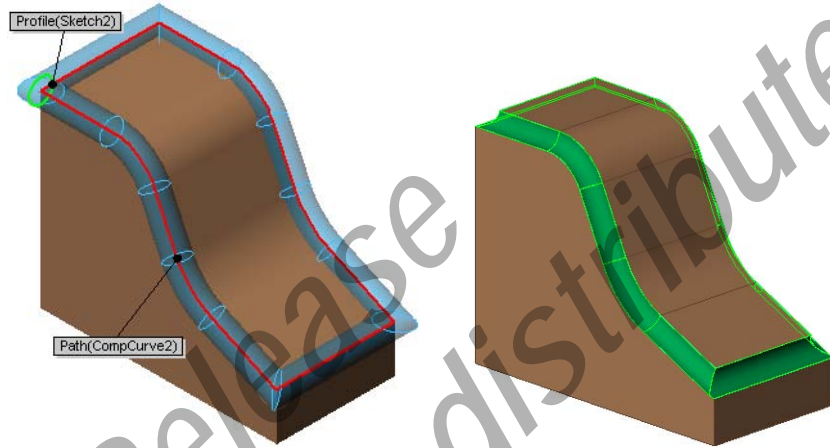
Click **OK** to create the composite curve. The curve is listed in the FeatureManager design tree with its own unique icon –  `CompCurve1`. You can edit the definition of the curve to add or remove edges.



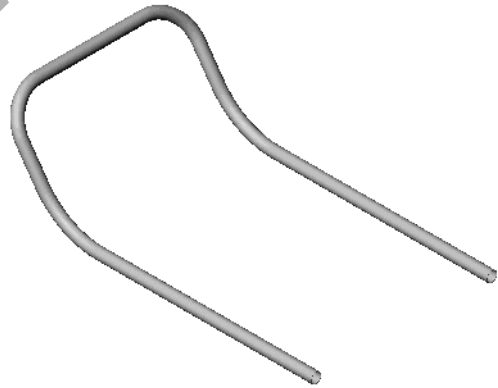
5 Sweep the cut.

Click **Swept Cut**  or click **Insert, Cut, Sweep**. Select the circle as the **Profile**. Select the composite curve for the **Path**.

Click **OK**.

**3D Sketches**

You can create 3D sketches by sketching on reference planes or planar faces in a model or assembly. In this example, the 3D sketch will be used as the path in a sweep.

**Plane At Angle**

An angled plane will be created using the **Plane At Angle** option. Note that planar model faces or user-defined coordinate systems can also be used, as this plane is, to orient the geometry of the 3D sketch.

1 New part.

Open a new part using the Part_IN template.



**Introducing:
Insert Axis**

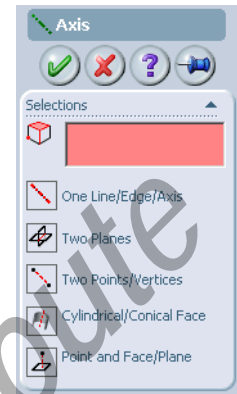
Insert, Reference Geometry, Axis creates an axis that appears in the FeatureManager. The axis can be renamed and dragged to any length. They are useful when any vector information is required.

Where to Find It

- Click **Axis**  on the Reference Geometry toolbar.
- Or, click **Insert, Reference Geometry, Axis**.

2 Axis tool.

Click the **Axis** tool  and click the **Two Planes**  option.

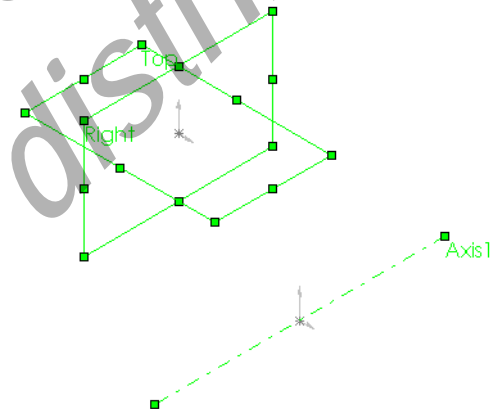


3 Plane geometry.


Select the Right Plane and Top Plane reference planes.

Click **OK** to add the axis.

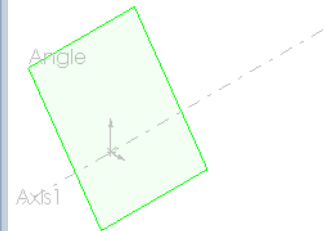
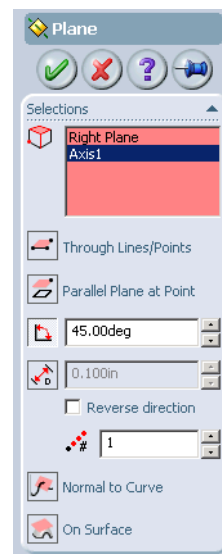
The axis appears as **Axis1** and will be used to define an angled plane.




4 Angled plane.

Click the **Plane** tool . Select the Right Plane reference plane and the axis. Set the **Angle** to **35°** and click **Reverse Direction** if necessary. Click **OK**.


Name the new reference plane **Angle**.

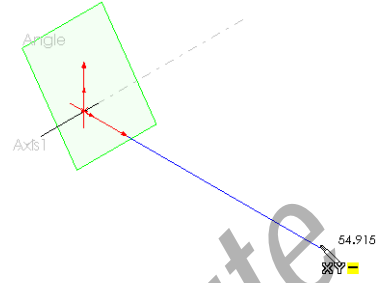


5 Open a new 3D Sketch.

Click the **3D Sketch** tool  to start a new sketch. Change to the Isometric view. Click **View, Axes** to see the axis.

6 Sketching a line.

Click the **Line** tool and start sketching at the **Origin** of the sketch. Drag the line using the **Horizontal** marker  to keep it on the X axis of the default **XY** plane. Make the line about **50"** long.

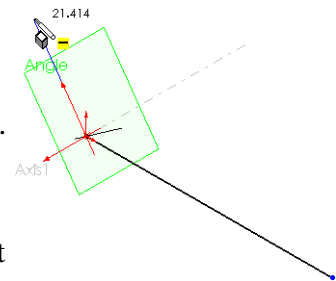
**7 Switch sketch planes.**

Deselect the Line tool. Press the **Ctrl** key and click the plane named **Angle** in the FeatureManager design tree.

When you start sketching the next line, the XY plane will be aligned with the reference plane named **Angle**.

8 Continue sketching.

Sketch the next line from the endpoint at the **Origin** and move along the axis of the selected plane. Make the line about **20"** long.

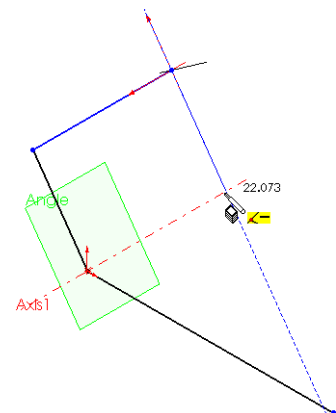
**Note**

Depending on how the plane named **Angle** was defined, you will be sketching along either the X or the Y axis. In the illustration at the right, the line is sketched along the horizontal X axis.

9 Continue to sketch in the plane.

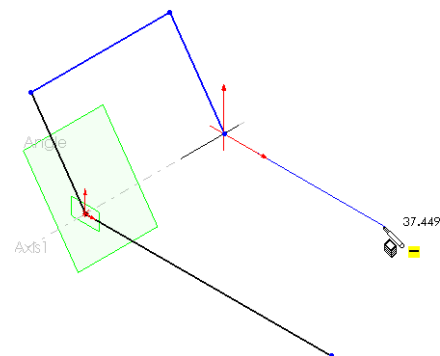
Continue to sketch the lines on the reference plane named **Angle**. The horizontal line should be about **25"** long.

Stop the last line on the axis. This adds a **Coincident** relation between the end of the line and the axis.

**10 Switch sketch planes.**


Deselect the Line tool. Hold down **Ctrl** and select the **Top** plane.

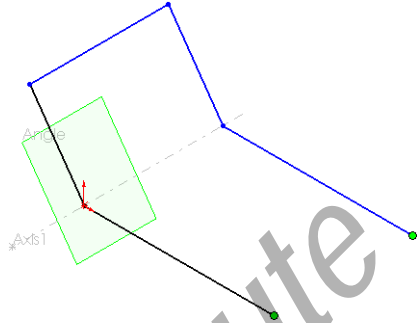
Click the Line tool again and sketch along the X axis of **Top**, stopping near the end of the first line.

**Tip**

To switch between standard planes (**Top**, **Front** and **Right**) you can also press the **Tab** key.

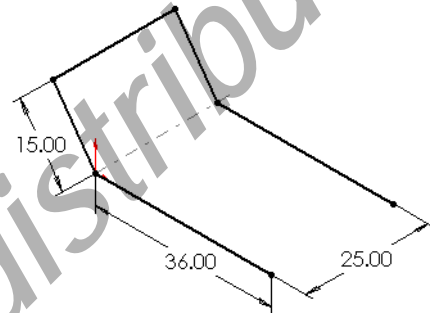
11 Along Z Relation.

Add a relation between the endpoints of the first and last lines. Use the **Along Z**  relation because they should both be aligned along the Z axis of the default plane, Front.



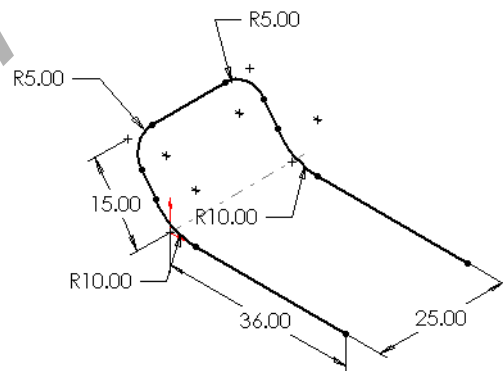
12 Dimensions.

Add dimensions as shown. Dimension the true length of three lines. Because of the relations, the sketch will be fully defined.



13 Fillets.

Add fillets to the set of lines using the sketch fillet tool. Add two pairs of fillets radius 5" and 10" as shown.



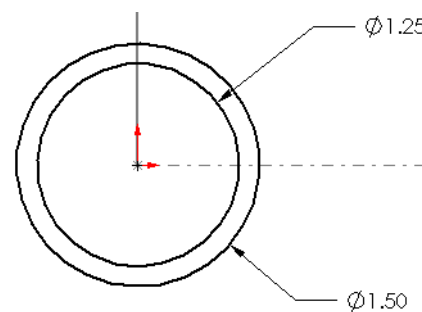
14 Exit the 3D sketch.

Multiple Contours in a Sweep

Sweep can use a multiple contour sketch as the **Profile**. The rules are similar to those used for an extrusion.

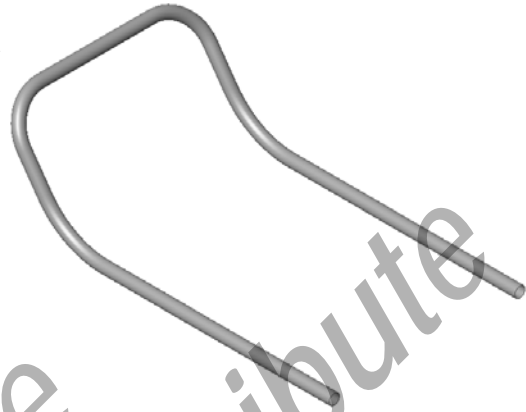
15 Profile.

Create a plane at the end of the sketch line and create two circles to represent the ID and OD of the tube.



16 Sweep.

Sweep a boss feature using the circles as the profile and the 3D sketch as the path.

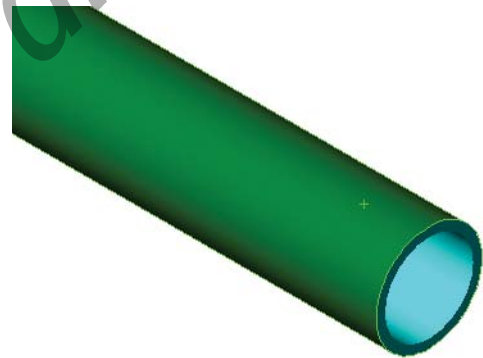


Using the Hole Wizard on Non-planar Faces

The **Hole Wizard** is used to create cuts in the form of standard holes. It can also be applied to non-planar faces with a **3D Sketch**.

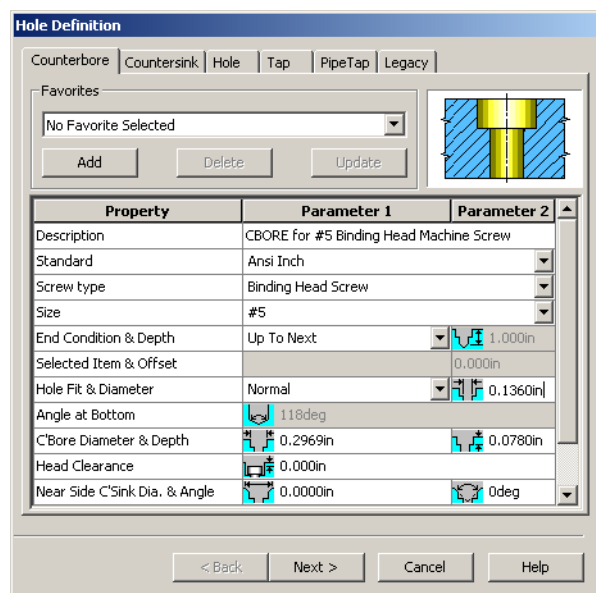
17 Zoom in.

Zoom in on the left open end of the model. Select the cylindrical face.

**18 Hole Wizard.**

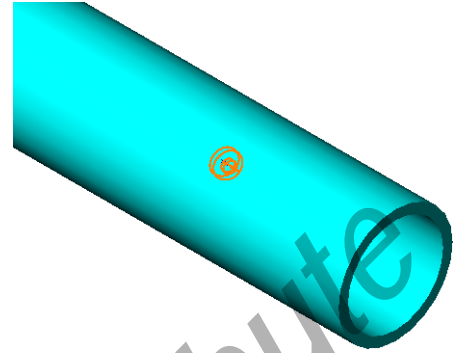
Click the **Hole Wizard** and set the properties for the counterbore as shown.

Click **Next**.



19 Hole center.

A point is located on the face of the model, **Coincident** to it. The sketch is a 3D sketch.



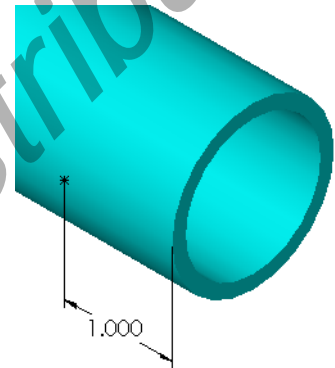
20 Location.

Locate the point **Coincident** to the Top reference plane and **1"** from the end face.

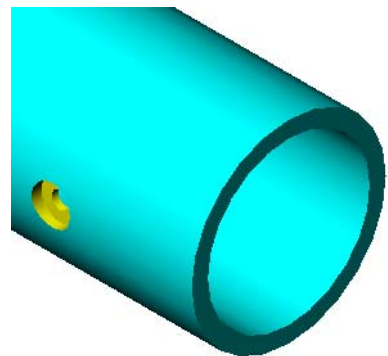
Note

In a 3D sketch, you can dimension directly to faces and edges of the model. To create the dimension, dimension between the point and the flat, end face of the model.

Click **Finish**.



21 Save and close the file.



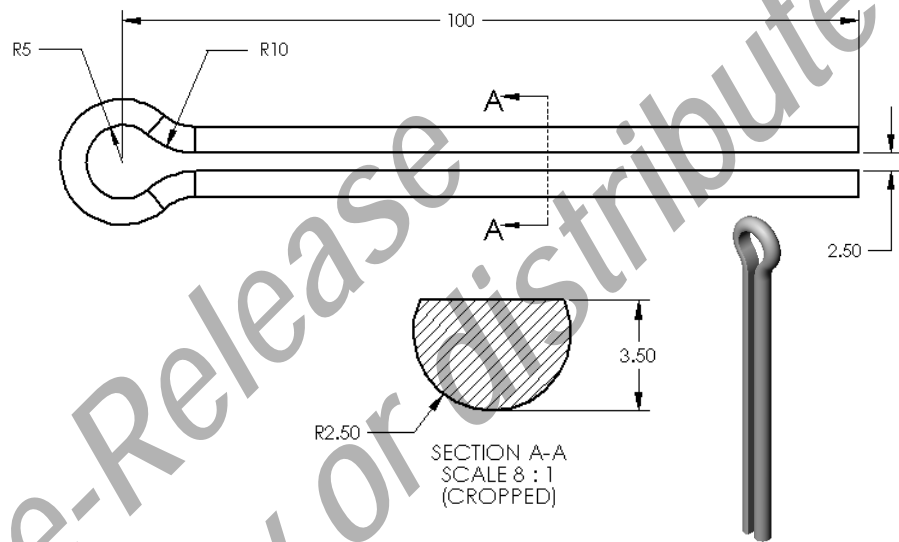
**Exercise 9:
Sweeps without
Guides**

Create these three parts using swept features. These require only a path and a section, no guide curves.

Units: **millimeters**

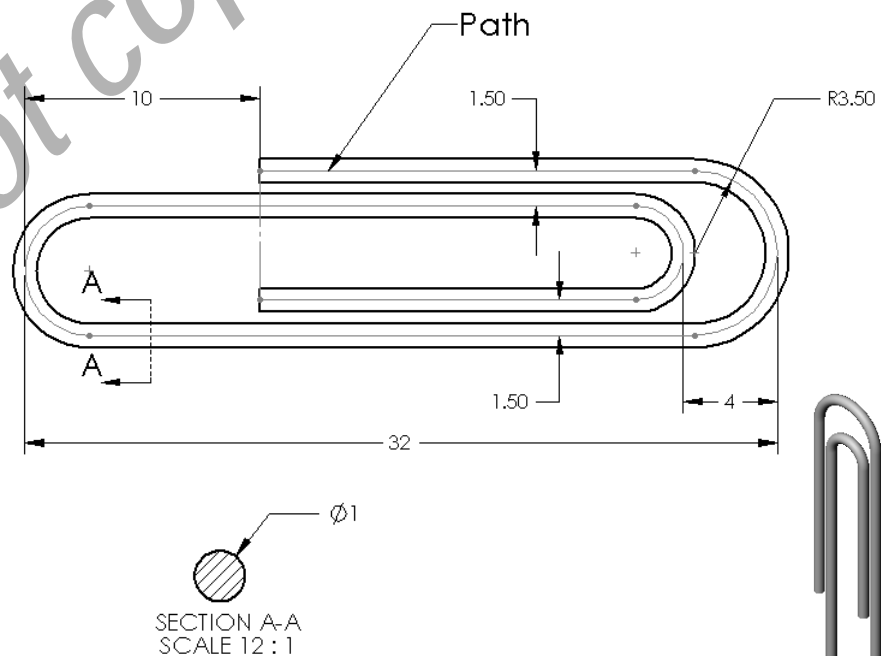
Cotter Pin

The Cotter Pin uses a path that describes the inner edge of the sweep.



Paper Clip

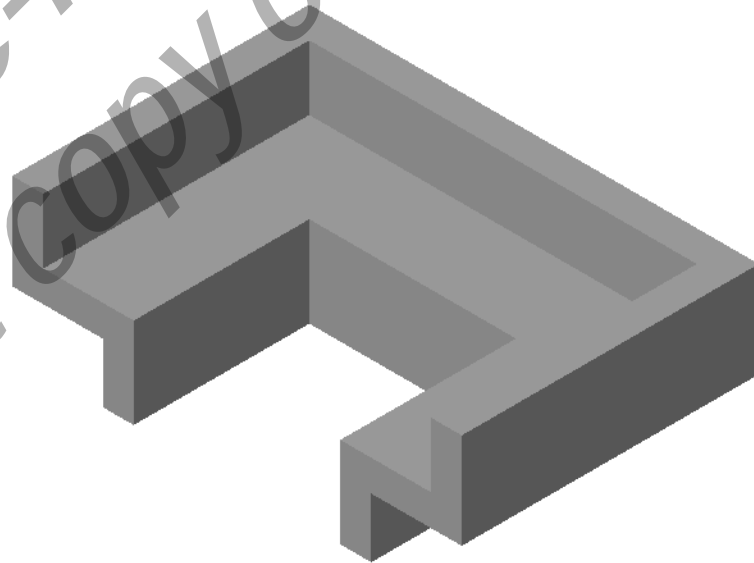
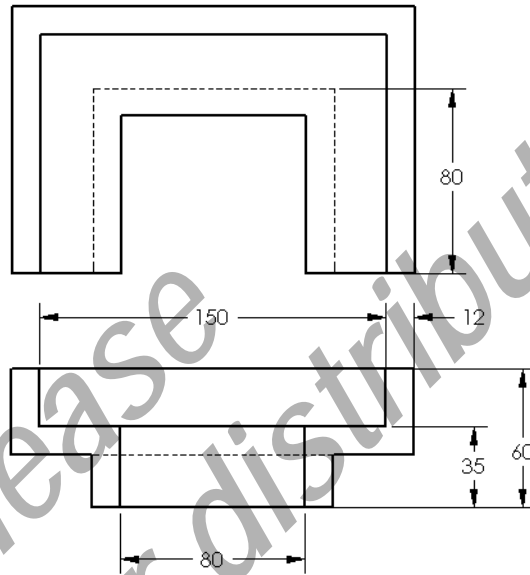
The Paper Clip is defined by a path that describes the centerline of the sweep.



Thanks to Paul Gimbel, TriMech Solutions, LLC for submitting these examples.

Mitered Sweep

The Mitered Sweep is defined by a path that describes the outer edge of the sweep.



Exercise 10: Attachment

Create this part using the step by step instructions provided. Use relations or link values where applicable to maintain the design intent.

This lab uses the following skills:

- Sketching
- Planes
- Extruding
- Sweeping
- Multi-thickness Shelling
- Variable-radius Fillet



Design Intent

The design intent for this part is as follows:.

1. Part is symmetrical.
2. Wall thickness is uniform.

Procedure

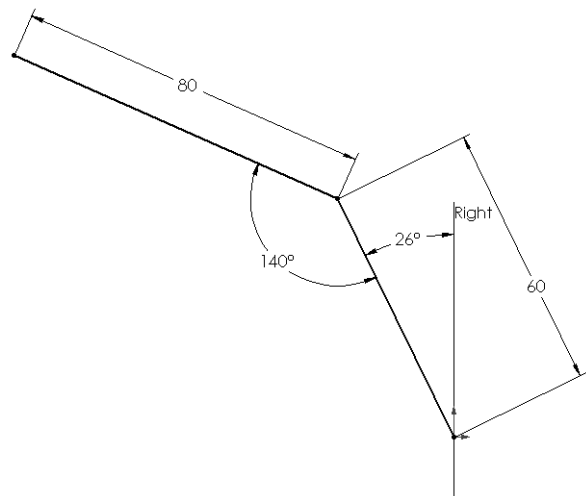
Open a new part using the Part_MM template and name it Attachment.

- 1 **Layout sketch.**
Sketch a layout of the part on the Front reference plane. The sketch sets the locations and dimensions for the two main features.

Note

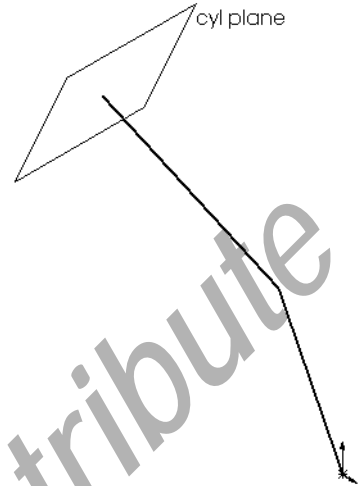
The 26° angle is dimensioned to the Right reference plane.

Name the sketch Layout.



2 Plane normal to curve.

Create a plane that is normal to the endpoint of the upper line of the Layout sketch.

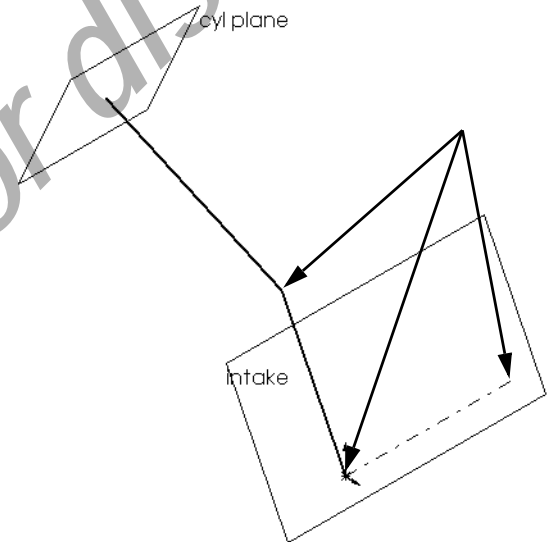
**3 Plane through 3 points.**

Create another sketch on the Top reference plane and add a short vertical line from the Origin.

Exit the sketch.

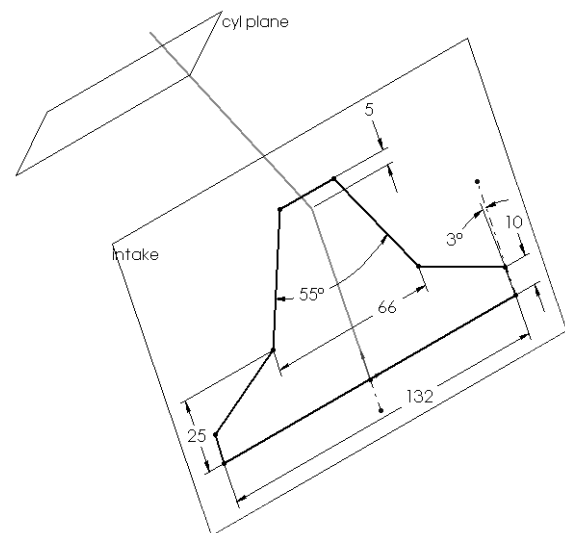
Using the **Through Lines/Points** method, select the endpoints of this line and the sharp corner of the Layout sketch, to define another plane.

Name this plane intake.

**4 Sketch the profile.**

Sketch on the intake plane to create the profile of the nozzle.

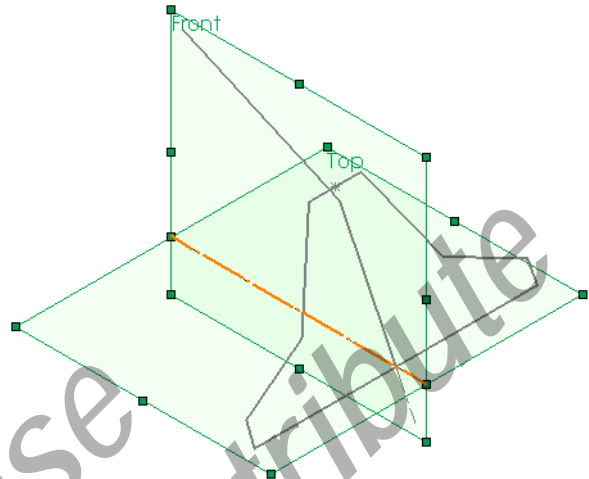
Use symmetry to create the sketch and tie it to the Layout sketch.



5 Axis.

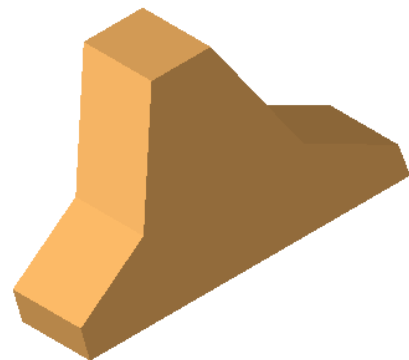
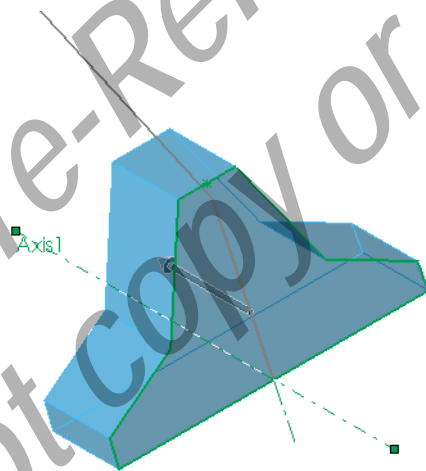
Create an axis defined by the intersection of the Front and Top reference planes.

This will be the vector for the extrude direction.



6 Extrude.

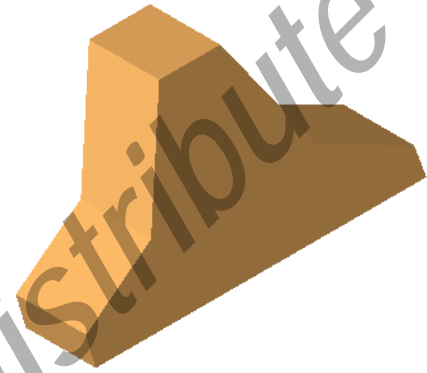
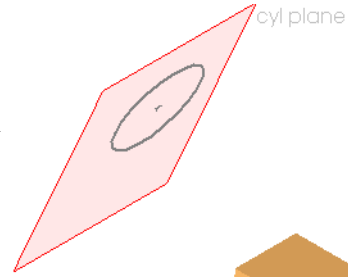
Extrude the profile sketch using the **Blind** end condition. Select the axis for the **Direction of Extrusion**. Set the **Depth** to **28mm**.



7 Cylinder.

On the cyl plane, sketch a **34mm** diameter circle, centered on the end of the upper line in the Layout sketch.

This circle will be used to extrude a cylinder.



Pre-Release
Do not copy or distribute

**End Condition:
Up to Surface**

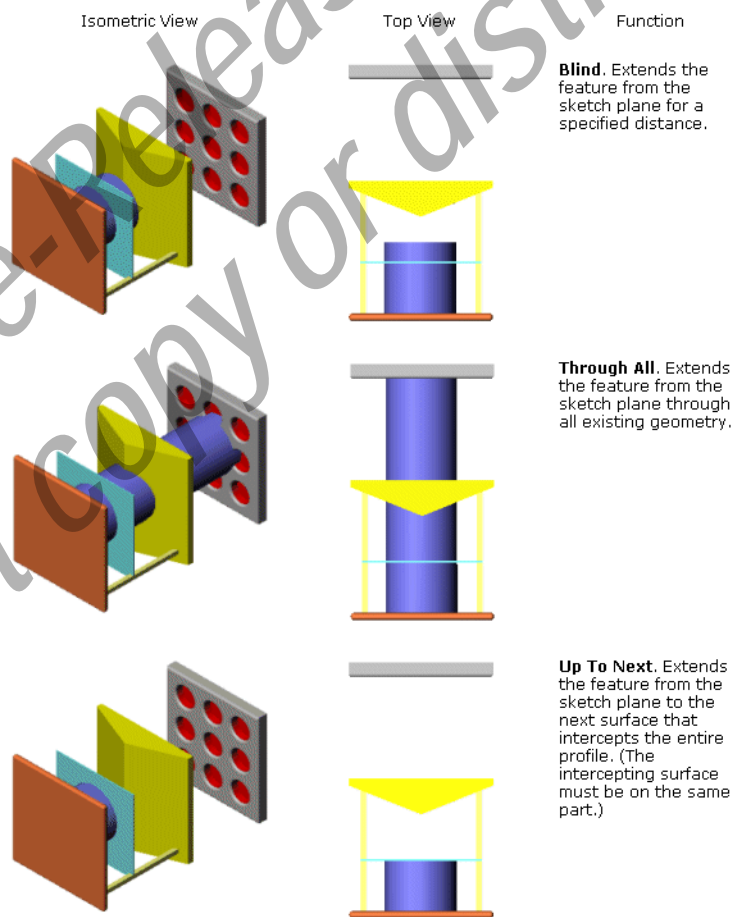
Ideally, the end condition of the cylinder should be such that it stops exactly flush with the front face of the first feature. The end condition most people think of in this type of situation is **Up to Next**. However, that will not work in this case.

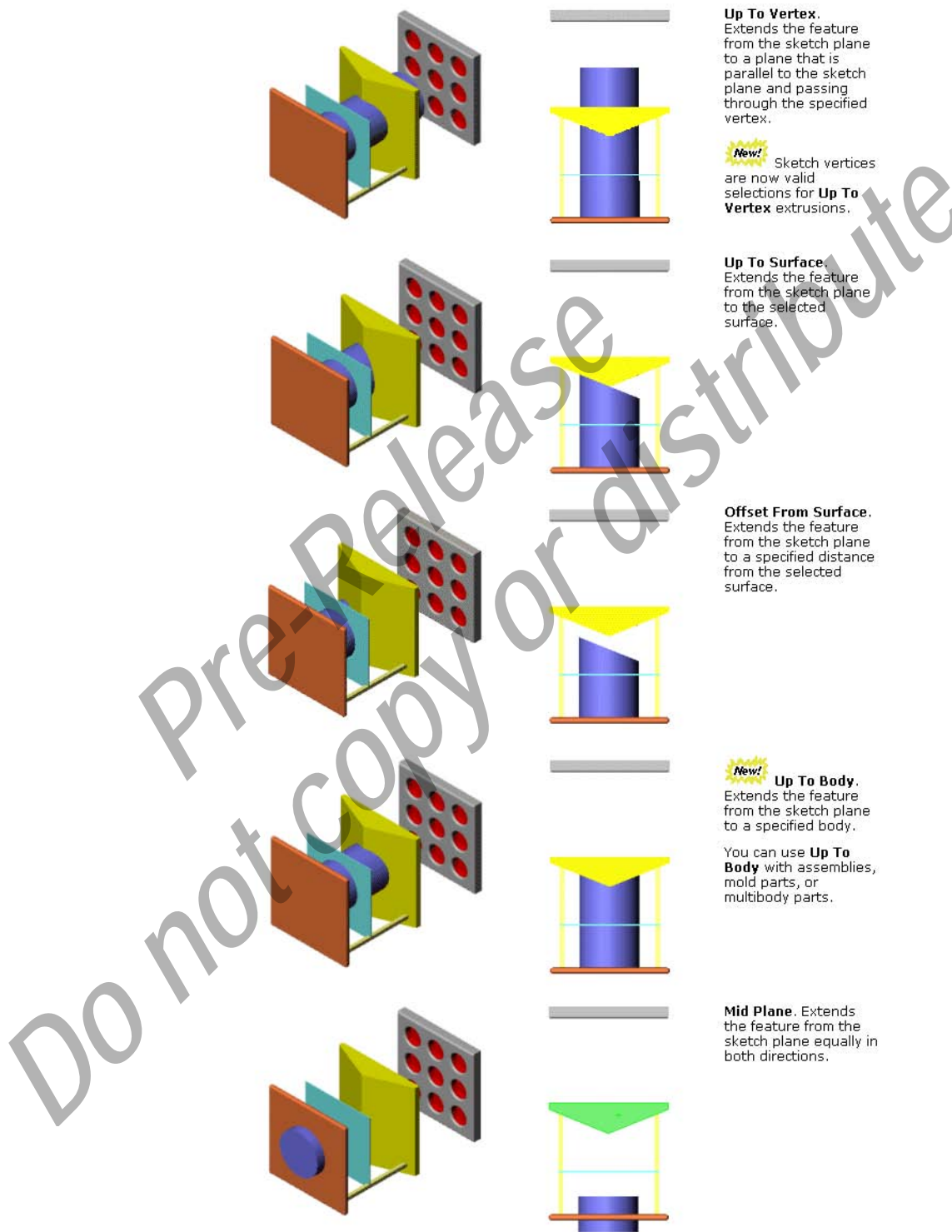
On-line Help

On-line help is a vital resource for learning more about the SolidWorks software. Refer to it whenever you need to find the answer to a particular question. In this case, use the on-line help to look up the text string “end condition extrude”. This will give you a concise explanation of the different end conditions for extruded features.

End Condition Extrude

When you extrude a profile, you need to select a **Type** from the **Extrude Feature** dialog box.





Up To Vertex.
 Extends the feature from the sketch plane to a plane that is parallel to the sketch plane and passing through the specified vertex.

New! Sketch vertices are now valid selections for **Up To Vertex** extrusions.

Up To Surface.
 Extends the feature from the sketch plane to the selected surface.

Offset From Surface.
 Extends the feature from the sketch plane to a specified distance from the selected surface.

New! **Up To Body.**
 Extends the feature from the sketch plane to a specified body.

You can use **Up To Body** with assemblies, mold parts, or multibody parts.

Mid Plane. Extends the feature from the sketch plane equally in both directions.

Up to Surface

From the on-line help it is easy to see that the end condition **Up to Surface** meets our needs. **Up to Surface** extends the extrusion from the sketch plane to the selected surface. The surface can be a face, a reference plane, or a stand-alone surface.

8 Up to Surface.

Click **Insert, Boss, Extrude**. Verify from the preview that the boss is extruding in the correct direction. If it is not, click **Reverse Direction**.

From the **End Condition:** list, select **Up to Surface**.

Select the front face of the swept first feature.

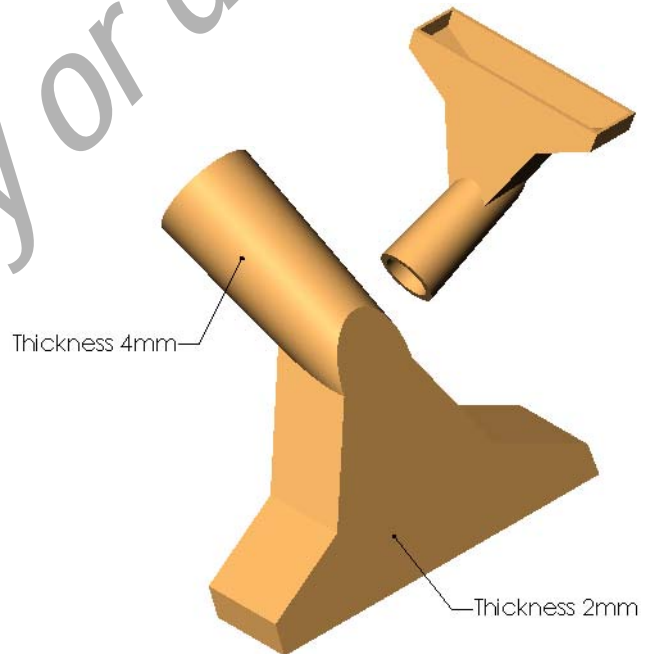
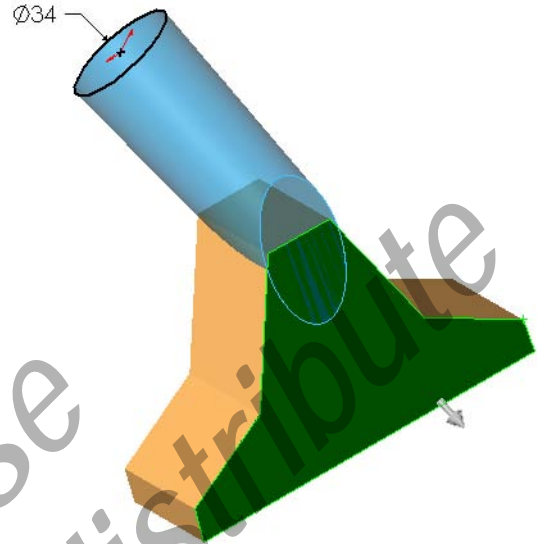
Select **Draft**, and set the angle to **2°**, and check **Merge result**.

Click **OK**.

9 Multiple-thickness Shell.

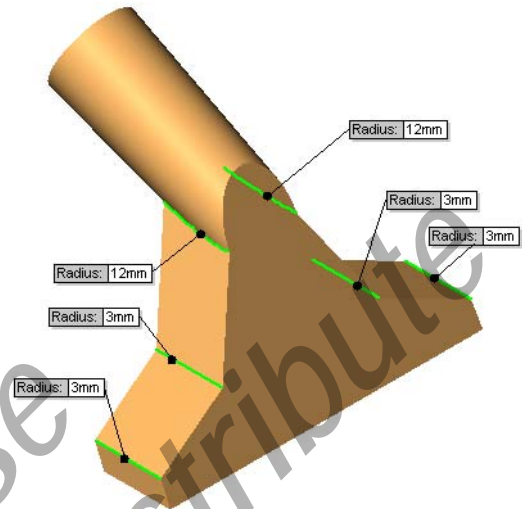
Shell the solid **2mm** to the inside, selecting the end faces for removal.

Select the cylindrical face and set it to **4mm**.



10 Fillets and rounds.


Add fillets and rounds to the outside of the solid body as shown.

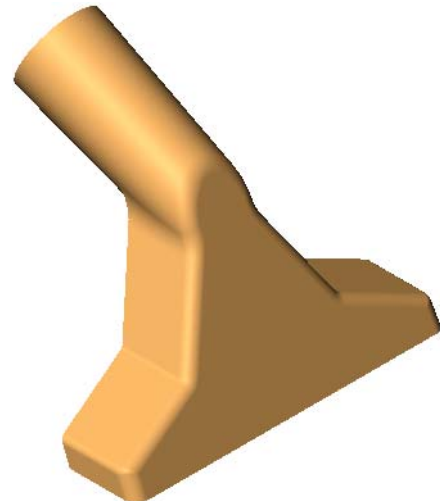
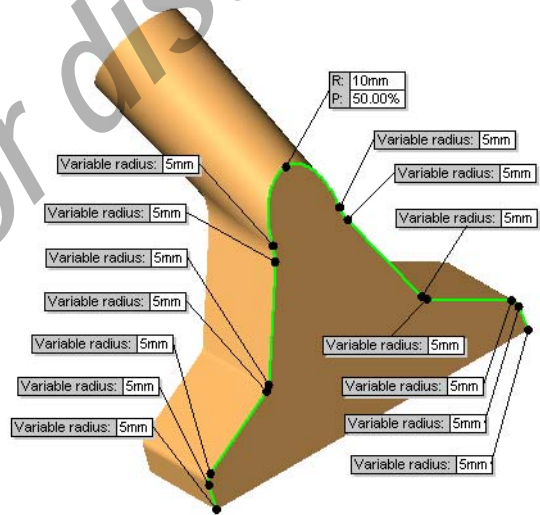
**11 Variable radius fillet.**

Add a variable radius fillet to the set of tangent edges shown. The fillet varies from **5mm** to **10mm** at the middle, and back to **5mm**.

Tip

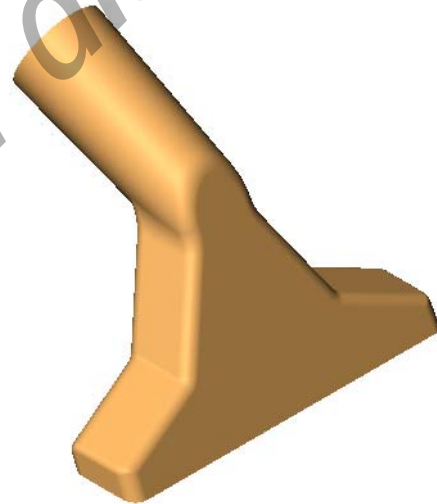
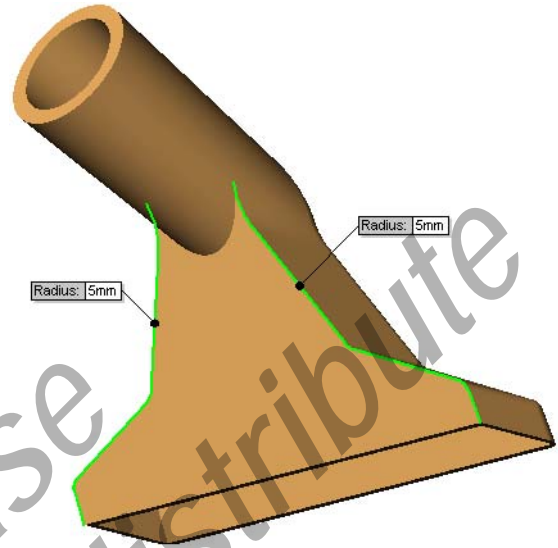
This technique simplifies assigning the values to the vertices:

- 1 Click **Fillet** .
- 2 Click **Variable radius**.
- 3 Right-click an edge, and pick **Select Tangency**.
- 4 Set the **Radius** to **5mm**, and click **Set All**.
- 5 Set the number of control points to **1**.
- 6 Click in the **Items To Fillet** list.
- 7 Use the arrow keys on the keyboard to cycle through the list of selected edges. As you do so, the control point will move from one edge to another.
- 8 When the control point appears on the correct edge, click it in the graphics area. Then, use the callout to assign the **10mm** radius.
- 9 Click **OK**.



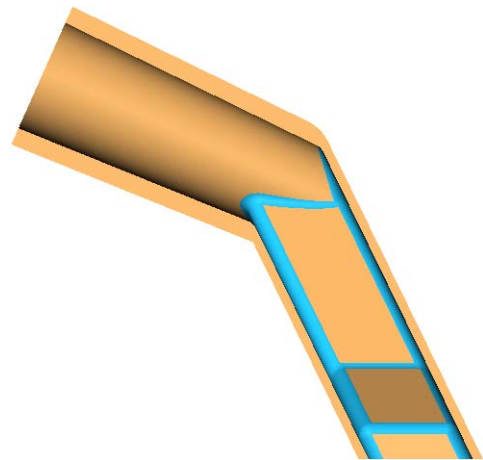
10 Fillet.

Add **5mm** radius fillets to the edges shown.



11 Inner fillets and rounds.

Add fillets and rounds of **3mm** on the inner edges of the part as shown in the section view at the right.



12 Save and close the part.

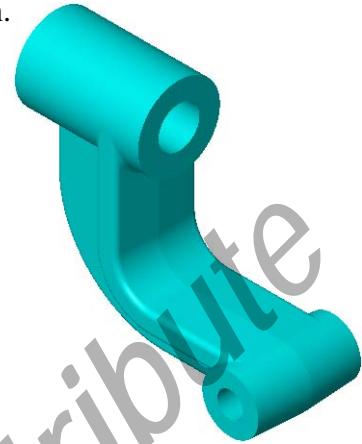
Exercise 11: Hanger Bracket

Create this by following the steps as shown.

This lab uses the following skills:

- Multibody solids
- Sweep using guide curves
- Merging bodies

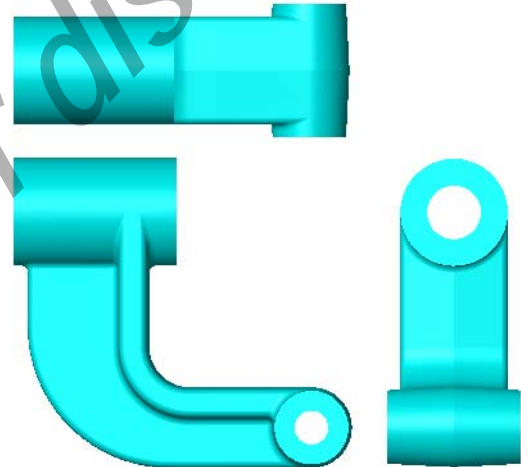
Units: **inches**



Design Intent

The design intent for this part is as follows:

1. All fillets and rounds are **0.125"**.
2. Part is symmetrical with respect to the parting line.
3. Draft is **3°**.

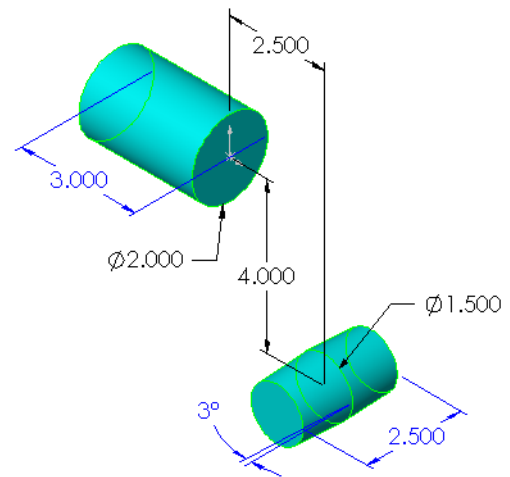


Procedure

Open a new part using the Part_IN template and name it Hanger Bracket.

1 Create sweep ends.

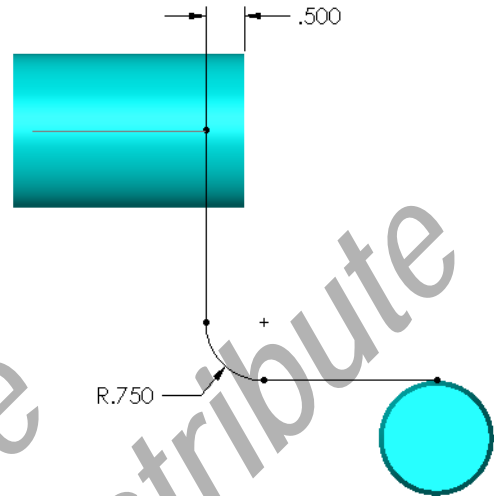
Create two extruded solid bodies to represent the ends of the sweep.



2 Create sweep path.

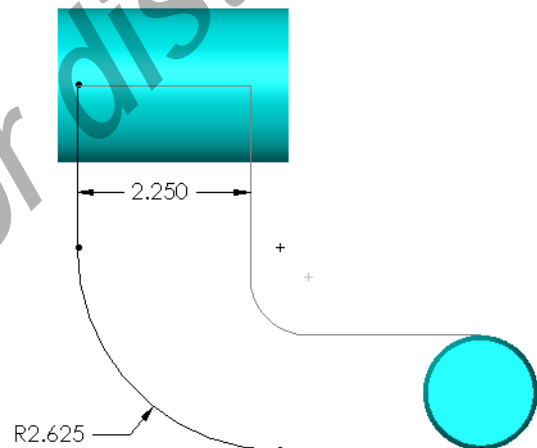
The path and the guide curve must each be in separate sketches.

Create the path sketch using the existing geometry.




3 Create guide curve.

Create the guide curve sketch using the existing geometry including the path sketch.

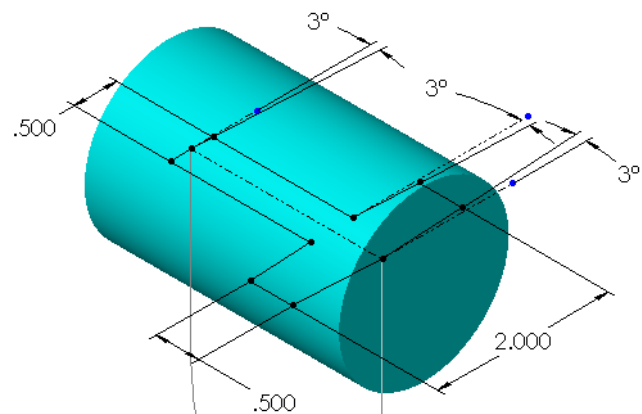


Tip

If you sketched *all* the geometry in one sketch, it can still be used. Change the two lines and the arc that form the guide curve to construction geometry. Open a new sketch for the guide curve. Use **Convert Entities**  to copy the guide geometry into the new sketch.

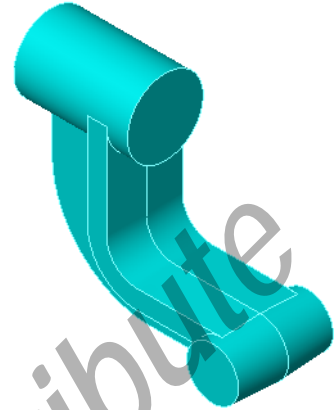
4 Create sweep section.

Create the sweep section as a sketch using the dimensions shown at the right.

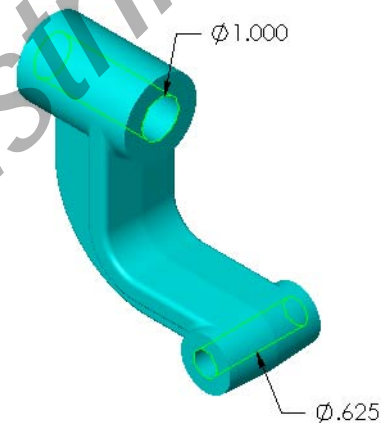


5 Insert sweep.

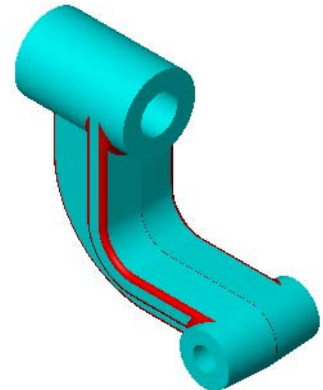
Using the sketches, sweep the feature. Use the **Merge result** option to combine all the solid bodies.

**6 Create through holes.**

Add two through holes cuts to the model.

**7 Insert fillets and rounds.**

Add **0.125"** fillets and rounds, shown here in **red**, to complete the model.

**Tip**

Filleting by feature works best.

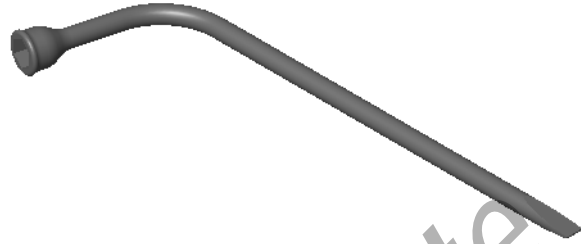
8 Save and close the part.

Exercise 12: Tire Iron

Create this by following the steps as shown.

This lab uses the following features:

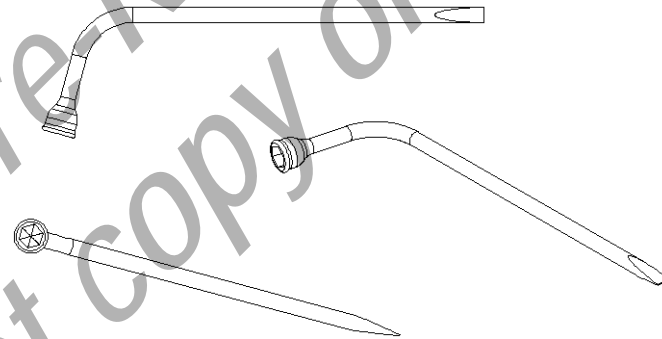
- Sweep feature
- Revolve feature
- Sketch fillets
- Polygon tool
- Dome feature
- Reference planes



Design Intent

The design intent for this part is as follows:

1. Regular end is symmetrical using angled cuts.
2. Wrench end is created using a hexagon cut.
3. Section is constant diameter.

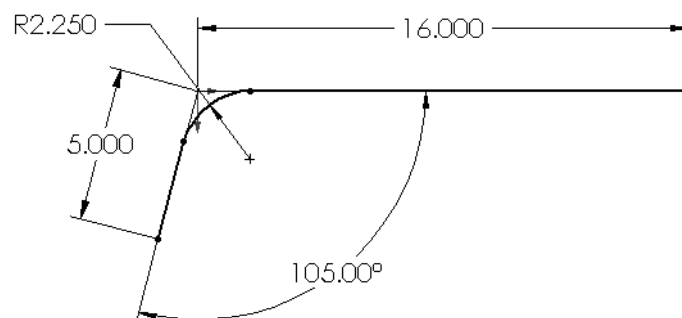


Procedure

Open a new part using the Part_IN template and name it Tire Iron.

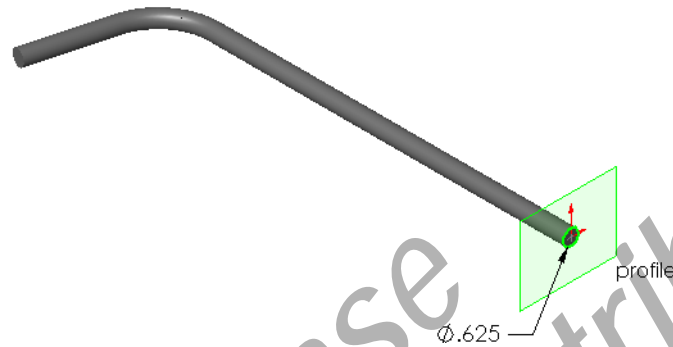
1 Create the sweep path.

Create the sketched lines then add the fillet.

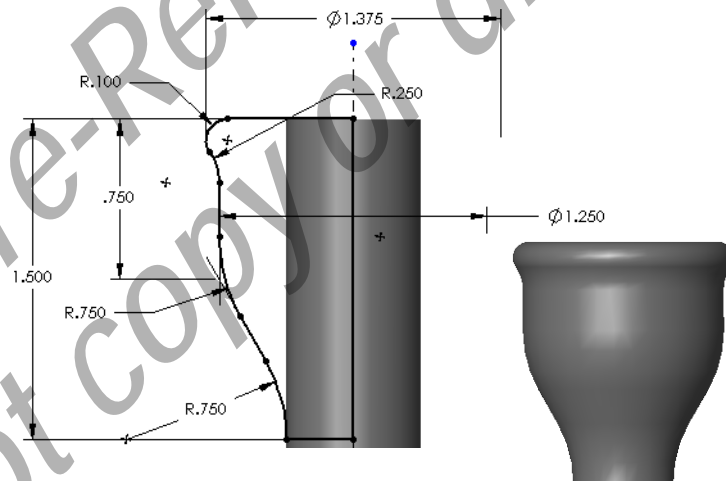



2 Insert sweep.

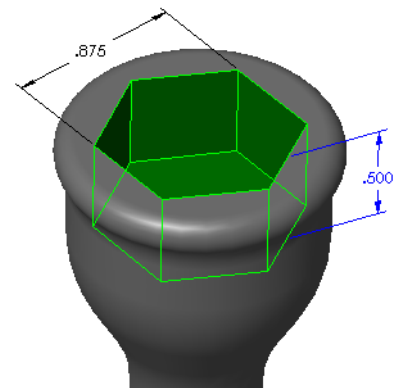
Create a new reference plane and use it to sketch the sweep section sketch. Sweep the profile along the path.

**3 Revolved feature.**

Create a revolved feature on the angled end of the sweep feature. This boss will hold the hexagon cut.

**4 Hexagonal cut.**

Create a hexagonal cut using the Polygon tool .



Dome Feature

The **Dome** feature lets you deform the face of a model creating either a convex (default) or concave shape.

Introducing: Dome

To create a dome, select the face or faces you wish to deform. Specify a distance and optionally, a direction. By default the dome is created normal to the selected faces. You can select faces whose centroid lies outside the face. This allows you to apply domes to irregularly shaped faces.

Where to Find It

- Click **Dome**  on the Features toolbar.
- Or, click **Insert, Features, Dome**.

5 Round the bottom of the cut using the Dome feature.

Click **Dome**  on the Features toolbar.

Clear the **Continuous dome** check box.

Select the hexagonal face at the bottom of the cut.

Specify a **Distance** of **0.25"**.

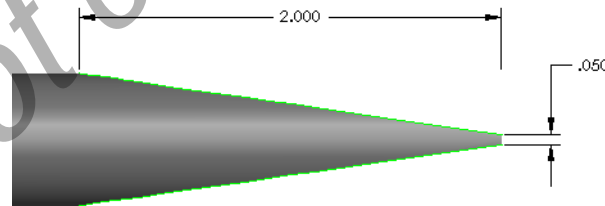
Click **Reverse Direction**  to make the dome concave.

Click **OK**.



6 Through all cut.

Create the flat end of the part using a sketch and a through all cut.



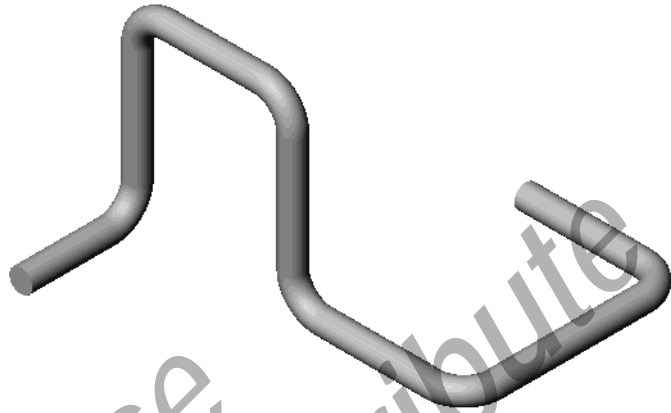
7 Save and close the part.

Exercise 13: 3D Sketching

Create this by following the steps as shown.

This lab uses the following skills:

- 3D sketching
- Lines and fillets
- Sweep



Procedure

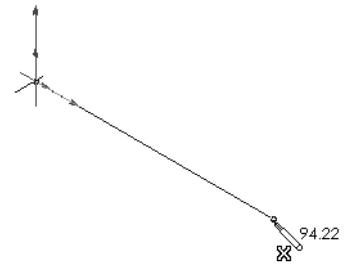
Open a new part using the Part_MM template and name it 3D Sketching.

1 New 3D sketch.

Create a new **3D Sketch** and change the view orientation to Isometric.

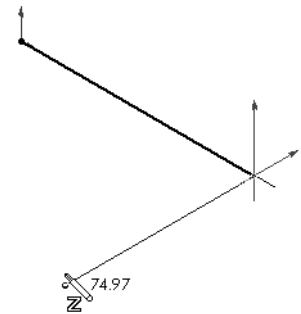
2 Sketch lines.

Click the **Line** tool and start the first line at the Origin. Sketch the line in the **X** direction.



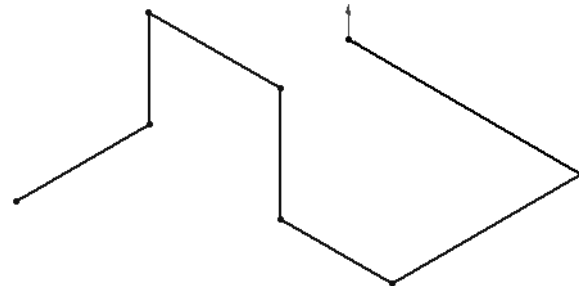
3 Switch planes.

Begin dragging the second line to see the axes. Press the **Tab** key to switch from the default Front plane to the others. Switch to the Right plane orientation and sketch along the **Z** axis.



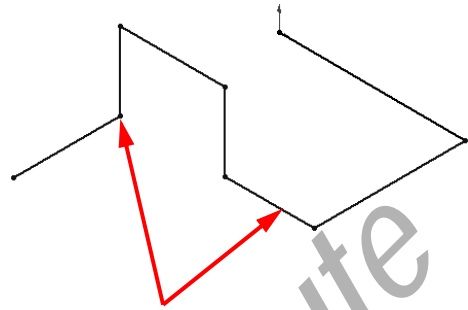
4 Continue lines.

Continue sketching lines and switching planes so that you are always sketching on **X**, **Y** or **Z** in the appropriate direction.



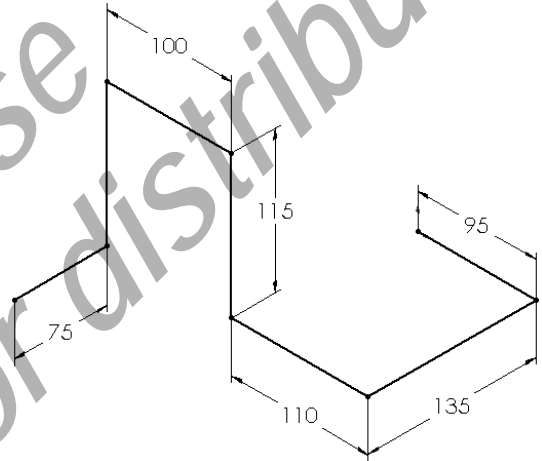
5 Relation.

Add a **Coincident** relation between the endpoint and line shown at the right.



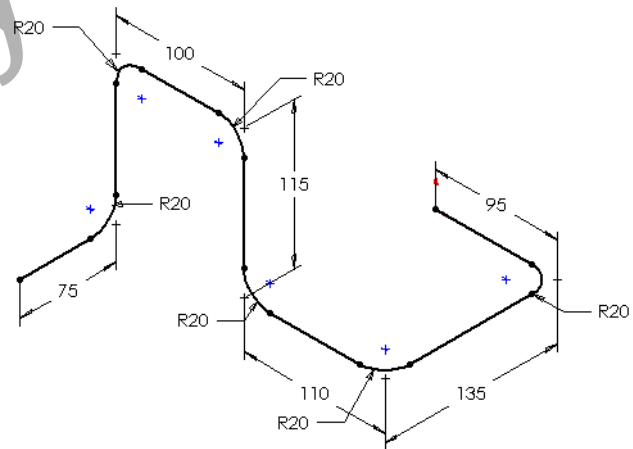
6 Dimensions.

Dimension the true length of the lines as shown to fully define the sketch. Select the endpoints of the lines or the lines themselves.



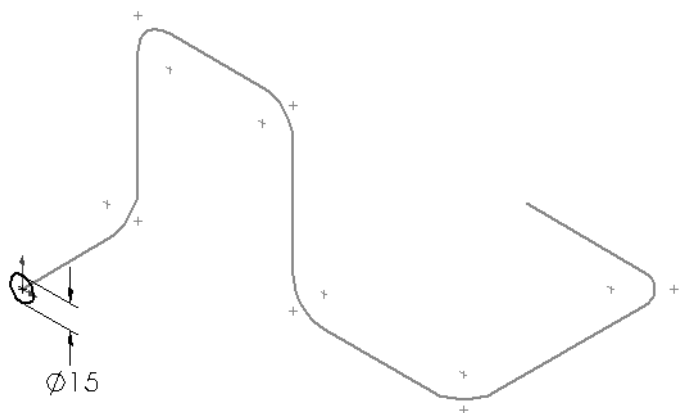
7 Fillets.

Add **20mm** fillets at the vertex points.

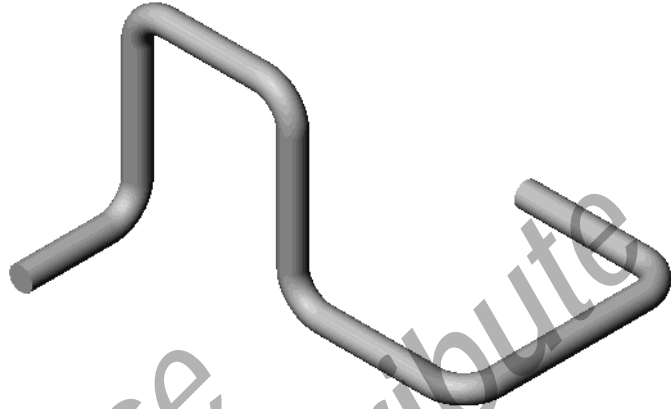


8 Profile sketch.

Create a new plane using **Normal to Curve** at the endpoint of the 3D sketch. Sketch a circle of diameter **15mm**.



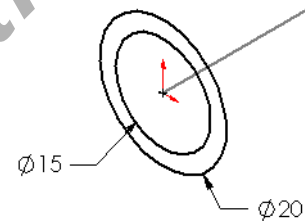
- 9 Sweep.**
Sweep the circle section along the 3D path.



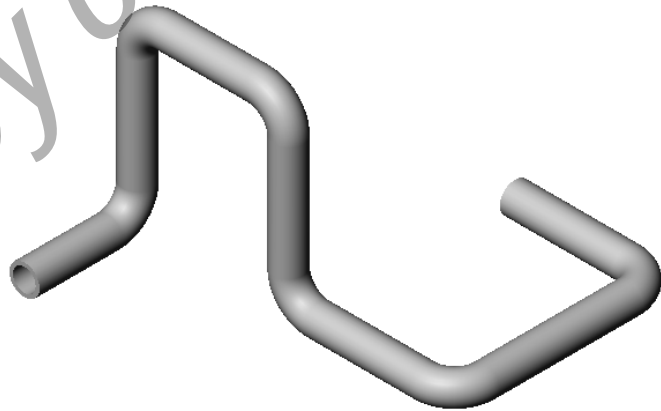
Optional

Edit the profile sketch of the sweep to create a thin wall.

- 10 Edit sketch.**
Edit the profile sketch and add a concentric circle, diameter **20mm**. Exit the sketch.



- 11 Modified sweep.**
The concentric circles form a thin wall in the sweep.



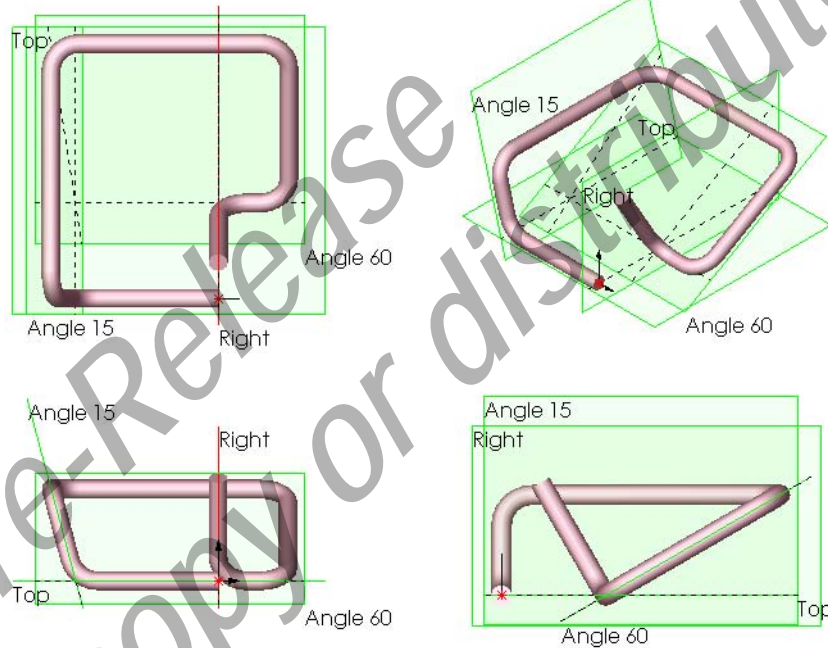
- 12 Save and close the part.**

Exercise 14: 3D Sketching with Planes

Create this part by following the steps as shown.

This lab uses the following skills:

- 3D sketching
- Lines and fillets
- Sweep



Procedure

Open an existing part named 3DSketchAngle.

1 New 3D sketch.

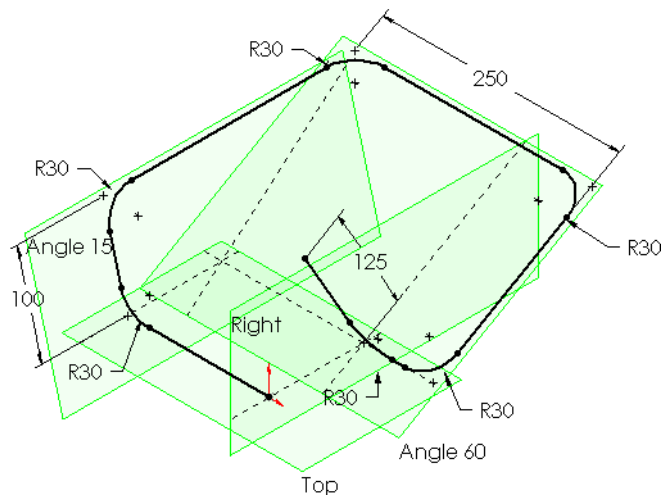
Create a new **3D Sketch** and change the view orientation to Isometric.

2 Sketch lines.

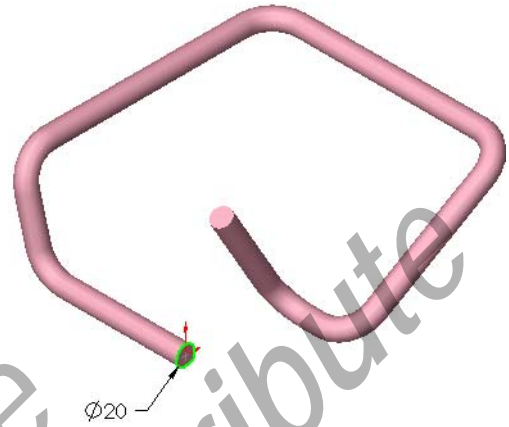
Click the **Line** tool and start the first line at the Origin.

Use the planes Angle 15, Angle 60 and Top to orient and constrain the lines of the sketch.

Add fillets. Use **Link Values** to make the fillet radii equal.



- 3 Sweep.**
Create a circle profile and use the sweep command to complete the exercise.



- 4 Save and close the part.**

Pre-Release
Do not copy or distribute

Exercise 15: Hole Wizard and 3D Sketches

Create this by following the steps as shown.

This lab uses the following skills:

- Hole Wizard
- Reference planes
- 3D sketching
- Patterning



Procedure

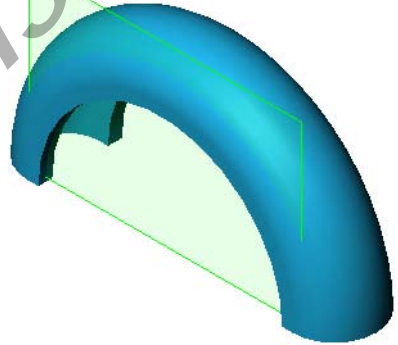
Open an existing part named HoleWizard.

1 Reference planes.

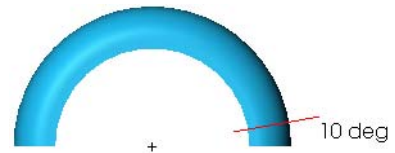
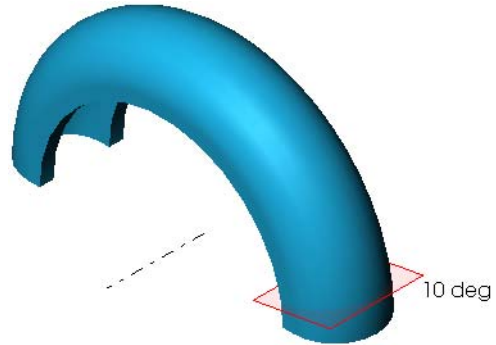
Create two new reference planes as follows:

- **Offset Distance** – offset **25mm** from the Front plane.


25 off



- **At Angle** – angle of **10°** using a temporary axis and a model face.

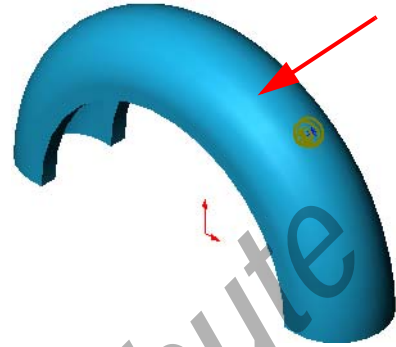


2 Hole size.

Select the curved face of the model and click **Hole Wizard** .

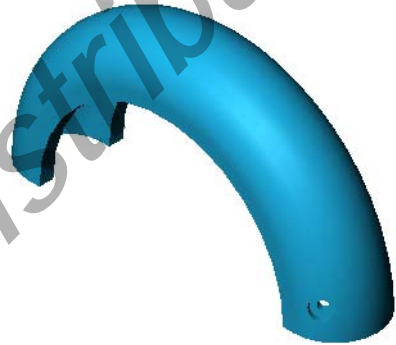
Choose the settings for the description “CBORE for M6 Hex Head Bolt”.

Use the **Up To Next** end condition.

**3 Placement.**

Position the hole's locating point in the 3D sketch by making it coincident to both the 25 off and the 10 deg planes.

The axis of the hole is now perpendicular to the face at that point.

**4 Patterns.**

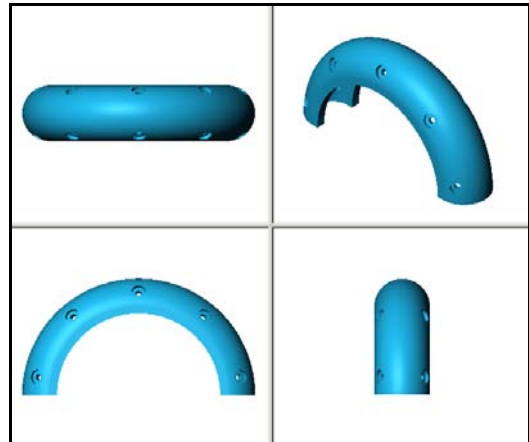
The objective is to have 5 holes equally spaced through a total angle of 160° , on both the front and back of the part, for a total of 10 holes. Do this by patterning the hole.

Question

Should you mirror the hole, and then make a circular pattern of the mirror feature? Or should you make a circular pattern, and then mirror the circular pattern?

Optional

Write an equation that determines the proper angle for the circular pattern based on the plane angle. At right, the plane angle is 20° .

**5 Save and close the part.**

Lesson 3

Lofts

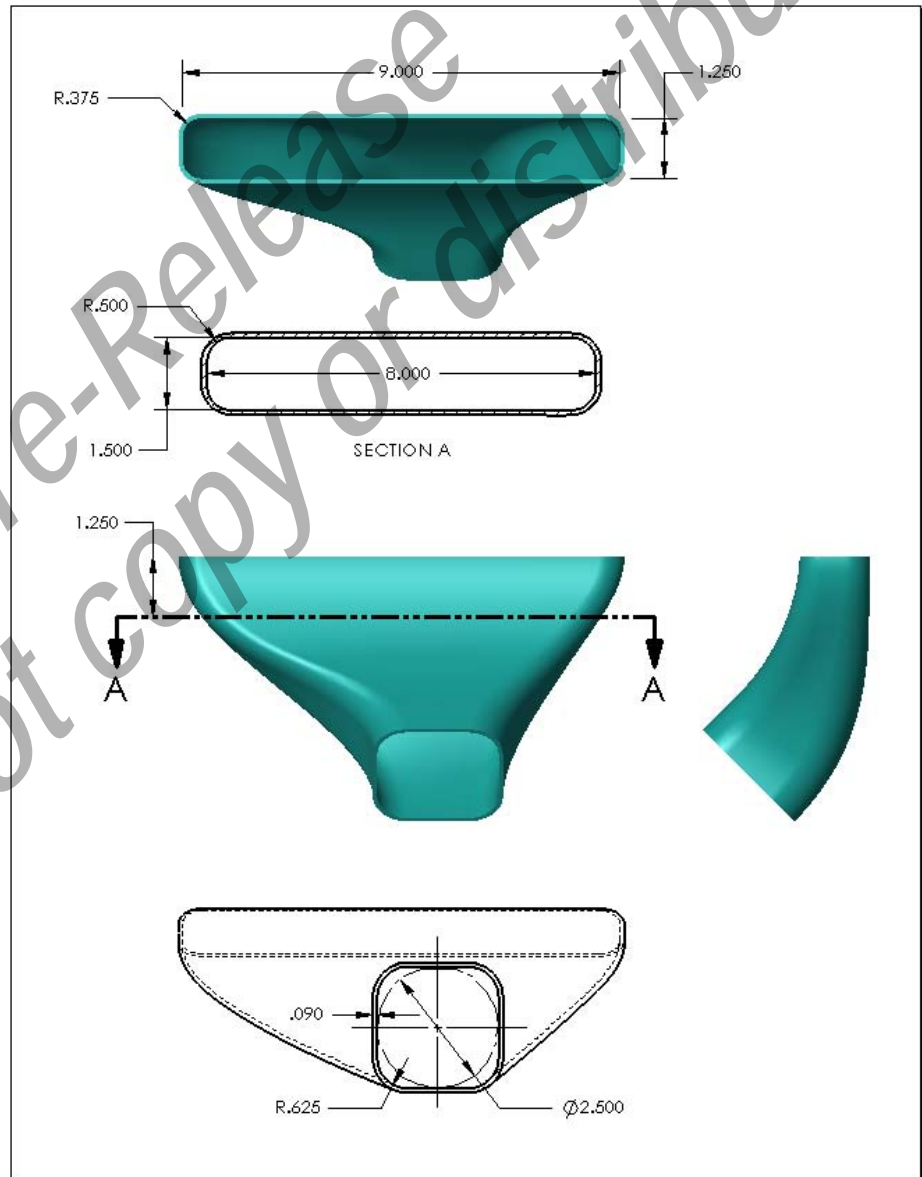
Upon successful completion of this lesson, you will be able to:

- Create a boss by lofting between profile sketches.
- Model free-form shapes using advanced lofting and filleting techniques.
- Use Split Entities to divide a sketch curve.
- Use the Deviation Analysis tool to compare faces along edges.
- Modify solid bodies using Flex.

Pre-Release
Do not copy or distribute

Basic Lofting

Lofting enables you to create features that are defined by multiple sketches. The system constructs the feature – either a boss or a cut – by building the feature between the sketches. We are given the dimensions of the bottom, top, and an intermediate section of the part, as well as its height. This sort of problem lends itself very well to lofting.



Stages in the Process

The major steps in this operation are:

- **Create the sketches.**
For best results they should be made up of the same number of entities and you should give some thought to how the entities will map one to the other during the loft. To save time, the sketches have already been created for this example.
- **Optionally create guide curves.**
Guide curves can optionally be used with lofting to give more control over the *transitions* between the profiles.
- **Insert loft between profiles.**
Where you select each profile and the order in which you select them is important.

Introducing: Loft

Inserting a **Loft** creates a boss, cut or surface using profiles and optionally, guide curves. The loft is first created between the profiles and optional guides provide additional control over how the shape in between the profiles is generated.

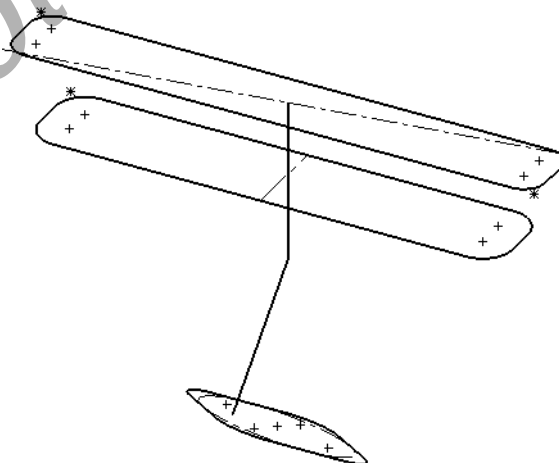
Where to Find It


- Click **Lofted Boss/Base** on the Features toolbar.
- Or, click **Insert, Boss/Base, Loft**.
- Or, click **Insert, Cut, Loft**.

Procedure

Consider the following procedure:

- 1 **Open the part Defroster Vent.**
The part consists of three sketches as shown.



- 2 **Insert a loft.**
Click **Insert, Boss/Base, Loft**, or click **Lofted Boss/Base**  on the Features toolbar.

3 Loft PropertyManager.

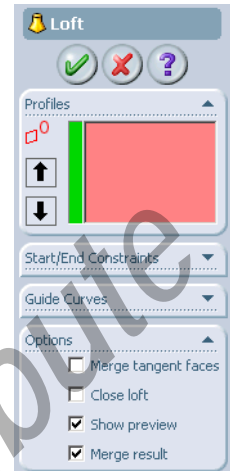
Click in the **Profiles** list and select the two sketches in the graphics window. You should pick in roughly the same location on corresponding entities in each sketch.

Note

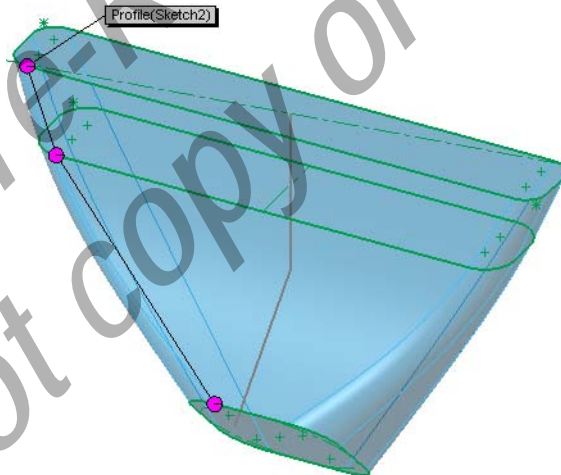
When lofting three or more sketches they have to be in the proper sequence. If the profiles are not in the correct order in the list, you can reposition them using the **Up** and **Down** buttons.

Tip

Although **Show preview** improves visualization as you select the profiles, with complex shapes, the preview tends to slow the system response.

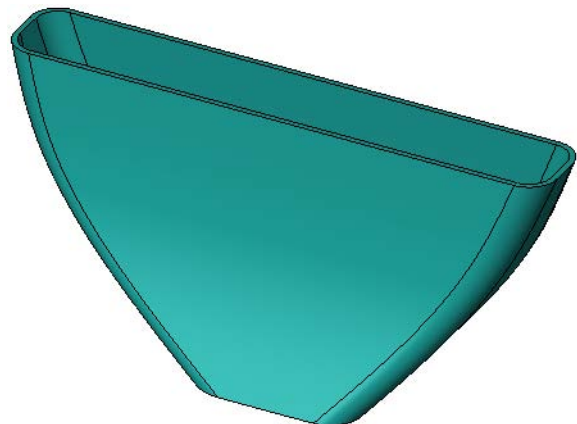
**4 Preview.**

As you select the sketches, the system generates a preview showing which vertices on the sketches will be connected during the loft. Pay close attention to this preview because it will show you if the loft is going to twist. A callout also appears to identify the profiles.

**5 Click Thin Feature.**

Set the **Thickness** to **0.090** inches. Make sure the thickness is added to the *outside* of the profiles.

Click **OK** to create the feature.



Merge Tangent Faces

The **Merge tangent faces** option causes the surfaces in the loft feature to be tangent if the corresponding segments in the profiles are tangent. Faces that can be represented as planes, cylinders, or cones are maintained. Other adjacent faces are merged, and the sections are approximated. Sketch arcs may be converted to splines.

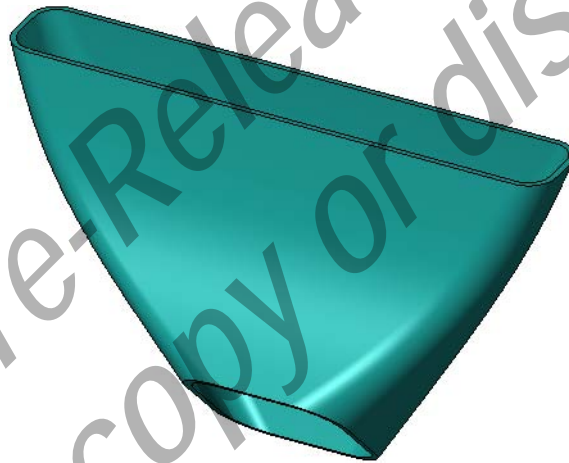
6 Edit the feature.

Edit the definition of the Loft feature.

Under **Options**, click **Merge tangent faces**.

Click **OK**.

Notice that the edges that corresponded to the ends of the lines and arcs in the profiles are now gone. Compare this to the results in step 5.

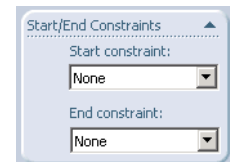


Start and End Constraints

When lofting, you can control how the feature is built by using options that influence how the system starts and ends the loft at the beginning and ending profiles. You can also control the length and direction of the influence at each end.

7 Edit the feature.

Edit the definition of the loft feature. Expand the **Start/End Constraints** group box. By default, no special tangency options were applied to the start and end of the loft.

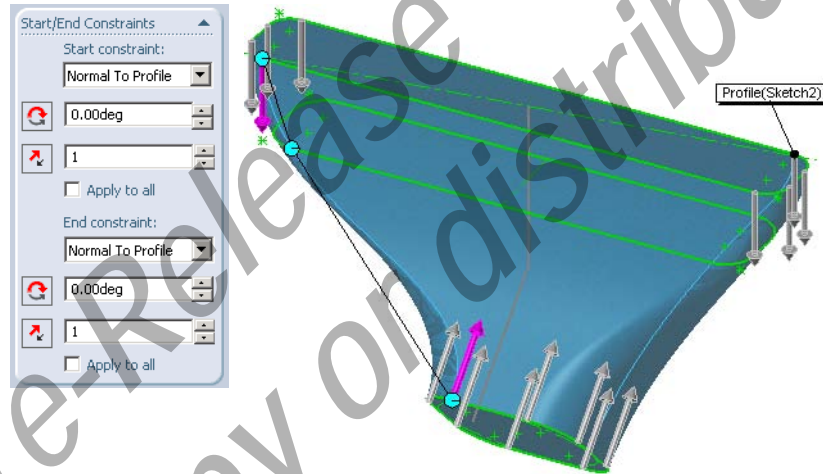


8 Normal to Profile.

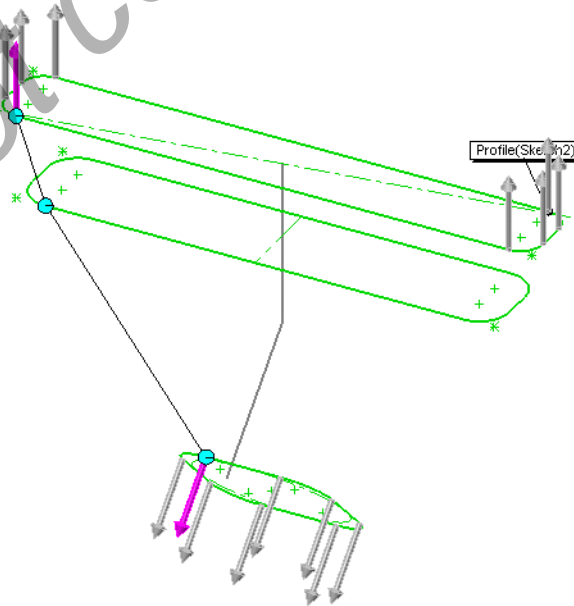
Select the options **Normal to Profile** for both the start and end of the loft. The tangent vector arrows should point in the directions shown.

If they do not, click **Reverse Direction**  to reverse the direction.

Leave the start and end tangent length values at the default **1**. Changing the tangent length change the influence on the shape of the loft. You can change all the **Tangent Length** values by typing a value and clicking **Apply to all**. Individually, a single tangent vector arrow can be dragged.



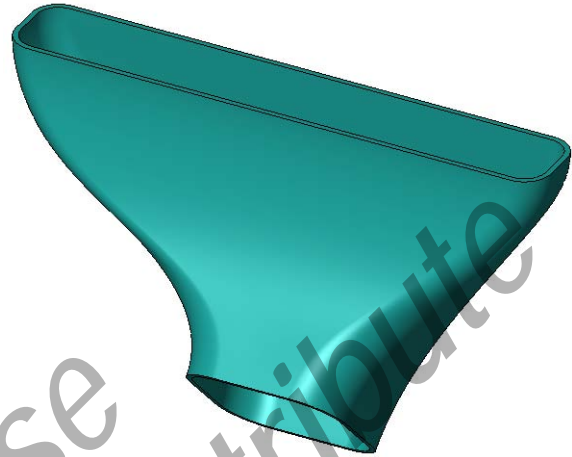
Pay attention to the preview. If the tangent arrows are in the incorrect direction, the preview will look something like the illustration below.



Click **OK**.

9 Results.

The result is that the shape of the loft is altered so that the faces of the feature start and end normal (perpendicular) to the plane of the profile sketches.



Note

The **Draft Angle** option



with **Normal to Profile**

applies draft with respect to the planes of the profiles. If it is used with the **Direction Vector** option, the draft is applied with respect to the direction vector.

10 Save and close the part.

Merging a Multibody with Loft

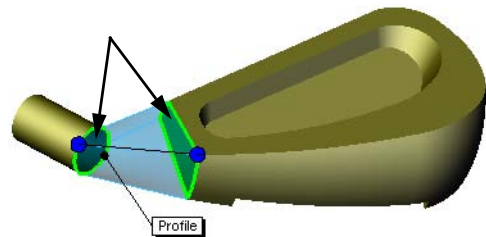
The **Merge result** check box can be used on any boss feature aside from the first feature. In this example, we will create the transitional feature from the head of a golf club into the shaft using a multibody.

1 Open Lofted Merge.

The part contains two solid bodies that cannot be merged.

2 Insert a loft feature.

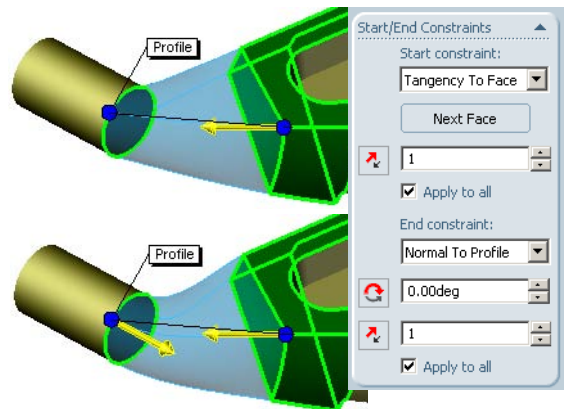
Insert a loft feature between the planar faces of the two bodies. Select the faces in similar areas.



3 Start/End constraints.

The two tangency options used are **Tangency To Face** for the selection on the *head* and **Normal to Profile** for the selection on the *shaft*.

The **Next Face** button is used to resolve any ambiguity as to which set of faces is used.



Merge result must also be checked.

Note The option **Curvature To Face** could be used in place of **Tangency To Face** to make the faces match in curvature.

4 Merged feature.

Once the feature is added, the part contains only one solid.

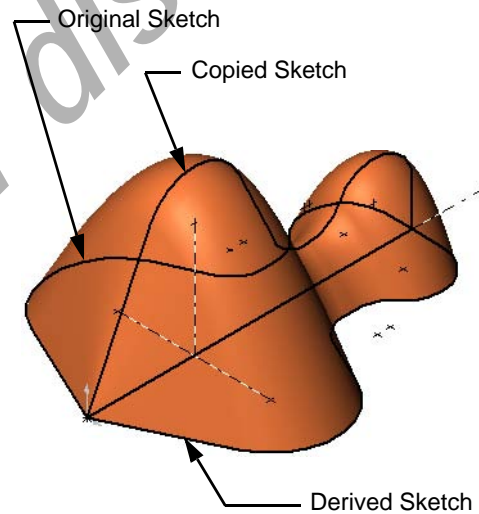


Using Derived and Copied Sketches

Lofted features may have many sketches to describe the **Profiles**, **Guide Curves** or **Centerlines**. Many of the sketches may be similar or exactly the same. Derived and copied sketches can help reduce the amount of sketching required.

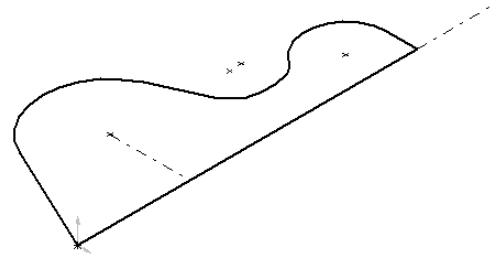
- **Derived Sketches** are exact duplicates of the original sketch and retain the link from the original to the derived. They can only be placed, not changed.
- **Copied Sketches** are also duplicates of the original sketch but can be changed in any way. There is no link back to the original.

Consider a decorative shape like the one shown in the illustration. Two sketches of the loft are the same (the original sketch and derived sketch) while the third is similar, but not identical.



1 Open part.

Open the part *Derive&Copy*. It contains a single sketch named *Source*.




Copying a Sketch

To create another profile of similar shape, copy and paste the existing sketch onto the desired sketch plane. Copied sketches can be edited in any way and are *not* linked back to the original. In this example, the sketch *Source* will be copied onto the plane *Right* and edited.

2 Select sketch.


Select the sketch **Source**. The sketch geometry will highlight on the screen.

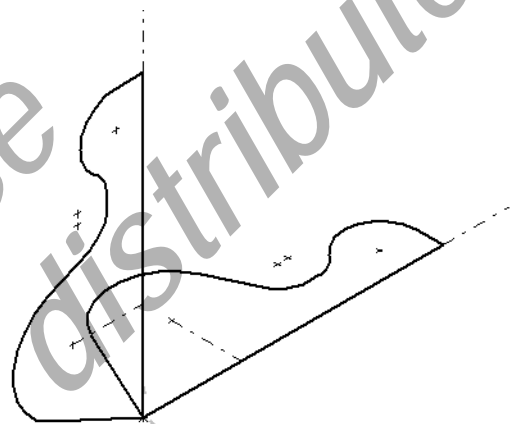
3 Copy sketch.

Using **Ctrl+C**, or **Edit, Copy** or the **Copy** tool  on the Standard toolbar, copy the sketch to the clipboard.

4 Select plane and paste.

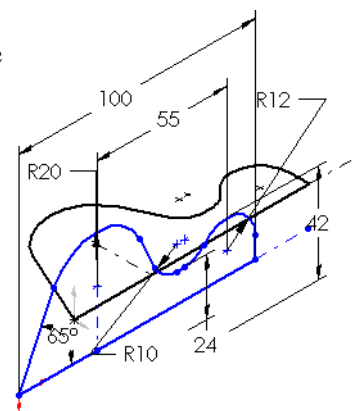
Select the plane **Right** from the FeatureManager design tree and click **Ctrl+V**, or **Edit,**

Paste or the **Paste** tool  on the Standard toolbar. The sketch will be pasted from the clipboard to the selected plane. It will appear on the screen in the plane's orientation.



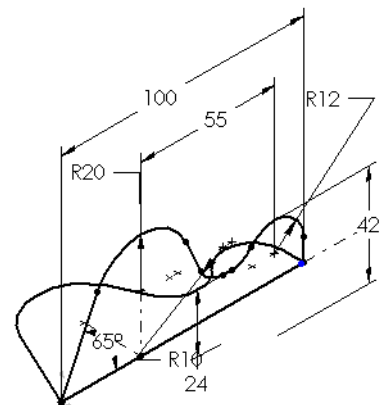
5 Edit sketch.

Select the new sketch and **Edit Sketch**. Use **Modify Sketch** to rotate and move the sketch geometry. Relations and dimensions will be needed to fully define the sketch.



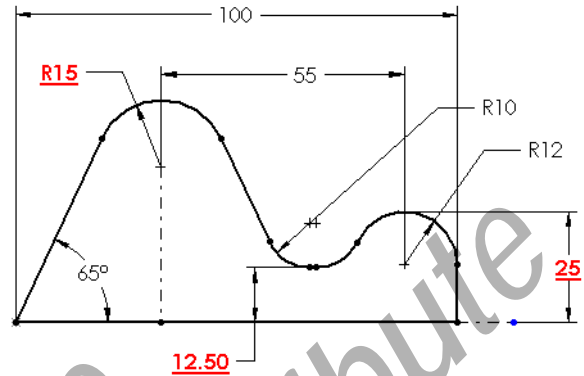
6 Add relations.

Add **Collinear** and **Coincident** relations between the profiles. The sketch is fully defined.



7 Make changes.

Make some changes to the dimensions in the sketch. Change the bold, red, underlined dimensions as shown. Note that two of them are also changed from **Diameter** dimensions.



Exit the sketch and rename it Copied.

Derived Sketches

Introducing: Insert Derived Sketch

A **Derived Sketch** is used to create a copy of the **Source** sketch on a different plane and location. The derived sketch will be a child of the original sketch.

Insert Derived Sketch is also used to create a copy of a sketch. Derived sketches are dependent on the original for size and shape but not location and usage. You cannot edit the geometry or dimensions of a derived sketch. You can only locate it with respect to the model. Changes to the original sketch propagate to the derived copies.

Where to Find It

- From the **Insert** menu, choose **Derived Sketch**.

Creating a Derived Sketch

Create the derived sketch on the plane **Top**. Once copied, the sketch can be rotated and repositioned if it is at the wrong orientation.

8 Select sketch and plane.

Hold down **Ctrl** and select the sketch **Source** and the plane you want it copied to (**Top**). The sketch will be copied to the selected plane in the next step.

9 Insert a derived sketch.


Click **Insert, Derived Sketch**. The sketch is inserted onto the selected plane, but it is under defined.

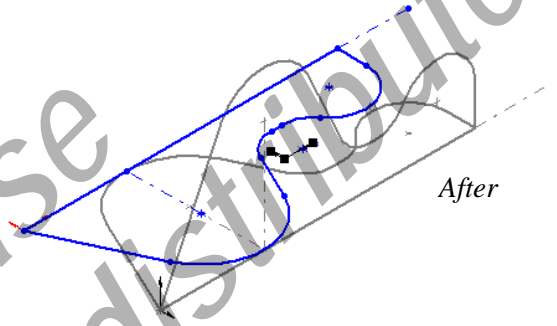
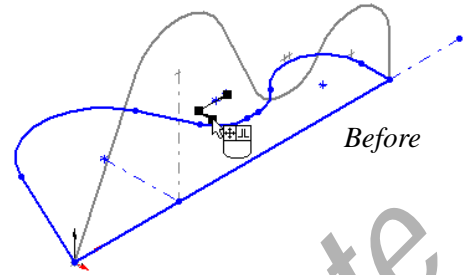
Unlike **Copy** and **Paste**, the system automatically puts you into the **Edit Sketch** mode. Also, notice that derived sketches are identified as such by the **derived** suffix appended to their names in the **FeatureManager** design tree.

Locating the Derived Sketch

Derived Sketches are inserted under constrained and often out of orientation.

10 Modify the sketch.

Click **Modify Sketch** .
Position the cursor over the black origin symbol as indicated. Click the right-mouse button to mirror the sketch.



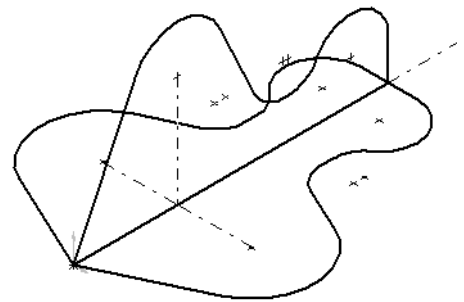
11 Drag.

Move the sketch to the right and close the **Modify Sketch** dialog.




12 Fully define.

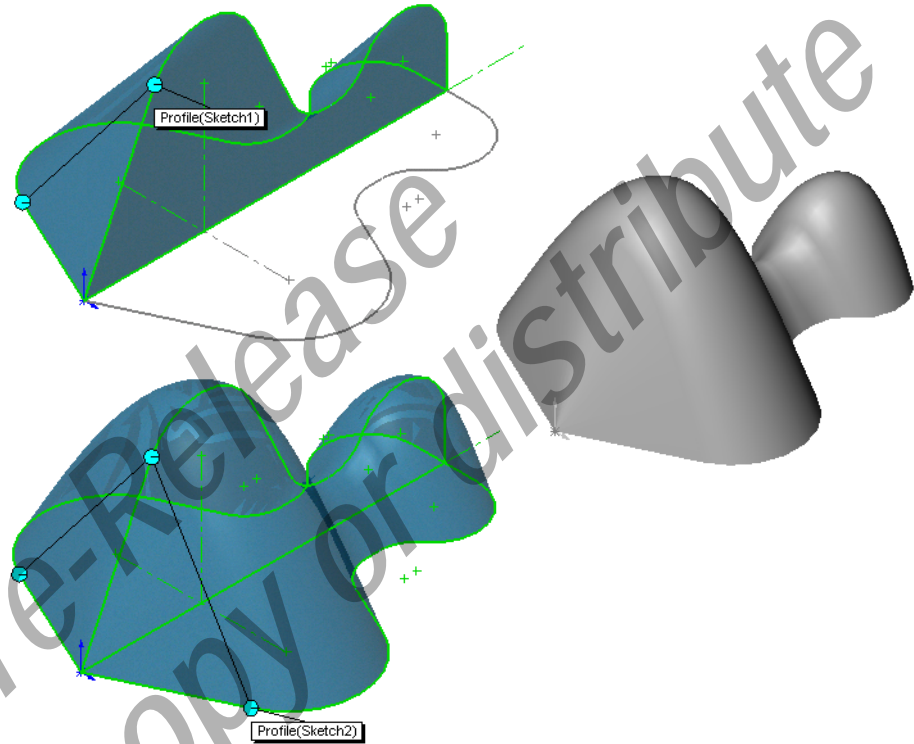
Add **Collinear** and **Coincident** relations similar to those used in step 6.



13 Insert a loft.

Click **Loft Boss/Base** . Select **Merge tangent faces**.

Loft the three profiles without using guide curves or centerlines. Select the profiles near a common vertex.

**Loft Viewing Options**

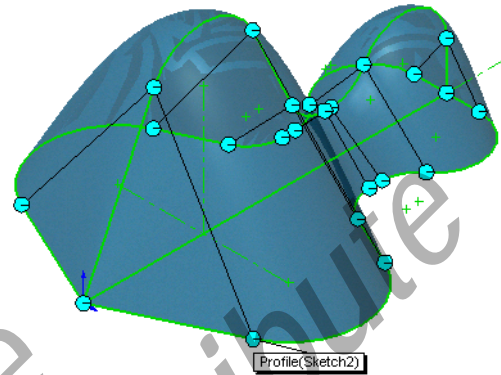
Loft features can be viewed with **Connectors** or **Mesh** previewed. By default, only the selection connectors are shown in the loft and no mesh is displayed.

Where to Find It

- Right-click in the graphics window while editing a Loft feature and choose **Show All Connectors** or **Hide All Connectors**.
- Right-click in the graphics window while editing a Loft feature and choose **Mesh Preview, Mesh All Faces** or **Mesh Preview, Clear All Meshed Faces**.

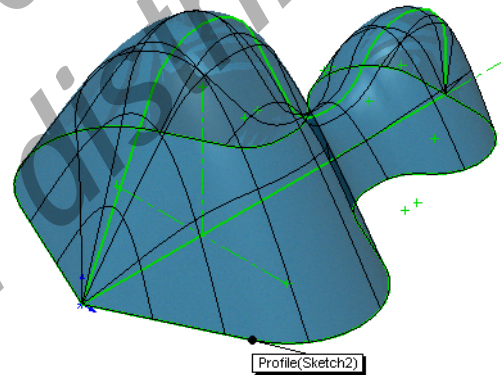
14 Show All Connectors.

Right-click **Show All Connectors** to view connections between all the profile endpoints. Each endpoint can be dragged and dropped. Right-click **Hide All Connectors**.



15 Mesh Preview.

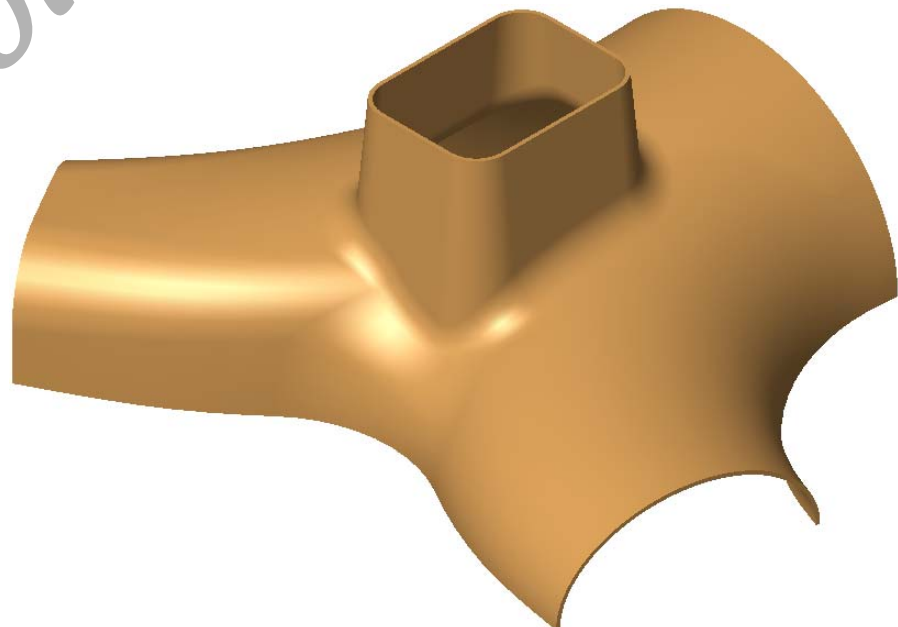
Right-click **Mesh Preview, Mesh All Faces**. A surface mesh overlays the shaded preview. Right-click **Mesh Preview, Clear All Meshed Faces** and click **OK**.



16 Save and close the part.

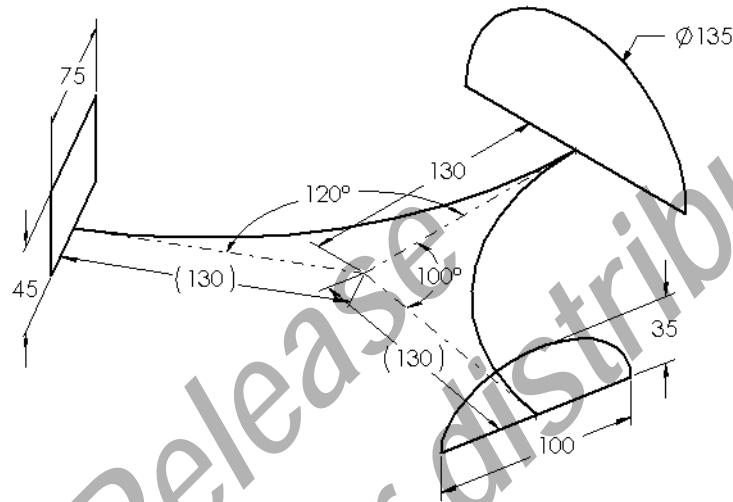
Advanced Lofting

The part shown at the right is a heat shield that goes over a hot gas manifold. It consists of several shapes – a semi-circle, a rectangle, a half ellipse – all of which must be smoothly blended together. Since the basic shapes are the result of blending two or more profiles, lofting is the approach of choice.



1 Open part.

Open the part Heat Shield. To save time, we will start with this part that already has the basic geometry defined.

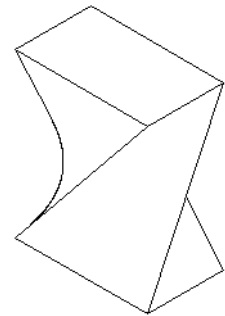
**Preparation of the Profiles**

When lofting, you have to give special consideration to the way you sketch the profiles, and how you subsequently select them in the Loft command. In general, there are two rules you should follow for good results:

- **Pick the same corresponding spot on each profile.**

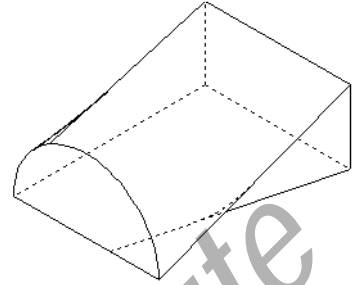
The system connects to points you pick. If you are careless, the resulting feature will twist.

If the profiles are circles there are no ends to pick such as there are on rectangles. That makes picking corresponding spots tricky at best. In this situation, put a sketch point on each circle and pick them when you select the profiles.



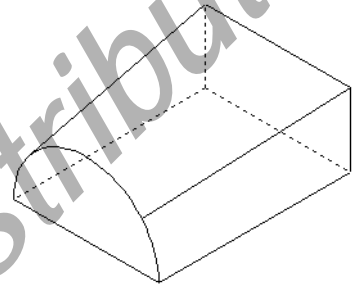
- **Each profile should have the same number of segments.**

In the example at the right, a closed semi-circle (2 segments) was lofted to a rectangle (4 segments). As you can see, the system blended one side of the rectangle into part of the arc, another side into the remainder of the arc, and so on. This does not give a good result.



You have two options:

- Interactively add or move connector points during the Loft command.
- Subdivide the arc manually so you can control exactly which portion of the arc corresponds to each side of the rectangle.

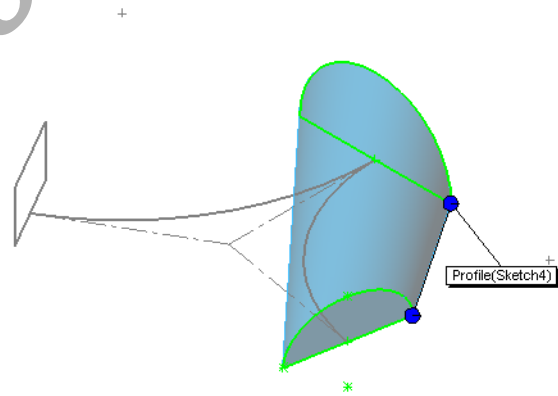


2 Insert a loft.

Click **Loft Boss/Base** , or click **Insert, Base, Loft**.

3 Preview.

Select the two profiles and notice the preview. Be careful to pick the same relative corner of each profile.



Tip

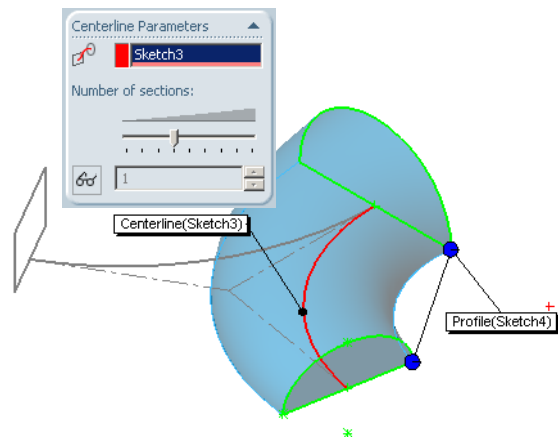
Because of the importance of where you pick the profiles, it is usually *not* a good idea to select them from the FeatureManager design tree.

4 Centerline.

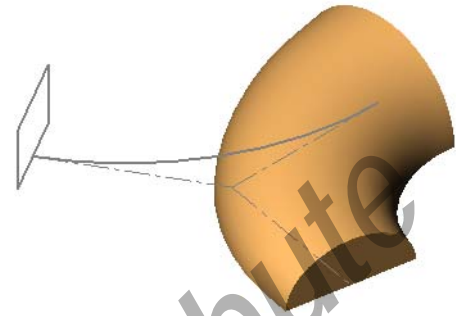
Expand the **Centerline Parameters** group box.

Select the centerline (Sketch3).

Click **OK** to create the feature.



5 Results.

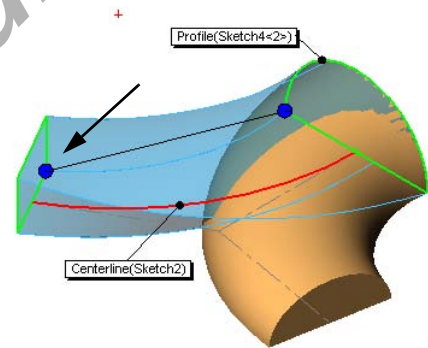


Sharing Sketches


Sketches that have been *absorbed* when used to create extrudes, revolves, sweeps and lofts can be used again to create additional features. They can simply be selected from the Feature Manager to become part of the new feature.

6 Insert another loft.

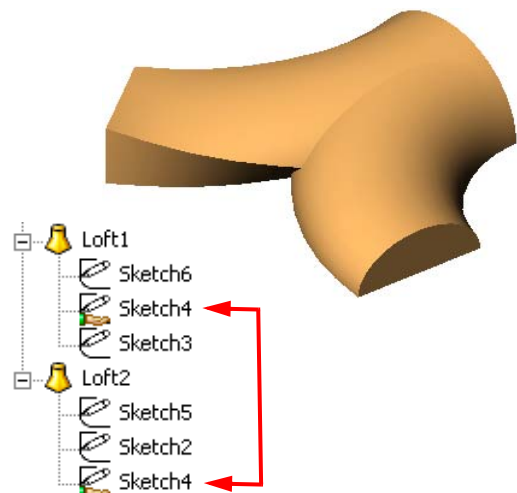
Loft between **Profiles** Sketch5 selected on the lower corner and the absorbed Sketch4 selected from the Feature Manager. Use Sketch2 as the **Centerline**.



7 Shared sketch.

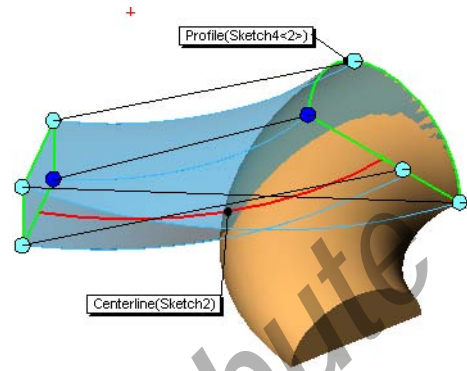
Sketch4 is shared by both loft features, as indicated by the name and the  symbol. Editing the sketch would change both features.

Unfortunately the loft shape, although valid, is not desirable. The fact that the profiles have different numbers of edges adversely influences the shape of the feature.



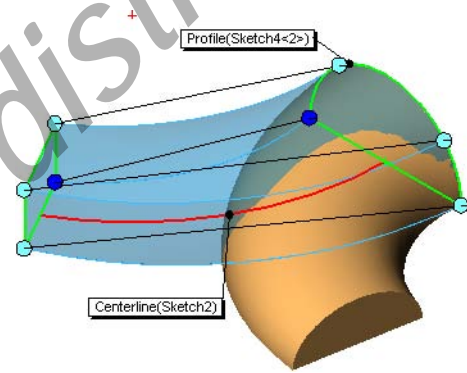
8 Show Connectors.

Right-click the **Loft2** feature and select **Edit Feature**. Right-click in the graphics area and select **Show All Connectors**. Colored circles appear at the endpoints of the segments of the profiles. Notice that a connector has been added to the semicircular profile. This is because both profiles must have the same number of segments. If you did not sketch them that way, the system breaks them for you.



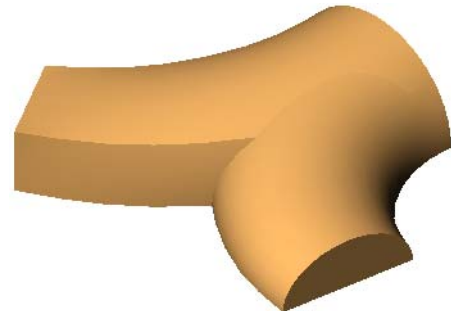
9 Synchronize the profiles.

Drag the connectors to improve how the rectangular profile maps to the semicircular profile. Click **OK** to rebuild the feature.



10 Results.

Although dragging the connectors is very interactive, it may not be precise enough for some applications. If precise control is needed over how the profiles map to each other, you should manually subdivide the profile.

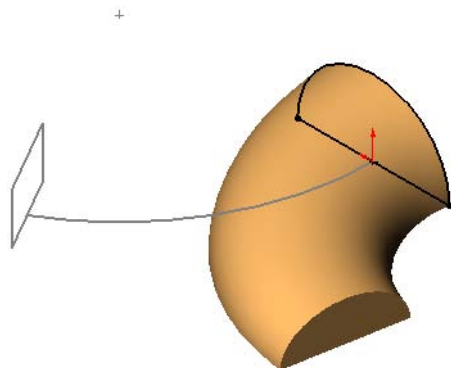


11 Delete.

Delete the **Loft2** feature and use a modified sketch with equal numbers of segments.

12 Recreate the sketch.


Select the flat face and open a sketch. Click **Convert Entities** to create copies of the arc and line edges in the sketch.



**Introducing:
Split Entities**

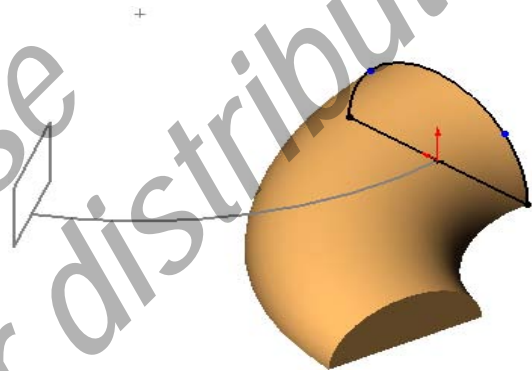
Split Entities breaks a single sketch curve into multiple pieces at selected locations.

Where to Find It

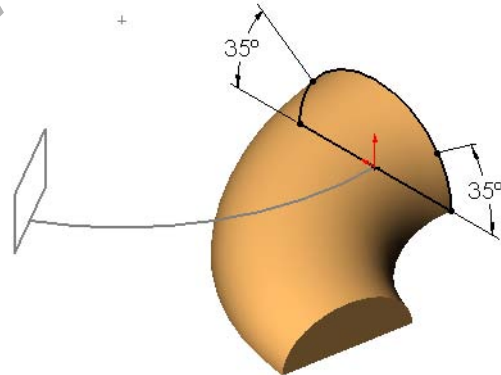
- On the Sketch toolbar click the **Split Entities** tool .
- Or, click **Tools, Sketch Tools, Split Entities**.
- Or, right-click a sketch segment and choose **Split Entities**.

13 Split entities.

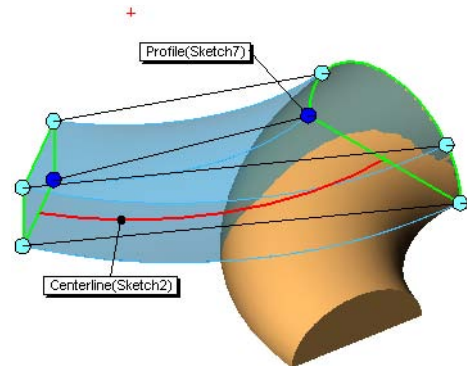
Divide the arc into three pieces by using **Split Entities** at two locations along its length. Position the breaks on either side of the center. All three arcs are coradial but their arc angles are under defined.

**14 Angular dimensions.**

Dimension the arcs at **35°** using 3 point angular dimensions. If you want, you can link the values of the angles so when you change one, they both change.

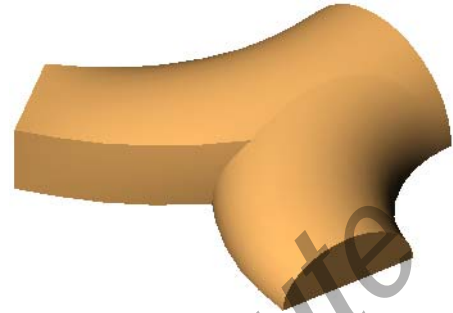
**15 Exit the sketch.****16 New Loft.**

Create a second centerline loft between the two four-sided sketches using the centerline curve. Right-click **Show All Connectors** to display the matching endpoints.



17 Results.


The second loft merges into the first, forming a single solid.




**Introducing:
Deviation Analysis**

The **Deviation Analysis** tool can be used to determine the angular difference between faces along common edges. A 90 degree value indicates perpendicular faces, 0 degrees indicates tangency.

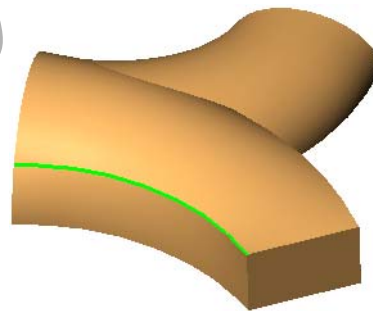
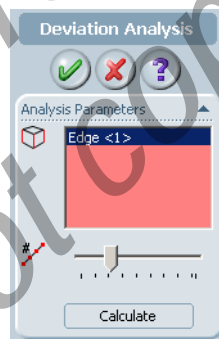
Where to Find It

- From the Tools toolbar click the **Deviation Analysis** tool .
- Or, click **Tools** menu choose **Deviation Analysis**.

18 Analysis parameters.

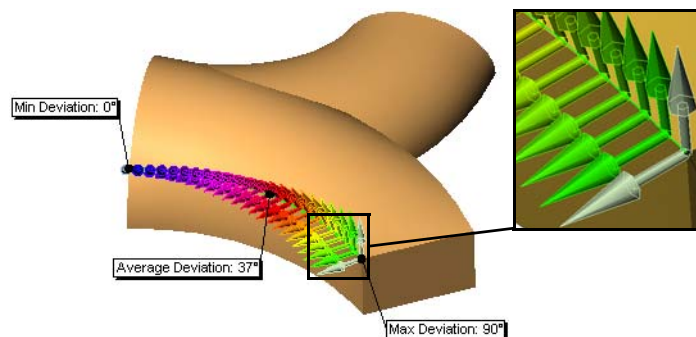
Click **Deviation Analysis**  and select the model edge shown. Set the number of sample points slider control to halfway.

Click **Calculate**.



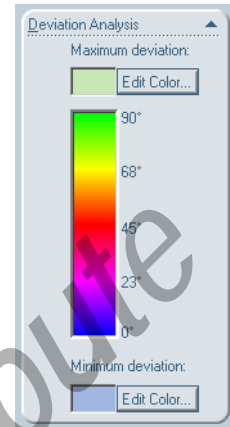
19 Deviation Analysis graphics.

The results of the deviation analysis appear as pairs of 3D arrows on the edge. They are color coded to show the change in angle between the faces along the common edge.

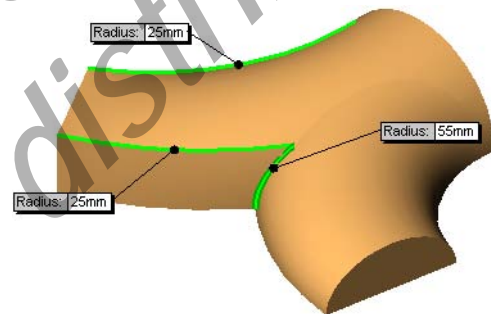


20 Color settings.

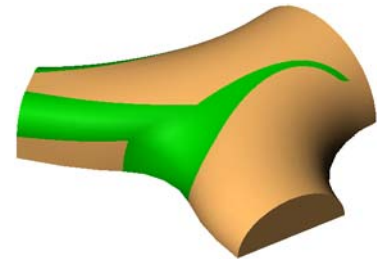
The color settings used for the arrows can be changed.

**21 Add fillets.**

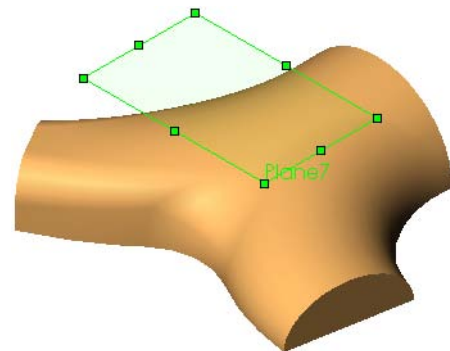
Run a **25mm** fillet down the two sharp edges of the second loft. Run a **55mm** radius fillet up the edge between the two lofts. You can use a multiple radius fillet if you wish, or create two separate fillets.

**Note**

Fillets shown in color for clarity.

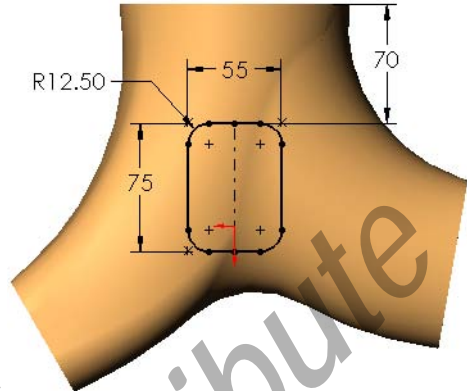
**22 Create an offset plane.**

Create a plane offset **100mm** from the Top reference plane. This will be used to sketch the profile of the rectangular inlet tube.



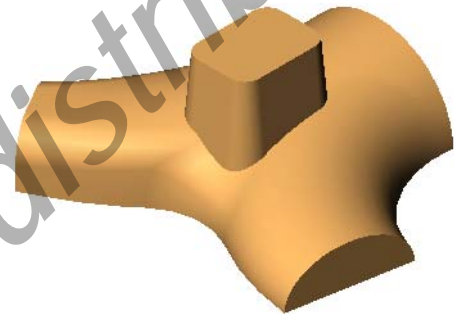
23 Sketch profile.

Sketch a rectangular profile as shown. Fillet the corners with sketch fillets. The profile is centered left-to-right with respect to the Origin.



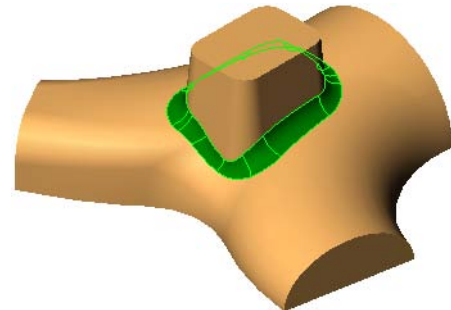
24 Extrude.

Extrude a boss using the end condition **Up to Next**, and **5°** of **Outward Draft**.



25 Add fillet.

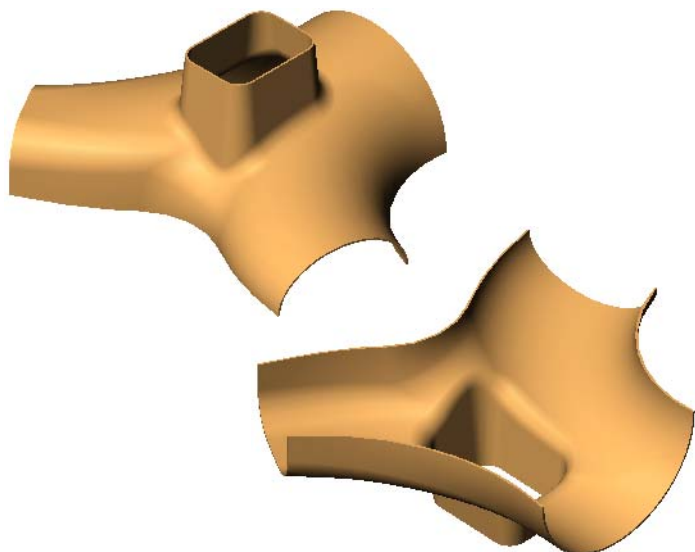
Run a **12.5mm** radius fillet around the base of the boss.



26 Shell part.

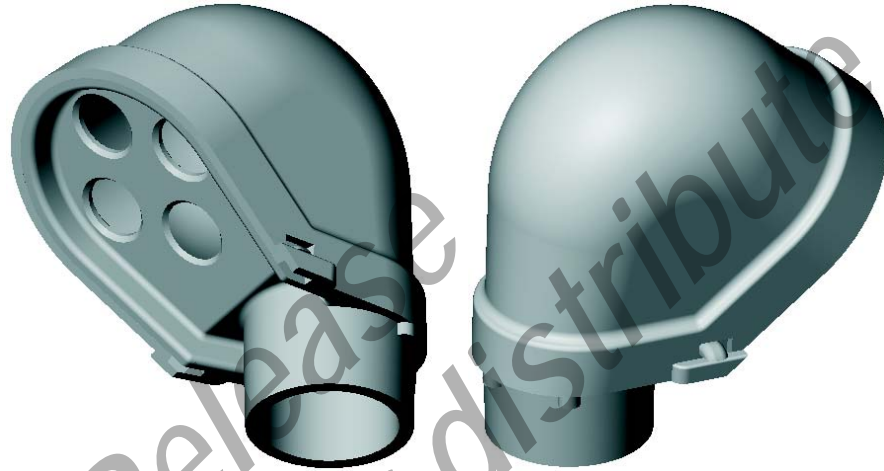
Shell the part towards the inside using a wall thickness of **1.5mm**.

Save and close the part.

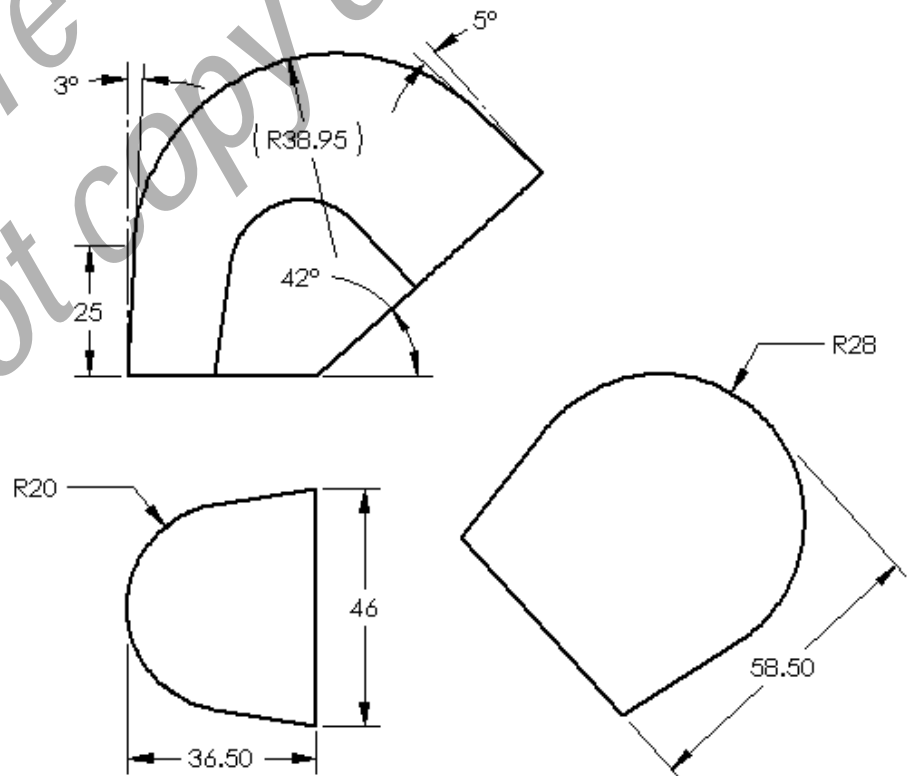


Other Techniques

Sometimes the best approach to modeling a free-form shape is not to use sweeping or lofting. Consider, for example, the two part assembly shown below. This is a weather-proof service head for an electrical conduit.



The cover presents an interesting modeling problem. Let's take a look at just its basic shape which is shown below in a simplified drawing.



We can see from the drawing that the shape is defined by two “teardrop” profiles that are blended together along the path shown in the front view.

Stages in the Process

Some of the key stages in the modeling process of this part are given in the following list:

- **Extrude up to surface.**
Having defined the basic profile and the angled plane, we extrude a boss up to the plane.
- **Advanced filleting.**
We will use some advanced filleting techniques to round off the part creating the smooth, blended transition between the two teardrop shapes.
- **Symmetry.**
Given the symmetry of the part, we want to take advantage of mirroring. We will model half of it and then mirror everything using **Mirror All**.
- **Shell.**
After mirroring the basic shape, we will shell it out to the desired wall thickness.

Procedure

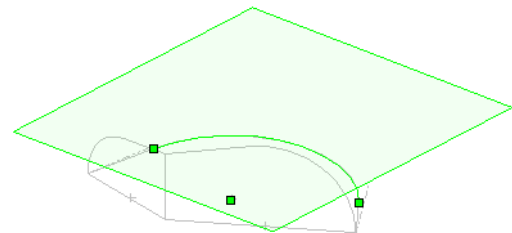
Begin by opening an existing part.

1 Open part.

Open the part
Cover Sketches.

There are three sketches used to form the profiles of the “teardrop” shape.

The plane Up To is generated from three endpoints of sketches and therefore is skewed.



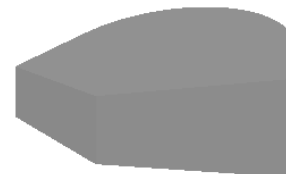
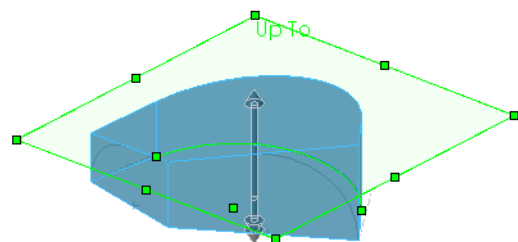
2 Up to Surface.

Using Sketch1, create an extrusion **Up To Surface** using the plane Up To as the surface.

Click **OK**.

This makes the basic shape.

Next, we have to round off the edge.



Advanced Face Blend Fillets

A face blend fillet differs from an edge fillet in that instead of selecting an edge, you select two sets of faces. The advanced options enable you to use geometry to define the radius of the fillet instead of specifying a numeric radius value. This is very powerful.

Introducing: Face Fillet

The **Fillet** command has an additional group box, **Fillet Options**, where a **Hold Line** can be assigned to define the fillet's tangent edge or rail. Defining the rail of the fillet defines the fillet's radius. In this case the bottom edge of the part will be used.

Where to Find It

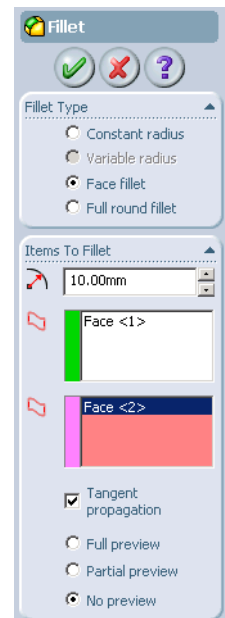
- **Face Fillet** is located on the **Fillet** PropertyManager.

3 Insert fillet.

Click **Fillet** . In the **Fillet Type** group box, choose the **Face Fillet** option.

Note

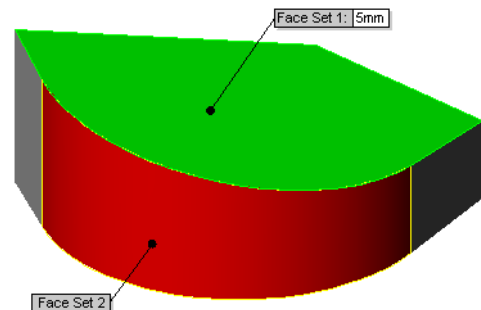
Since the **Hold line** will define the radius, you do not need to enter a radius value. Also, when you expand the **Fillet Options** group box and select the **Hold lines**, the radius field disappears.

**4 Select faces.**

Verify that the **Face Set 1** selection list is active and select the top face of the part.

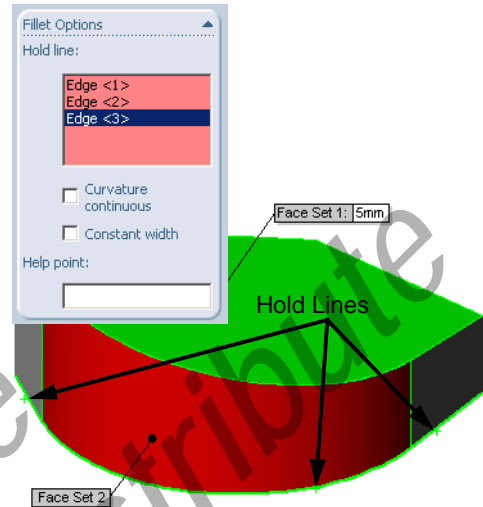
Activate the selection list for **Face Set 2** and select one of the three side faces.

With the default condition **Tangent propagation** enabled, picking one face will select all three.

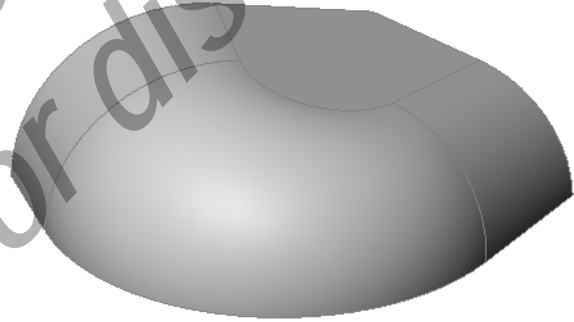


- 5 Add fillet options.**
Expand the **Fillet Options** group box. Click in **Hold line** selection list, and select the three edges as shown in the illustration.

Click **OK** to create the fillet.

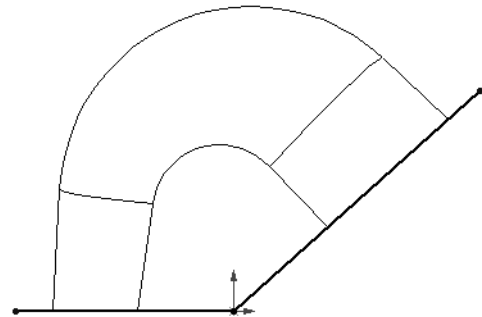



- 6 Results.**
The three vertical faces (Face Set 2) are completely removed. The fillet is created with a variable radius defined such that the fillet ends exactly on the hold lines.



- 7 Convert and drag.**
Switch to a Front view and open a new sketch on the Front reference plane. Select and convert the two straight edges of the first feature.

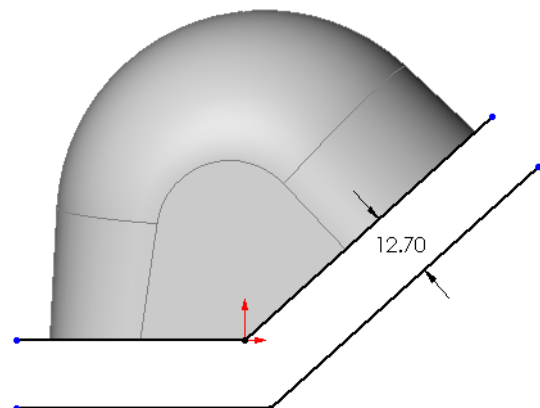
Although converted edges are fully defined, you can drag the endpoints, making the lines longer and therefore, under defined.



- 8 Offset sketch geometry.**
Click **Offset Entities** , and select one of the two converted edges.

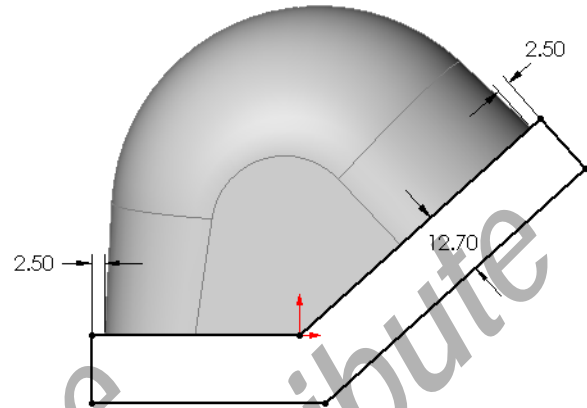
Set the offset value to **12.7mm** and use **Select chain** to offset both connected edges.

Click **OK**.



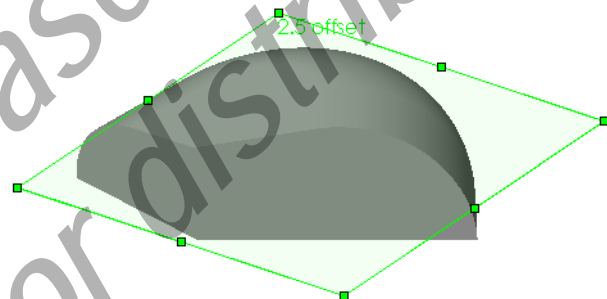
9 Dimensions.

Add lines to close off the ends and dimensions to fully define the sketch.

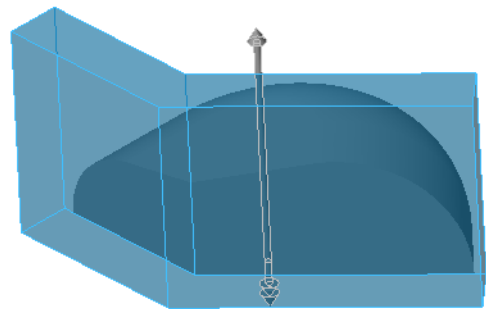
10 Exit sketch.**11 Offset plane.**

Create a new plane named 2.5 offset that is offset **2.5mm** from the Up To plane that was used for the base extrusion.

This plane will serve as the termination surface for the boss.

**12 Extrude using blind.**

Extrude the new sketch **50mm** or more.



**Introducing:
Cut With Surface**

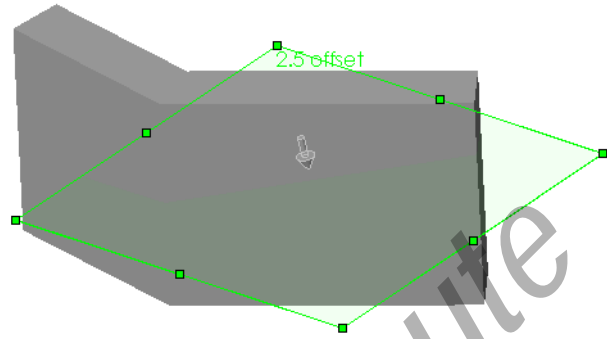
Solids can be cut with a *surface*, that is a true surface, a face or a reference plane. In this example a blind extrusion will be cut using a reference plane.

Where to Find It

- On the **Insert** menu click **Cut, With Surface**.
-

13 Cut with surface.

From the **Insert** menu, choose **Cut**, **With Surface** and select the plane 2.5 offset.

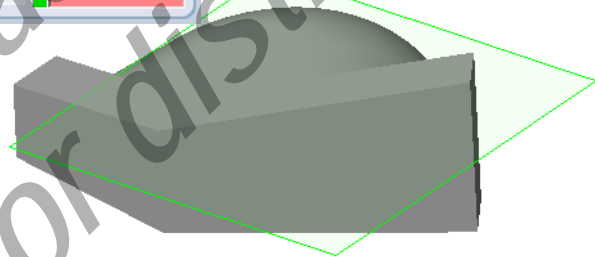


14 Direction.

Flip the direction arrow to point in the direction indicating the portion to remove.

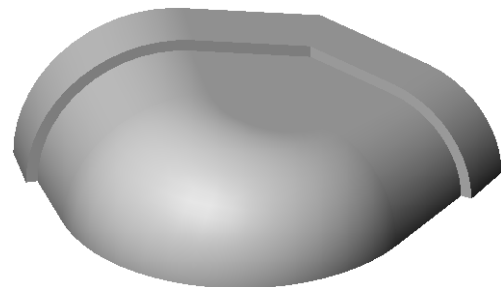


Click **OK**.



15 Add fillet.

Fillet the two ends using the same **Face Fillet** technique that you used in Steps 3 through 5.



Note

Face blend fillets do not work across discontinuous faces. Therefore, you will have to create these fillets in two operations, one for each end.


Introducing: Mirror

Not counting mirroring within sketches, there are four types of mirroring in SolidWorks:

- **Mirror Part:** Creates a new part that is the mirror image of a previously constructed (and saved) part. The copy has an external reference back to the original (like a derived part) so that changes to the original propagate to the copy.
- **Mirror Feature:** Creates a copy of a feature (or multiple features), mirrored about a plane.
- **Mirror Faces:** Allows you to mirror features by selecting all of their faces. This is ideal for imported, non-parametric parts.
- **Mirror Body:** Creates a symmetrical part by mirroring an existing solid body with respect to a planar face.

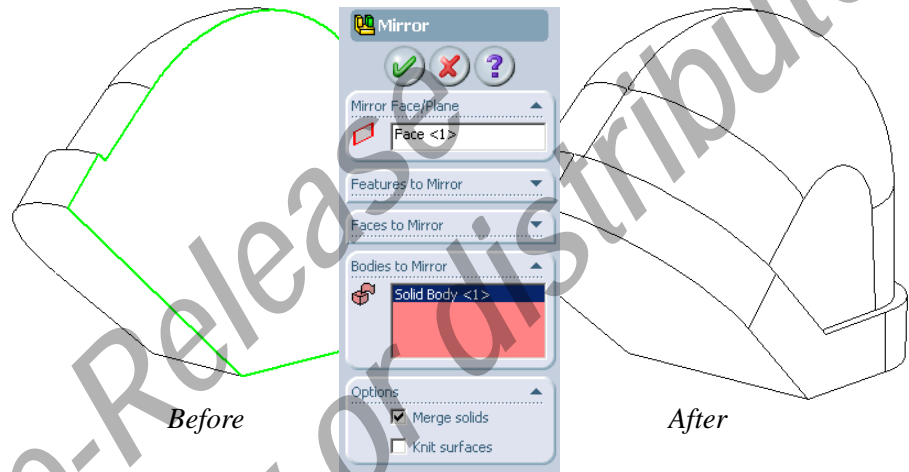
Since this part is symmetrical, we will use **Mirror Body**.

Where to Find It

- Click **Mirror**  on the Features toolbar.
- Or, click **Insert, Pattern/Mirror, Mirror**.

16 Mirror body.

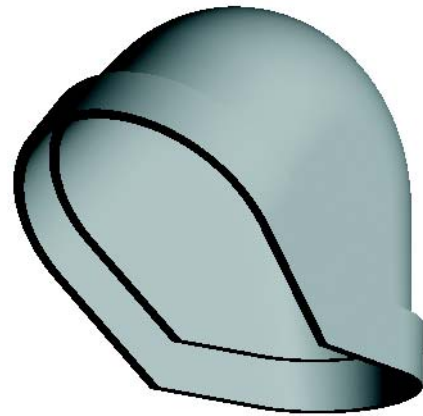
From the **Insert** menu, choose **Pattern/Mirror, Mirror** and select the planar face to mirror about. Under **Bodies to Mirror**, select the solid. Click **OK**.

**17 Shell part.**

Remove the two flat faces by shelling the part with a wall thickness of **2.5mm**.

Conclusion


The rest of the features are fairly simple and basic so we will not take the time to go into them here. In fact, if we were to complete building this part, we would probably postpone the mirroring operation until the end. This would simplify the process of creating the fillets and the hole and boss on the side.



Using Flex


The **Flex** feature is used to bend, twist, taper or stretch selected solid bodies. The feature is applied to the geometry *between* the Trim Planes.

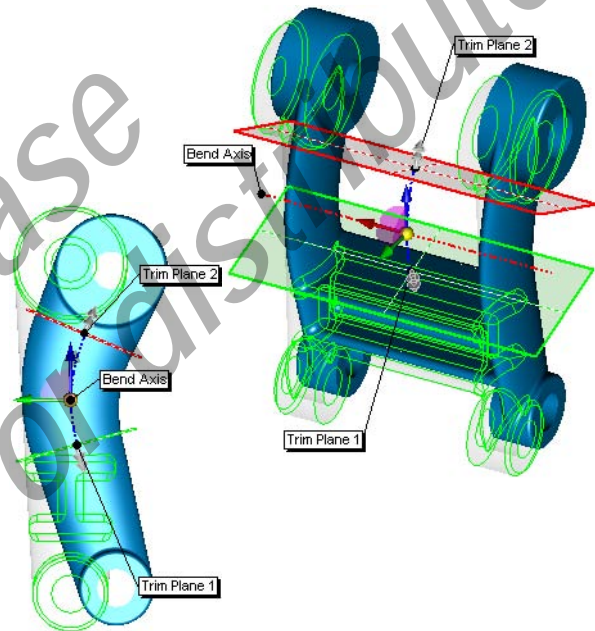
Where to Find It

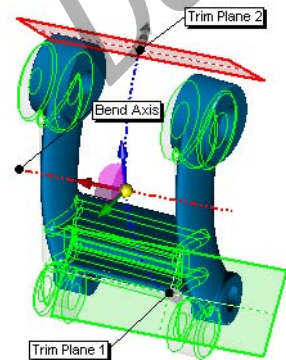
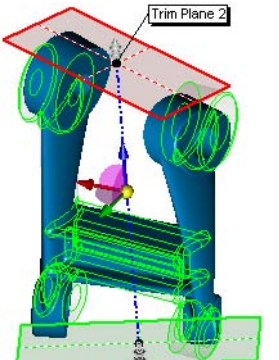
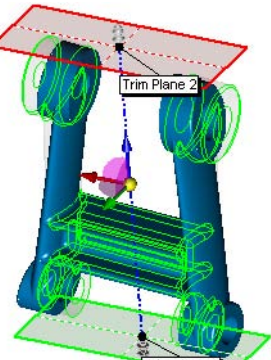
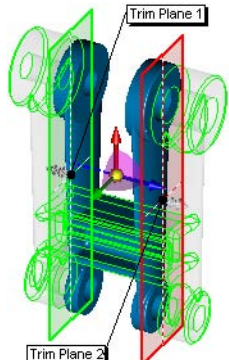
- Click **Flex**  on the Features toolbar.
- Or, click **Insert, Features, Flex**.

Triad and Trim Planes

The Flex is controlled using the Triad and Trim Planes. These components appear during the command.

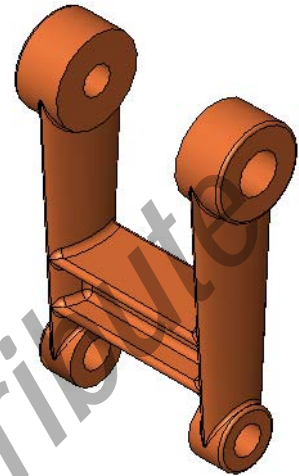
- **Trim Planes**
Trim Planes are created at the extents the part but can be moved by dragging the  arrow.
- **Triad**
The Triad is a coordinate system that sets the center of the flex and the orientations of the Trim Planes.
- **Flex Input**
The Flex Input determines how the geometry is flexed. The options are shown in the chart below.




Bending	Twisting	Tapering	Stretching
Bending occurs about the (red) Bend Axis between the Trim Planes .	Twisting occurs about the (blue) Z axis between the Trim Planes .	Tapering occurs along the (blue) Z axis between the Trim Planes .	Stretching occurs along the (blue) Z axis between the Trim Planes .
			

1 Open part.

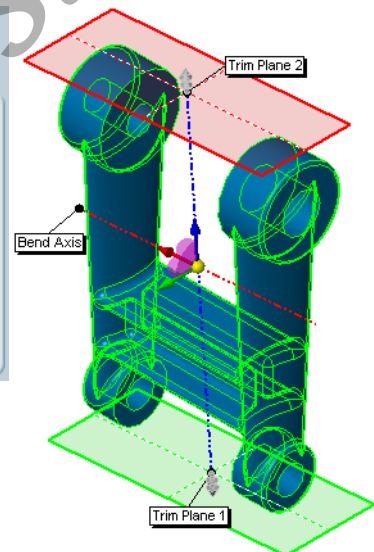
Open the existing part named Flex. The part is an imported solid body.


**2 Flex Input.**

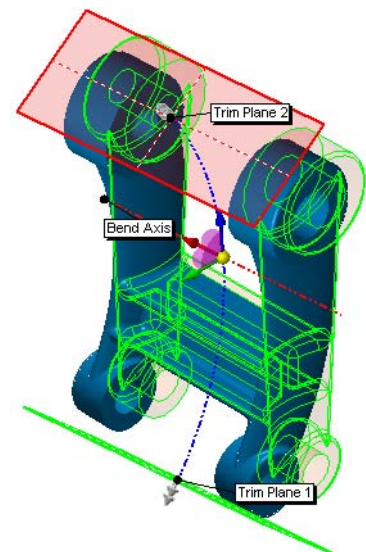
Click **Flex**  and select the solid body. Select **Bending** as the **Flex Input**.

Select **Hard edges** to avoid creating spline-based geometry whenever possible.

The Trim Planes appear at the upper and lower extents of the part with the Triad and Bend Axis between them.

**3 Bend.**

Position the cursor over Trim Plane 1 or Trim Plane 2 to display the bending cursor . Drag the plane to produce a bend similar to that show at right.



Flex Options

There are several useful options available to you while the Flex preview is active. These options are available by right-clicking the centerpoint of the Triad.

- **Reset Flex**

Removes the bending, twisting, tapering or stretching and resets the graphics to the pre-drag state.

- **Align Trim Plane Axis to Selection**

Aligns the trim plane normal to selected geometry: normal to a planar face, parallel to a line or through a point and the origin.

- **Align Bend Axis to Selection**

Aligns the bend axis to selected geometry: normal to a planar face or parallel to a line.

- **Center and Align to Component**

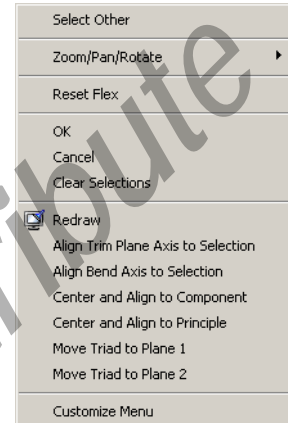
Move the triad to the centroid position and align the axes to the global axes.

- **Center and Align to Principle**

Move the triad to the centroid position and align the axes to the principal axes.

- **Move Triad to Plane 1 or 2**

Move the triad to align with the Trim Plane 1 or 2 position to place the bend axis on the trim plane.



Note

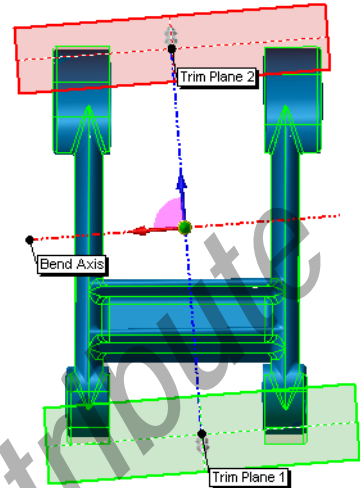
The Triad can also be dragged and dropped onto faces and edges to change its orientation. It can also assume the orientation and position of an existing coordinate system.

4 Reset.

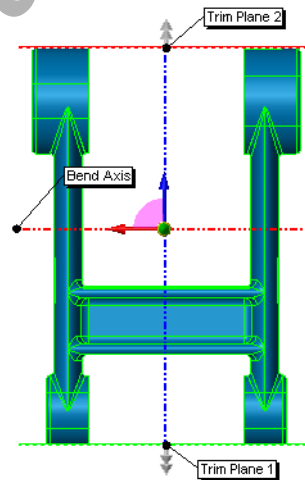
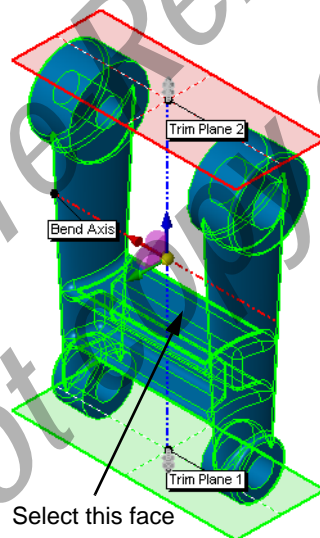
Right-click the Triad and choose **Reset Flex** to remove the bending created by the drag.

5 Alignment.

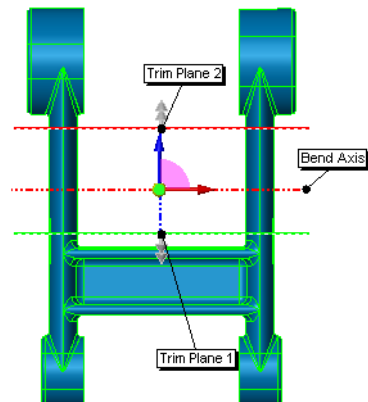
Change the view orientation to Front. A closer look at the trim planes and axes shows that they are not aligned to the principal axes of the part. The next steps will set the alignment and reposition the trim planes.

**6 Align Trim Planes.**

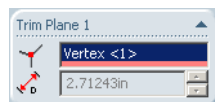
Right-click the Triad and choose **Align Trim Plane Axis to Selection**. Select the planar face as shown.

**7 Move Trim Planes.**

Drag Trim Planes 1 and 2 to the approximate positions shown using the handles.

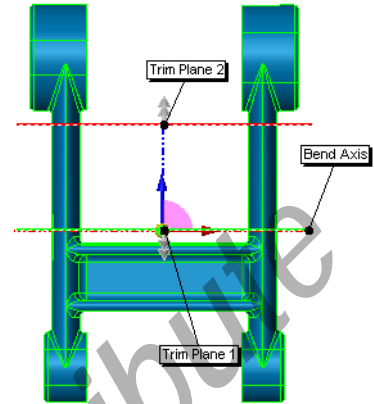
**Tip**

The trim planes can also be located to a specific location by selecting a vertex or endpoint, or by entering a **Trimming Distance** value.



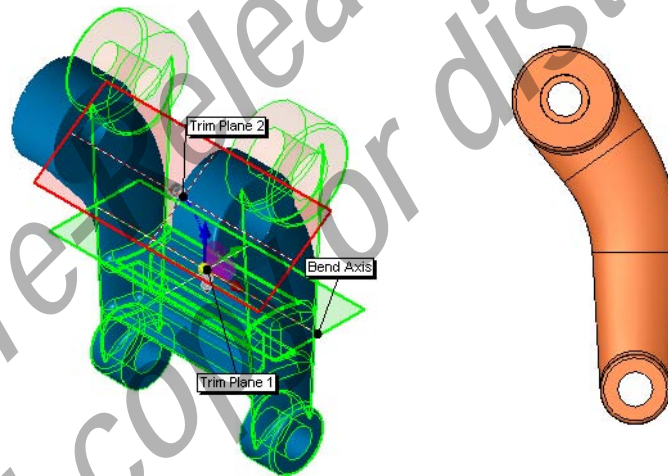
8 Move Triad.

Right-click the Triad and select **Move Triad to Plane 1**. The Triad is aligned to the trim plane position.



9 Bending.

Drag Trim Plane 2 to bend the model.



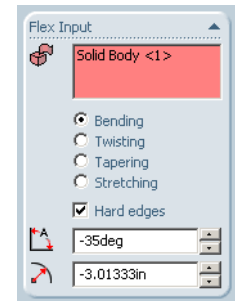
Note

Because the bend axis is on the trim plane, the portion of the model below the trim plane remains fixed.

10 More precise control.

You can set either the **Angle** or **Radius** option to achieve more precise control over the bend.

Click **OK** to create the flex feature.



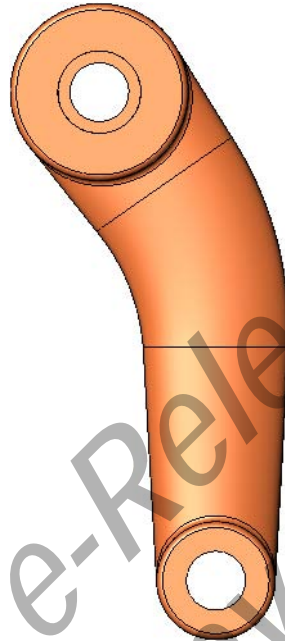
Controlling Direction

This command does not have a **Reverse Direction** button like other commands do. Instead, enter a negative or positive value to control the flex direction.

11 Save and close the file.

Hard Edges

The **Hard Edges** option creates analytical surfaces (cones, cylinders, planes, and so on) whenever possible, and often results in split faces where the trim planes intersect the bodies. If this option is cleared, the results are spline-based, so surfaces and faces may appear smoother and original faces remain intact.



Hard Edges Selected



Hard Edges Cleared

Pre-Release
Do not copy or distribute

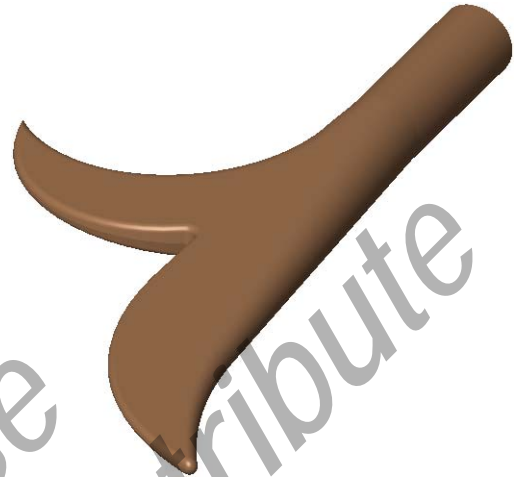
Exercise 16: Poker

Create this part using the dimensions provided. Use relations and equations where applicable to maintain the design intent.

This lab uses the following skills:

- Sketching
- **Up To Next** extrusion
- Face and Edge Fillets
- Mirror All

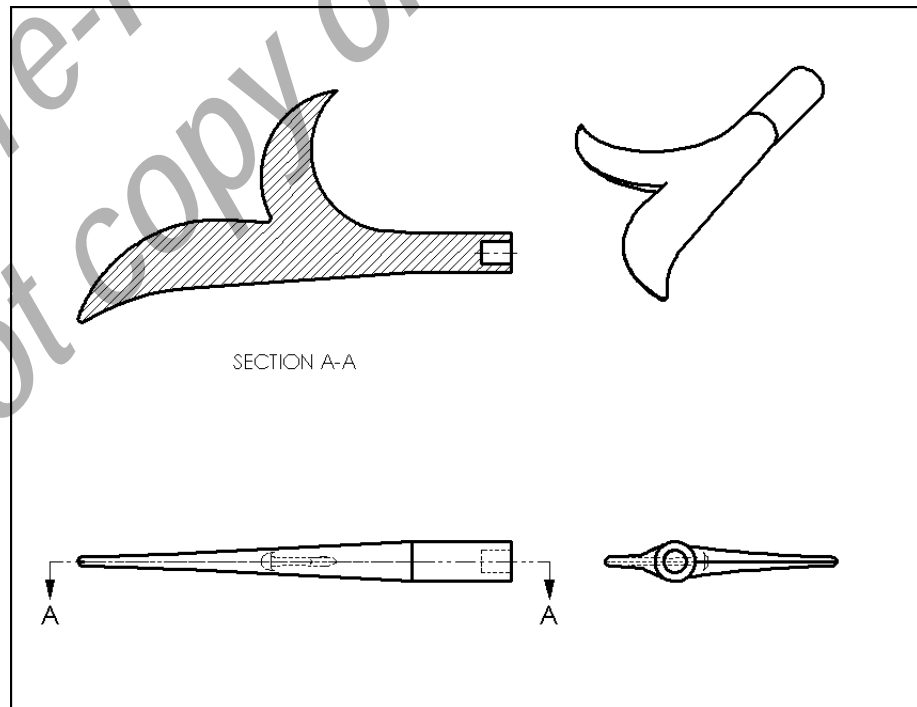
Units: **millimeters**



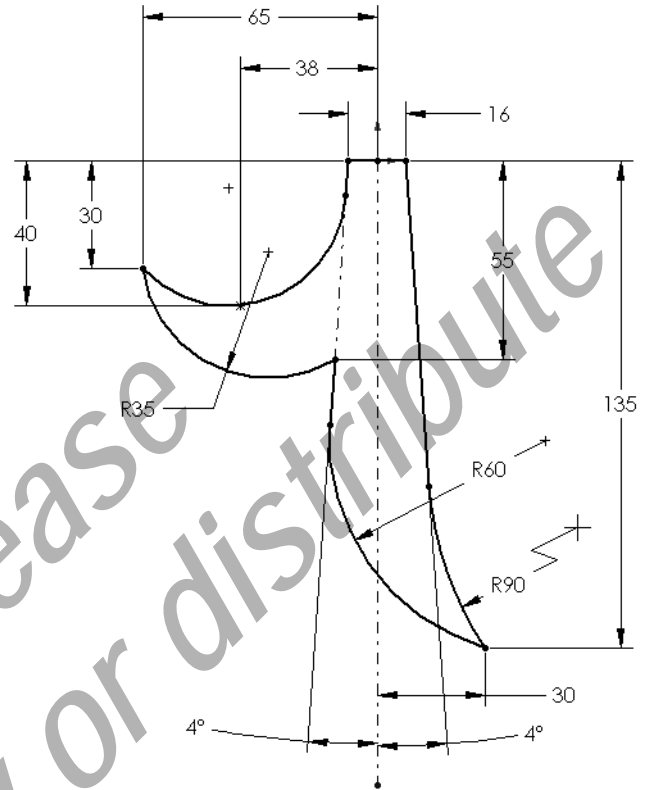
Design Intent

The design intent for this part is as follows:

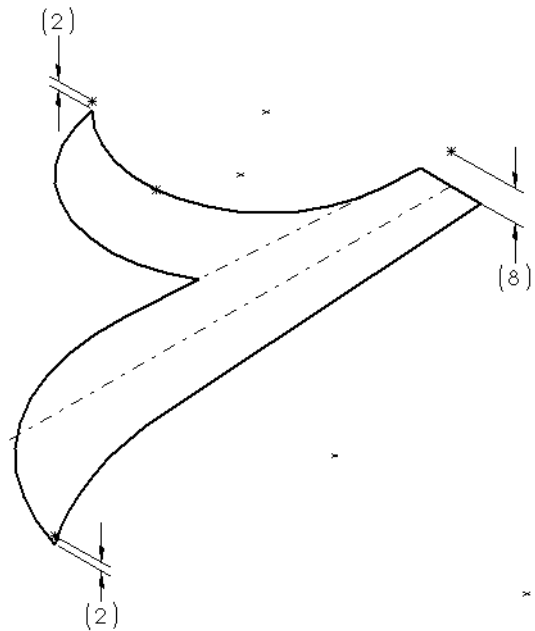
1. The part is symmetrical.
2. Circular hole is on the centerline.



- 1 **Open part.**
Open the part
Poker.sldprt.



- 2 **Points for the plane.**
Create three points located above the sketch positions as shown (these are reference dimensions only). Create the required planes and use one sketch for each point.

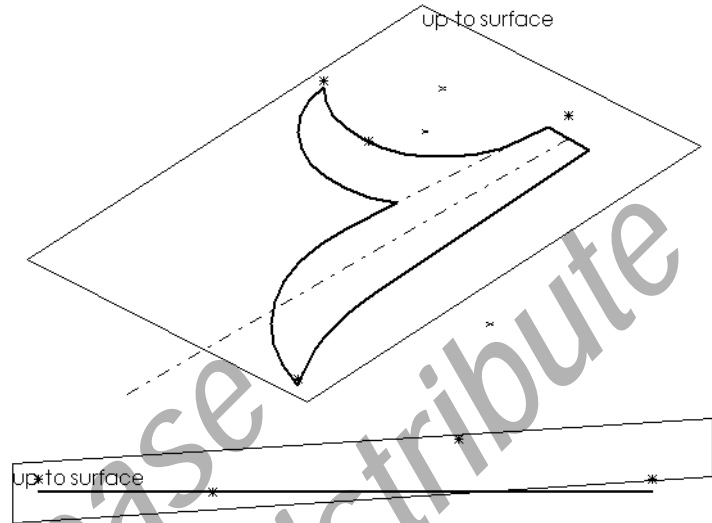


Question

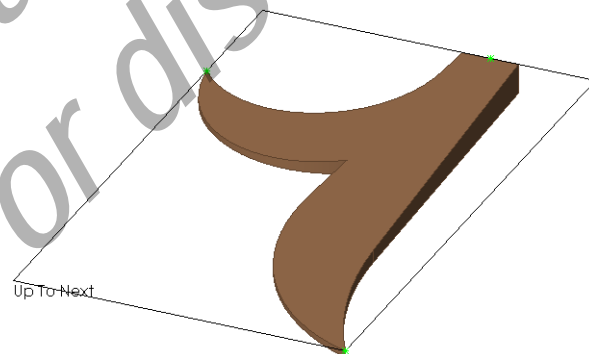
Instead of creating reference planes and individual sketches, can you create one 3D sketch containing all the necessary points?

- 3 Up to surface plane.**
Using the three points, create a plane that passes through them.

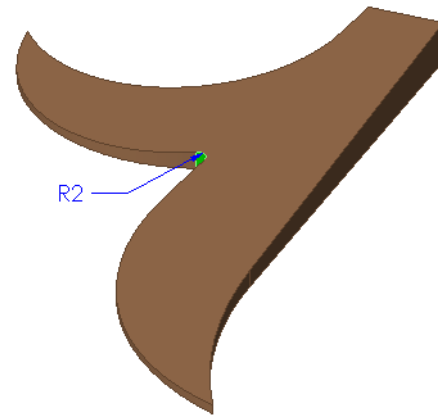
The Isometric and Front views are illustrated.



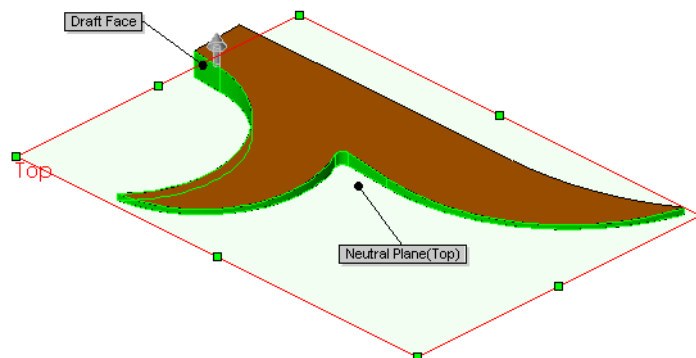
- 4 Extrusion.**
Extrude the sketch **Up To Surface**, selecting the plane as the surface. The extrusion is terminated by the plane.



- 5 Fillet.**
Add a fillet, radius **2mm**, on the inner edge. This fillet could also have been added in the profile.



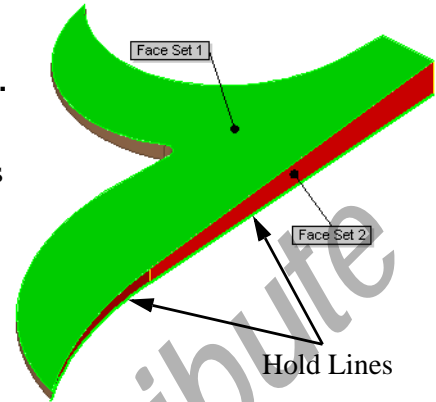
- 6 Draft.**
Add draft to selected faces using the Top reference plane as the neutral plane. Use **7°** of draft. Note the arrow indicating the pull direction.



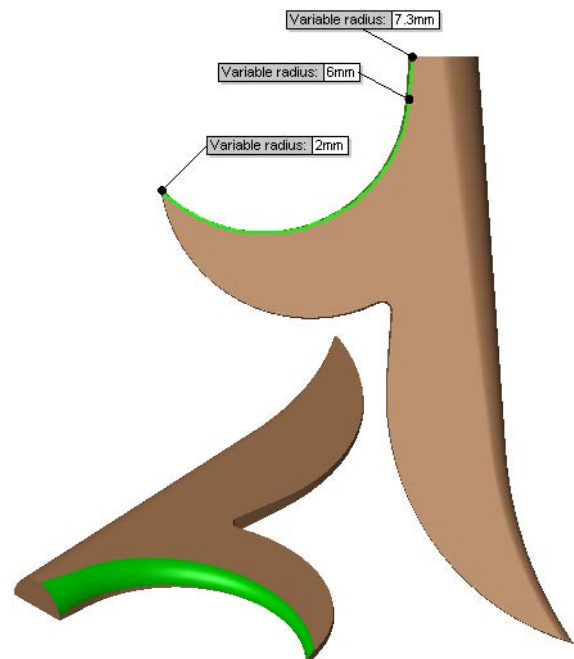
7 Face fillet.

Using faces of the model, create a **Face Fillet** that includes **Hold Lines**.

Select the faces in two sets as shown in the illustration. Use the outer edges to stop the fillets.

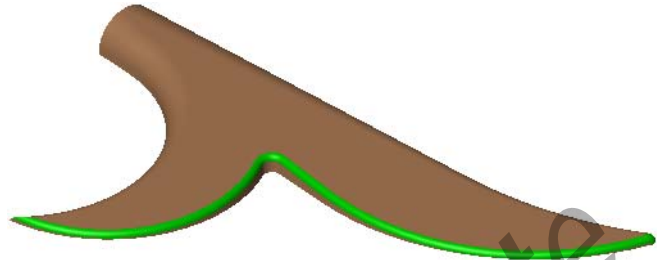
**8 Variable radius fillet.**

Select the two inner edges and use a variable radius fillet. Vary the fillet from **7.3mm** to **2mm** along the selected edges.



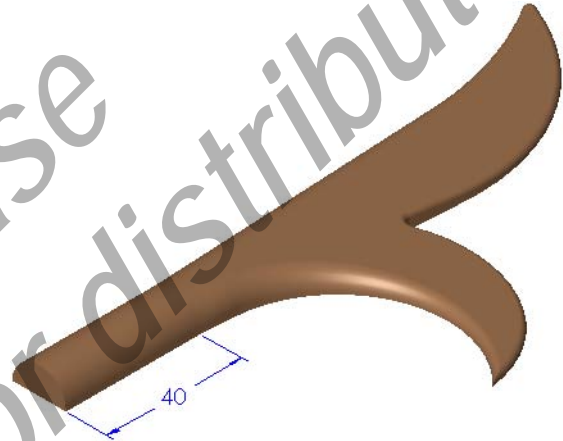
9 Edge fillet.

Add **1.5mm** fillet along the edges indicated in the illustration.



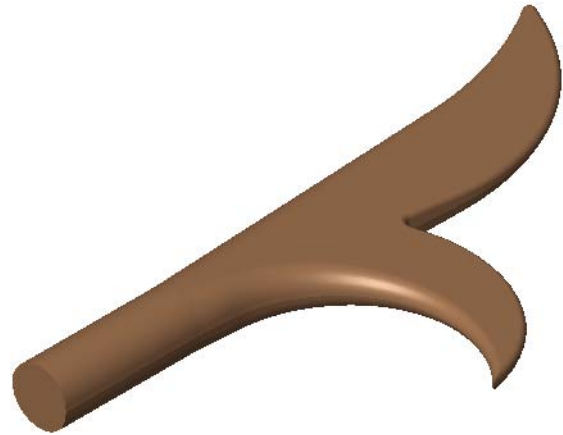
10 Extend the handle.

Using the flat face as a sketch plane, copy the edges and extrude them **40mm** to extend the handle.



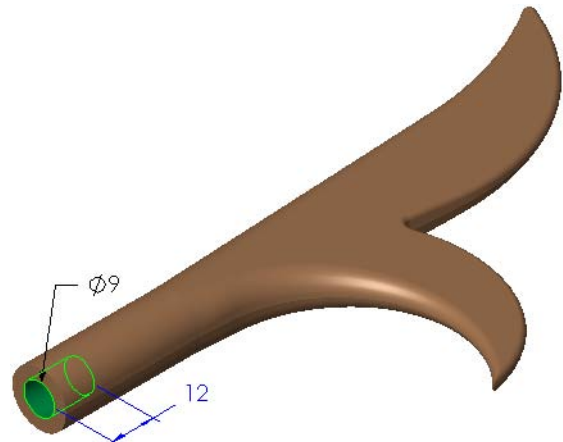
11 Mirror body.

Use **Mirror** to create the other half of the part.



12 Cut.

Create a cut diameter **9mm**, **12mm** deep to complete the model.



13 Save and close the part.

Exercise 17: Derived Sketch

Create this part using the dimensions provided. Use relations and equations where applicable to maintain the design intent.

This lab uses the following skills:

- Derived Sketch
- MidPlane Extrusion

Units: **millimeters**



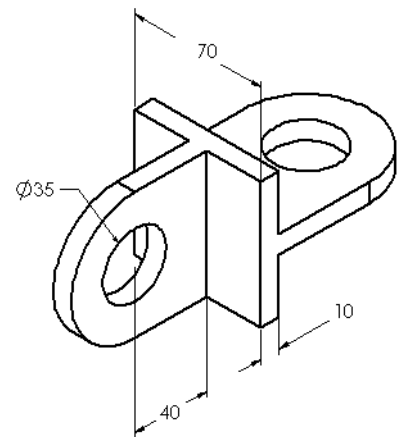
Design Intent

The design intent for this part is as follows:

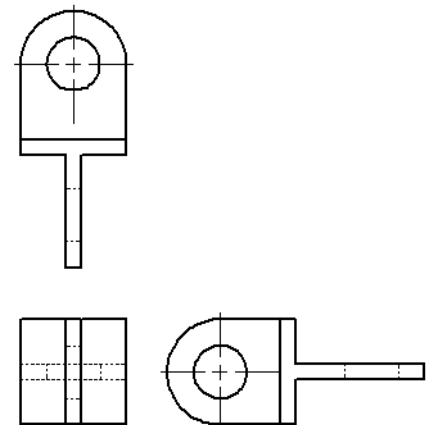
1. All material thickness for flanges is equal to that of the square plate.
2. Part is symmetrical.
3. Round holes are equal diameter and placement.
4. All fillets and rounds **3mm**.

Dimensioned Views

Use the following graphics with the design intent to create the part.



Three views.



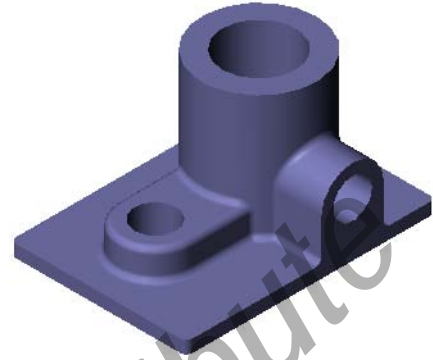
**Exercise 18:
Copy Sketch**

Create this part using the dimensions provided. Use relations and equations where applicable to maintain the design intent.

This lab uses the following skills:

- Up To Next Extrusion
- Copy Sketch for similar features

Units: **millimeters**



Design Intent

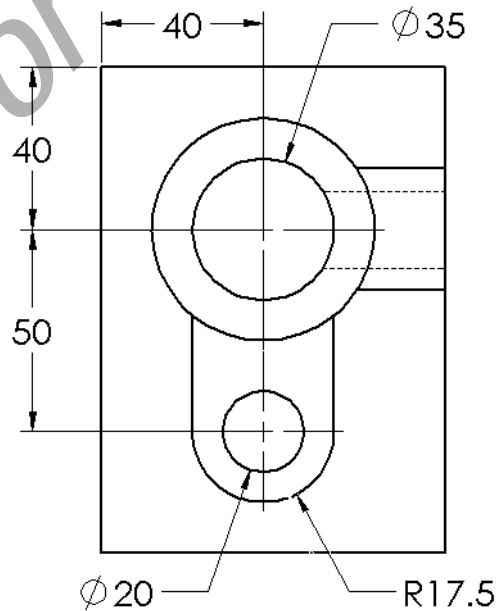
The design intent for this part is as follows:

1. Part is *not* symmetrical.
2. Vertical holes are through all.
3. All fillets and rounds **3mm**.

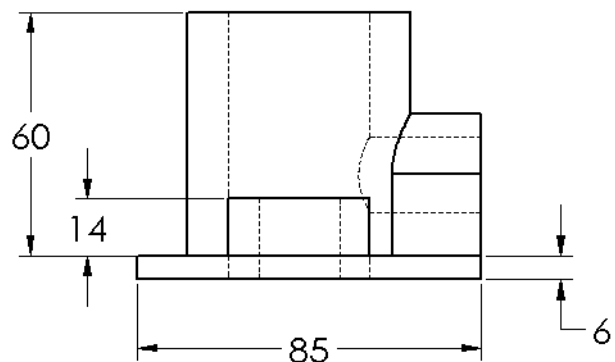
Dimensioned Views

Use the following graphics with the design intent to create the part.

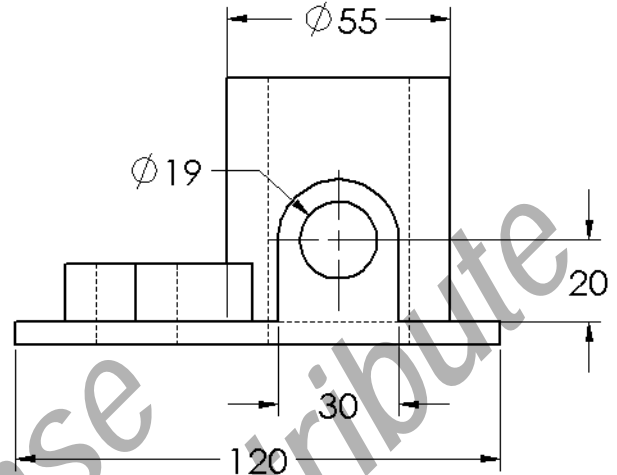
Top view.



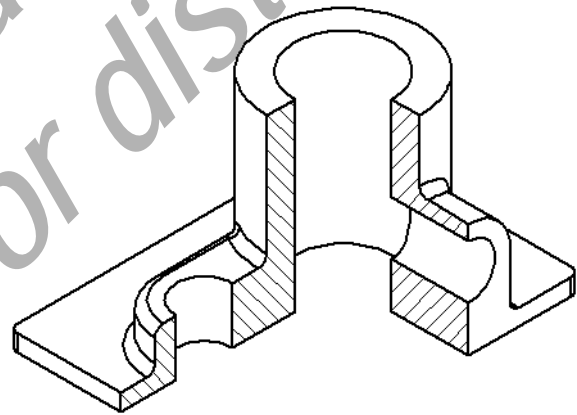
Front view.



Right view



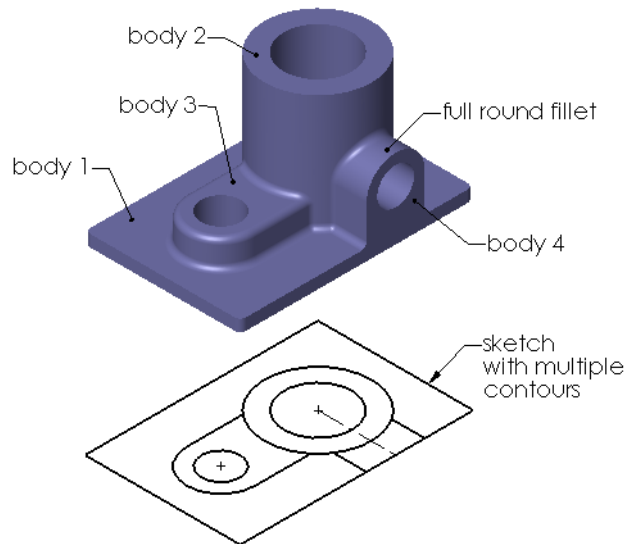
Cutaway view



Optional

Build this part using a different approach:

1. Use multibodies.
2. Use selected contours.
3. Add a full round fillet.



**Exercise 19:
Funnel**

Create this part using the information and dimensions provided. This lab reinforces the following skills:

- Lofting
- Shelling
- Sweeping

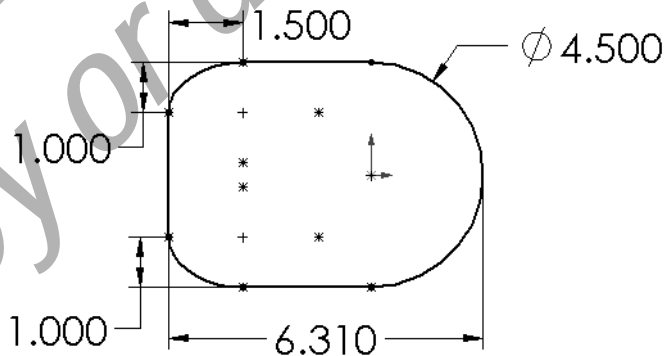


Procedure

Open a new part using the Part_IN template and name it Funnel.

1 Sketch the first profile.

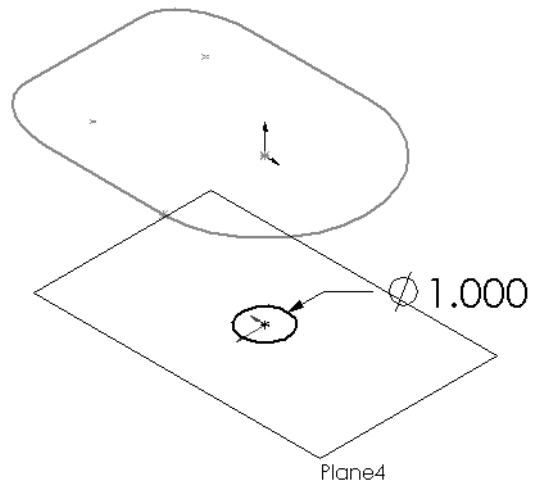
Use ellipses, lines and arcs to create this profile.



2 Second profile.

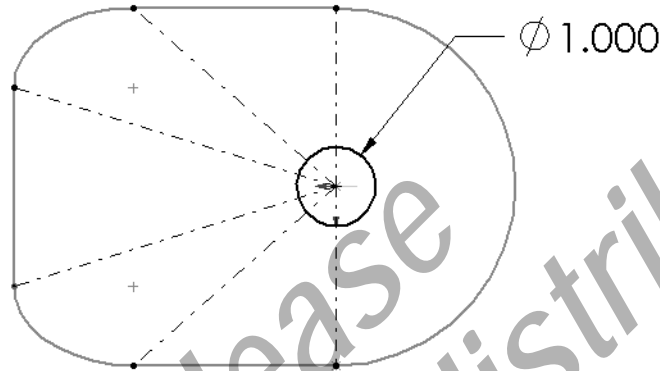
Create a new plane that is parallel to the Top reference plane **3.25"** below it. Sketch a circle lined up with the Origin.


This circle will be used as the second profile in a loft, after it is divided up into sections that match the first profiles endpoints. If the circle is not broken up, the loft will decide what the breakup of the circle should be.

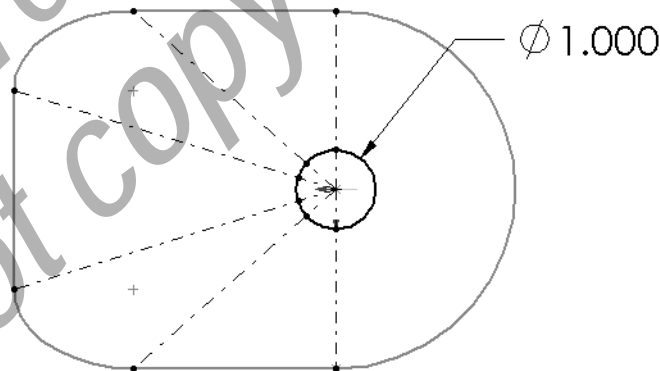


3 Breakup.

Add centerlines radially from the circle's center to the endpoints of the first profile. This geometry will cross the circle's circumference at several places.

**4 Divide circle.**

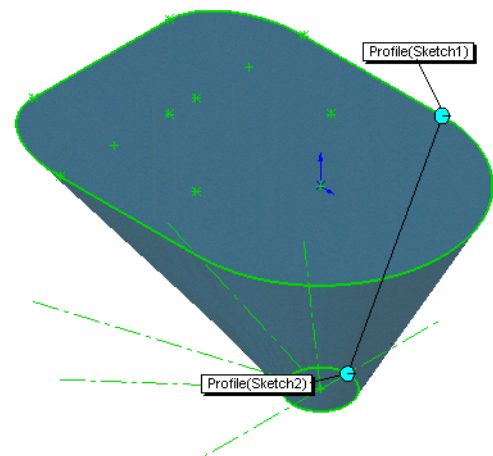
Using the **Split Entities**  command, add six split points, breaking the arc into pieces. Make each split point coincident with a centerline. You can add **Coincident** relations or you can drag and drop them onto the centerlines.

**5 First loft.**

Exit the sketch and loft between the two profiles. Select two endpoints that will match position, one from each sketch. This will ensure that the "start point" of the loft will be positioned correctly.

The option **Merge Tangent Faces** should be used.

An extra callout was added to the illustration for clarity.

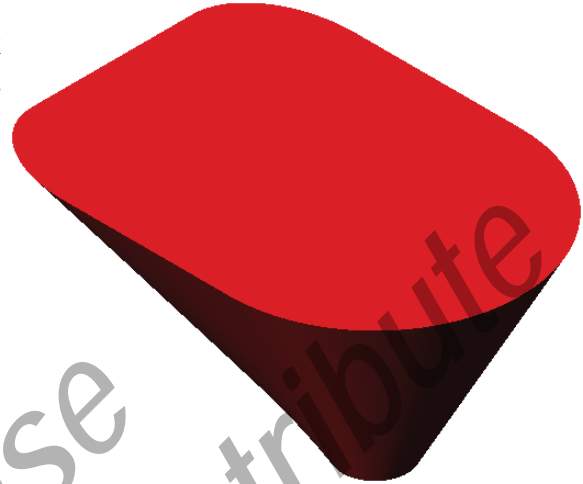


Important!

Note

6 Resulting loft.

The loft solid should look like this when completed.



7 Initial neck sketch.

The funnel neck is formed by another loft, this time using two sketched circles. Flip the model over and sketch a circle on the end face, making it **Conradial** with the circular, outer edge.

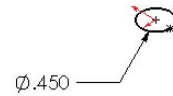
Add a point related to the Origin with a **Vertical** relation on the edge.



8 Neck end sketch.

Create a new reference plane offset from the circular face by **2"**.

Sketch a circle that is lined up with the Origin. Add a point on the circumference of the circle that is related to the Origin with a **Vertical** relation. The points are used to "line up" the profiles just as the centerlines were used in the first loft.



9 Neck loft.

Using the point entities to select the sketches, loft between the profiles.

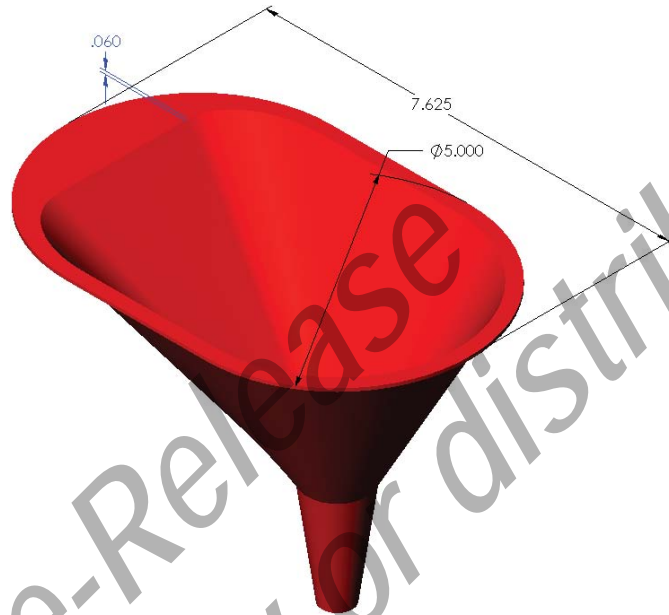
**10 Shell the funnel.**

The dimensions are given for the *inside* of the funnel. Create a thin walled part by shelling to the *outside*, a thickness of **0.06"**.



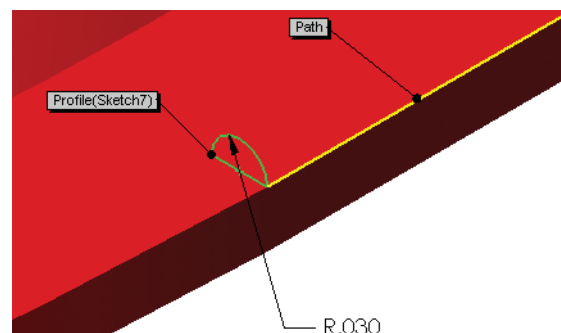
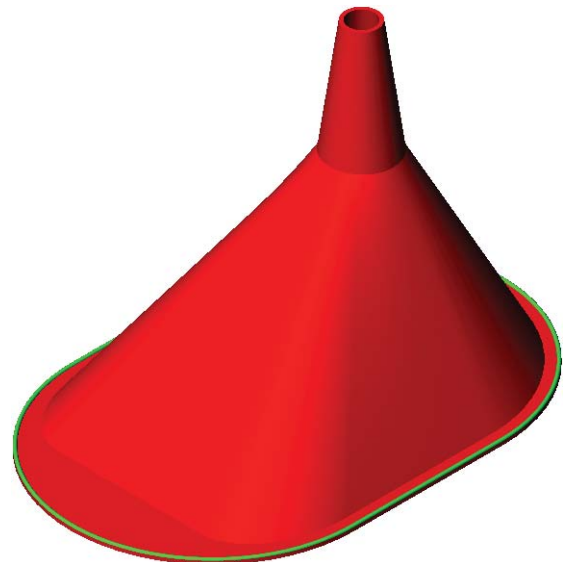
11 Build the rim.

Sketch the outline of the rim using the dimensions given. Use **Convert Entities** to create the inner outline. Extrude the rim to a depth of **0.06"**. If desired, use **Link Values** to tie the two thickness values together.



12 Sweep a lip on the underside of the rim.

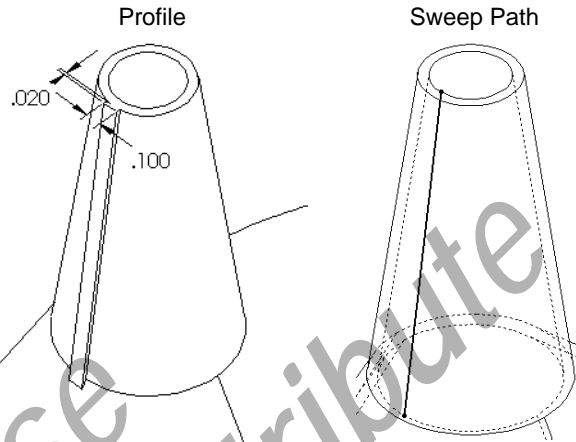
The cross-section of the lip is a semi-circle, **0.060"** in diameter. Use the model edge of the rim as the sweep path.



13 Make a rib on the neck of the funnel.

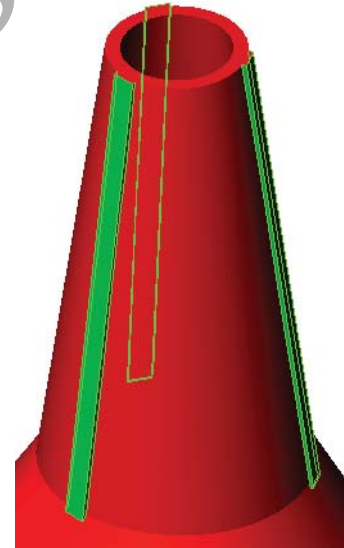
Funnels don't work well if air can't get out of the bottle. Sweep the section along a curve that lies on the inner face of the funnel neck.

An easy way to construct this curve is to sketch a line and constrain it with **Pierce** relations to model edges at the opening and where the *inside* of the neck meets the main body.



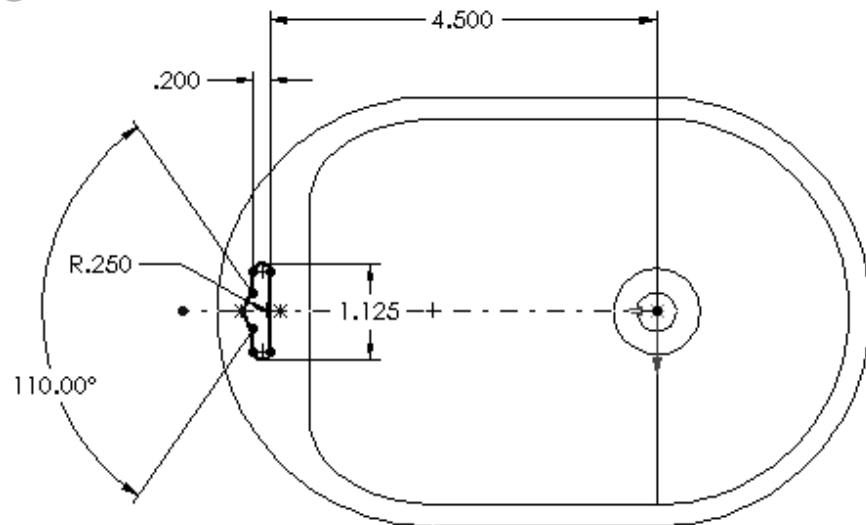
14 Pattern the rib.

Make a total of three ribs, equally spaced, using a circular pattern.

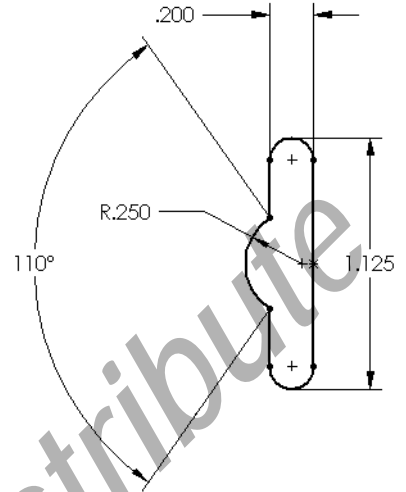


15 A hole in the rim.

Using the dimensions provided, sketch a profile to cut through the rim so the funnel can be hung on a hook. Notice the use of an angular dimension on an arc. This can be created by picking the arc's centerpoint and its two ends.



Detail of the hole.



16 Save and close the part.



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Do not copy or distribute

Lesson 4 Surface Modeling

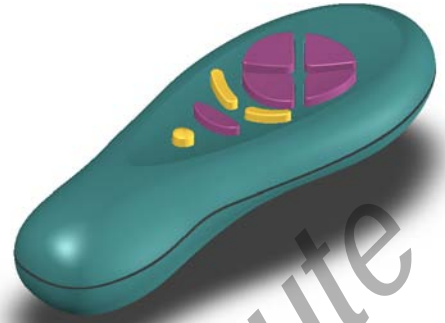
Upon successful completion of this lesson, you will be able to:

- Create extruded, ruled, lofted, and planar surfaces.
- Modify surfaces by trimming.
- Create filled surfaces for blending.
- Convert surfaces into solids.
- Use surface intersections to create 3D curves.
- Create surfaces to fill gaps in imported models.
- Delete and patch model faces.

Pre-Release
Do not copy or distribute

Working with Surfaces

There are a number of different situations when it is necessary to work with surfaces. One is when you import data from another CAD system and the result is a collection of surfaces, not a solid model. Another situation is when the shape you want to create is best modeled using free-form surfaces that are then knit together to form a solid. In this case study we will explore using surfaces to model a shape – a remote control – that would be difficult to model using only solid modeling techniques.



What are Surfaces?

The outer skin of a solid model is made up of surfaces. Surfaces are what define the shape of the faces of a solid – whether they are flat or curved. The difference between a surface model and a solid model is one of intelligence and completeness. Solid models are always closed. There are not any gaps or overlapping edges. Surface models can be open. Multiple surfaces may not meet along their edges. They might overlap or fall short.

Solid models are intelligent. The system knows what space lies “inside” the solid and what lies “outside”. Surface models lack that intelligence. You might consider a surface to be the ultimate “thin feature”. It has a shape, but no thickness. When multiple surfaces are put together so that the edges all meet and there are no gaps, the result can be “filled”, transforming it into a solid.

Stages in the Process

Some of the key stages in the modeling process of this part are given in the following list:

- **Capture the design intent.**

The industrial designer provided concept sketches of the remote control. These were scanned to create image files that can be inserted into a sketch. The sketch pictures will serve as a guide when modeling the remote control.

- **Parting lines and draft angles.**

As a general rule you should begin modeling by defining the parting line and setting up the draft angles using reference surfaces. With the vast majority of free-form parts, you must build draft in as you model. Generally you cannot add draft later as a local feature.

- **Splines.**
Consumer products are characterized by smooth, curvature continuous shapes that cannot be modeled using lines and arcs. Splines are the curves that in turn create the surfaces.
- **Lofted and swept surfaces.**
One portion of the remote control will be lofted using a series of profiles and guides. Another portion will be swept using guide curves.
- **Blending surfaces to fill in gaps.**
Not all the necessary surfaces can be created using loft or sweep. The remaining portion will be created as a filled surface.
- **Knitting.**
Once the surface model is complete, the surfaces are knitted into a solid.
- **Symmetry.**
The knitted solid is mirrored.
- **Associativity and design changes.**
After evaluating the model, we will change the underlying curves.
- **Breaking the master model into separate parts.**
From the master model we will break out individual piece parts.
- **Specialized features for plastic parts.**
Mounting bosses, snap hooks, and snap hook grooves can be built easily using specialized fastening features.

Surfaces Toolbar

The Surfaces toolbar contains shortcuts for all the surfacing commands. These commands can also be accessed via the **Insert, Surface** menu.



Using Sketch Picture to Capture Design Intent

We will start the modeling process with a couple of sketches of the design concept provided by the industrial designer. These will be used as guides as we create the basic curves.

Where to Find It

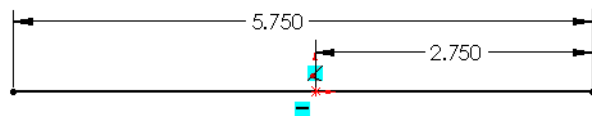
- Click **Tools, Sketch Tools, Sketch Picture**.
- Click **Sketch Picture**  on the Sketch toolbar.

Procedure

Begin by opening a new part with units set to inches.

1 Side view sketch.

Open a sketch on the Right reference plane.



Sketch a horizontal line as shown. This reference line will be used to in subsequent operations.

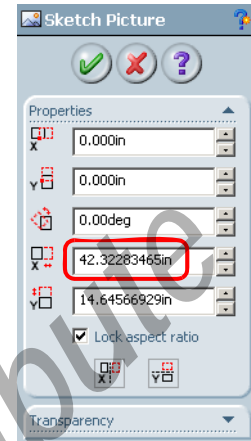
2 Sketch picture.

Click **Tools, Sketch Tools, Sketch Picture**.

In the Case Study folder for this lesson, browse to the Remote Control \Sketches from ID folder.

Select the image Remote-side-view.tif and click **Open**.

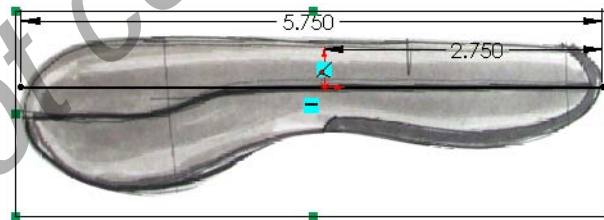
The picture will come in very large. Note that the **Width** is over 42 inches.

**3 Resize the picture.**

Make sure **Lock aspect ratio** is selected and scale the image to approximately the correct size by setting the **Width** to **5.75in**.



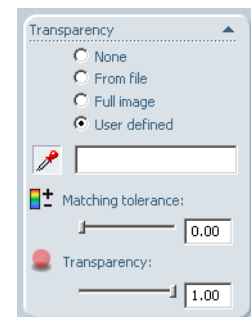
Fine tune the position of the picture by dragging and resizing it. The objective is to line the picture up with the sketched reference line.

**4 Transparency.**

Expand the **Transparency** options. Select **User defined** and click the white background area of the picture to define the transparent color.

Set the **Transparency** slider to **1.00**.

Click **OK**.



5 Top view sketch.

This one will also come in large. And it is rotated.

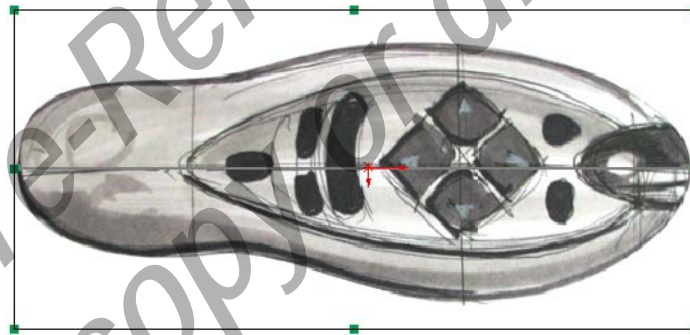
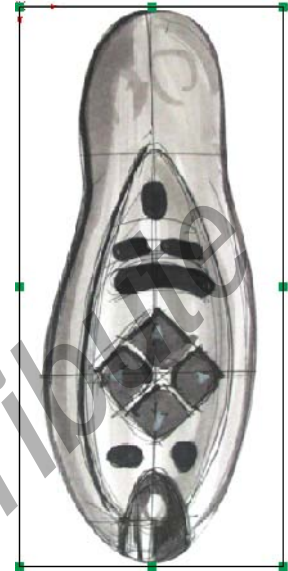
Rotate the image by setting the **Angle** to **90°**.

Make sure **Lock aspect ratio** is selected and scale the image to approximately the correct size by setting the **Width** to **5.75in**.

Fine tune the position of the picture by dragging and resizing it.


Line it up with the reference line in the first sketch.

Set the **Transparency** to **1.00** and select the white background of the picture as the transparent color.

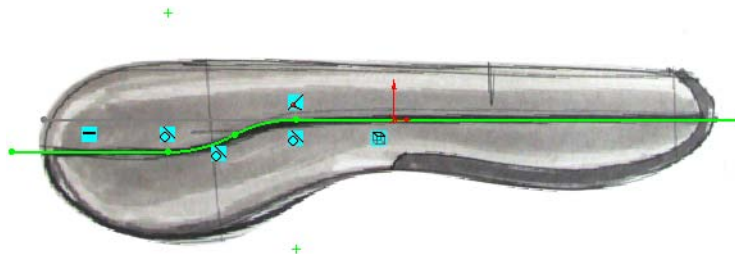


6 Sketch the parting line.

Open a new sketch on the **Right** reference plane.

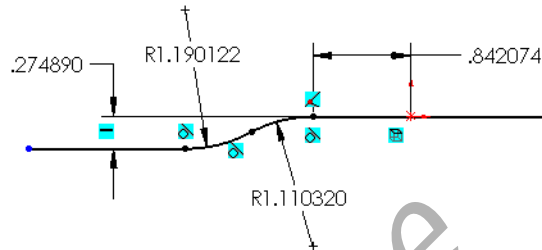
Use **Convert Entities**  to copy the reference line from **Sketch1** into the active sketch.

Using tangent arcs and lines, sketch the parting line shown here in green for clarity.



7 Dimension the sketch.

Sketch1 is hidden for clarity. Do not worry about the values of the dimensions. Your values may vary. The goal right now is to constrain the sketch.

**Note**

The dimensions are shown in 6 decimal places just to illustrate that we are not worrying about the exact dimension values at this time.


**Introducing:
Fit Spline**

Fit Spline creates a spline that follows, or fits, sketch segments within a tolerance that you specify. Fit splines are parametrically linked to underlying geometry so that changes to the geometry update the spline.

Where to Find It

- Click **Tools, Spline Tools, Fit Spline**.
- Click **Fit Spline**  on the Spline Tools toolbar.


8 Fit spline.

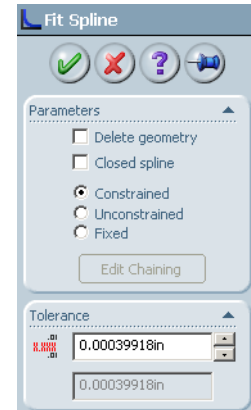
Click **Fit Spline**  on the Spline Tools toolbar.

Clear the **Closed spline** check box.

Right-click the line and select **Select Chain**.

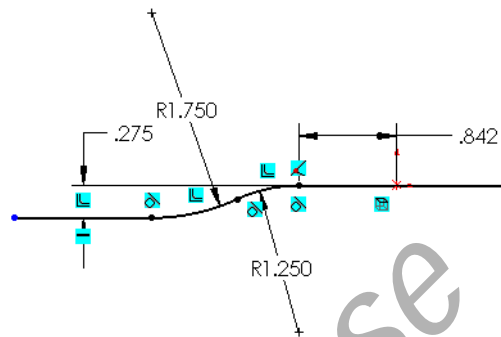
The system creates a spline and converts the original sketch entities to construction geometry.

The spline is related to the original sketch entities by a **FitSpline** relation as indicated by the  symbol.



9 Change the dimensions.

Fine tune the parting line geometry by editing the dimension values as shown below. Notice that the spline updates accordingly.



Similarities Between Solid and Surface Modeling

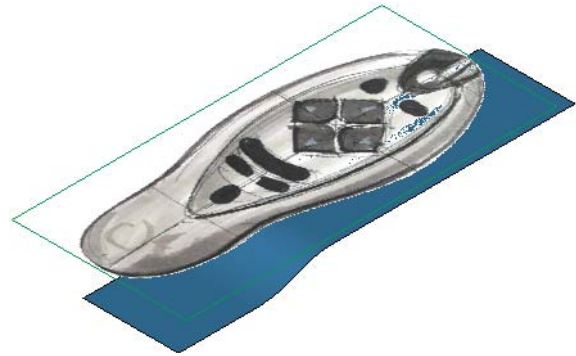
While surface modeling has many unique and specialized commands, a number of surface commands are identical to their solid modeling counterparts. For example:

- Insert, Boss, Extrude = Insert, Surface, Extrude
- Insert, Boss, Revolve = Insert, Surface, Revolve
- Insert, Boss Sweep = Insert, Surface, Sweep
- Insert, Boss, Loft = Insert, Surface, Loft

10 Extrude a surface.

Extrude the parting line sketch so that it extends beyond what will be the edge of the model. A distance of **1.5"** works well.

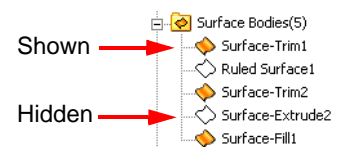
It is only necessary to extrude in a single direction because we are going to take advantage of the part's symmetry and use mirroring.



Introducing: Hide/Show Surface Body

When you model using surfaces, it is not uncommon for you to create extra surfaces as reference geometry or construction aids; surfaces that are not part of the finished model but rather are a means to an end.

These surfaces are grouped in the **Surface Bodies** folder. These surfaces can sometimes get in the way. **Hide/Show Surface Body** is used to control the visibility of these surfaces.



Where to Find It

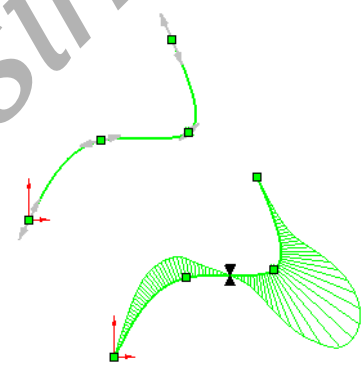
- Right-click on surface in the graphics area, and select **Body, Hide**.
- Expand the **Surface Bodies** folder in the FeatureManager design tree. Right-click the surface and select either **Hide Surface Body** or **Show Surface Body**.
- Right-click the **Surface Bodies** folder to hide or show everything in the folder.

Splines

A spline is a curve that can have a locally very simple form, yet at the same time be globally flexible and smooth. Splines are very useful for modeling free-form shapes that are smooth and fair. [Fair is a term often used in boat building. A “fair curve” is one that is as smooth as it can be as it follows the path it must take around the hull of a boat; it is free of extraneous bumps or hollows.]

**Introducing:
Spline**

Splines are used to sketch curves that are not arcs, or conic sections such as ellipses or parabolas. Splines are defined by a series of interpolant points. Interpolant means that the curve passes through the points. You can modify a spline by adding or deleting points, moving the points, dimensioning the points or adding geometric relations. The spline can also be changed by modifying the spline handles (arrows) that control the tangency of the curve at the interpolant points.

**Where to Find It**

- Click **Spline**  on the Sketch toolbar.
- Or, click **Tools, Sketch Entities, Spline**.

11 Hide surface.

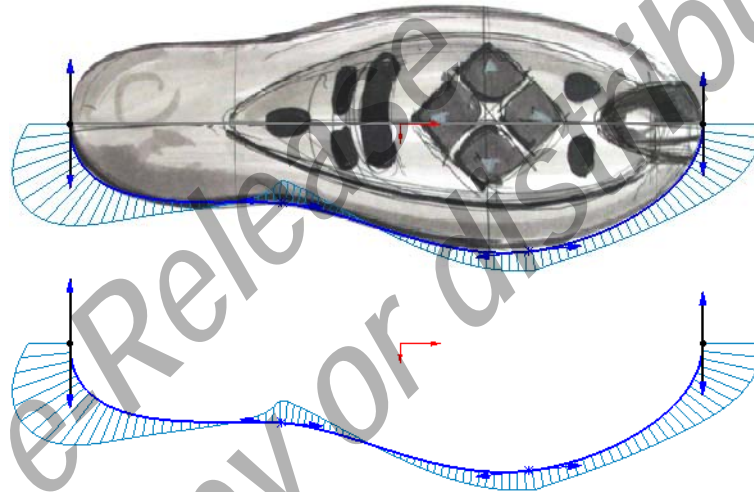
In the graphics area, right-click the extruded surface and select **Body, Hide** from the shortcut menu. This will make it easier to see what we are sketching in the next step.

12 Sketch a 4-point spline for top view of parting line.

Make both ends **Coincident** to the ends of the reference line in Sketch1.

Make the tangent handles at both ends **Perpendicular** to the reference line in Sketch1.

Turn on the curvature combs. Adjust the positions of the points and drag handles until you are satisfied with the spline and how it fits the sketch. When finished, exit the sketch.



Trimming Surfaces

**Introducing:
Trim Surface**

When you add features to a solid model, all the overlapping faces are automatically trimmed. When you work with a surface model, the trimming has to be done manually.

Surfaces can be trimmed to their intersection with other surfaces, the face of a solid, or reference planes. Additionally, you can select a sketch that will be projected onto the surface to create a trim boundary. The system highlights the various solutions to the trimming operation. You have the option to select what you want to *keep* or what you want to *remove*.

Where to Find It

- Click **Insert, Surface, Trim**.
- Or, click **Trim Surface**  on the Surfaces toolbar.

13 Trim the parting surface.

Click **Trim Surface** .

For **Trim Type**, click **Standard**.

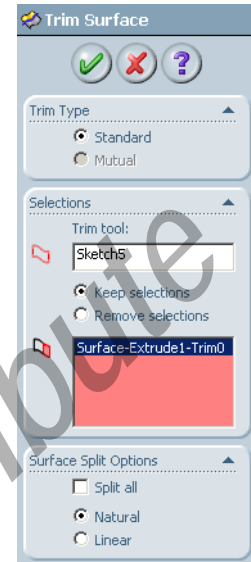
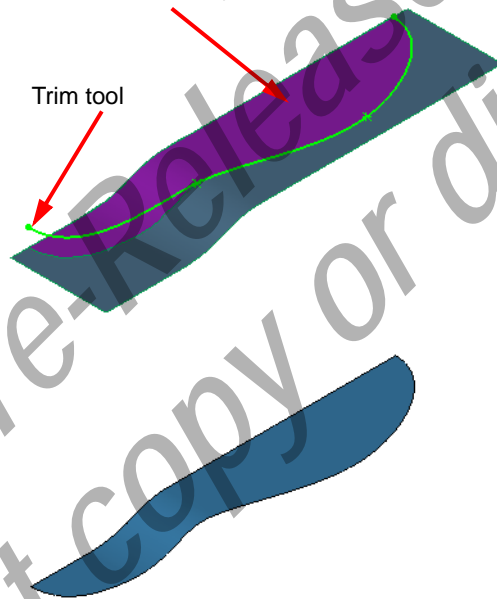
For the **Trim tool**, select the sketch we just created in step 12.

Click **Keep selections** and click in the selection list. Identify the portion of the parting surface that you want to *keep*.

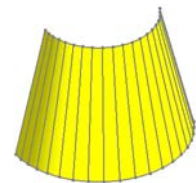
Click **OK** to complete the trimming operation.

Portion of surface you want to keep

Trim tool

**Ruled Surfaces**

In general, a **Ruled Surface** can be thought of as an infinite number of line segments connecting corresponding points on opposite sides of the surface. In the case of a SolidWorks ruled surface, one edge is defined by the edges or edges of existing surfaces. The other edge is calculated by the system based on the options you choose. Unlike other types of surfaces, for the **Ruled Surface**, you do not need to create sketches.

**Introducing:
Ruled Surface**

Use the **Ruled Surface** to create surfaces that are either perpendicular or tapered away from the selected edges.

Where to Find It

- Click **Ruled Surface**  on the Surfaces toolbar.
- Or click **Insert, Surfaces, Ruled Surface**.

14 Ruled surface.

In this case we want to create a reference surface that follows the edge of the parting surface and that has 3° of draft with respect to the Top reference plane. We will use this surface in subsequent steps to help define the geometry of the part.

For **Type**, select **Tapered to Vector**.

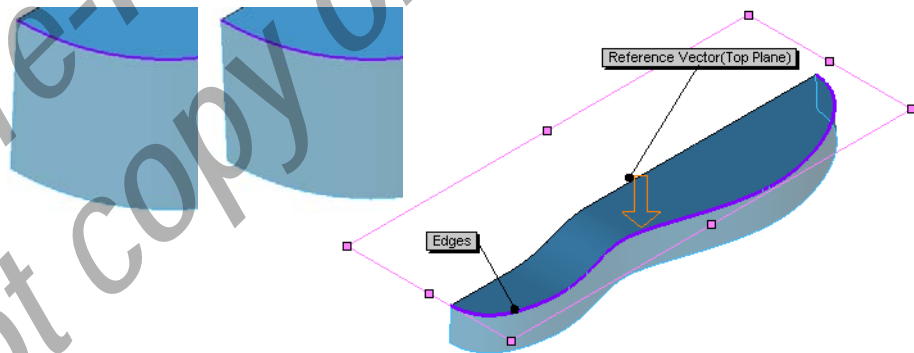
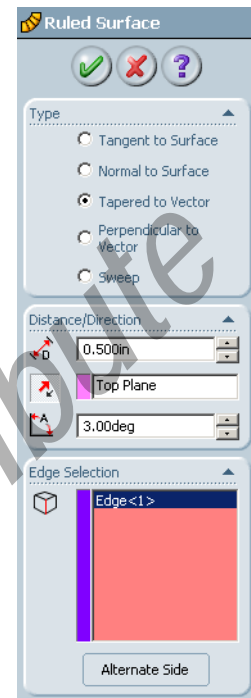
For **Distance** enter **0.5"**. The distance is not critical. We just need something big enough to work with easily.

For the **Reference Vector**, select the Top reference plane and click **Reverse Direction**.

Set the **Angle** to **3.00°**.

For **Edge Selection**, select the edge of the trimmed surface.

Verify that the ruled surface tapers *outward*. If it does not, click **Alternate Side**.

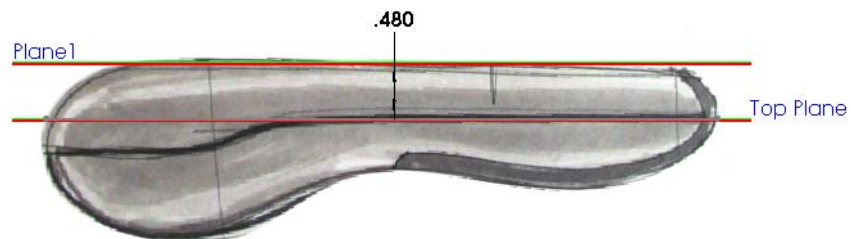


Click **OK**.

15 Offset plane.

Create a plane offset from the Top plane. This will be used for sketching the area around the keypad.

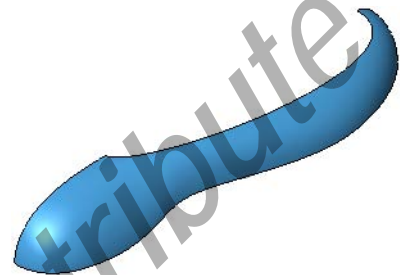
In this case, the offset was 0.480". Depending on how you scaled the sketch picture, your results may differ.



Note From the looks of the sketch picture, it appears the upper face of the remote control is angled with respect to the **Top** plane. However, we checked with the industrial designer and were told that the two should indeed be parallel.

Lofting Surfaces

The surface that will actually be part of the finished model is one half of the upper part of the housing. This will be a lofted surface and to create it, we need several profile and guide curves.

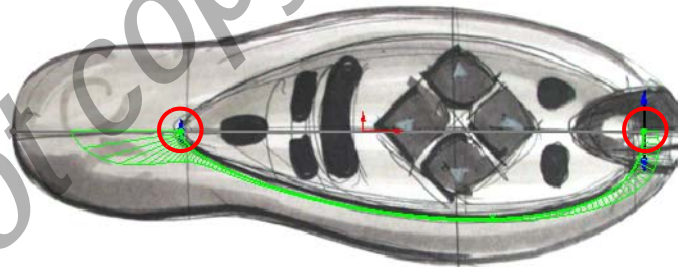


16 Sketch a 3-point spline for outline of the keypad area.

Make both ends **Coincident** to the ends of the reference line in Sketch1.

Make the handles at both ends **Perpendicular** to the reference line in Sketch1.

Turn on the curvature combs. Adjust the positions of the points and drag handles until you are satisfied with the spline and how it fits the sketch. When finished, exit the sketch. This will be the guide curve.



Note Since the spline is not dimensioned, it is under defined and appears blue in the sketch. It is shown here highlight in green for clarity.

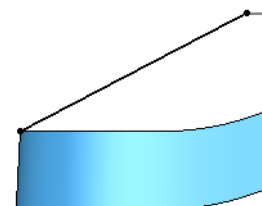
17 First profile curve.

Create a new sketch on the **Right** reference plane.

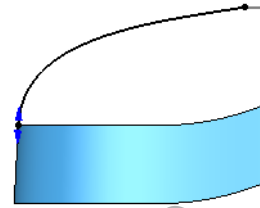
The profile is a 2-point spline. Creating this is a multistep process:

1. Sketch the spline. The ends are **Coincident** to the end of the guide curve (step 16) and the corner of the ruled surface.

Note: For clarity, the sketch picture is not shown.



2. Make the spline tangent to the edge of the ruled surface. This is necessary to maintain the 3° draft angle when we loft the surface.

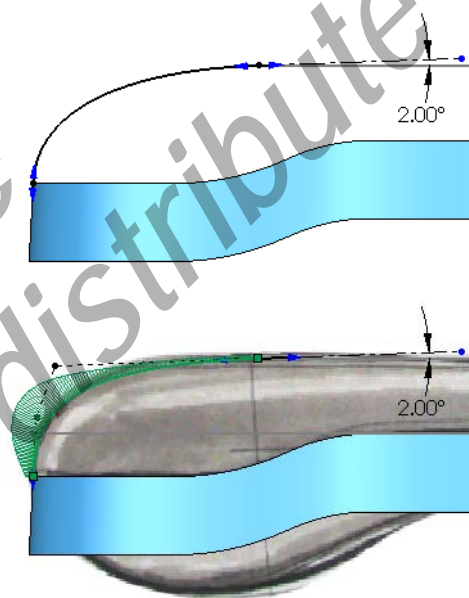


3. Sketch a construction line tangent to the other end of the spline. Create an angular dimension between it and the plane the guide curve is on (step 15).

Set the angle to **2.00°**.

4. Display the curvature combs and show the sketch picture.

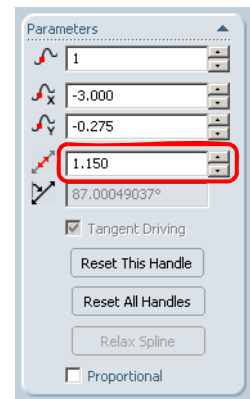
Adjust the lengths of the tangent handles until you are satisfied with the shape of the spline.



Tip

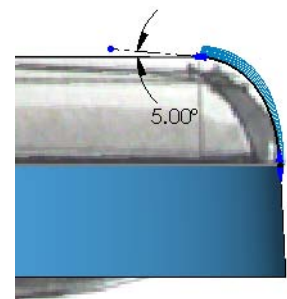
The PropertyManager is very useful for making small adjustments to the length of the tangent handles.

5. Exit the sketch.



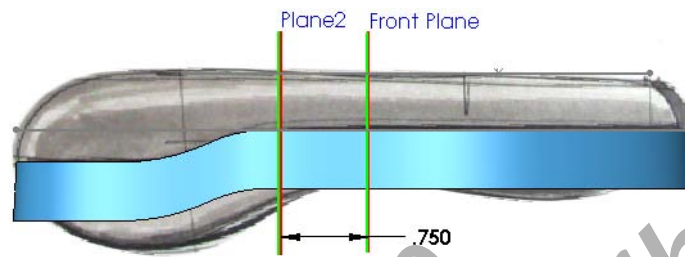
18 Second profile curve.

Repeat the preceding procedure for the profile curve on the front end of the remote control.



19 Offset plane.

Create a plane offset **0.75"** from the Front plane. This will be used for sketching a third profile curve.

**20 Third profile curve.**

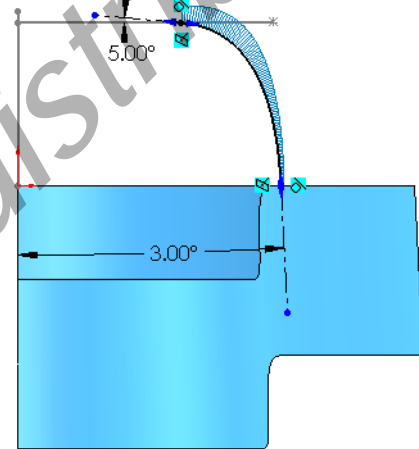
Create a new sketch on Plane2.

Switch to a Front view orientation.

Sketch a 2-point spline. Add **Pierce** relations between the ends and the guide curve and the edge of the ruled surface.

Sketch two construction lines tangent to the spline and dimension their angles as shown.

Display the curvature combs and adjust the lengths of the tangent handles until you are satisfied with the shape of the curve. In this case, the sketch pictures do not offer any guidance use your best judgement.

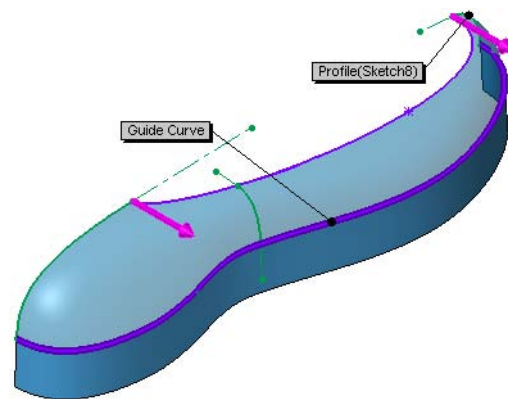
**21 Loft the surface.**

Select the three profile curves.

For **Start/End Constraints**, select **Normal To Profile** for both.

For **Guide Curves**, select Sketch6 (step 16) and the edge of the ruled surface.

For the edge tangency, select **Tangency to Face**. For Sketch6, select **None**. Click **OK**.

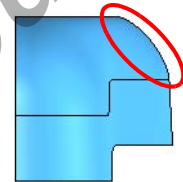
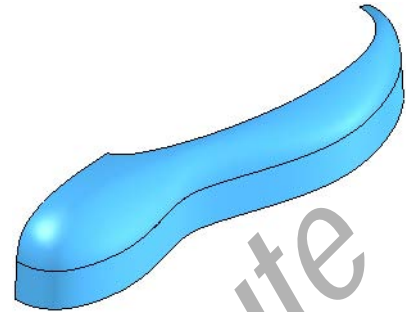


22 Evaluate the results.

Use any of the techniques that were introduced in *Analyzing Geometry* on page 87 to evaluate the resulting loft. These include **Display Curvature** and **Zebra Strips**.

Sometimes it is also helpful to add another directional light to give more illumination to the side of the model.

Looking at the Front view, the surface does not look rounded enough in the area indicated.

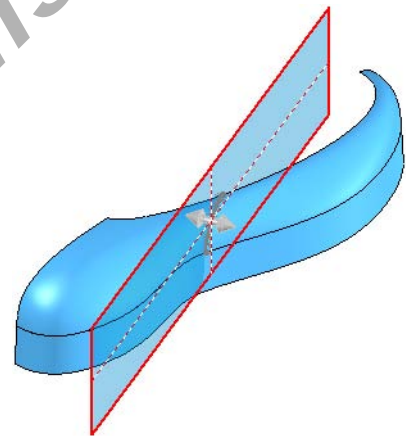


23 Add a loft section.

Right-click the lofted surface, and select **Add Loft Section** from the shortcut menu.

The system generates a section plane and a profile curve through the surface.

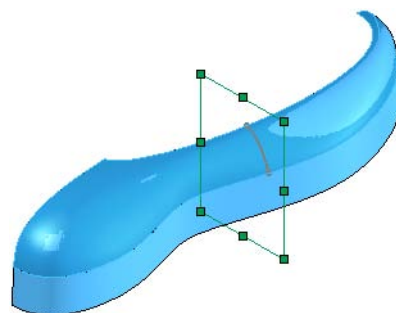
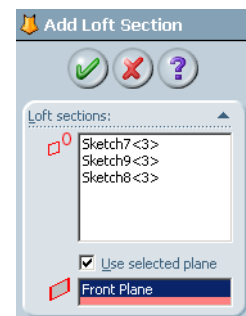
You can move and rotate the plane by dragging it.



24 Use selected plane.

In the PropertyManager, select the **Use selected plane** check box.

Select the Front reference plane and click **OK**.



25 Show sketch.

In the next step we will edit the new loft section. Before we do that, show the sketches for the second profile and the guide curve.

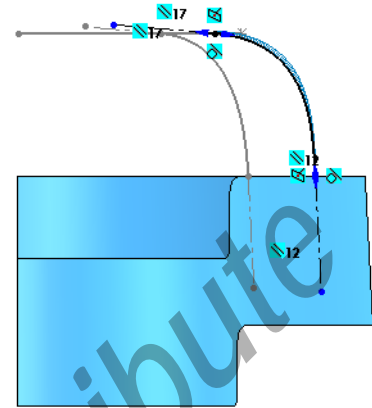
26 Edit the new loft section.

View the sketch relations. If there are not already **Pierce** relations between the ends and the guide curve and the edge of the ruled surface, add them.

Sketch construction lines tangent to each end of the spline. Add **Parallel** relations between them and the construction lines in the second profile.

Display the curvature combs and adjust the spline until you are satisfied with the shape.

Exit the sketch to rebuild the lofted surface.



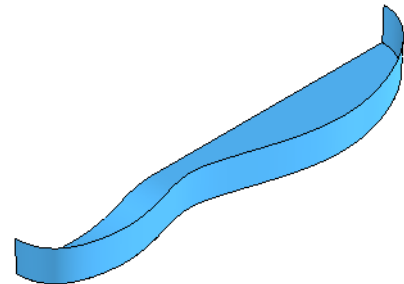
Modeling the Lower Half

We will use a similar approach modeling the lower half as we did for the upper half. Namely, we will use the sketch picture as a guide to help establish the shape of the part. However, instead of lofting, we will use Sweep with Guide Curves and Fill Surface.

27 Ruled surface.

Create a second ruled surface also with 3° of draft. This time, it should extend upwards from the edge of the parting surface.

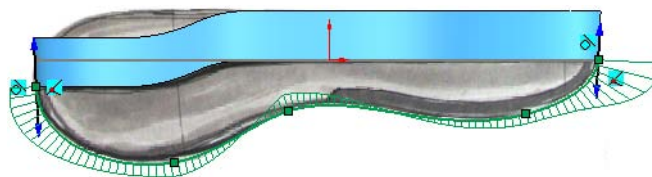
This will be used as a reference when modeling the lower half of the remote control.

**28 Spline.**

Open a new sketch on the Right reference plane. Show the side view sketch picture.

Create a 5-point spline. You need **Coincident** relations between the endpoints and the corners of the ruled surface. Add **Tangent** relations between the spline and the edges of the ruled surface.

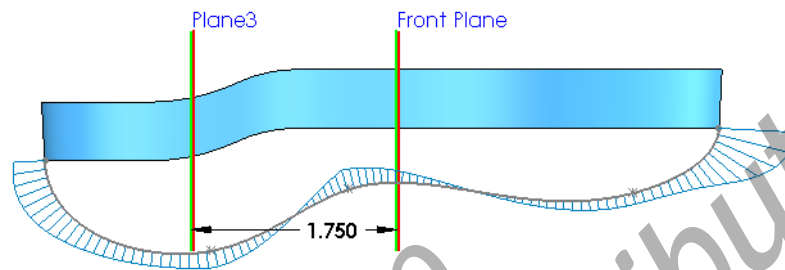
Display the curvature combs and adjust the shape of the spline until you are satisfied. Then exit the sketch.



This is the guide curve for the sweep.

29 Offset plane.

Create a plane offset **1.750"** from the Front plane. This will be used for sketching a the sweep profile.

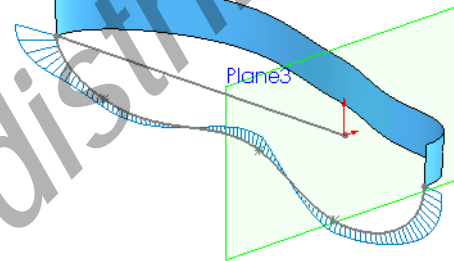


30 Sketch the sweep path.

Open a new sketch on the Right reference plane. Sketch a horizontal line through the origin.

One end of the line is coincident with the end of the spline. The other end is coincident with Plane3.

Exit the sketch.



Introducing: Partial Ellipse

Sketching a partial ellipse is similar to sketching a centerpoint arc:

- Position the cursor where you want the center and drag the mouse to establish the length of the major axis. Then release the mouse button.
- Next, drag the outline of the ellipse to establish the length of the minor axis.
- Finally, click where you want the ellipse to start, and drag the mouse to establish the length of the circumference.

Important!

To fully define an ellipse you must dimension or otherwise constrain the lengths of the major and minor axes. You must *also* constrain the orientation of one of the two axes. One way to do this is with a **Horizontal** relation between the ellipse center and the end of the major axis.


Where to Find It

- Click **Tools, Sketch Entities, Partial Ellipse**.
 - Or, click **Partial Ellipse**  on the Sketch toolbar.
-

31 Sketch the sweep profile.

Open a new sketch on Plane3.

The sweep profile is a partial ellipse. Sketching this is a multistep process:

1. Click **Partial Ellipse**  on the Sketch toolbar. Sketch a partial ellipse as shown. It should be approximately the lower-right quarter of a complete ellipse.

It is good if the start point of the ellipse is below the end of the minor axis.

Sketch it out in space so as not to inadvertently capture and unwanted relations.

2. Add a **Horizontal** relation between the center and the point at the end of the minor axis.

Note: Sketch relations have been viewed on for illustration purposes.

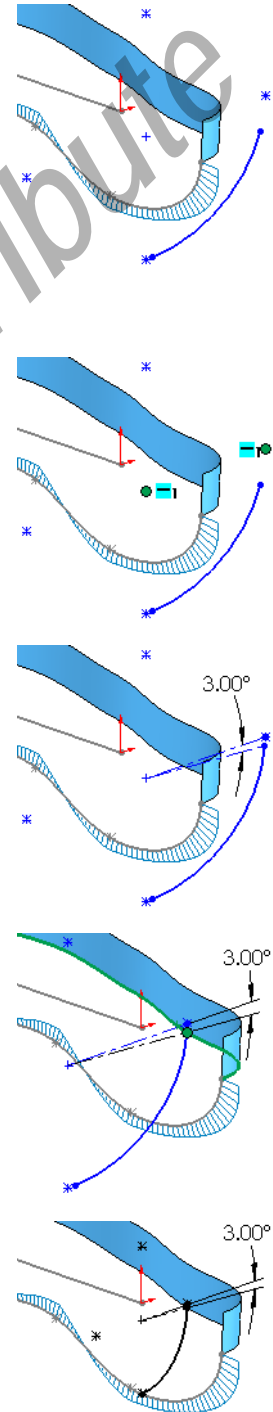
3. Sketch construction lines from the end of the minor axis to the center and then to the end point of the ellipse.

Dimension the angle between them and set the value to **3.00°**.

4. Add a **Pierce** relation between the end point of the ellipse and the bottom edge of the ruled surface.

5. Add a **Coincident** relation between the other end point of the ellipse and the end of the major axis.

Then add a **Pierce** relation between the end point of the ellipse and the sketched guide curve.

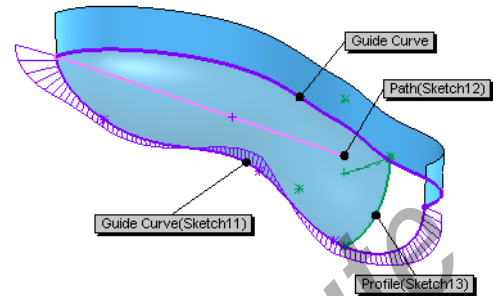


32 Sweep the surface.

Select the profile, path, and both guide curves to sweep the surface.

Note

An extra guide curve callout has been shown for illustration purposes.



Filling in Gaps

There are situations where special tools are needed to fill in areas of a model with surfaces. For example:

- **Blending shapes.**
Sometimes the shape you need cannot easily be created using fillets, sweeps, or lofts.
- **Repairing gaps or incorrect geometry in imported surfaces.**
Sometimes imported surfaces lack the completeness or precision to be knit into a solid. In these situations a tool is needed to fill in missing surface patches.
- **Closing holes in a part.**
In preparation for modeling a core and cavity mold, through holes in the part have to be closed off. Surfaces are used to do this. However, when the edges of the hole are not planar, creating a surface patch requires a special tool.

Introducing: Filled Surface

The **Filled Surface** feature constructs a surface patch with any number of sides, within a boundary defined by existing model edges, sketches, or curves.

Where to Find It

- Click **Filled Surface**  on the Surfaces toolbar.
- Or, click **Insert, Surface, Fill**.

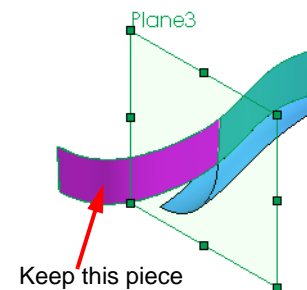
Preparation for Using Filled Surface

To properly blend the filled surface to its adjacent boundaries, you should not rely on using curves for boundaries. It is much better to use the edges of surfaces. This however, usually requires you to create reference surfaces prior to using the **Filled Surface** command.

33 Trim surface.


Trim the 3° draft reference surface using Plane3 as the trimming tool.

This will serve as one of the reference surfaces for the filled surface.



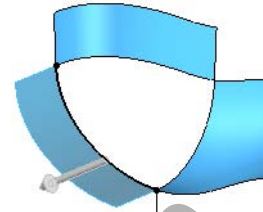
34 Extrude a surface for the second reference.


Open a new sketch on the Right reference plane.

Use **Convert Entities**  to copy the sketched guide curve into the active sketch.

Sketch a vertical construction line, coincident to Plane3, and use it to trim the converted curve.

Extrude a surface **0.5"** in the direction shown. Do not use draft.

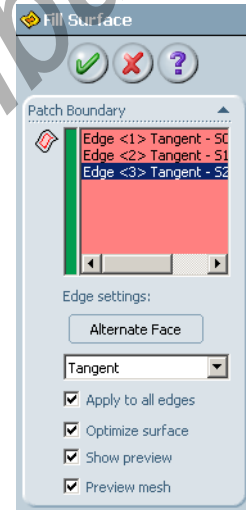
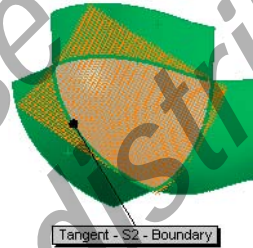
**35 Filled surface.**

Click **Filled Surface**  on the Surfaces toolbar.

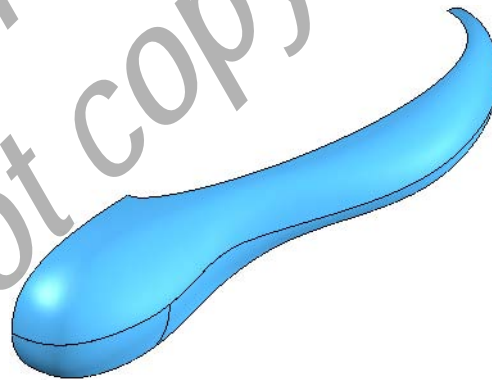
For **Edge settings**, select **Tangent**.


Select the edges of the three surfaces.

Click **OK**.

**36 Hide and show surfaces.**

Hide the reference surfaces and show the lofted surface.

**37 Zebra stripes.**

Click **Zebra Stripes**  on the View toolbar. Evaluate the quality and smoothness of the surfaces. Pay particular attention to the filled surface and how it blends with the swept surface.

To review **Zebra Stripes**, see *Zebra Stripes* on page 93. To review other techniques for evaluating the quality of surfaces, see *Analyzing Geometry* on page 87.



**Introducing:
Planar Surfaces**

You can create a planar surface from either a closed single contour non-intersecting sketch, or a closed set of planar edges.

Where to Find It

- Click **Insert, Surface, Planar**.
- Or, click **Planar Surface**  on the Surfaces toolbar.

**Introducing:
Curve Through
Reference Points**

Curve Through Reference Points creates a spline through sketch points, vertices, or both.

Where to Find It

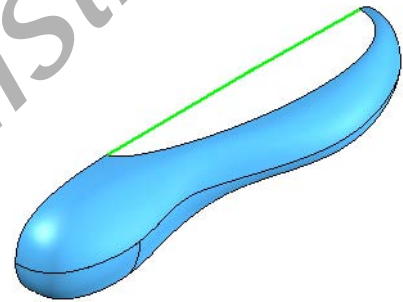
- Click **Insert, Curve, Curve Through Reference Points**.
- Or, click **Curve Through Reference Points**  on the Curves toolbar.

Note


38 Click Curve Through Reference Points .

Select the two vertices shown, creating a straight spline.

We could just as easily have sketched a line.

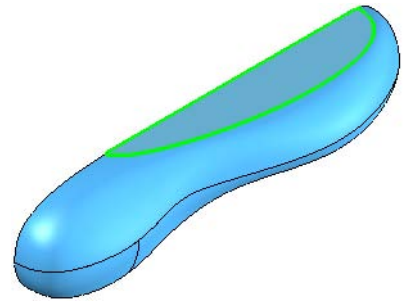


39 Planar surface.

Click **Insert, Surface, Planar** or click **Planar Surface**  on the Surfaces toolbar.

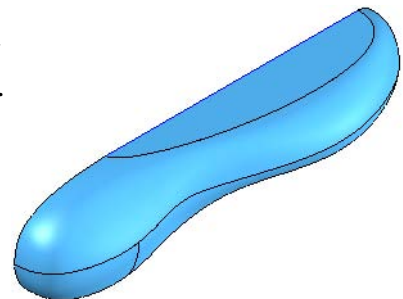
Select the curve you just created and the open edge of the lofted surface.

Click **OK**.




40 Results.

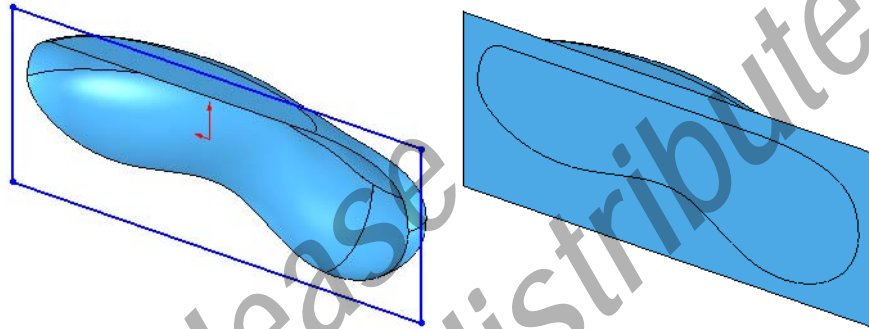
The resulting planar surface fits exactly across the opening of the lofted surface.



41 Another planar surface.

Open a new sketch on the **Right** reference plane and sketch a rectangle somewhat larger than the outline of the part.

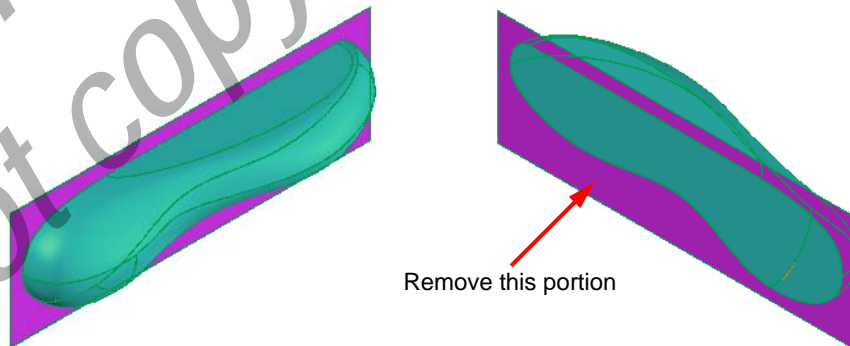
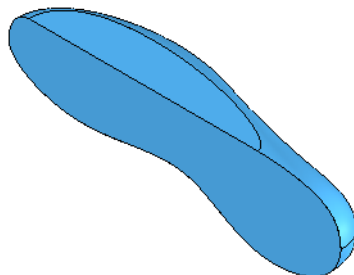
Click **Planar Surface** . The system automatically creates a planar surface using the active sketch. Click **OK**.

**42 Mutual trim.**

Click **Trim Surface** . For **Trim Type**, click **Mutual**.

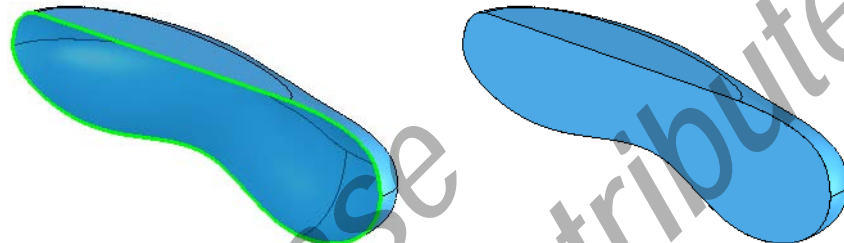
For **Trimming Surfaces**, select all five surfaces: the lofted surface, the swept surface, the filled surface, and both planar surfaces.

Click **Remove selections** and click in the selection list. Identify the portion of the planar surface that you want to *remove*.

**43 Results.**

Note

In this particular example, instead of creating an oversized planar surface and trimming it, we could have simply created the planar surface by selecting the edges of the existing surfaces. However, sometime those edges might not be planar, or they might extend beyond where they should. In those cases it is better to create an oversized surface and use **Mutual Trim**.



It's Not a Solid – Yet

Although the collection of surfaces looks solid, it is not. It is hollow. To transform these surfaces into a solid, two more steps are required:

1. All the surfaces must be combined into a single composite surface.
2. The resulting composite surface must be filled to make a solid.


Creating a Knit Surface

Knit Surface is used to combine or sew several surfaces into one composite surface. If the knit surface encloses a complete volume, with no gaps, it can be filled to become a solid.


**Introducing:
Knit Surface**

Use **Knit** to combine two or more reference surfaces or faces into one. The edges of the surfaces or faces must be adjacent and not overlapping. Use the option **Try to form solid** to form the knit surface into a solid provided the surfaces form a closed volume.

Where to Find It

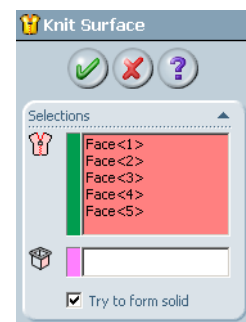
- Click **Insert, Surface, Knit**.
- Or, click **Knit Surface**  on the Surfaces toolbar.

44 Knit surfaces.

Click **Insert, Surface, Knit** or click **Knit Surface**  on the Surfaces toolbar. Select the trimmed surface and the two planar surfaces by either clicking them in the graphics window or the Feature-Manager design tree.

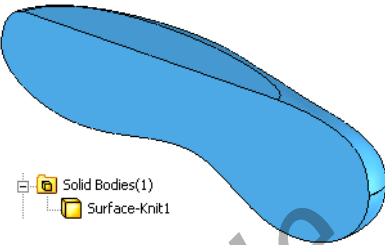
Select the **Try to form solid** check box.

Click **OK**.




45 Results.

The resulting solid doesn't look much different from the surfaces. However, the FeatureManager design tree indicates that a solid body now exists in the part.



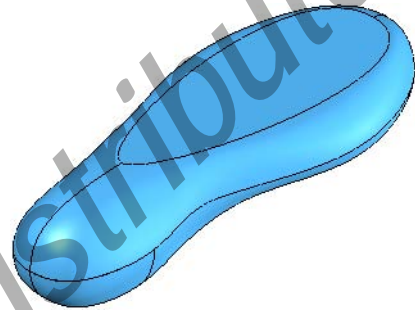
A **Solid Bodies** folder appears.

46 Mirror.

Click **Mirror**  on the Features toolbar. Select the planar face (step 41) as the **Mirror Face/Plane**.

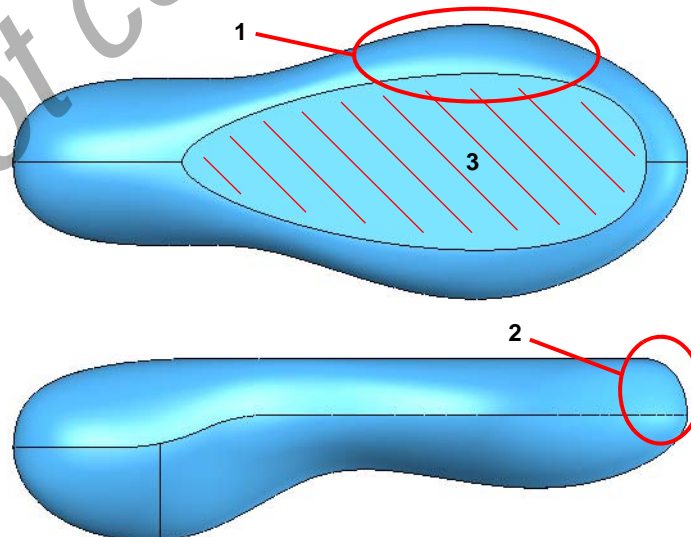
Expand the **Bodies to Mirror** list and select the solid body.

Make sure **Merge solids** is selected and click **OK**.

**Design Changes**

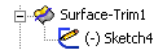
Let's evaluate the design so far. There are three areas that don't look quite right.

1. The curves of the parting line and the edge of the area where the keypad goes do not compliment each other well.
2. Also, the front end of the remote control isn't rounded enough.
3. The area where the keypad goes is boring – it is flat.

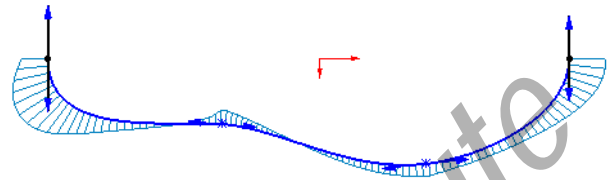


Dynamic Feature Editing

The curve that ultimately controls the outline of the remote control is the parting line and it is embedded under the trimmed surface.



When you edit this sketch, the part is rolled back and all the geometry disappears. Fixing the overall shape of the remote control would take a long process of trial and error because you would be working blind.



Dynamic feature editing enables you to make changes to features and sketches without rolling back the part. This way you can see the effects of the changes as you make them.

Introducing: Move/Size Features

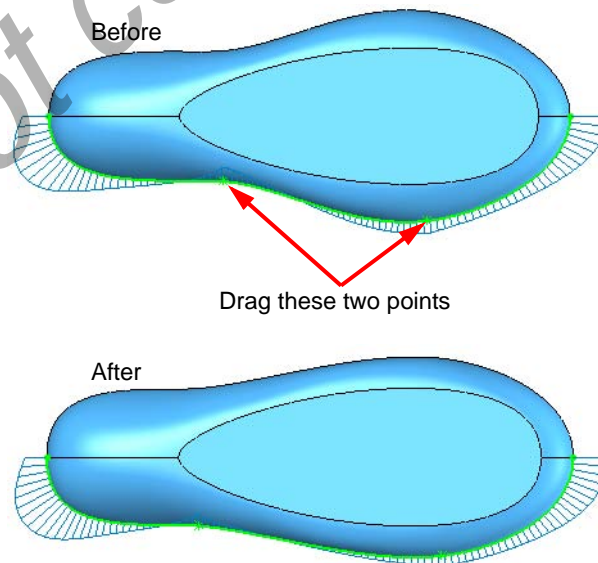
Move/Size Features enables you to dynamically edit features. When you drag the entities of a sketch, either with or without opening the sketch itself. The preview updates when you release the mouse button after dragging.

Where to Find It

- Click **Move/Size Features**  on the Features toolbar.

- 1 Click **Move/Size Features** .

Expand the trimmed surface and show the underlying sketch. Adjust the shape of the spline by dragging the interpolant points.

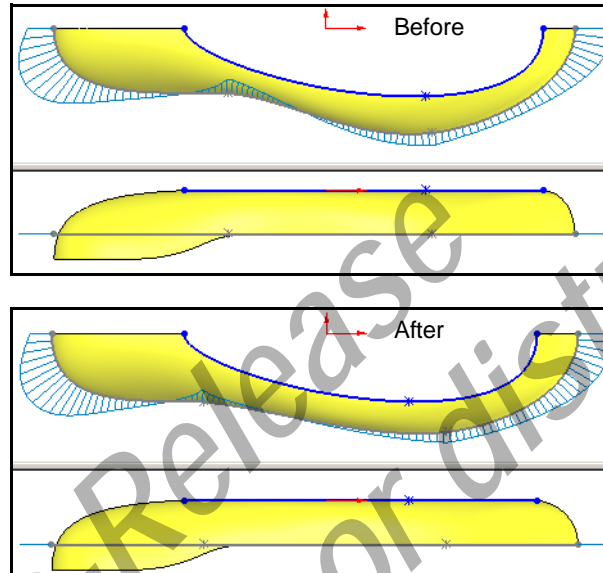


2 Dynamically edit a sketch.

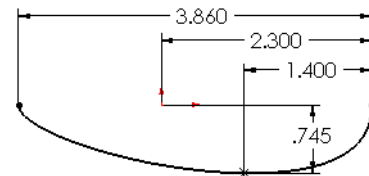
Expand the lofted surface feature and double-click the sketch that defines the edge of the flat area where the keypad will go.

Tip

Use viewports to see the top and front views at the same time.

**Tip**

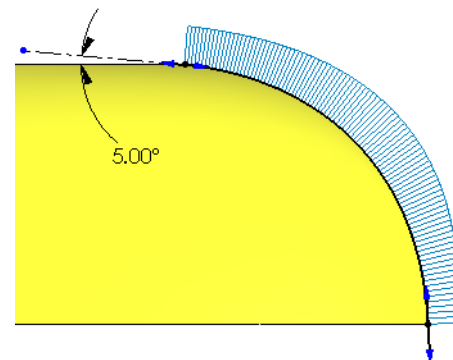
If you decide to dimension the sketch, turn off **Move/Size Features** for improved performance. With **Move/Size Features** on, the model will rebuild each time you add a dimension.

**3 Edit the other sketches.**

Repeat this procedure as necessary to edit the other sketches that make up the lofted surface.

Note

This is an exercise in judgement and aesthetics. There is no unique right or wrong solution.




Replacing a Face

We will create a new, concave face to replace the planar face.

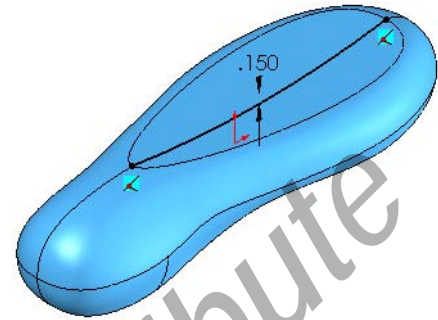
4 Sketch an arc.

Open a new sketch on the **Right** reference plane.

Sketch a **3 Point Arc**  and dimension it as shown.

The endpoints have **Coincident** relations with the vertices at the ends of the planar face.

Exit the sketch.



5 Create a plane.

Create a reference plane parallel to the **Front** plane, passing through the centerpoint of the arc you just sketched in step 4.

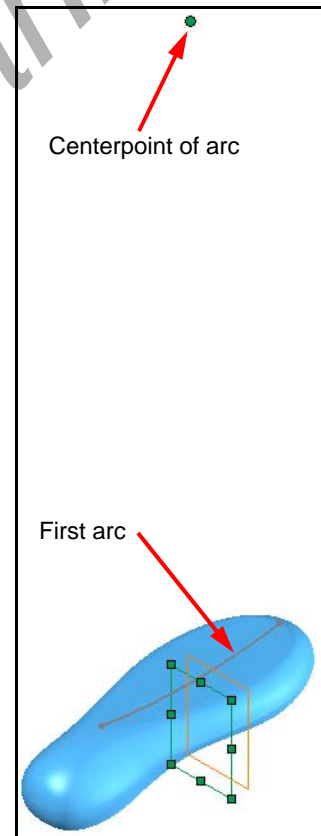
6 Sketch a second arc.

Create a new sketch on **Plane4**, the plane you just created.

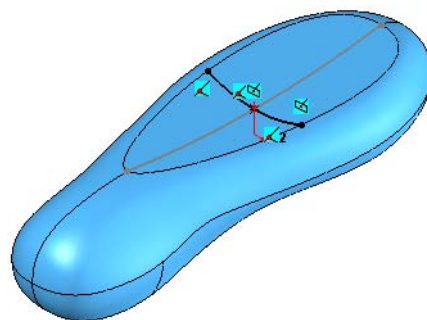
Sketch a **Centerpoint Arc** . The two endpoints have **Pierce** relations with the edges of the planar face.

Create a reference point on the arc. Relate it to the arc in the previous sketch with a **Pierce** relation.

Add a **Coincident** relation between the arc's centerpoint and the **Right** reference plane.




+ K2



7 Exit the sketch.

8 Filled surface.

Click **Filled Surface**  on the Surfaces toolbar.

For **Edge settings**, select **Contact**.

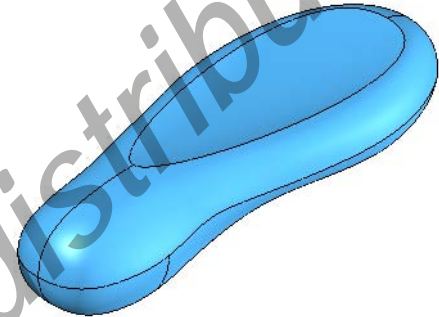
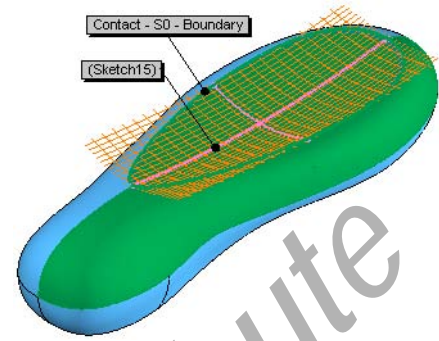
Select the two edges of the planar face.

Under **Constraint Curves**, select the two arcs.

Under **Options**, select **Merge result**.

Click **OK**.

The planar face is replaced with the concave face.

**Merge Result**

The behavior for this option depends on the boundaries. When all the boundaries belong to the same solid body, you can use the filled surface to replace a face of the solid. This streamlines your work, eliminating the need to use the **Replace Face** command. For more information about **Replace Face**, see *Exercise 23: Using Import Surface and Replace Face* on page 255.

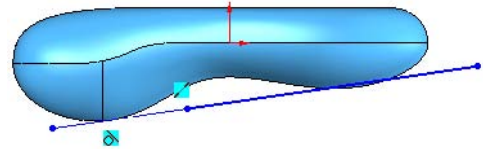
9 Sketch.


Open a sketch on the Right reference plane.

Sketch a line tangent to the silhouette edge as shown.

Split the line and change the left-most portion to construction geometry.

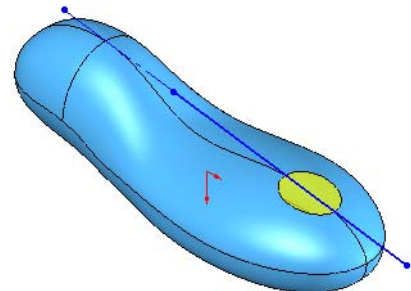
Adjust the angle of the line so it barely intersects the bottom of the front portion of the remote control.

**10 Cut through all.**

Click **Extruded Cut** . Since this is an open profile, the end condition will be set to **Through All** automatically.

The goal is to create a small flat spot to the remote can be set on a table without falling onto its side.

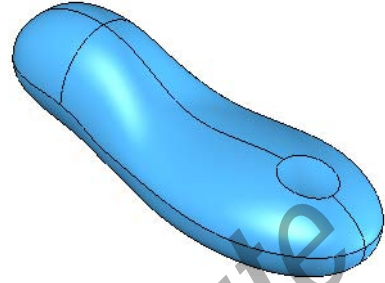
If the area of the cut is too big or too small, use **Move/Size Features** to adjust the sketch dynamically.



11 Dome.

Create a **Dome** feature about **0.065"** deep. The exact depth is not critical.

To review the **Dome** feature, see *Dome Feature* on page 129.



Finishing Touches

In this next section we will:

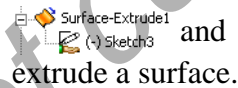
- Split the part into separate bodies, each representing a major component of the remote control;
- Shell the part;
- Define the basic geometry and shape of the keypad;
- Create specialized features called fastening features;
- Save the individual bodies as part files.

Splitting the Part

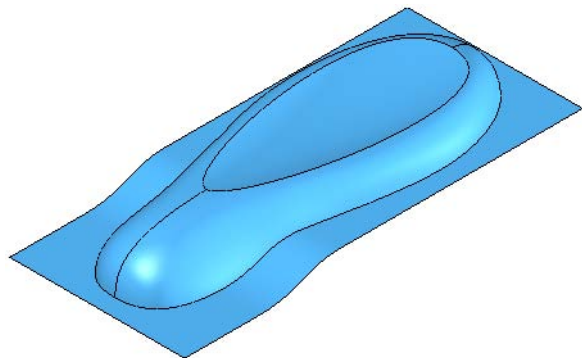
Splitting a part into multiple bodies was covered in *Lesson 1: Multibody Solids*. To review this topic, see *Splitting a Part into Multibodies* on page 42.

1 Extrude the parting surface.

Reuse the original parting sketch

 and extrude a surface.

Use **Mid Plane** as the end condition and set the **Depth** is such that it extends beyond the body of the part.



2 Split the part.

Click **Split**  or click **Insert, Features, Split**.

Select the parting surface as the trim tool.

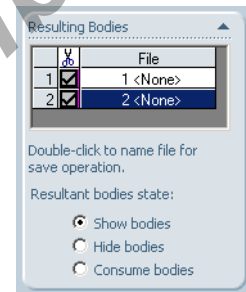
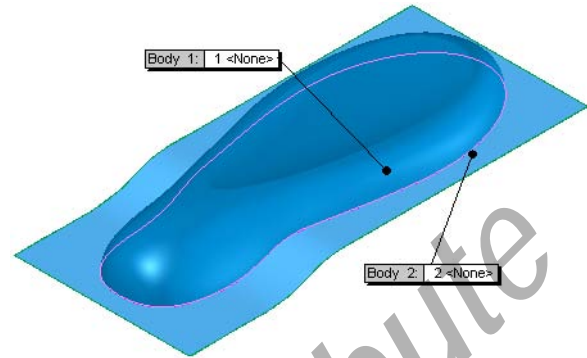
Click **Cut Part**. The system computes the intersection of the trim tool with the part and calculates the results.

We want to create both bodies but we do not want to save them as separate part files at this time.

Select the check boxes for both bodies but leave the file name set to <None>.

For **Resultant bodies state**, select **Show bodies**.

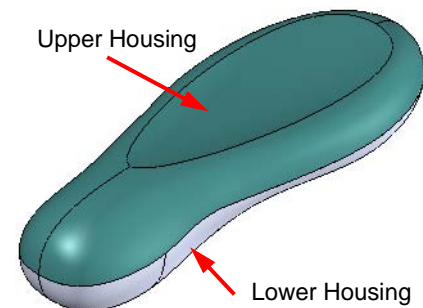
Click **OK**.

**3 Hide the parting surface.****4 Rename the solid bodies.**

Expand the Solid Bodies folder.

Rename the bodies Upper Housing and Lower Housing.

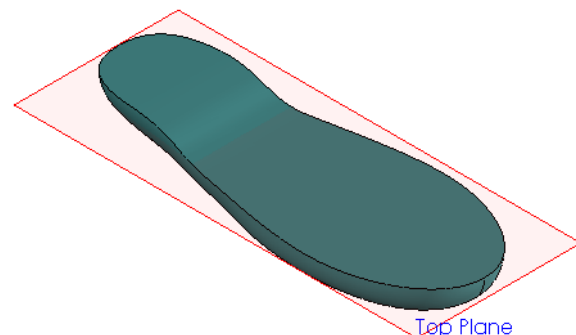
Change the colors of the upper and lower housing so it will be easier to tell them apart.

**5 Hide the Lower Housing.****Modeling the Keypad**

To save time we will use a library feature for the sketch of the holes for the keypad. The sketch is straightforward and creating it step-by-step contributes nothing to this case study about surfacing.

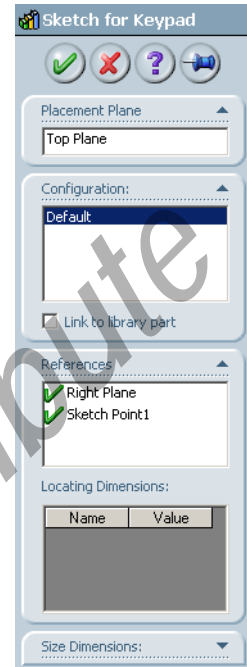
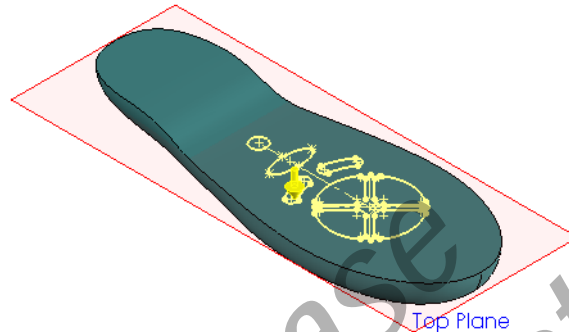
1 Reference plane.

Show the Top reference plane. This is the plane onto which we will insert the library features (sketch).



2 Library Feature.

Drag the library feature named Sketch for Keypad from the Design Library and drop it onto the Top reference plane.



Associate the external references to the target part's Right reference plane and origin.

Click **OK**.

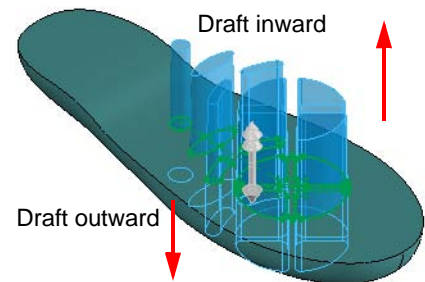
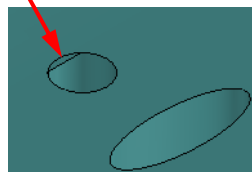
3 Dissolve the library feature.

Right-click the library feature and select **Dissolve Library Feature** from the shortcut menu.

4 Extrude a cut.

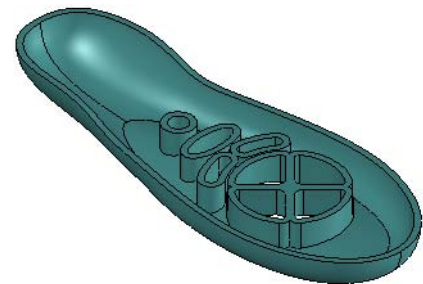
Extrude a cut **Through All** in *both* directions. Use **1.00°** of draft.

A sliver face is left if the cut is not extruded in both directions.



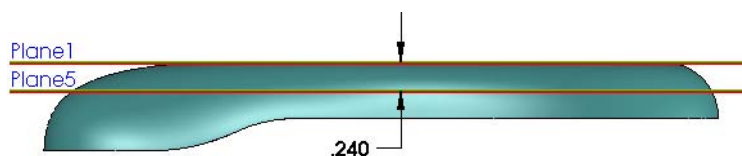
5 Shell.

Shell the Upper Housing using a **Thickness** of **0.080** inches.



6 Reference plane.

Create a reference plane offset **0.240"** from the plane that was used to sketch the area around the keypad (step 15 on page 200).



Note

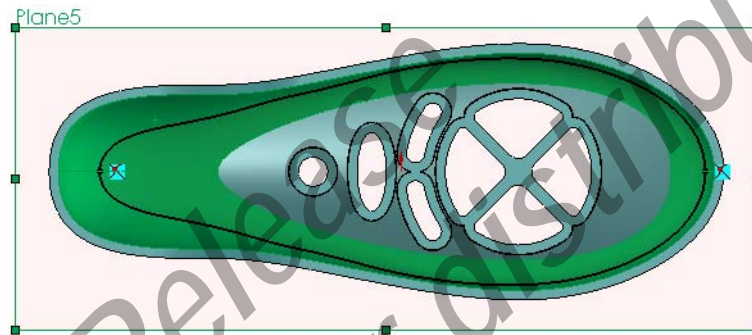
The 0.240" dimension was obtained by adding 0.010" to the sum of 0.080" (the shell thickness) and 0.150" (the dimension on the arc in step 4 on page 216).

7 Intersection curves.

Open a sketch on Plane5.

Click **Intersection Curve**  on the Sketch toolbar.

Select the two faces as shown on the inside of the Upper Housing.



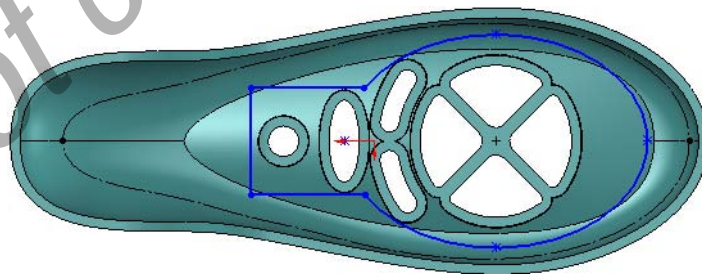
Turn off the **Intersection Curve** tool and hide Plane5.


8 Keypad.

Change the two intersection curves to construction geometry and sketch the outline of the keypad as shown. Use an ellipse and a rectangle and trim as necessary.

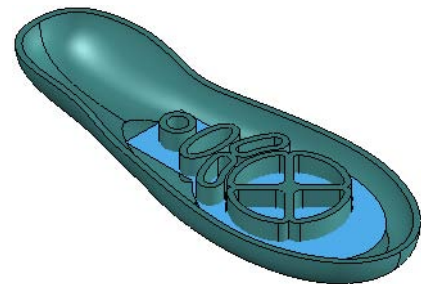
Note

The intersection curves are used as a guide to make sure the keypad doesn't interfere with the inside of the housing.

**9 Planar surface.**

Click **Planar Surface**  on the Surfaces toolbar.

Create a planar surface using the active sketch.




**Introducing:
Cut with Surface**

You can cut a solid model by removing material with a surface or a plane. In a multibody part, you use the **Feature Scope** to determine which body or bodies to cut.

Where to Find It

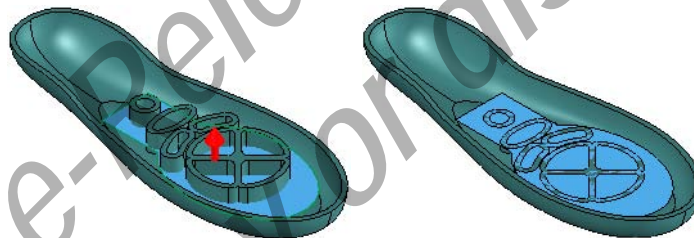
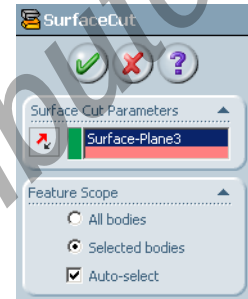
- Click **Cut With Surface**  on the Features toolbar.
- Click **Insert, Cut, With Surface**.

10 Cut with surface.

Click **Cut With Surface**  on the Features toolbar. Under **Surface Cut Parameters**, select the planar surface and the cutting surface.

Under **Feature Scope**, click **Selected bodies** and select the **Auto-select** check box.

Click **OK**.

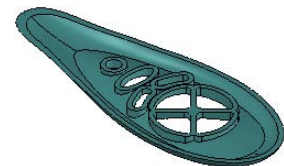


Question:

Since the surface we are using is planar, why not just cut using the reference plane?

Answer:

The advantage of using a surface rather than a plane is that the extent of the cut is limited by the boundaries of the surface. If we cut with the reference plane, the entire body would have been cut, not just the areas around the keypad holes.




**Introducing:
Thicken**

Creates a solid feature by thickening one or more adjacent surfaces. If the surface you want to thicken is comprised of multiple adjacent surfaces, you must first knit the surfaces together before you thicken the surface.

Where to Find It

- Click **Thicken**  on the Features toolbar.
- Click **Insert, Boss/Base, Thicken**.

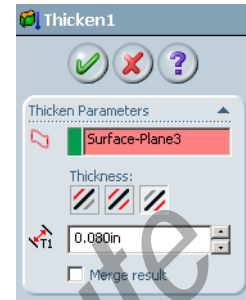
11 Thicken.

Click **Thicken**  on the Features toolbar.

Select the planar surface.

Set the **Thickness** to **0.080** inches and clear the **Merge result** check box.

Examine the preview.



Select either **Thicken Side 1** or **Thicken Side 2** as necessary so that the surface is thickened *away* from the solid body.

Click **OK**.

12 Rename.

Name the solid body Keypad.

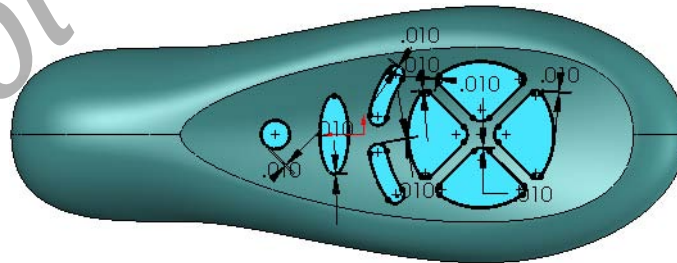
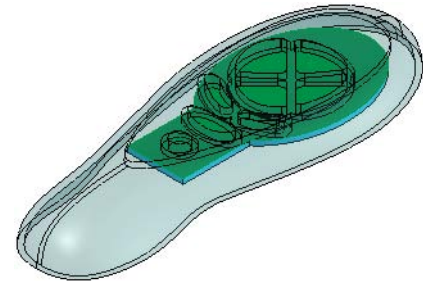
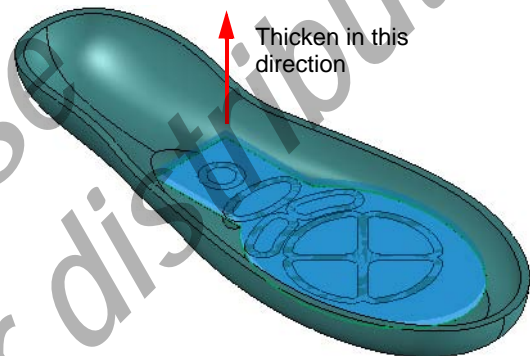
13 Offset the edges.

Open a new sketch on the uppermost face of the Keypad. This will be the sketch for the buttons.

Note

The Upper Housing is shown transparent for illustration purposes.

Click **Offset Entities** . Offset the edges of the keypad holes **0.010**".

**Tip**

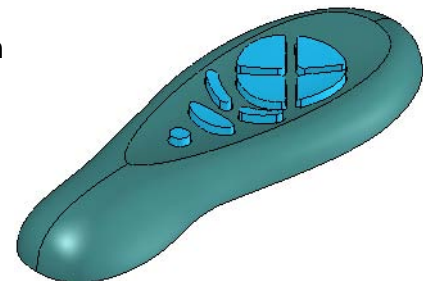
Press the **Enter** key to repeat the previous command.

14 Extrude.

Extrude the sketch using **Offset From Surface** and an **Offset Distance** of **0.100**".

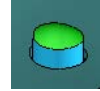
Set the **Draft Angle** to **1.00°** and make sure the draft is inward.

Select **Merge result** and use the **Feature Scope** to select the Keypad.



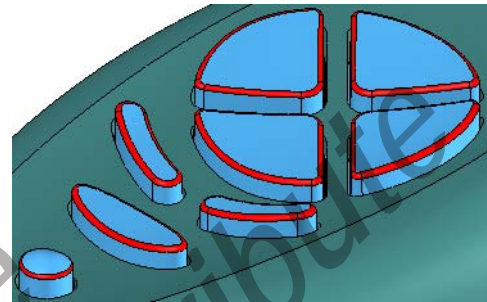
15 Dome.

Create a **0.050"** dome on the top of the round button.



16 Fillet.

Add **0.020"** radius fillets to the edges of the keypad buttons, shown here in red for illustration purposes.



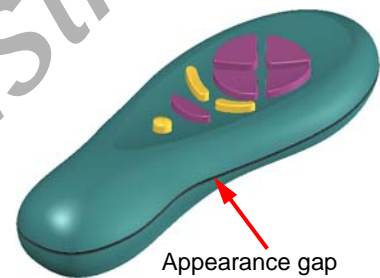
Appearance Gap

The next step in the process is to sweep a cut to create an appearance gap between the upper and lower housings.

First we will create two 3D curves:


- The sweep path
- The guide curve

Then we will sketch the sweep profile.




1 Hide the Keypad body.

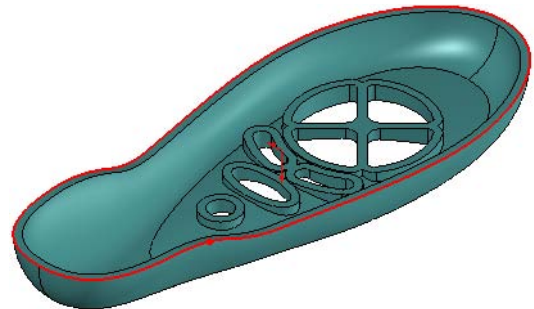
2 3D sketch.

Click **3D Sketch**  on the Sketch toolbar to open a new 3D sketch.

3 Fit spline.

Click **Fit Spline**  on the Spline Tools toolbar.

Right-click the outermost edge of the Upper Housing and select **Select Tangency** from the shortcut menu.



Tighten the **Tolerance** until the **Actual Deviation** is less than **0.001"**.

Click **OK**. This is the path for the sweep.

Note

The resulting spline is shown here in red for illustration purposes only. It does not mean the spline is over defined.

4 Exit sketch.

5 Repeat.

Repeat steps 2 through 4, fitting a second spline to the inside edge of the Upper Housing. This is the guide curve for the sweep.

Note

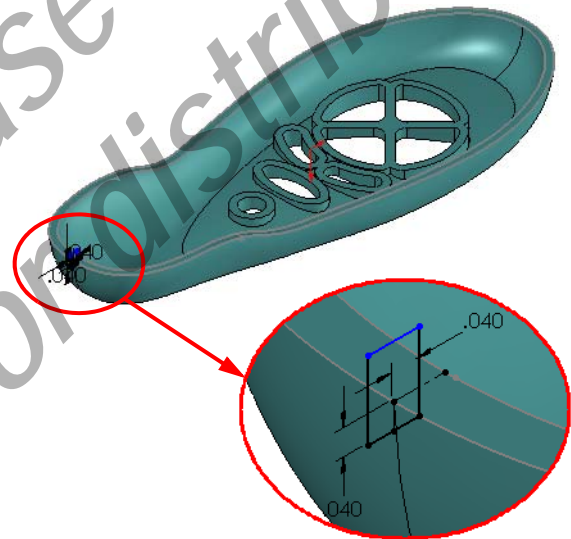
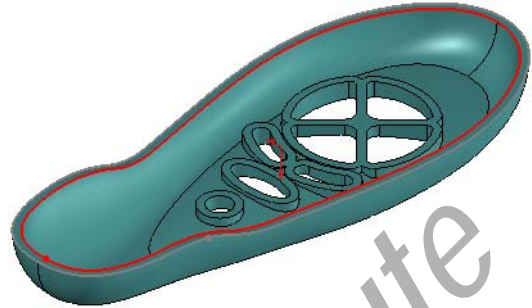
We could also have used **Composite Curve** for the path and guide. To review **Composite Curve**, see *Introducing: Composite Curve* on page 106.

6 Profile sketch.

Open a sketch on the Right reference plane.

Sketch a rectangle as shown. This is the profile for the swept cut feature.

The uppermost line in the rectangle does not need to be fully defined.

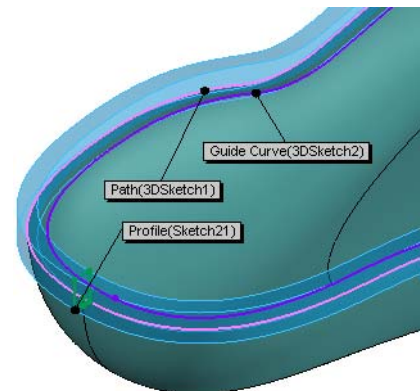
**7 Sweep a cut.**

Select the **Profile**, **Path**, and **Guide Curve** as shown in the illustration.

Expand the **Options** listing.

For **Orientation/twist type**, select **Follow part and 1st guide curve**.

Click **OK**.

**Draft**

We could have built the required draft angle into the profile sketch. However, in this case we will add draft using the **Draft** feature.


Introducing: Draft

The **Draft** feature tapers selected faces in the model by a specified angle with respect to the pull direction of a mold. You can add draft using a **Neutral Plane** or a **Parting Line**.

Where to Find It

- Click **Draft**  on the Features toolbar.
- Click **Insert, Features, Draft**.

1 Parting line draft.

Click **Draft**  on the Features toolbar.

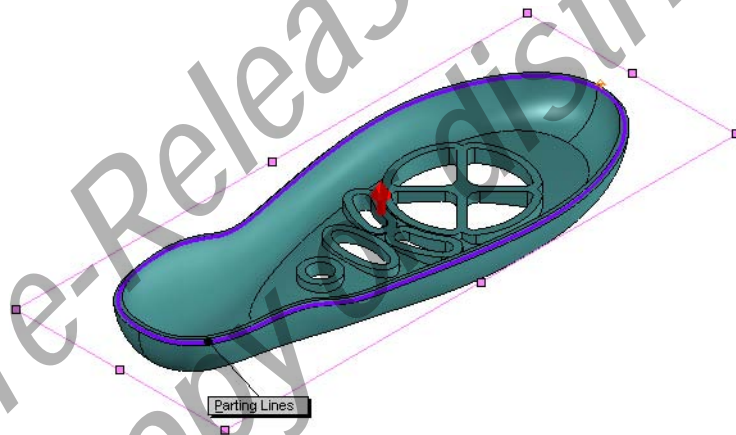
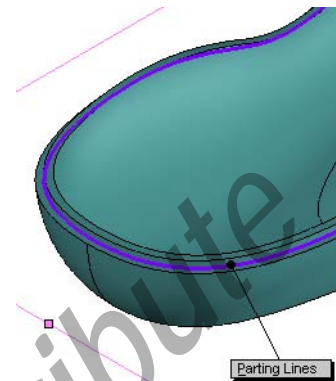
For **Type of Draft**, select **Parting Line**.

For the **Draft Angle**, enter **1.00°**.

For **Direction of Pull**, select the Top reference plane.

Click **Reverse Direction**.


For **Parting Lines**, select the model edge shown and click **OK**.




Draft Analysis

The **Draft Analysis** tool is useful in determining whether the part has sufficient draft to be removed from the mold based on a set draft angle.

Where to Find It

- Click **Draft Analysis**  on the Mold Tools toolbar.
- Or, click **Tools, Draft Analysis....**

2 Draft analysis.

Click **Draft Analysis**  on the Mold Tools toolbar, or click **Tools, Draft Analysis**.

For **Direction of Pull**, select the Top reference plane.

Click **Reverse Direction**.

Set the **Draft Angle** to **1.00°**.

Select the **Face classification** check box.

Click **Calculate**.

The green faces have positive draft with respect to the pull direction. The red faces have negative draft.

Click **Cancel**.

Note

For a more in-depth discussion of draft analysis, see *Analyzing Draft on a Model* on page 271.

3 Hide and show bodies.

Hide the Upper Housing. Show the Lower Housing.

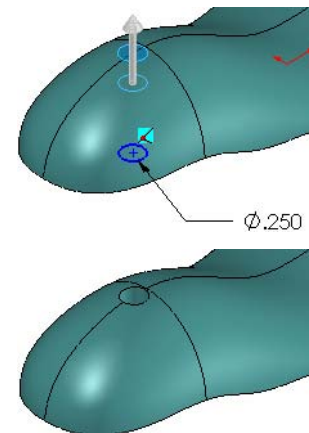
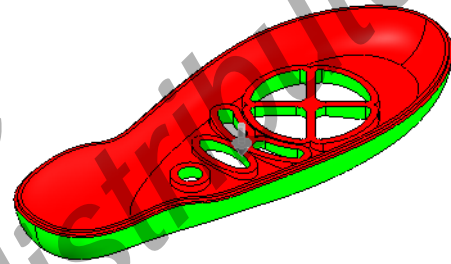
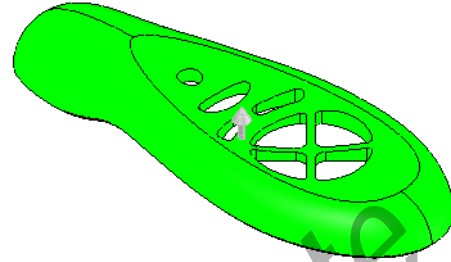
4 Hole for fastener.

Open a sketch on the Top reference plane and sketch a **0.250"** diameter circle as shown. The distance from the origin is not critical but it should be located near the rear of the remote.

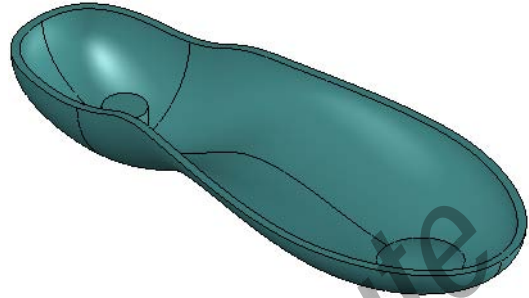
Add a **Coincident** relation between the circle's center and the Right reference plane.

Extrude a cut as follows:

- The **From** position is **Offset 0.75"** from the sketch plane.
- The **End Condition** is **Through All**.
- The **Draft Angle** is **1.00°**.
- Select the **Draft outward** check box.
- For **Feature Scope**, select the Lower Housing.



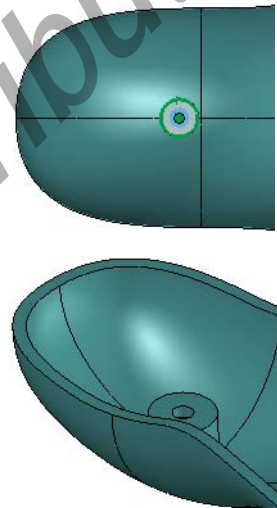
- 5 **Shell.**
Shell the Lower Housing using a **Thickness** of **0.080** inches.



- 6 **Clearance hole.**
Click **Hole Wizard**  on the Features toolbar.

- **Standard = ANSI Inch**
- **Type = Screw Clearances**
- **Size = #4**
- **Fit = Normal**
- **End Condition = Through All**
- **Feature Scope = Lower Housing**

Add a **Concentric** relation between the locating point and the edge of the cut feature.







Fastening Features

Fastening Features streamline creation of common features for plastic parts. You can create:



- Mounting Boss
- Snap Hook
- Snap Hook Groove
- Vent (also useful in sheet metal parts)

Where to Find It

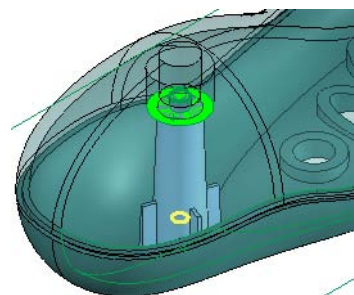
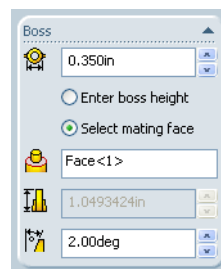
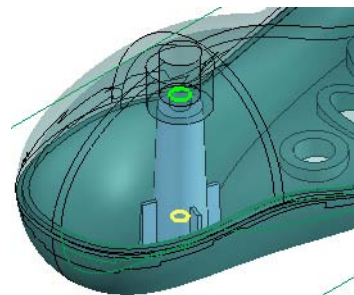
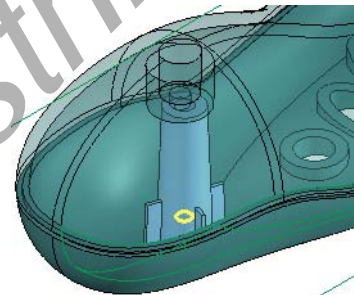
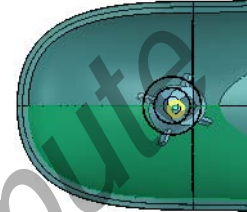
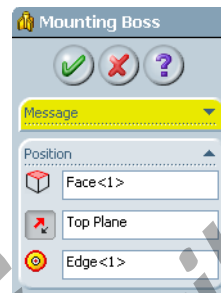
- Click **Mounting Boss** , **Snap Hook** , **Snap Hook Groove** , or **Vent**  on the Fastening Features toolbar.
- Click **Insert**, **Fastening Feature**, and select either **Mounting Boss**, **Snap Hook**, **Snap Hook Groove**, or **Vent**.

-
- 1 **Appearance.**
Show the Upper Housing.
Make the Lower Housing semi-transparent. A transparency of 0.75 works well.

2 Mounting boss.

Click **Insert, Fastening Feature, Mounting Boss**. Creating a mounting boss is a multistep process:

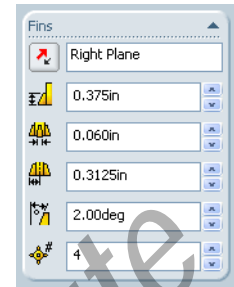
1. Change to a bottom view orientation and select the inside face of the Upper Housing. One technique is to select the face through the fastener clearance hole.
2. To define the direction of the mounting boss, select the Top reference plane and click **Reverse Direction**. This orients the mounting boss correctly with respect to the pull direction of the mold.
3. To position the mounting boss, select the edge of the clearance hole.
4. To define the height of the mounting boss, select the planar face on the inside of the Lower Housing as shown.



Set the **Diameter** to **0.350"** and the **Draft Angle** to **2.00°**.

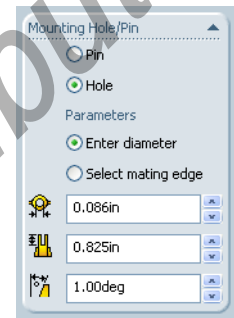
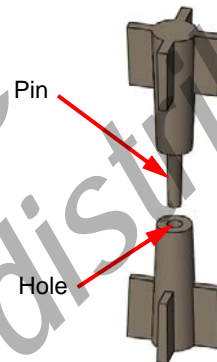
5. To orient the fins, select the Right reference plane. Set the other **Fins** options as follows:

- Height = 0.375"
- Width = 0.060"
- Length = 0.3125"
- Draft Angle = 2.00°
- Number of fins = 4



6. A mounting boss can have a pin or a hole. In this case we want a hole.

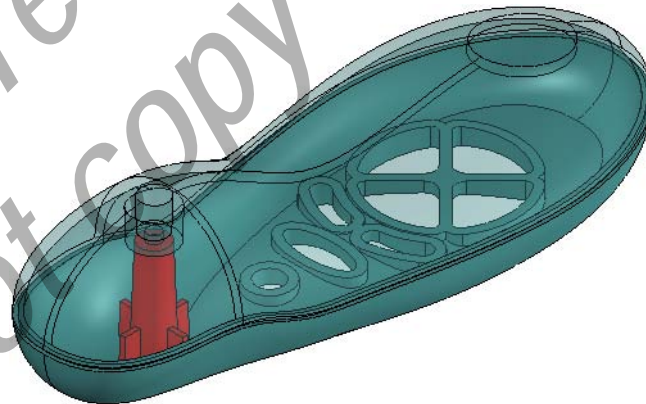
- Select **Hole**
- Select **Enter diameter**
- Diameter = 0.086"
- Depth = 0.825"
- Draft Angle = 1.00°



7. Click **OK**.

3 Results.

The mounting boss is added to the inside of the Upper Housing.



Note

The mounting boss is shown in red for illustration purposes.

4 Appearance.

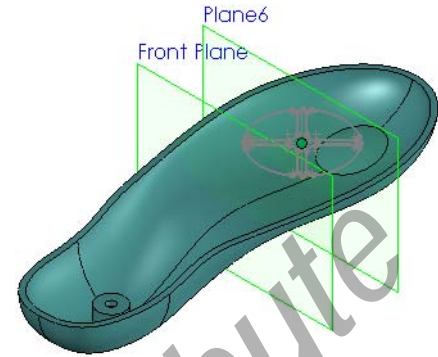
Hide the Upper Housing.

Remove the transparency from the Lower Housing.

5 Offset plane.

Show the sketch that was inserted as a library feature for the keypad cutout (step 2 on page 220).

Create a plane that is parallel to the **Front** reference plane and that passes through the point at the center of the circular keypad.

**6 3D sketch.**

Open a new 3D sketch.

Insert two points. Make them **Coincident** with the inside edges of the Lower Housing and also coincident (**On Surface**) with the offset plane.

**7 Snap hook.**

Click **Insert, Fastening Feature, Snap Hook**.

Select one of the points in the 3D sketch.

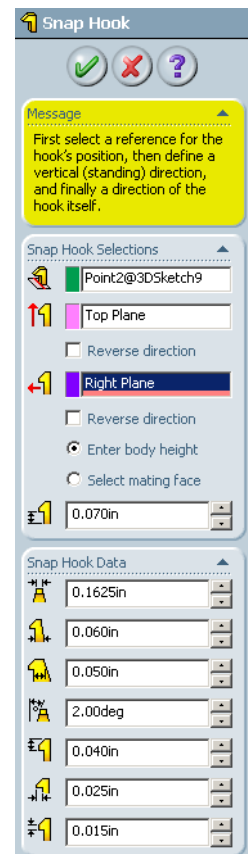
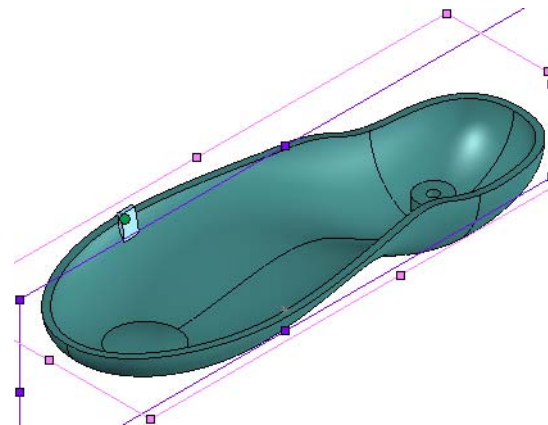
Select the **Top** reference plane to define the vertical direction of the snap hook.

Select the **Right** reference plane to define the direction of the hook.

Set the **Body height** at **0.070"**.

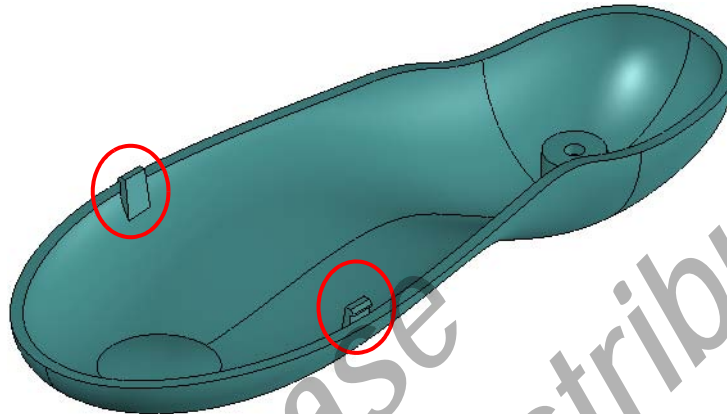
Enter the **Snap Hook Data** as shown.

Click **OK**.



8 Repeat.

Create a second snap hook using the second point in the 3D sketch.



9 Show solid body.

Show the Upper Housing.

10 Snap hook groove.

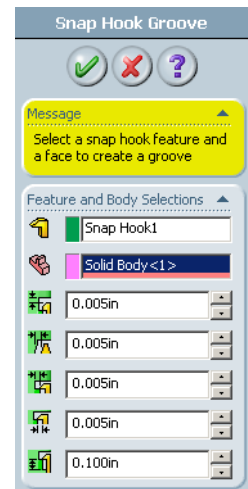
You must create a snap hook *before* you can create a snap hook groove.

Click **Insert, Fastening Feature, Snap Hook Groove.**

Select the Snap Hook1 feature.

Select the Upper Housing as the solid body that the groove will be applied to.

Enter the dimension values as shown.



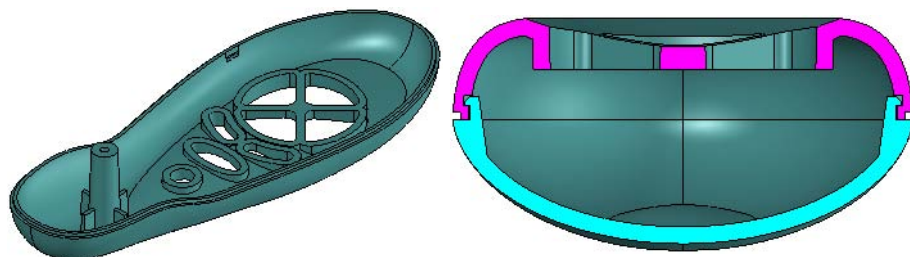
Note

The dimensions of the snap hook groove are driven by the snap hook. The values in the PropertyManager are offsets, or clearances, so you can make the groove slightly larger than the hook.

Click **OK.**

11 Second snap hook groove.

Repeat this process for Snap Hook2. The results are shown below.



Section View

Note

The cut faces of the section view have been colored for clarity.

Saving the Bodies and Creating an Assembly

Save Bodies allows you to save individual solid bodies as part files. You can indicate which bodies you want to save. Optionally you can generate an assembly from the saved parts.

To review **Save Bodies** and **Create Assembly**, see *Introducing: Save Bodies* on page 39.

12 Save bodies.

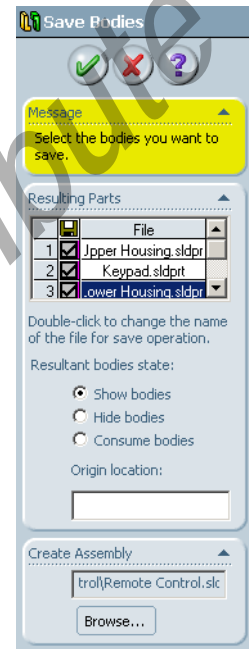
Right-click the **Solid Bodies** folder and select **Save Bodies** from the shortcut menu.

Save the three solid bodies as:

- Upper Housing
- Lower Housing
- Keypad

If you want to create an assembly do the following:

- 1 In the **Create Assembly** group box, click **Browse**. The **Save As** dialog opens.
- 2 Browse to where you want to save the assembly.
- 3 Give the assembly a name and click **Save**.



13 Save and close all the files.

Rapid Prototyping

By using rapid prototypes early in the product development cycle, you can receive critical feedback early in the design process. Rapid prototyping is sometimes called 3D printing.

The 3D printing process often takes advantage of a rapid prototyping process known as stereolithography, or layered object manufacture. 3D printers come with special software that imports the CAD file and slices it into thin horizontal layers 0.003 inches to 0.01 inches thick. Each thin cross-section is sent to the 3D printer, which builds up the model, layer by layer, starting from the bottom of the part and moving upward. In a matter of minutes or hours, the model is complete.




Print3D

Print3D is a web portal linked to the SolidWorks software. Using **Print3D**, you can contact selected rapid part and prototype vendors to request price quotes or place an order for rapid prototypes of the currently open part document. Some vendors provide instant price quotes; others will contact you via e-mail.


Print3D automates the process of requesting a quote or ordering a prototype, eliminating the need to search for reliable services, save parts as STL files, FTP the files to vendors, or perform other operations. Model data is encrypted prior to transmission, so your data is always secure.

Where to Find It

- Click **Print3D**  on the Standard toolbar.

Note

You may have to use **Tools, Customize** to add the Print 3D icon to the Standard toolbar.



The screenshot displays the Print3D web portal interface. At the top, there is a navigation bar with the SolidWorks logo and the Print3D title. Below the navigation bar, there are three vendor listings:

- Quickparts.com, Inc.**: An Inc. 500 Company; Instant Quotes ~ Single Source for Quick Parts. Services include Stereolithography (SLA), Fused Deposition Modeling (FDM), Selective Laser Sintering (SLS), QuickTurn Injection Molding, Silicone Molding, and CNC, Sheet Metal, Castings. Contact information: (p) 1-877-521-8683, (e) qphelp@quickparts.com. Links: [Print3D at Quickparts](#), [Tell me more](#).
- The Protomold Company, Inc.**: Protomold's Rapid Injection Molding service produces real injection molded parts in engineering grade resins within 5 days of receipt of your native SolidWorks file, starting as low as \$1,995 for 25 parts. Real Parts. Real Fast. Real Savings!. Link: [Print3D at Protomold](#), [Tell me more](#).
- Xpress3D, Inc.**: Xpress3D instantly provides price quotes from multiple rapid prototyping services. Easily choose based on price, delivery time, rapid prototyping process, or material. Prototyping includes SLA (Stereolithography), FDM (Fused Deposition Modeling), and 3DP (3D Printing). Xpress3D's automatic file compression, encryption, and upload make this the fastest, easiest, and securest way to order prototypes. Link: [Print3D at Xpress3D, Inc.](#), [Tell me more](#).

At the bottom of the page, contact information for SolidWorks Corporation is provided: 300 Baker Avenue - Concord, MA 01742. Phone: 800-693-9000 - International: +978-371-5000 - Email: info@solidworks.com. Copyright © 2003 SolidWorks Corporation. All rights reserved.

Intersection Curves and Splines

One of the keys to any sweep operation is creating the required curves to use as the path or guides. In this example, a decorative piece of wrought iron is modeled by sweeping a circle along a curved path. The path is created by finding the intersection between two reference surfaces.



Thanks to Jason Pancoast at Computer-Aided Products, Inc. for submitting this example.

Stages in the Process

The major steps in this operation are:

- **Create a revolved surface.**
This will use a sketched spline.
- **Create a helical surface.**
This is done by sweeping a line along a straight path, with a helical guide curve.
- **Generate intersection curve.**
Find the intersection between the two reference surfaces. This is the path for the twisted sweep.
- **Sweep one of the “spokes”.**
A circular profile is swept along the intersection curve.
- **Pattern the “spokes”.**
A circular pattern of the swept feature completes the part.

Design Intent

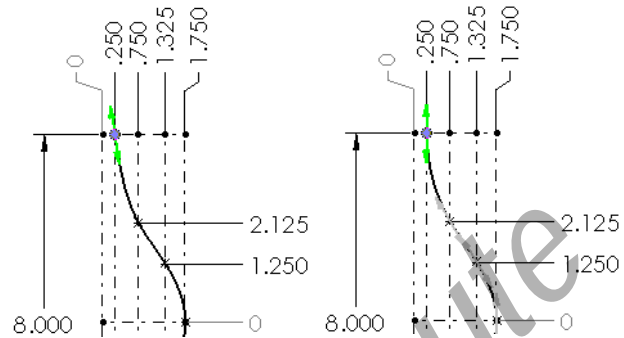
Some of the design intent we have to consider includes:


1. The diameter of the helix has to be equal to or greater than the diameter of the revolved surface.
2. The height of the helix and the height of the revolved surface have to be equal.
3. The helix is defined by its height and the number of turns. The system will calculate the pitch.

6 Vertical relation.

Select the upper end spline handle (arrow) and add a **Vertical** relation.

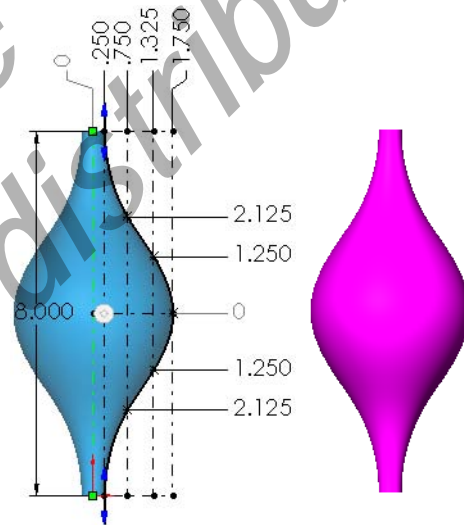
Repeat the procedure for the lower end spline handle.

**7 Revolve surface.**

Select the vertical centerline at the zero datum, and click  on the Surfaces toolbar.

Set the **Angle** to **360°**.

Click **OK**.

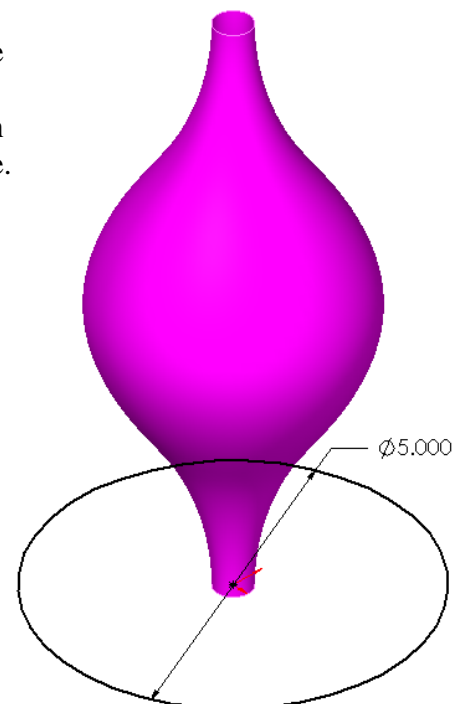
**8 Circle for the helix.**

Open a sketch on the Top reference plane, and sketch a circle.


Dimension it so that it is larger than the diameter of the revolved surface.

Note

An equation can be used to ensure that the diameter of the circle is always greater than the diameter of the revolved surface.



9 Add helix.

With the sketch active, click **Helix/Spiral** .

Insert a helix with the following parameters:

- **Defined by = Height and Revolution**
- **Height = 8.00"**
- **Revolution = 1**
- **Starting angle = 90°**
- **Clockwise**

This helix will be used as the guide curve for a swept surface.

Note

An equation can be used to set the height of the helix equal to the height of the revolved surface.

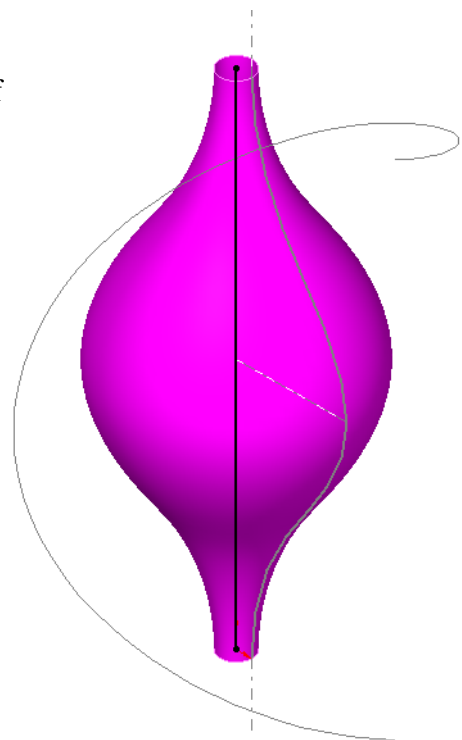


10 Sketch the sweep path.

Open a new sketch on the **Front** reference plane. Show the sketch of the revolved surface.

Select the vertical centerline, and click **Convert Entities** to copy it into the sketch.

11 Exit the sketch.



12 Sketch the sweep profile.

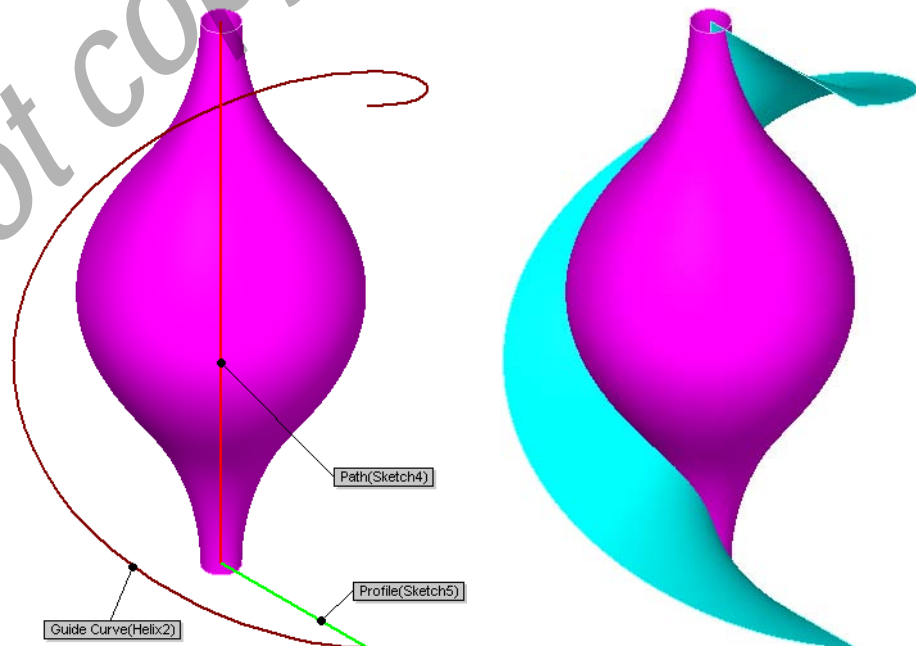
Open a new sketch on the Top reference plane. Sketch a line from the bottom end of the sweep path to the end of the helix.

Important!

Be sure to add a **Pierce** relation between the end of the line and the helix. Add the line *without* capturing a horizontal relation.

**13 Exit the sketch.****14 Sweep a surface.**

Sweep a surface using the path, section, and guide as shown below.



15 Intersection curve.

Open a new **3D Sketch**. Hold down **Ctrl** and select the two surfaces.

Click **Intersection Curve** .

The system generates the intersection in a 3D sketch, and automatically puts you into **Edit Sketch** mode.

16 Exit the sketch.

Exit the 3D sketch and hide the two surface bodies.

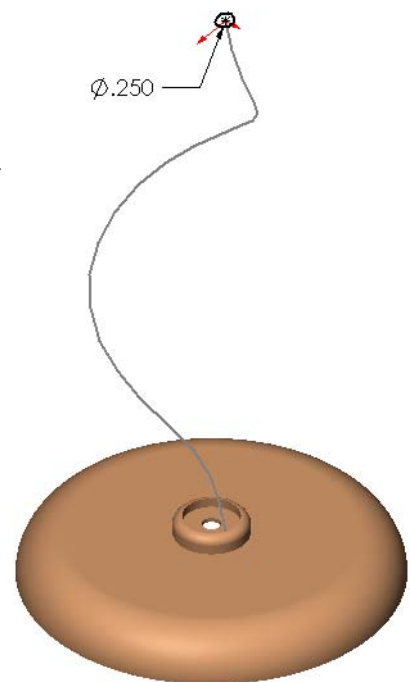


17 Show the solid body.

Right-click **Revolve1** and select **Show Solid Body**.

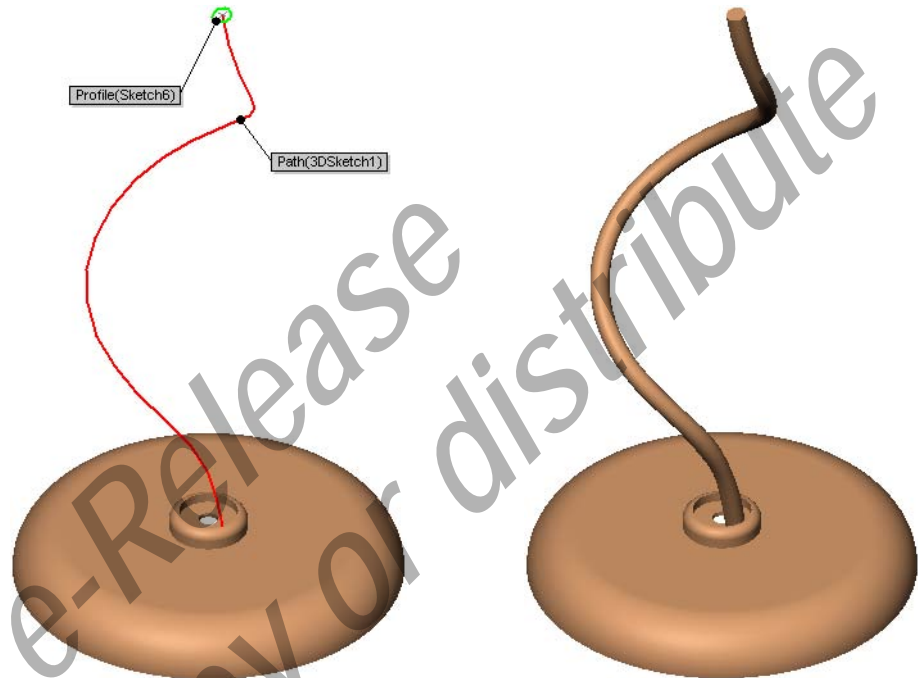
18 Sketch the sweep profile.

Create a plane normal to the top end of the intersection curve, and sketch a **0.25"** circle.

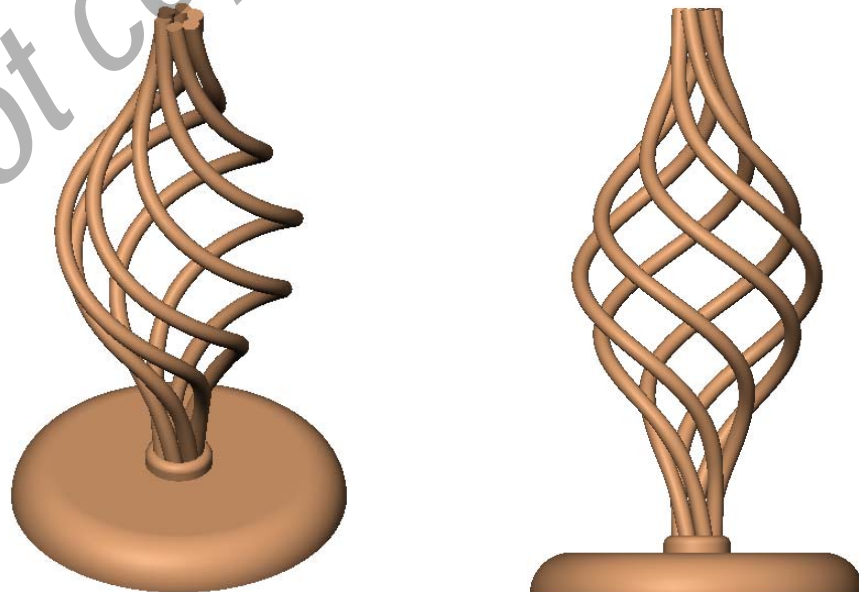


19 Sweep.

When sweeping the boss, use the option **Align with end faces** and **Merge result** to ensure that the boss completely merges with the revolve feature.

**20 Circular pattern.**

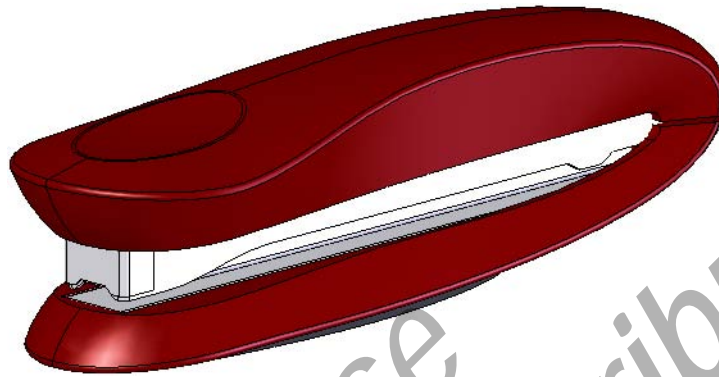
Create a circular pattern with six equally spaced instances.

**21 Save and close the file.**

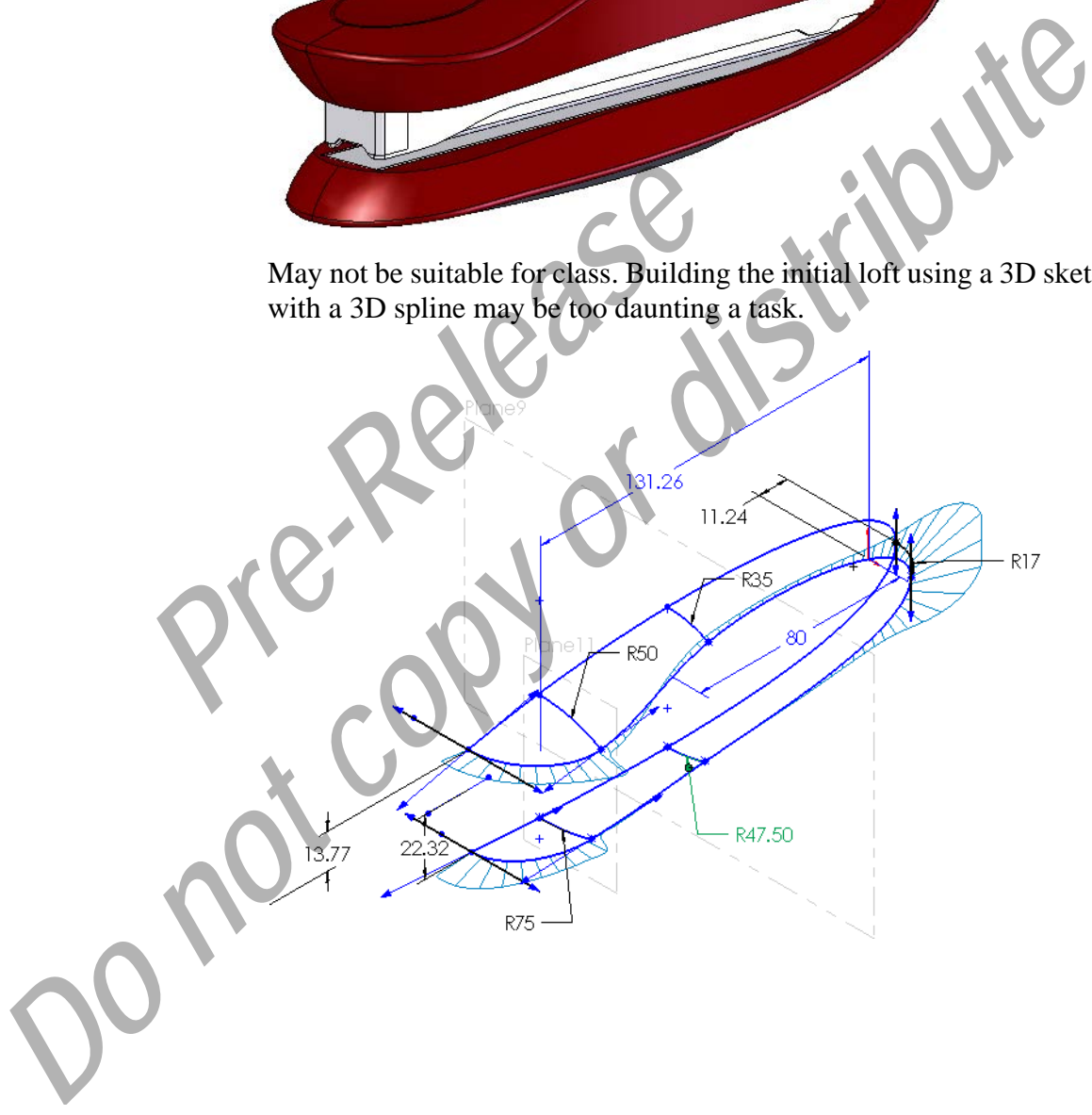
Pre-Release
Do not copy or distribute

**Exercise 20:
Stapler**

Under Construction



May not be suitable for class. Building the initial loft using a 3D sketch with a 3D spline may be too daunting a task.



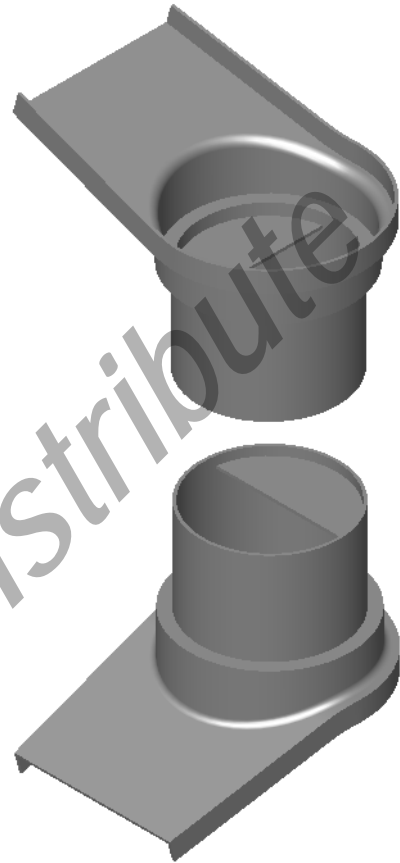
Exercise 21: Surface Modeling

Use surface commands to create a thin walled solid model.

Disclaimer: The primary purpose of this exercise is to give you the opportunity to practice using some of the surfacing commands. In reality, there is no compelling reason to build this part using surfaces. The steps in the procedure may be somewhat contrived so that certain commands will be utilized.

This lab reinforces the following skills:

- Surface extrude, revolve and sweep
- Knit surface
- Surface fillet
- Trim and extend surface
- Thicken surface

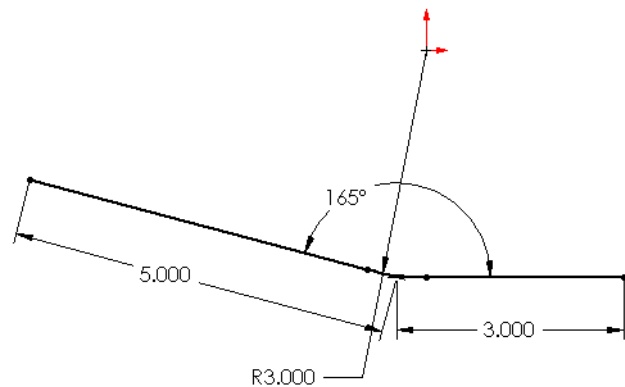


Procedure

Open a new part using the Part_IN template and name it Baffle.

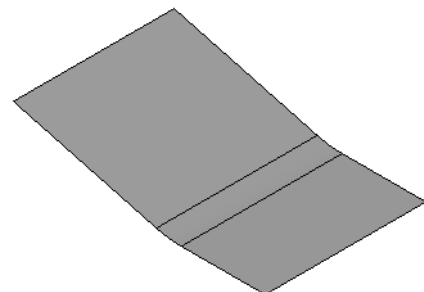
1 Sketch for extrude.

Create a sketch on the Front reference plane using this geometry.



2 Extruded surface.

Extrude a *surface* 5" using the end condition: **MidPlane**.

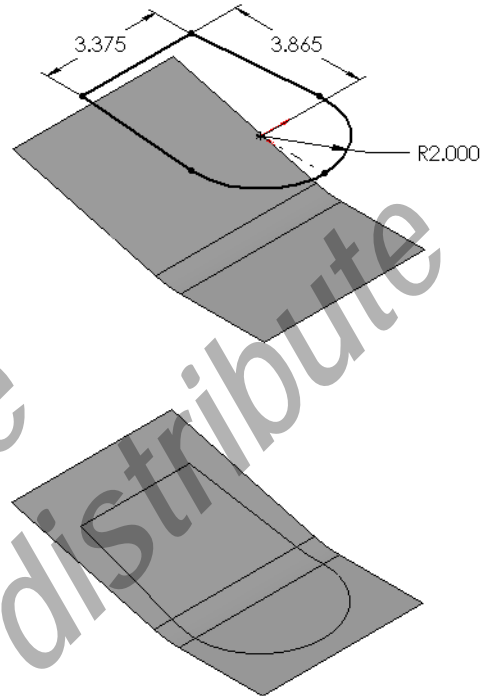


3 Split line.

Create a sketch on the Top reference plane as shown.

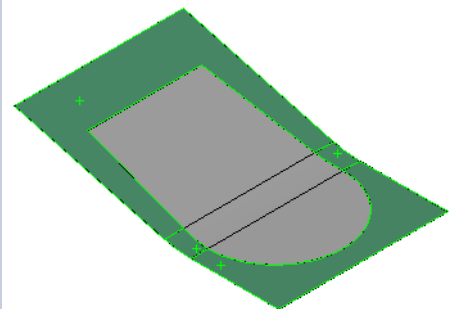
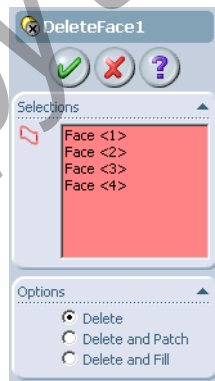
Using the sketch, project a **Split Line** onto the extruded surface.

This creates four additional faces.




Delete Face

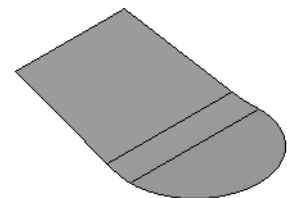
If you click the surface and press **Delete**, you will delete the split line feature. To delete selected faces of a surface, right-click a surface and select **Delete Face**. The PropertyManager will open and you can select the faces you want to delete.




4 Delete faces.

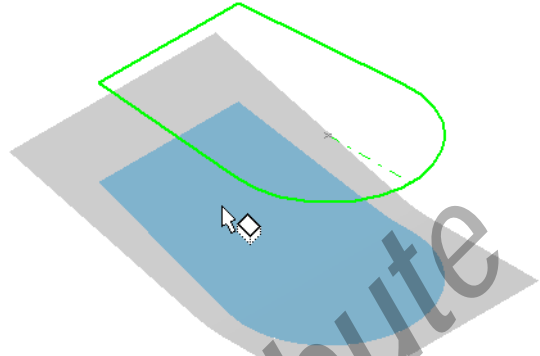
Delete the faces that fall outside the split line shape. Use the **Delete** option, *not Delete and Patch*.

A DeleteFace1 feature  DeleteFace1 is added to the FeatureManager design tree.



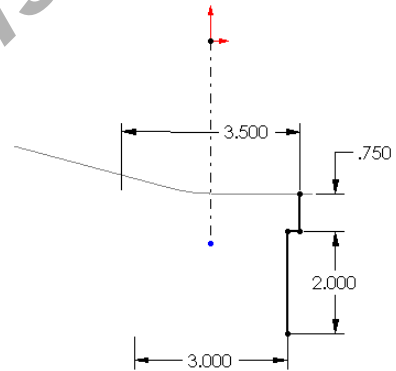
A Different Approach: Trim

Instead of manually splitting the faces of the extruded surface and then deleting them, you could have used **Trim Surface** to achieve the same result in a single operation. You would have clicked **Trim Surface**  and selected the sketch as the **Trim tool**. The technique of splitting the surface and then deleting the unwanted faces was used in this exercise to illustrate how to delete selected faces from a surface.



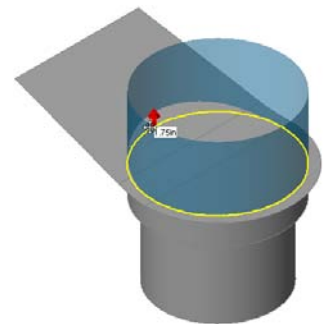
5 Revolved surface.

Sketch on the Front reference plane and revolve the geometry as a surface.



6 Extend surface.

Extend the top edge of the revolved surface so that it extends well beyond the extruded surface.



7 Trim surface.

Trim both the extruded and revolved surfaces, leaving the portions shown.

Tip

Mutual Trim can be used.



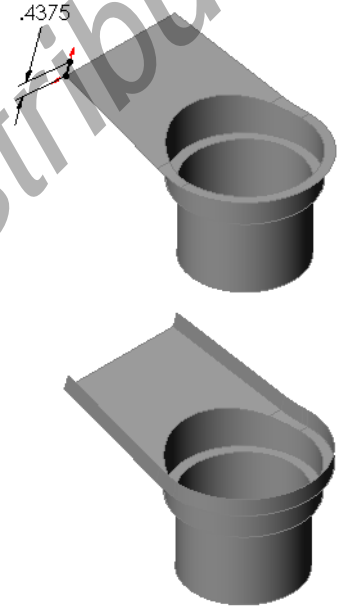
8 Sweep surface.

Create a reference plane normal to the surface edge and sketch a line as shown.

Using the line as the sweep section and the surface edge as the sweep path, create the surface shown.

Tip

Create a composite curve from the edges of the surfaces.




Filleting Surfaces

Surfaces are filleted using the same command as for solids. However, surface behave a little different than solids. The difference depends on whether the surfaces are separate, discrete surfaces, or whether they have been knit together.

Rules

There are a couple of simple rules that make filleting surfaces very straightforward:

- If the surfaces are knit, select and fillet the edge, just like you would with a solid. This is the simplest, most straightforward case.
- If the surfaces are *not* knit, use a **Face Fillet** between the individual surfaces.
- If the surfaces are *not* knit, after they are filleted, the result *is* knit. Filleting will automatically trim the filleted surfaces and knit them together with the fillet, forming a single, composite surface.
- When using a **Face Fillet** on surfaces, preview arrows will appear indicating the side of the surface to which the fillet will be applied. This is because when filleting untrimmed surfaces, there can be multiple solutions. Click **Reverse Face Normal**  to reverse the

arrows. For example, as illustrated on the following page, an intersecting cylinder and curved surface can yield four different results, depending on which side of the surfaces the fillet is located.

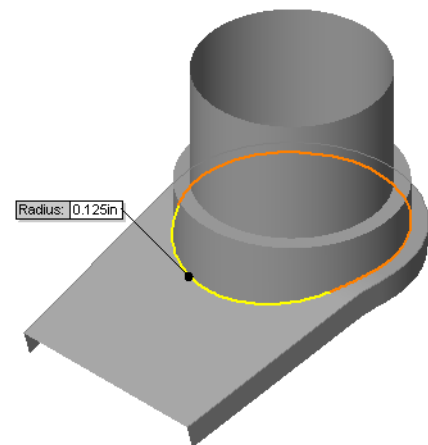


9 Knit surface.

Combine the trimmed and swept surfaces into a single surface using **Knit Surface**.

10 Surface fillets.

Add a fillet of **0.125"** radius to the surface edges as shown in the illustration.



Making it Solid

Similar to a thin feature, you can thicken a surface by adding material on either side or equally, on both sides. If there are no solid features in the model, the thickened surface will be a boss, or more specifically, the first feature. If the surface that you select is a knit surface that encloses a complete volume, you have the option of filling the volume completely.

**Introducing:
Thicken Feature**

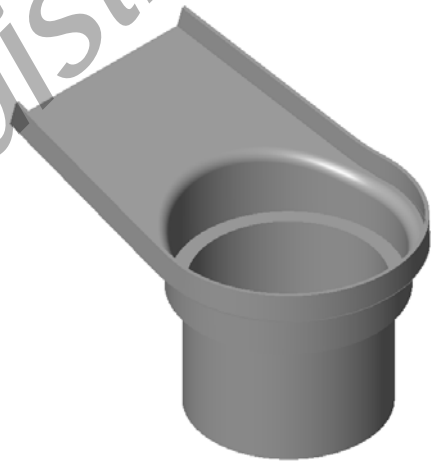
A thickened surface feature can be created as either a boss or a cut feature.


Where to Find It

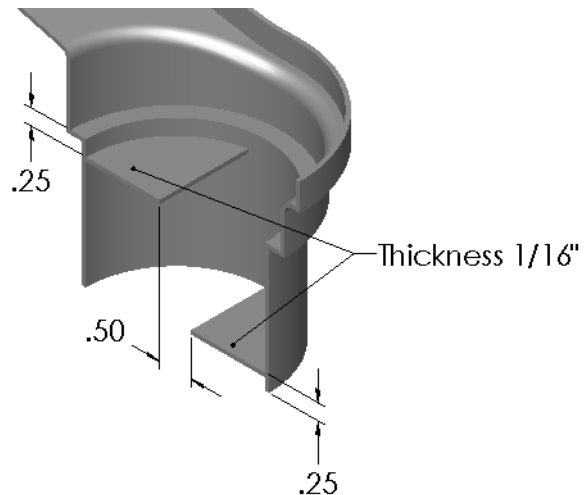
- Click **Thicken**  on the Features toolbar.
- Or click **Insert, Boss/Base, Thicken**.

11 Thicken surface.

Create the first feature by adding **0.0625"** thickness to the *inside* of the surface with **Insert, Boss/Base, Thicken**.

**12 Baffles.**

Create two symmetrical baffles as shown using **Planar Surface**  and **Thicken**. Note that the baffle plates are shown in a sectioned view.

**13 Save and close the part.**

Exercise 22: Halyard Guide

Use surface commands to model the halyard guide.

This lab reinforces the following skills:

- Surface Sweep
- Surface Trim
- Creating Planar Surfaces
- Knit Surface
- Surface Fillet
- Thicken Surface

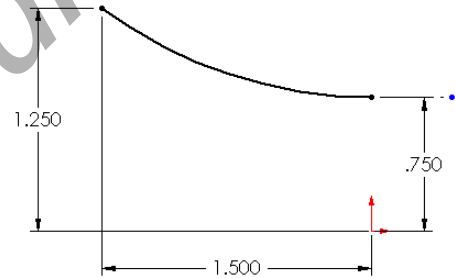


Procedure

Open a new part using the Part_IN template and name it Halyard Guide.

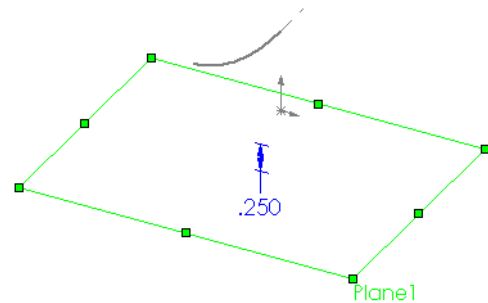
1 Sketch first guide curve.

Open a sketch on the Right reference plane, and create the sketch shown at the right.



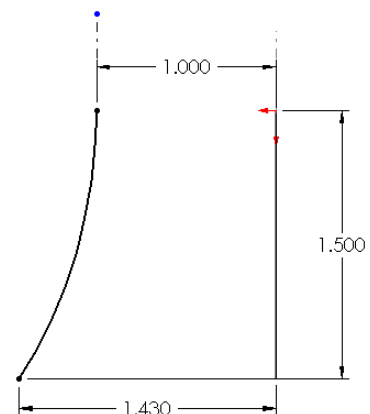
2 Offset plane.

Create a plane offset **0.25"** below the Top reference plane.



3 Sketch second guide curve.

Open a sketch on the offset plane (Plane1 in the illustration above), and create the sketch shown at the right.



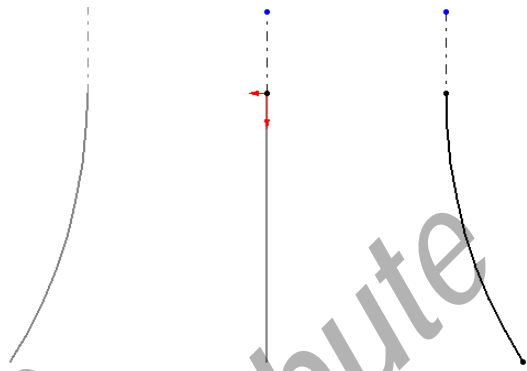
4 Sketch third guide curve.

Open another sketch on the offset plane, and sketch a vertical centerline from the Origin.

Sketch a second vertical centerline whose lowermost end is aligned with the Origin.

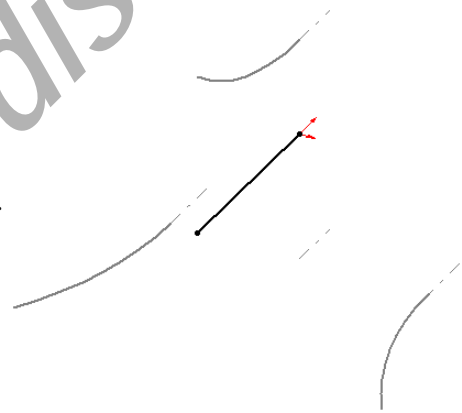
Sketch an arc tangent to the centerline.

Add **Symmetric** relations between the arc in this sketch and the arc in the sketch of the second guide curve.



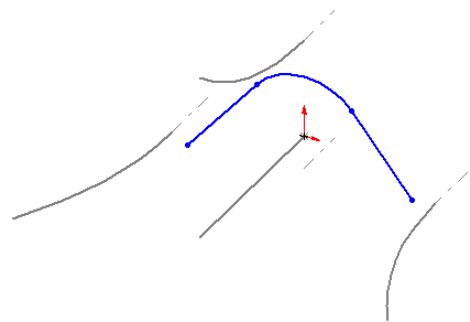
5 Sketch the path.

Open a sketch on the Top reference plane, and sketch a vertical line starting at the Origin. Add a relation so the length of the line is driven by the guide curve sketches.



6 Sketch the sweep profile.

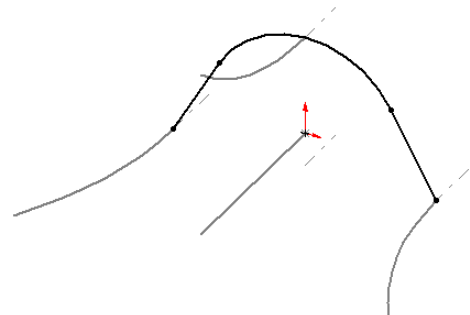
Open a sketch on the Front reference plane, and sketch an arc centered on the Origin. Sketch two tangent lines as shown.



7 Add relations.

Add **Pierce** relations between the ends of the tangent lines and the second and third guide curves.

Add a **Coincident** relation between the arc and the end of the first guide curve. The sketch should be fully defined.



Important!

8 Sweep a surface.

Using the profile, path, and three guide curves, sweep a surface.

Use **Path Tangent** for the **Start tangency type**.

9 Trim the surface.

Trim the swept surface using the Top reference plane as the trim tool. Keep the uppermost portion of the surface.

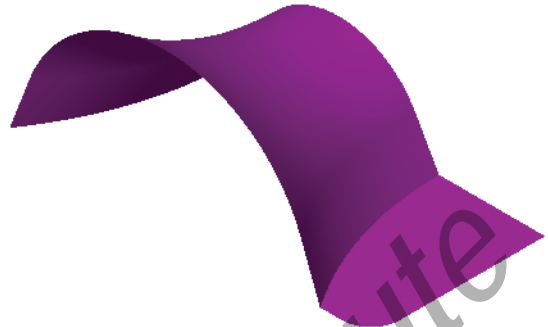
10 Sketch.

Open a sketch on the Top reference plane. Convert the edge of the trimmed surface, and complete the sketch using the dimensions given.



11 Planar surface.

Click **Planar Surface**  to create a planar surface using the active sketch.



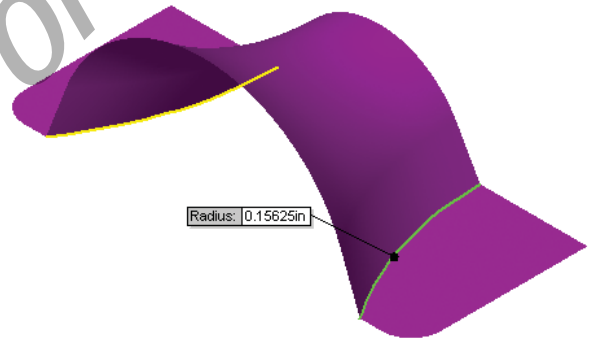
12 Second planar surface.

Mirror the first planar surface to create the second one.



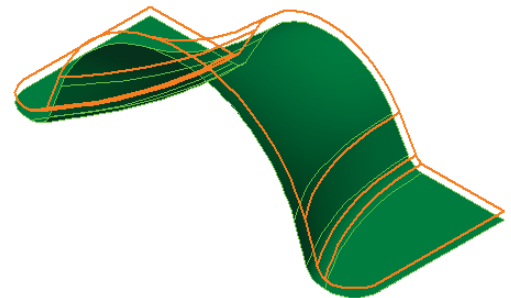
13 Knit the surfaces and fillet the edges.

Knit the three surfaces together, and then fillet the edges shown with a **532"** radius fillet.



14 Thicken.


Create the first feature by thickening the surface **0.08"**. Check the preview to ensure the material is added to the correct side.



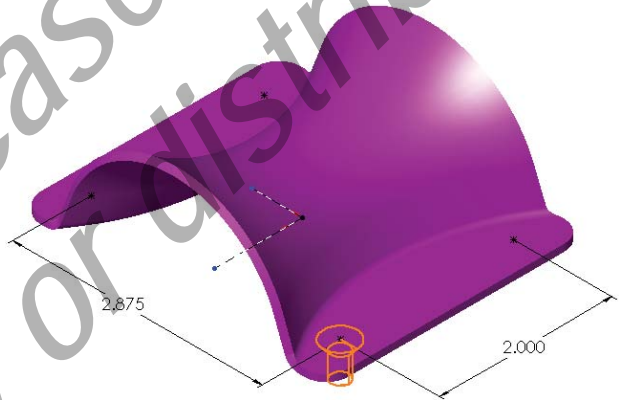
15 Mirror body.

Use **Insert, Pattern/ Mirror, Mirror** to create the other half of the guide and **Merge** result.

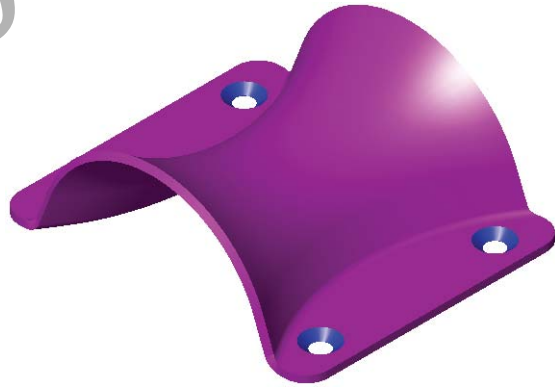
**16 Countersunk hole.**

Add 4 countersunk holes. Select the flat face of the model and click .

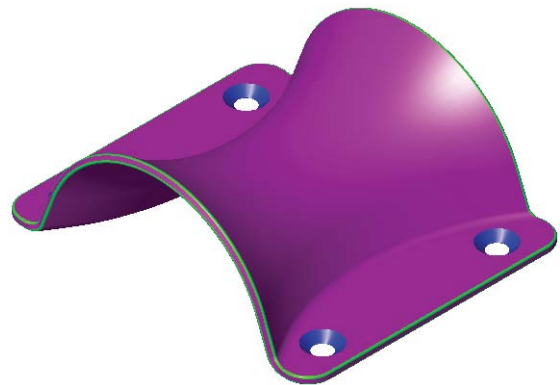
Choose the settings for the description “ANSI #10 Flat Head Machine Screws (100)”.

**Tip**

Use mirroring in the sketch to facilitate creating all four holes in one feature.

**17 Fillet the edges.**

Add a **0.020"** radius fillet to the edges of the part.

18 Save and close the file.

Exercise 23: Using Import Surface and Replace Face

This demonstrates some techniques for modifying imported models. The lab uses a surface imported from a Parasolid (x_t) file. The surface is moved to a new position and used to replace a face in the solid.

This lab uses the following skills:

- Delete Face
- Import Surface
- Move Surface
- Replace Face

1 Open existing file.

Open the existing Parasolid file named `Button.x_t`. It is found in the `Replace Face` folder.


Note

If you are prompted to select a template, choose `Part_IN`.

The face to be replaced is highlighted in green.

2 Delete faces.

Before we can replace the face, some fillets have to be deleted. Click

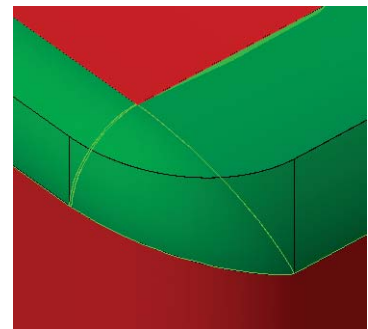
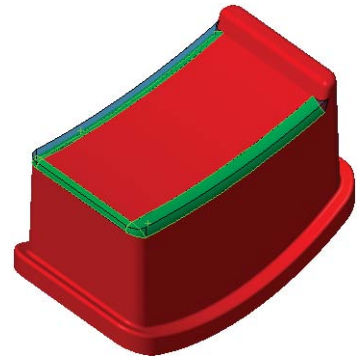
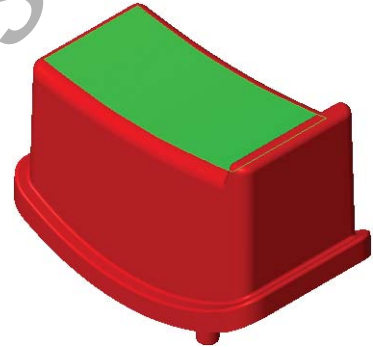
Delete Face  on the Surfaces toolbar. Select the faces shown.

Be sure to zoom in on the corners. There are some small faces there.

Tip

Drag a selection box around the corners to be sure to select the small faces.

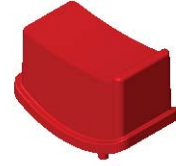
Select the option **Delete and Patch**, and click **OK**.




3 Import surface.

Import a surface into the part using **Insert, Features, Imported**. Select the Parasolid file named New Surface.

The surface color was changed for clarity.

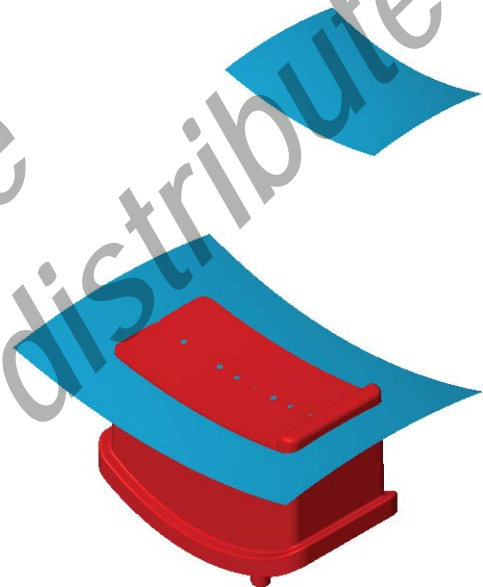
**4 Move the surface.**

Click **Insert, Features, Move/Copy**, or click **Move/ Copy Bodies**  on the Surfaces toolbar.


Use the **Translate** option.

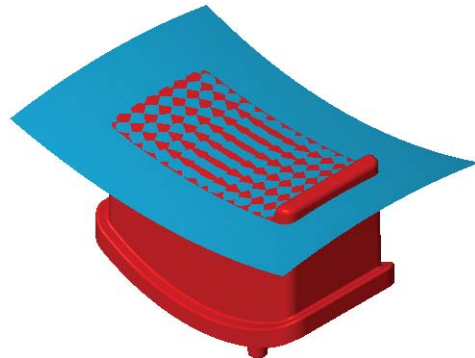
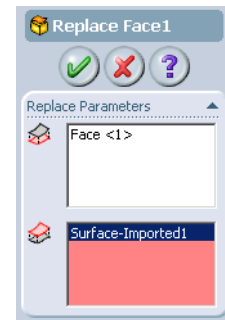
Enter **2.5"** for **Delta Y**.

Click **OK**.

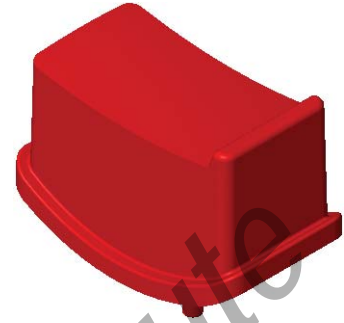
**5 Replace face.**

Replace the top face of the part with the imported surface.

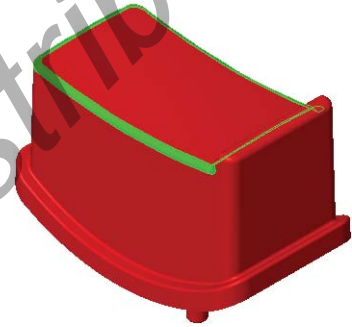
Click **Insert, Face, Replace**, or click **Replace Face**  on the Surfaces toolbar.



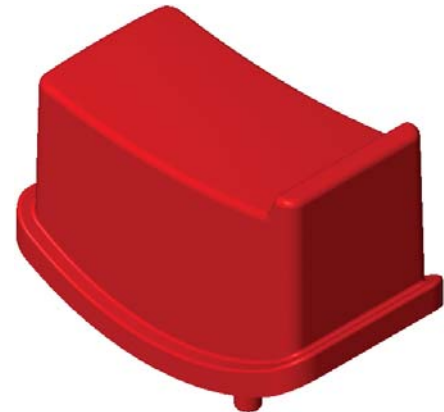
- 6 Hide the surface.**
Right-click the surface, and select **Hide Surface Body**.



- 7 Fillet.**
Add a **0.025"** fillet as shown.



- 8 Save and close the file.**



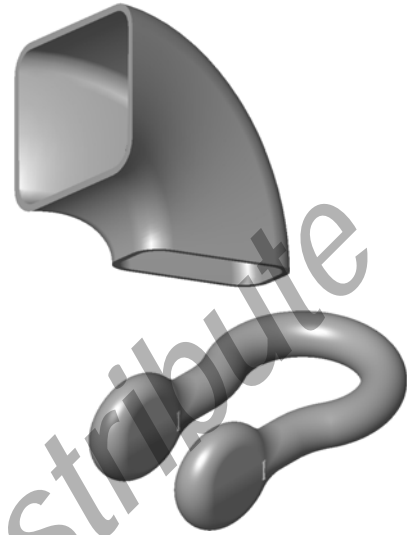
Exercise 24: Using Surfaces

This lab includes two small exercises in using surfaces to create solids.

- The first one creates a solid by lofting between two surfaces.
- The second uses the method of knitting surfaces to combine multiple bounding surfaces into a solid.

This lab uses the following skills:

- Lofting between surfaces
- Importing an IGES file
- Repairing missing surfaces
- Knitting surfaces

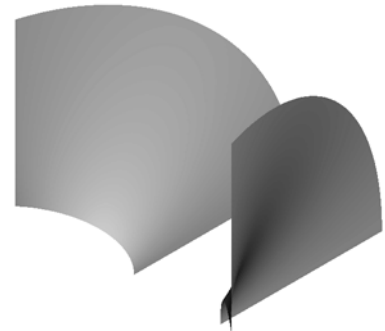


Lofting Between Surfaces

Lofting can be accomplished using sketches, faces or surfaces. In this example, lofting is performed between two surfaces to form a solid.

1 Open the part.

Open the existing part named LOFT_SURF. The part consists of two imported surfaces.

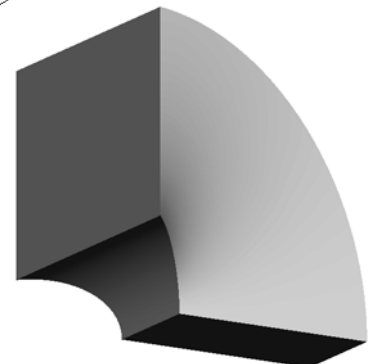
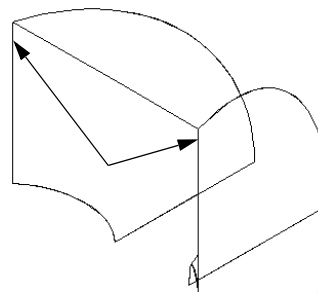


2 Insert loft.

Using **Insert, Boss/Base, Loft**, select the two surfaces as the **Profiles** of the loft.

Pick the surfaces near mating corners, like you would using sketches.

The result is a single solid body.



3 Fillets and shell.

Add fillets of radius **0.5"** and a shell of **0.125"** to complete the body.

4 Save and close the file.



Repair and Knit Surface

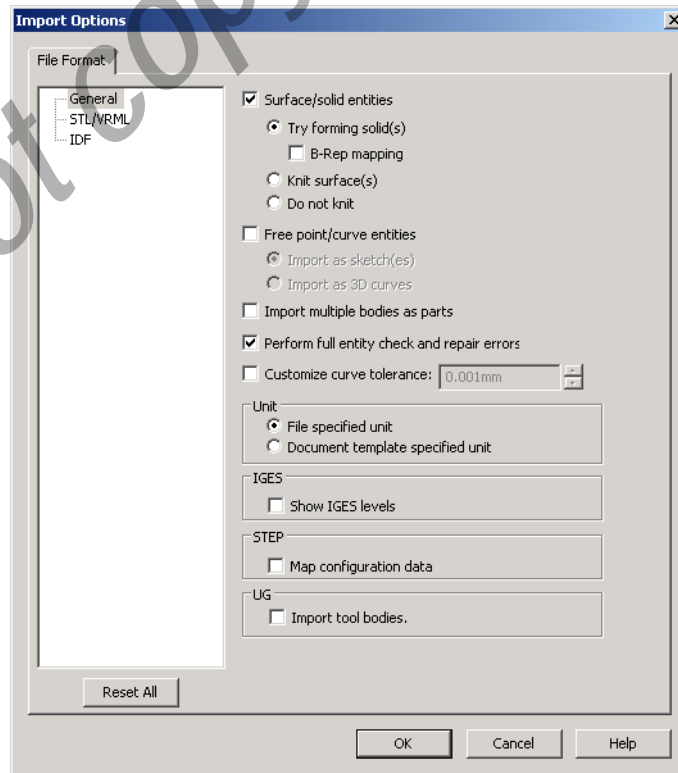
Knit surface allows you to combine several surfaces into a single, larger surface or in some cases, a solid. For a solid, the surfaces must comprise a closed volume. If surfaces are missing from the imported data, the gaps must be filled.

1 Import an IGES file.

Click **File, Open**, or click **Open** . Set **Files of type:** to IGES Files (*.igs;*.iges). Select the file Surface Repair.IGS.

2 Click Options.

Verify that the option **Try forming solid(s)** is selected and click **OK**.

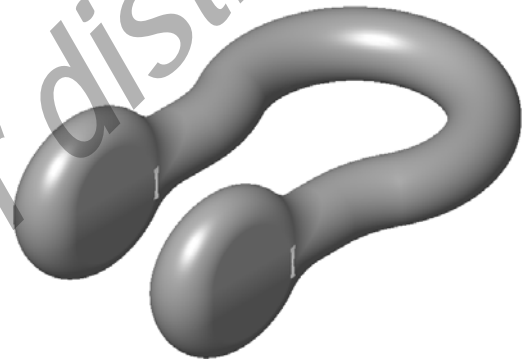
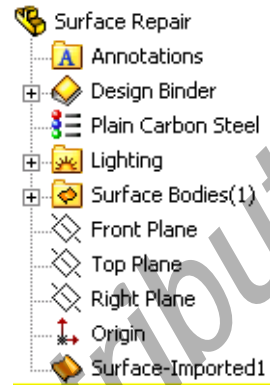


3 Click Open from the Open dialog.

If you are prompted to select a template, choose Part_IN.

4 Results.

The individual surface patches are knit into a single imported surface. However, there are some gaps.

**5 Click Filled Surface .**

Set **Edge settings** to **Tangent**.

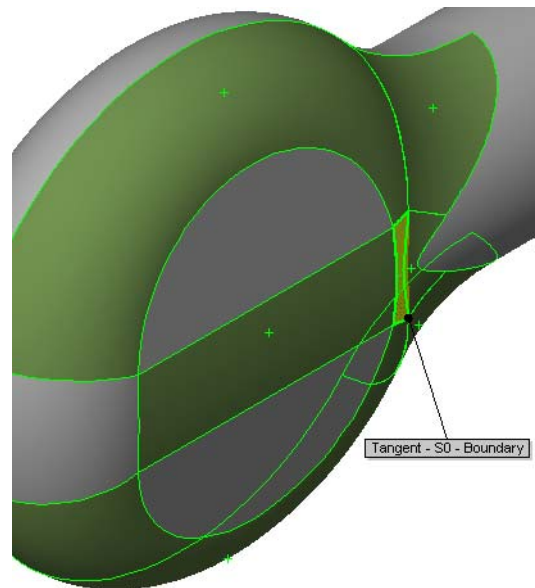
Select the **Apply to all edges** check box.

6 Select edges.

Right-click one of the edges of the opening, and select **Select Open Loop**.

Select the **Merge result** check box.

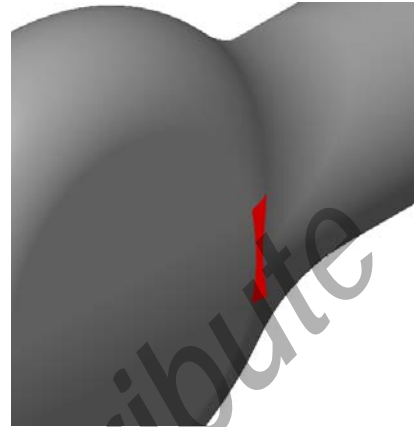
Click **OK**.



7 Results.

A surface patch is created to fill in the opening. It is shown here in a different color for illustration purposes.

Since the **Merge result** option was selected, the new patch has automatically been knit to the existing surface.



8 Repeat.

Repeat this process for the remaining three openings.

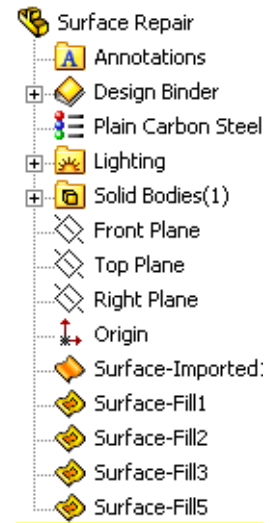
Important!

When doing the *last* opening, also select the option **Try to form solid**. This will thicken the resulting knit surface into a solid.

9 Results.

Although the graphics look the same, a solid has been formed. Only by looking at the **Solid Bodies** folder can you tell the model is now a solid.

10 Save and close the file.



Exercise 25: Inserting a Picture and Combining

This lab demonstrates a technique for using image files in a sketch. The lab uses a JPEG file that is “traced” in a sketch using splines and other geometry.

This lab uses the following skills:

- Insert picture
- Splines
- Combine

Procedure

Open a new part using the Part_IN template and name it Fork.

1 Images.

Create a new sketch on the Front reference plane.

From the **Tools** menu, choose, **Sketch Tools, Sketch Picture** and insert the FORK SIDE . jpg image file.

Set the **Width** (the X dimension) to **6”**.

Using the Top plane and the FORK TOP . jpg file, create another sketch.

Size the image to the same width.

2 Front sketch.

Edit the sketch with the FORK SIDE . jpg image and “trace” the lower edge of the image with a spline.

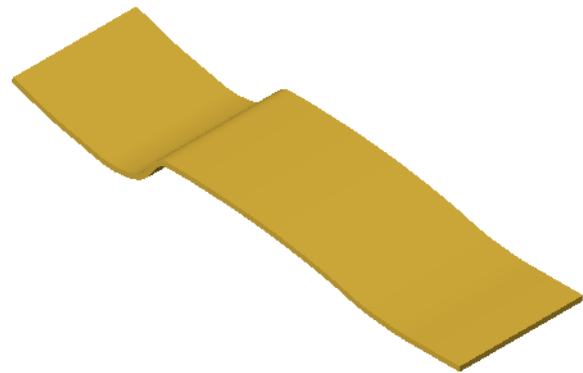
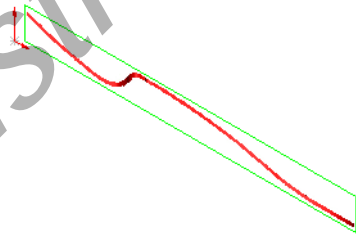
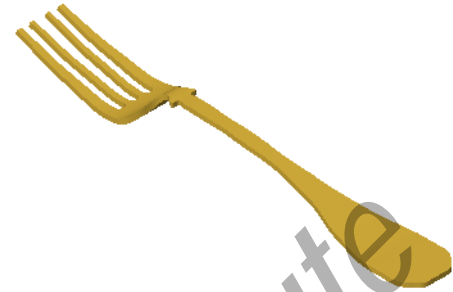
Note

Zoom in after creating the spline to move or add more spline points. The sketches can be fully defined at a later time, if necessary.

3 Extrude.

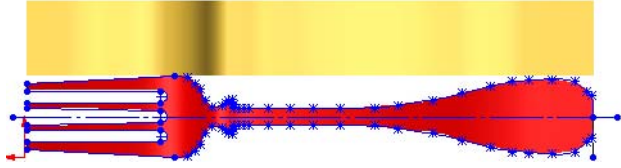
Using a thin feature, extrude the sketch with a thickness of **0.0625”**.

The image file in the sketch can be suppressed.



4 Top sketch.

Using lines, arcs and splines, trace the image shape. Use symmetry where it is appropriate.



5 Extrude and combine.

Extrude the boss feature and combine the solid bodies into one.

6 Save and close the part.



Pre-Release
Do not copy or distribute

Pre-Release
Do not copy or distribute

Lesson 5 Core and Cavity

Upon successful completion of this lesson, you will be able to:

- Apply shrinkage to resize a plastic part.
- Analyze a model to check the draft angles of model faces.
- Fix un-drafted faces on the plastic part.
- Determine parting line edges to build parting line surfaces.
- Create shutoff surfaces.
- Create parting surfaces.
- Create interlock surfaces.
- Create a tooling split.
- Utilize multiple parting lines and parting surfaces
- Create side cores, lifters, and core pins.
- Model an electrode.

Pre-Release
Do not copy or distribute

Case Study: A Simple Two Plate Mold Design

Designing mold tooling is a multi-step process. Once you create the model for which you want to design a mold, you need to follow several steps to create the core and cavity. This case study demonstrates how to create a simple two plate mold for this plastic dust pan.



Rendered With Real View Graphics

Stages in the Process

The key steps in this lesson are listed below. Each of these topics describes a section in the lesson.

- **Fixing File Translation Errors.**
Many times mold designers will need to build a mold for a plastic part that was designed in another CAD system. Use the **Import Diagnostics** command to find and fix errors on translated CAD models.
- **Check the plastic part for correct draft.**
A solid model of a plastic part is provided to create the mold tooling. The model must be drafted correctly or the molded part will not eject from the tooling. Use the **Draft Analysis** command to determine if the part can eject from the mold.
- **Fix the un-drafted faces.**
When a plastic part is not drafted properly, the mold designer must fix the plastic part model to ensure that the part ejects from the mold.

- **Scaling the plastic part.**

When the hot injected plastic cools during the molding process, it hardens and shrinks. Before creating the mold tooling, the plastic part is scaled slightly larger to compensate for plastic shrinkage.
- **Establish the parting lines.**

Parting lines must be established on the plastic part. The parting lines are the edges of the plastic part from which the parting surfaces are created. They are the boundary edges between the core and the cavity surfaces.
- **Create shut-off surfaces for holes in the plastic part.**

After the parting lines are established, shut-off areas on the plastic part are sealed with surfaces. A shut-off area is where two pieces of mold tooling contact each other to form a hole or a window in the plastic part. Holes molded in the plastic require a shut-off surface. Not all plastic parts require this.
- **Create the parting surfaces.**

Once the shut-off surfaces are created, parting surfaces can be created. The parting surfaces are projected away from the parting line edges all around the parting lines' perimeter. Typically, these surfaces are perpendicular to the direction of pull, although there are other techniques to model them. These surfaces are used to define and separate the tooling boundaries.
- **Develop the interlock surfaces.**

Around the perimeter of the parting surfaces, tapered surfaces are created to help lock the tooling components together when the mold is closed. These tapered surfaces are tapered 5° from the direction of pull. This angle keeps the steel from galling as the mold closes and opens. Not all tooling requires these special surfaces. If you do create interlock surfaces, these surfaces are knitted to the parting line to help separate and establish the boundaries between the tooling bodies.
- **Separate the tooling into separate solid bodies.**

The last step of the tooling design is to separate the solid bodies of the mold tooling from the plastic part and from the parting surfaces.
- **Side Cores and Lifters can be established.**

When necessary, an optional design step is applied to separate "side cores" and "lifters" from the core and cavity bodies. This creates tooling that does not travel in the same direction as the primary parting direction of the tooling.
- **Electrode modeling.**

Surface modeling techniques are demonstrated to model electrodes that are used to burn complicated geometry on the tooling. The **Move Face** command is used to quickly clear back an electrode.


Problematic File Translations

A common problem for mold designers is data translation errors. Sometimes a plastic part is designed in one brand of CAD system and then sent to another brand of CAD system for the tooling to be designed in. Many times the translation is not successful. To successfully design the tooling, the translated data must be free of gaps and errors. The SolidWorks application has tools to help find and repair these problematic areas on translated models. The dustpan in this case study is not a water-tight solid model. The tooling cannot be created until the model is fixed and becomes a solid body.

Introducing: Import Diagnostics

The **Import Diagnostics** command is used to fix problems with the geometry on an imported body or surface body.

Where to Find It

- Click **Import Diagnostics**  on the Tools toolbar.
- Click **Tools, Import Diagnostics...**
- Right-click the imported body in the Feature Manager design tree and choose **Import Diagnostics**.

1 Open the file named **Translated_Dustpan**.

This part was imported from an IGES file. It could not be knit into a solid body.

Right-click **Surface-Imported1** and click **Import Diagnostics** from the popup menu.

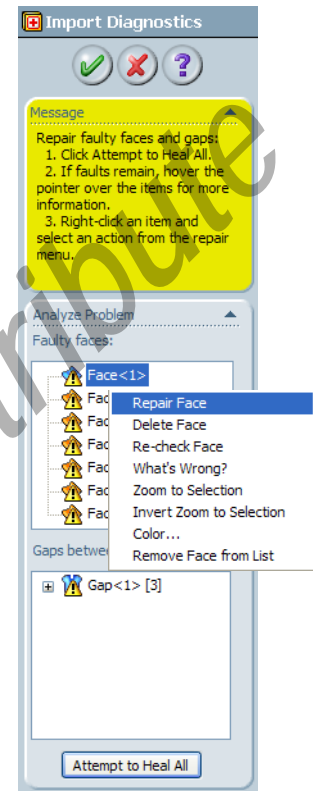
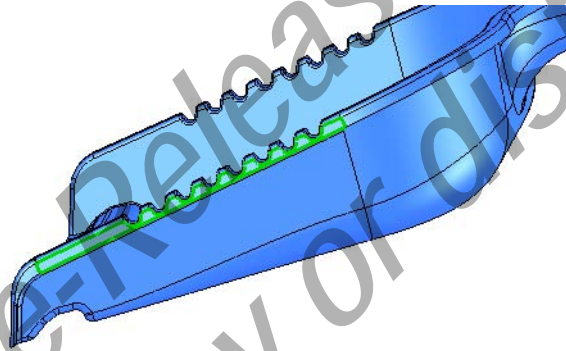


2 Examine the results.

Right-click the first face in the **Faulty face** list.

The popup menu allows several options for working with faulty faces.

Click **Zoom To Selection** from the popup menu.



3 Click What's Wrong.

Right-click again the first face in the **Faulty face** list and click **What's Wrong** from the popup menu.

The message indicates that this face is overlapping other faces in the model. This is preventing the model from knitting into a water tight solid body.

4 Inspect the Gap.

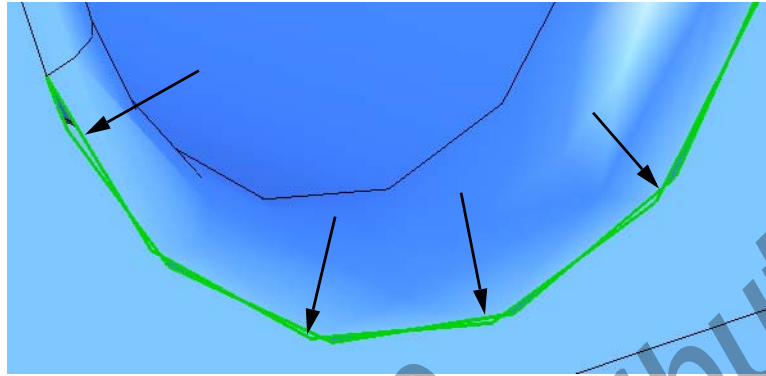
Right-click Gap<1> in the **Gaps between faces** list.

Click **Zoom To Selection** from the popup menu.

Inspect the highlighted edges on the model.

Zoom in closer if necessary.

Notice the gaps where these edges come together.



5 Repair the face.

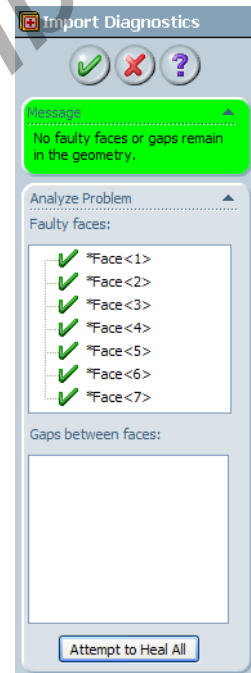
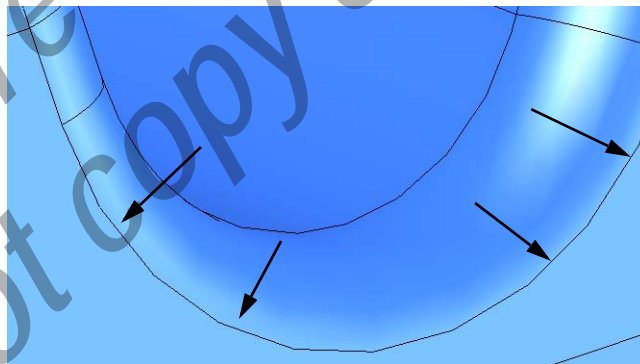
Right-click the first face in the **Faulty face** list.

Click **Repair Face** from the popup menu.

Inspect the edges.

The edges between the faces are more precise and the gap has been closed.

Notice also the model is now a water tight solid body.



Tip

Use the **Attempt to Heal All** button to automatically fix the problems on an imported model. If the results are not satisfactory, use the commands on the popup menus for the **Faulty faces** list, or the **Gaps between faces** list to fix the problems individually.

Analyzing Draft on a Model

To create the tooling for a mold, the plastic part must be engineered and drafted properly so that it ejects from the surrounding tooling. To analyze the draft on a molded part, use the **Draft Analysis** command to help find draft and design errors.

Checking the Mold-ability of a Plastic Part

If the faces of the plastic part are not drafted properly, a plastic part may be scored or even get stuck in the tooling when it is ejected from the mold. To determine if a part is moldable, an analysis of all the faces on the model must be done to verify that the part was properly drafted, and also to see if enough draft was applied.

In the following diagram, a cupcake is used as a simple illustration to explain the *direction of pull*. Notice that the bottom of the cupcake is drafted. That is done so that the cupcake does not get stuck in the pan. The same idea is used on plastic parts. They must be drafted properly or the part may get stuck in the surrounding tooling. To run a **Draft Analysis** on a plastic part, the term *direction of pull* needs to be explained.

Determining the Direction of Pull

The *direction of pull* is the direction in which the plastic part is going to be ejected from the tooling. A simple way to think of this, is to think of a cup cake and the direction it falls out of the cupcake pan. The direction vector of the top plane of the pan represents the direction of pull. The direction of pull is also analogous to the “*path of least resistance*.” Keeping this in mind, mold designers design a mold so that the plastic part easily comes out of the mold with the least amount of tooling as possible. This will help keep the cost of the mold down.



Cupcake and cupcake pan.

The direction of pull is shown by the arrow.

Note

Complicated molds can have more than one direction of pull. This situation will be covered in *Case Study: Multiple Parting Directions* on page 311.

Introducing: Draft Analysis

The **Draft Analysis** command is used to make sure that all faces on the plastic part have enough draft on them. When the **Draft Analysis** is run, all of the faces in the plastic part are traversed and colors are assigned to the faces to show the amount of draft, and to specify the piece of tooling that the face should be molded by.


The **Draft Analysis** displays:

- Faces that are not drafted.
- Faces that are drafted incorrectly.
- Faces that do not have enough draft on them.
- Faces that straddle the parting line.
- Faces that have draft, but include areas with insufficient draft.

Where to Find It

- Click **Draft Analysis**  on the Mold Tools toolbar.
- Or click **Tools, Draft Analysis**.

6 Check the part for proper draft.

Click **Draft Analysis**  on the Mold Tools toolbar.

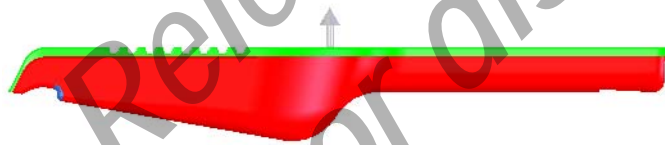
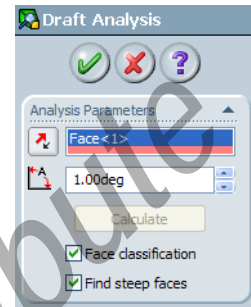
Select the top planar face of the dust pan for the **Direction of Pull**.

Set the **Draft Angle** tolerance to **1°**.

Select the **Face Classification** check box.

Select the **Find Step Faces** check box.

Click the **Calculate** button.



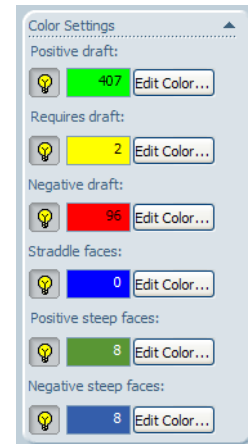
The direction of pull is the top face of the dust pan.


Draft Analysis Colors

In the PropertyManager for the **Draft Analysis** command, six **Color Settings** are used to display what the draft looks like on the model.

The default colors are shown in the illustration at the right and they are used and described in this example. Click **Edit Color** to change any of the colors.

The color settings are described in detail over the next pages.

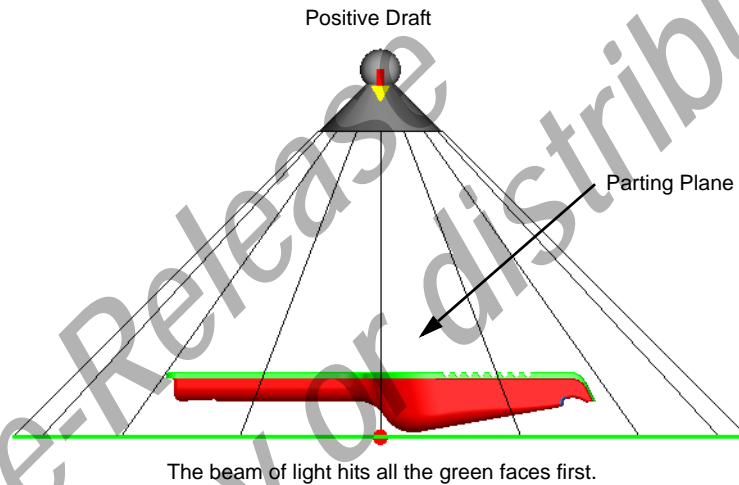
**Tip**

In the **Draft Analysis** command, use the  **Show/Hide** buttons in the **Color Settings** area to hide or show the faces with different types of draft. Sometimes these surfaces are very small and hard to find on the part when all the surfaces are visible.

Positive Draft

Positive draft displays the faces that can be ejected away from the positive side of the parting plane.

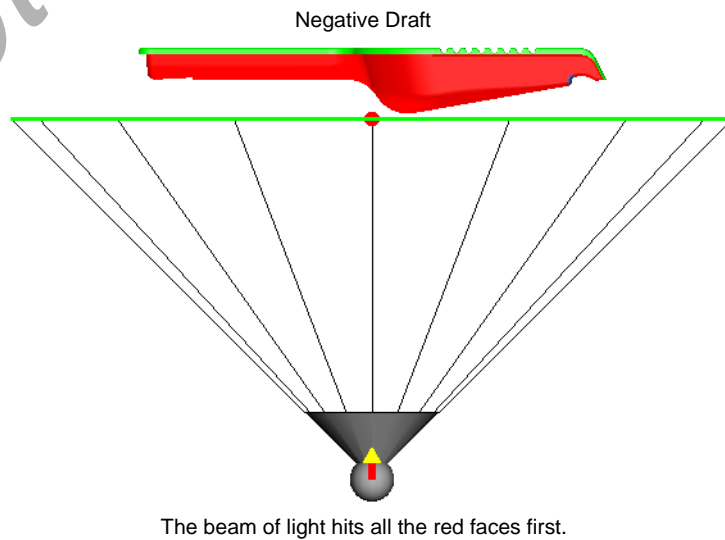
Imagine shining a beam of light at the plastic part, parallel to the direction of pull. If the light can illuminate the face, it has positive draft. The green faces in the illustration below all get hit with the light beam. They have positive draft. The red faces do not get hit with the light beam because the green faces block the light from the surfaces under the part.



Negative Draft

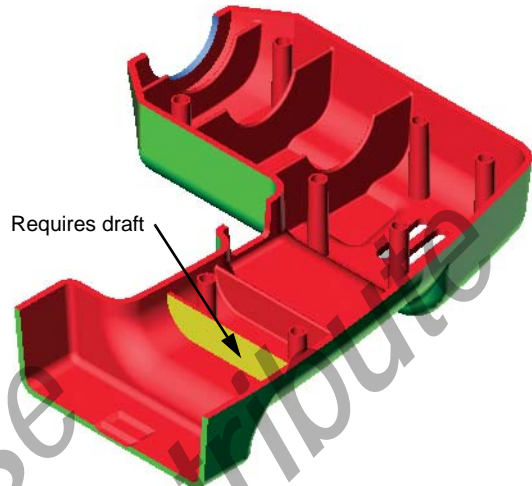
Negative draft displays the faces that can be ejected away from the negative side of the parting plane.

Imagine a light beam shining in the opposite direction. All of the red faces of the model now get hit with the light beam. They have negative draft.



Requires Draft

When **Draft Analysis** identifies a face that has less than the required draft angle, that face is colored yellow and is classified as **Requires draft**. The face either has no draft, or needs to have more draft added. The plastic part must be adjusted to ensure that it ejects properly from the tooling.

**Note**

The plastic part in the previous diagram shows a face that requires draft. The next case study will analyze this plastic cordless electric drill cover and explain how to deal with faces that do not have the correct amount of draft on them.

Straddle Faces

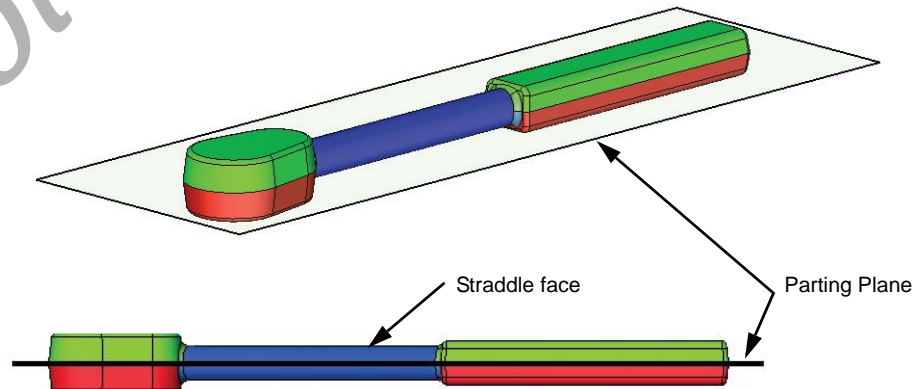
Straddle faces are faces that straddle the parting line. You must split straddle faces into two pieces to separate the tooling surfaces. Splitting the face can be done manually with the **Split Face** command or it can be done with the **Parting Line** command, automatically, by clicking on the **Split Faces** option.

Note

There are no straddle faces in the dustpan example.

An example of a part with a straddle face, the Forged Ratchet Body, is shown in the following diagram.

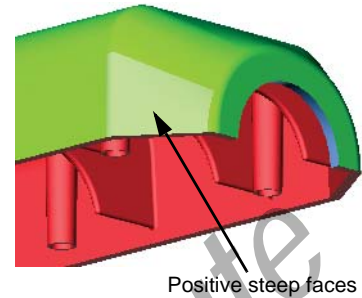
This face must be split into two faces where the parting plane bisects it.



One piece will be formed by the cavity and the other will be formed by the core.

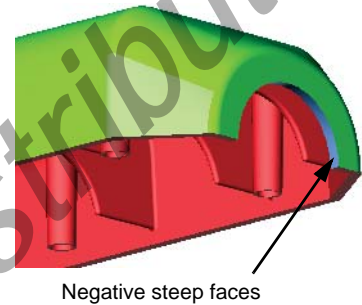
Positive Steep Faces

These faces include *portions* of the face that have less than the required draft. If the *entire* face had less than the required draft, it would be classified as **Requires draft**. These faces are found on the positive side of the mold.



Negative Steep Faces

These faces include *portions* of the face that have less than the required draft. These faces are found on the negative side of the mold.



Scale the Plastic Part to Allow for Shrinkage

Mold tooling is manufactured slightly larger than the plastic part produced from the mold. This is done to compensate for the shrinkage that results after the hot, ejected plastic cools. Before the tooling is created from the plastic part, mold designers scale the plastic part larger to account for shrinkage. Different plastics, geometry, and molding conditions all have an effect on the shrink factor.

Scale the Plastic Part

You can use the **Scale** command to grow or shrink the model geometry. Scale the part slightly larger so that when the molded part cools and shrinks, all of the molded features are the correct size.

Introducing: Scale

The **Scale** command applies a scaling factor. The scaling can be **Uniform** or vary in the **X**, **Y**, and **Z** directions. In this example, the body is uniformly scaled larger by 5%.


Note

The **Scale** command changes the size of the part, but it does *not* change the dimensions of preceding features.


Important!

When scaling a part with non-uniform scaling, remember that cylindrical holes may no longer be cylindrical. You may have to make changes to the model to compensate for this before you create the mold tooling.

Where to Find It

- From the **Insert** menu, click **Features, Scale**.
- Or, click **Scale**  on the Mold Tools toolbar.

7 Scale the plastic part.

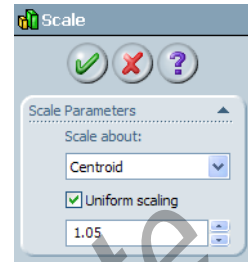
Click **Scale**  on the Features toolbar.

The scaling type can be either about the **Centroid**, about the **Origin**, or about a **Coordinate System**.

Select **Centroid**

Select the **Uniform Scaling Factor** check box.

Set the **Scale Factor** to **1.05** (5% larger). Click **OK**.



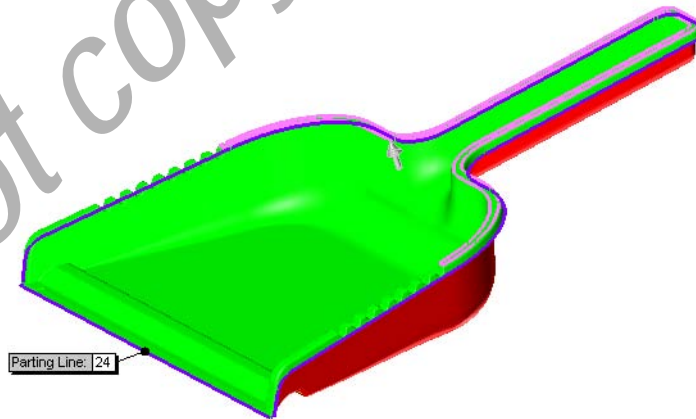
Determine the Parting Lines

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

Establish the Parting Lines

Now that the part is properly drafted and scaled, the parting line can be established. After another **Draft Analysis** is run, the parting lines are typically identified as the edges on the model that share two faces, classified as positive and negative draft.

In the next illustration, the cavity surfaces (positive draft) are green and the core (negative draft) surfaces are red. Any edge that shares a red and green face is a parting line edge.




Introducing: Parting Lines

The **Parting Line** command allows the designer to automatically or manually establish the parting edges. Later, this **Parting Line** feature will be used to create parting surfaces. In case the plastic part geometry was changed, the **Draft Analysis** is done as part of the **Parting Line** command.

- Click **Insert, Molds, Parting Lines**.
- Or, click **Parting Lines**  on the Mold Tools toolbar.

8 Establish the parting lines.

Click **Parting Lines**  on the Mold Tools toolbar.

Click in the **Direction of Pull** field.

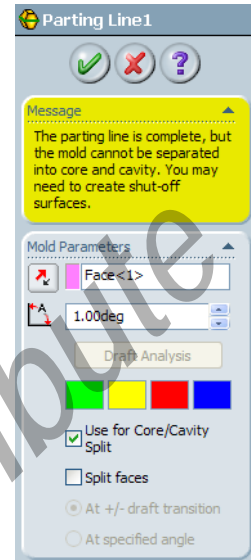
Select the top face of the dustpan.

Set the **Draft Angle** to 1°.

Click the **Use for Core/Cavity Split** option.

Clear the **Split Faces** option.

Click **Draft Analysis**.

**Tip**

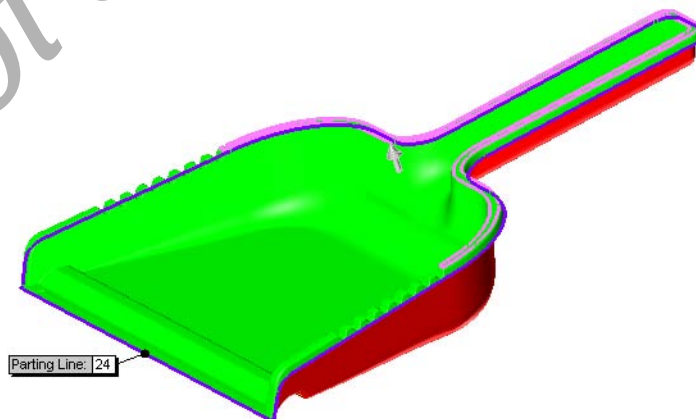
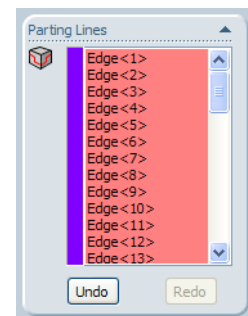
Use the **Split faces** option to automatically split straddle faces into two pieces before selecting your parting line.

9 Selecting all of the parting edges.

When the **Draft Analysis** is complete, all of the edges that are shared by green and red faces are automatically selected and added to the **Parting Lines** list.

Click **OK**.

The parting line feature is added to the model.




**Note**

There may be more than one parting line feature in a model. The option **Use for Core/Cavity Split** is used to specify which parting line should be used as the primary parting line for the **Tooling Split** command.

Manual Selection Of Parting Lines

In this example, the parting line edges are automatically selected when the **Parting Lines** command is run. Because this is a simple parting line boundary, the edges are automatically added to the **Edges** list in the **Parting Line** PropertyManager.

Sometimes the parting line may be more complex and the software will not automatically find the parting line. When this happens, use the edge selection buttons that appear next to the **Edges** list box to manually select the parting line.

-  **Add selected edge.**
-  **Select next edge.**
-  **Zoom to the selected edge.**
- **Undo**
- **Redo**

Tip

Remember that the **Select Tangency**, **Select Loop**, and **Select Partial Loop** commands can all be used when establishing parting lines. Access these commands from the shortcut menu when right-clicking in the graphics area.

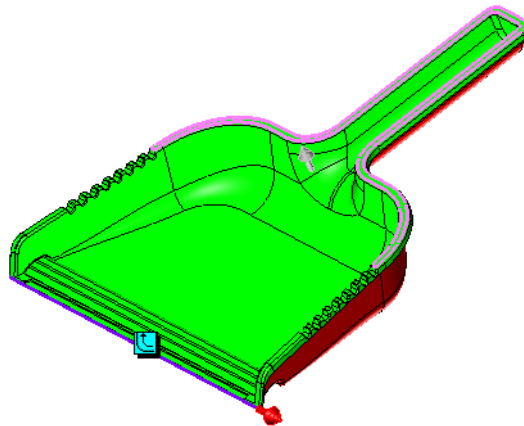
10 Edit the parting line feature.

Edit the **Parting Line1** feature. Right-click in the **Parting Lines** list box and select **Clear Selections** from the shortcut menu.

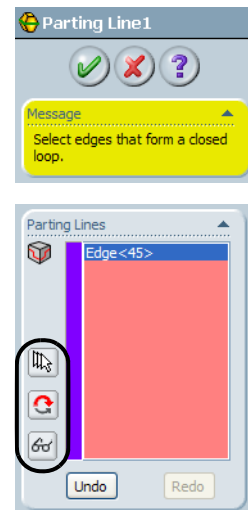
11 Select an edge on the model.

Select an edge on the model that shares a red and a green surface.

Notice that the message at the top of the PropertyManager has changed to instruct the designer to select the edges that represent the parting line. Notice also that the edge selection buttons appear in the **Parting Lines** list box.






When you select the edge, the edge is added to the list box.



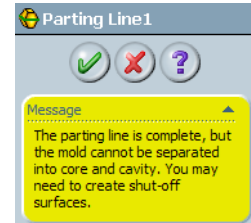
Manual Selection of Parting Line Edges

Designers will use the buttons next to the **Parting Lines** list to select the next edge, or pick another candidate. The message at the top of this PropertyManager provides feed back to let them know when they have a continuous parting line selected.

Additionally, the next edge that is a candidate for the **Parting Lines** list will be marked with a 3D arrow in the model view. If the next edge is acceptable, click **Add selected edge** . If the next candidate is not satisfactory, use the **Select next edge**  button to select a different edge that shares the same end point as the last edge added to the list.

A designer can use the **Zoom to the Selected Edge**  button, and the model view continues zooming into the next edge selection automatically as they continue selecting edges.

If a completed loop is selected, the message on the PropertyManager changes to inform the designer that they have completed selecting a closed loop that can be used for a complete parting line.



12 Cancel the dialog.

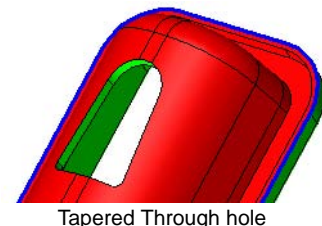
Click **Cancel** to discard the changes.

Note

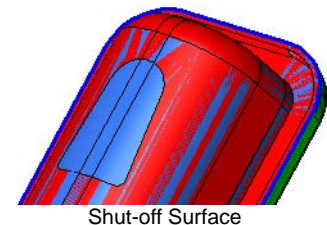
A complete loop is not required to create a parting line feature. Parting lines can be incomplete and finished later in the mold design process.

Shutting Off Holes or Windows in the Plastic Part

After the parting lines are established, the next step is to determine any open molding areas on the plastic part that need **Shut-off Surfaces**. An open molding area is either a hole or window in the molded part where two pieces of tooling touch coincidentally to form the hole. The illustration shows a simple shut-off surface. It is created on the smaller end of the tapered window.



The **Shut-Off Surfaces** command automatically shuts off the open holes in a plastic part.



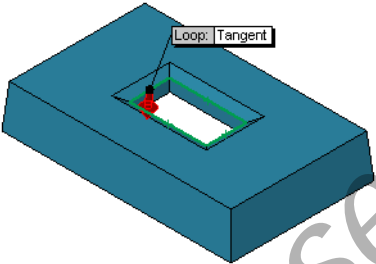
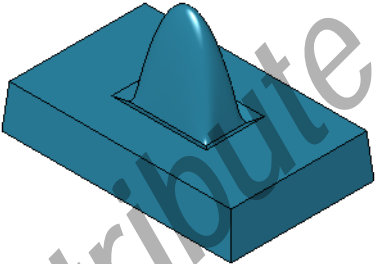
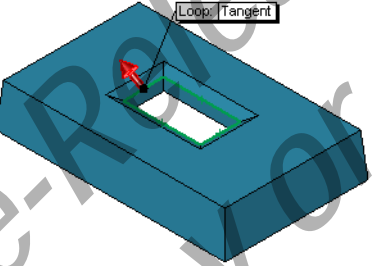
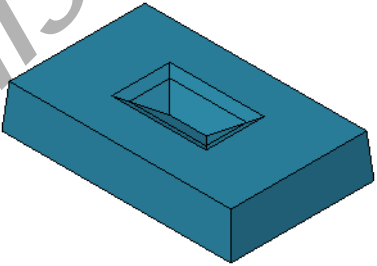
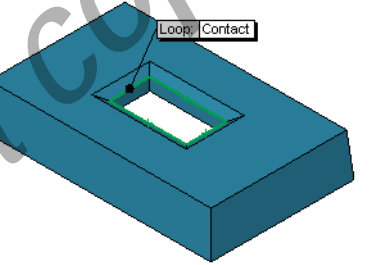
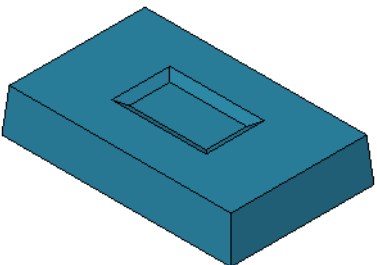
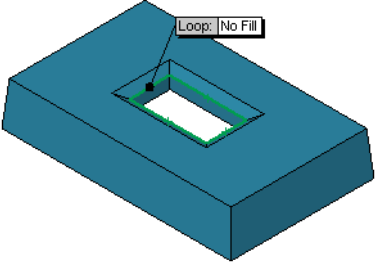
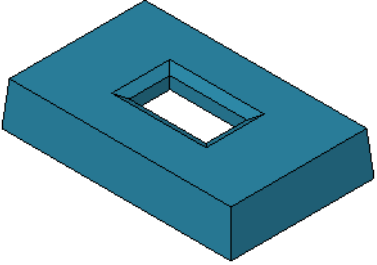
Shut-off Surface Patch Types

Callouts are used to choose what type of shut-off surface to create.

- Tangent
- Contact

■ **No Fill**

The following table shows the results of the different patch types.

Tangent patch type – tangent to faces <i>below</i> the loop	
	
Tangent patch type – tangent to face <i>above</i> the loop	
	
Contact patch type	
	
No Fill patch type	
	


Introducing: Shut-off Surfaces

The **Shut-off Surfaces** command allows designers to automatically or manually shut-off any open holes or windows in the plastic part. The shut-off surfaces are stored as a feature in the FeatureManager design tree. Shut-off surfaces are later used to help separate the mold tooling surfaces. To select the different options click the *callout* in the graphics area. The patch types can be globally changed by selecting the appropriate type from the **Reset All Patch Types** options.

Where to Find It

- Click **Shut-off Surfaces**  on the Mold Tools toolbar.
- Or, click **Insert, Molds, Shut-off Surfaces**.

13 Create the shut-off surfaces.

Click **Shut-off Surfaces**  on the Mold Tools toolbar.

Rotate the part and zoom in on the area in the handle that requires a shut-off surface.

If necessary, manually select the loop shown in the diagram.

Set the **Patch Type** as **Tangent**.

Toggle the tangency arrow if needed.

Click **OK**.

The shut-off surface is created.

The surface bodies for the cavity and core are created and organized in the `Surface Bodies` folder.



Note

Established parting line features may be used to define the boundaries for the shut off surface.

Automation

A great deal of automation was built into the process of creating tooling for a molded part. Some examples already examined include:

- Automatically selecting the edges for the parting line based on the common edges between the positive and negative draft faces.
- Automatically knitting two surface bodies – one for the core and one for the cavity.

Note

If shut-off surfaces were not needed, the knit core and cavity surface bodies would be created when a complete **Parting Line** feature is added. In this example, shut-off surfaces *are* required, and the **Shut-Off Surfaces** command knit the surfaces and organized them in the `Surface Bodies` folder.

Later in this chapter, the **Tooling Split** command is used to automatically create the tooling. This command requires that three surface body folders, each with the appropriate surface bodies in them, must exist. The folders are:

- Cavity Surface Bodies
- Core Surface Bodies
- Parting Surface Bodies

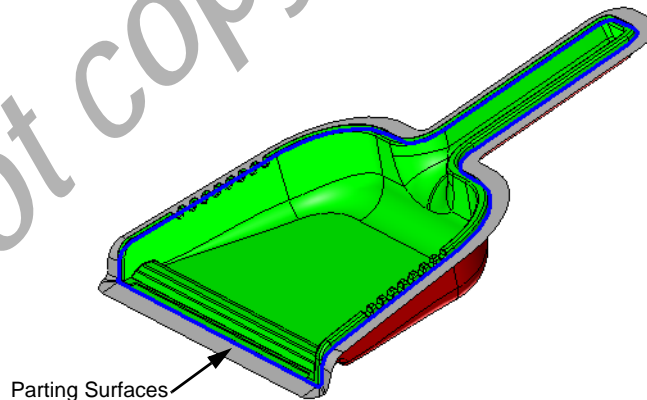
At this point two folders and surfaces exist that represent the core and cavity surfaces. This third required surface is created with the **Parting Surfaces** command.

Modeling the Parting Surfaces

The next step is to create parting surfaces around the perimeter of the parting lines. Currently the surfaces are organized into two core and cavity surface bodies. The **Shut-off Surfaces** command separated the knit surfaces. Another knit surface body, the **Parting Surface**, needs to be added.

Parting Surfaces

Parting surfaces are ribbon-like, knit surface bodies that generally extrude perpendicular to the pull direction, away from the parting line edges on the plastic part. This parting surface helps split the tooling blocks where the cavity and core faces touch around the perimeter of the plastic part. Use the **Parting Surfaces** command to create this knit surface geometry that separates the mold tooling blocks.



Introducing: Parting Surfaces

The **Parting Surfaces** command allows designers to automatically create parting surfaces. The **Parting Surfaces** command creates surfaces that extrude from the parting line in a perpendicular direction to the direction of pull. The parting surfaces form the splitting surfaces that separate the mold cavity faces from the mold core faces.

Where to Find It

- Click **Parting Surfaces**  on the Mold Tools tool bar.
- Or, click **Insert, Molds, Parting Surfaces**.

Smoothing the Parting Surface

When creating mold tooling, remember that the tooling is manufactured directly from the design. There are several processes that take place in the machining of the tooling. Two of these processes are CNC milling and EDM machining.

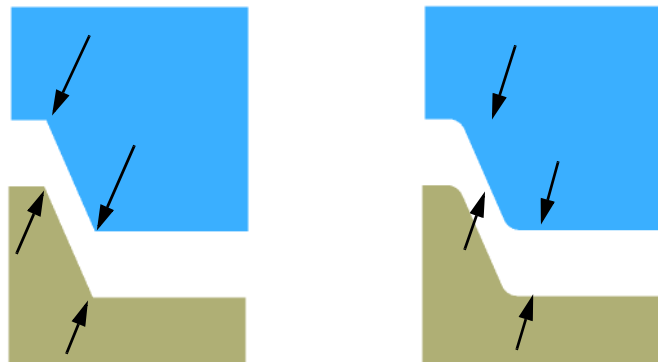
CNC milling requires end-mills with rounded tips, called ball-mills, to machine the 3D shapes into the metal. When there are tight or sharp transitions in the 3D shape, a ball-mill cutter may not fit in the area to machine it. When an end-mill cannot fit into the more complicated geometry transitions, another manufacturing process called EDM machining is used to eliminate the material that the end-mill could not remove. EDM machining is a very time consuming process. The more EDM machining you can eliminate from the manufacturing process, the faster the mold can be manufactured.

To address this, the **Parting Surfaces** command includes a **Smoothing** option to adjust the parting line geometry, minimizing sharp corners that are inaccessible to the ball-mill. Although it may not completely remove the sharp areas, it can drastically cut down on the amount of EDM machining needed to create the tooling.



This Ball-mill Does Not Fit In The Corner Smoothing Allows Ball-mill Into Corner

Another benefit of smoothing the parting surfaces is to eliminate the sharp edges on the parting surfaces. Sharp edges on the tooling wear out faster than rounded corners. The smoothing process allows longer lasting tooling to be designed.



Sharp Edges Wear Faster

Rounded edges last longer

14 Create the parting surfaces.

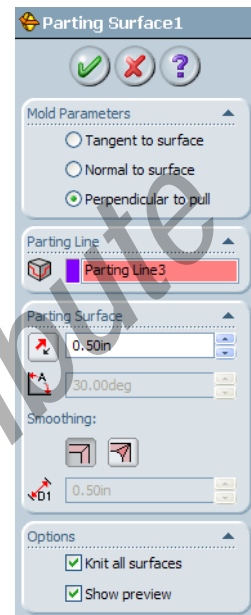
Click **Parting Surfaces**  on the Mold Tools toolbar.

Select **Perpendicular to pull** from the **Mold Parameters** options.

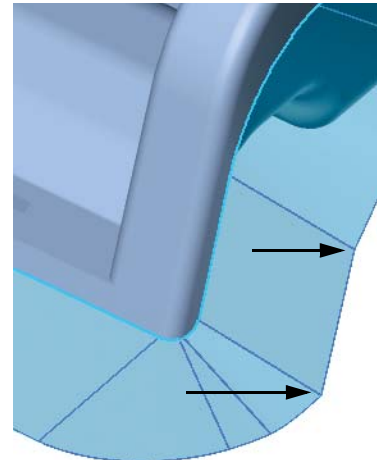
Set the **Distance** to **0.5"**. The default **Smoothing** option is set to **Sharp**.


Select the **Knit all surfaces** and the **Show preview** check boxes.

The preview of the parting surfaces is displayed around the perimeter of the parting lines.

**15 Examine the sharp corners.**

Zoom to view the sharp corners on the parting surfaces.

**16 Use the smoothing option.**

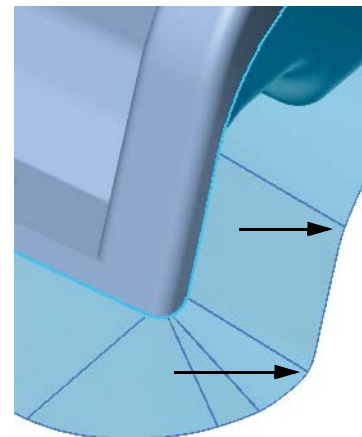
Under the **Smoothing** options, click **Smooth** . Set the **Distance** to **0.25"**.

Now examine the same area.

The sharp corners have been rounded.

This option provides better machining conditions, and makes the parting surfaces last longer when the mold is in production.

Click **OK**.



Interlocking the Mold Tooling

The next step is to create interlock surfaces around the perimeter of the parting surfaces. The interlock surfaces are tapered from the parting surfaces, usually at a 5° angle. Tapered surfaces help the mold seal properly and they help guide the tooling into place when the mold closes. The interlocks also keep the tooling aligned when the mold is closed. This ensures that the tooling does not shift, creating uneven, unpredictable wall thicknesses on the parts created in the mold. The 5° taper also keeps the steel that forms these surfaces from galling when the mold is open or shut.

Automatic Interlock Surface Creation

When using the **Tooling Split** command, select the **Interlock surfaces** option to automatically create the interlock surfaces. This works well when the parting line doesn't include any sudden radical jogs that require extra surface modeling to fill.

Creating the Mold Tooling

All the surfaces required to create the mold tooling are now organized in the correct surface body folders. You can now create the mold tooling.

Automatic Tooling Separation

The **Tooling Split** command automates the creation of the solid bodies that represent the cavity and core of the mold tooling. In a few mouse clicks, the tooling bodies are created and organized as multi-body solids in the **Solid Bodies** folder.


Introducing: Tooling Split

The **Tooling Split** command creates solid bodies from the tooling blocks based on the surfaces in the **Surface Bodies** folder.

The core surface bodies and the parting surface bodies are combined and used to cut a solid block that encompasses these surface bodies.

Simultaneously, a mold cavity is created by combining the cavity surface bodies with the parting surface bodies. These surface bodies are cut from the same solid block.

Where to Find It


- Click **Tooling Split**  on the Mold Tools toolbar.
- Or, click **Insert, Molds, Tooling Split**.

17 Create an offset plane.

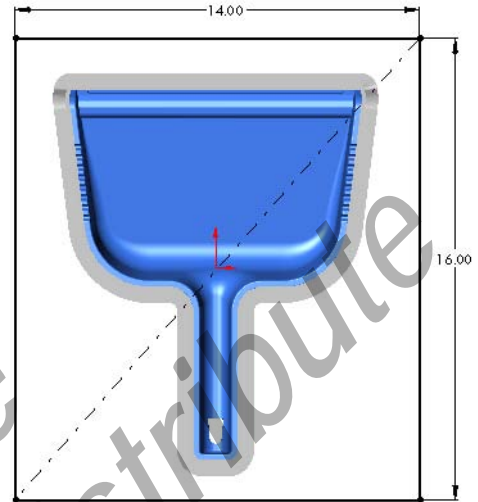
Select the planar top face of the dustpan and create an offset plane **1.00"** above it.



18 Split the tooling.

Click **Tooling Split**  on the Mold Tools toolbar.

Create a rectangular sketch around the perimeter of the dustpan on the plane that was just created.

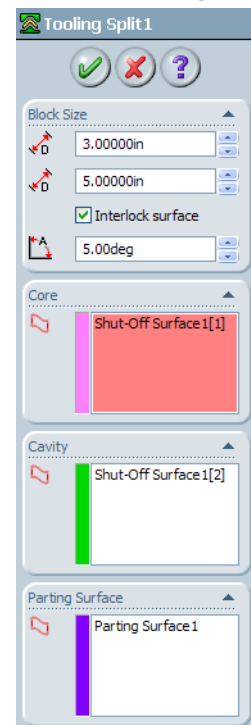
**19 Use the Interlock Surface option.**

Set the block sizes to **3.00"** and **5.00"**.

Select the **Interlock Surface** check box.

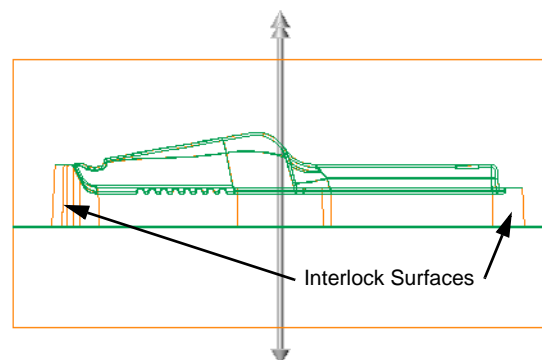
Set the **Draft Angle** to **5°**.

Notice that the surface bodies for the core, cavity and parting surface have automatically been placed in their appropriate list boxes.

**20 Examine the preview.**

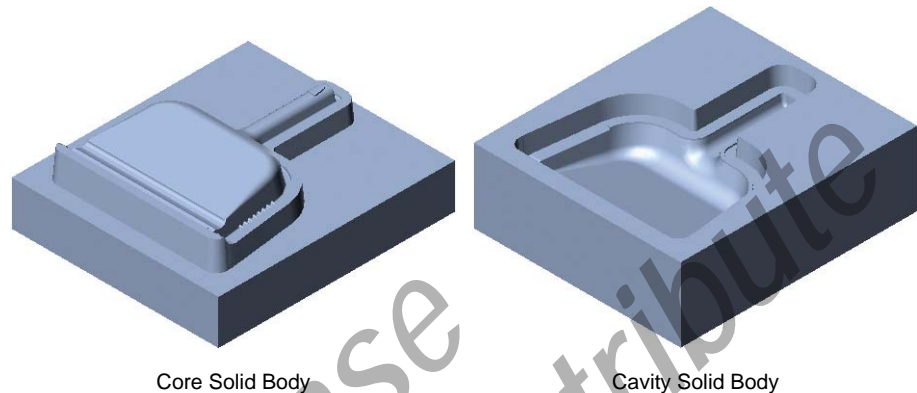
Notice the interlock surfaces are generated automatically.

Click **OK**.



21 Hide all surface and solid bodies.

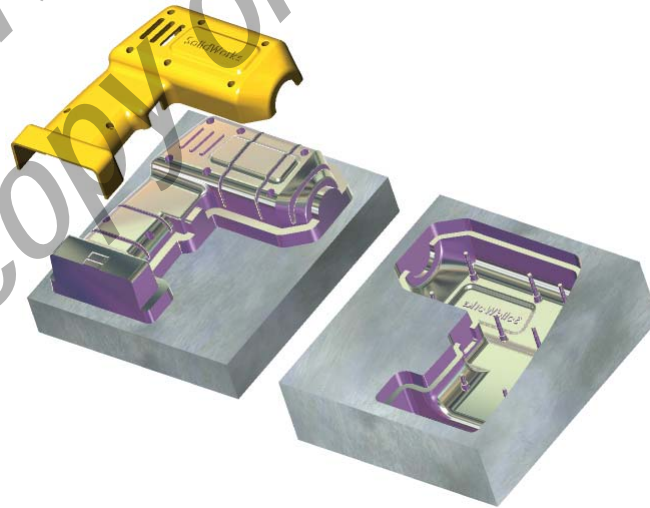
Show the solid bodies one at a time to examine the tooling.



Save and close all files.

**Case Study:
Plastic Bezel of
a Cordless Drill**


The objective of this case study is to create the tooling for the plastic bezel of a cordless drill. The parting line for this plastic part is more complex than the last example.



The following topics will be explained:

- Fixing un-drafted faces on imported geometry.
- Using the **Ruled Surface** command.
- Thickening the surface body into a solid
- Fixing steep model faces.
- Inverting the current selection.
- Creating complex shut-off surfaces.
- Creating interlock surfaces manually.
- Selecting a partial loop.
- Using the **Lofted Surface** command.

1 Open the part named Cordless Drill.

Click **Draft Analysis**  on the Mold Tools tool bar.

Select the Top Plane for the **Direction of Pull**.

Set the **Draft Angle** tolerance to 1°.

Select the **Face Classification** and the **Find Steep Faces** check boxes.

Click **Calculate**.

2 Examine the draft analysis results.

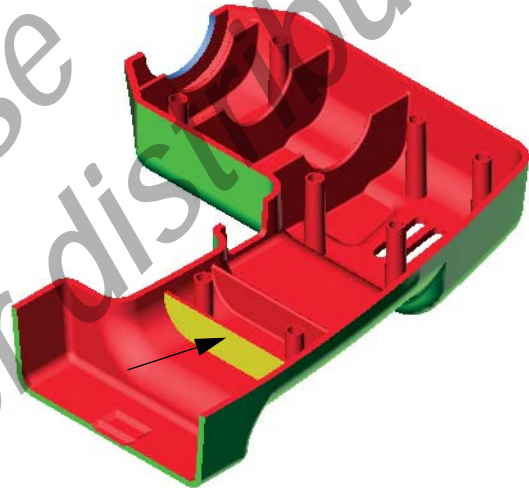
The arrow in the illustration shows a face that requires draft.

The draft analysis found two faces that must be fixed.

Rotate the part and find the other yellow face. It is parallel to the yellow face in the illustration.

Click **OK** to exit the PropertyManager.

When closing the PropertyManager a message asks you if you want to keep the face colors. Click **Yes**.



Creating New Drafted Faces

The yellow faces cannot be molded. The part designer added a strengthening rib to this model, but did not apply draft to the rib. If this file were engineered with SolidWorks software, you could just edit the rib feature and add draft. However, many mold designers work with data that is imported from other CAD software. When using an imported file, all of the design history is lost, and you must resort to surface modeling. To fix this part designers will:

- Delete the un-drafted faces.
- Construct new faces with draft.
- Trim them back to the faces of the engineered part.

Delete Faces that Do Not Have Draft

The first step to fix the draft, is to delete the un-drafted faces from the solid body. This process turns the solid model into a surface model.

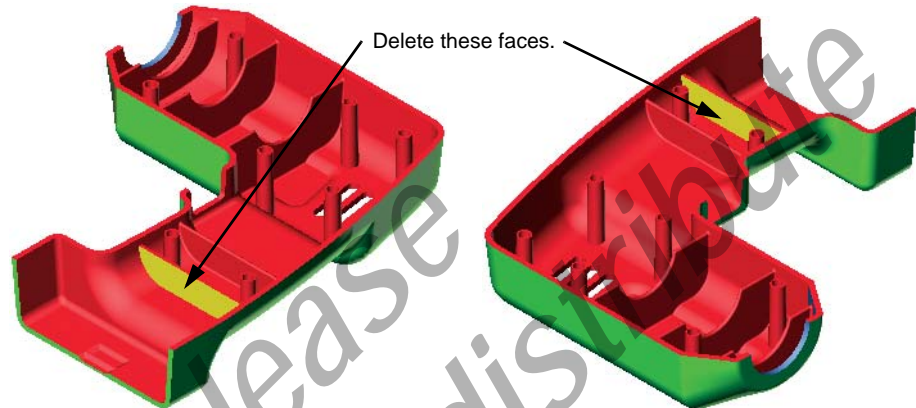
Disclaimer

In this example, the geometry of the rib is simple enough that you can add draft using the **Draft** command. Typically, things are rarely this simple. The more general approach to this situation is to delete and to surface model the faces that require more draft.

3 Delete face.

Click **Delete Face**  on the Surfaces toolbar.

Select the two yellow faces. Select the **Delete** option and click **OK**.

**4 Examine the Surface Bodies folder.**

When the un-drafted faces were deleted from the part, the part became a surface body. Look in the FeatureManager design tree, and notice that now there is one body in the Surface Bodies folder named DeleteFace1. The Solid Bodies folder is gone.

Color

The part has lost all of the colors assigned to the model faces during the draft analysis. The draft analysis colors are no longer valid because the geometry of the body has changed. Another draft analysis will be required after the new faces are built and knit back to the model.

Create New Drafted Surfaces

To create new drafted surfaces, use the **Ruled Surface** command.

Introducing: Ruled Surface

Use the **Ruled Surface** to create surfaces that are either perpendicular or tapered away from the selected edges. The ruled surface tool has many uses for mold design. In this step, it is used to create new drafted faces that were deleted them from the model. Later, this command will be used to create interlock surfaces around the parting surfaces perimeter.

Where to Find It

- Click **Ruled Surface**  on the Mold Tools toolbar.
- Or click **Insert, Molds, Ruled Surface**.

5 Create new ruled surfaces.

Click **Ruled Surface**  on the Mold Tools toolbar.

Select the **Tapered To Vector** option.

Set the **Distance** to **1.0"**.

Click in the **Reference Vector** field.

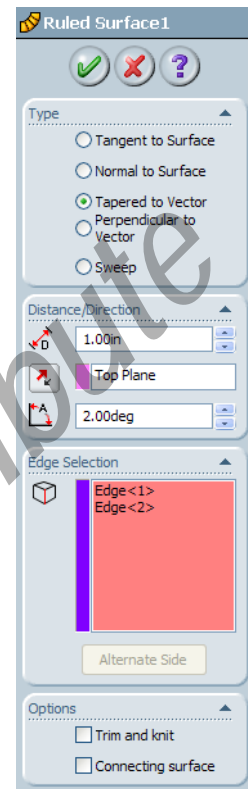
Select the Top Plane from the FeatureManager design tree.

Set the **Draft Angle** to **2.0°**.

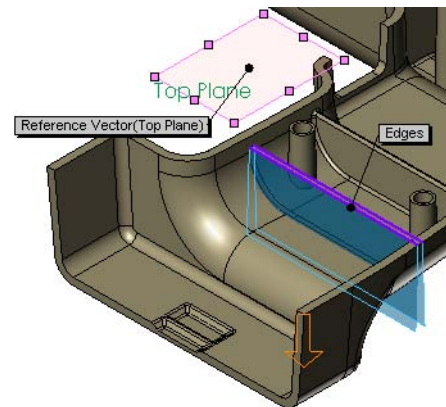
Click in the **Edge Selection** list.

Select the two horizontal edges on the remaining face of the rib.

*Do not click **OK** yet.*

**6 Preview.**

Examine the preview and verify that the surfaces draft outward. If one or both do not, select the edge or edges in the **Edge Selection** list and click **Alternate Side**.

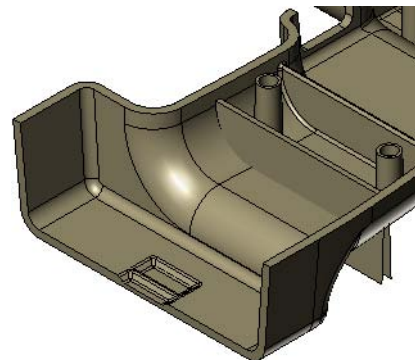
**7 Choose Ruled Surface options.**

At the bottom of the PropertyManager, clear the **Trim and Knit** check box.

Clear the **Connecting Surface** check box.

Click **OK**.

Two surfaces are created.



Trim the New Surfaces

Now trim back the two new ruled surfaces to the underside of the drill housing. Then trim back the surfaces on the drill housing to the new ruled surfaces. This is done using the **Mutual** option in the **Trim Surface** command. Trimming these surfaces requires two steps.

8 Trim the ruled surfaces.

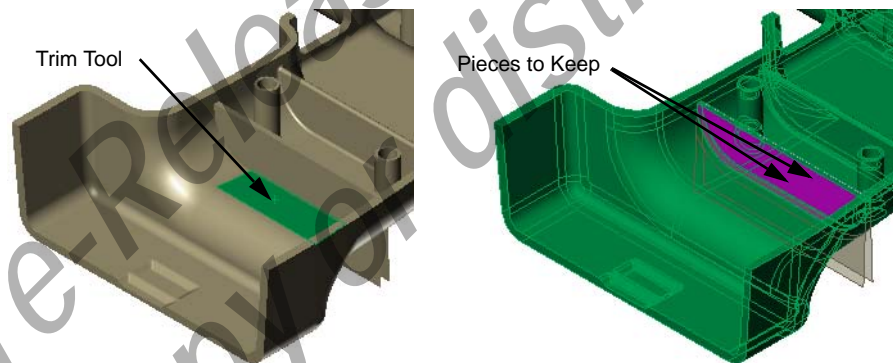
Click **Trim Surface**  on the Surfaces tool bar.

Select **Standard** from the **Trim Type** options.

Click in the **Trim Tool** field.

Select an inside face of the surface body.


Select **Keep selections** and select the two ruled surfaces by clicking on the portion you want to keep, and click **OK**.



Tip

9 Mutual trim the surfaces.

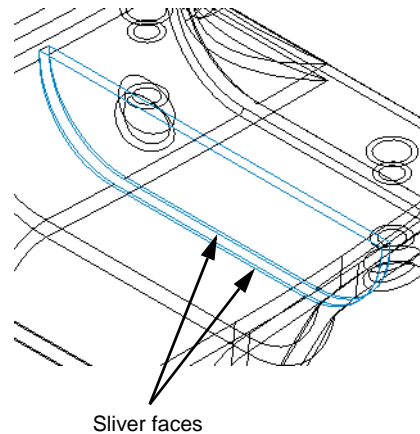
Switch to wire-frame mode to better view the trimming operation.

Click **Trim Surface**  on the Surfaces toolbar.

Select **Mutual** from the **Trim Type** options.

Click in the **Trimming Surfaces** list.

Select the two ruled surfaces and the inside face of the drill housing.

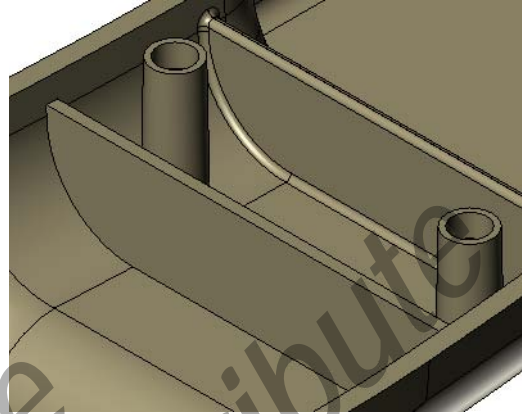


10 Select the surface pieces to keep.

Select **Keep selections** and identify the portions of the three surfaces you wish to keep.

Select the two trimmed ruled surfaces and the inside face of the drill housing.

Click **OK** and examine the results.

**Thicken the Surface Body**

The surface modeling required to correct the faces with insufficient draft is complete. Note that the **Trim Surface** command automatically knit all of the surfaces into a single surface body. The surface body will now be converted back into a solid body by thickening the surface body. After thickening, the **Draft Analysis** will be repeated.

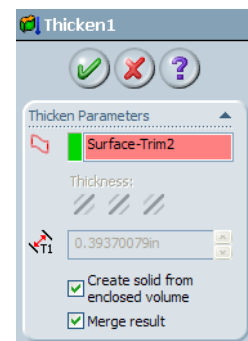
11 Thicken the surface body.

Select the **Surface-Trim2** feature from the FeatureManager design tree.

Click **Insert, Base/Boss, Thicken**.

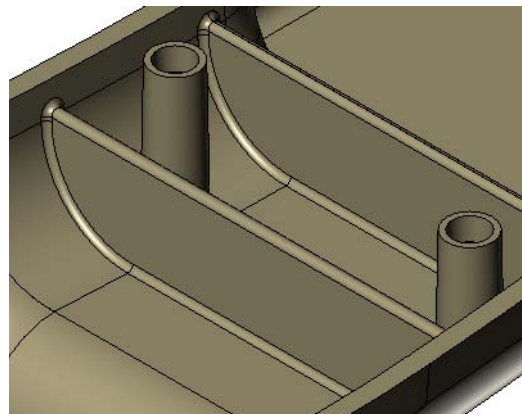
Click **Create solid from enclosed volume** and **Merge result**.

Click **OK**.

**12 Fillet the rib.**

Put a **Full round fillet** on the top of the new rib.

Run a **0.030"** radius fillet around where the rib intersects with the body of the drill housing.



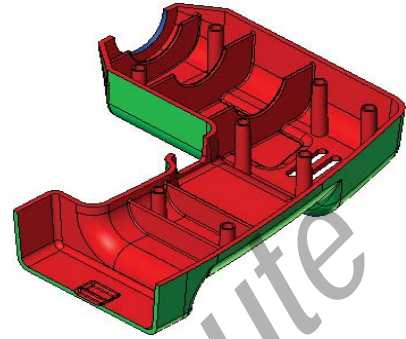
13 Check the part for draft.

Click **Draft Analysis**  on the Mold Tools toolbar.

Use the same analysis settings that were used in step 1 on page 289.

The faces of the rib are now classified as having negative draft.

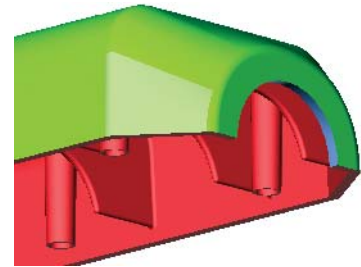
Click **OK** and click **Yes** to save the face colors when prompted.

**Fixing the Steep Faces**

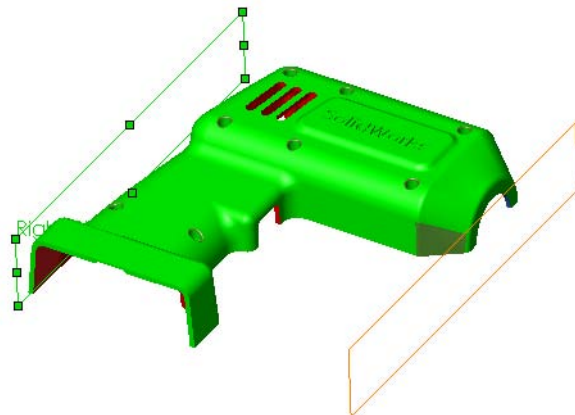
During the **Draft Analysis** some steep faces were found. Sometimes these can be ignored as long as they have some draft on them. In other cases, modifications are needed if the steep face is going to be part of the parting line that is also an interlock surface. In this model the barrel has a steep face that should be adjusted. A draft angle of 5° is usually required on a surface that will be part of an interlock surface. This keeps the steel from one side of the tooling from galling the steel on the other side of the tooling when the tooling opens and closes. See *Automatic Interlock Surface Creation* on page 286 for a thorough explanation of interlock surfaces.

14 Find the negative steep face on the model.

The steep face in the opening of the barrel must be adjusted because it is right on the parting line, and it is also used to develop the interlock surfaces.

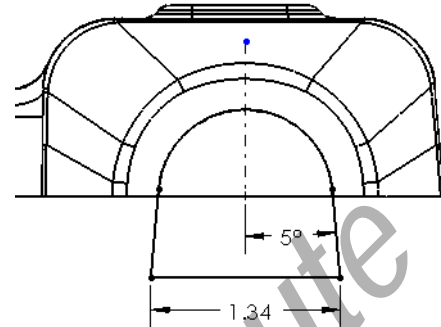
**15 Create an offset plane.**


Create an offset plane **7.0"** away from the Right Plane so that it is in front of the barrel.



16 Fix the steep faces.

Open a sketch on the new reference plane. Create the sketch as shown in the illustration. The intent is to create some draft at the very bottom of the circular edge.



Use the **Convert Entities**  command to convert the arc. Then create angled lines tangent to the converted arc.

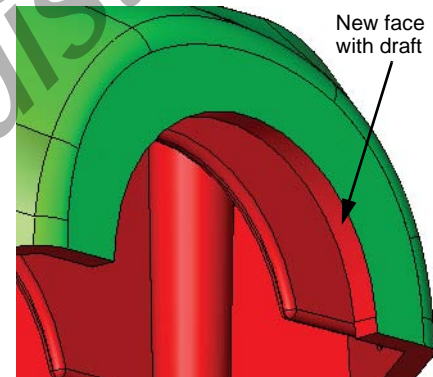
17 Cut the sketch into the model.

Extrude a cut a **Depth** of **1.00"** into the part. This will create faces with draft on the inside of the barrel.

18 Recheck the draft.

The face was divided into three separate faces. All three faces are now classified as negative draft and are no longer classified as steep faces.

The part can now be molded and the tooling can be created.


**19 Scale the Part.**

Scale the part larger by **1.05%** to allow for shrinkage

20 Examine the results.

Look in the FeatureManager design tree and see that the **Scale1** feature was added.

21 Create the Parting Lines.

Click **Parting Lines**  on the Mold Tools toolbar.

Click in the **Direction of Pull** field.

Select the **Top Plane** from the FeatureManager design tree.

Set the **Draft Angle** to **1°**.

Click the **Use for Core/Cavity Split** option.

Clear the **Split Faces** option.

Click **Draft Analysis**.

All the parting lines are automatically found.

Click **OK**.

Introducing: Invert Selection

The **Invert Selection** command will un-select the currently selected objects and then select every un-selected object on the model document. **Invert Selection** will be used to select all of the faces on the model so that the colors can be removed from them. The appropriate selection filter should be used when using the **Invert Selection** command.

Where to Find It

- Click **Tools, Invert Selection**.
- Click **Invert Selection** from the right mouse button popup menu.

22 Remove the colors assigned by Draft Analysis.

Click **Filter Faces**  on the Selection Filter toolbar.

Select any face on the model.

Remember what face you selected.

Right-click that face and click **Invert Selection** from the popup menu.

Hold down the control key and reselect the original selected face.

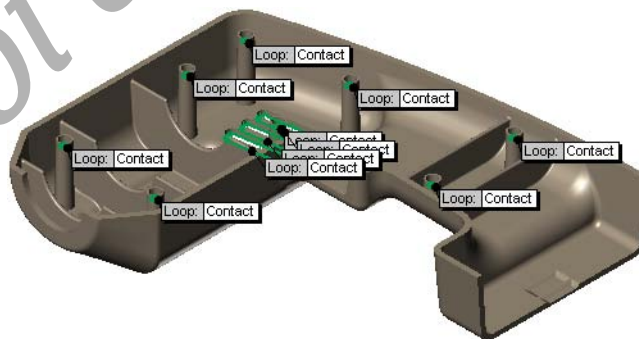
Click **Edit Color** .

Click **Remove Color** and then click **OK**.

All of the colors are removed from the model faces.

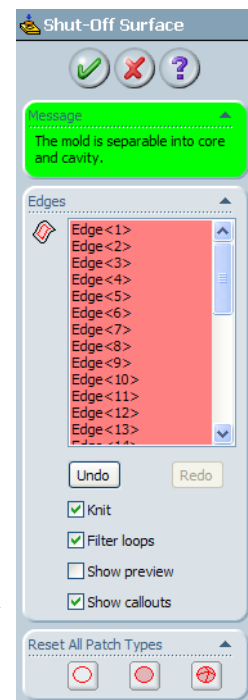
23 Find the shut-off surface areas.

Click **Shut-off Surfaces**  on the Mold Tools toolbar.




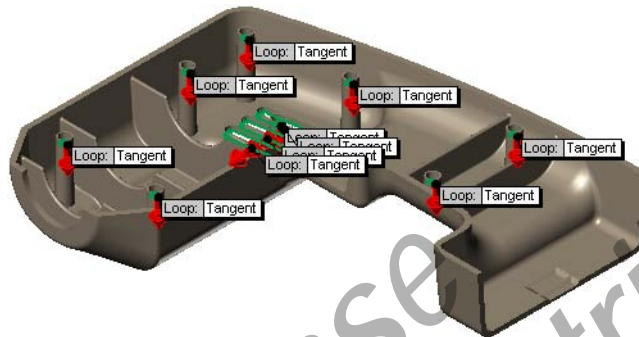
The solid model is analyzed for areas that require shut-off surfaces. The colors assigned by the **Draft Analysis** were removed to make the selected green loops more visible.

The **All Contact**  patch type is used by default.



Note

If the **All Tangent**  patch type is used, potential shut-off loops are displayed with *red arrows* and a callout. Use the red arrows to toggle the faces that you want the shut-off surfaces to be tangent to.

**24 Create the shut-off surfaces.**

The shut-off areas on this part are all planar. Therefore, the **Tangent** option is not needed.

Click the **All Contact** patch type.

Make sure that you select the **Knit** option, and then click **OK**.

25 Examine the results.

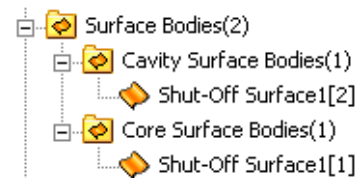
Shut-off surfaces were created for the three vent holes in the side of the bezel. Also, there are shut-off surfaces for all of the through holes in the part.

**Note**

You may also use **Parting Line** features as the boundaries for shut-off surfaces.

26 Examine the surface bodies.

The FeatureManager design tree now contains a **Solid Bodies** and a **Surface Bodies** folder. The **Surface Bodies** folder in turn includes two other folders.



27 Expand the Surface Bodies folder and its sub-folders.

Expand the Cavity Surface Bodies and the Core Surface Bodies folders. Note that the **Shut-off Surfaces** command created two surface bodies: one representing the core and one representing the cavity of the mold.

28 Hide the solid bodies.

The model contains both surface and solid bodies. To work on the surfaces only, right-click the solid body in the Solid Bodies folder and select **Hide Solid Body** from the shortcut menu.

**Tip**

To hide all of the surface bodies in the Surface Bodies folder, right-click on the folder named Surface Bodies, and select **Hide Bodies** from the shortcut menu. This technique can also be used to hide all the bodies in the Solid Bodies folder.

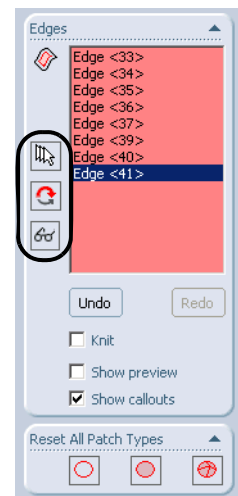
29 Hide the surface bodies.

Hide all of the surface bodies and show the solid body again.

Complex Shut-off Surfaces

The **Shut-off Surfaces** command automatically found all of the shut off surfaces on this part. There are many cases where shut-off surfaces are more complex. In such cases, use the selection tools from the **Edges** list box to select shut-off surface boundaries. If an edge is selected that is not a closed loop, the selection buttons appear next to the **Edges** list.

These buttons work the same way as they do in the **Parting Line** PropertyManager. See *Manual Selection Of Parting Lines* to review how these buttons work.



Important!

Sometimes a shut-off surface is too complex to use this command. When this happens, choose the **No-Fill** shut-off surface type. After the shut-off surfaces are established, manually surface model the complex shut-off.

If a tooling split is created from a part where there were manually created shut-off surfaces, a copy of the manually created shut-off surfaces must be created. This is done with the **Move/Copy Body** command.

Drag one copy into the Cavity Surface Bodies folder, and the other copy into the Core Surface Bodies folder. These surface folders are referenced when using the **Tooling Split** command.

Any surface in the Cavity Surface Bodies folder is automatically added to the cavity surfaces list when the **Tooling Split** command is used. The same is true for core surfaces and any parting surfaces that were created manually.

Tip

The **Ruled Surface** command has many options for creating complex shut-off surfaces. The **Tapered to Vector** option and **Sweep** options are particularly useful for creating complex shut-off surfaces. Refer to *Exercise 27: 80mm Fan Bezel* for a complete example of how ruled surfaces were used to model complex shut-off surfaces. This example also shows how the surfaces were copied and placed in the appropriate surface bodies folder.

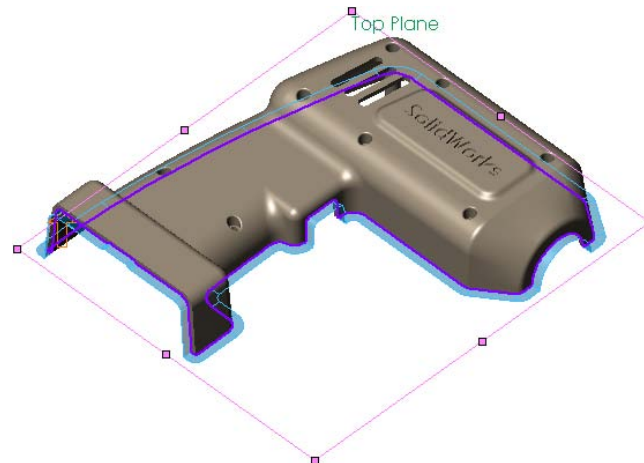
30 Create the parting surfaces.

Click **Parting Surfaces**  on the Mold Tools toolbar.

Under **Mold Parameters** select **Perpendicular to pull**.

Set the **Distance** to **0.1875"**.

Select the **Knit all surfaces** and the **Show preview** check boxes.

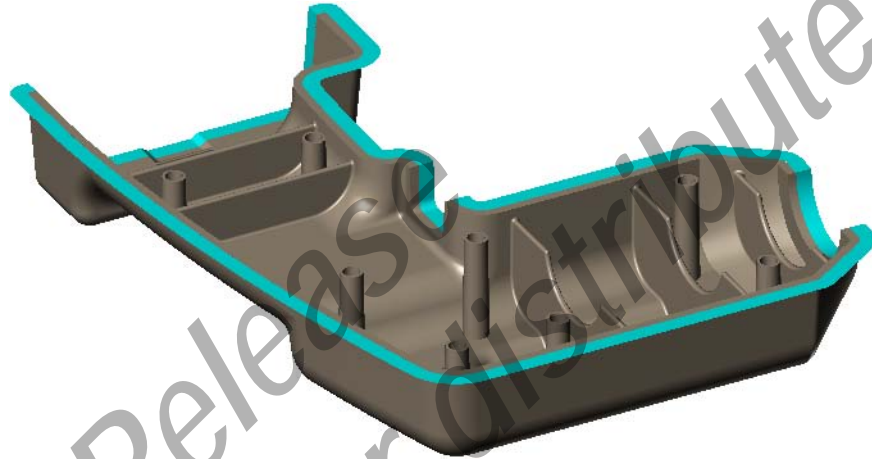
31 Examine the preview.

Note

In certain cases, the distance or other **Parting Surface** options may need to be adjusted to achieve an acceptable parting surface.

32 Click OK.

The parting surfaces are created and the Parting Surface1 feature is added to the FeatureManager design tree.



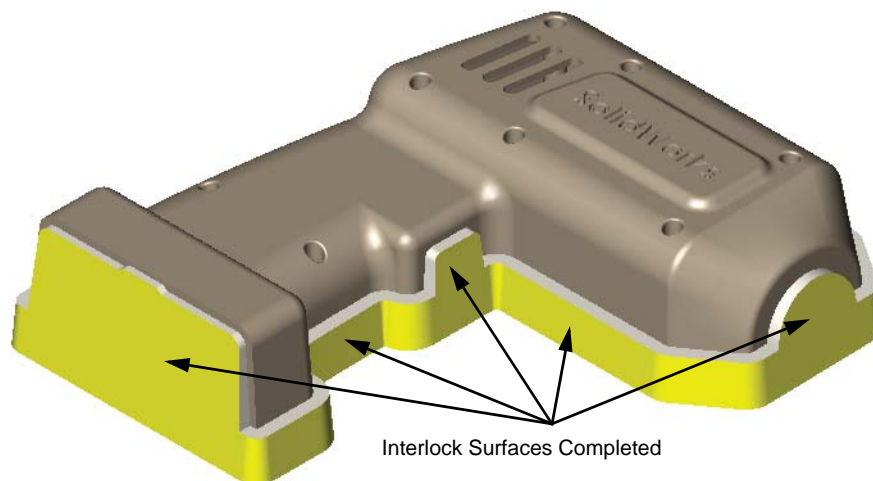
The parting surfaces are shown in color for clarity

Note

Although this process is automated, some manual surface modeling is sometimes required to adjust the surfaces created in this step. SolidWorks software allows you to trim, model, and knit new surfaces to this parting surface feature.

Interlock Surfaces

Sometimes, depending upon the complexity of the parting surface, interlock surfaces cannot be created automatically. In this example, they are created manually because of the sudden changes in parting surface geometry. The battery pack mounts, the trigger, and the barrel areas of the bezel are areas where some simple surface modeling can create interlock surfaces



Modeling the Interlock Surfaces

Select Partial Loop

Use the **Ruled Surface** command to create the tapered, ribbon like, surfaces that form the interlocks.

The parting surfaces can contain many small edges. To construct ruled surfaces along edges of parting surfaces, a series of connecting edges will need to be selected. To facilitate this process, use **Select Partial Loop** to select a chain of connecting edges. The chain direction is based on where you select the *second* edge:

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

33 Create a ruled surface.

Click **Ruled Surfaces**  on the Mold Tools toolbar.

Select the **Tapered to Vector** option.

Set the **Distance** to **0.625"**.

Click in the **Reference Vector** field and select the Top Plane from the FeatureManager design tree.

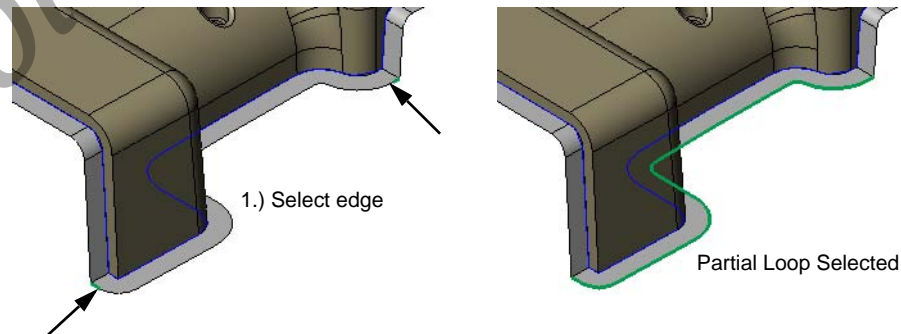
Set the **Angle** to **5°**.

34 Select a partial loop of edges.

Click in the **Edges** selection list. Select the first edge on the parting surface as shown in the illustration.

Right-click on the second edge as shown. Select it near the end that is closest to the first edge you selected.

Click **Select Partial Loop** from the shortcut menu.

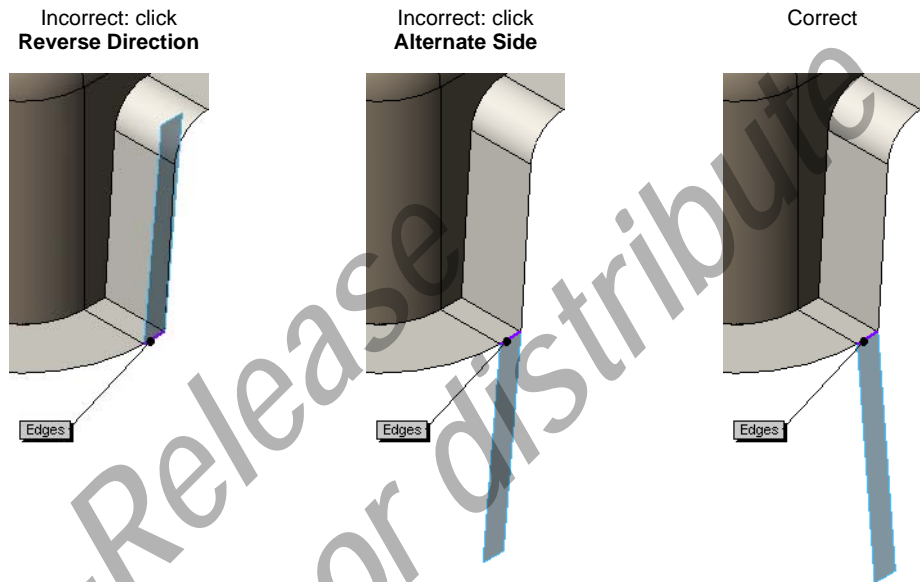


Click the **Trim and knit** option.

Clear the **Connecting Surface** check box.

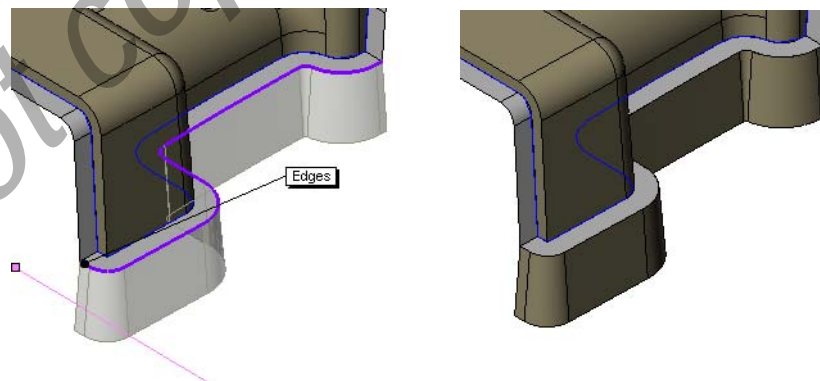
Tip

When selecting the first edge, examine the preview. If the preview of the ruled surface points in the wrong direction with respect to the pull direction, click **Reverse Direction**. If the preview tapers inward towards the parting surfaces instead of outward, click **Alternate Side**.

**35 Examine the preview.**

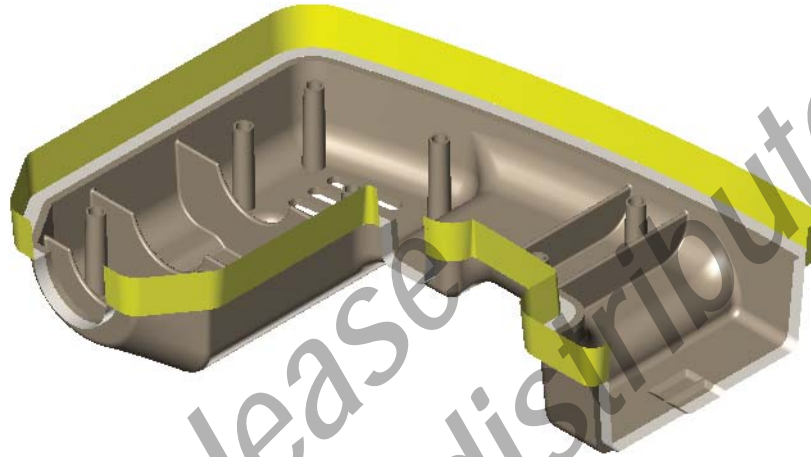
Make sure that the surfaces are drafted outwards.

Click **OK**. The results are shown in color below.



36 Create two more ruled surfaces.

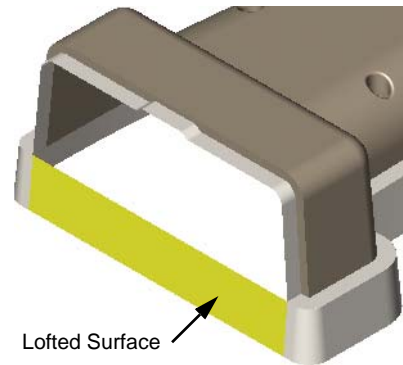
Use the same technique to create the remaining interlock surfaces around the perimeter of the parting line.



The three ruled surfaces are completed.

Fill in the Gaps With Lofted Surfaces

Now that the ruled surfaces are complete, fill the gaps in the interlock surfaces. Use the **Lofted Surface** command to create surfaces that connect the open edges of the ruled surfaces.




Introducing: Lofted Surface

Use the **Lofted Surface** command to create more interlock surfaces. Create lofted surfaces using the two edges of the ruled surfaces that are open. Select the two edges near the same starting point to keep the surface from twisting.

Where to Find It

- Click **Lofted Surface**  on the Surfaces toolbar.
- Or, click **Insert, Surface, Loft...**

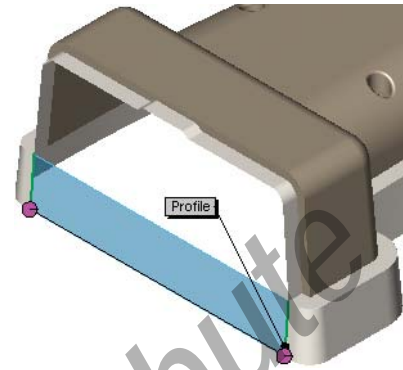
37 Create a lofted surface.

Click **Lofted Surface**  on the Surfaces toolbar.

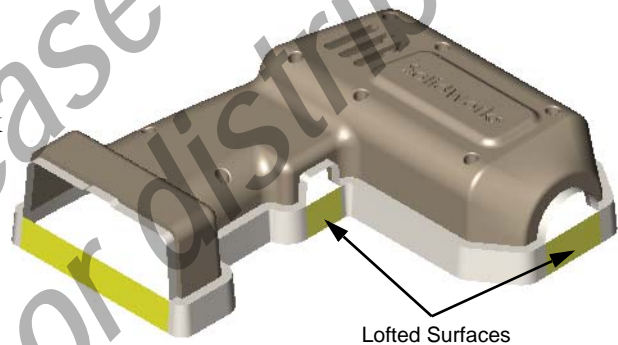
Select the two edges as shown in the illustration.

Select both the edges near their bottom or top endpoints to prevent the surface from twisting.

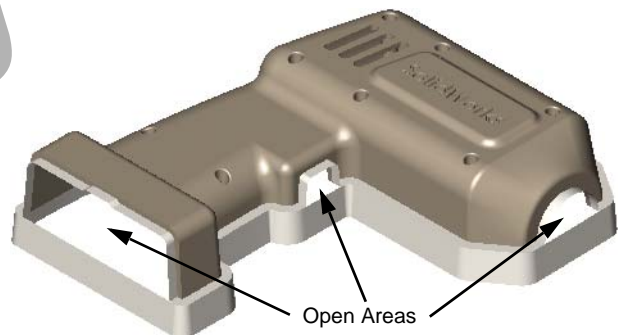
Click **OK**.


**38 Create two more lofted surfaces.**

This completes the all of ribbon-like interlock surfaces.

**Completing the Interlock Surfaces**

There are three more open areas that need to be filled with surfaces. These areas are where a major jog in the parting line occurs. The next few steps use the **Extend Surface** and **Trim Surface** commands.

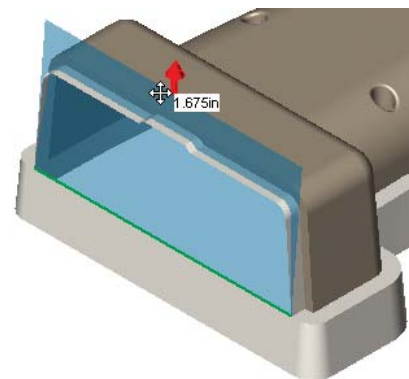
**39 Fill in the open interlock areas.**

Click **Extend Surface**  on the Surfaces toolbar.

Select the uppermost edge of the surface.

Drag the handle so that the surface extends past the highest point on the parting surface. The exact distance is not critical.

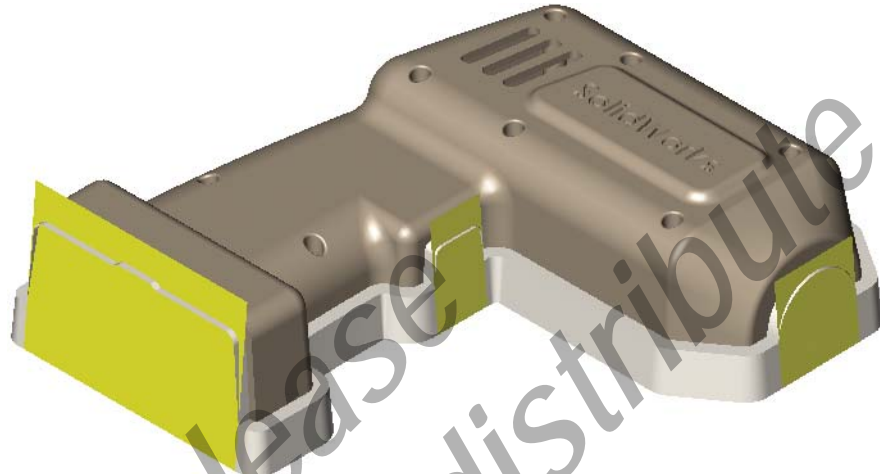
Click **OK**.

**40 Repeat.**

Repeat this procedure for the other open areas with lofted surfaces.

41 Examine the results.

The resulting surfaces should extend past the highest points of the parting surfaces.

**42 Trim the extended surfaces.**

Click **Trim Surface**  on the Surfaces toolbar.

Under **Trim Type** select **Mutual**.

Click in the **Trimming Surfaces** list.

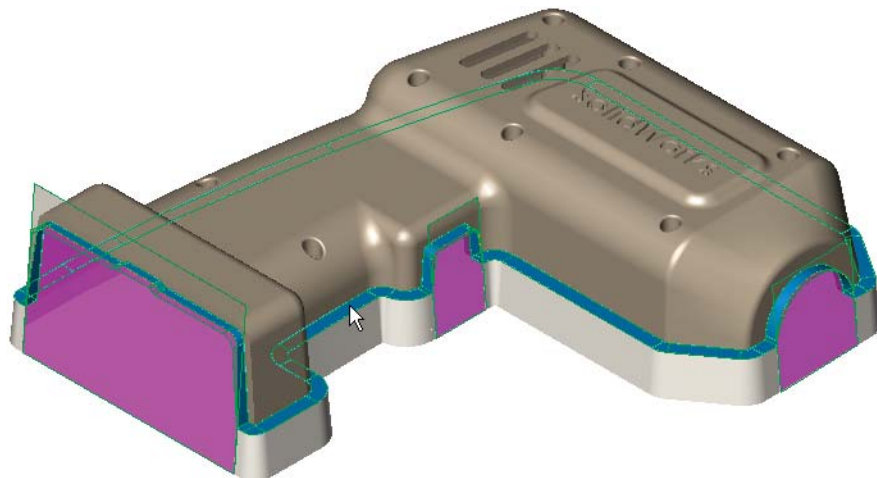
Select each of the extended surfaces from the graphics area and the Parting Surface1 feature.

Select the **Keep Selections** option.

Click in the **Pieces to Keep** list.

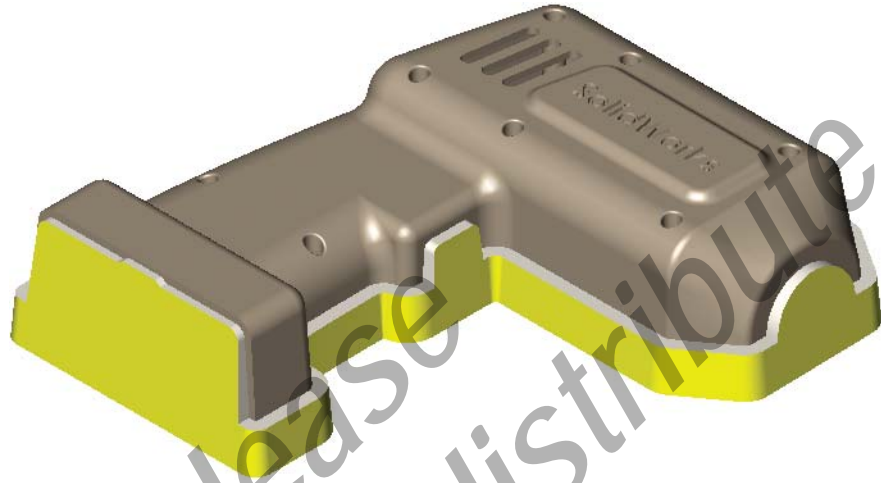
Now reselect the same surfaces in the appropriate areas to mutually trim them.

Click **OK**.



43 Results.


Examine the results of the **Trim Surface** command.



Knit the Interlock Surfaces to the Parting Surfaces

All of the interlock surfaces are now complete. The next step is to knit the interlock surfaces and the parting surfaces together. Knitting the interlock surfaces and the parting surfaces creates a complete surface body to split the mold tooling. The **Mutual** trim option knit the three extended surfaces to the parting surfaces. However, the other portions of the interlock surfaces are still separate surface bodies.

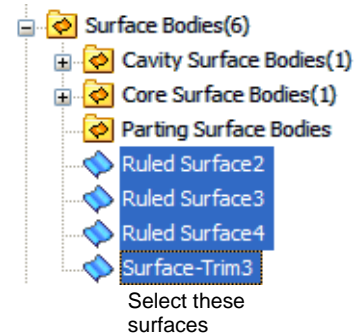
44 Knit the all of the surfaces together.

Click **Knit Surface**  on the Surfaces toolbar.

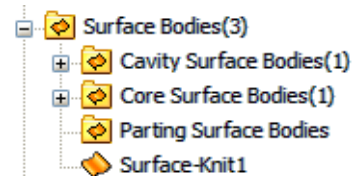
Select all the surfaces in the Surface Bodies folder.

Clear the **Try to form solid** check box.

Click **OK**.

**45 Examine the Surface Bodies folder.**

The Surface Bodies folder is updated to show the knit surface body.



Preparations for the Tooling Split

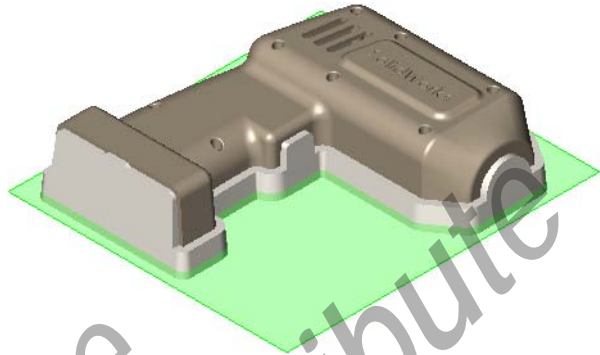
To create the *tooling split*, the perimeter of the parting surface body must be larger than the outside profile of the tooling blocks. A planar face is created that is larger than the tooling blocks and the parting surfaces. This face is used to cut and form the top faces of the tooling.

46 Create an offset plane.

Create a reference plane offset **0.5"** below the Top plane.

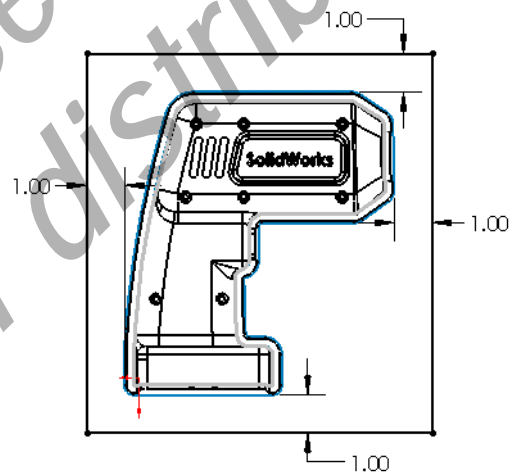
Use this plane to create a large planar surface


Name this plane Tooling Plane.

**47 Sketch the outside perimeter of the tooling.**

Create a new sketch on the Tooling Plane.

Sketch a rectangle that is **1.0"** larger than the edges of the interlock surfaces.

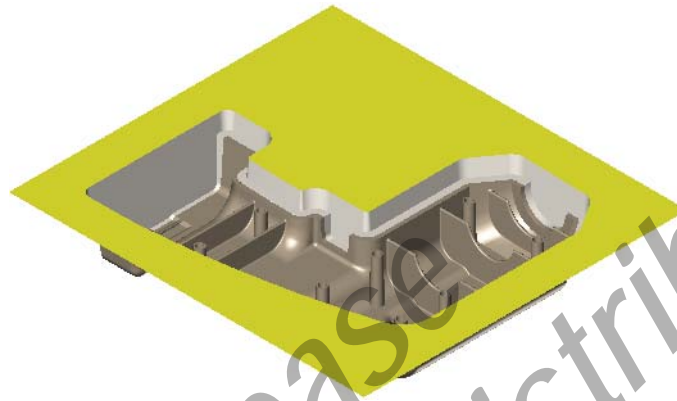
**48 Create a planar surface.**

Click **Planar Surface**  on the Surfaces toolbar to create the surface using this sketch profile. Click **OK**.

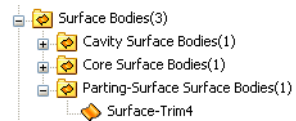


49 Trim the planar surface.

Use the **Mutual** option to trim the new planar surface to the bottom of the interlock surfaces. The result is the surfaces are knit together.

**50 Parting Surface folder.**

Drag and drop the resulting surface into the Parting Surface folder.

**51 Create the tooling.**

Click **Tooling Split**  on the Mold Tools toolbar.

The PropertyManager appears prompting you to choose a plane, surface, or a sketch to use for the perimeter of the tooling.

Select the large planar surface from the graphics area.

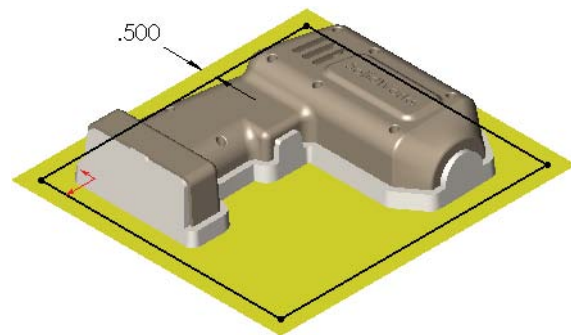
The part is now in sketch mode.

52 Create an offset sketch.

Create an offset **0.5"** to the *inside* of the planar surface as shown.

Click **Exit Sketch** to continue.

The **Tooling Split** PropertyManager appears.



53 Adjust the sizes of the tooling blocks.

Change the tooling **Block Sizes**.

Set the **Depth in Direction 1** to **3.0"**.

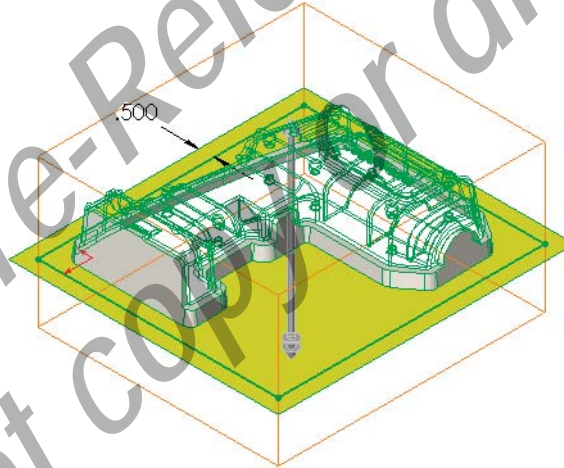
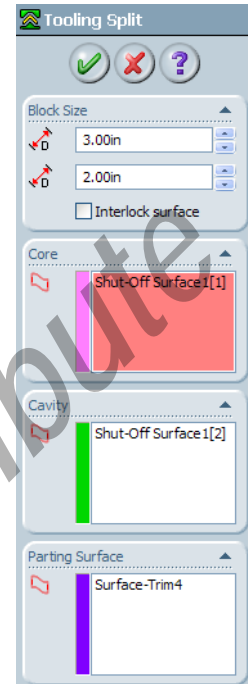
Set the **Depth in Direction 2** to **2.0"**.

Make sure the **Interlock Surface** check box is cleared.

The **Core**, **Cavity**, and **Parting Surfaces** selection lists are automatically populated by their corresponding surfaces in the Surface Bodies folder.

Change the view to an ***Isometric** or ***Front** view to get a better angle of the tooling blocks.

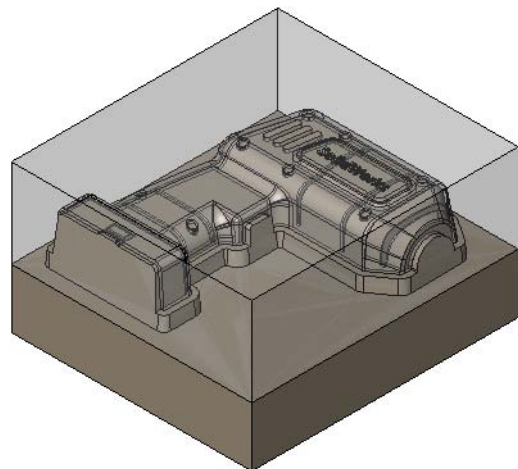
Click **OK**.

**54 Examine the tooling.**

The **Tooling Split** is finished.

The core, cavity and plastic part bodies are organized in the Solid Bodies folder.

Additionally, the **Tooling Split1** feature was added to the end of the FeatureManager design tree.



55 Hide the surface bodies.

Hide all the surface and solid bodies. Show them one at a time to examine the results.

Solid Bodies	Surface Bodies
 <p data-bbox="456 674 574 699">Molded Part</p>	 <p data-bbox="1049 737 1198 762">Parting Surface</p>
 <p data-bbox="444 1199 561 1224">Cavity Solid</p>	 <p data-bbox="1052 1100 1195 1125">Cavity Surface</p>
 <p data-bbox="451 1619 555 1644">Core Solid</p>	 <p data-bbox="1058 1541 1188 1566">Core Surface</p>

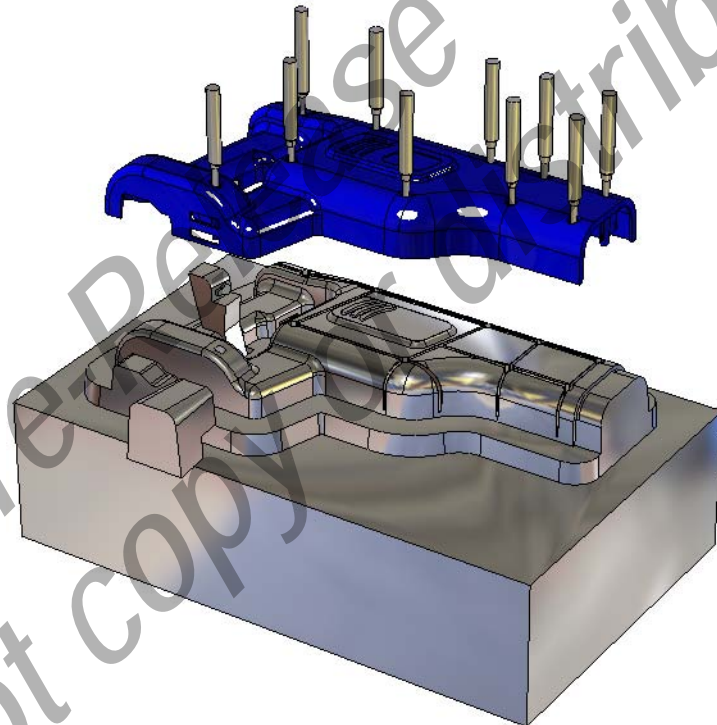
56 Save and close all files.

Tip

An assembly of the tooling can be created by right-clicking on the **Solid Bodies** folder and selecting **Create Assembly**. Refer to the topic *Saving Solid Bodies as Parts and Assemblies* on page 36 for more information.

Case Study: Multiple Parting Directions

The previous exercises created molds with only two pieces of tooling. Molds can be more complicated. Some molding areas require tooling that does not travel in the same direction that the plastic part ejects from the mold. This requires engineering more than just a cavity and a core. Other pieces of tooling such as *side cores* and *lifters* are required to form molding areas that cannot be ejected from the primary parting line. SolidWorks software provides commands to help create tooling that travels in a different direction than the primary parting plane.



1 Open the part that requires side cores.

Open Power Saw with Side Actions.

The tooling split for this part was already created.

In the next steps, you roll back the model, and determine how the tooling split was created.

An **Undercut Analysis** will be done to find molding areas where additional tooling needs to be created.

2 Rollback the part.

Right-click `Scale1` in the FeatureManager design tree and select **Rollback** from the shortcut menu.

**Tip**

The body for the plastic part is shown with transparency. Right-click the body in the solid bodies folder and choose **Appearance, Color...**

In the **Optical Properties** group box, use the transparency slider to remove the transparency from the part.

**Introducing:
Undercut Detection**


The **Undercut Detection** command helps determine where there are trapped molding areas. A trapped molding area is an area on the plastic part that cannot be released from the tooling using the primary direction of pull. This command will help locate areas that will need tooling such as lifters and side cores.

Where to Find It

- Click **Undercut Detection**  on the Mold Tools toolbar.
- Or, click **Tools, Undercut Detection**.

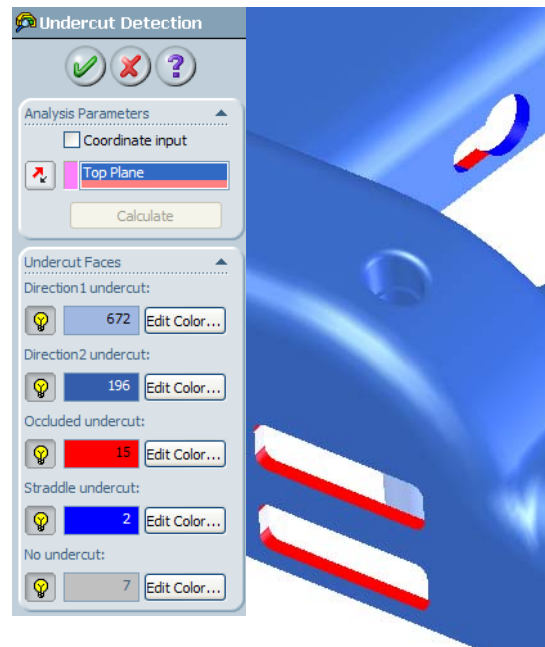
3 Check the model for undercuts.

Click **Undercut Detection**

 on the Mold Tools toolbar. Choose the Top Plane as the **Direction of Pull**. Press **Calculate**.

Zoom in to the battery pack and the trigger location to see the faces that are colored red.

These areas require tooling that travel perpendicular to the direction of pull. **Close** the dialog *without* saving the face colors.



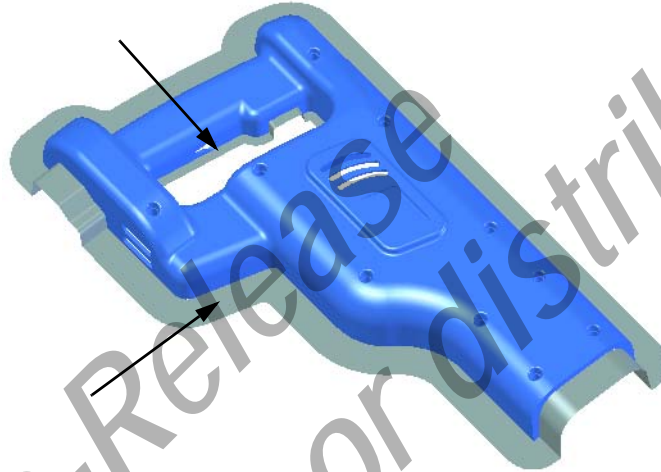
4 Examine the parting lines.

Right-click *Curve1* in the FeatureManager design tree and select **Roll Forward** from the shortcut menu.

5 Examine the parting surfaces

Notice that this part has two parting lines and two parting surfaces.

SolidWorks allows the use of multiple parting lines.

**6 Roll to end**

Right-click anywhere in the FeatureManager design tree and select **Roll to End** from the shortcut menu.

Trapped Molding Areas

After the undercut analysis is complete, SolidWorks software colors certain faces on the model red. These areas trap the plastic part from coming out of the tooling. Ideally plastic parts should not include any trapped areas. When there are no side cores or lifters, the mold is less expensive to design and manufacture. However, trapped molding areas cannot always be avoided. In such cases, additional tooling needs to be created to form the trapped molding areas.


Side Cores

A *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.

Introducing: Side Core

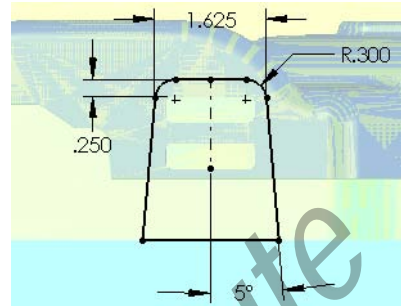
The **Core** command creates side cores based on the active sketch. Sketch around the area that requires new tooling. Create the sketch on a plane or a face parallel or perpendicular to the direction in which the tooling travels away from the plastic part.

Where to Find It

- Click **Core**  on the Mold Tools tool bar.
- Or, click **Insert, Molds, Core**.

- 7 Examine the Side Core Sketch.**
Select and edit the sketch named Side Core Sketch.


This sketch was created on an inside face of the cavity body. The face is drafted 5° from the direction that this side core travels. This side core travels perpendicular to the direction of pull.



Note

This sketch can be created on a face that is not parallel to the direction that the side core will travel.

- 8 Exit the sketch.**
Exit the sketch without changes.
- 9 Create the side core.**
Select the Side Core Sketch from the FeatureManager design tree.

Click **Core**  on the Mold Tools tool bar.

Click the **Front** plane for the **extraction direction**.

Click on the Cavity to select the **Core/Cavity body**.

Set the **Draft Angle** to 5° with the **Draft outward** option.

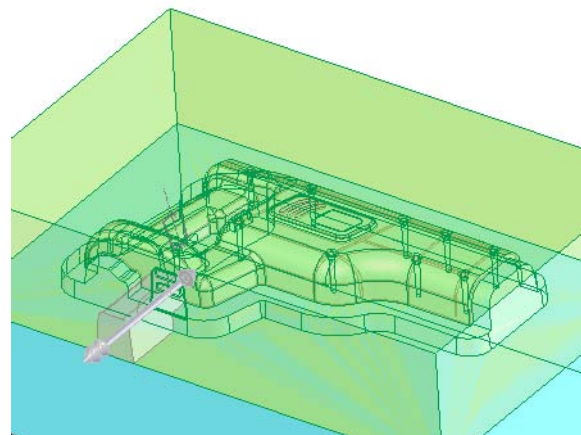
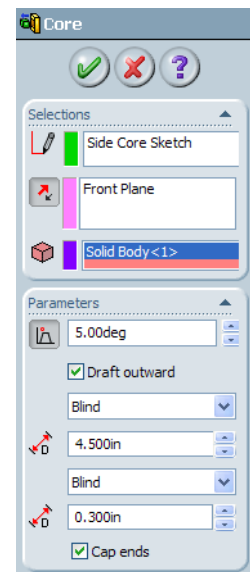
Set the first **End Condition** to **Blind**.

Set the first **Distance** to **4.5"**.

Set the second **End Condition** to **Blind**.

Set the second **Distance** to **0.3"**.

Click **OK**.



Creating the Side Core

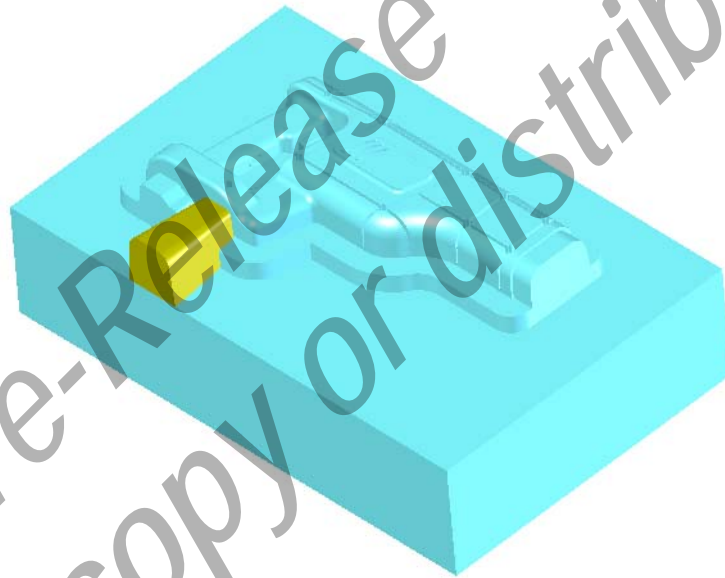
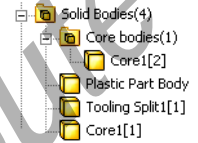
10 Examine the Solid Bodies folder.

Notice that there is now a new folder named *Core bodies*.

The **Side Core** command created a new solid body for the side core.

This command created the body, and then subtracted it from the cavity body.

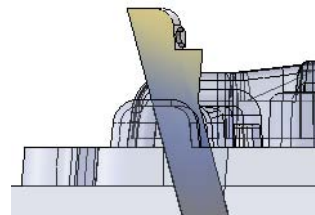
Any bodies created by the side core command are stored in this new folder in the FeatureManager design tree.



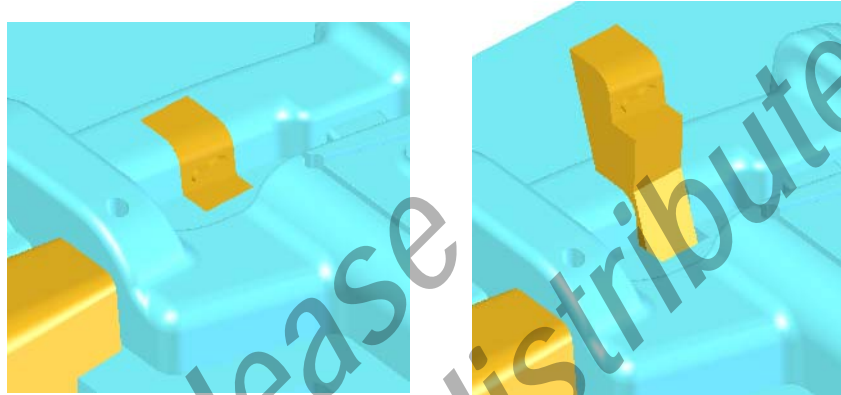
The cavity was hidden to show the resulting body of the side core command

Lifters

Lifters need to be created when there is a trapped molding area that not even a side core can create. Look to the trigger area of the saw bezel. There is a key shaped opening that is used for a safety lock. Because there is limited room in the trigger area, adding a side core is problematical. In this situation, mold designers create a mechanical device called a *lifter*.



This piece of tooling is moved by the ejector box. As the ejector box strokes forward, it pushes the lifter upwards and backwards on an angle, leaning away from the molding area. As it slides up and away from the molding area, it helps to *lift* the plastic part off of the core.



The lifter slides upwards and away from the molding area.

11 Edit the Lifter Sketch.

The shank of the lifter is leaned back 15° from the direction of pull.

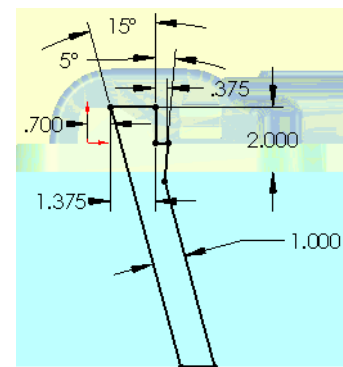
Notice also the 5° angle on the front of the profile.

This acts as an interlock, and keeps the part from sliding all the way through the bottom of the core.

Exit the sketch without changes.


Hide the cavity body and the plastic part body.

Show the core body.



12 Create the lifter.

Select the **Lifter Sketch** from the FeatureManager design tree.

Click **Core**  on the Mold Tools tool bar.

Select the core for the **Core/Cavity body**.

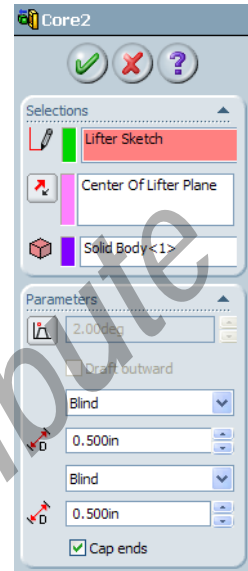
Click **Draft off**.

Clear the **Draft outward** option.

Set both **End Conditions** to **Blind**.

Set both of the **Depth along extraction direction** values to **.500"**.

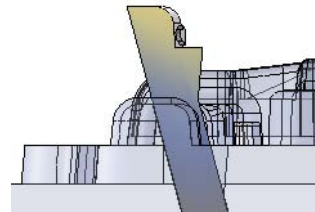
Click **OK**.

**13 Examine the results.**

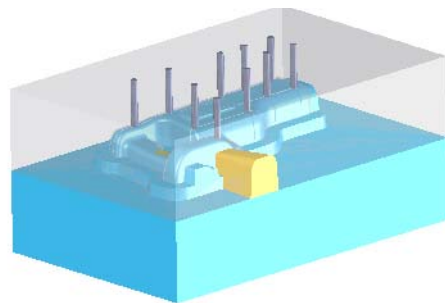
Hide all of the bodies except for the new lifter.

Notice that this new body is listed in the Core bodies folder.

Rename this feature **Lifter**.


**Core Pins**

The **Side Core** command can also be used to separate the *core pin* molding areas from the tooling. *Core pins* are created to form detail areas in the plastic part. These molding areas are areas that can wear faster than the other faces of the tooling. By creating molding areas with core pins, the mold can be easily repaired by switching out core pins, rather than replacing an entire piece of tooling.



14 Core pins.

Show the cavity body and make it transparent.

Select the Core Pin Sketch. Click **Core**  on the Mold Tools tool bar.

Click the top face of the cavity as the **Extraction Direction**.

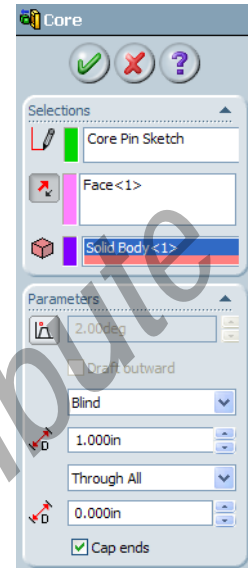
Click the cavity as the **Core/Cavity body**.

Click **Draft off**.

Set the first **End Condition** to **Blind** and set the **Depth along extraction direction** to **1.000"**.

Set the second **End Condition** to **Through All**.

Click the **Cap Ends** option.

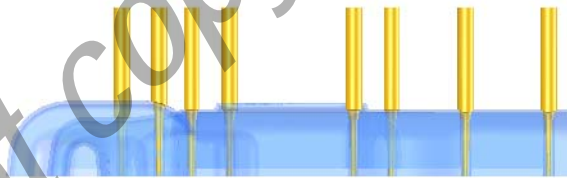
**Note**

It may be necessary to reverse the extraction direction.

15 Examine the results.

All of the core pins (10 solid bodies) are added to the model and the Core bodies folder.

Rename the last feature to CorePins and hide all of the solid bodies except the plastic part and the core pins.

**16 Save and close all files.**

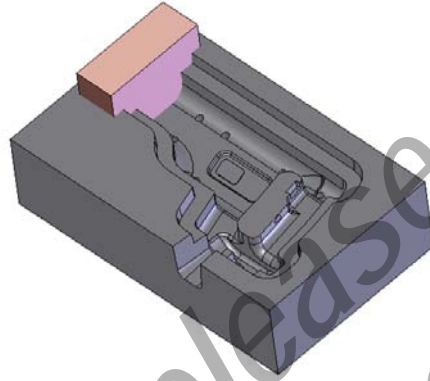
**Case Study:
Electrode
Design**

Electrode design is another challenging part of a mold design and manufacturing. Electrodes are used to remove steel from areas on the tooling that cutting tools like end-mills and ball-mills cannot reach or fit into. SolidWorks provides great modeling tools to produce accurate and complicated electrodes. This case study demonstrates how to use multi-body solids to create electrodes. Afterwards the **Move Face** command is demonstrated to show how to quickly clear back the material on the electrodes that will interfere with the areas of tooling that should not get EDM machined.

1 Open the part named *Electrode*.

This part has two solid bodies. One represents the cavity for the power saw bezel mold, and the other represents an electrode.

The electrode is needed here because there are sharp internal corners into which an end-mill cannot fit to machine this tooling properly.



2 Hide the electrode blank.

Zoom in to view the *Electrode Body*.

Right-click the *Electrode Body* in the *Solid Bodies* folder.

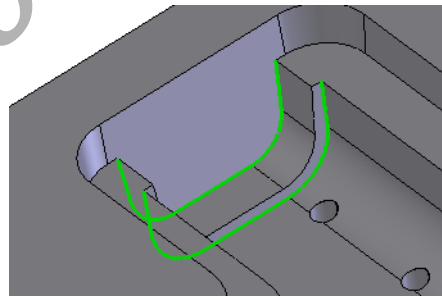
Click **Hide Solid Body** from the popup menu.

3 Examine the area that the electrode will burn.

End mill cutters cannot be used to machine the highlighted edges.

The cutters are round and these corners are dead sharp. Electrode machining is the only way to accurately machine these parts of the cavity.

Show the *Electrode Body*.



4 Make a copy of the cavity body.

Use the **Move /Copy** command to make a copy of the cavity body.

Note

A copy is needed because in the next step, the **Subtract** option in the **Combine** command is used to subtract the copied cavity geometry from the electrode blank. This eliminates the copied cavity body from the *Solid Bodies* folder. Later, the original cavity body will be used to visualize the clearance between the cavity and the electrode.

5 Subtract the copied cavity body from the electrode body.

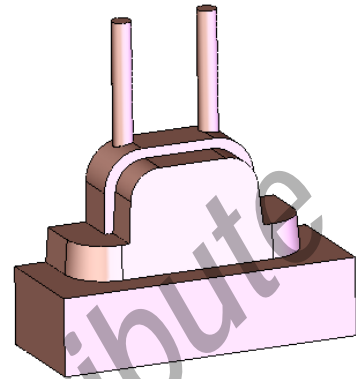
Click **Insert, Features, Combine...** from the menu.

Use the **Electrode Body** as the main body, and **subtract** the copied cavity body from it.

Examine the resulting body.

More work needs to be done to finish the electrode.

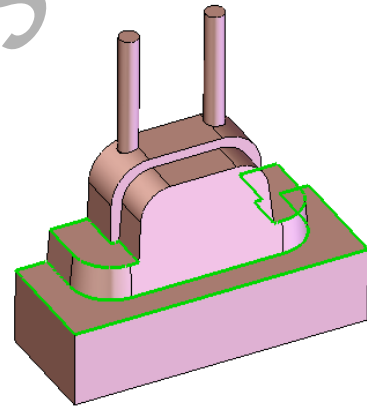
Electrode clearances will need to be modeled.



Electrode Clearances

Now that the electrode shape has been extracted from the cavity, certain areas of this electrode need to be removed. Other areas require clearance between the electrode and the tooling.

The highlighted faces in the diagram to the right are faces that can be cleared back, or in other words, "*pushed away from the tooling.*" These faces can be cleared back because they can easily be machined on the cavity without using EDM machining.



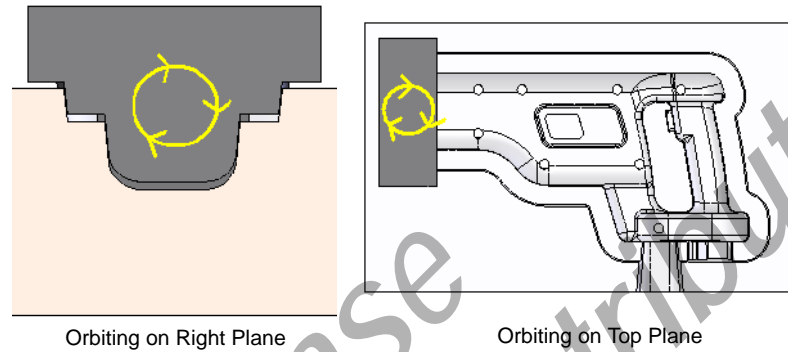
Over-burn

Even though the electrode geometry is the reverse of the cavity, the electrode faces that contact the tooling should be offset away from the tooling because of *over-burn*. Over-burn allowance must be considered because EDM machining requires that clearance exists between the electrode and the tooling to allow for *flushing*. As the electrode burns the shape into the metal, EDM fluid is used to flush out the burned metal. There should be clearance between the electrode and part to allow the flushing to get in and clear the scrap metal.

Orbiting

To make up for the offset geometry, the electrodes are *orbited* in the area that they are to machine. Orbiting the electrode will help the machinist achieve the exact dimensions of the shape in the steel being machined. Also, the wider the orbit, the faster the unwanted metal can be removed from the tooling.

The section views below shows different ways this electrode might be orbited. As the orbit is made larger, more steel will be removed wherever the electrode makes contact with the steel.



Tip

Offsetting the electrode geometry can either be done on the CAD model, or the tool paths can be offset in the CAM system to achieve the same effect.

Moving Back the Faces

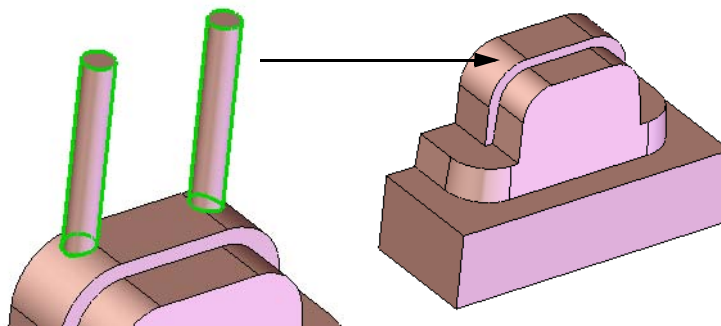
The **Move Face** command moves or rotates model faces. It will be used to clear back the faces that do not need to be burned in the cavity. The adjacent surfaces will be automatically extended and trimmed to the new position of the moved faces.

6 Remove the pins.

Use the **Delete Face** command to remove the faces that make up the pins and the radii around them.

Use the **Delete and Patch** option to heal the model into a solid.

There are 6 faces total to delete.



Use the Delete Face command to remove these faces.

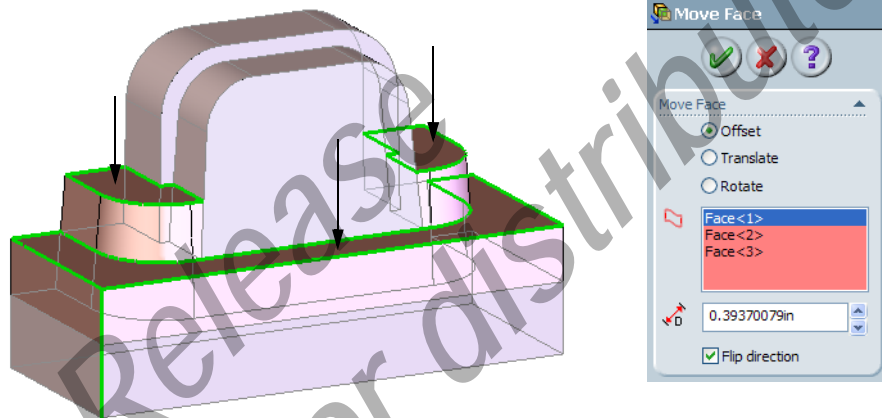
7 Move the faces.

Click **Insert, Face, Move...**

Click **Offset**.

Set the **Distance** to **.875**.

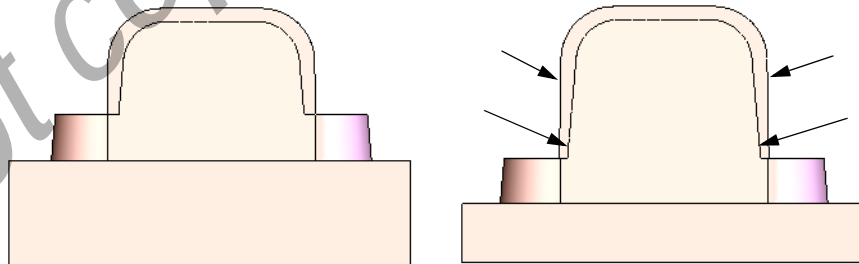
Select the 3 faces in the diagram below and flip the direction if necessary.



Move these three faces downwards

8 Inspect the adjacent faces.

Notice how the adjacent faces were extended and trimmed to the new moved faces.



Before

After

Note

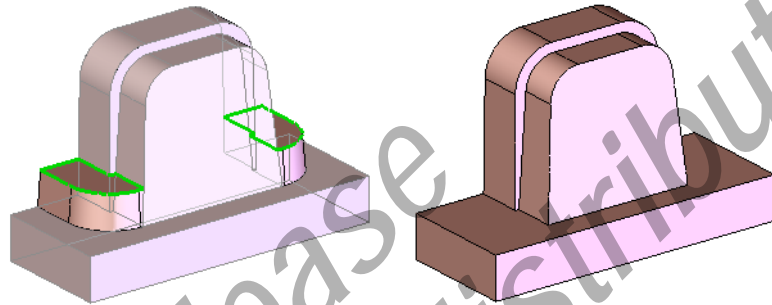
If the electrode was cleared back by extruding cuts straight down without extending the angled surfaces, witness lines could appear in the cavity where these surfaces originally ended. The witness lines would show up in the cavity after the EDM machining was completed.

9 Move two more faces.

Move the two highlighted faces down another **.875"**.

This electrode can now be orbited with out burning the lands for the interlocks.

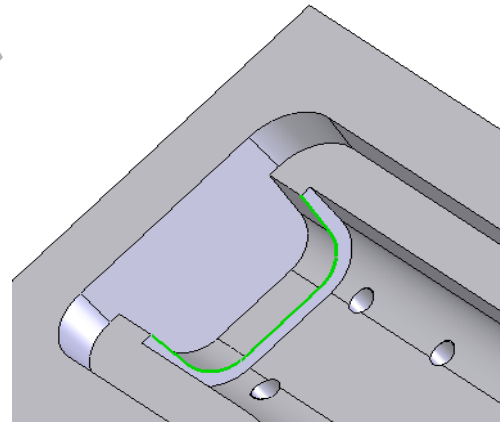
Show the cavity again and inspect the electrodes clearances.



The electrode now has plenty of clearance

Keeping the Sharp Edges

One more thing to keep in mind when modeling electrodes is that sharp edges on the tooling must be kept sharp. This electrode is currently burning too much of the cavity, and will cause some critical sharp edges to become dull, or rolled over. If this electrode was orbited from the top plane, these critical edges would become rounded or dulled.



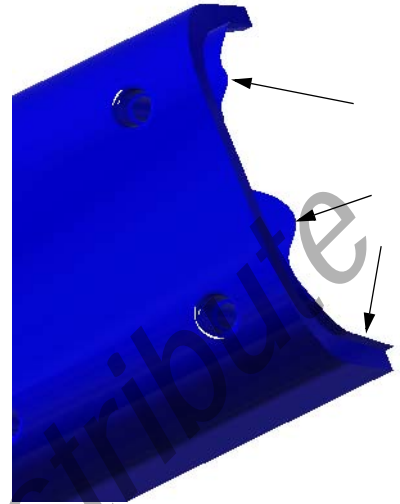
Critical Edges Must Stay Sharp!

Flash

The highlighted edges in the previous diagram are critical tooling edges. These edges need to be kept sharp or the plastic part may acquire *flash* around it's edges during the molding process.

Flash is unwanted plastic that forms around the parting lines when the sharp edges are not created properly or when the mold does not seal properly.

To avoid this situation, this electrode will be cleared back so that it only burns one area of this cavity. The area that is not burned by this electrode would then have to be modeled on another electrode and then burned separately. Burning these areas separately will ensure that these edges stay sharp. To keep the edges sharp, the first electrode can be orbited from the top plane, and then the secondary electrode can be orbited from the side plane.

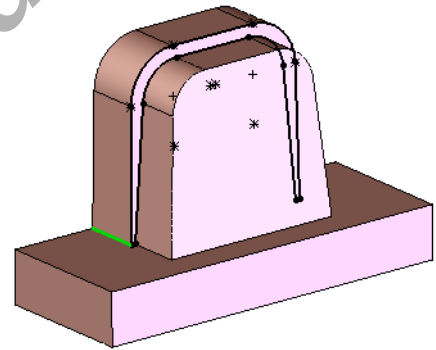


This Plastic Part has Flash.

10 Clear back the electrode.

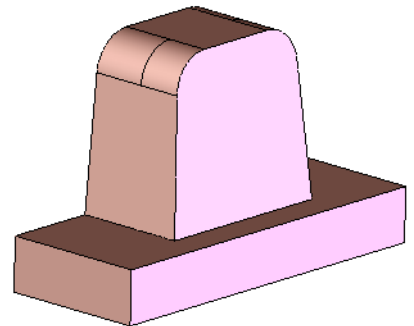
Open a sketch on this face and convert the edges.

Cut-Extrude specifying the **Through All** end condition using the highlighted edge as the **Direction of Extrusion**.



11 Examine the finished Electrode.

The electrode can now be used to machine this area in the cavity and the critical edges will remain sharp.



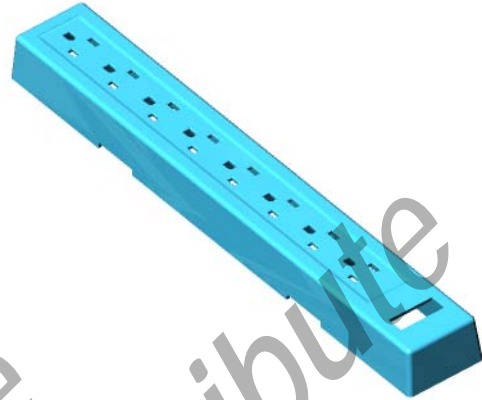
12 Save and close all files.

Exercise 26: Tooling for Plastic Power Strip

Create the tooling for the plastic power strip.

This lab reinforces the following skills:

- Checking for correct draft.
- Establishing the parting line edges.
- Closing off open windows and holes.
- Creating parting surfaces.
- Creating interlock surfaces.
- Splitting the tooling into separate bodies.

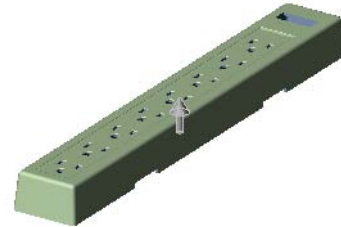


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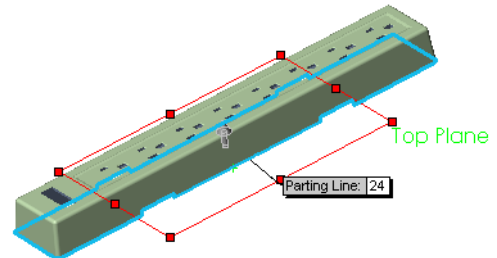
Procedure


Open the part named Power Strip.

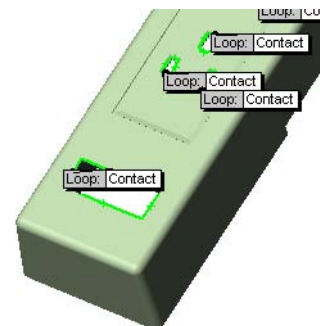
- 1 **Check the part for the correct draft.**
Use **Draft Analysis** to check the draft on the part. Make sure all surfaces have at least **2°** of draft on them. Use the **Top Plane** for the **Direction of Pull**.



- 2 **Determine the parting line edges.**
Use the **Parting Lines** command to establish the parting lines around the perimeter of the part.



- 3 **Fill in the open areas on the plastic part.**
Click **Shut-off Surfaces**  on the Mold Tools tool bar and examine the patch callouts on the model. Set all patches to **Contact** by clicking the button from the **Reset All Patch Types** options.



4 Create the parting surface geometry.

Create the **Parting Surfaces** with a **Distance** of **0.5"**. Make them perpendicular to the pull direction of the mold.



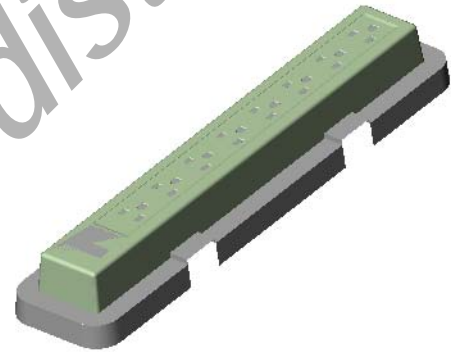
5 Create tapered interlocks.

Create **Ruled Surfaces** around the perimeter of the parting line.

Set the **Angle** to **5°**.

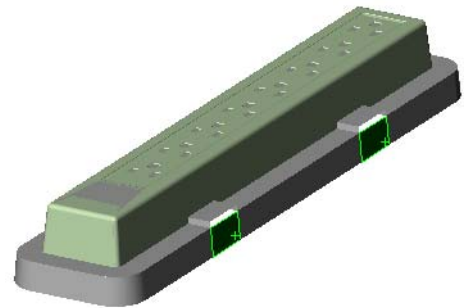
Set the **Distance** to **0.625"**.

You can create all four ruled surfaces in one step.



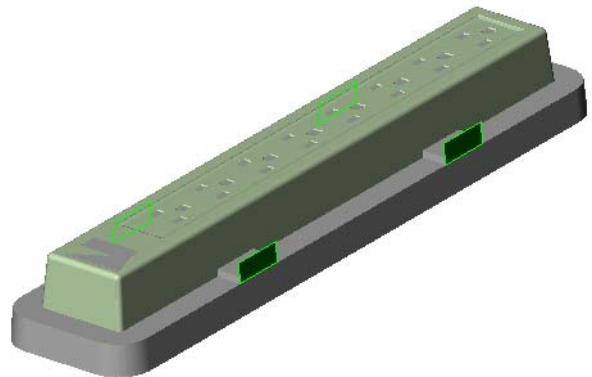
6 Complete the interlock surfaces.

Use the **Lofted Surface** command to fill in the openings in the interlock surfaces.



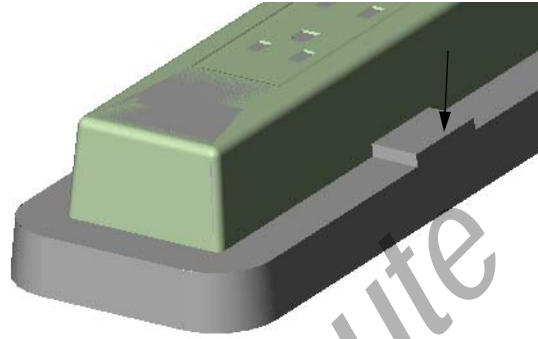
7 Fill in the remaining gaps.

Extend the ruled surfaces to fill the remaining openings where the parting line jogs upward.

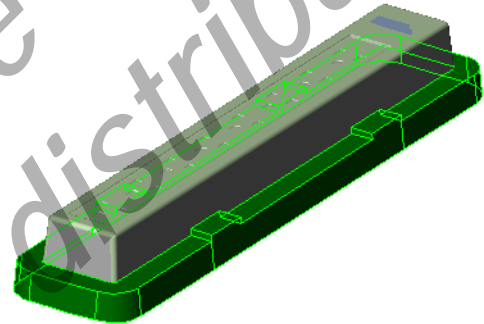


Tip

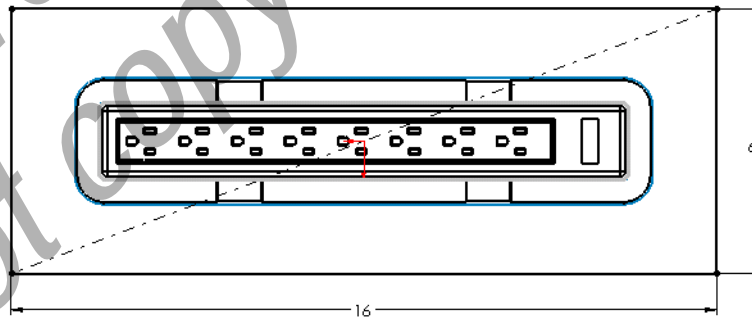
- 8 Mutual trim the surfaces.**
Use the **Trim Surface** command to trim the excess surfaces back to the parting surfaces.



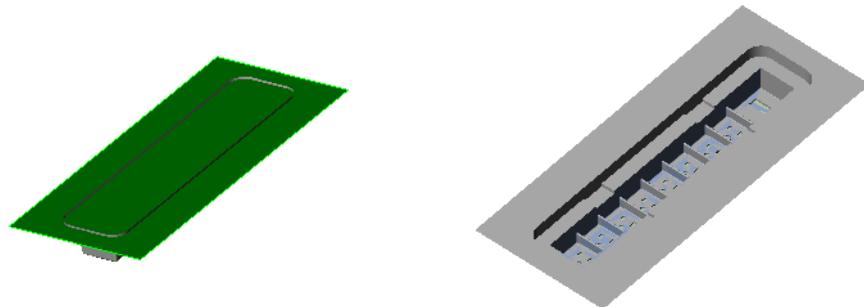
- 9 Knit all the surfaces together.**
Select all the surfaces in the Parting Surface Bodies folder and **Knit** them together.



- 10 Create a planar surface.**
Create a planar surface on a reference plane **0.5"** below the Top plane.



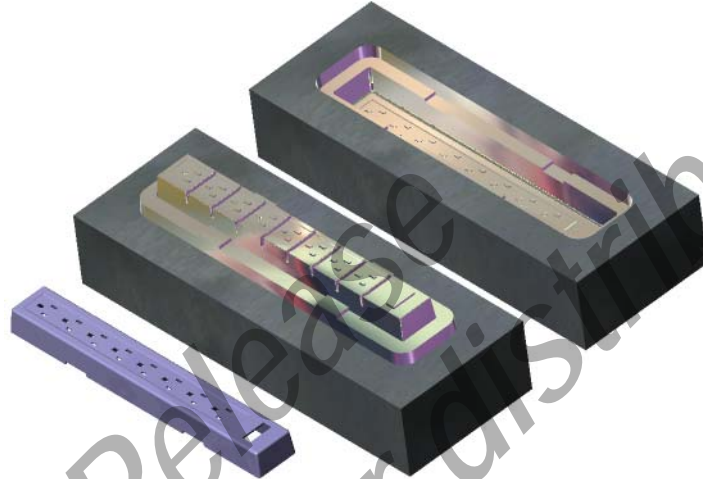
- 11 Mutual trim the surfaces.**
Rotate the part and mutually trim the planar surface to the interlock surfaces.



12 Split the tooling into separate bodies.

Use the **Tooling Split** command and create the core and the cavity for this tooling.

Optionally, create an assembly from the resulting bodies.



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**Exercise 27:
80mm Fan Bezel**

This exercise is an excellent example of a mold with multiple parting directions. The tooling, already created from the **Tooling Split** command, consists of a cavity and a core. In this case think of the tooling as an upper and lower core. This example also shows how you can create complex shut-off surfaces to shut off the primary tooling. These shut-off surfaces are used to interlock the upper and lower cores.



This lab reinforces the following skills:

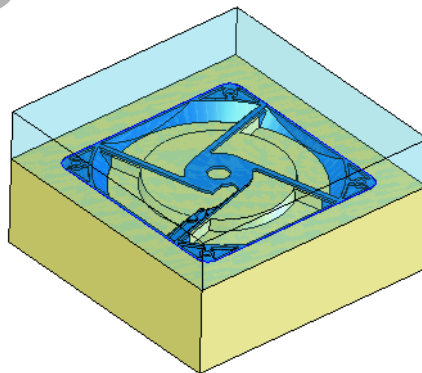
- Checking for undercut.
- Creating side cores.

Procedure

Open the tooling already created for the 80mm Fan Bezel.

1 Open an existing part.

Open the 80mm Fan Bezel. The tooling for this part was created except for the side cores.



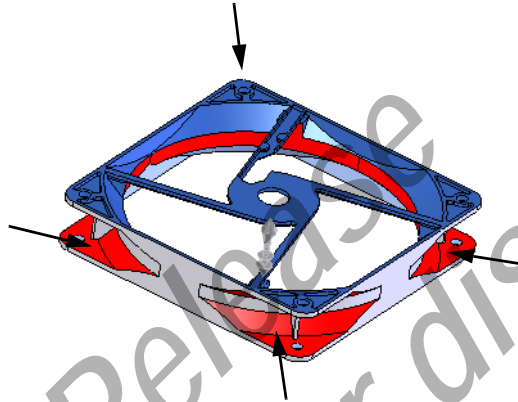
2 Roll back to the Parting Line1 feature.

Rolling the model back to this position puts the model back to the state it was in before any tooling was created.

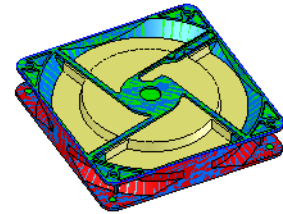
3 Undercut analysis.

Perform an undercut analysis on this model. Use the Top Plane as the **Direction of Pull**. The analysis finds several areas with red faces. The red faces on the outside faces of the part require side cores. The red faces on the inside of the part are formed by the core and cavity, consequently, side cores are not necessary in these areas.

Click **OK** and keep the face colors when prompted.

**4 Show Surface Bodies.**

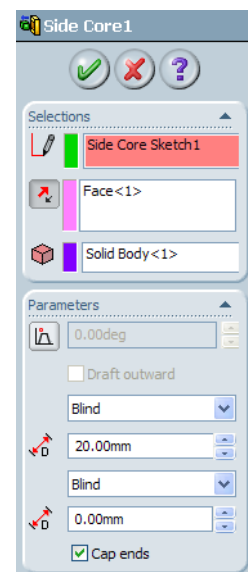
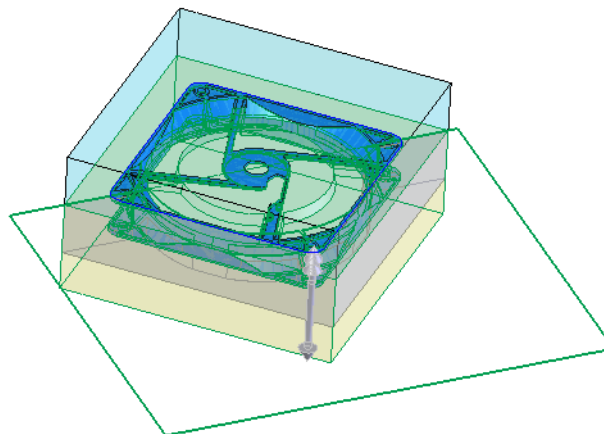
Roll the FeatureManager design tree forward past the Complex Shutoff folder. Show all of the surface bodies in the Surface Bodies folder to understand how this complex shut-off was surface modeled.

**5 Roll to End.**

Roll the model forward to the end of the FeatureManager design tree.

6 Create a side core.

Select Side Core Sketch1 and create a side core with the **Core** command. Use the main core as the solid body to subtract the side core.



Important!

Notice how the side core sketch was created. Seemingly, 4 side cores

are required. However, this geometry is forgiving enough, allowing you to create two side cores instead of four.

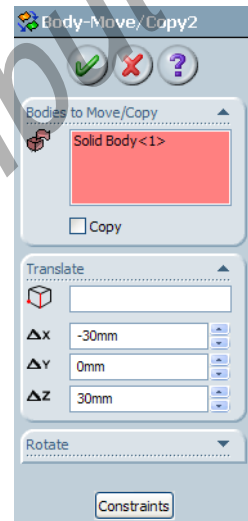
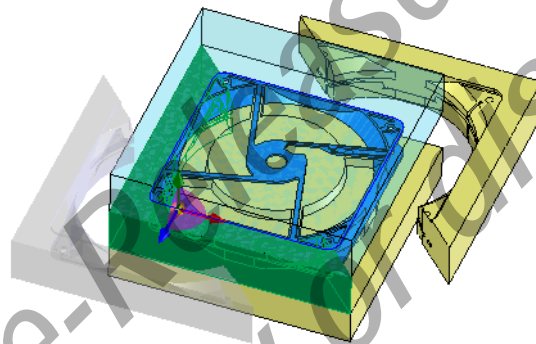
Creating less tooling will reduce the engineering and manufacturing costs needed to create the mold.

7 Create another side core.

Select Side Core Sketch2 and create another side core using the same settings in the previous step.

8 Examine the solid bodies.

Use the **Move / Copy** command to move the side cores away from the tooling.



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