

# SolidWorks®

## SolidWorks Workbook Advanced Modeling

Version 2009

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# Introduction

SolidWorks modeling software is used around the world to design products, develop machinery, and create production systems. The success of the software's application in industry has led to its growing presence in education. Mechanical engineering, industrial design, and transport technologies are just a few of the functions in which SolidWorks software is successfully used as an advanced tool by designers and engineers.

The software enables you to define the shape and exact dimensions of a product design, analyze it with integrated simulation tools, and generate photorealistic images. New users can learn quickly and be up and running in no time thanks to a very intuitive user interface.

This tutorial covers advanced topics and presumes you already have experience using the program. If SolidWorks software is new for you, you can acquire the basic skills necessary for completing this tutorial by completing the lessons in the **Student Workbook** which is part of the educational materials published by the Dassault Systèmes SolidWorks Corp.

Lessons 1 and 2 of this tutorial cover the modeling of thin-walled parts. Industrial products for the consumer markets — such as cars, mobile phones, and mp3 players — frequently contain multiple plastic components. These components typically have small thicknesses (1-3 millimeters) compared to their overall size. In these two lessons you will learn techniques for modeling these types of parts successfully with SolidWorks software.

Lessons 3 and 4 of this tutorial cover the modeling of parts with complex shapes. Introductory lessons deal with creating primarily rectangular products, which are refined with fillets or chamfers. In these chapters, however, you will learn how to create more “free-form” geometries, which are commonly used in the design of modern products.

As its name suggests, SolidWorks software is a solid-modeling system for modeling design volumes in a three-dimensional space. It can, however, be used quite successfully for modeling surfaces and creating surface models. Lesson 5 shows you how you can define a product's shape by starting with sketched edges and cross-sections, combining these into surfaces, and connecting the surfaces to a complete free-form surface model.

## Structure of this Workbook

The lessons in this workbook cover the skills you need to create more complex models. Basic training guides deal with more rectangular or circular shapes. In this manual you will learn the techniques that will allow you shape your designs more freely.

### Complex Shapes

The Advanced Modeling workbook contains three major topics contained in five lessons. Lessons 1 and 2 explain the creation of a thin-walled product. Lessons 3 and 4 contain a tutorial that explains the use of sweep and loft features. Lesson 5 shows you how to build a model from surfaces.

#### Lesson 1, Thin-Walled Parts: Part One

This chapter explains how you can model thin-walled parts. Topics include draft angles, the use of library features, using references to other parts, and more.

#### Lesson 2, Thin-Walled Parts: Part Two

This lesson continues the work from Lesson 1, and explains techniques for creating ribs, extruding thin-walled features, making cross-sections, and adding recessed text.

#### Lesson 3, Advanced Shapes: Part One

This lesson involves modeling a part using sweep and loft features. You will then add a screw thread to the model using a helix feature. Finally, you will learn how to round the part with sweep operations along model edges.

#### Lesson 4, Advanced Shapes: Part Two

This lesson includes more modeling exercises with extensive use of the loft feature. Also, direction is provided on how to complete modeling tasks when sweep or loft features initially do not provide a solution.

#### Lesson 5, Surface Modeling

In this lesson, you will use surface modeling techniques to create a model of a detergent bottle. Certain parts have such complicated curvatures (e.g. cars, housings, plastic parts, etc.) that it is easier to create them using a surface-modeling approach rather than with the traditional and typically effective feature-extrusion approach. First, you will create surfaces. You will then continue with trimming operations to ensure correct connections. Finally, you will transform your surfaces into a solid model.

## Prerequisites

To complete the tutorials provided in this workbook, you will need the following:

- A computer with SolidWorks and the SolidWorks Online User Guide installed.
- The set of files needed for these exercises, which are available from the same source as this document.
- This document, *SolidWorks Workbook Advanced Modeling*.

## Where to Start and How Much Time Will It Take?

SolidWorks is a very user-friendly design program, but you will more than likely be able to use it more efficiently with some practice. You can increase your basic skills by executing assignments from the *SolidWorks Student Workbook*. Its eleven lessons will take about one hour each to complete.

The lessons in this workbook cover three advanced modeling topics: thin-walled parts, complex (bi-curved) shapes, and the art of surface modeling. There are five lessons. On average, each one takes two hours to complete.

The lessons are written in tutorial style, providing a step-by-step explanation of your actions. Completing the instructions without focusing on the ideas involved will enable you to grasp the purpose and utility of the commands you use. It is important to understand the commands, so you are able to use them independently in your own designs.

## Other Sources of Information

- SolidWorks Student Workbook
- SolidWorks User's Guide, Dassault Systèmes SolidWorks Corp.



# Thin-Walled Parts: Part One

Most plastic parts are injection-molded. To use this manufacturing process successfully, the parts must be thin-walled. Creating thin-walled parts involves some common sequences and operations, whether they are cast or injection-molded. Both shelling and draft capabilities are used in this exercise, as well as ribs and other thin features.

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

- Create thin-walled parts.
- Create parting lines and apply draft to model faces.
- Use base parts.
- Make cut features with an open contour sketch.
- Find dependencies and external references for parts.
- Perform shelling operations.
- Create and insert Library Features.
- Create linear patterns of features.
- Add virtual sharp symbols.
- Resize features by dragging.



## Case Study: Mouse Cover

This example will follow the steps for creating a mouse cover, starting with a **Base Part**.

### Stages in the Process

Some of the key stages in the modeling process of this part follow:

- **Creating a Base Part**

When you insert an existing part into a new part, it is inserted as a **Base Part**. Changes to the original will propagate to the copy.

- **Draft with a Parting Line**

Draft can be defined with respect to a parting line and pull direction.

- **Using a Base Part**

Two copies of the **Base Part** will be used: one for the top half of the mouse cover and one for the bottom.

- **Creating a Centered Plane**

This part contains several features that are aligned to the centerline of the part itself. A centered plane is used for locating features.

- **Shelling**

Shelling opens up one or more faces of a part to hollow it out. A shell feature is a type of applied feature.

- **Design Library Features**

Library features allow you to create and reuse commonly used cuts and bosses.



### Design Intent

The design intent of this model follows.

- The button holes are all of equal size and evenly spaced.
- The shell uses two different wall thicknesses.
- The boss on the underside is centered.
- The ribs are of the same general shape, but some are different sizes.



# Creating a Base Part

Using **Base Parts** allows you to use one part to form several others. Changes to a base part propagate out to the parts into which it is inserted. These are called **Derived Parts**. In this example, the base part is the shape of the upper and lower halves of a mouse. We will insert the base part into two new parts and use cuts to form the upper and lower halves. The base part is constructed from a sketch on the Right Plane, which is extruded.

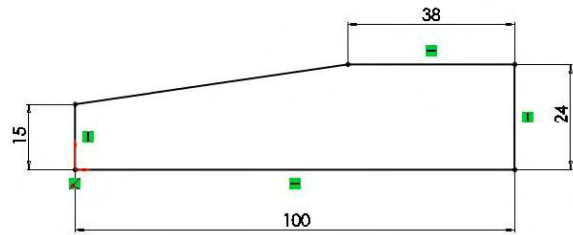
## Create the Main Body

Create the main body using a line sketch and an extrusion.

### 1. Open a New Part with Model Units Set to Millimeters

### 2. Initial Sketch

Create the line geometry that represents the entire body, upper and lower halves included. Sketch this on the **Right Plane**. The lower left corner should be the origin.



### 3. Extrude the Sketch

Extrude the sketch as a boss 75 mm as shown.




### 4. Save the Part

Save the part as **M-Base.SLDPRT** in your own working directory.

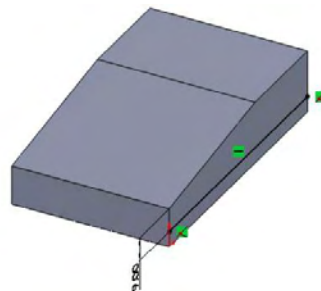
## Review of Draft

In this lesson we will use a **Split Line**. We then apply a draft to faces with respect to a neutral plane. SolidWorks also allows you to apply draft with respect to a parting line and the pull direction of the mold.

### 5. Sketch the Split Line

Select the right face of the base feature and pick the **Sketch** icon to begin sketching. Create a line across the right face, dimensioned from the upper corner of the model. Watch the cursor as you sketch the line and be careful not to add any unintentional relations, such as midpoint .

### 6. Close the Sketch



## Creating the Split

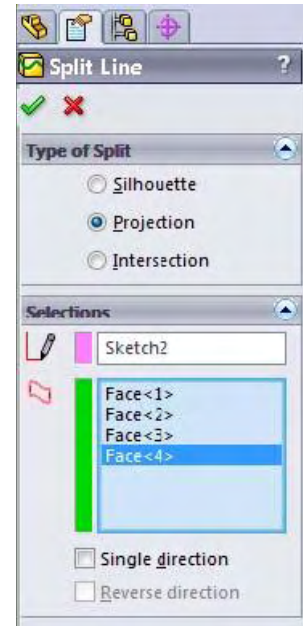
Once the split line curve(s) have been sketched, they can be used to split the faces.

### 7. Projection Split Line


Click on **Insert, Curve, Split Line...** and select the **Projection** option in the **Type of Split** area. This option projects the curve through the model faces.

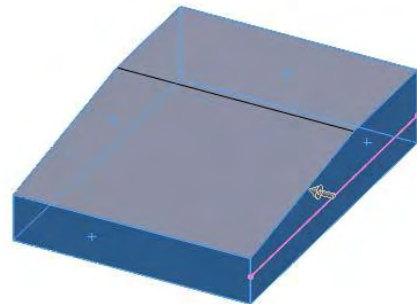
### 8. Select Faces

In the **Selections** area click in the **Sketch to Project** box and select **Sketch2** from the FeatureManager design tree (click the title of the PropertyManager to display the Fly-out FeatureManager design tree). Click in the **Faces to Split** list and choose the faces of the model that will be split as the curve is projected through them. Select all four faces around the perimeter of the model.



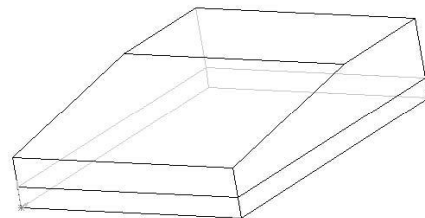
### 9. Faces Selected

Click the **Single Direction** option. The arrow should point into the model, if not select **Reverse direction**. Click **OK**  to split the faces.



### 10. Resulting Faces

The selected faces are split in two by the projected curve. The solid remains a single solid.

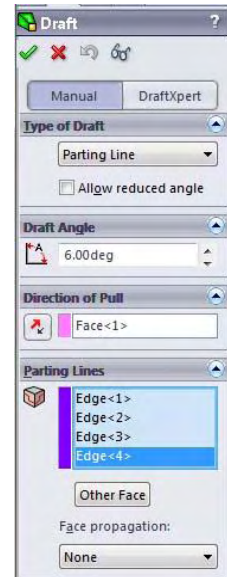


## 11. Draft PropertyManager

Click **Insert, Features, Draft...** from the menu or pick

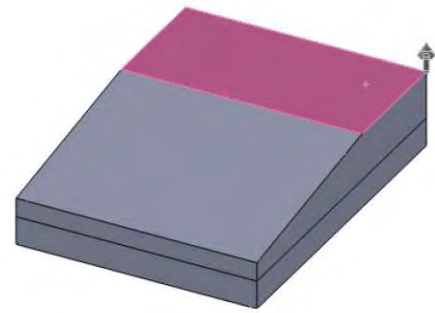


from the Features tab. Choose the **Parting Line** option from the **Type of Draft** pull-down menu. Set the **Draft Angle** to **6°**.



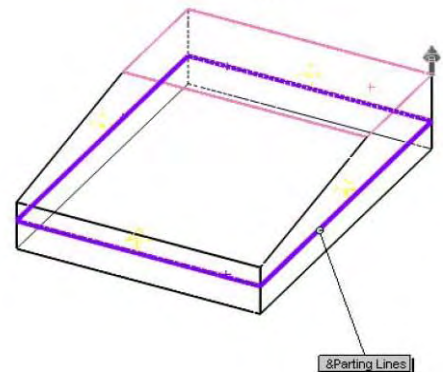
## 12. Direction of Pull

Select a face on the model as shown to set the **Direction of Pull**. The arrow should point away from the volume and the entry **Face<1>** should appear in the **Direction of Pull** list. In the graphics area the face is identified by a different color. This is the direction that the mold will be pulled to remove the part from the mold.



## 13. Parting Lines

Click in the **Parting Lines** list and select *the edges* created in the Split Line operation. The system figures out on which faces to apply draft based on the parting lines and the direction of pull. Press **OK** to create the draft.

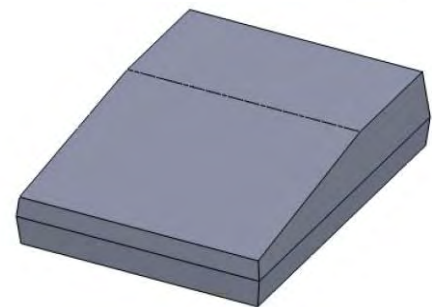


## 14. Complete Draft

The draft is added to the appropriate faces. All of the faces are drafted the same amount with respect to the pull direction.

## 15. Repeat for Bottom Four Faces

The picture shows the result after the bottom four faces (with an opposite direction of pull) have been drafted.



## 16. Save and Close the Part

# Using a Base Part

A **Base Part** lets you use a previously created part as the base feature for a new part. In this example, the part with split lines and draft will be inserted into two new parts: one for the top half of the mouse, the other for the bottom half. The excess portion of each will be cut off.

## Insert Part

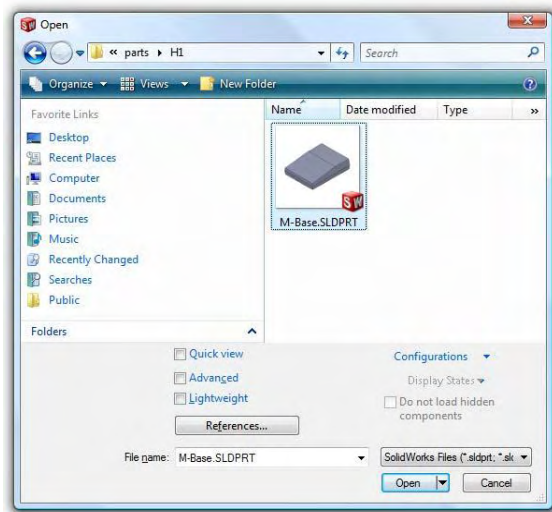
**Insert Part** allows you to insert one or more base parts multiple times into a new part. The original part becomes a single feature in the new part. Changes to the base part are transferred into the existing part. To find it choose from the pull-down menu: **Insert, Part....**

### 17. Open a New Part with Model Units Set to Millimeters

This is the empty part into which we will insert the base part.

### 18. Inserting a Base Part

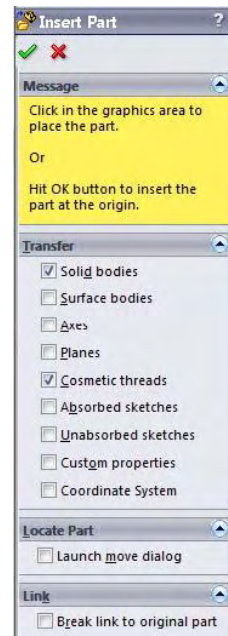
Click **Insert, Part...** from the pull-down menu. Use the browser to locate and select the part **M-Base.SLDPRT** that you just created. Click **Open**.



The **Insert Part** PropertyManager appears.

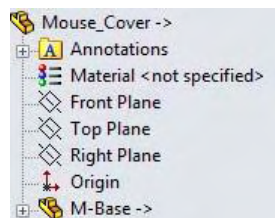
For this lesson, the options listed are not important, so we leave them all unchecked. See the *SolidWorks Online User's Guide* for a description of the listed options.

Click **OK** .



### 19. The Base Part

The base part is inserted into the active part. Save this part as **Mouse\_Cover.SLDPRT**. The FeatureManager design tree lists it as a single feature: **M-Base part ->**. The arrow -> indicates that the feature references another source. The part itself has the same arrow.





## Cutting with an Open Contour Sketch

Open contours and single lines can be used as extruded cuts on a model. In this example, a single line is used to split the part at the parting line. This splitting method will ask you to identify which side of the model is to be removed. This will be indicated with an arrow.

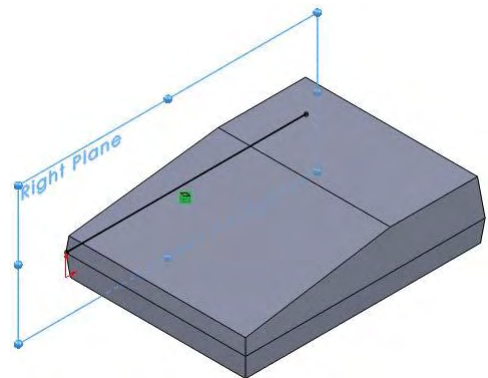
### Convert Entities

**Convert Entities** enables you to copy model edges into your active sketch. These sketch elements are automatically fully defined and constrained with an **On Edge** relation.

Convert Entities can be found on the Sketch tab, using the  icon; or from the pull-down menu by choosing **Tools, Sketch Tools, Convert Entities**.

## 20. Sketching the Cut Line

Click the **Right Plane**, insert a sketch, and use **Convert Entities** to copy the parting line into the sketch. **Convert Entities** creates a fully defined sketch element. If the dimensions of the part change, the converted edge will automatically change with it.

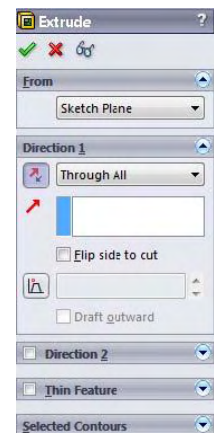


## 21. Extruding the Cut

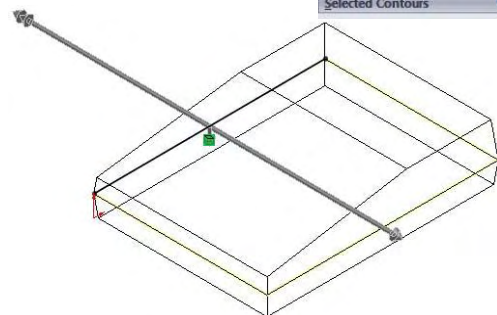
Click **Insert, Cut, Extrude...** from the menu or click the



**Extruded Cut** icon. Using a single line as a **Cut** forces some options to be set automatically. The **End Condition** of **Direction1** is set to **Through All**. Options such as Blind and Depth are not available. Since the sketch line is on the outermost side of the part, the cut only needs to be made in one direction. So, if **Direction 2** is checked, clear it.

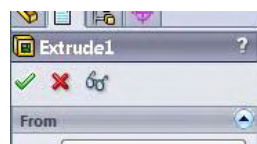


**Flip Side to Cut** and **Reverse Direction** must be closely monitored. The line will extrude through the model in a direction normal to the sketch plane. The preview arrow indicates which side of the model will be removed. Since we are creating the top half, make sure the arrow is pointing down.



### Disjoint Feature

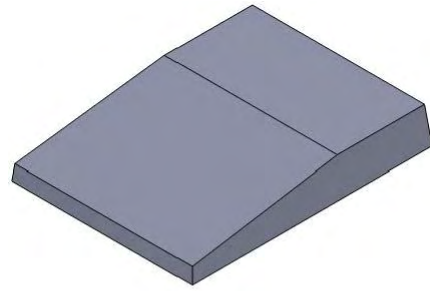
If, when you press **OK**, you get a warning message, toggle **Reverse Direction** and press **OK** again.



**Rebuild Errors**  
The intended cut does not intersect the model. Please check the sketch and direction.

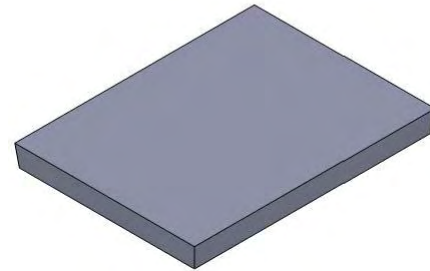
## 22. Top-Half Model

The result of the cut is the top half of the model, including everything above the parting line. We will work this into a thin shell solid, but it will remain related to the base part from which it was copied.



## 23. Bottom-Half Model

Using the same procedure, open another new part, and insert the base part. This time the upper half is removed to create the model of the bottom half. Name this one **Mouse\_Base.SLDPRT**.

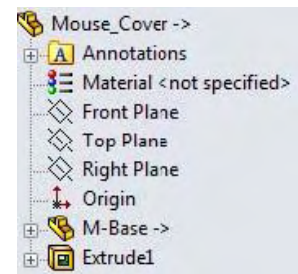


## Finding External References

The system automatically established references between the base part and the derived parts. You can search for any existing references by using **List External References**.

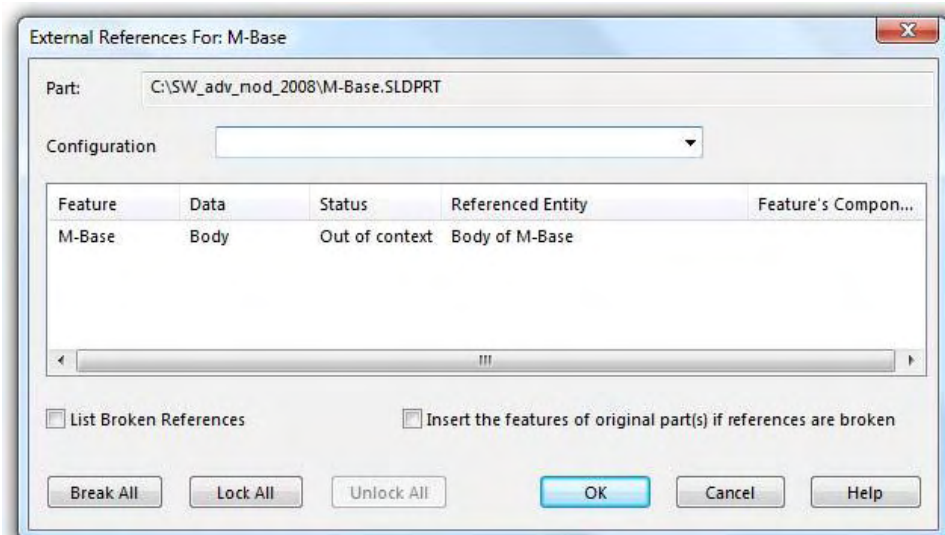
## 24. FeatureManager Design Tree

The FeatureManager design tree indicates the relationship to another part, the base part, by arrows next to the part itself (**Mouse Cover->**) and the base part (**M-Base->**) feature. The full path names of these references can be determined by using **List External References**.



## 25. External References

Select the **Mouse Cover->** (top level) feature name and select **List External Refs...** from the shortcut menu. The title bar lists the feature/part. Its information follows. Information includes the full path name of the reference, the feature name, data (type of information), and referenced entity.





## Creating a Centered Plane


This part contains several features that are aligned to the centerline of the part itself. We need to create a centered plane that can be used for locating features and for measuring. Midpoints will be used to define the plane.

### Plane through Three Points

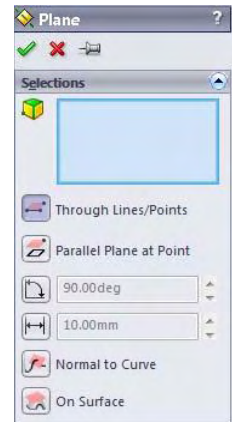
Another way to create a plane is by using three non-collinear points. These points can be defined as actual sketch points, midpoints of lines or edges, or vertices (endpoints). Make sure to select three locations.

#### 26. Insert Plane

Click **Insert Reference Geometry, Plane...** from the menu

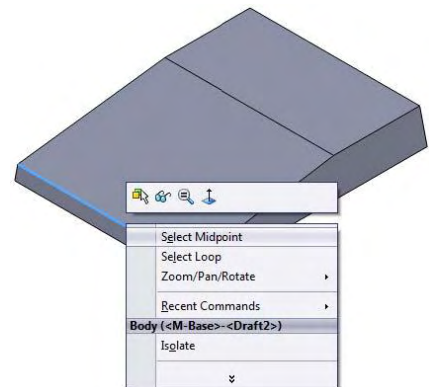
or use the  icon from the **Reference Geometry** dropdown box. The Plane PropertyManager will appear.

Select the **Through Lines/Points**  option.



### Selecting Midpoints

Rather than making sketches containing individual point entities to create centered locations, midpoints of edges can be selected directly from the model. **Select Midpoint** selects the midpoint of an edge. It is particularly useful with the **Insert Plane** command. Use the right mouse-button menu while the cursor is over an edge to choose **Select Midpoint**.

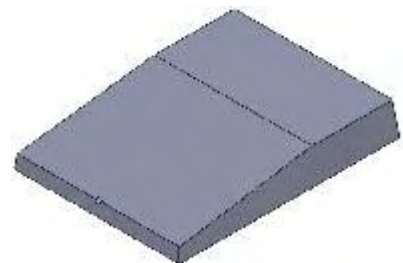


#### 27. Midpoint

Select the midpoint of the front upper edge by floating the cursor over the edge and clicking **Select Midpoint**.

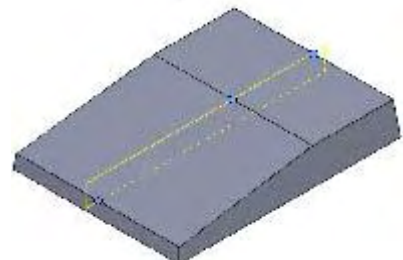
#### 28. Resulting Midpoint

The midpoint appears on the edge as a blue dot.



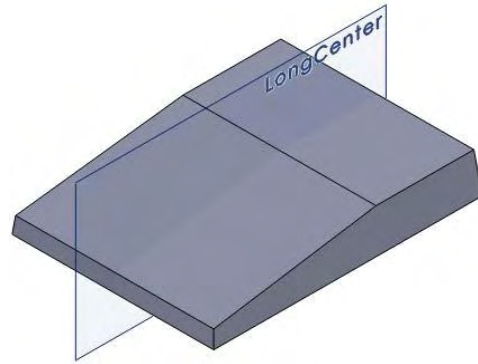
#### 29. Select a Total of Three Midpoints

Select two more midpoints. As you select them, they are added to the selection list. Click **OK**.



### 30. The Completed Plane

A plane can be resized by dragging its corners or sides. Resize the plane to resemble the illustration on the right. Name the new plane **Long Center** in the FeatureManager design tree.



**Question:** Given the options shown in the **Plane** PropertyManager, is there another way we could have defined the centered plane?

**Answer:** Yes. We could have selected the Right reference plane and one midpoint and used the

**Parallel Plane at Point**  option.


### Testing the Plane

Check the effectiveness of the plane by changing the base part. The change makes the part narrower, forcing the plane to update. The plane should remain centered regardless of the value chosen.

### 31. Open the Base Part


Open the base part by selecting the M-Base feature and **Edit In Context** from the right mouse-button menu. This will automatically open the referenced part.

### Move/Size Features

Drag handles can be used to lengthen/shorten the extrusion distance dynamically, or to rotate/scroll the sketch. They appear when you turn on **Move/Size Features** and double-click on an extruded feature. **Move/Size Features** activates the display and use of drag handles for an extruded solid. Dragging the appropriate handle will change the extrusion depth, or move or rotate the feature's sketch. It can be found: From the Features tab pick the  icon. If the icon does not appear click **Tools, Customize, Commands**. Select **Features** and drag the icon to the Features tab.

### 32. Move/Size Features

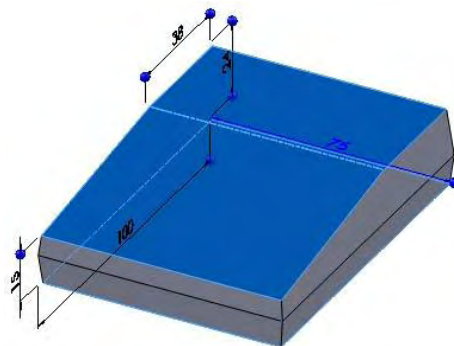
Toggle on **Move/Size Features** using the toolbar icon. The icon will appear in the

Pressed-in mode: .

### 33. Feature Handles

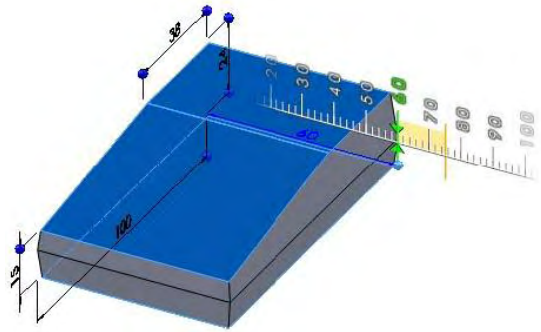
Rotate the part and click the base feature to view the drag handles (A double-click on the top face will show the dimensions and the drag handles).

The drag handles are used to lengthen or shorten the extrusion distance dynamically, or to rotate/move the sketch.



### 34. Change the Extrusion Depth

Drag the distance arrow towards the center of the model. The dynamic display at the cursor will show the current depth setting. Release the drag handle at **60 mm**. The part immediately rebuilds.



### Fine Control over the Value

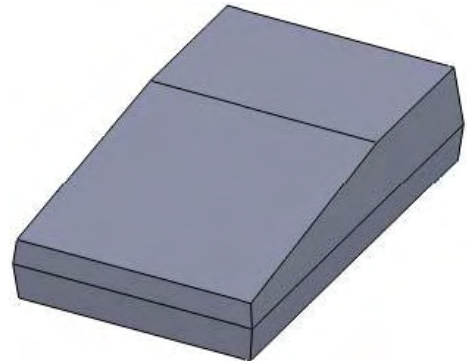
Notice the line extending from the cursor at right angles to the drag direction. If the cursor is moved directly in-line with the drag direction, the depth value changes by large increments - of 10 mm, for example. If the cursor is moved out along the perpendicular line, farther off from the drag direction, the value changes in smaller increments of 5 mm, 1 mm, or 0.1 mm. This gives you an interactive way to control the drag distance precisely.

### 35. Toggle Off

Click the **Move/Size Features** again to turn off the functionality. You could leave it on, but every time you click on a feature, it will bring up the drag handles.

### 36. Changes to the Base Part

The depth of the extrusion has been changed from 75 mm to 60 mm. The change will transfer to both the **Mouse Cover** and **Mouse Base** parts, making them the same width as the base part.



### 37. Return to the Mouse Cover Part

Use the Window pull-down menu to switch back to the **Mouse Cover** part.

# Measuring

Use **Measure** to check the actual distance from plane to edge. The **Measure** option can be used for many measurement tasks. Here, it is used to measure the shortest distance between an edge and a plane. **Tools, Measure** can calculate distances, lengths, surface areas, angles, and XYZ locations of selected vertices.

## 38. Measure the Distance

Flip the view over from **Isometric** using the key sequence **Shift-Up Arrow** twice.


Select one bottom edge and the **Long Center** plane. Click **Tools, Measure...** to display the distance. The **Normal Distance:** reads **30 mm**.


The **Measure** dialog stays active until you click **Close**, this allows you to make multiple measurements.

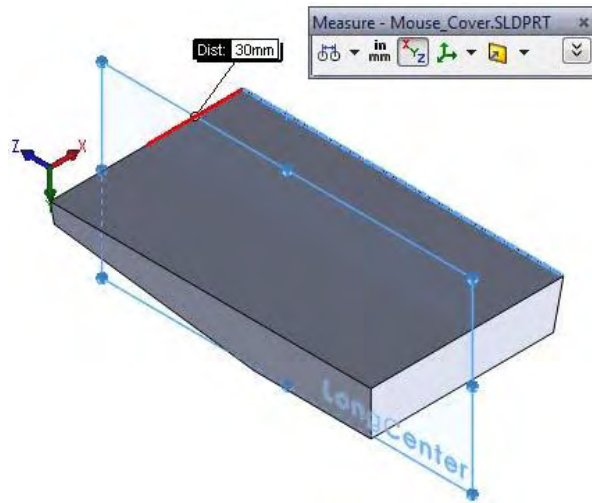
Click **Close** to shut down the dialog.

You can also temporarily deactivate the **Measure** dialog by switching into **Select** mode. This enables you to turn the measurement function off and continue modeling.

There are a couple of ways to switch into **Select** mode. You can:

1. Pick **Select** from the right mouse-button pop-up menu.
2. Pick the **Select**  icon from the Sketch toolbar.

To turn the measurement function back on, simply click inside the **Measure** dialog. When the measurement function is on, the cursor looks like this: .

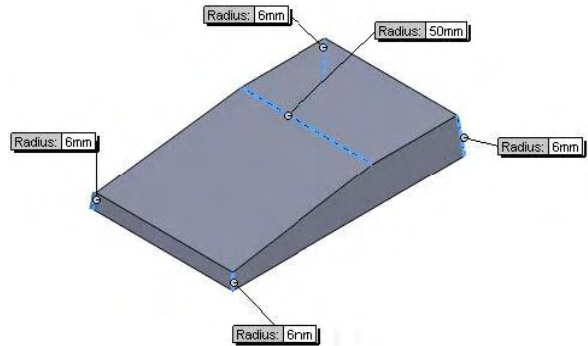


# Shelling

The shelling operation is used to "hollow out" a solid by applying a wall thickness to some faces and removing others. In this case, the top faces will have a greater thickness (3 mm) than the sides (2 mm).

## 39. Adding Fillets

Add fillets of **6 mm** (4 times) and **50 mm** to the solid before shelling if fillets are to be created on the inside. The inside fillets will be smaller than the outside by the wall thickness. The edge fillets are left off in this case because the top faces are to have a larger thickness value.



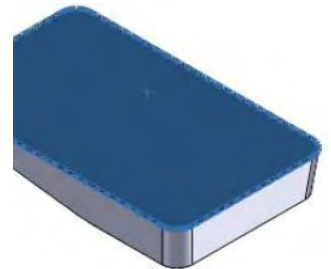
## Insert Shell

**Insert Shell** removes selected faces and adds thickness to others to create a thin-walled solid. You can find this shell command:

- By choosing **Insert, Features, Shell...** from the pull-down menu;
- Or, on the Features Tab by clicking .

## 40. Select the Face to Be Removed

Flip the view over from Isometric using the key sequence **Shift-Up Arrow** twice. Select the bottom face of the model as a face to be removed.

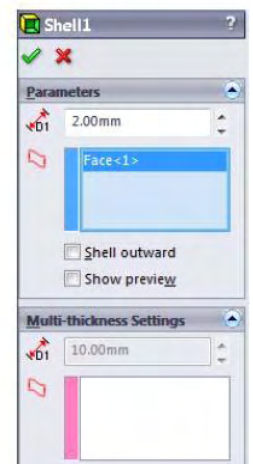


## 41. Shell Command

Click **Shell...** from the **Insert, Features** menu. Set the Thickness to 2 mm as the default. For a constant thickness shell, you would finish the command now. Do *not* click **OK** yet!

Thickness

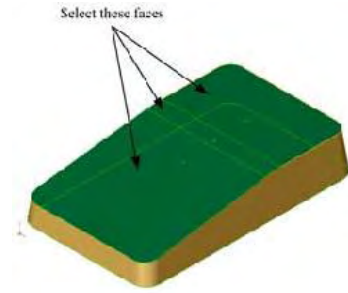
Select the faces.



Click in the **Multi-thickness Faces** list to indicate the additional selections will not be the default thickness.

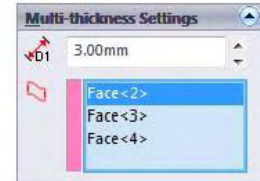
#### 42. Select Thicker Faces

Switch back to Isometric view and select the three top faces. Each selection will add an entry in the **Multi-thickness Faces** list. The entries are **Face<2>**, **Face<3>**, and **Face<4>**.



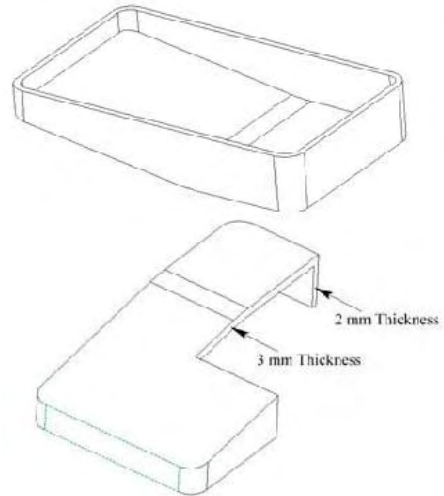
#### 43. Set the Thickness

Click on the **Face<2>** entry and set its value to **3 mm** in the Multi-thickness field. Do the same for **Face<3>** and **Face<4>**. Press **OK** to create the shell.



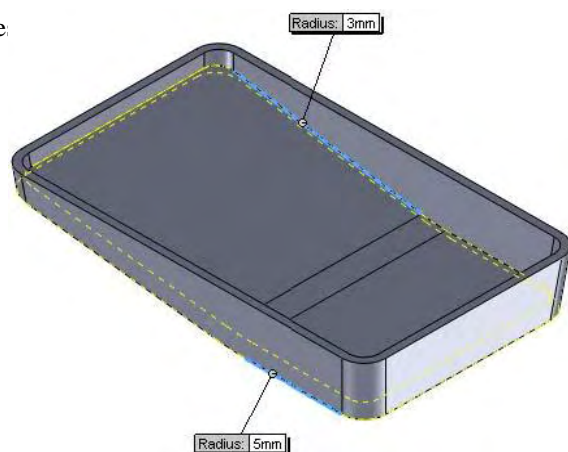
#### 44. Resulting Shell

The shell operation removed the bottom face and applied the 2 mm thickness to all the other faces in the model except the three that were set thicker. Since the **Shell Outward** option was *not* selected, the thickness was applied to the inside of the original solid.



#### 45. Complete Filleting the Shell

Add fillets to the inner (**3 mm**) and outer (**5 mm**) edges of the solid. The fillets require selection of only one edge each using the **Tangent Propagation** option.



#### 46. Save

Save the changes.

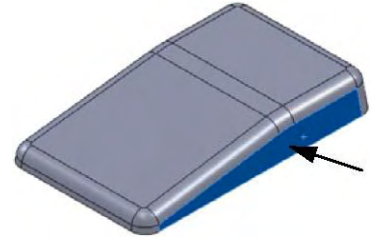


## Adding Virtual Sharps

A **Virtual Sharp** symbol will be added to help locate the sketch of the library feature. The **Virtual Sharp** option generates a symbol at the intersection of two model edges, representing the corner that filleting has removed. There is no icon for this command. The symbols are generated in a sketch by inserting a point entity with the selection of two intersecting edges. There are several styles of symbols available under **Tools, Options, Document Properties, Detailing, Virtual Sharp**.

### 47. Insert Sketch


Create a new sketch on the angled outer face of the part.

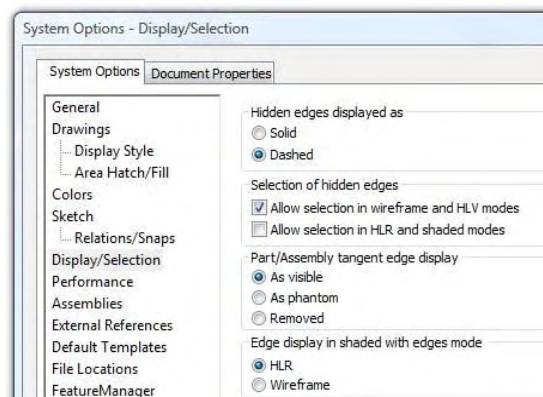


### 48. Shaded with Edges

Hidden-line removed edges can be displayed in the shaded mode. Use **Tools, Options, System Options, Display/Selection** and click **HLR Edges** on. The normal display is **No Edges**.


As a shortcut for toggling the display of HLR edges in shaded mode, click **View, Display, HLR Edges in**

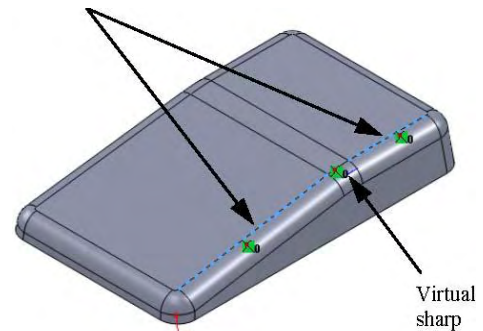
**Shaded Mode**, or click the  tool on the View toolbar. The tool looks just like the Shaded View tool except it is blue instead of yellow.



### 49. Selections

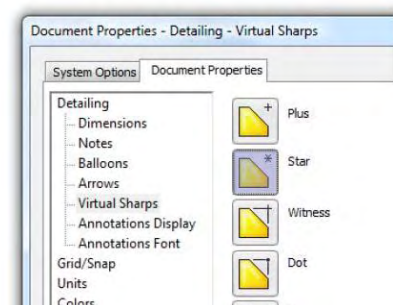
**Ctrl**-select the two edges indicated and

(click) insert a **Point** . The **Virtual Sharp** symbol appears at the apparent intersection of the two edges, on the sketch plane. Name the sketch **virtual sharp**.



### 50. Other Styles


The symbol can have several styles. Under **Tools, Options** on the **Document Properties, Detailing, Virtual Sharps** tab, you can see the available styles. Select the star (\*) and click **OK**.



# Library Features




**Library Features** allow you to create and use commonly sketched shapes as cuts or bosses. These features are named and created as separate files on the system so that they can be inserted into any part. In this example, a rounded slot shape that exists in the feature folder of the **Design Library** will be used in the part.

## Design Library

The Design Library tab  in the Task Pane is the central repository for reusable elements, such as:

- Annotations
- Assemblies
- Library Features
- Parts




The Design Library tab contains the following folders:

-  **Design Library**
-  **Toolbox**
-  **3D ContentCentral**
-  **SolidWorks Content**

A library feature can be added to a part by simply dragging it from the Design Library to the part. Positioning the feature occurs as part of the command.

## Design Library Tools

The following tools are available under the Design Library tab:

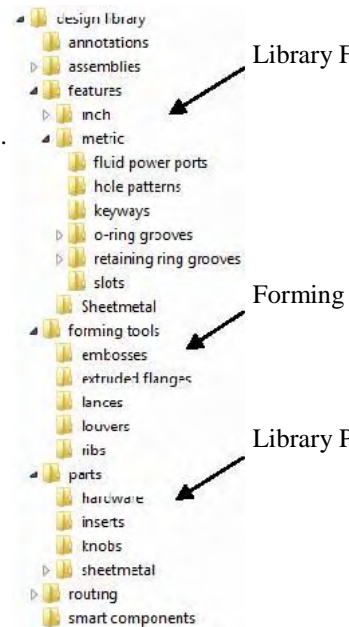
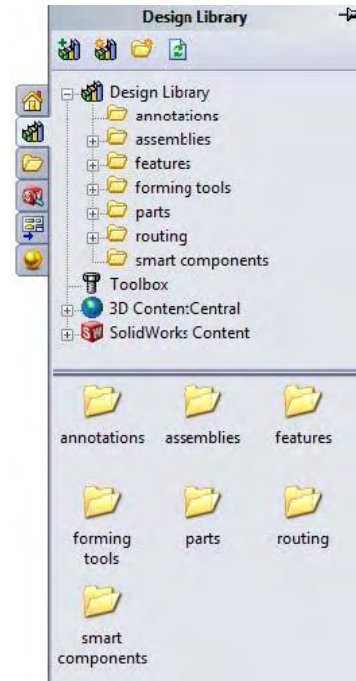
-  **Add to Library**
-  **Add File Location** Add an existing folder to the Design Library.
-  **Add New Folder** Create a new folder on disk and in the Design Library.
-  **Refresh** Refresh the view of the Design Library tab.

## Essentials of Using the Design Library

Taking full advantage of the Design Library requires an understanding of its file structure. Although some library features and parts come with SolidWorks software, the real power of the Design Library is in creating and using your own folders and libraries.

## Main Directory Structure

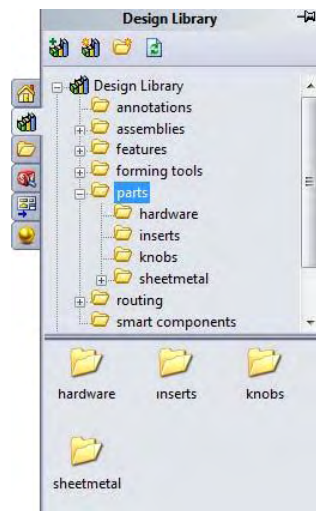
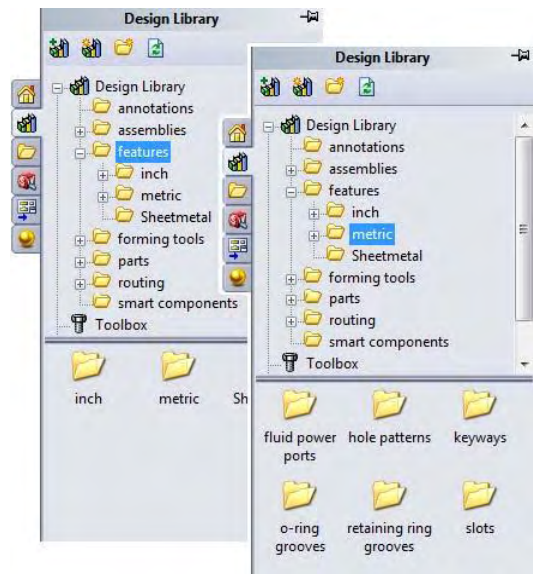
Using the Windows Explorer<sup>®</sup>, browse to the **SolidWorks** folder and open the **design library** folder. The content of this folder matches the content of the Design Library in the SolidWorks Task Pane.





## Features

The **features** directory contains all of the *library features* that are shipped with the Design Library. There are subdirectories for features with imperial units (**inch**) and metric units (**metric**), as well as a **Sheetmetal** subdirectory. The inch and metric subdirectories contain the same features, the only difference is units used. The content of the metric subdirectory is shown at right. All features must be **\*.sldlfp** files.

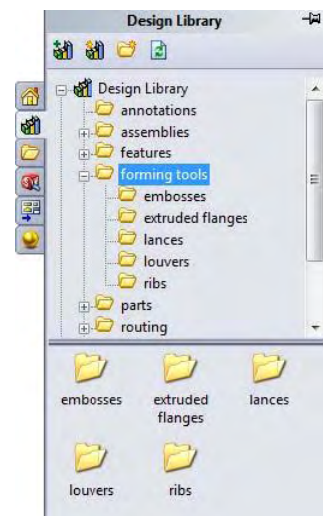


## Parts

Library *Parts* shipped with the Design Library are found in the subdirectories located under the **parts** folder. All of these must be **\*.sldprt** files.

## Forming Tools

For sheetmetal parts, SolidWorks has provided a set of *Forming Tools*. These include various ribs, dimples, louvers, and lances. These files must be **\*.sldprt** files.



## Adding to the Folders

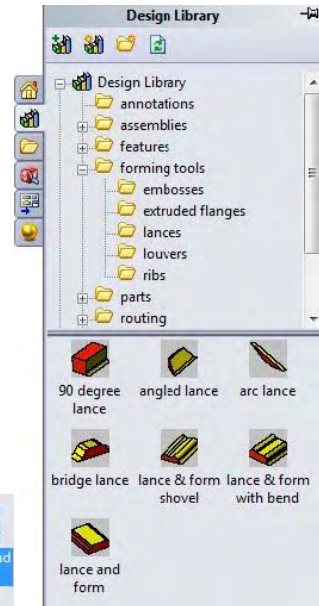
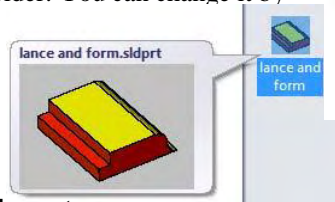
You can add any library feature or part to these folders, and they will appear in the Design Library. They will appear as icons showing their preview pictures. You can drag and drop files into the Design Library from Explorer. Press the delete key to remove them from the palette.

## Sub-Folders and Tabs


Each sub-folder, such as **Lances**, holds the appropriate type of files (\*.sldprt in this case). The files appear as icons in the lower part of the Design Library. You can also add your own sub-folders to the directories in the Design Library.

## Icons

Graphics for the icons are taken automatically from the last saved image of the library feature or part. They can be shaded or wireframe images, but you should zoom in on them for best results. Placing the cursor on the icon will cause a larger preview to appear for a better view. The name of the icon comes from the name of the library feature or part as it appears within the folder. You can change it by clicking on it.

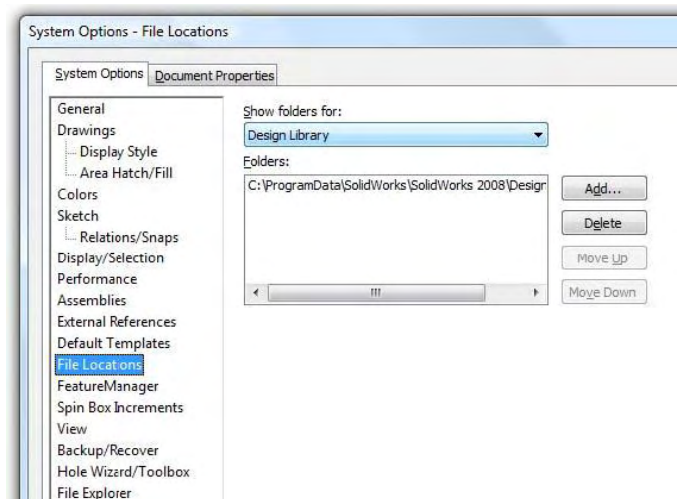


## Setting Up a New Directory

The Design Library comes with several preset directories, each containing several library features or parts. You can add more files to these existing directories, or you can create your own. The tabs are created by adding folders to the **design library** directory structure or by clicking **Add File Location**  and browsing the desired directory on your system.

## Organizing Your Libraries

You can control where SolidWorks software looks for your libraries by setting a search path in **Tools, Options, File Locations**.



## Two Schools of Thought

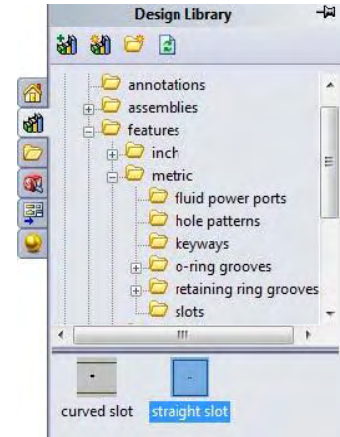
There are two schools of thought regarding the creation of library features. One is to include the necessary locating dimensions and references in the library feature and then “repair” them when they dangle after you drag the feature from the Design Library. The other approach is to *not* include external references in the library feature and then add the necessary ones during the **Edit Sketch** portion of the dialog. In this example, there are no external references to repair, but we do need to orient the sketch and locate it with dimensions and relations.

## Using the Design Library

A slotted hole will be created in the part using the Feature Palette. The feature will represent a button hole in the mouse cover.

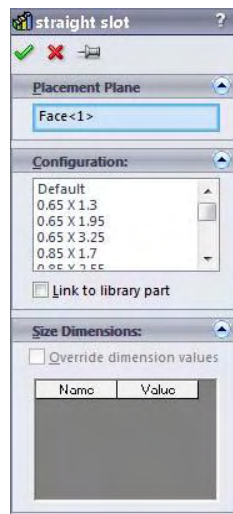
### 51. Open the Design Library

Go to the **features, metric, slots** folder. The icon **straight slot** is the library feature that will be used.

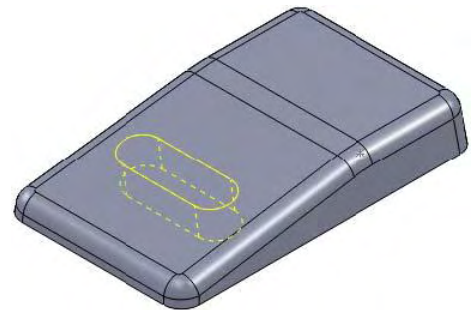


### 52. Drag and Drop

Drag and drop the feature from the Design Library onto the angled face.

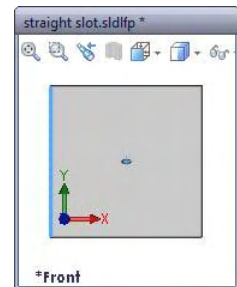


A PropertyManager appears. The **Placement Plane** field lists the face on which the feature was dropped.



In the **Configuration** field select **Default**.

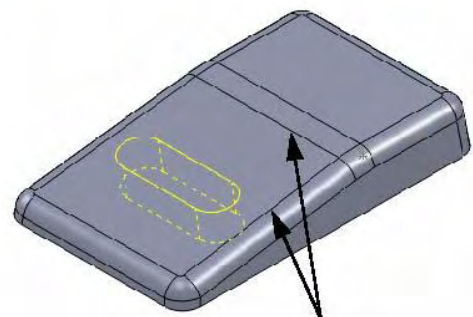
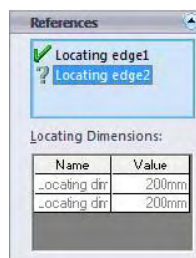
A new window is opened displaying the library feature and the references required for positioning the feature.



### 53. Position Feature

In this step, the orientation and position of the library feature will be set in relation to the part.

First, select the upper edge of the tilted plane. Next, select the right edge.

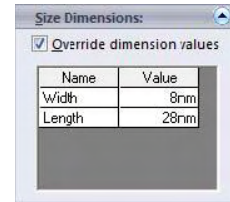


Select these edges.

## 54. Change Feature Dimensions

The default configuration selected earlier for the feature does not have the right sizes. With the PropertyManager we can change the dimensions of the library feature, even before it is created.

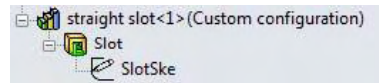
Check the option **Override dimension value** in the **Size Dimensions** field. Enter **8 mm** and **28 mm** respectively for **Width** and **Length**.



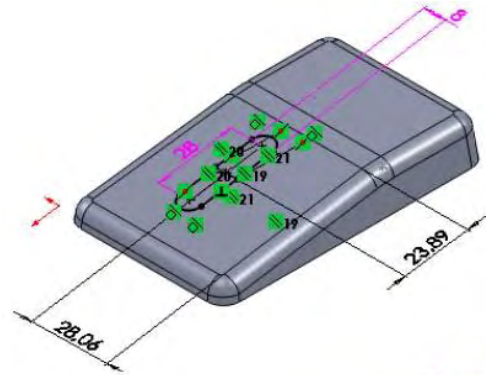
Click **OK**.

## 55. Modify Sketch

Open the sketch **SlotSke** you find within the **straight slot** library feature.

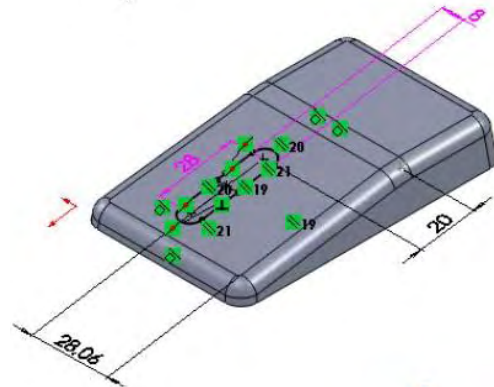


Remove the two position dimensions. They will be replaced by new dimensions.



## 56. Dimension to the Virtual Sharp

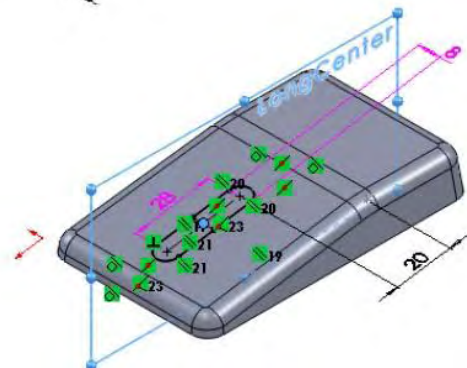
Create a dimension between the virtual sharp and the arc. The virtual sharp behaves like a sketch point.



## 57. Relations

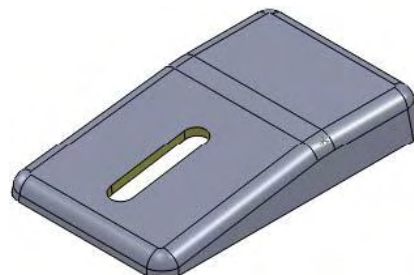
Add a **Collinear** relation between the centerline and the plane **Long Center**.

Leave the sketch.



## 58. Library Feature Completed

The creation and insertion of the library feature is complete. Later, it will be copied to other locations as part of a pattern. The feature is listed as **Slot straight1**, the same name as the icon. Rename the library feature to **Button Hole**.



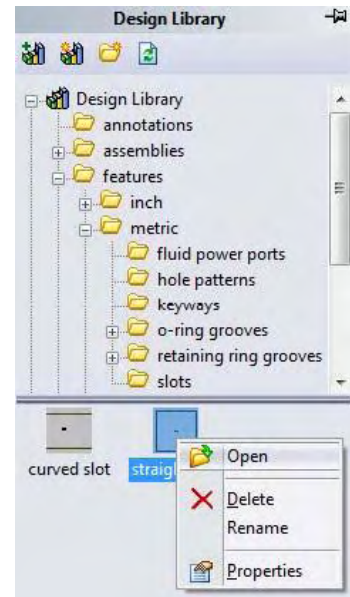


## Editing the Library Feature

Library features can be opened and edited directly from the Design Library. Changes that are made will affect all future copies of the feature, but not those that have already been inserted.

### Edit Library Item

Any Library item (part, feature, or forming tool) can be opened for editing. Position the cursor over the icon and choose **Open** from the right mouse-button menu. The library feature or part will be opened for editing.

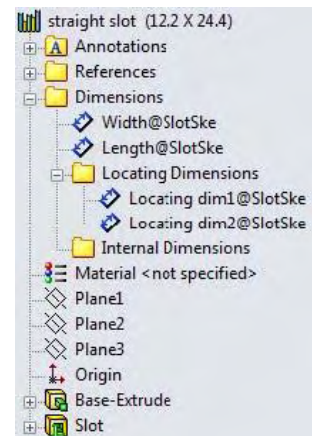


### Dimension Control

You can control access to dimensions in the library feature. Using **Edit Dimension Access**, dimensions can be flagged as **Internal** or **User Dimensions**. To do this, use the right mouse-button menu and **Edit Dimension Access** when editing the library feature.

- **Internal Dimensions** do not appear in the inserted feature, only in the original.
- **User Dimensions** appear in the inserted feature and can be changed.

You can drag the dimensions from one folder to the other.



## Multiple Contour Profiles


In this example, two circles will be created and extruded together as a single boss feature. These features act as pins that connect a button to the **Mouse Cover**.

### 59. Open a Sketch on the Inside Angled Face

Turn the part over so you can see inside. Select the angled face and open a sketch.




### View Normal To

The **View Normal To** option on the **View Orientation** dialog (or the **View Normal To** option  on the **Standard Views** toolbar) is used to change the view to be normal to the selected face or plane. Select the face and double-click **View Normal To**. This orients the view so you see the true size and shape of the face. For more control you can **Ctrl**-select a second face/plane to set as the screen's "Y" direction.

### 60. Normal To

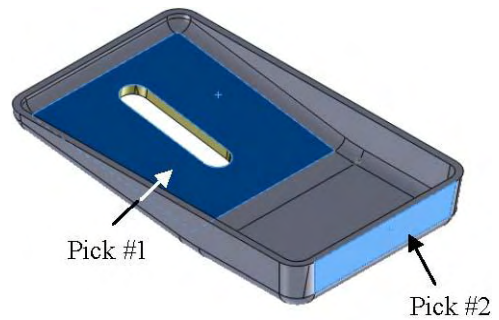
Select the sketch face and double-click **View Normal To**.

Return to the previous view state by clicking **Previous View**  on the **View** toolbar.



### 61. Alternate Normal To

**Ctrl**-select the sketch face (Pick #1) and the orientation face (Pick #2), in that order. Double-click **View Normal To**.



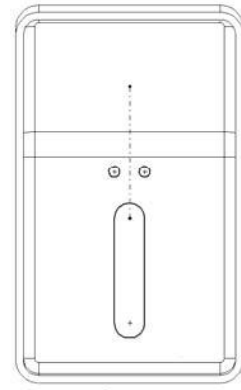
### 62. Result

The second selected face acts as the "Y" direction for the view, rotating the view to put the second face above the first.



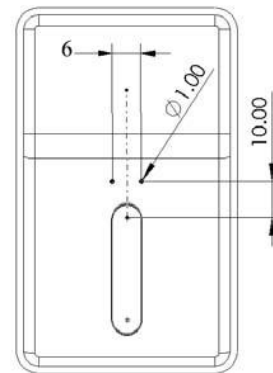
### 63. Sketch-Mirrored Circles

Using a centerline as a mirror axis, create a pair of symmetrical circles. The mirror relation will hold the locations and radii as equal. Note that one centerline endpoint is at the **Button Hole** arc centerpoint. This way if the **Button Hole** moves, the circles will also move.



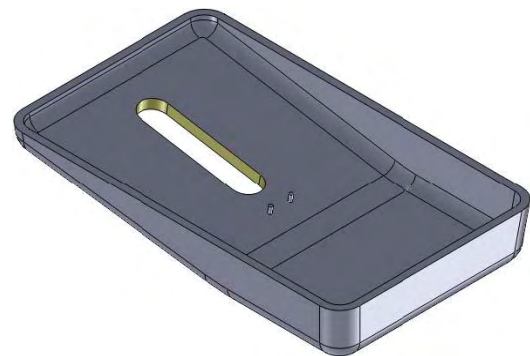
### 64. Dimensions

Add dimensions to the sketch, noting that only one dimension is required to the centerline and one for the diameter.



### 65. Extrude as a Boss

Using **Insert, Boss/Base, Extrude...**, create a **2 mm** boss from the two profiles in the sketch. The two contours are considered as a single feature in the FeatureManager design tree. Name this feature the **Pin Connector**.



# Linear Patterns

SolidWorks software supports **Sketch Driven**, **Table Driven**, **Curve Driven**, **Linear**, **Circular** and **Mirror** patterns. Here we will use a **Sketch-Driven Pattern**.

## Pattern types

There are several types of patterns that are linear in nature. The one you use depends on the nature of the pattern you want to create. A pattern in SolidWorks can be a seemingly random collection of locations.

## Sketch Driven Patterns

The sketch driven pattern uses the positions of points in a sketch as the location of pattern instances. The sketch must precede the pattern.

## Table Driven Patterns

**Table Driven Patterns** enable you to create a pattern of features at a series of X-Y locations. You can enter these locations directly into the spreadsheet-like dialog box, or you can read them from an ASCII text file. The file must have the file extension **\*.sldptab** or **\*.txt**.

If you enter the coordinates directly in to the **Table Driven Pattern** dialog box, you can save the location list as a file for reuse. You have to make a coordinate system before you create the pattern because the coordinate system determines the direction of X and Y.

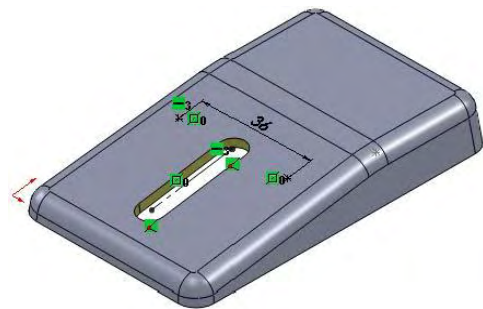
## Linear Patterns

Linear patterns can be generated as a full array of equally spaced instances in a selected direction. One or two directions can be used. Individual instances can be deleted from a pattern after it is created.

### 66. Sketch for the Pattern

Create a sketch for the pattern by placing two points symmetrically about a centerline. Tie the centerline and the points to the position of the arc centerpoint.

Close and rename the sketch to **PatternSketch**.



### 67. Show sketch

Show the sketch of the Button Hole feature. The centerpoint of the arc is needed in the pattern.

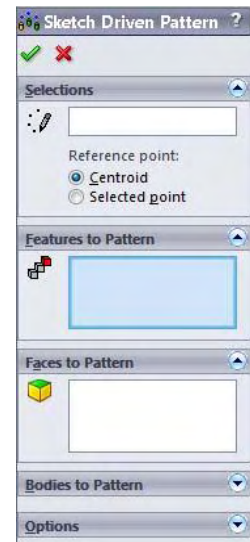




**Insert, Pattern/Mirror, Sketch Driven Pattern** creates instances of selected features positioned by the points in a sketch. The instances retain their associativity with the original feature(s) and will update if the original changes.

### 68. The Sketch Driven Pattern Command

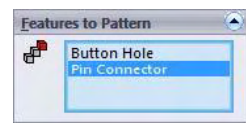
From the **Insert** menu, choose **Pattern/Mirror, Sketch Driven Pattern**. The PropertyManager shown at right opens.



### 69. Select the Features

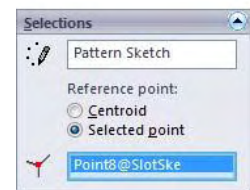
One or more features can be patterned in a single command. Select the **Button Hole** and **Pin Connector** features from the FeatureManager design tree (click the title of the PropertyManager to show the FeatureManager).

The **Features to Pattern** list should indicate two features.



### 70. Reference Sketch

Click in the **Reference Sketch** field and select the sketch **PatternSketch** from the FeatureManager design tree.



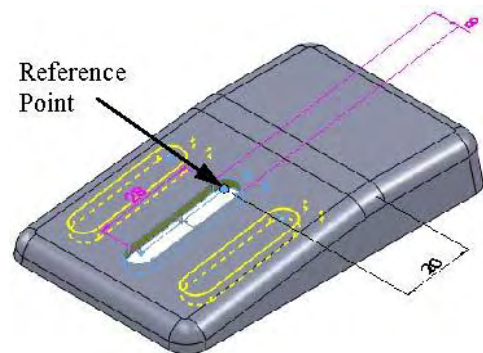
### 71. Reference Point

Click **Selected point**. Then, click in the **Reference Point** field and select the upper arc centerpoint of the **Button Hole** sketch.

### 72. Preview

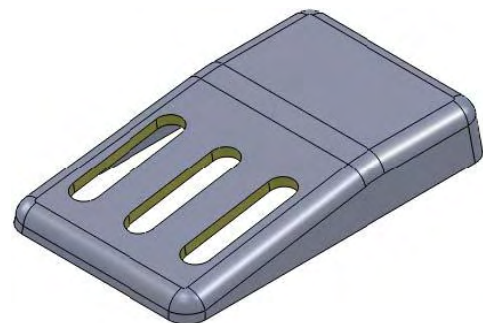
The preview shows the locations of the two instances.

Click **OK**.



### 73. Completed Pattern

After using **Hide Sketch** on the Button Hole sketch and the PatternSketch, the pattern looks like this. The pattern feature is listed as Sketch Pattern1 in the FeatureManager design tree.



### 74. Save the Part

You need to save your work because the next part of this lesson picks up where this one leaves off and completes the modeling of the mouse cover.



## Thin-Walled Parts: Part Two

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

- Employ various techniques for making ribs.
- Mirror features.
- Copy and paste a sketch.
- Use derived sketches.
- Create thin features by extruding.
- Use the Hole Wizard.
- Create section views of the model.
- Edit a sketch plane.
- Create a cut with text.

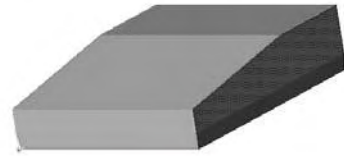


# Review

In the previous lesson, the **Mouse Cover** part was created to the point at which the **Button Holes** were copied using a linear pattern. Here is a review of the major steps you have completed so far:

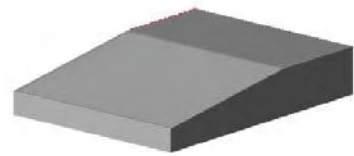
- **Base Part Created**

The base part was created as a solid. Draft was applied using parting lines.



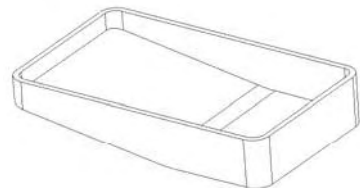
- **Inserted Base Part**

A new part was created, and the base part was inserted as the first feature. The lower portion of the body was removed with a cut.



- **Shelling**

Multi-thickness shelling was used to hollow out the body. A larger thickness value was used on the upper faces and the bottom face was removed completely. The edges were filleted.



- **Library Features**

Using a library feature, a cut was inserted into the model. The sketch was edited to align and size it.



- **Patterning**

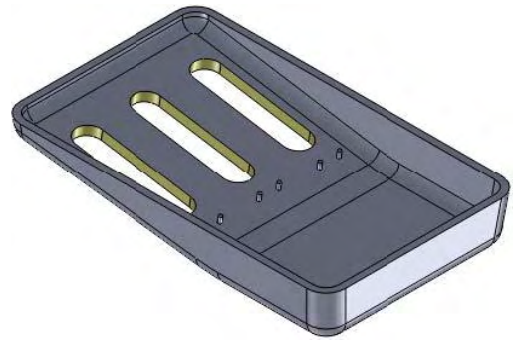
The library feature and an associated boss were copied using a Sketch Driven Pattern.



This lesson continues the work on the part, adding the internal ribs, bosses, and other features that will complete it.

## Creating a Rib

The cross rib braces the shell from left to right. This rib has draft, is symmetrical, and has a rounded top. In a situation in which the rib needs to merge into two walls, the profile should be sketched somewhere in between the walls. The **Long Center** plane will be used to sketch the rib between the faces to which it connects.



### Locating Plane

Create a new plane that will be used to locate the center of the rib.

#### 1. Reopen the Part Mouse\_Cover.SLDPRT

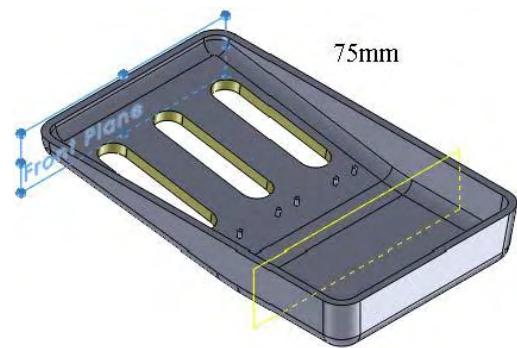
This is the part that we were working on in the previous chapter: Thin-Walled Parts: Part One..

#### 2. Create an Offset Plane

Use **Insert, Reference Geometry, Plane...** to create a new plane.

**Offset Distance** 

**75 mm** from the **Front** reference plane.  
Name the plane **CrRib**.

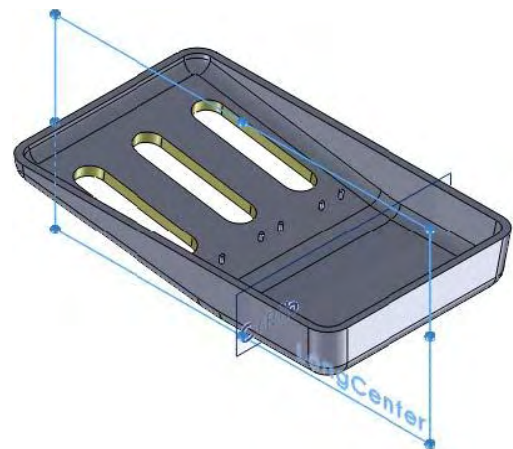


### Rib Geometry

Create the rib as a symmetrically sketched profile. This profile will include draft and mirroring.

#### 3. Select the Sketch Plane

Select the **Long Center** plane and open a sketch. Because this plane runs down the center of the part and stays centered, even if the dimensions of the part change, using it ensures that the rib sketch will always lie inside the walls of the part.



#### 4. Change View Orientation

Change the orientation of the view so you can see the plane's true size and shape with the bottom of the part facing up. Change the Display Style to Hidden Lines Visible.

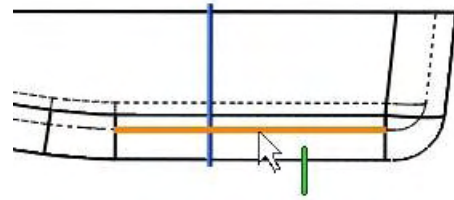


## 5. Copy an Edge

Select an edge of the inner wall thickness, visible through the outer wall. This edge is the physical inside face. Use **Convert Entities**



to create the base of the rib. If you have trouble selecting the edge, you can also try selecting it using section view.



## 6. Rib Geometry

Using a vertical centerline and mirroring, sketch an angled line. Connect it to its mirrored copy with a tangent arc.

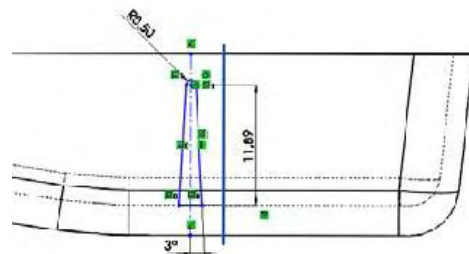


## 7. Trim and Add Dimensions

Add dimensions for the draft angle, arc radius, and height. When you dimension the height, be sure to select on the circumference of the arc, not the center or the endpoints.



**Trim** the two ends of the converted line.



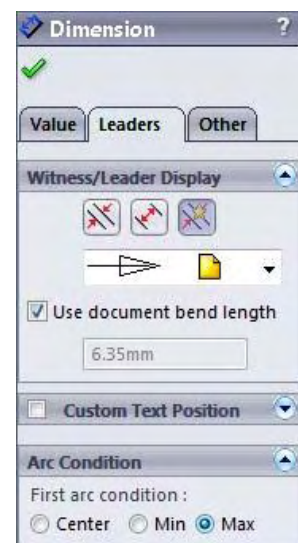
## Arc Conditions

When you place a linear dimension on an arc or a circle, you should select the circumference, not the centerpoint. By default, the dimension will reference the center, but you can change the reference point. If you select the center, you cannot change the reference point later.

Change the measurement point on an arc or a circle by editing the properties of the dimension. The dimension properties will allow the dimension to measure to the center, minimum, or maximum position. For the rib, changing the arc condition to maximum measures to the top of the arc.

## 8. Change Arc Conditions

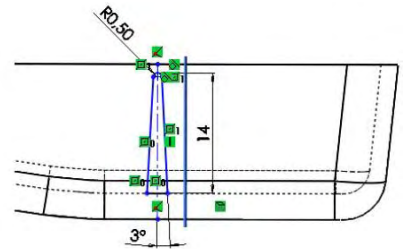
Select the linear dimension, switch to the **PropertyManager** and select the **Leaders** tab. Change the **First Arc Condition** from **Center** to **Max**. In the **Value** tab, change the **Value** to **14 mm** and **Arrows** to **Inside**. Press **OK**.





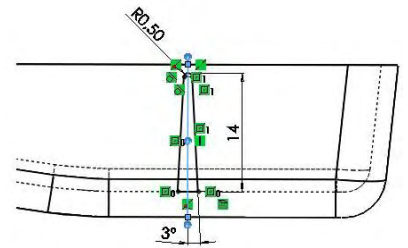
## 9. Maximum Condition Measurement

The dimension now measures to the true top of the rib.



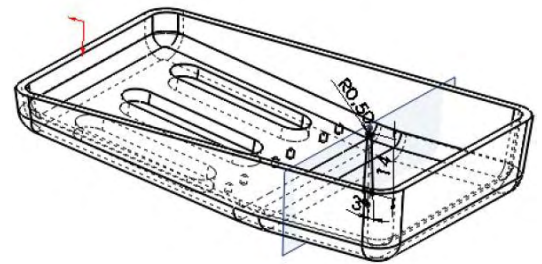
## 10. Relation to a Plane

Relate the centerline to the **CrRib plane** with a **Collinear** relationship. The sketch is now fully defined.



## 11. Completed Sketch

The fully defined rib sketch sits between the two walls of the cover.

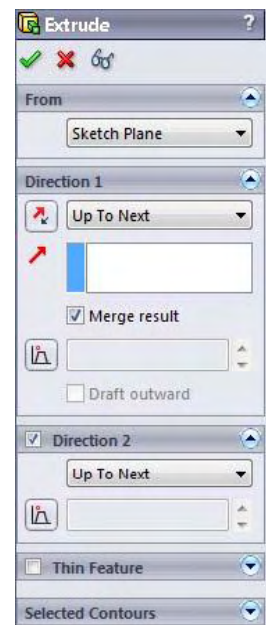


## Rib End Conditions

Ribs or any other features that need to merge into other faces (the walls) must use the **Up To Next** end condition in both directions. This will ensure that the rib connects to both walls and all adjacent faces.

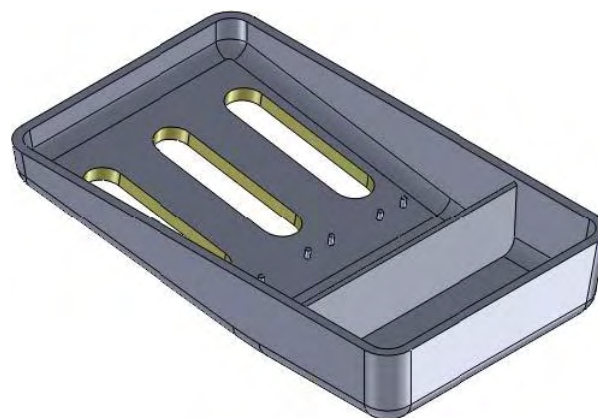
## 12. Access Insert Boss

Click **Insert, Boss/Base, Extrude...** and choose the **Up To Next** end condition under **Direction1**. Select **Direction2** by clicking its check box. Set the **Direction2** end condition to **Up To Next** as well.



## 13. Completed Extrusion

Press **OK** to create the rib. It merges into both walls, the fillets, and the bottom face, creating a single solid. Name this feature **Cross Rib**. Switch the Display Style back to **Shade With Edges**.

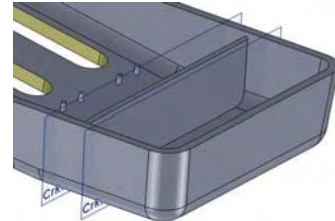


## Copying a Sketch

To create another rib 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 for the previous feature **Cross Rib**, will be copied onto the plane **CrRib\_1** and edited. This rib will run normal to the first from the sketch plane to the end wall.

### 14. Create Offset Planes

Use **Insert, Reference Geometry Plane** to create two new planes **Offset Distance**  $\pm 5$  mm from the **CrRib** reference plane. Name the planes **CrRib\_1** and **CrRib\_2**.




### 15. Select the Cross Rib Sketch

Expand the feature Cross Rib, and select the sketch. The sketch geometry will highlight on the screen.




### 16. Copy

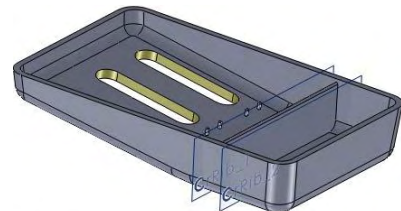
Using **Ctrl+C**, or **Edit, Copy**, or the **Copy** icon

 from the main toolbar, copy the sketch to the clipboard.

### 17. Select Plane and Paste

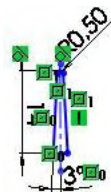
Select the plane **CrRib\_1** from the FeatureManager design tree and click **Ctrl+V**, or **Edit, Paste**, or the **Paste** icon  from the Main toolbar.

The sketch will be pasted from the clipboard to the selected plane. It will appear on the screen in the plane's orientation.



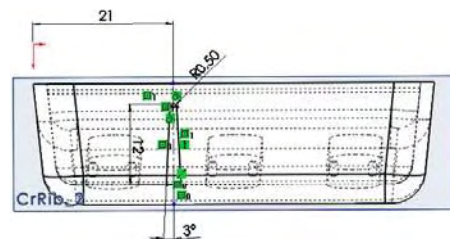
### 18. Edit the Sketch

Select the new sketch and **Edit Sketch**. Orient the view to look at the part from the back, with the bottom facing up. The geometry was pasted somewhere outside the model and is sized improperly. You will need to add relations and dimensions to fully define the sketch.



### 19. Relations and Dimensions

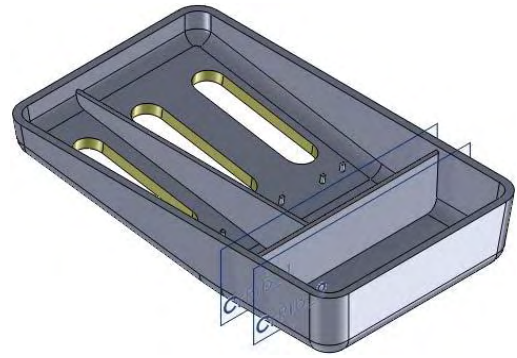
Add a **Collinear** relation between the horizontal line in the sketch and the model edge at the bottom of the Cross Rib. Add a **21 mm** dimension between the centerline and the Right reference plane. Change the height to **12 mm**. The ends of the centerline can be left under-defined or you can tie them to model edges.





## 20. Create a Boss

Using the **Up To Next** option in both directions 1 and 2, extrude a new boss feature. Name the feature **Length Rib**.



## Mirroring Features

Mirror patterns are used to create copies of selected features using a mirror plane. The copy is the same distance yet opposite hand across the mirror plane. The rib just created will be mirrored.

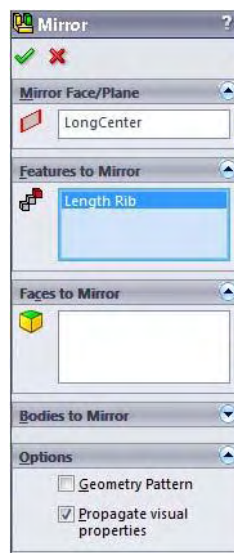
**Mirror Feature** copies features across a plane or planar face. The copies retain their associativity to the original and will update if the original changes. To execute this command, from the **Insert** menu, choose **Pattern/Mirror, Mirror...**

## 21. Mirror Dialog

Click **Insert, Pattern/Mirror, Mirror...** from the menu.

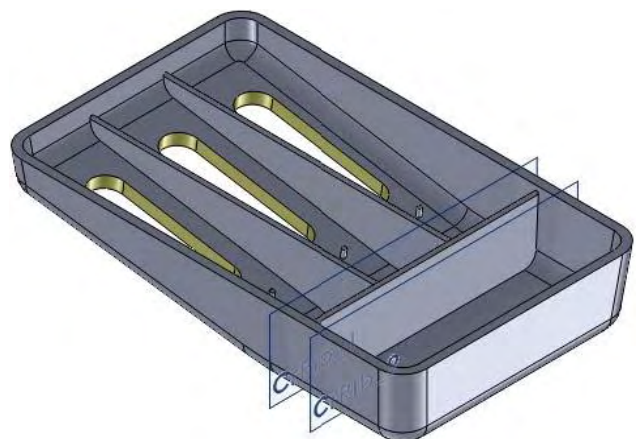
## 22. Select the Feature and Mirror Plane

The selection list for **Mirror Face/Plane** is active by default. Select the plane **Long Center**. Its name will appear in the selection list. Click in the **Features to Mirror** list to activate it. Select the **Length Rib** feature. This is the feature that will be mirrored. Notice the preview of the mirrored rib. Press **OK**.



## 23. Completed Copy

The copy retains the shape of the original. Changes to the original will force changes in the copy.



# Derived Sketches

A **Derived Sketch** is used to create a copy of the **Length Rib** feature on a different plane and location. The derived sketch will be tied to 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.

## Creating a Derived Sketch

Create the derived sketch on the plane **CrRib\_2**. Once copied, the sketch can be rotated and repositioned if it is at the wrong orientation.

### 24. Sketch and Plane

Select the *sketch* of the **Length Rib** feature and **Ctrl**-select the plane on which you want it copied (**CrRib\_2**). The sketch will be copied to the selected plane in the next step.

### 25. Insert 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.

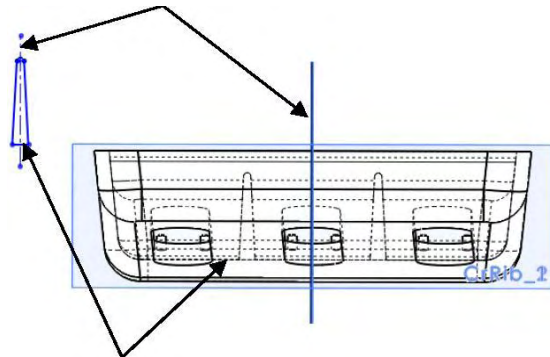
## Locating the Derived Sketch

Like library features, **Derived Sketches** are inserted under-constrained. In this example, the rib is tied to an edge and a plane.

### 26. Show the Center Plane

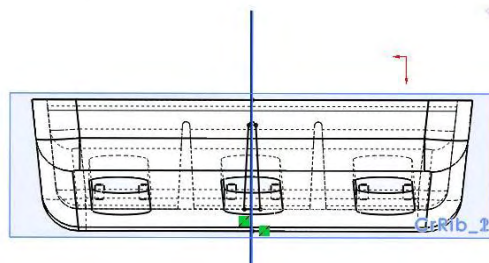
Show the plane **Long Center**.

We are going to use this plane to relate to the sketch centerline.



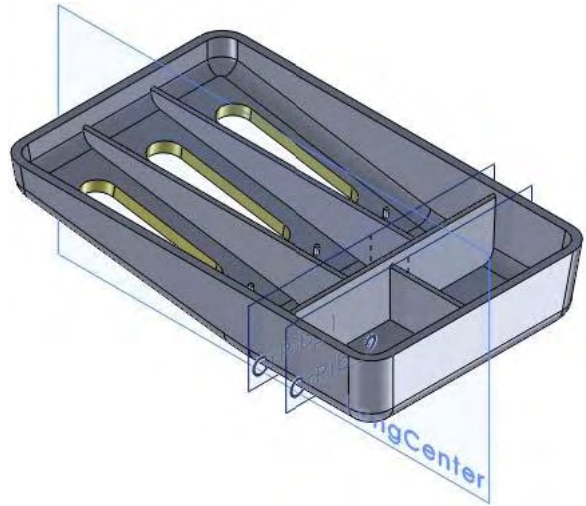
### 27. Fully Defined

Add a **Collinear** relation between the centerline and the plane. Add another **Collinear** relation between the base and the shell bottom.



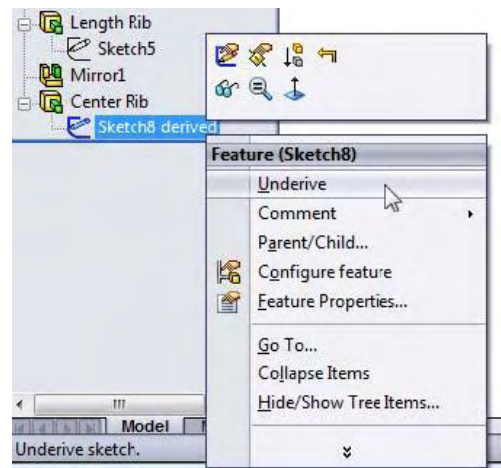
## 28. Complete the Rib

Extrude the sketch **Up To Next** towards the rear wall and the **Cross Rib**. The rib created is a copy of the Length Rib section that is centered in the shell. Name this feature **Center Rib**.



## What if I Change My Mind?

You can break the link between a derived sketch and the original by selecting **Underive** from the right mouse-button pop-up menu. Once this is done, changes to the original will no longer propagate to the copy. However, once you underive it, there is no way back. You cannot re-establish the link once it's removed. Also, notice that derived sketches are identified as such by the **derived** suffix appended to their names in the FeatureManager design tree.

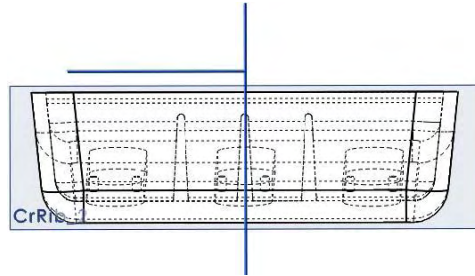


# Tapered Boss

We will create a tapered boss at the intersection of the **Cross Rib** and the **Center Rib**. The boss is sketched above the model and extruded down with draft to the bottom face of the shell.

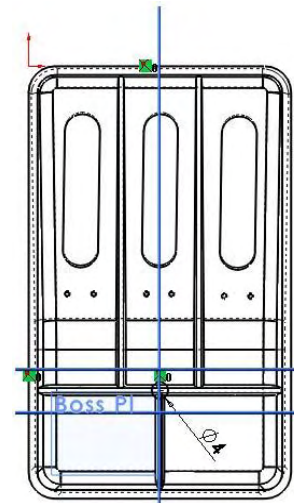
## 29. Plane for Sketch

Create a plane offset **18 mm** from the inner face. This plane will be used to sketch the boss profile. Name it **Boss PI**. Open a new sketch.

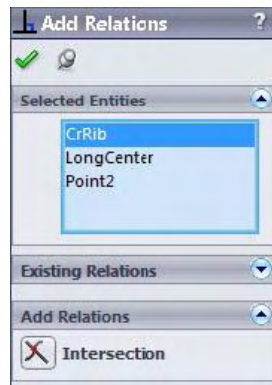


## 30. Relations and Dimension

The circle is sketched, and its center is related with an **Intersection** relation between the circle centerpoint and the intersecting planes **CrRib** and **Long Center**.



If you get a message saying your sketch is over-defined, you have to delete the last geometric relation you made.



After the diameter dimension is added, the sketch is fully defined.

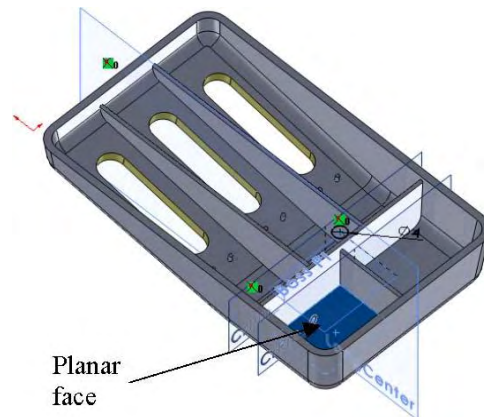
## Up to Surface End Condition

Once again we could use the end condition **Up To Next**. However, this is also a good opportunity to show the **Up To Surface** end condition. **Up To Surface** allows you to select a face to terminate the extrusion.

## 31. Extruding Up To Surface

Click **Insert**, **Boss** or the  icon and choose the **Up To Surface** end condition. Click in the **Selected Items** box and select the planar face indicated at the right. Set the face **Draft** to **3°** and click **Draft Outward**.

icon and choose the **Up To Surface** end condition. Click in the **Selected Items** box and select the planar face indicated at the right. Set the face **Draft** to **3°** and click **Draft Outward**.



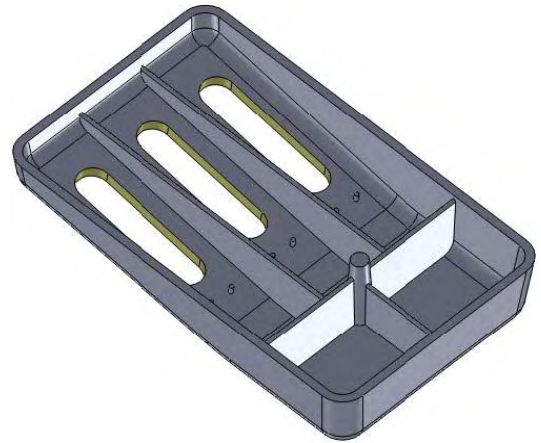
**Important:** Verify the preview because by default, bosses extrude away from the part. **Reverse Direction** if needed and press **OK**.





### 32. Completed Boss

The boss extends down to the bottom face of the shell. Rename the feature **Tapered Boss**. You may hide the visible planes at this point (toggle **View Planes**).

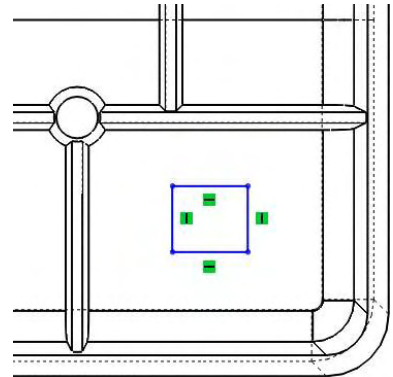


## Thin Features


**Thin Features** are made by extruding or revolving an *open* sketch profile and applying a wall thickness. The thickness can be applied to the inside or outside of the sketch, or equally on both sides of the sketch.

### 33. Rectangle

Create a sketch and rectangle on the inner face of the model. This example requires an “L” shape rather than a rectangle.



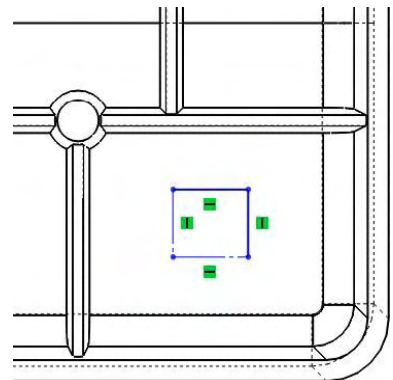
## Construction Geometry

**Construction Geometry** converts sketch geometry from standard to construction and *vice versa*. Note that the PropertyManager **For Construction** checkbox performs the same task. You can find it on the **Context Toolbar** using the **Construction Geometry** button 

### 34. Construction

Select the lower and left lines. Click the **Construction Geometry** tool to change them to centerlines.


**Note:** This procedure can be performed on any sketch geometry type.



## Ordinate Dimensions

Ordinate-type dimensions can be created in the sketch as the driving dimensions, replacing the standard linear, radial, diametric, and angular dimensions. The ordinate type uses a datum position of “zero” and measures locations from that position. There is no signage for relative direction from the zero position. All positions are listed as positive.

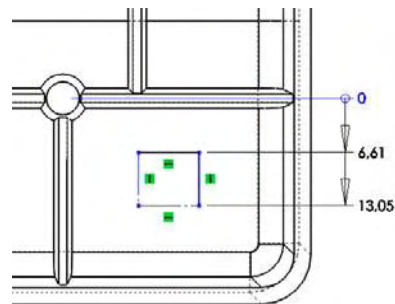
### Where to Find It

- Click the arrow below the **Smart Dimension**  tool from the Sketch tab.  
Choose **Ordinate Dimension**, **Horizontal Ordinate Dimension**, or **Vertical Ordinate Dimension**.
- When the **Smart Dimension** tool is active, right-click and choose **Ordinate**, **Horizontal Ordinate**, or **Vertical Ordinate** from **More dimensions**.

#### 35. Vertical Ordinate

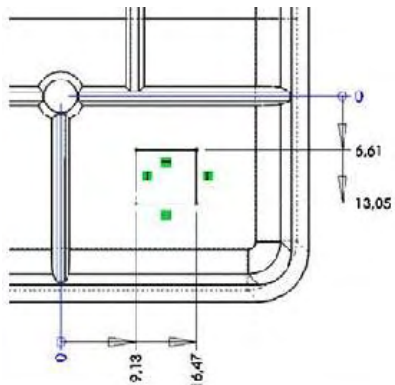
Click **Vertical Ordinate** and select the edge of the Tapered Boss as the datum position. Click a location to the right of the model to place the dimension.

Select the two horizontal lines to create dimensions from the datum. Ordinate dimensions are automatically positioned aligned with the datum dimension.



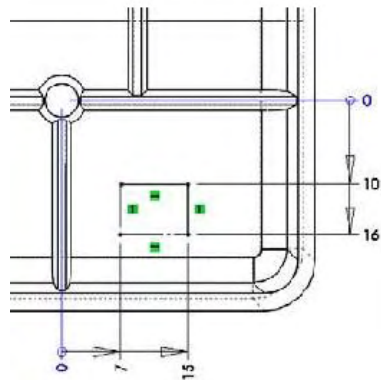
#### 36. Horizontal Ordinate

Click **Horizontal Ordinate** and again select the edge of the Tapered Boss as the datum position. Click locations on the two vertical lines.



#### 37. Final Size

Set the dimensions to the values shown. Note that the datum (0) dimensions are *driven* dimensions.



### 38. Extrude the Boss

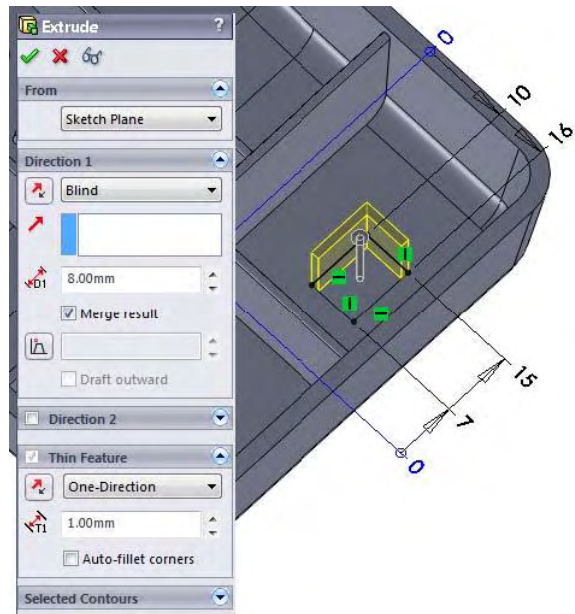
Extrude as a blind boss with a depth of **8 mm**. Under **Thin Feature** set the **Thickness** to **1 mm**.

### 39. Offset Graphics

The thin feature offset is previewed when the wall thickness value is given.

Whether it defaults to inside or outside depends on the order in which you sketched the lines.

If need be, use **Reverse Direction** to put the offset on the inside. Press **OK** to create the thin feature.



### 40. Completed Thin Feature

The thin feature is created as an extrusion.



# Drafting Faces


Individual faces can be drafted with respect to a plane or planar face of the model using **Insert Draft**. In this example, draft is added after the boss is extruded because only two of the faces require draft.



## 41. Open the Insert Draft PropertyManager

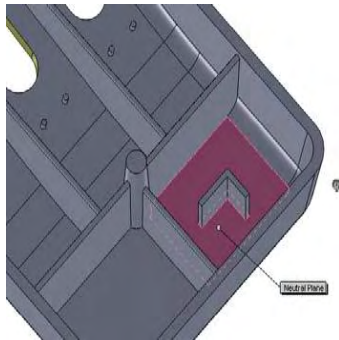
⇒Tip:

Open the **Insert Draft** dialog before you select anything. Then you can click in the proper selection list and pick the required geometry.

Pick the icon  from the feature tab, or pick **Insert, Features, Draft**. Set the **Draft Angle** to **3°**.

## 42. Neutral Plane

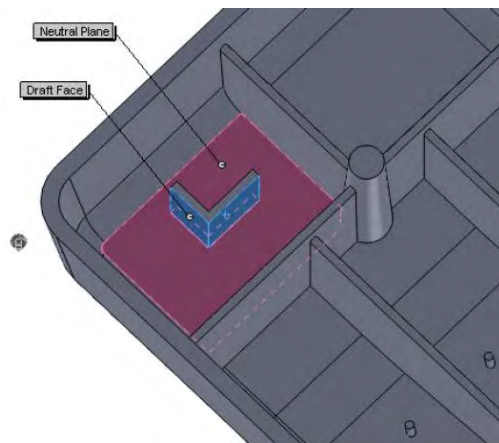
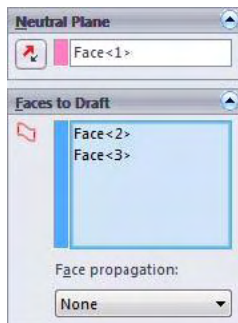
Click in the **Neutral Plane** selection list and pick the face indicated in the illustration below. Verify that the pull direction arrow is pointing up. This is what determines whether the draft angle goes in or out.



## 43. Faces to Draft

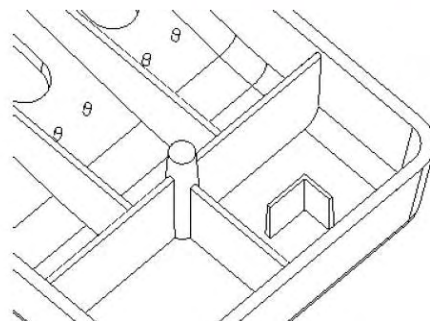
Click in the **Faces to Draft** list and select the two outer faces of the thin feature as shown.

Remember you can use **Select Other** as a quick way of selecting hidden or obscured faces.



## 44. Completed Draft

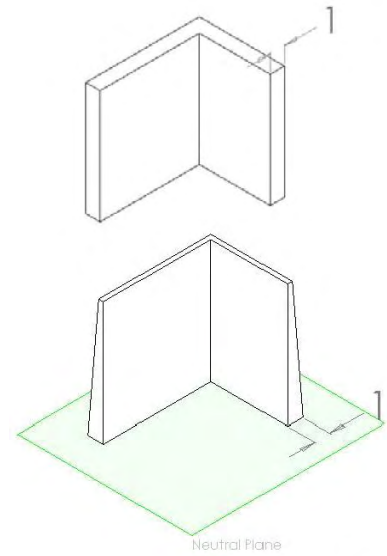
The two selected faces are drafted, after clicking **OK**.



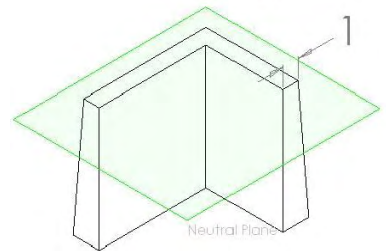
## More about the Neutral Plane

The choice of neutral plane determines more than just the pull direction for the draft. It also controls how the faces are "rotated" when draft is applied to them. Consider this example of a thin feature extruded with a wall thickness of 1 mm.

If the neutral plane is selected as shown, the drafted faces will rotate about their bottom edges. This maintains the 1 mm dimension at the bottom while the walls get thinner at the top.



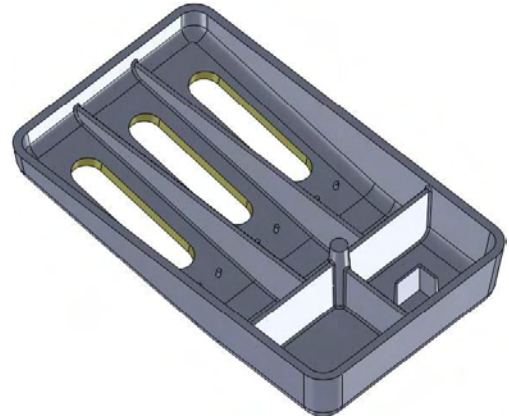
If the top face of the feature is used as the neutral plane, the drafted faces are rotated about their upper edges. This maintains the 1 mm dimension at the top while the walls get thicker at the bottom.



### 45. Add Fillets

Using a **0.5 mm** fillet, fillet everything inside the cover *except*:

- Top edges of the thin feature,
- Inside edges of the thin feature,
- Top edge of the circular boss,
- Edges of the Button Holes,
- The Connecting Pins.



## Using the Hole Wizard

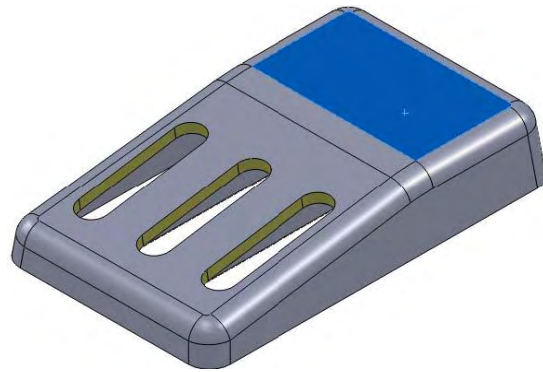
The **Hole Wizard** is used to create specialized holes in a solid. It can create simple, tapered, counter-bored, and countersunk holes using a step-by-step procedure. In this example, the **Hole Wizard** will be used to create a countersunk hole.

### Creating a Countersunk Hole

You can choose the face into which to insert the hole and then define the dimensions of the hole using the **Hole Wizard**. During the process you can also position the location of the hole on the face.

#### 46. Select a Face for the Hole

The hole will enter the solid through the selected face. Selecting the face is similar to selecting a sketch plane.



### The Hole Wizard

The **Hole Wizard** creates shaped holes, such as countersunk and counter-bored types. The process creates two sketches: one defines the shape of the hole; the other, a point, locates the center. To do this:

1. From the menu choose: **Insert, Features, Hole, Wizard...** or,



2. Choose the **Hole Wizard** tool from the Features tab.

#### 47. Start the Hole Wizard

From the **Insert** menu, choose **Features, Hole, Wizard...** or pick the **Hole Wizard** tool on the Features tab.



## 48. Hole Types




Choose the **Countersink**  Hole Type.

## 49. End Condition

Choose the **End Condition** type **Through All**. Check also if the **Standard** is set to **Ansi Metric**.

## 50. Hole Dimensions

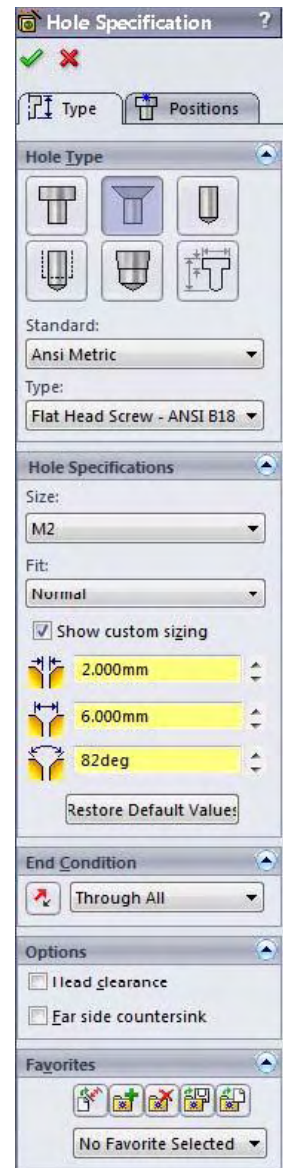
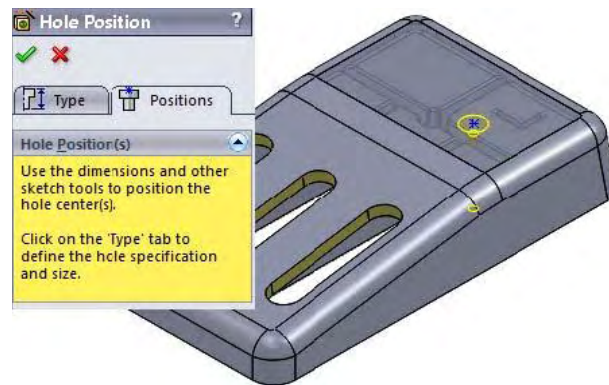
Click in the **Custom Sizing** field to set the dimensions of the hole as follows:

- Diameter  = 2 mm,
- C-Sink Diameter  6 mm,
- C-Sink Angle  = 82°.

Press the **Positions** tab to move on to locating the feature.

## 51. Hole Placement

A message in the PropertyManager alerts you that you are now in **Edit Sketch** mode. In this mode you should add relations and dimensions to fully define the location of the point.

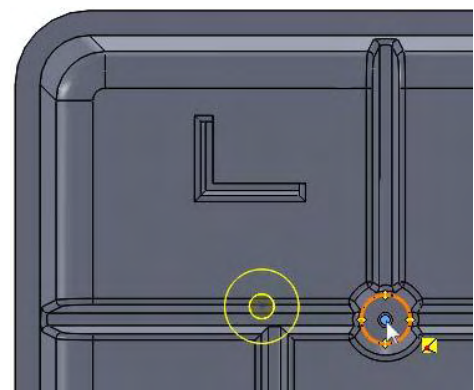


The Point tool is automatically turned on in the event you want to add more points for additional holes. Turn off the Point tool before locating the existing point.

## 52. Locate Point

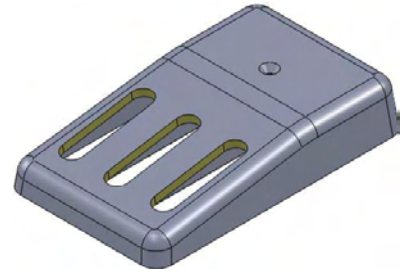
Drag the centerpoint of the Hole to the centerpoint of the **Tapered Boss**, which appears automatically while dragging. If it doesn't automatically appear, go to the tree, select it, and select the midpoint of the Tapered Boss together with the point with Ctrl. Then, coincident them.

Click **OK**.




### 53. Countersunk Hole

A countersunk hole with the specified dimensions is created on the face, centered on the **Tapered Boss**.



## View Sectioning


Section the view to see the result of the hole operation. We will use only view display tools. The model is *not* cut. **Display Section View** cuts the view using one or more sectioning planes. To do so:

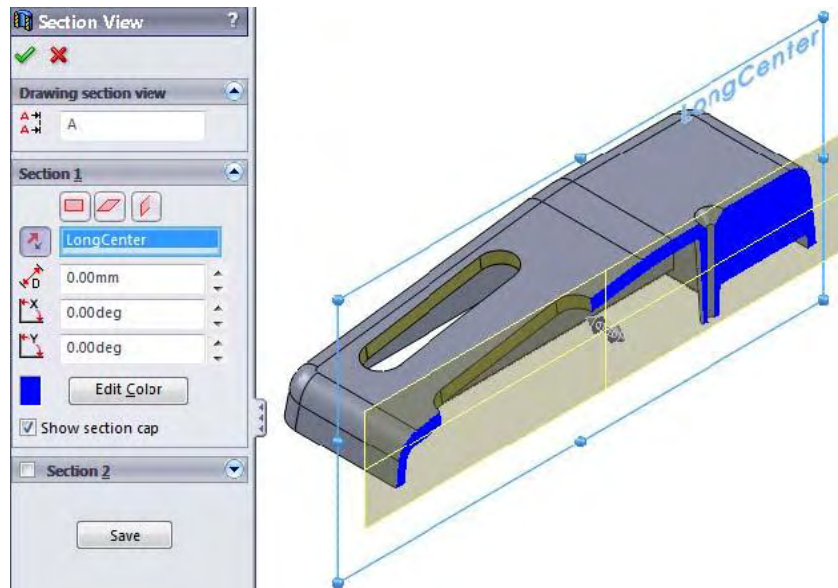
1. From the menu choose: **View, Display, Section View...** or,
2. Click the  icon on the View toolbar.

### 54. Section Plane

Click on the plane **Long Center** as the plane with which to cut.

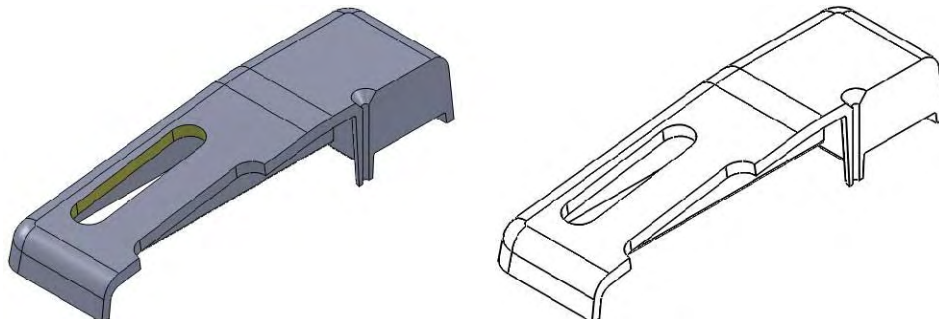
Click  or **View, Display, Section View** to access the **Property Manager**.

Press **Reverse Section Direction**  if necessary. Press the **OK** button to accept the result.



### 55. Section View

The view is displayed as a section. Return to the unsectioned view by selecting the icon a second time, turning it off. You can display any type of view as a section – hidden line, shaded, or wireframe.



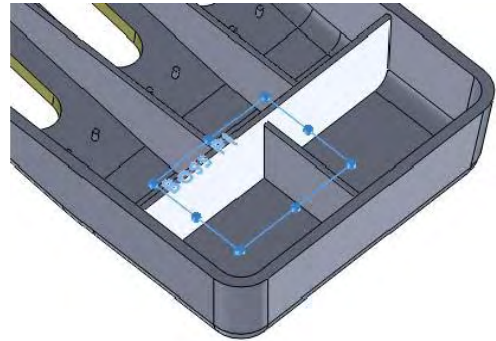


## Editing a Sketch Plane

You can change the plane a sketch was created on by using the **Edit Sketch Plane** option. The feature is rebuilt as if the sketch was created on that plane in the first place. In this example, the tapered boss sketch is edited to be flush with the bottom face of the shell.

### 56. Select Sketch and Edit

Click on the sketch of the **Tapered Boss**. Choose the option **Edit Sketch Plane** from the right mouse-button menu.



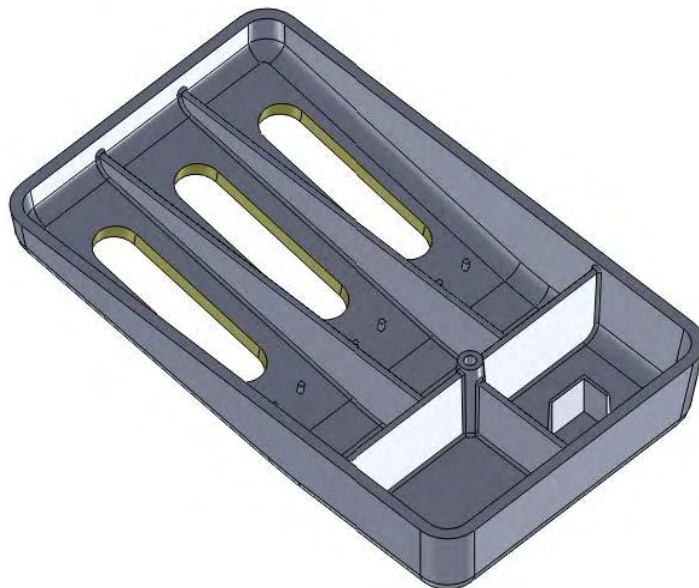
### 57. Choose the Face that is the Edge of the Shell

Select the face and press the **OK** button. The model will be rebuilt as if the sketch had been created on the selected face in the first place.



### 58. Result

The top of the **Tapered Boss** is now flush with the wall.



## Using the Rib Tool

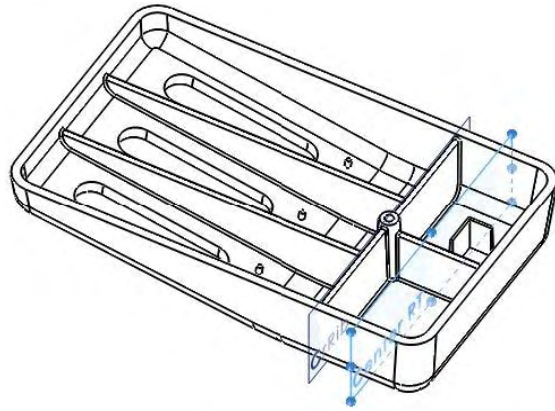
The rib tool — **Insert, Features, Rib** — allows you to create ribs using only a sketched line. The tool prompts you for thickness, placement, and direction of the rib material.

### Rib Sketch

A simple sketched line forms the rib centerline, which is all you need to base the rib. A reference plane is useful for placement of the rib centerline.

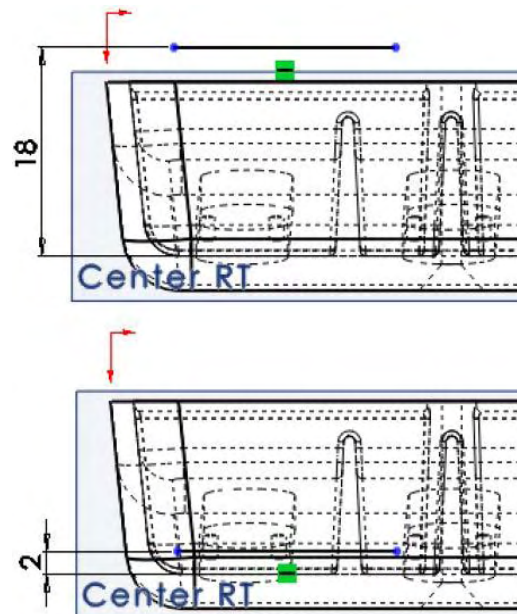
#### 59. Create a New Plane

Create a new plane, offset from the plane **CrRib** by **10 mm**. Name it **Center RT**.




#### 60. Sketch Line

**Open** a new sketch in **Center RT** and sketch a horizontal line above the model as shown. This avoids accidentally capturing an unwanted reference in the model that would prevent you from moving the line later. This line will represent the centerline and top of the rib. The line does not have to reach the wall or rib. Change the value of the dimension to **2 mm** to place it properly. The ends of the line should fall inside the area where you want to place the rib, although it does not have to meet the inner walls exactly.



### Insert Rib




**Insert Rib** creates a flat-topped rib either with or without draft. The rib is based on a sketched contour line that defines the path of the rib. To insert a rib:

1. Choose from the pull-down menu: **Insert, Features, Rib...** or,
2. Pick the  icon from the Features toolbar.

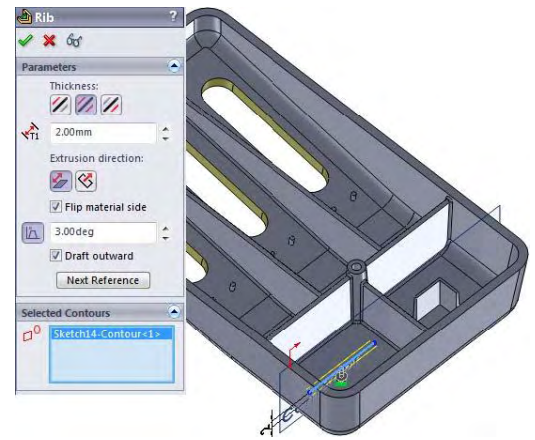


## 61. Rib Tool

Click the **Rib** tool on the Features tab and set the parameters as:

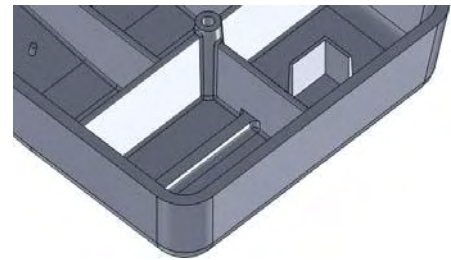
- Thickness: 2 mm**  
**Create rib on both sides of sketch** 
- Extrusion direction: Parallel to Sketch** 
- Draft** : **3° Outward**

Look at the preview arrow, which indicates the direction the rib will be extruded. If necessary, select **Flip material side** to reverse the direction.

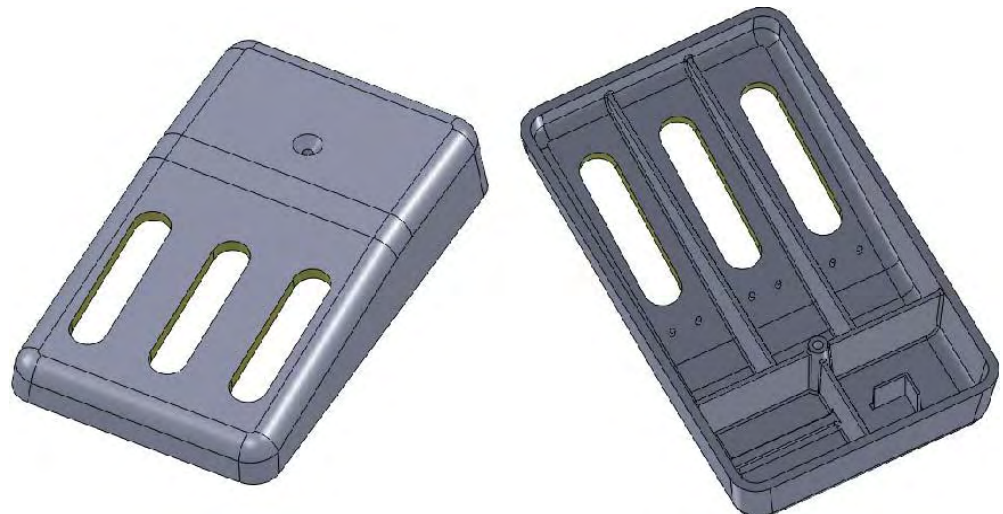


## 62. Add Fillets to Complete the Rib

The finished rib is merged to the bottom, the wall, and the existing rib, including the fillets. Add **1-mm** fillets to the bottom edges of the rib to complete the part.



## Result



# Adding Text



Text can be added to a sketch and extruded as a cut or boss with the **Text Tool**. The text can be positioned freely; located using dimensions or geometric relations; or made to follow sketch geometry or model edges.

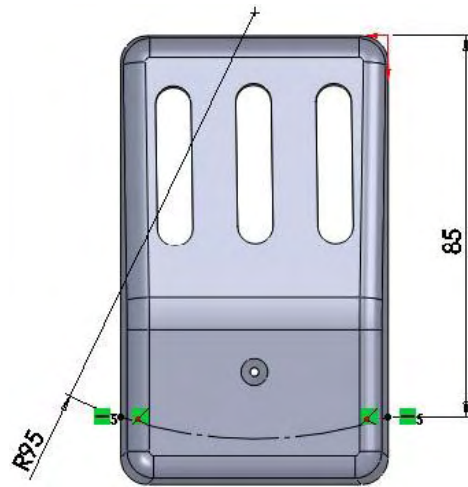
## Text Tool

The text tool allows you to insert text into a sketch and use it to create a boss or cut feature. Because SolidWorks software is a true Windows application, it supports whatever fonts you have installed on your system. To add text, choose from the pull-down menu: **Tools, Sketch Entity, Text**.

### 63. Construction Geometry


Sketch on the top, flat face and add a construction arc as shown:


- Sketch a **Three Point Arc**  on the face.
- Add a **Horizontal** relation between the two endpoints.
- Add a **Coincident** relation between the endpoints of the arc and the long outer edges of the *Mouse\_Cover*.
- Add dimensions as shown.
- Select the arc and change it into construction geometry by checking **For construction** in the **PropertyManager**, or by clicking **Construction Geometry**  from the Context toolbar.




Close the sketch.

### 64. Text on a Curve

Create another sketch on the top face. Click  or **Tools, Sketch Entities, Text...** The Sketch Text PropertyManager appears.

In the **Curves**  list select the arc from the previous sketch.

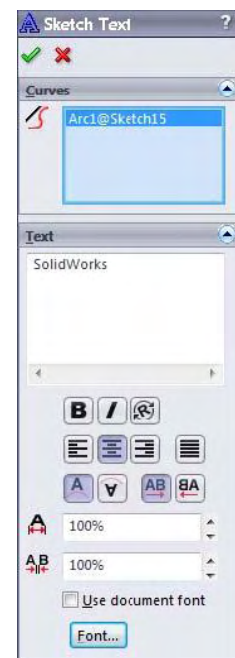
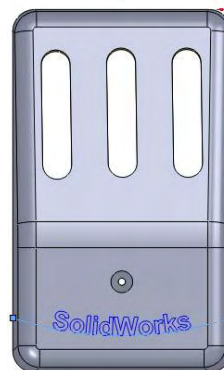
Type the text in the **Text** area. Click **Center Align**  to center the text on the curve (also try the other options and watch the preview for their effects).

You have the option of changing the default font by picking the

 **Font** button.

This example was done using **20 point Arial Bold**.

Click **OK**.



## 65. Extrude a Cut

Extrude the sketched text as a cut feature to a depth of **0.5 mm**.

Hide the sketch we used to position the text.



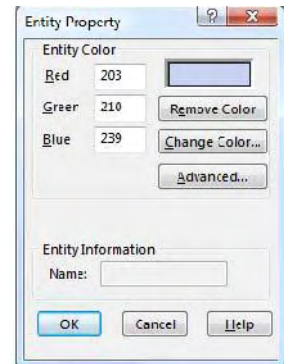
## 66. Edit Properties

Edit the properties of the cut feature to change the **Name**. Also change its color, making the text more visible on the finished part. Click the **Color** button.



## 67. Color Settings

Change the color by picking the **Change Color** button. To reset the feature back to its original color, pick **Remove Color**.



## 68. Color Settings

Pick one of the pre-defined colors in the palette or click **Define Custom Colors** to create your own color. When you are through, click **OK**.



## 69. Click OK a Couple More Times

Click **OK** to apply and close the **Entity Property** and **Feature Properties** dialogs.

## 70. Results



## Searching in the FeatureManager Design Tree

The **Go To** option can be used to find sketches, bosses, cuts, and other features by name in the FeatureManager design tree.

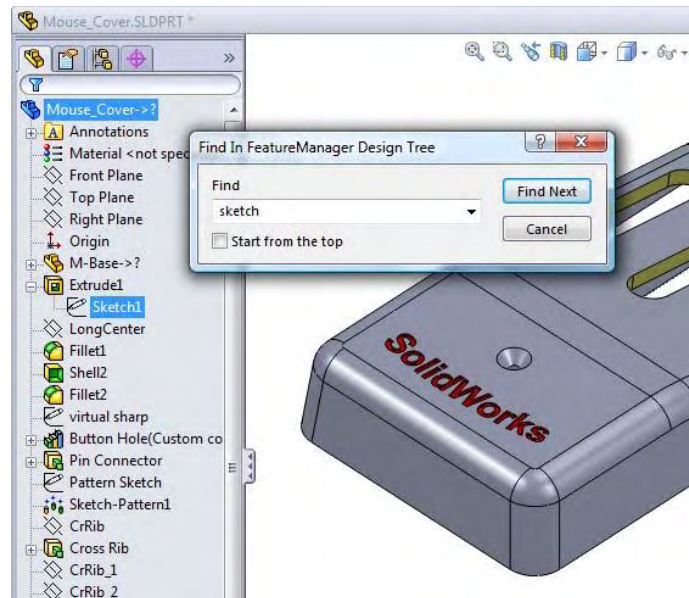
### 71. The Find Dialog

From the top level feature (part name), access the **Go To...** option from the right mouse-button menu. Set the search string for the characters **sketch**. Click the **Find Next** button.



### 72. Search Results

By default the search starts from the top of the FeatureManager design tree. Each click of **Find Next** finds the next occurrence of the text string. If that feature is collapsed, the system expands and highlights it. It will continue searching until it reaches the bottom of the design tree or until you click **Cancel**.



### 73. Save Your Work and Close the File

Congratulations, you have successfully finished the first two lessons of this workbook.

# Modeling Advanced Shapes: Part One

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 non-planar curve by projecting a sketch onto a surface.
- Create a variable radius fillet.
- Create boss and cut features by sweeping.
- Model threads.

## Introduction

This lesson contains a couple of case studies. They 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
- Lofting
- Advanced filleting capabilities



## Case Study: Bottle

Modeling free-form shapes requires some techniques for creating features that are quite unlike the extruded or revolved shapes built earlier. This example will go through the steps for 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 are:

- **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 9.5 mm at the two sides to 6.3 mm at the center of the front and back.
- **Shell the bottle**  
The bottle has two different wall thicknesses. The neck has to be thicker (1.5 mm) because of the threads. The body is thinner (0.5 mm).
- **Model the threads**  
This is another sweeping operation. However, this time a different sort of path is used: a helix.



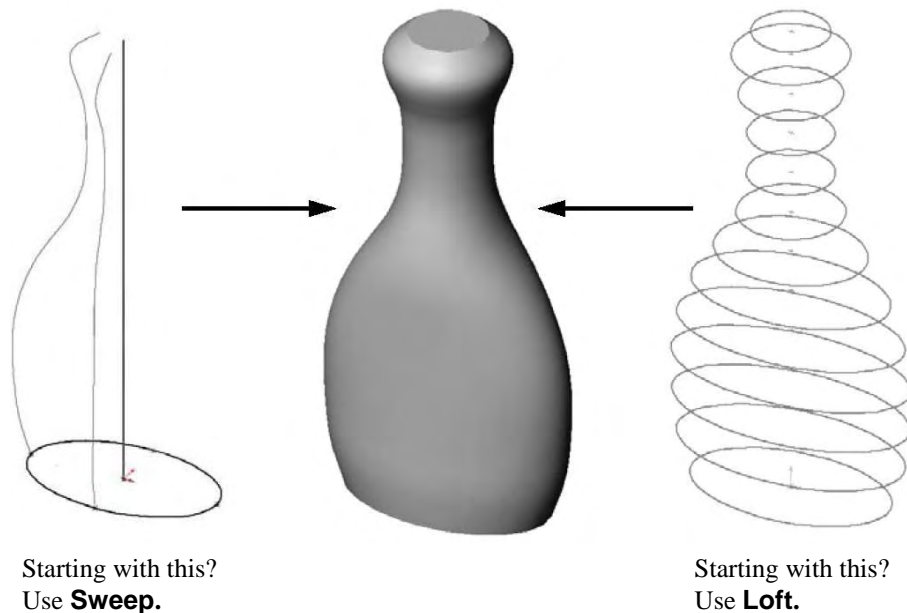
## Sweeping and Lofting: What's the Difference?

Both sweeping and lofting are capable of creating many complex shapes. The method you choose to use to build a particular part depends primarily on what design information you have. There are also some general differences between sweeping and lofting that will influence the approach you utilize. In essence:

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



Consider the base feature of a plastic bottle such as the one shown in this illustration. If the design data you have 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.



## Sweeping

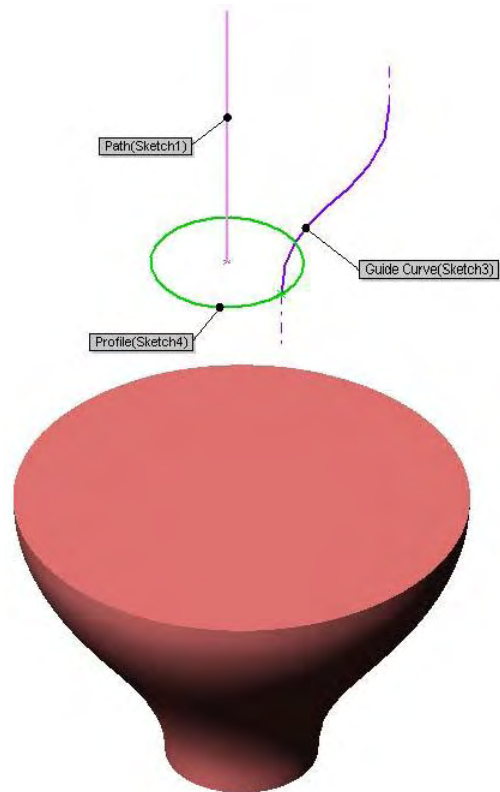
The simplest sweep is a **Boss Extrude** feature, which occurs when a 2D contour sketched on a plane is extruded in the perpendicular direction to that plane. The next simplest sweep is a **Base Sweep**, where the path is a 2D sketch and the sweep section (cross-section) is a simple shape that does not vary along the length of the path. But sweeping can be more complex than presented in this example, offering much more control to the designer. A more sophisticated sweep feature can also incorporate 3D 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, as shown in the above figure on the left.



## Sweep Components

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

- **Sweep Section**  
This is the profile sketch. Sweeping only supports a single profile. It must be a closed, non-self-intersecting boundary.
- **Guide Curves**  
Sweeps can contain multiple guides, which are used to shape the solid. The guide curves must be related to the profile with the **Pierce** relation. 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 guide curve is attached to the profile. As the profile is swept along the path, the radius of the circle changes and follows the shape of the guide.
- **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 Control** 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 Free Points** enables you to create a 3D 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 Points**.

□ Or, click on  the Curves toolbar.

### Entering points "On the Fly"


If you have not 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 the following procedure:

**Note:** This process is not actually part of the case study. Entering the points directly into the **Curve File** dialog is quite time consuming. This information is included here so you will know how the process works. To proceed with the case study, skip to **Reading Data From a File**.

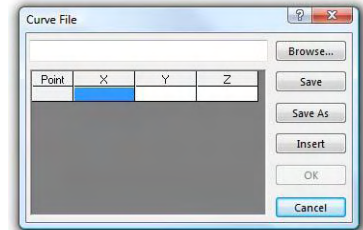
## Procedure

Begin by opening a new part with the units set to mm.

### 1. Insert Curve

Click  or **Insert, Curve, Curve Through Points....** The dialog box **Curve File** will appear. This dialog gives you several options:

1. Browse for an existing file and insert the curve using it "as is".
2. Modify an existing file before inserting the curve.
3. Insert XYZ coordinates "on the fly" with the option of saving them into a file.



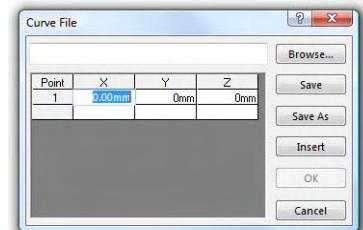
**Note:** The curve is created *outside* of a sketch. Therefore, the X, Y, and Z coordinates are interpreted with respect to the coordinate system of the **Front** reference plane.

### 2. Data for First Point

Double-click in the upper-left cell (top row, under the heading **Point**).

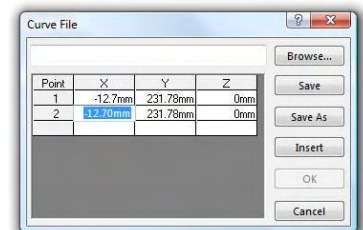
The system will open a row for the first coordinate point using the default values of X=0.0, Y=0.0, and Z=0.0.

Type 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.



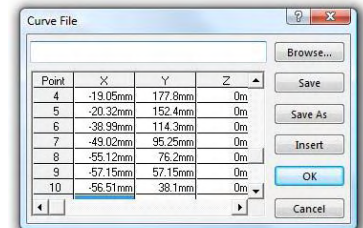
### 3. Adding Another Point

Double-click in the cell *below* Point #1. The system will add a second row using the same values as the preceding one. This is handy when one or more of the coordinates stay fixed from one point to the next. Watch the graphics window for a preview of the curve as you build it (zoom if necessary).



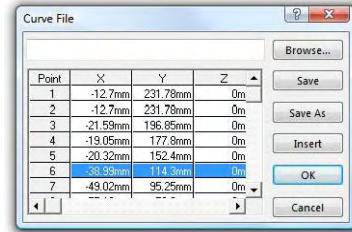
### 4. Repeat as Needed

Add the coordinates for the remaining locations.



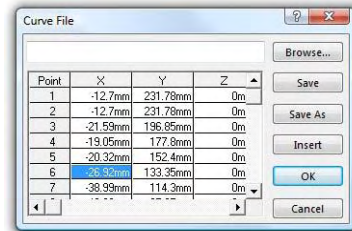
## 5. Inserting a Row

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. In this case, we want to add a new location *before* Point # 6.



## 6. Click Insert

When you press the **Insert** button, the system creates a copy of the selected row, moving the rest of the rows down one position. Edit these values to the correct coordinates.



## 7. Finish the Procedure

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.

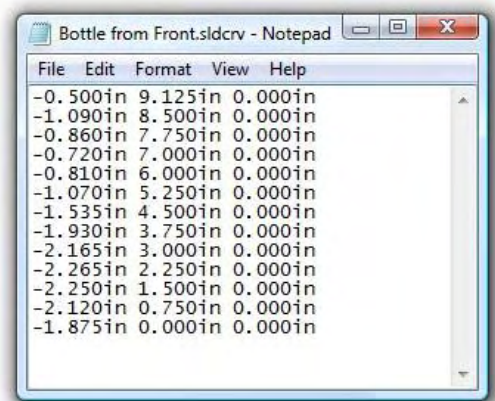
Whether you choose to save the file or not, you click the **OK** button to create the curve. We will use a file, however, to enter the points, so do not press **OK** yet.

## 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** reference plane coordinate system.



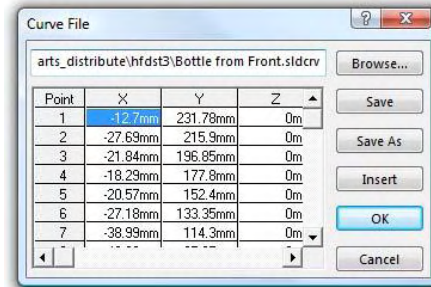
## 8. Insert Curve

The **Curve File** dialog was already opened in the previous steps. We will now overwrite the data you just entered.

## 9. 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. The data you have entered before has been overwritten.

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



## 10. 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 a **Front** view. A feature named **Curve1** appears in the FeatureManager design tree.

## Editing the Curve

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

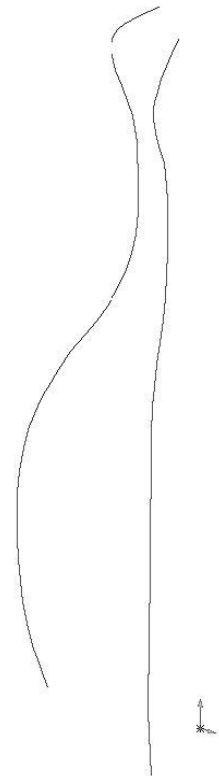
1. Browse for and substitute a replacement file.
2. Edit the existing point list.
3. Edit the original file and read it in again.

## 11. Create the Second Guide Curve

From the **Insert** menu, choose **Curve, Curve Through Points...**

From the browser, select the file **Bottle from Side.sldcrv** and click **Open**. Click **OK** to create the second guide curve. This curve represents the shape of the bottle when viewed from the side.

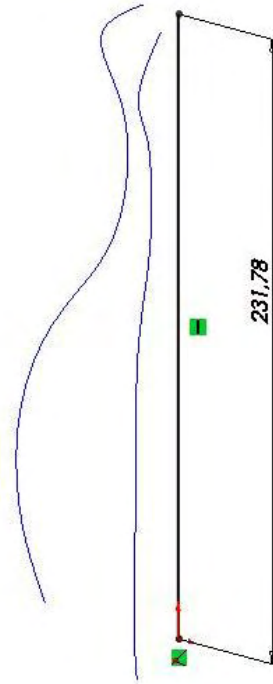
The illustration at the right shows both guide curves in a **Trimetric** view orientation.



## 12. 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 **231.78 mm**.


This line will be used as the sweep path.



## Introducing: Insert Ellipse


Sketching an ellipse is similar to sketching a circle. You 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. 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 the major axis. One way to do this is with a **Horizontal** relation between the ellipse center and the end of the major axis.

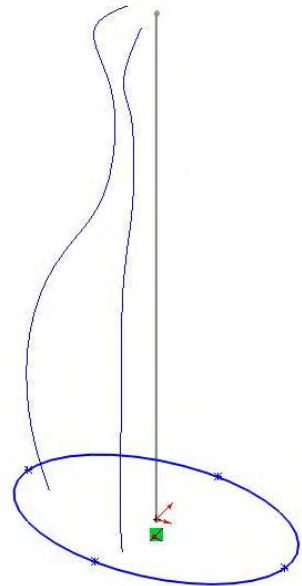
### Where to Find It?

1. From the menu: **Tools, Sketch Entity, Ellipse.**
2. Or, from the Sketch Tools tab pick the tool: .

## 13. Sweep section

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

From the Sketch Tools tab, pick the **Ellipse** tool   
and sketch an ellipse with its center at the origin. Don't close the sketch. You are first going to relate the ellipse to the two guide curves.



## 14. Relating the Sweep Section to the Guide Curves

The profile of the sweep section has to be related to the guide curves using the **Pierce** relation. Thus, the guides had to be created *before* the profile.

Press the **Ctrl** key, and select the point at the end of the ellipse's major axis and the first guide curve. Select **Pierce** from the PropertyManager, or right-click and select **Make Pierce** from the shortcut menu. Repeat this procedure for the minor axis and the second guide curve.

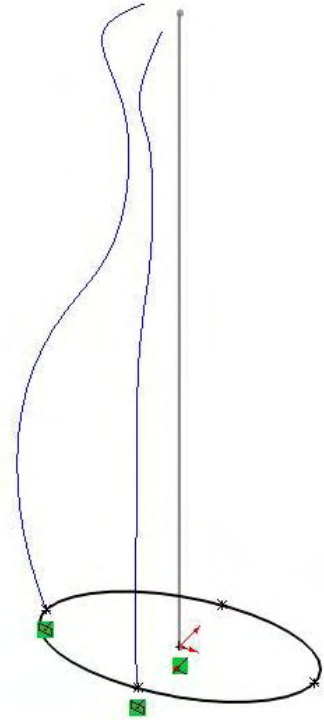
When adding a Pierce relation, you should pick the point first and *then* select the curve that pierces the sketch plane.

## 15. 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.

## 16. Exit the Sketch

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

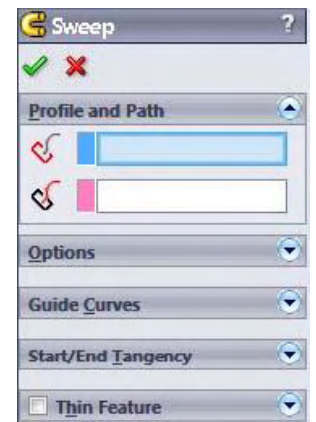


## Sweep Dialog

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

The dialog is divided into five sections of group boxes:

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



The **Show preview** option has advantages and disadvantages. While it is very nice to see a shaded preview as you select the profile, path, and guide curves, there are performance considerations. Each time you select one of the sweep components, the system has to regenerate the preview. This takes time. The more complex the sweep, the longer it takes.

For this example, the **Show preview** option is checked.



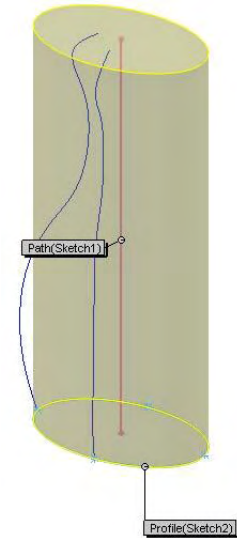
## 17. Sweep PropertyManager

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

## 18. Select Profile and Path

Make sure that 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 results without the effect of any guide curves.

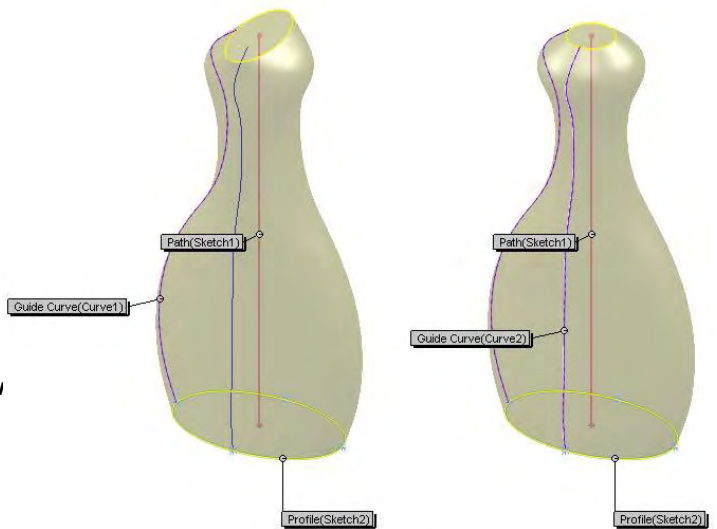


## 19. Guide Curves


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

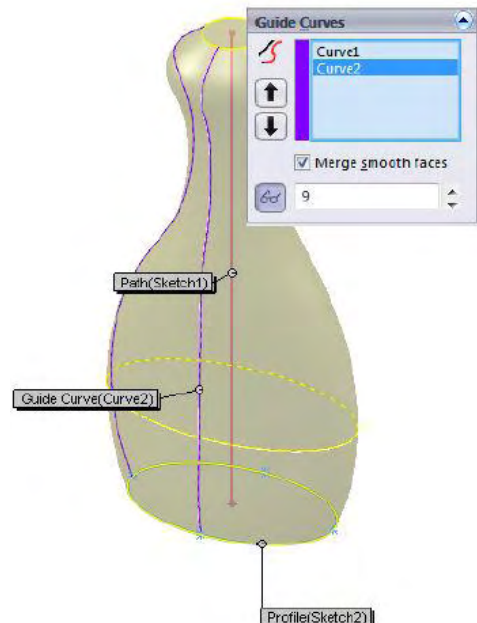
A callout appears only on the last guide you select.

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



## 20. 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.





## 21. Options

Expand the **Options** group box, and make sure that the default **Follow path** is selected.  
Click **OK**.



## 22. Finished Sweep

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

# Insert Design Library Feature


Library Features are used to add cuts, bosses, or sketches to a part. Although palette features are a kind of library feature, the regular library feature is somewhat different. The library feature is a little more versatile because:

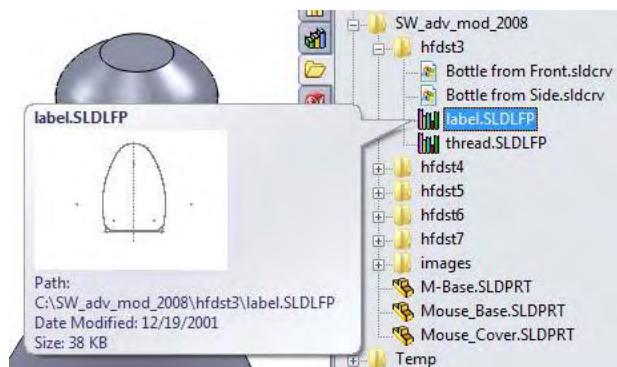
- It is inserted using a different method.
- It can have multiple **Mandatory References**. Palette features can have only one.
- It can be inserted onto reference planes. Palette features can only be inserted onto planar faces.

## Introducing: Insert Library Feature

**Library Features** are used to add pre-defined cuts, bosses, or sketches to a part. They are a special file type of **\*.sldlfp** file. Although we do not normally think of sketches as features, when working with library features, they are.

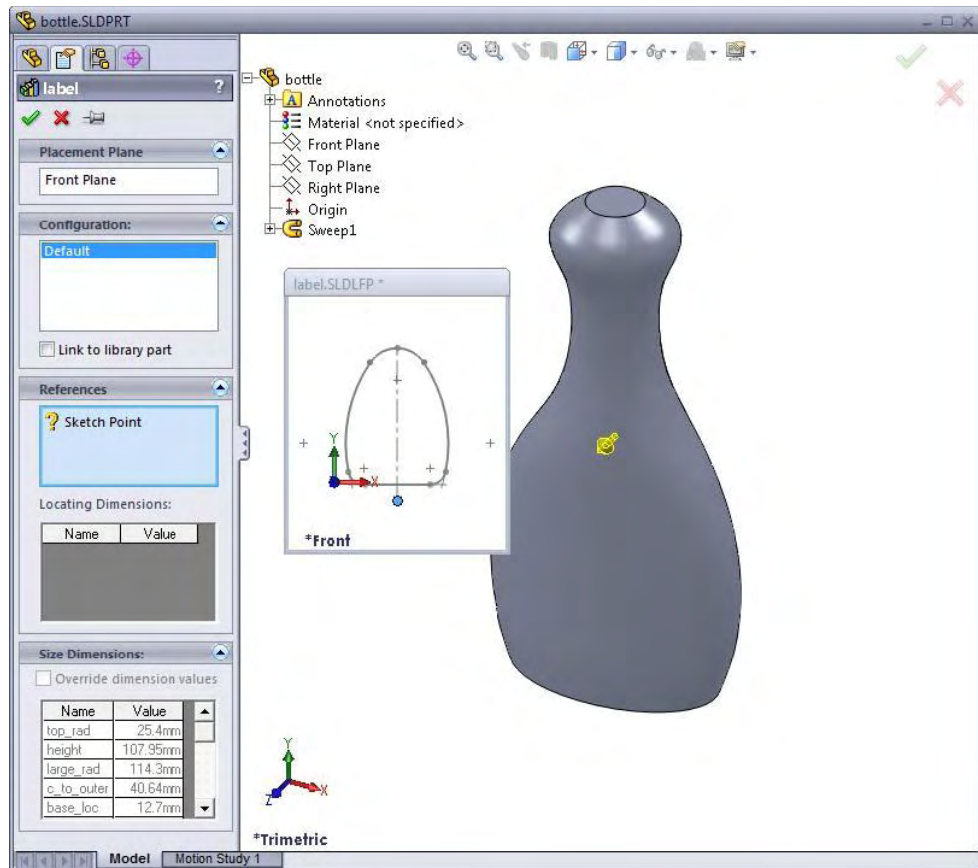
## 23. Insert Library Feature

Click  and select the name of the feature **Label.sldlfp** from the browser. Drag and drop the Library Feature into space and then select the front plane in the fly-out Feature manager.



## 24. Multiple Windows

A new window will open with a preview of the feature. The blue point shows the reference point of the sketch.

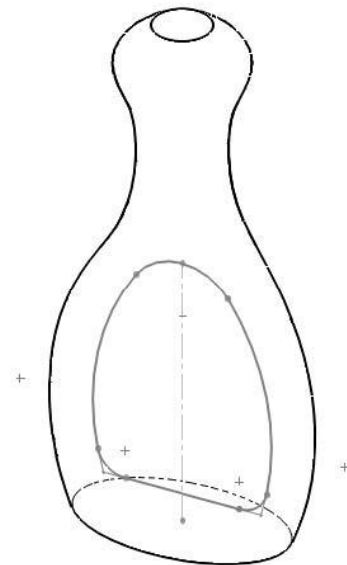


## 25. Selections

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

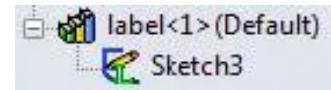
## 26. Results.

The library feature is inserted into the part and is related to the **Front** plane and the origin. The sketch is fully defined. If the **Reference** had not been satisfied, the sketch would have had a dangling relation.



## The Library Feature Folder

The sketch appears in the FeatureManager design tree in a folder named **label1**. The actual sketch cannot be used in this form. It must be removed from the library feature folder.



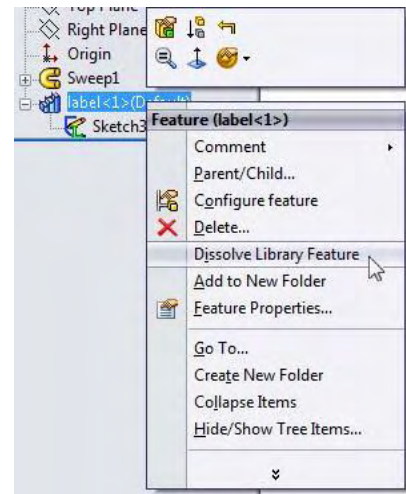
## Introducing: Dissolve Library Feature

**Dissolve Library Feature** is used to break down the **label<1>(Default)** folder. This removes the library feature icon and causes each of the features it contained to be listed individually in the FeatureManager design tree.

### 27. Dissolve

Right-mouse-click the library feature select. Choose **Dissolve Library Feature** from the menu.

The **label<1>(Default)** folder is removed and the sketch it contained is listed individually in the FeatureManager design tree. It can now be used to create a projected curve.



## 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 is created using a **Library Feature**.


### Introducing: Insert Projected Curve

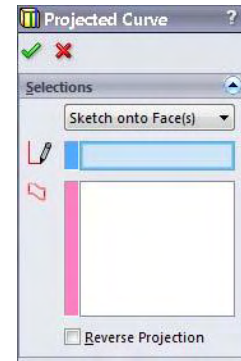
**Projected Curve** projects a sketch onto a face or faces of the model. When these faces are curved, the result is a 3D curve. This command can also merge two orthogonal sketches into one 3D curve.

### Where to Find It?

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

## 28. Projected Curve Dialog and Preview

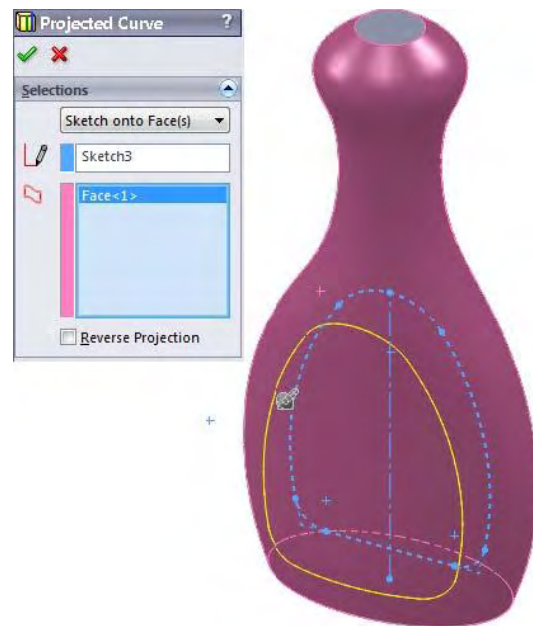
Click , or on the **Insert** menu, choose **Curve, Projected...** Choose the **Sketch onto Face(s)** option from the list.



## 29. Selection

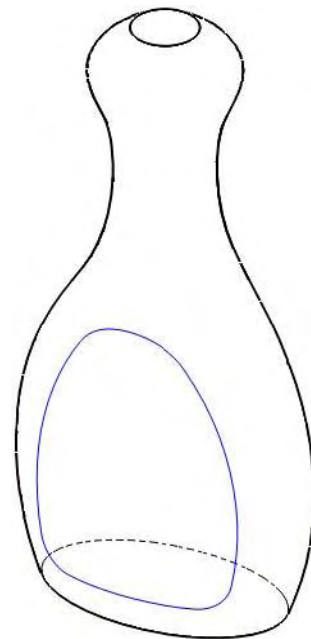
Click in the Sketch to Project list and select the sketch. Click in the Projection Faces list and select the model face by default, the system projects the sketch normal to the sketch plane (along the positive Z-axis). If you want to project the curve onto the back of the bottle, click **Reverse Projection**.

Click **OK**.



## 30. 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 label area on the bottle.



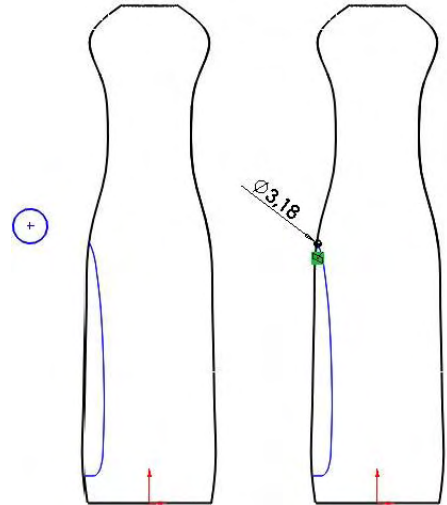
### 31. Sketch the Profile

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

### 32. Pierce Relation


Add a **Pierce** relation between the center of the circle and the projected curve to define its location. Dimension the circle to **3.18 mm** 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.



### 31. Sweep the Boss for the Label Outline

Exit the sketch.

Click  **Sweep Boss/Base**, or on the **Insert** menu, choose **Boss/Base, Sweep**. 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.



### 32. Add the Neck

Select the top face of the base feature and open a sketch. Use **Convert Entities** to copy this edge into the active sketch. Extrude the sketch upward a distance of **15.88 mm**.




# Variable Radius Filleting

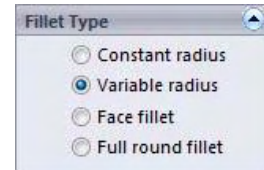
A variable radius fillet runs around the bottom of the bottle. Variable radius fillets are defined by specifying a radius value for each vertex along the filleted edge with the option of adding more control points along the edges. Variable radius control points operate as follows:

- The system defaults to three control points, located at equidistant increments of 25 percent, 50 percent, and 75 percent 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 is only a single vertex on the bottom edge of the bottle. Therefore, we will use control points.

### 33. Fillet the Bottom

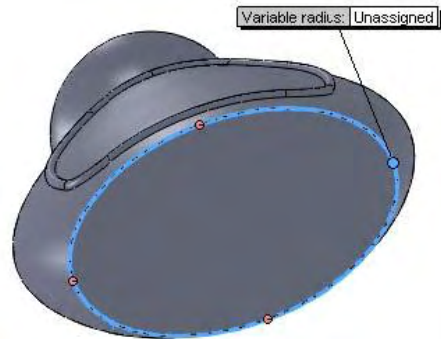
Click  on the Feature tab. For **Fillet Type**, choose **Variable radius**.



### 34. 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.

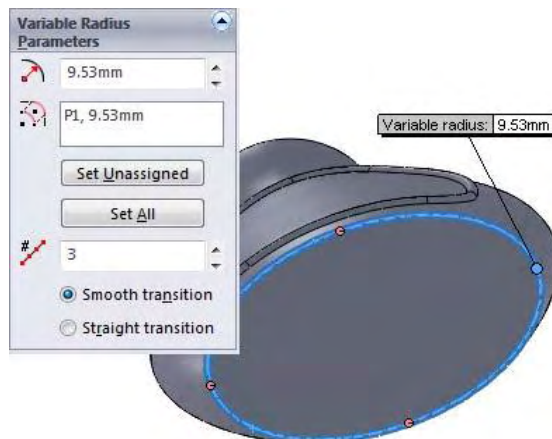


### 35. Assign Radius Value to the Vertex

Click the callout and enter a radius value of **9.53 mm**.

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

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.

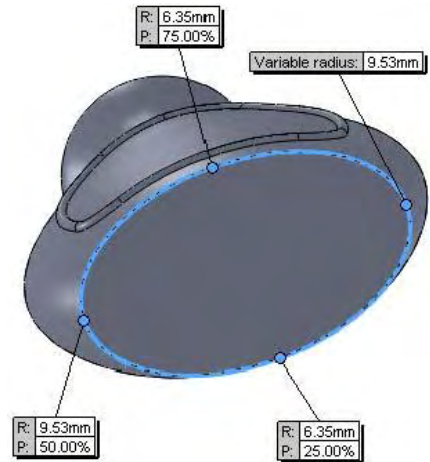




### 36. Radius Values

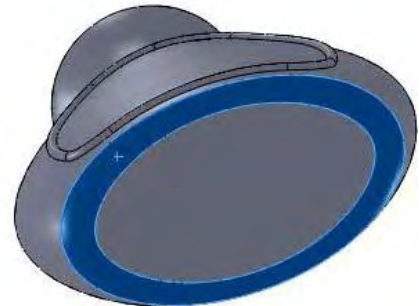
Click the control points and use the callouts to set the radius **R** to **6.35 mm** and **9.53 mm** as shown. Leave the positions **P** at their default values of **25%**, **50%**, and **75%** as shown in the illustration at right.

Click **OK** to create the fillet.



### 37. Result

The result of the variable radius fillet is shown in the illustration at right. The fillet forms a closed loop varying smoothly from 9.53 mm to 6.35 mm to 9.53 mm to 6.35 mm and back to 9.53 mm 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 9.53 mm from the bottom face. In other words, rather than have the fillet define the rail, we will define where the rail should be, and let the system compute the fillet radius.





### Adding a Split Line

A split line is used to divide the 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 line

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

#### Where to Find It?

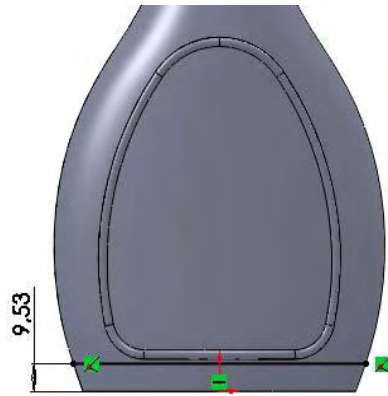
- Click **Ins**  **Curve, Split Line...**
- Or, click  on the Curves toolbar.

### 38. Delete the Fillet


Right-click the variable radius fillet, and select **Delete** from the shortcut menu.

### 39. 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.



### 40. Projection Split Line

Click , or **Insert, Curve, 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.

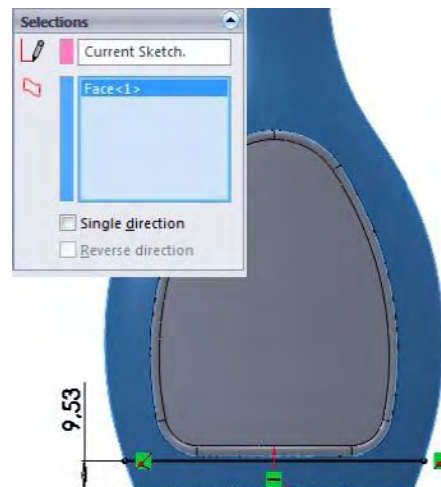


### 41. Select Faces

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.



### 42. 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 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

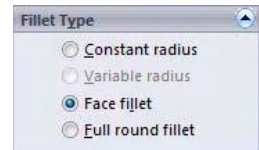
The **Fillet** command has an additional group box, **Fillet Options**, where **Hold Line** can be assigned to define the fillet’s tangent edge or rail. Defining the rail of the fillet defines the radius of the fillet. In this example, the edge created by the split line will be used.

## Where to Find It?

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

## 43. Insert Fillet

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

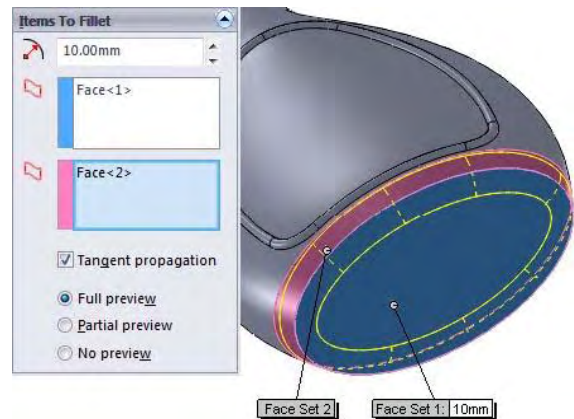


Since the **Hold Line** will define the radius, you do not need to enter a radius value. Also, when you expand the **Fillet Options** group box and select the **Hold Line**, the radius field disappears.

## 44. Select the faces

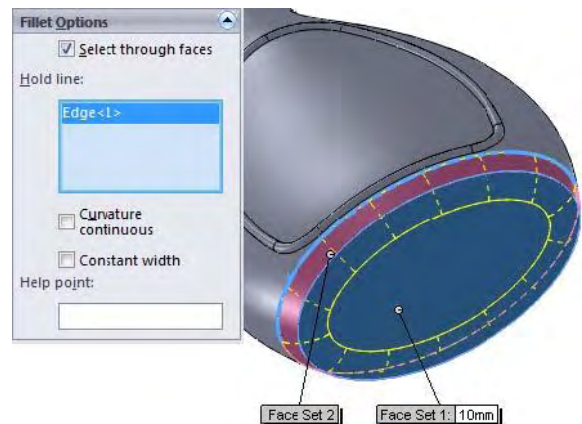
Verify that the **Face Set 1** selection list is active (the list has a blue bar) and select the bottom face of the bottle. Activate the selection list **Face Set 2** (the list has a purple bar) and select the face created by the split line.

Enable the Full preview option to get a better view of the fillet being created.



## 45. Fillet Options

Expand the **Fillet Options** group box. Click the **Hold Line** selection list, and select the edge created by the split line. Click **OK** to create the fillet.



## 46. Results

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



# Analyzing Geometry

SolidWorks software has several tools that are used to obtain information and to assess the quality of curves and surfaces. Some of these tools are:

- **Display Curvature**
- **Inspect Curvature**
- **Zebra Stripes**

## What is Curvature?

To avoid getting too deep into mathematics, we will establish a 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 value of curvature, the more flat a surface is.

## Introducing: Display Curvature

**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).

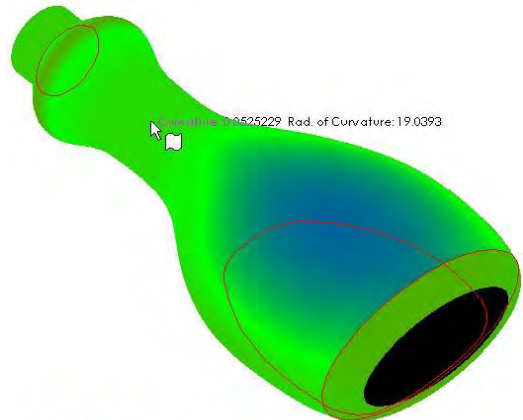
## Where to Find It?

- Click **View, Display, Curvature**.
- You can display the curvature for selected faces by right-clicking the face and selecting Face Curvature from the shortcut menu.

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.

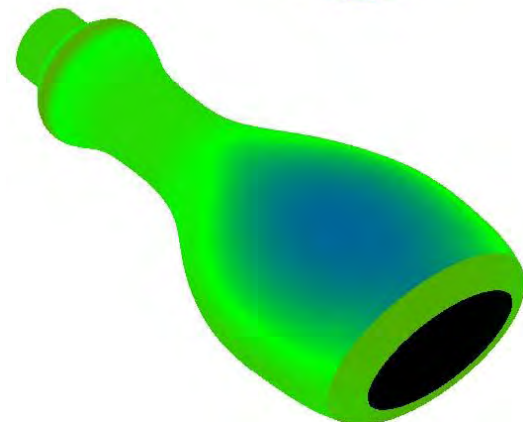
## 47. Display Curvature

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



## 48. Look at the Fillet

Notice the dramatic change of 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



## 49. Turn off Curvature Display

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

## Inspect Curvature

**Inspect Curvature** provides visual representation of the slope and curvature of most sketch entities. You can use **Inspect Curvature** 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: Inspect Curvature

**Inspect Curvature** gives a graphic representation of the curvature in the form of a series of lines called a *comb*. The length of the lines (teeth) represent the curvature. The longer the lines, the greater the curvature and the smaller the radius will be.

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 **Inspect Curvature** to learn other things about how curves are connected. Look at the illustration at 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 at common endpoint are:

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

In the illustration at 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 **Remove Curvature Information** from the shortcut menu.

### Where to Find It?

- Right-click the sketch entity, and select **Inspect Curvature** from the shortcut menu.

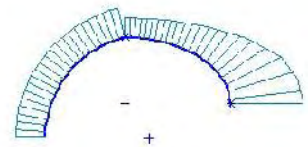
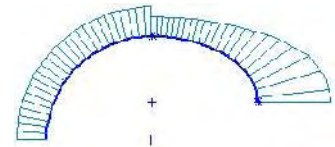
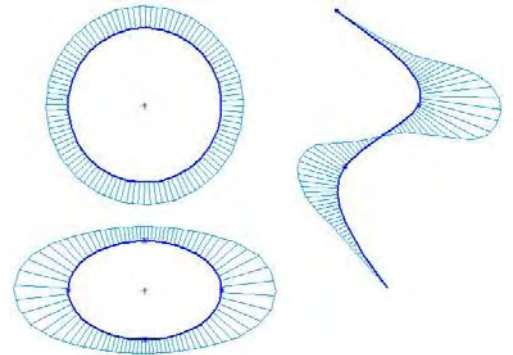
## Intersection Curve

**Inspect Curvature** 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 kind 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**  under **Convert Entities** on the Sketch Tools tab.
- Or, click **Tools, Sketch Tools, Intersection Curve**.

### 50. Intersection Curve

Select the Front reference plane and open a sketch.

Click .


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

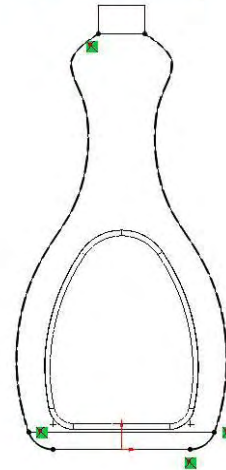


### 51. Results


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.

### 52. Turn Off the Intersection Curve Tool

Click  again to turn off the tool.



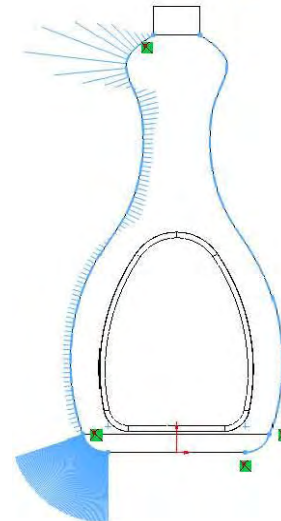
### 52. Inspect Curvature

Right-click one set of the intersection curves in the sketch, and select **Show Curvature Combs**  from the shortcut menu (or **Tools, SplineTools, Show Curvature**). Note the following:

- The fillet has a circular cross-section.
- 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.

### 53. Turn Off the Curvature Display

Right-click the intersection curves, and clear **Show Curvature Combs** from the shortcut menu.



### 54. Exit the Sketch

### 55. Rollback

Right-click the sketch; and select **Rollback**  from the shortcut menu.



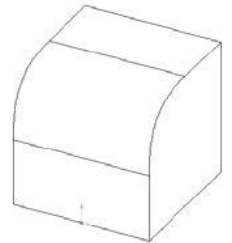
## Zebra Stripes

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 the 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 stripes 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 the 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.

Singularity

Contact

Tangent

Curvature Continuous



### Where to Find It?

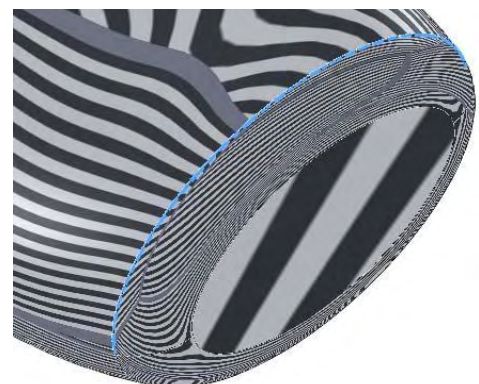
- Click **View, Display, Zebra Stripes**.

### 56. 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**.

## 57. Turn Off Zebra Stripes

## 58. Rollback

Right-click the fillet, and select **Rollback**  from the shortcut menu.

## 59. Second Split Line

Open a sketch on the bottom face and create an offset of **9.53 mm**. Use this sketch to split the bottom face.

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

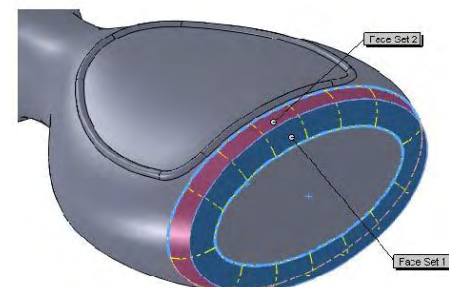


## 60. Roll Forward and Edit Feature

One of the face set lists will contain two faces. Click in that list, and deselect the middle face created by the split line.

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

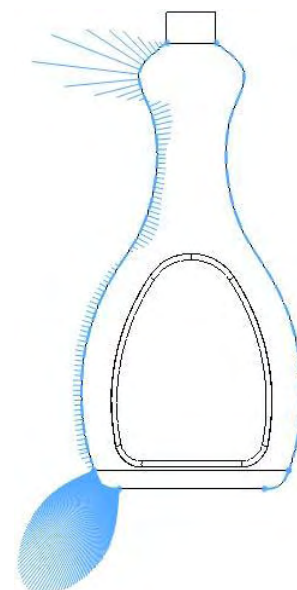
Click **Curvature continuous**, and **OK**.



## 61. Inspect 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.

**Note** You do not have to edit the sketch to inspect the curvature. Just right-click the sketch entities, and select **Show Curvature Combs** from the shortcut menu.



## 62. Delete the Sketch

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

## 63. Zebra Stripes

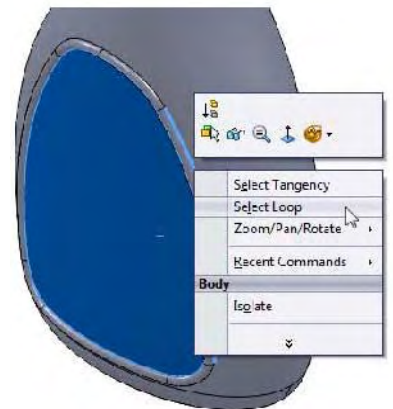
Click **View, Display, Zebra Stripes**. Examine how the stripes blend from the body of the bottle to the fillet.



## 64. Turn Off Zebra Stripes Display

## 65. Fillet the Label Outline

Run a **1.5 mm** radius fillet around the inside *and* outside edges of the swept label outline. This fillet has to be added *before* the bottle is shelled. Use a **Face** selection on the inner edges and **Edge** selection on the outer (right-click the edge, and select **Select Loop** from the shortcut menu).



## 66. Shell the Bottle

Create a multi-thickness shell, removing the top of the bottle neck. Use a wall thickness of **1.5 mm** for the neck and **0.5 mm** for all the other faces. The illustration at right shows a section view, viewed from the back.



## 67. Save Your Work

You have invested a lot of time into this case. 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.

## 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. Do not suppress a feature if you will need to reference its geometry later.


## Accessing the Suppress Command

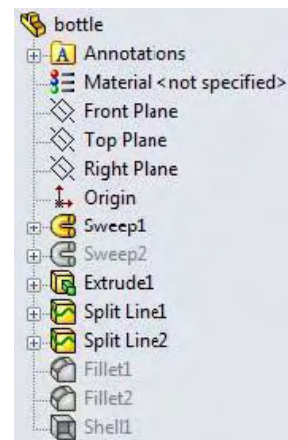
There are several ways to access the **Suppress** command:

- From the menu: **Edit, Suppress, This Configuration**
- From the right-mouse menu: **Feature Properties...**
- From the Context menu: **Suppress** 

## 68. Suppress Features

In the FeatureManager design tree, select the features for the label outline (**Sweep2**), the face fillet (**Fillet1**), the fillet around the label outline (**Fillet2**) and the multi-thickness shell (**Shell1**).

From the Context menu, pick **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: the Helix Command

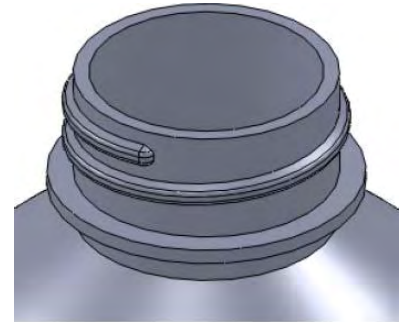
**Insert, Curve, Helix** creates a helical 3D curve based on a circle and definition values, such as pitch and number of revolutions. The curve can then be used as a sweep path.

## Where to Find It?

From the pull-down menu choose: **Insert, Curve, Helix....**

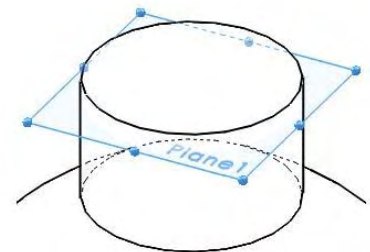
## Procedure

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



### 69. Offset Plane

Create a reference plane offset **2.5 mm** below the top of the bottle neck. This is where the threads will start.

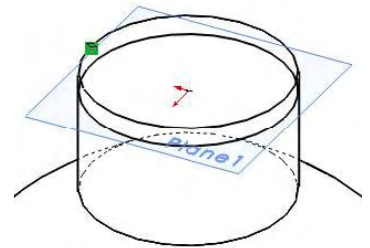


### 70. Insert Sketch

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**, or sketch a circle tangent to edge of the neck. This circle will determine the diameter of the helix.

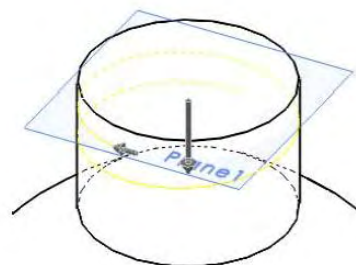


### 72. Create the Helix

From the **Insert** menu, choose **Curve, Helix....** The **Helix Curve** dialog is used to specify the definition of the helix. The threads have a **Pitch** of **3.81 mm** 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.

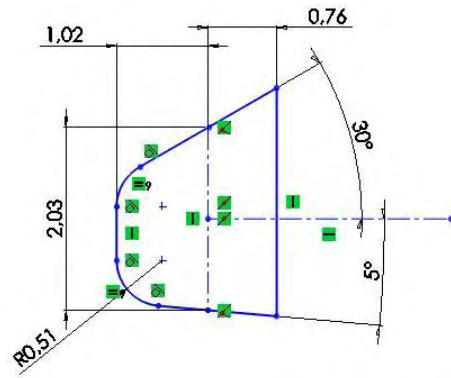




### 73. Insert a Sketch

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

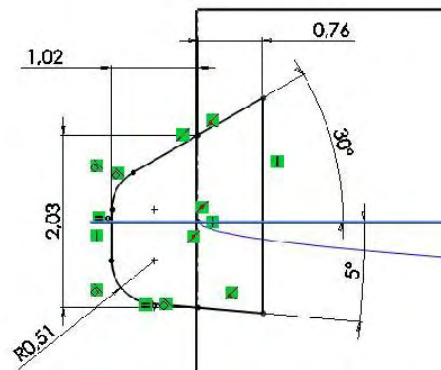
Dissolve the library feature and edit the sketch.



### 74. Relations

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

Add another **Collinear** relation between the vertical centerline and the outer silhouette edge of the model. If you are experiencing problems you can also try to define another relation by converting an entity and setting it to 'for construction'.

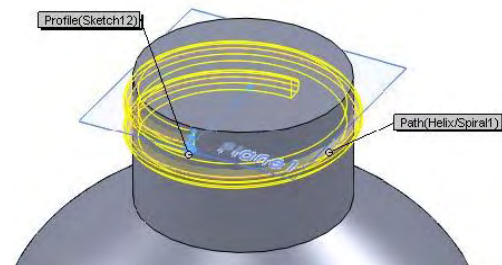


### 75. Exit the Sketch

### 76. Sweep the Threads

Select the sketch and the helix. Open the **Sweep** dialog. Verify that the sketch is being used as the sweep section and the helix as the sweep path.

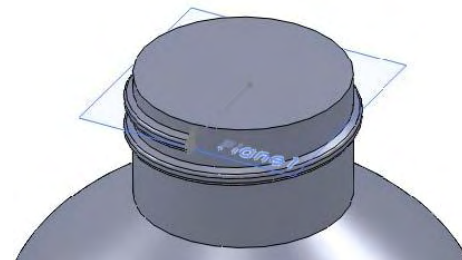
Verify that the **Align with End Faces** option is deselected and click **OK**.



**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.

### 77. Results

The result of sweeping the thread is shown in the illustration at right.

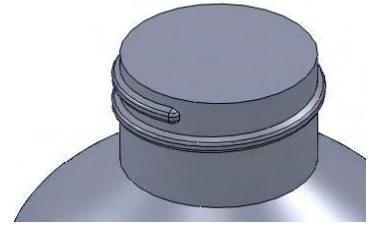




## 78. Add the Finishing Details

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.



## 79. The Finished Bottle

**Unsuppress** the label outline, the fillets, and the shell to show the completed bottle.

**Note** 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.

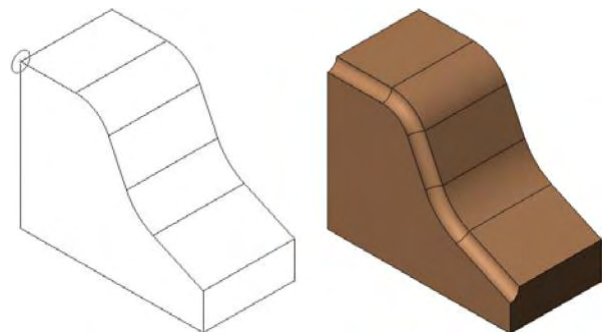


## Sweeping Along Model Edges

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.

### Align with End Faces

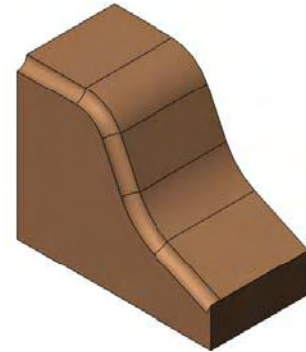
You are probably wondering what the option **Align with End Faces** does. Consider this simple example. Suppose you wanted to create a cut by sweeping a profile along the edge of a model as illustrated at right.



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.

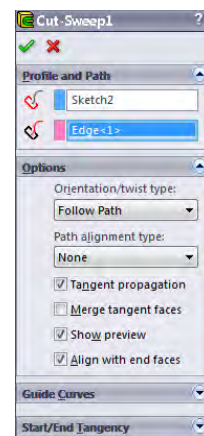
The reason we did not use **Align with End Faces** when sweeping the threads is because there were no end faces with which the boss could align. Using it in that case could have forced the system to give an incorrect result. Fortunately, **Align with End Faces** is not selected by default when sweeping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



## Propagate Along Tangent Edges

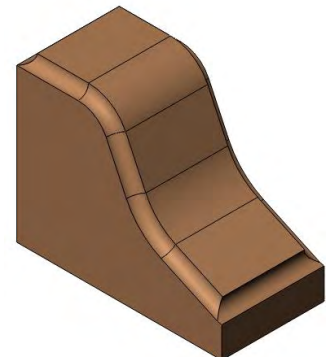
When you select a model edge as a sweep path, an additional option becomes available in the **Sweep** dialog. This option is **Tangent Propagation**, and it serves the same function as the similar option in filleting. If you select a single segment of the edge, this option causes the sweep to continue 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-button menu option **Select Tangency**.



## What if the Edges Are Not 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 in the dialog only accepts one selection. There is no way to select multiple edges. And since some of the edges are not tangent, they will not 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?

1. From the **Insert** menu, pick **Curve, Composite Curve**
2. Or, from the Curves toolbar, pick the tool: 

### 1. Composite Curve Dialog

Click **Open** and select the file **Align end Faces** from the directory.

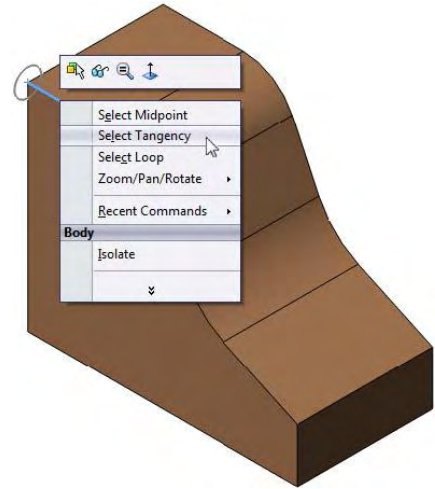
Click the tool  to open the **Composite Curve** dialog.



## 2. Select the Edges

Right-click one of the side edges and choose **Select Tangency**. **Select Tangency** is used to select a tangent-continuous chain of edges.

All the tangent edges are chosen.




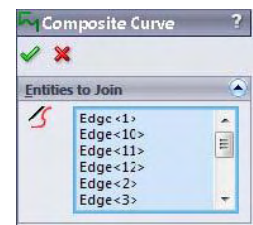
## 3. Select Remaining Edges

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:

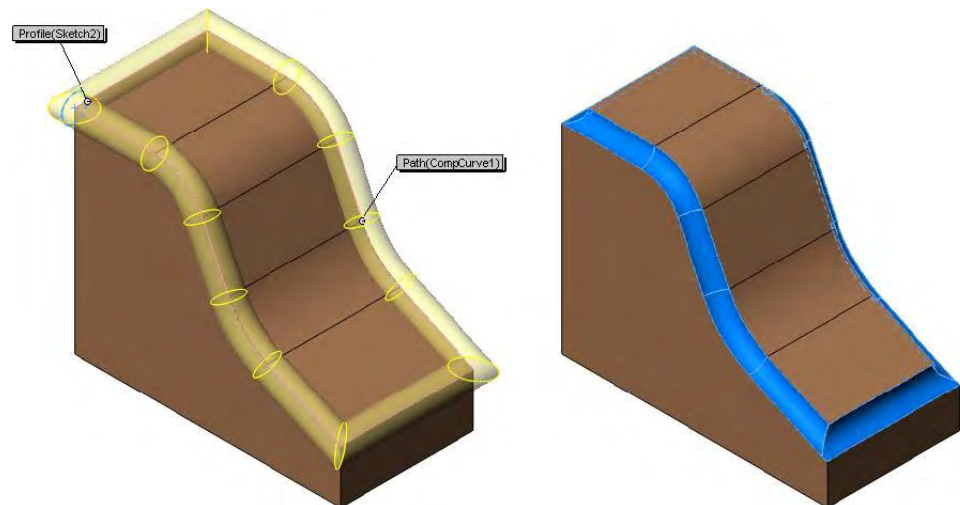
 **CompCurve1**. You can edit the definition of the curve to add or remove edges.



## 5. Sweep the Cut

Use **Insert, Cut, Sweep** and select the circle as the **Sweep Section**. Select the composite curve for the **Sweep Path**.

Click **OK**.





# Modeling Advanced Shapes: Part Two

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.

# 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 and top of the tapered boss, as well as its height. This sort of problem lends itself to lofting. We will begin by creating two sketches – one for the bottom and one for the top. The top sketch will be drawn on a plane offset from the base. This offset is the height of the boss.



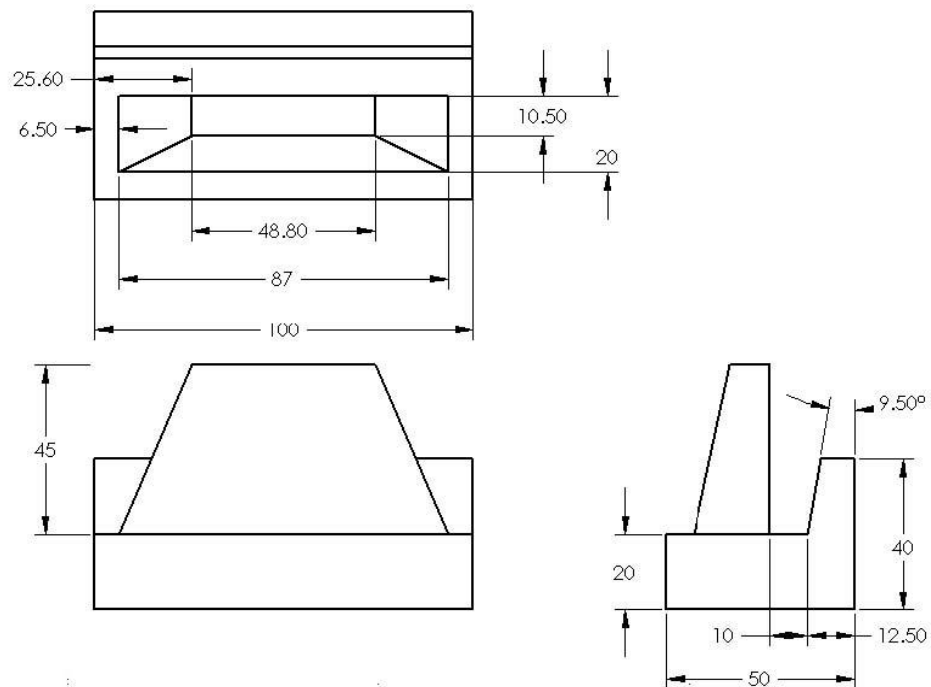
## Stages in the Process

The major steps in this operation are:

- **Create the start and end 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.
- **Take advantage of the option of creating guide curves.**  
Guide curves are an effective option with lofting to obtain greater control over the *transitions* between the profiles.
- **Insert Loft between profiles.**  
The locations where you select each profile and the order in which you select them are important.

## Example

The critical feature in this example is the tapered boss built on the top surface of the base. The drawing below shows the overall design intent. The dimensions of the bottom and top of the tapered boss are given, as is the height of the boss.





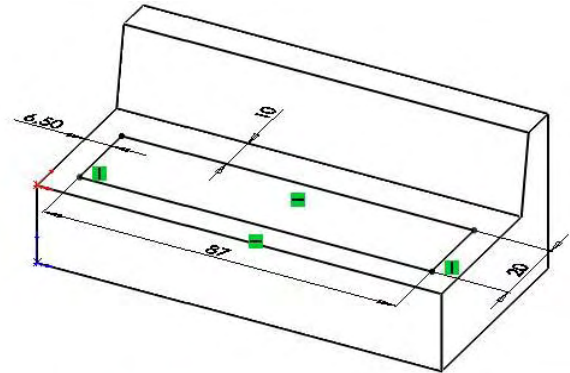
The way that the drawing is dimensioned makes lofting the obvious choice. If the tapered feature had been dimensioned showing the angles of the sides, we would pursue a different strategy. Given angular dimensions, we would extrude a simple rectangular boss and apply draft to its sides.

## Procedure

Consider the following procedure:

### 1. Sketch the First Profile

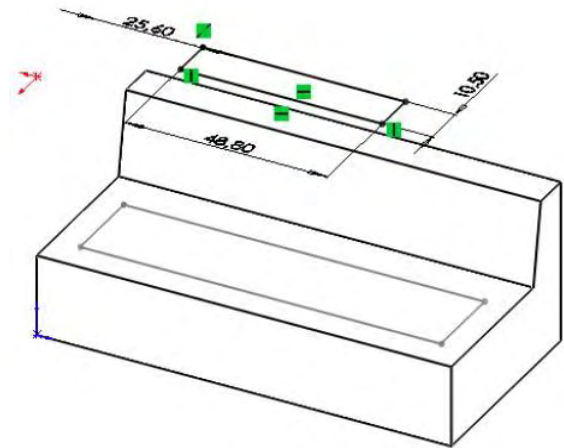
Open the part **BasicLofting.SLDPRT** that came with this workbook. Select the upper face of the base and sketch the bottom profile of the tapered boss using the dimensions given.



### 2. Define an Offset Plane

Create a plane offset **45 mm** above the face of the base.

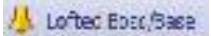
On this plane, sketch the top profile of the tapered boss using the dimensions shown.



## Introducing: Insert Loft

**Insert Loft** creates a boss or a cut using profiles and sometimes guide curves. The loft is created between the profiles. The optional guides give additional control over how the shape between the profiles is generated.

### Where to Find It?

- From the Features tab choose the Loft tool .
- Or, click **Insert, Base/Boss/Cut, Loft...**

### 3. Exit Sketch

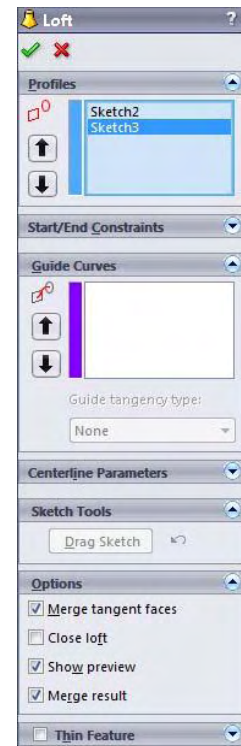
From the **Insert** menu, choose **Boss/Base, Loft...**

## 4. 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 only two sketches, the order does not matter. It is only when lofting three or more that 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.

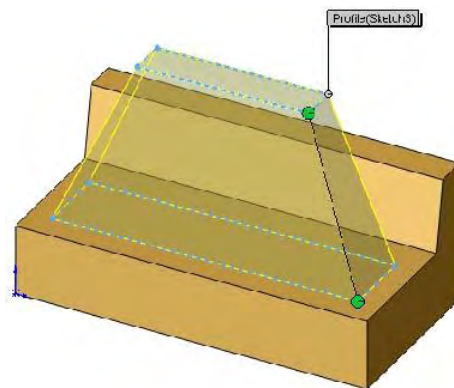
**Tip** Although **Show preview** improves visualization as you select the profiles, with complex shapes the preview tends to slow the system response.



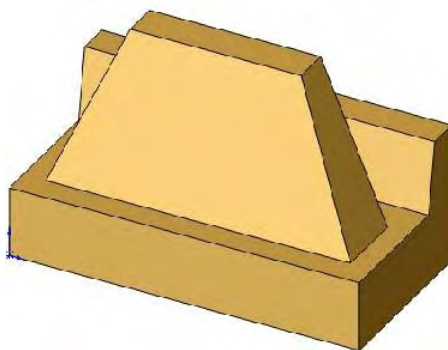
## 5. Preview

As you select the sketches, the system generates a preview line showing the vertices on the sketches that will be connected during the loft. Pay close attention to this preview because it will show you whether the loft will twist or not.

Click **OK** to create the feature.



## 6. Result



## Tangency Control


When lofting, you can control how the feature is built through 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 these influences at each end. Given the original drawing, the tangency options are not needed here. However, this is an opportune time to illustrate how they can affect the results.

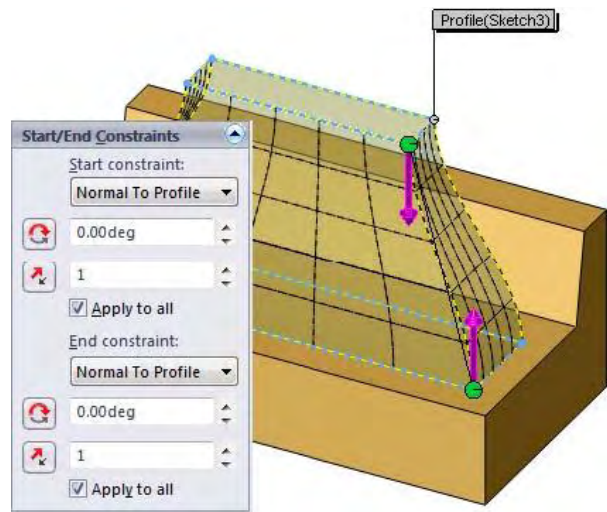
## 7. Edit Feature

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

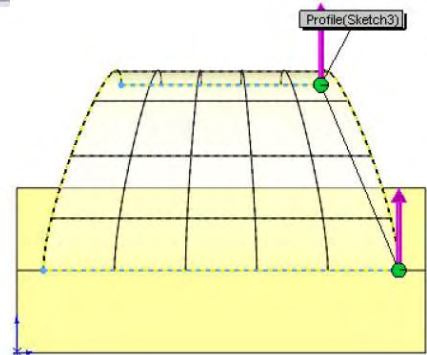
Select the options **Normal to Profile** for both the start and end of the loft.

Leave the **Start** and **End Tangent Length** values at the default **1**. The tangent vector arrows should point in the directions shown. If they do not, click  to reverse the direction.



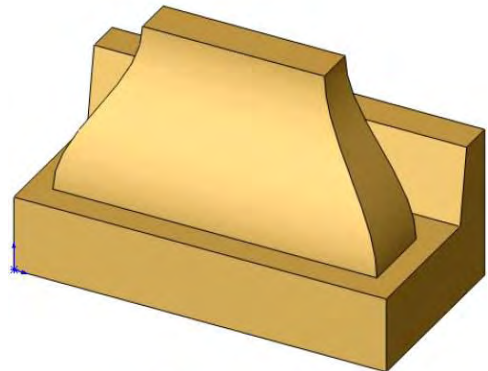
Pay attention to the preview. If the tangent arrows are incorrect, the preview will look something like the illustration at the right.

Click **OK**.



## 9. Results

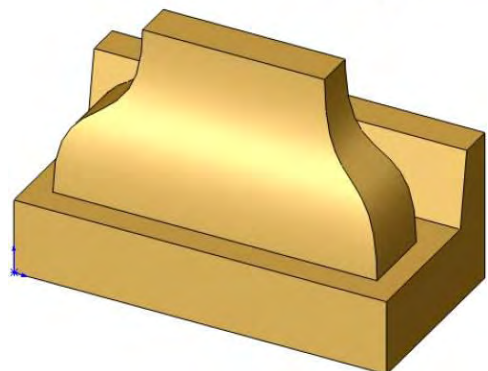
The result of this feature edit is that the shape of the loft changes so that the faces of the feature start and end normal (perpendicular) to the plane of the profile sketches.



## 10. Increase influence

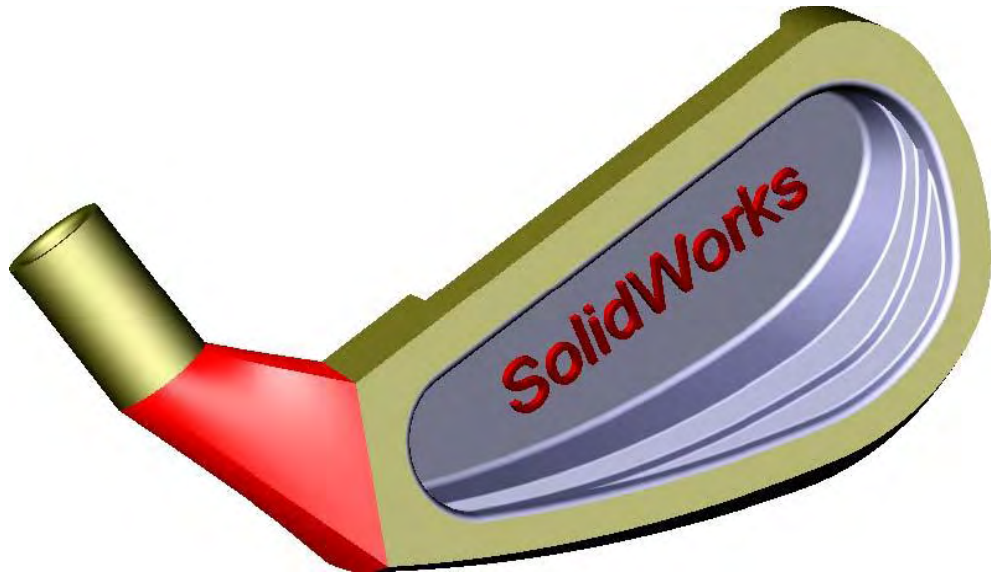
Use edit feature again to increase the influence of the start and end constraint from **1** to **2**.

**Tip** You can change the **Tangent Length** values by typing or by dragging the purple tangent vector arrows.

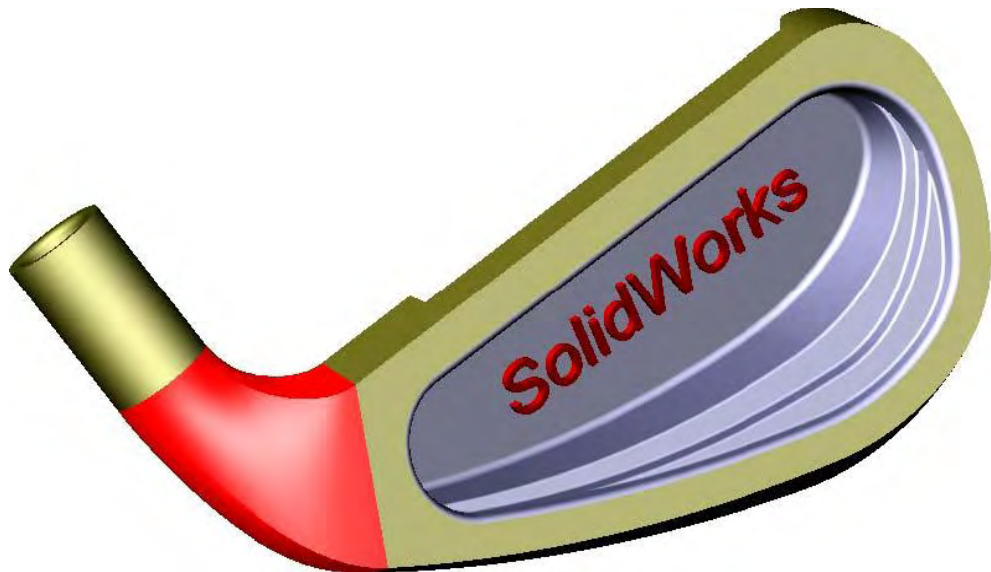


## Practical Example

The following is an illustration of how the tangency option can be applied when modeling a real part, in this case, a golf club. Consider the transition from the head of the golf club to the round shank of the shaft. Without using any tangency options, the result looks like the picture below.

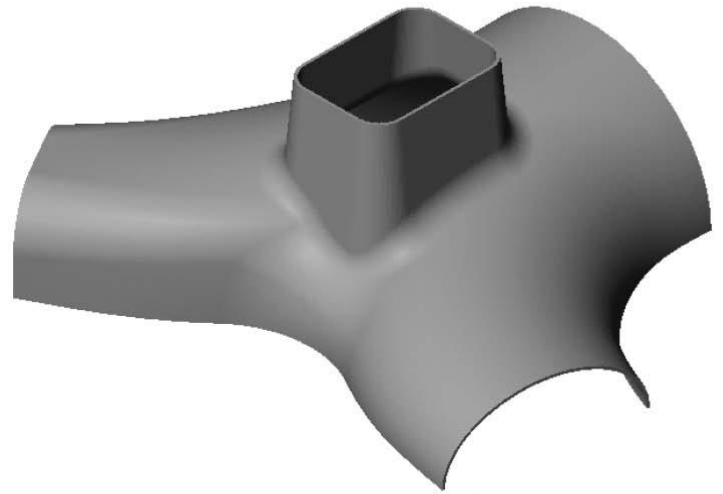


When tangency options are applied, the result is a smooth transition. The two tangency options used are **Normal to Profile** and **All Faces**.



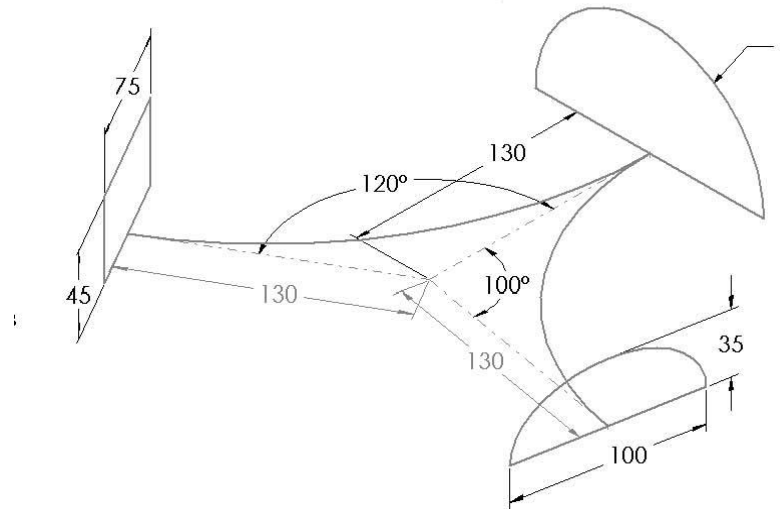
## Advanced Lofting

The part shown at right is a heat shield that goes over a hot gas manifold. It consists of several shapes – a semi-circle, a rectangle, and 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 obvious approach and best choice.



### 1. Open an Existing Part

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



### Preparation of the Profiles

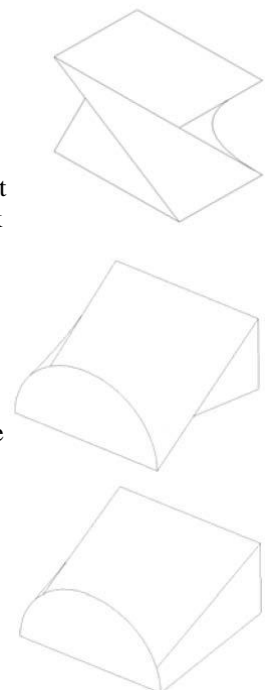
When lofting, you have to give special consideration to the way you sketch the profiles and how you subsequently select them in the Loft command. In general, there are two rules you should follow for good results:

- 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 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.

However, if you subdivide the arc, you can control exactly which portion of the arc corresponds to each side of the rectangle.

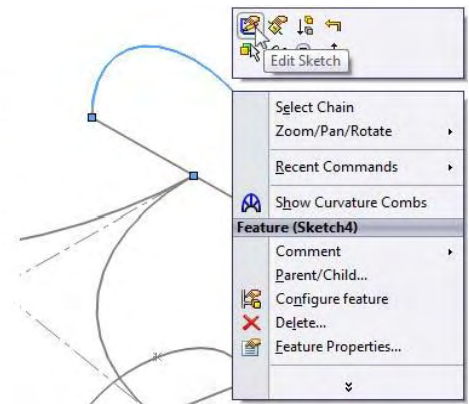




## 2. Edit Sketch

Edit the sketch of the semi-circle. The easiest way to do this is to position the cursor over the


sketch entity and choose **Edit Sketch** from the Context menu. This way you don't have to search through half a dozen sketches in the FeatureManager design tree trying to figure out which one to edit.



## Introducing: Split Entities

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

### Where to Find It?

- From the Sketch Tools tab click the split curve tool .
- Or, click **Tools, Sketch Tools, Split Entities....**

## 3. 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 co-radial, but their arc angles are under-defined.



## 4. Angular Dimensions

Dimension the arcs at  $35^\circ$  using three-point angular dimensions (select endpoint, then centerpoint, then endpoint). If you want, you can link the values of the angles so when you change one, they both change.

Exit the sketch.



## 5. Insert Loft

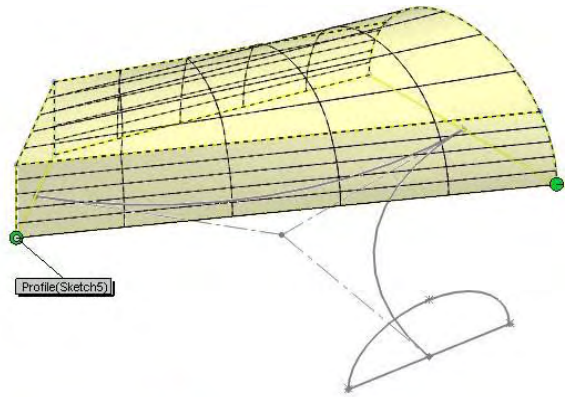
Click  **Lofted Boss/Base**, or click **Insert, Base, Loft....**



## 6. Preview

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

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.

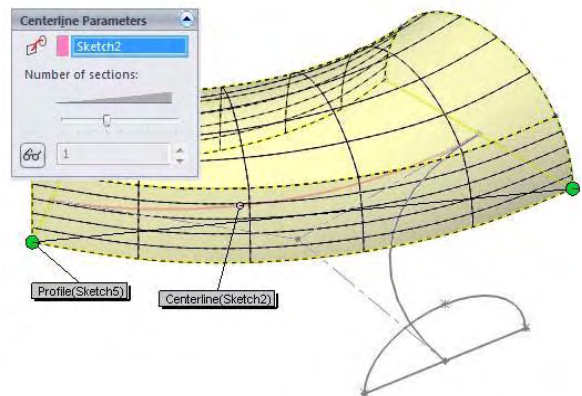


## 7. Centerline

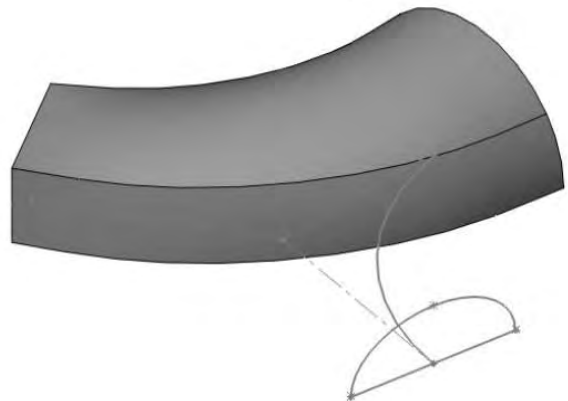
Expand the **Centerline Parameters** group box.

Select the centerline (**Sketch 2**).

Click **OK** to create the feature.



## 8. Results

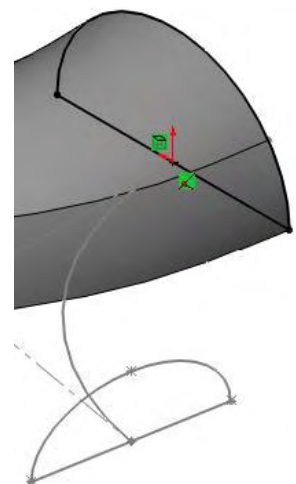


## 9. Recreate the Sketch

Although SolidWorks allows sketch entities to be used multiple times within the same model, the semi-circular sketch on **Plane 4** cannot be reused for the second loft because the circle is divided into three pieces. The profile we are getting ready to loft is not. We want the profiles to have the same number of segments.

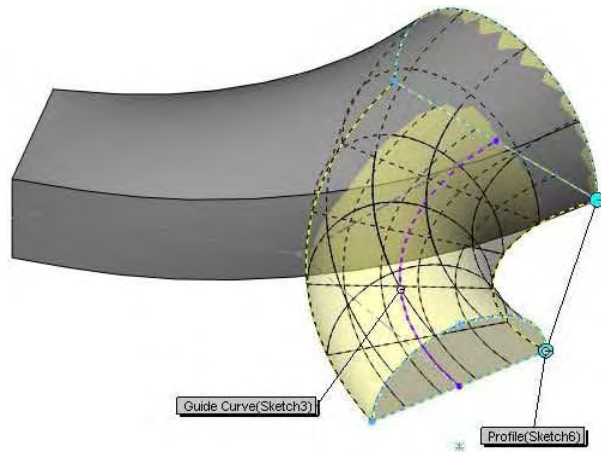
Select the plane with the semi-circle sketch (**Plane 4**) and open a sketch. Copy the horizontal line using **Convert Entities**. In the message box that appears select **Single Segment** and click **OK**. Then sketch a **180° Centerpoint Arc**.

The reason we do not simply use **Convert Entities** on the entire face is because the arc-shaped edge is also divided into three pieces.



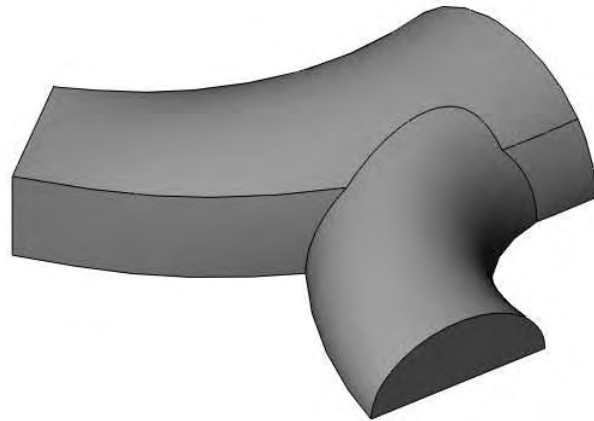
## 10. Loft

Create a centerline loft between the new sketch and the half-ellipse (**Sketch6**). Use **Sketch3** as Guide Curve.



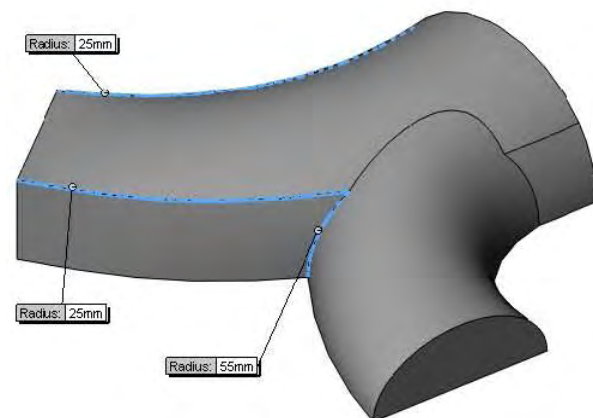
## 11. Result

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

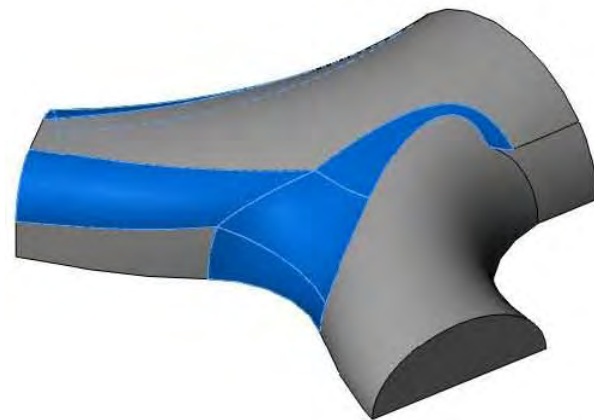


## 12. Fillet

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

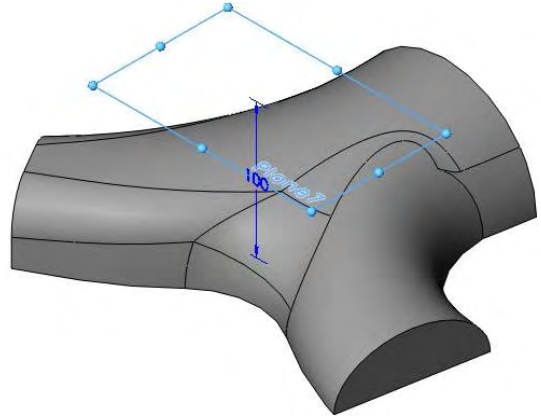


Fillets are shown in color for clarity.



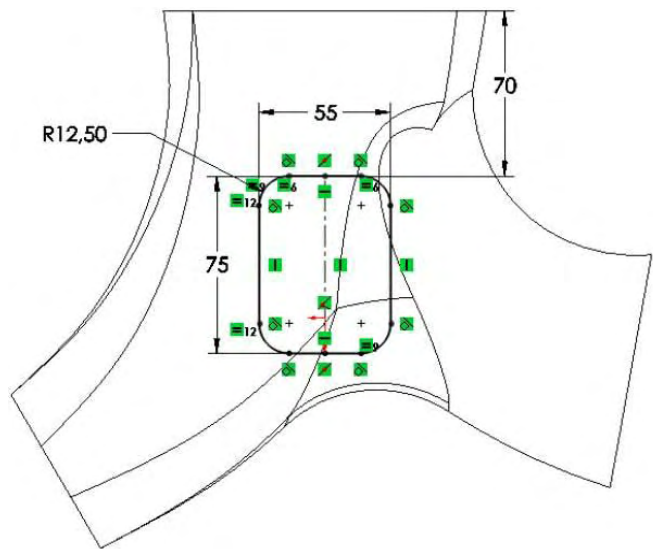
### 13. Offset Plane

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



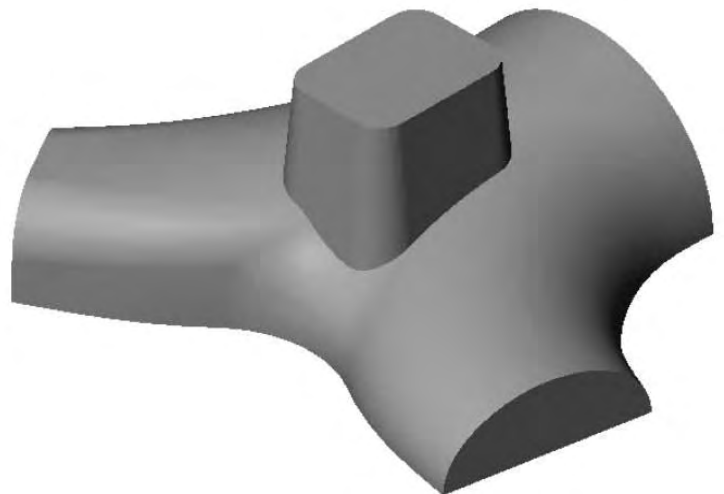
### 14. Sketch Profile

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



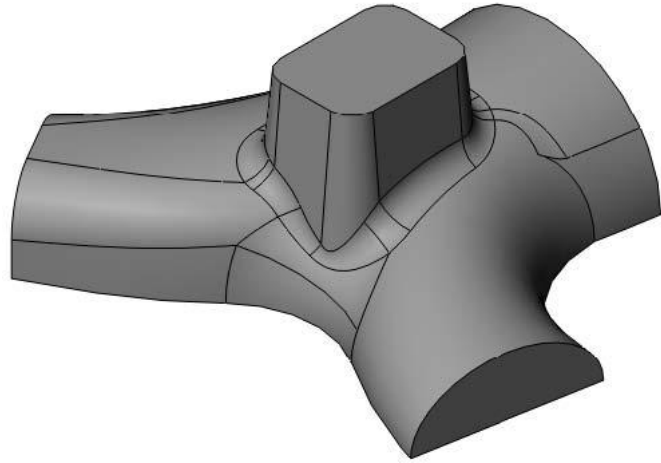
### 15. Extrude

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



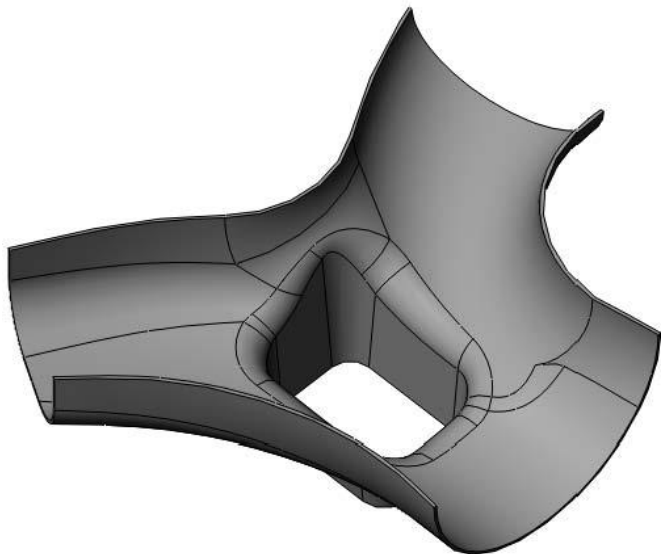
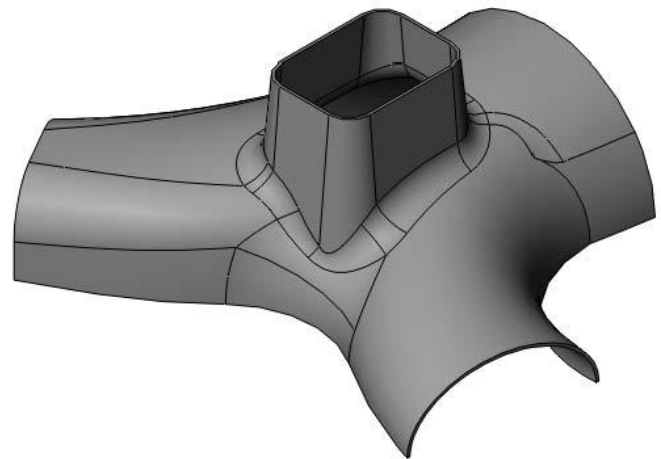
### 16. Fillet

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



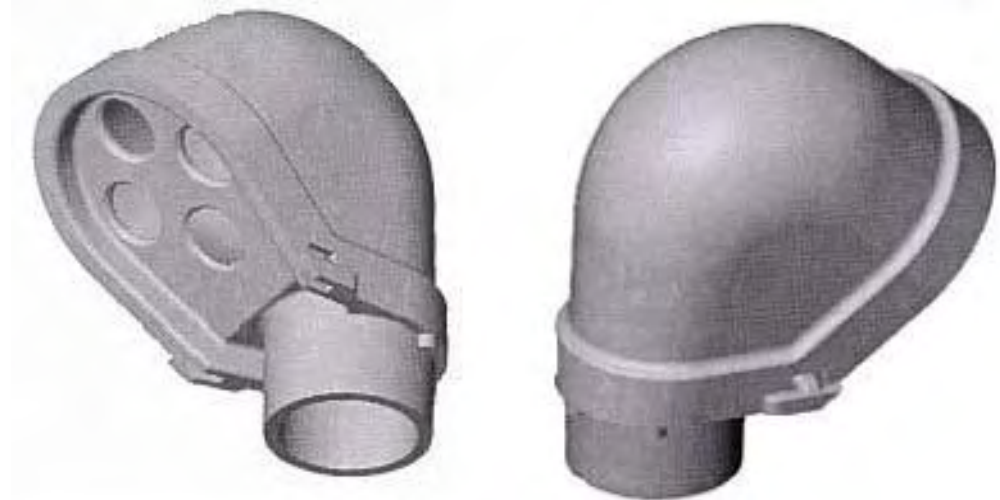
### 17. Shell

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

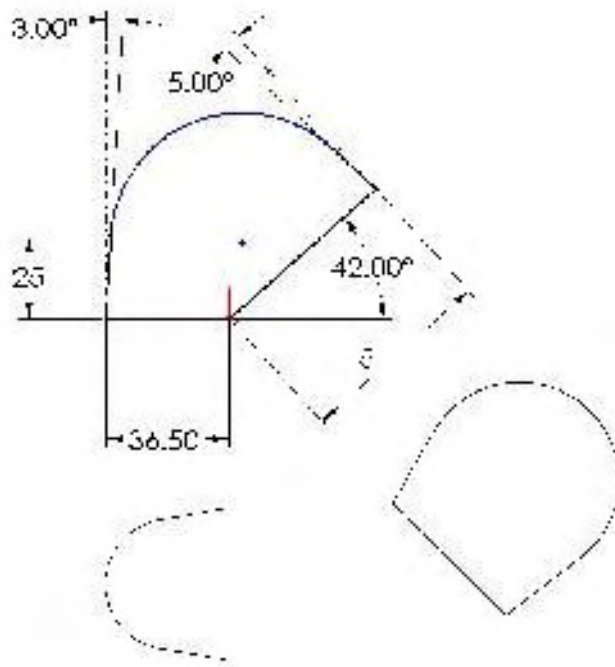


## Other Techniques

Sometimes, the best approach to modeling a free-form shape is not to use sweeping or lofting at all. 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:

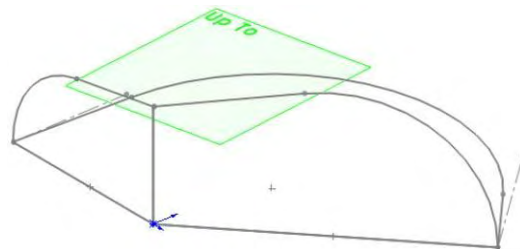
- **Extrude up to surface**  
Having defined the basic profile and the angled plane, we extrude a boss up to the plane.
- **Advanced filleting**  
We will use some advanced filleting techniques to round off the part, creating the smooth, blended transition between the two teardrop shapes.
- **Symmetry**  
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 the Part

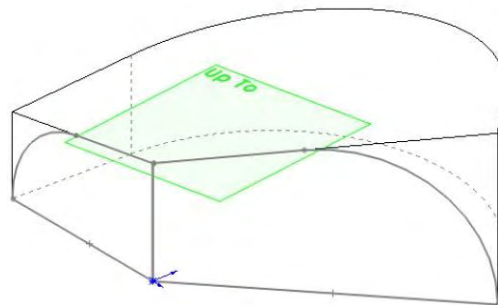
Open the part **CoverSketches**. There are three sketches used to form the profiles of the "teardrop" shape. The plane **Up To** is generated from three endpoints of sketches and therefore is skewed.



### 2. Up to Surface Extrusion

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

This makes the basic shape. Now we have to round off the edge.



## Advanced Face Blend Fillets

A face blend fillet differs from an edge fillet because 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: Advanced Face Fillets

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 base feature will be used.

### Where to Find It?

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

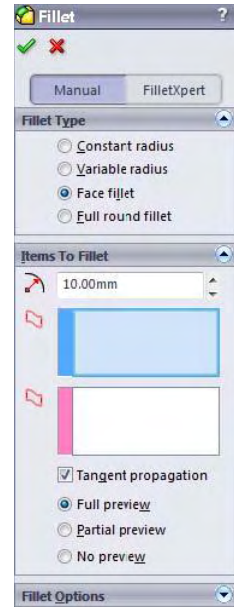


### 3. Insert Fillet



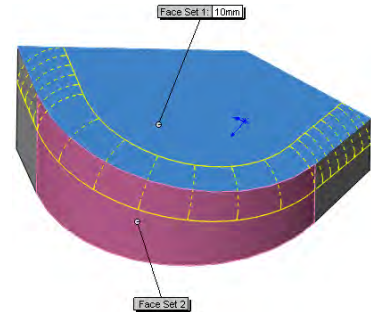
Click **Fillet**. In the **Fillet Type** group box, choose the **Face Fillet** option.

**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 line, the radius field disappears.



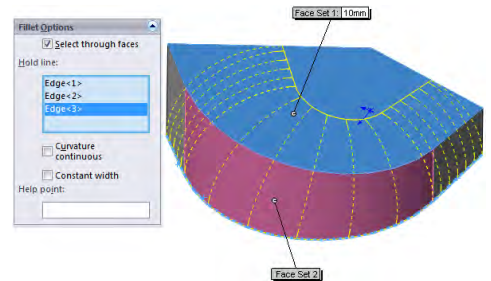
### 4. Select the Faces

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. Fillet Options

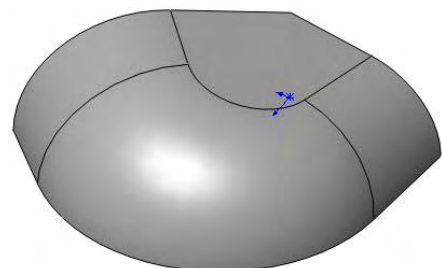
Expand the Fillet Options group box. Click in the Hold line selection list, and select the three edges as shown in the illustration.



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.

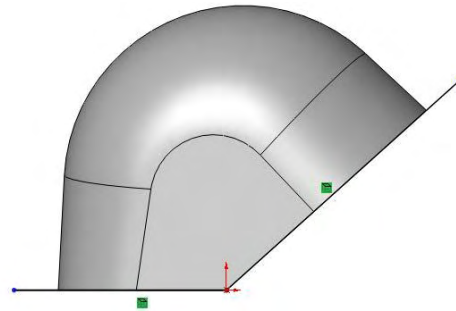


## 7. Sketch for the Boss


Switch to a **Front** view and open a new sketch on the **Front** reference plane. Select and convert the two straight edges of the base feature.

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

In the message box that appears select **Single Segment** and click **OK**.

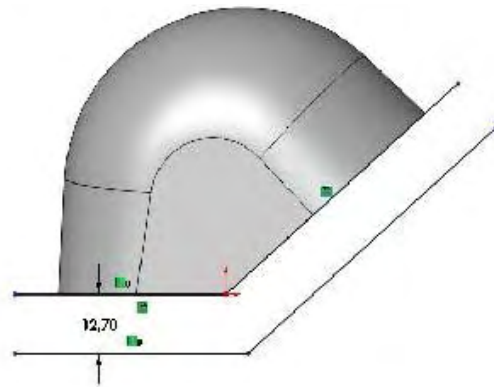


## 8. Offset Sketch Geometry

Click **Offset Entities** , and select one of the two converted edges.

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

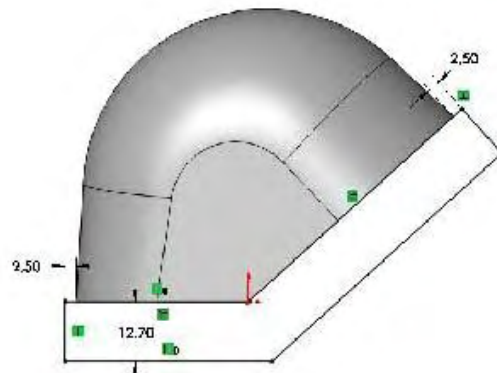
Click **OK**.



## 9. Dimensions

Add the dimensions of 2.5 as shown.

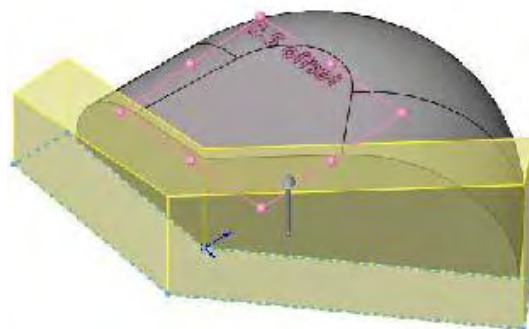
## 10. Exit the Sketch



## 11. Offset Plane

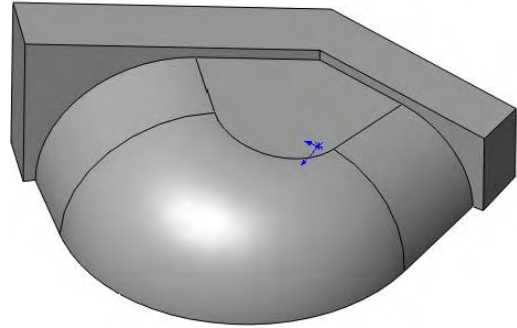
Create a plane that is offset **2.5 mm** from the **Up To** plane that was used for the base extrusion.

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



## 12. Extrude Up to Surface

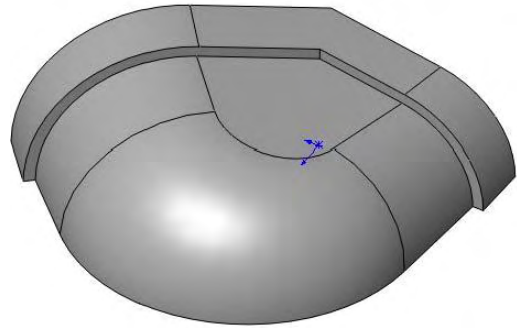
Extrude the sketch to the offset plane.



## 13. Fillet

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

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 three types of mirroring in SolidWorks:

1. **Mirror Feature:**  
Creates a copy of a feature (or multiple features), mirrored about a plane.
2. **Mirror Face:**  
Creates a copy of a face (or multiple faces), mirrored about a plane.
3. **Mirror Body:**  
Creates a copy of a solid body (or multiple features), mirrored about a plane.

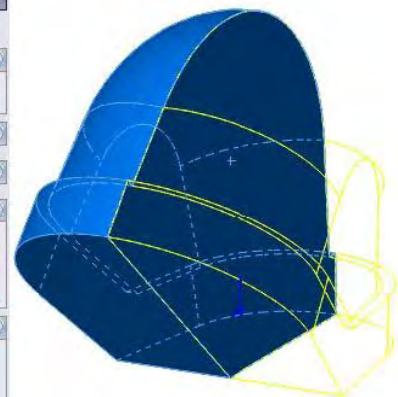
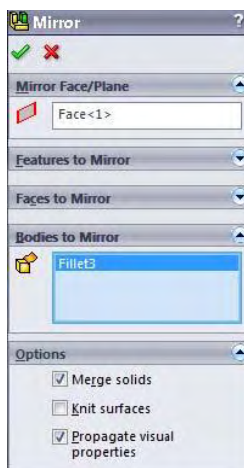
Since this part is symmetrical, we will mirror everything created so far by using **Mirror Body**.

## Where to Find It?

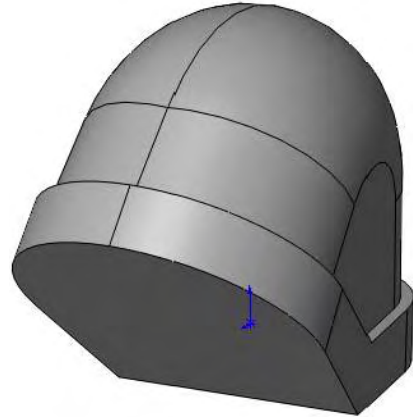
- On the **Insert** menu, click **Pattern/Mirror, Mirror...**

## 14. Mirror

From the **Insert** menu, choose **Pattern/Mirror, Mirror...** and select the planar face as the mirror face/plane. Make sure that the **Bodies to Mirror** box is active and select solid body in the graphics area. Click **OK**.

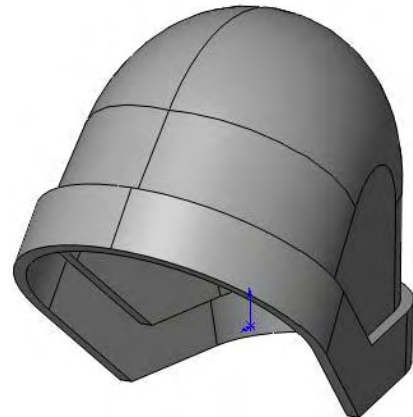


## 15. Result of Mirror



## 16. Shell

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



## 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 Surfaces: Modeling a Bottle

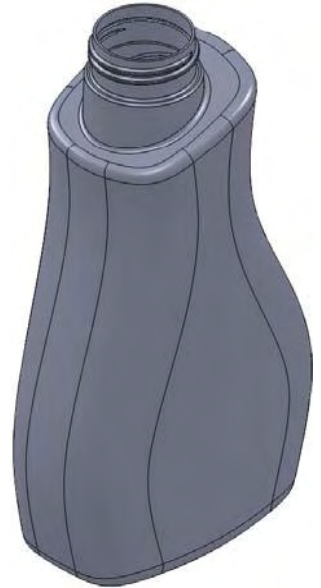
Upon successful completion of this lesson, you will be able to:

- Create lofted and extruded surfaces.
- Modify surfaces by trimming and extending.
- Create offset surfaces.
- Convert a surface model into a solid.
- Use surface intersections to create 3D curves.



# Working with Surfaces

There are a number of different situations in which 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 bottle with a shape that might be difficult to achieve using 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 always define a closed and correct volume. There are no 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. By their definition, the system knows what space lies “inside” the solid and what lies “outside”. Surface models lack that information. 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 key stages in the modeling process of this part are given in the following list:

- **Lofted surfaces**  
With the use of supplied sketches, the main lofted surfaces of the bottle are created.
- **Extruded surfaces**  
Two sides of the bottle have a single curvature. In one direction they are curved; in the other direction, in this case the vertical, they are straight. Single-curved surfaces can be created with the extrude feature.
- **Trimming surfaces**  
In surface modeling you often create faces that extend further than needed to define a closed volume. Surplus surface areas can be removed by trimming with other surfaces.
- **Face fillets**  
If major smooth transitions between surfaces are required, they must be added early in the design process. Here we will use the face fillet feature.
- **Planar surface**  
Planar surfaces are used to create a flat bottom and planar top plane for the bottle.
- **Make it solid**  
You can thicken a surface by adding material on either or on both sides. This creates a solid model.

## Surface Toolbar

The Surface toolbar contains shortcuts for all the surfacing commands. These commands can also be accessed via the **Insert, Surface** menu. To use the shortcuts on the Surfaces toolbar, right-mouse-click on any toolbar and select **Surfaces** .



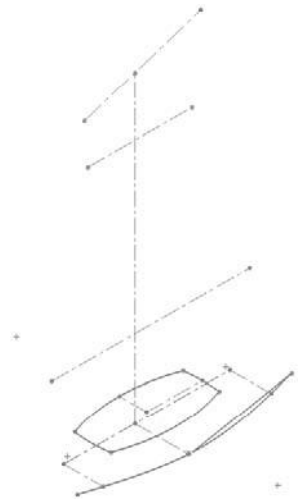


# Procedure

Begin by opening an existing part.

## 1. Open the Part Named Bottle

To save some time, this part already contains a number of sketches that will be used to create the surfaces and other features.



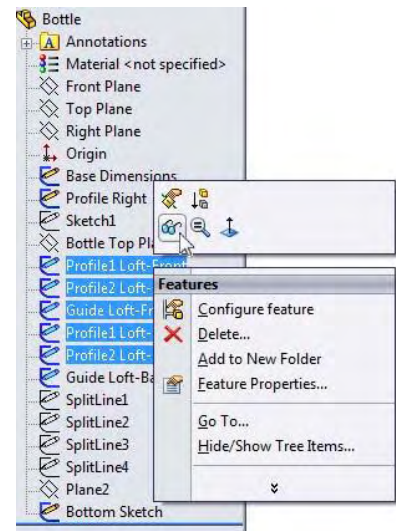
## 2. Hide and Show Sketches

Hide the following sketches.

- Base Dimensions
- Bottom Sketch

Show the following sketches:


- Profile1 Loft-Front
- Profile2 Loft-Front
- Guide Loft-Front
- Profile1 Loft-Back
- Profile2 Loft-Back
- Guide Loft-Front



## 3. Lofted Surface – Front Surface

Create the front surface for the bottle using a surface loft between two splines. Surface lofts include the same options as solid lofts. You can specify **Start/End Constraints**, use **Guide Curves**, and so on.

Click **Lofted Surface**

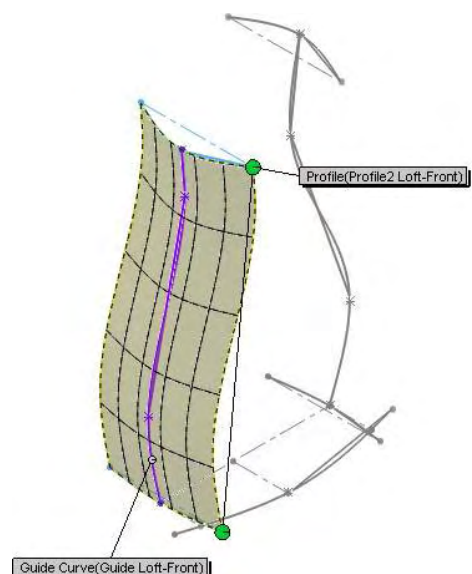
 on the Surfaces toolbar.

Select **Profile1 Loft-Front** and **Profile2**



**Loft-Front** for **Profiles** .

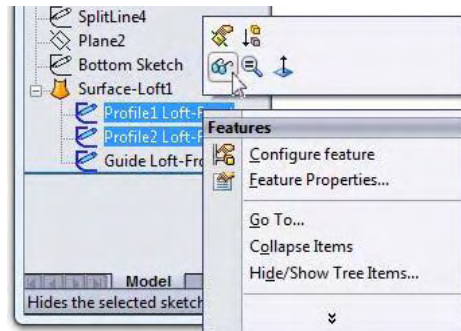
Select **Guide Loft-Front** for **Guide Curves** .

Click **OK** .




#### 4. Hide Sketches

The constructed feature appears at the bottom of the FeatureManager design tree. Expand **Surface-Loft1** by clicking on the plus sign  in the FeatureManager design tree. Select the three sketches contained in the feature, right click on the selection and select **Hide**  .



#### 5. Lofted Surface – Back Surface

Click **Lofted Surface**

 on the Surfaces toolbar or click **Insert, Surface, Loft...** from the menu.

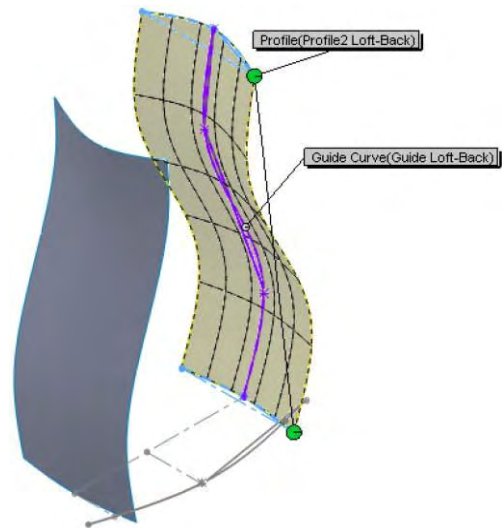
Select **Profile1 Loft-Front** and **Profile2**

**Loft-Front** for **Profiles** 

Select **Guide Loft-Front** for **Guide**

**Curves** 

Click **OK** 




#### 6. Result

Hide the sketches used for **Surface-Loft2**. The result is shown at right.



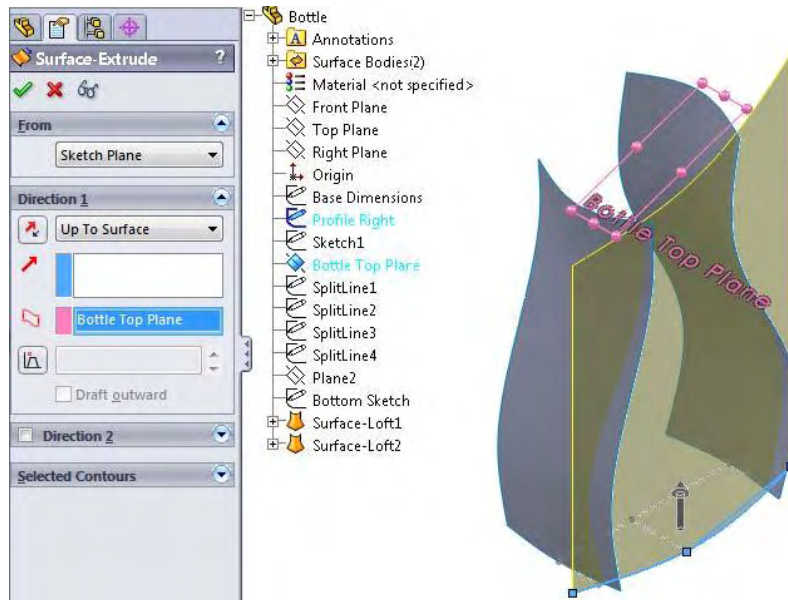
## 7. Extruded Surface – Right Surface

With the **Extruded Surface** tool, create the first surface of the right side of the bottle. The sides of the bottle are not created using lofted surfaces; instead, the **Extruded Surface** tool is used.

Select **Profile Right**. Click **Extruded Surface**  on the Surfaces toolbar.


Under **Direction 1**:

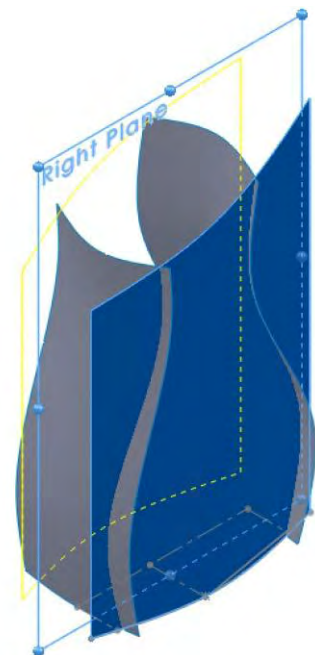
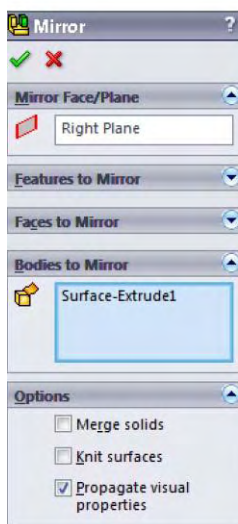
- For **End Condition** select **Up to Surface**.
- Set **Face/Plane**  to **Bottle Top Plane**. Select **Bottle Top Plane** by extending the Fly-out FeatureManager design tree.



Click 

## 8. Mirror

Use **Mirror** with the right reference plane to copy the extruded surface. Select the surface in the **Bodies to Mirror**  list.




# Trimming Surfaces

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.


## Introducing: Trim Surface

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 and you select the piece or pieces you want to *keep*.

### Where to Find It?


- Click **Insert, Surface, Trim**.
- Or, click the  tool on the Surfaces toolbar.


## 9. Mutual Trim

Click **Trim Surface**  on the Surfaces toolbar.

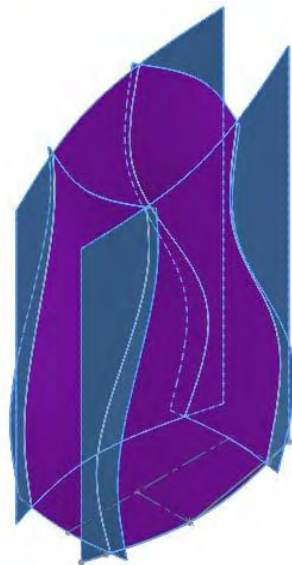
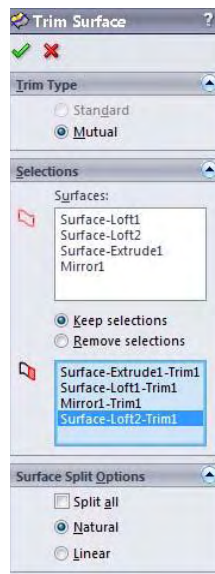
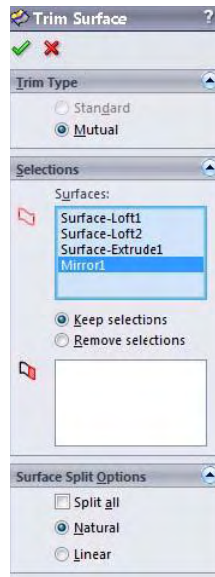
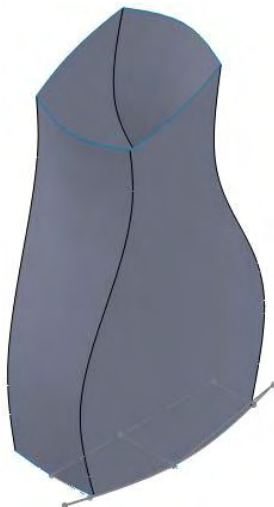
In the PropertyManager, under **Trim Type**, select **Mutual**.

Under **Selections**:

- For **Trimming Surface**  select **Surface-Loft1**, **Surface-Loft2**, **Surface-Extrude1** and **Mirror1**, in the graphics area.
- Select **Keep selections**.

For **Pieces to Keep**  Select the four surfaces as shown (in purple) in the figure at right.

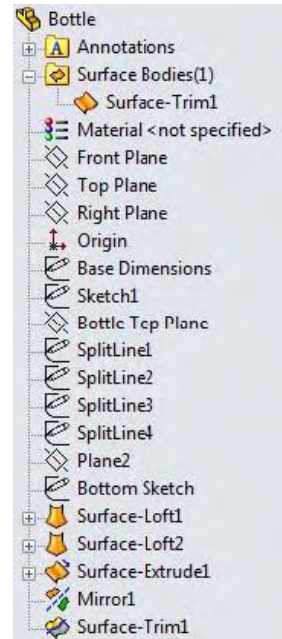
Click **OK** .





## FeatureManager Design Tree

The trimming operation created a single trimmed surface, not four individual surfaces. The system automatically joined, or knit the individual faces into a single feature: **Surface-Trim1**. Certain types of surfaces features – trimmed and knit surfaces particularly – are treated somewhat differently in the FeatureManager design tree than solid features. Even though the four individual surfaces were used to create the trimmed surface, they are not absorbed into it, the way, for instance, the profile sketches (**Profile1 Loft-Front** and **Profile2 Loft-Front**) were absorbed into **Surface-Loft1**.



### 10. Save the Model



Now is a good time to save your work.

### 11. Hide and Show Sketches

Hide the sketch used for **Surface-Extrude1**. Show the sketches **SplitLine1** through **SplitLine4**.



### 12. Split Line – Front and Back Right

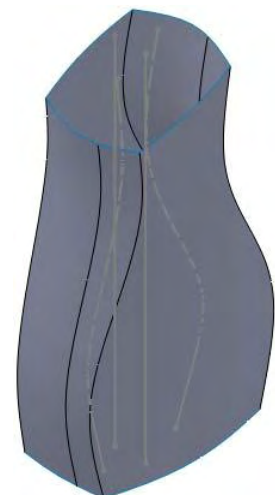
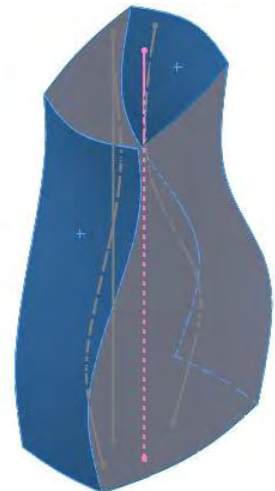
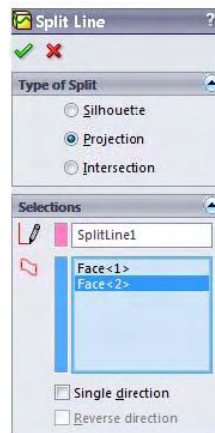
Click **Split Line**

 on the Curves toolbar, or click **Insert, Curve, Split Line**. To use the shortcuts on the Curves toolbar, right-mouse-click on any toolbar and select **Curves** .

In the PropertyManager, under **Type of Split**, select **Projection**.

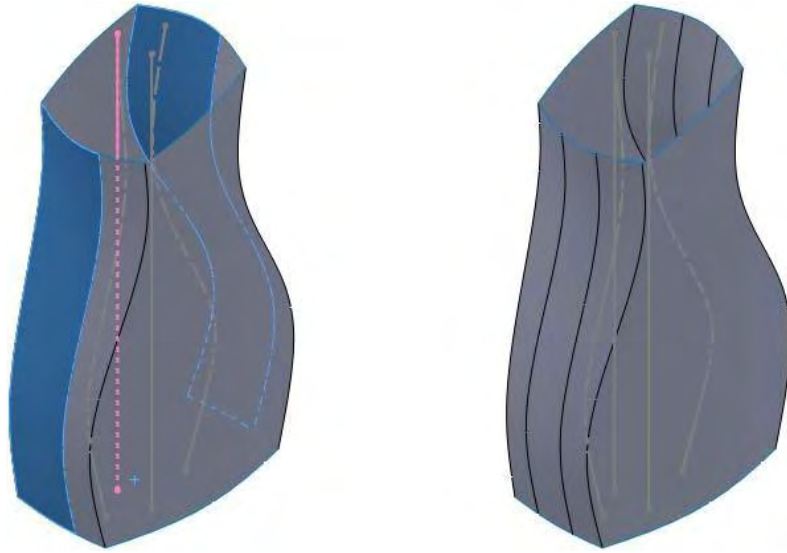
Under **Selections**:

- For **Sketch to Project**  select **SplitLine1**.
- For **Faces to Split**  select the front and back face as indicated.




### 13. Split Line – Front and Back Left



Using the same method as in the previous step, you can split the largest front and corresponding back face, as indicated below.



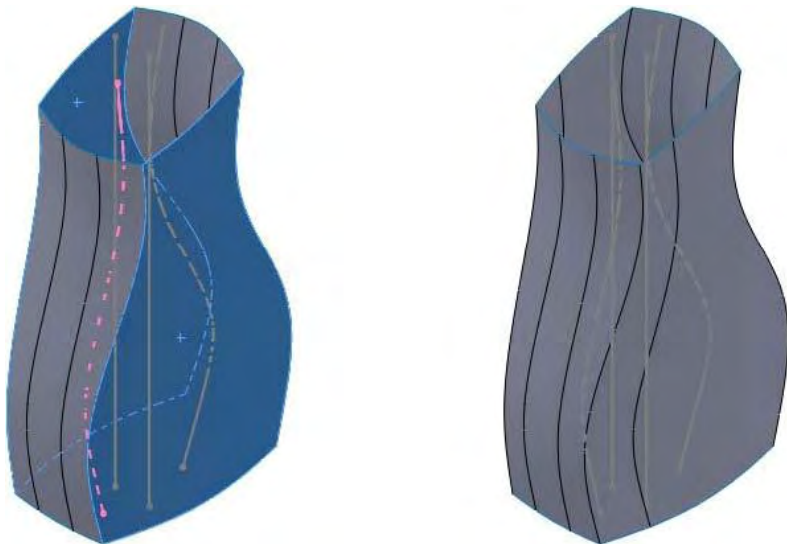
### 14. Split line – Left and Right Side

Click **Split Line**  on the Curves toolbar, or click **Insert, Curve, Split Line**. In the PropertyManager, under **Type of Split**, select **Projection**.

Under **Selections**:

- For **Sketch to Project**  select **SplitLine3**.
- For **Faces to Split**  select the left and right side faces as indicated.

Click **OK** .



For a clearer view of the model, hide the three sketches used for creating the split lines.



## 15. Face Fillet

The front side of the bottle is smoothly connected to the left and right side by a fillet. Because this fillet is an important part of the total shape of the bottle's body, the fillet is made early in the modeling process.


Click **Fillet**  or **Insert, Surface, Fillet/Round...**

Under **Fillet Type** select **Face Fillet**.

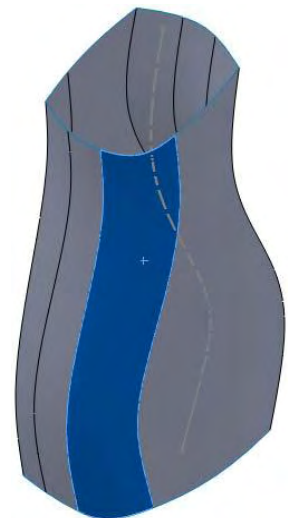
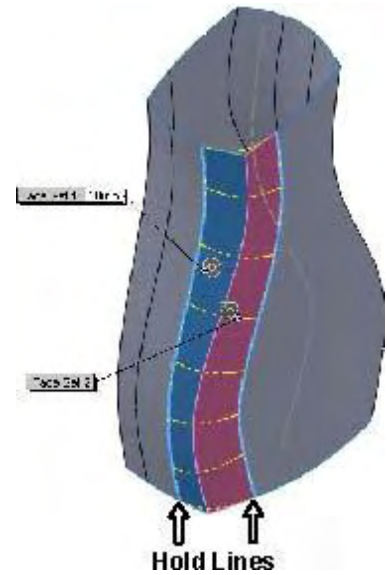
Under **Items to Fillet**, select the surfaces adjacent to the corner edge:

- In **Face Set 1** select the right of the front faces.
- In **Face Set 2** select the left face of the right side.

A fillet of 10 mm appears. You want to maximize the fillet. Extend the **Fillet Options** box, click in **Hold Line Edges** and select the two hold lines as marked in the figure. Finally, select **Curvature Continuous**. The correct fillet appears.

**Note** Make sure both surface normals (the grey arrows) point inward. It might be necessary to click the **Reverse Face Normal**  to ensure the fillet can be made.

Click **OK** .



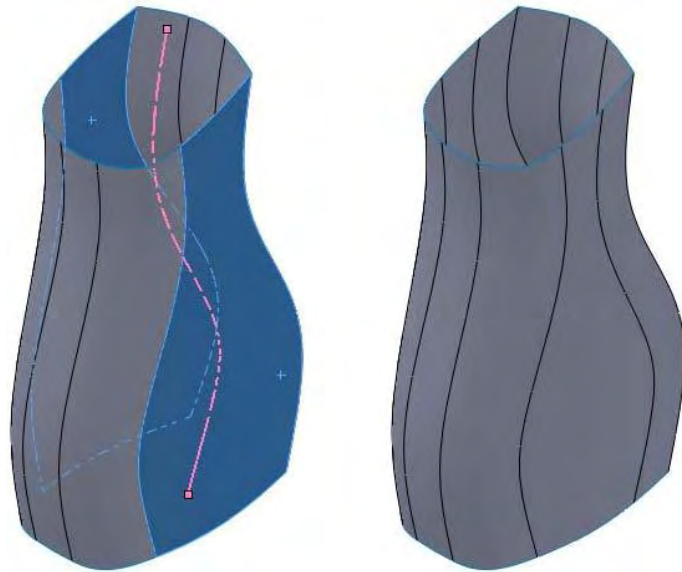
## 16. Another Face Fillet

Fillet the left side of the bottle the same way. The figures at right show the selections and the result.



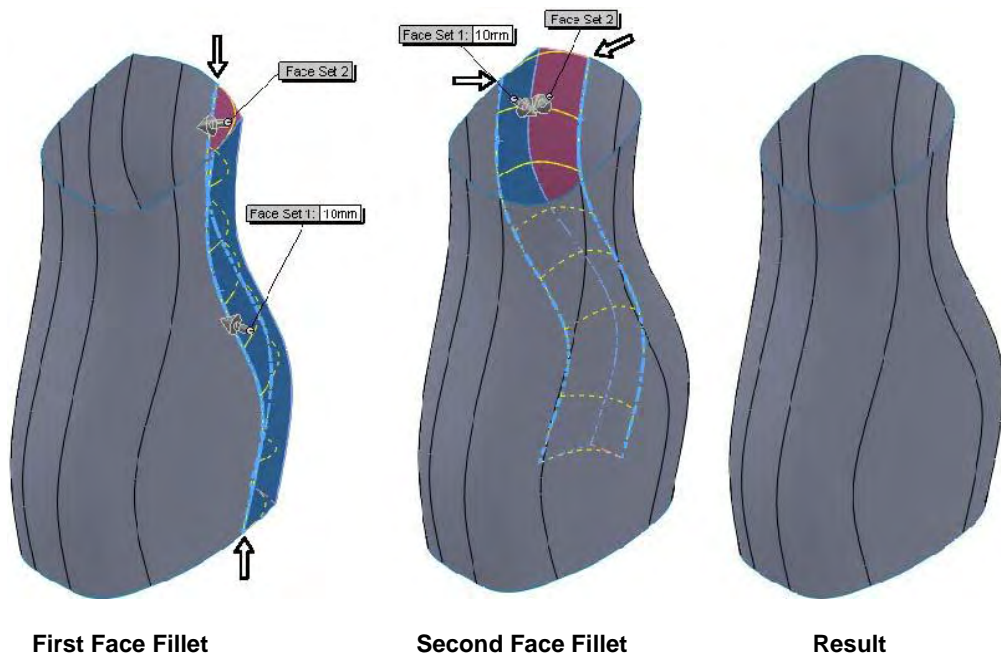
## 17. Split Line – Left and Right Side

Split the left and right face of the bottle using **SplitLine4**, as show to the right.



## 18. Two Face Fillets


Following the instruction of steps 15 and 16, create two more face fillets at the back of the bottle using the faces and hold lines as indicated in the figures below.



# Planar Surface

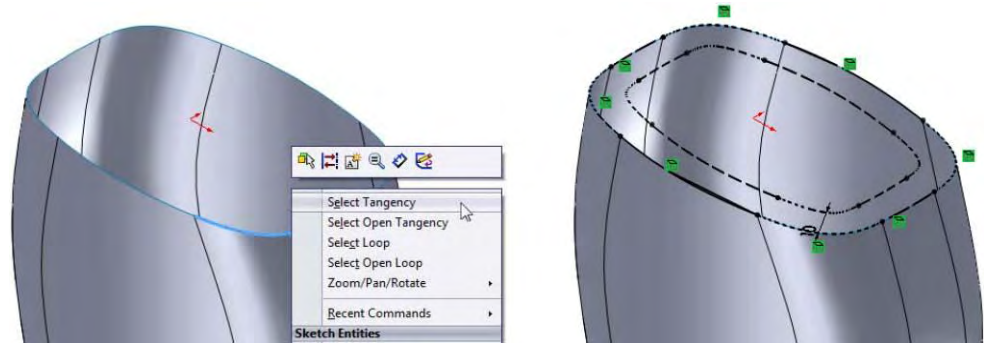
You can create a planar surface from either a closed, single contour, or non-intersecting sketch, or a closed set of planar edges.

## Where to Find It?

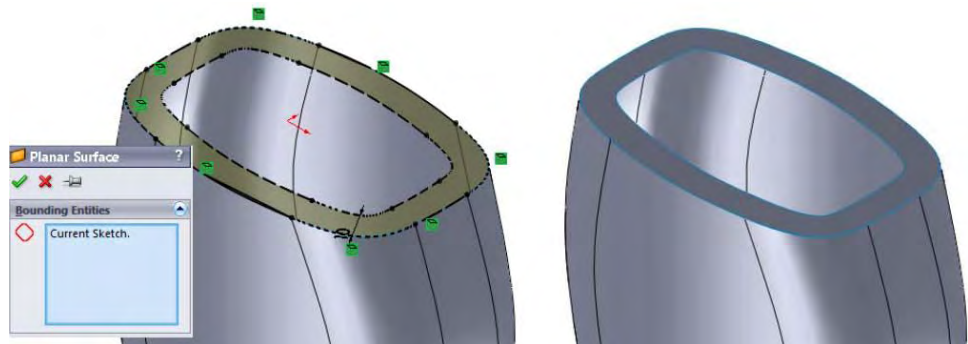
- Click **Insert, Surface, Planar**.
- Or, click  on the Surfaces toolbar.

## 19. Sketch the Surface Contours

Open a sketch on the standard reference plane **Top Plane** (actually the bottom of the bottle).

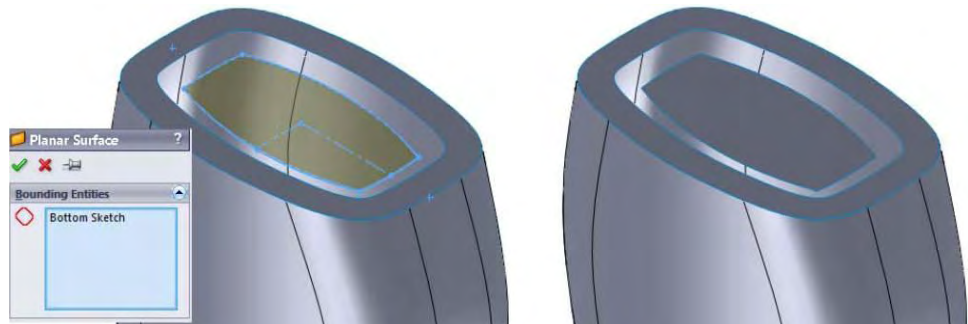


## 20. Planar Surface




## 21. Another Planar Surface

Create the recessed part of the bottom using the **Bottom Sketch**.



## 22. Filled Surface


Fill the gap between the two parts of the bottom using the **Filled Surface** tool.

Click **Filled Surface**  or **Insert, Surface, Fill...**

Under **Edge settings**:

- Select **Show Preview** and **Preview Mesh**.
- Deselect **Apply to all edges**.


Set **Curvature Control** to **Tangent**. Make sure the **Patch Boundaries**

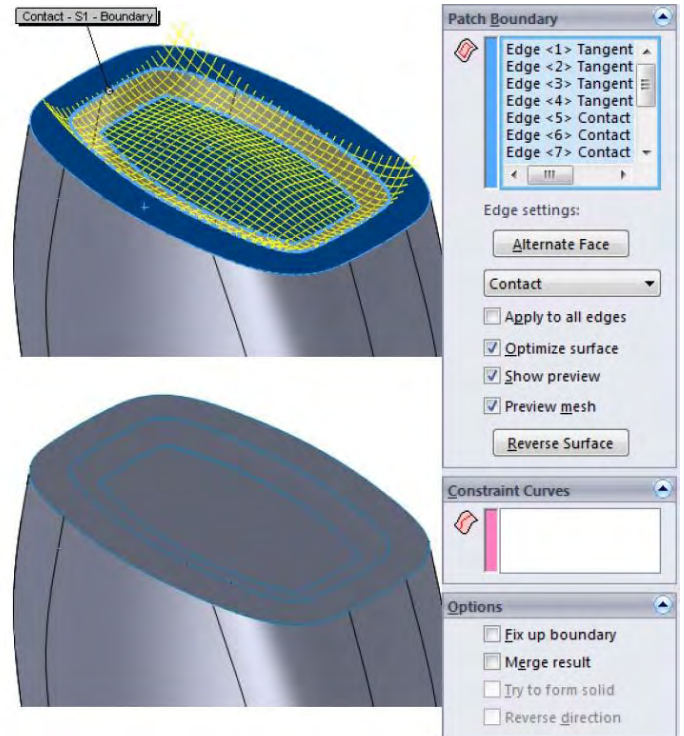
 field is active and select an edge of the inside planar surface (Surface-Plane2) by right-clicking and choosing **Select Open Loop**.

If a warning is displayed, click **OK**.

Set the **Curvature Control** to **Contact**. Select an edge on the inside of the bottom's bottom surface (Plane2) by right-clicking and choosing **Select Open Loop**.

If a warning is displayed again, click **OK**.

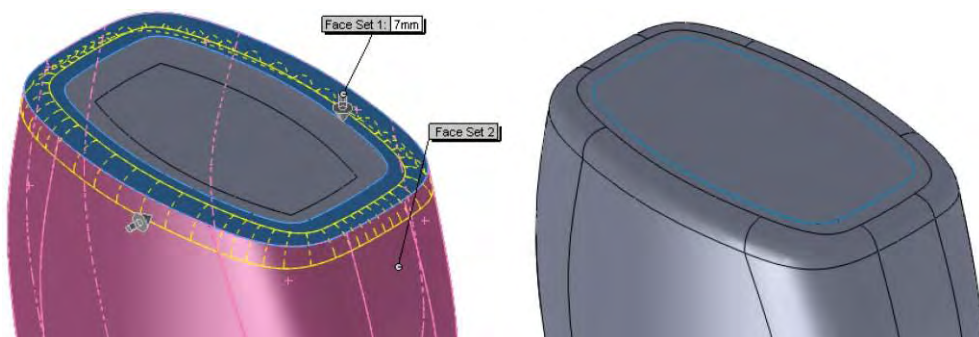
Create the face by clicking **OK** .



## 23. Fillet the Bottom

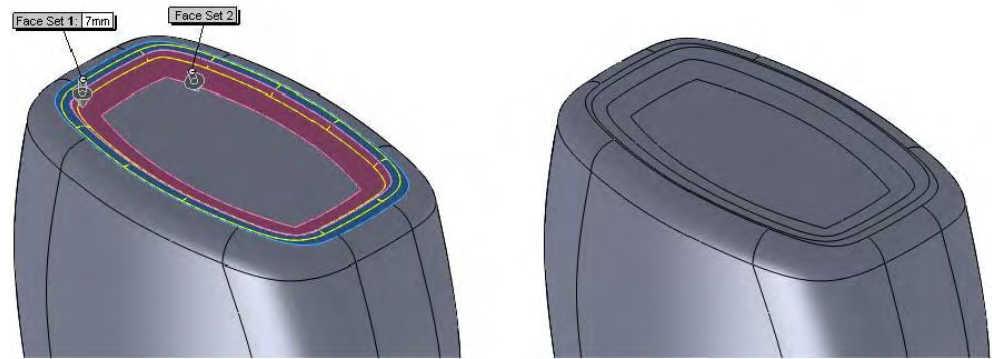
Click **Fillet**  or **Insert, Surface, Fillet/Round...**

Add a **7-mm Face Fillet** between the two faces shown below.





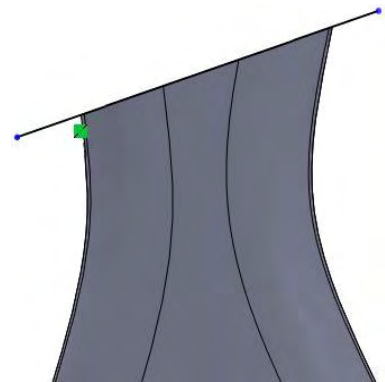
Add another **7-mm Face Fillet** between the two faces shown below.




## 24. Create Sketch

Open a sketch on the **Right** reference plane and sketch a single line segment. Add a **Collinear** relation between the line segment and the top center line in the **Base Dimensions** sketch (you may need to make the **Base Dimensions** sketch visible before you can add this relation).


Make sure that the line extends beyond the bottle's contours as shown in the figure to the right. This line will be extruded to create the top surface of the bottle's body.

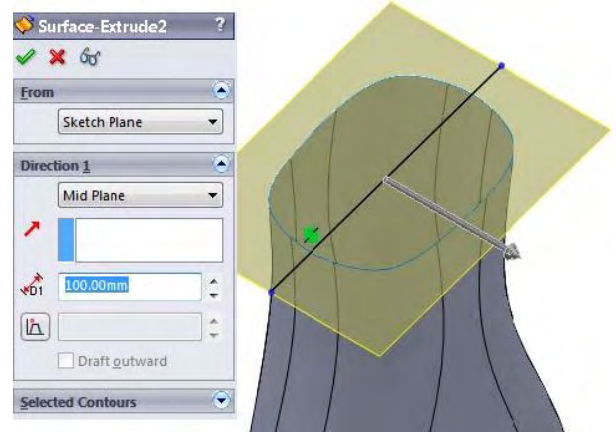


## 25. Extrude Surface

Click **Extruded Surface**  on the Surfaces toolbar

Under **Direction 1**:


- For **End Condition** select **Up to Surface**.
- Set **Face/Plane**  to **Bottle Top Plane**. Select **Bottle Top Plane** by extending the Fly-out FeatureManager design tree.



Click **OK** .



## 26. Surface Trim

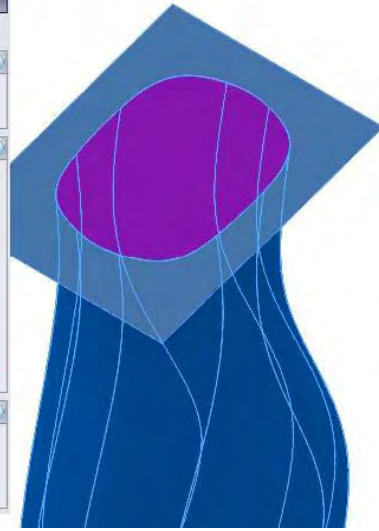
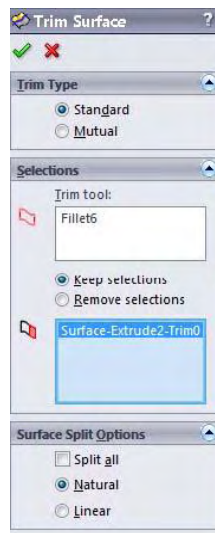
Now we remove the excess surface from the bottle's top.

Click **Trim Surface**  on the Surfaces toolbar, or **Insert, Surface, Trim....**

In the PropertyManager, under **Trim Type**, select **Standard**.

Under **Selections**:

- For **Trim tool**  select bottle's body (**Fillet6**) in the graphics area.
- Select **Keep selections**.
- For **Pieces to Keep**  select the portion we want to keep of the surface we created in the previous step (shown in purple in the figure on the right).

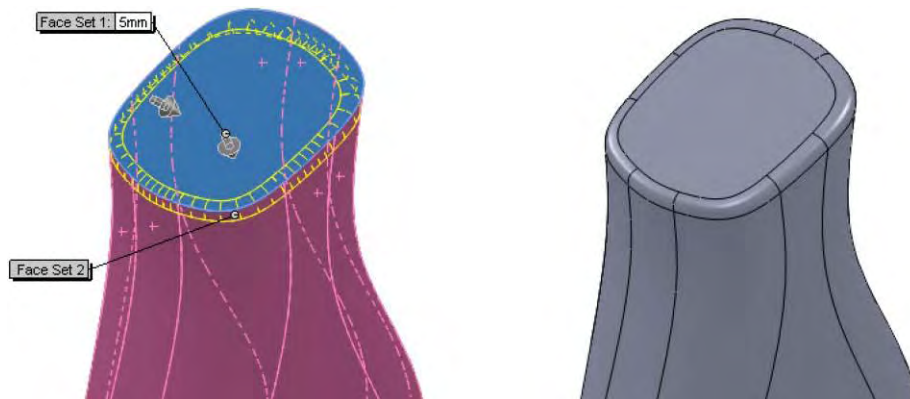


Click **OK** .

## 27. Face Fillet

Click **Fillet**  or **Insert, Surface, Fillet/Round....**


Add a **5-mm Face Fillet** between the bottle's body and the newly created top face of the bottle.



## Introducing: Revolved Surface

Creating a revolved surface is exactly like creating a revolved boss or cut feature. You have to have a centerline to define the axis of revolution, and you can specify the angle of revolution.

### Where to Find It?

- Click  on the Surfaces toolbar.
- Or, click **Insert, Surface, Revolve....**

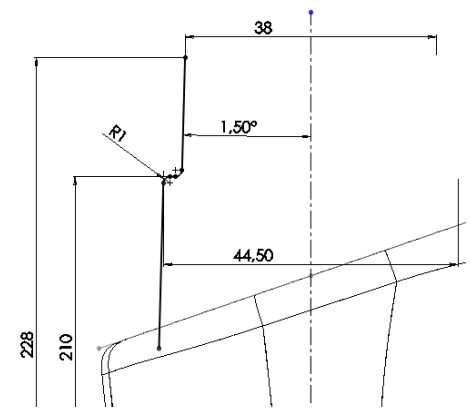
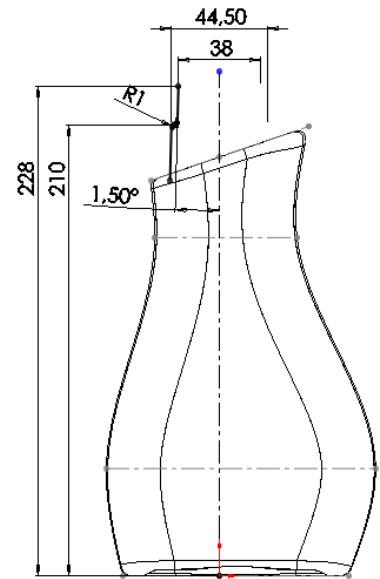


## 28. Sketch the Profile


Open a sketch on the **Right** reference plane and create the open line contour shown below. This contour will be revolved to form the bottle's neck.

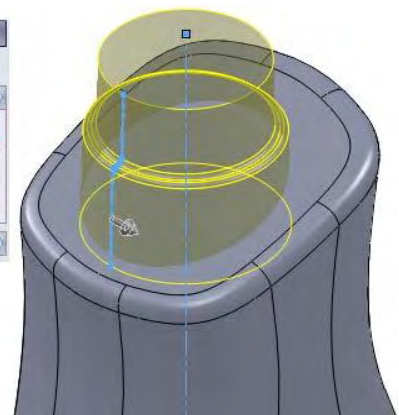
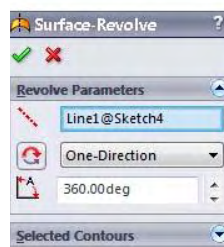
While sketching the contour, pay attention to the following:

- The two 'vertical' lines have a **Parallel** relation.
- The bottom end is **Coincident** with the left endpoint of the top center line in the **Base Dimensions** sketch.
- The upper end is at distance **228 mm** from the **Top** reference plane.
- The middle section is at distance **210 mm** from the **Top** reference plane.




## 29. Revolve Surface

Click , or click **Insert, Surface, Revolve....** Set the **Angle** to **360°**, and click **OK**.





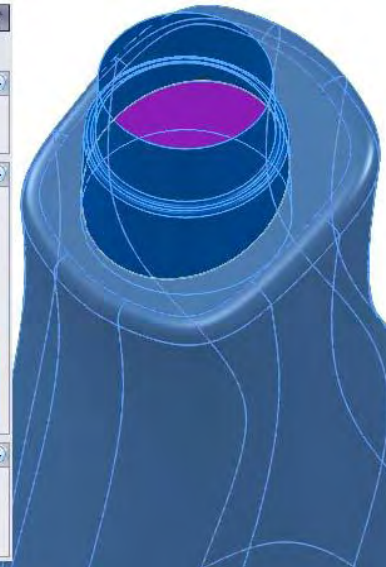
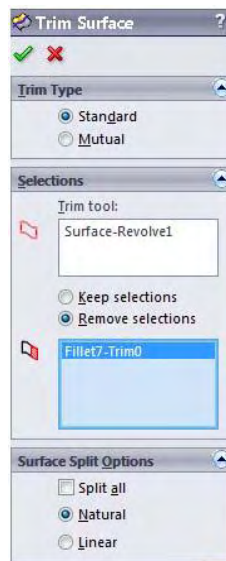
### 30. Surface Trim

Click **Trim Surface**  on the Surfaces toolbar, or **Insert, Surface, Trim....**

In the PropertyManager, under **Trim Type**, select **Standard**.

Under **Selections**:

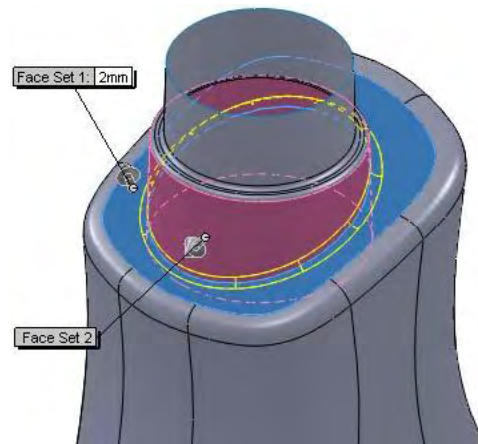
- For **Trim tool**  select the revolve surface from the previous step.
- Select **Remove selections**.
- For **Pieces to Remove**  select the surface inside the revolve surface (shown in purple).



Click **OK** .

### 31. Face Fillet

Connects the two surface bodies by adding a **2-mm Face Fillet** between them.




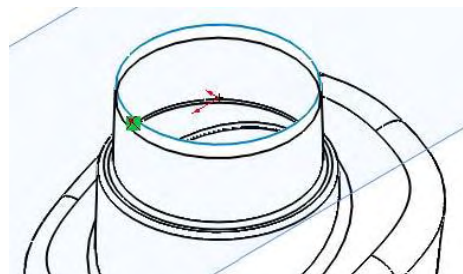
### 32. Offset Planes

Create a reference plane offset **225 mm** from the **Top** reference plane.


### 33. Insert Sketch

With the top plane selected, open a new sketch. Because a draft is applied to the bottle's neck, we cannot convert the top edge of the neck to obtain a circle. Instead, we will use an **Intersection Curve** to realize this circle.

Click **Intersection Curve** , choose **Tools, Sketch Tools, Intersection Curve**. Select the bottle's neck. A circle intersecting the neck will be created. Click **Intersection Curve** again to disable the **Intersection Curve** tool.



### 34. Create the Helix

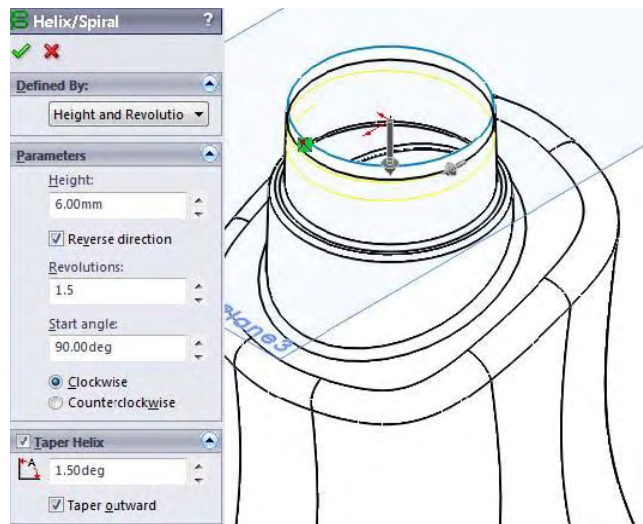
Click  **Helix and Spiral**, or choose **Insert, Curve, Helix/Spiral...**


For the **Defined By** type, choose **Height and Revolution**.

In the **Parameters** section set the following:

- Height: 6 mm.**
- Revolutions: 1.5.**
- Start angle: 90°.**

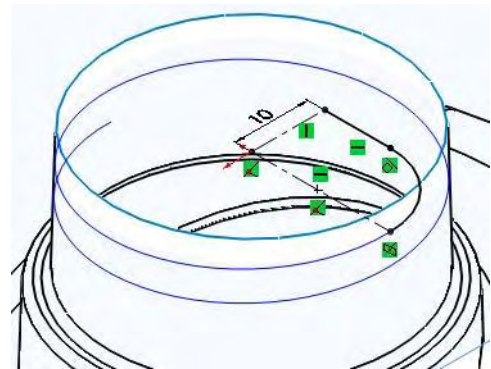
The threads are **Clockwise** and go down the neck (select **Reverse direction** if necessary).



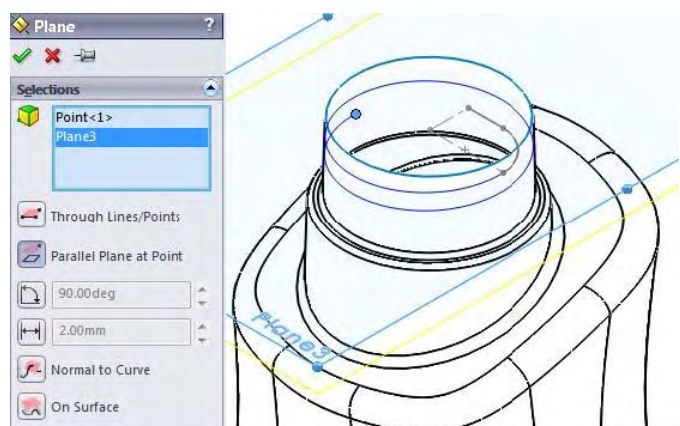
Click **OK**  to create the helix.

### 35. Lead In

Create lead in.

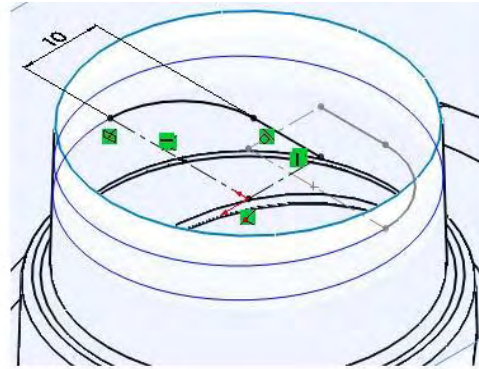


### 36. Parallel Plane at Point

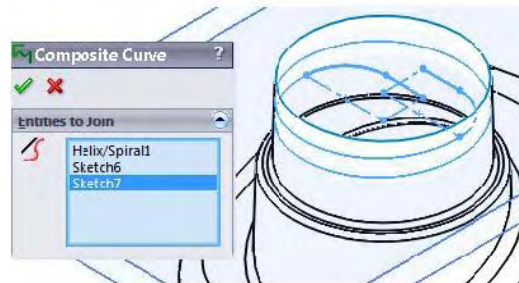


### 37. Lead In

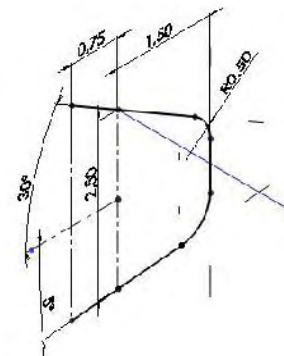
Create lead in.



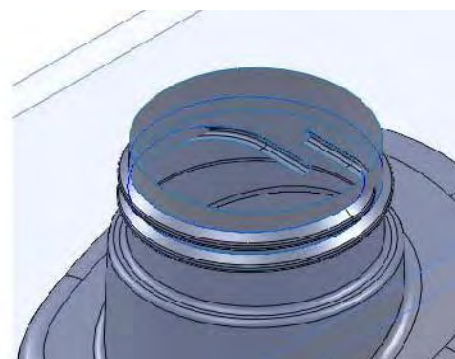
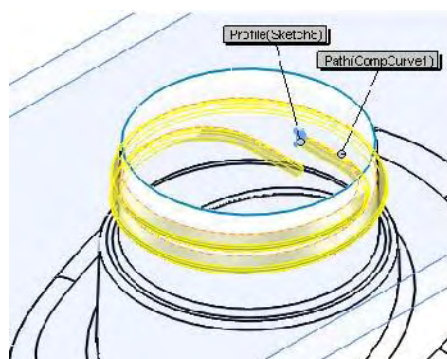
### 38. Composite Curve



### 39. Sketch the Thread Profile




### 40. Create the Thread







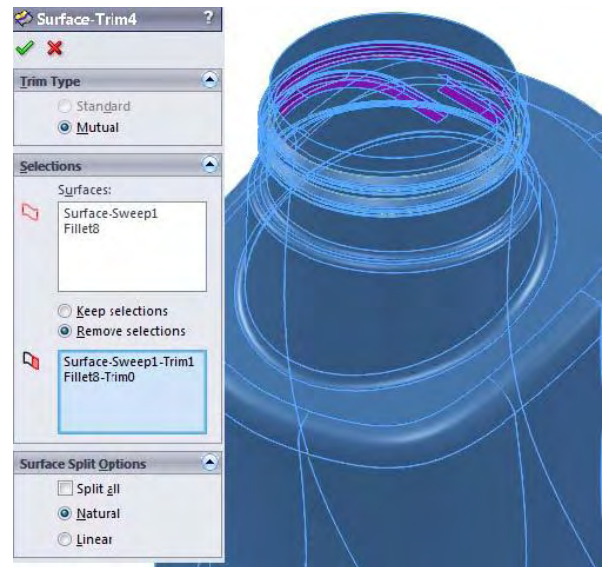
## 41. Mutual Trim

Click **Trim Surface**  on the Surfaces toolbar, or **Insert, Surface, Trim...**

In the PropertyManager, under **Trim Type**, select **Mutual**.

Under **Selections**:

- For **Trimming Surfaces**  select the surface sweep (Surface-Sweep1) and the bottle's main body (Fillet8).
- Select **Remove selections**.
- For **Pieces to Remove**  select the thread ends and the inside of the thread as shown in purple on the right.



Click **OK** .

## 42. Results



## Making it Solid

As with 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 base 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 can be created as either a boss or a cut feature.




### Where to Find It?

- Click **Insert, Boss/Base, Thicken...**, or **Insert, Cut, Thicken**.

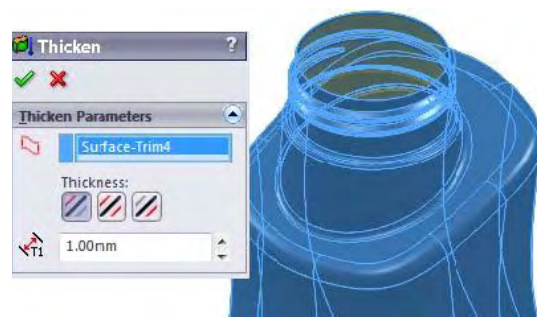
### 43. Thicken Feature

Click **Thicken**  on the Features tab.

In the PropertyManager, under **Thicken Parameters**:

- Select Surface-Trim4 for **Surface to Thicken** .
- Click **Thicken Side 1** .
- Set **Thickness**  to 1 mm.

Click **OK** .



**Note** Thicken features can take a lot of computational effort and may take a while (up to several minutes) before the result becomes visible. The difficulty is with the fillets created in the model. SolidWorks offsets every surface and then fills the space between the existing and offset surfaces. If the model has a fillet of 0.5 mm, an offset of 0.6 mm to the inside of the fillet cannot be constructed. Fillets with a radius smaller than the offset distance have to be made after the Thicken.

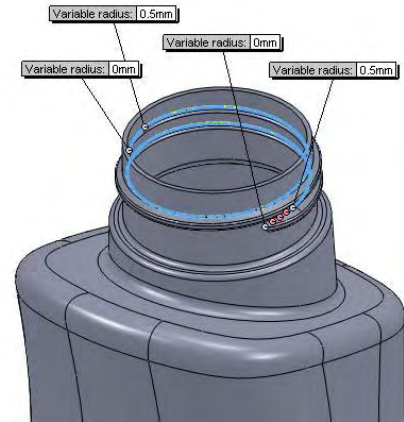
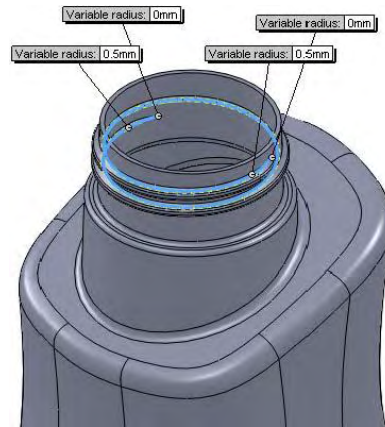
### 44. Section View



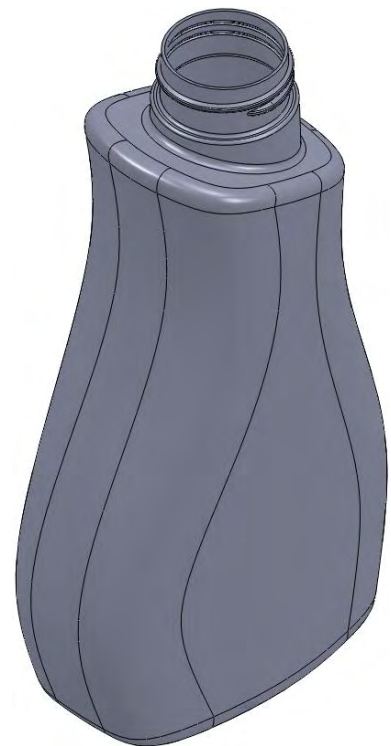
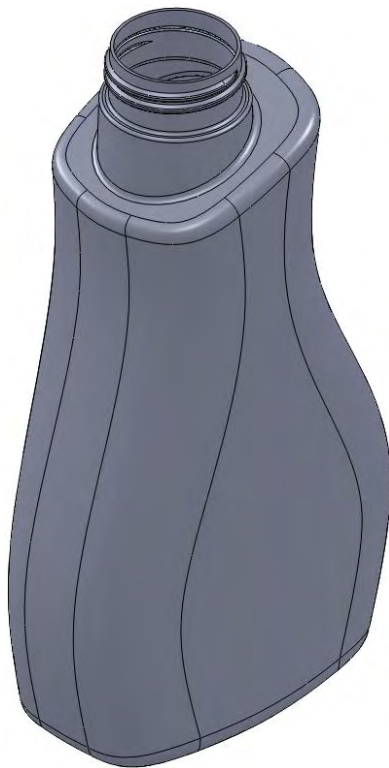


## 45. Variable Radius Fillet

Round of the inside edges of the bottle's thread



## 46. Finished Model



**47. Rendered with PhotoWorks**

