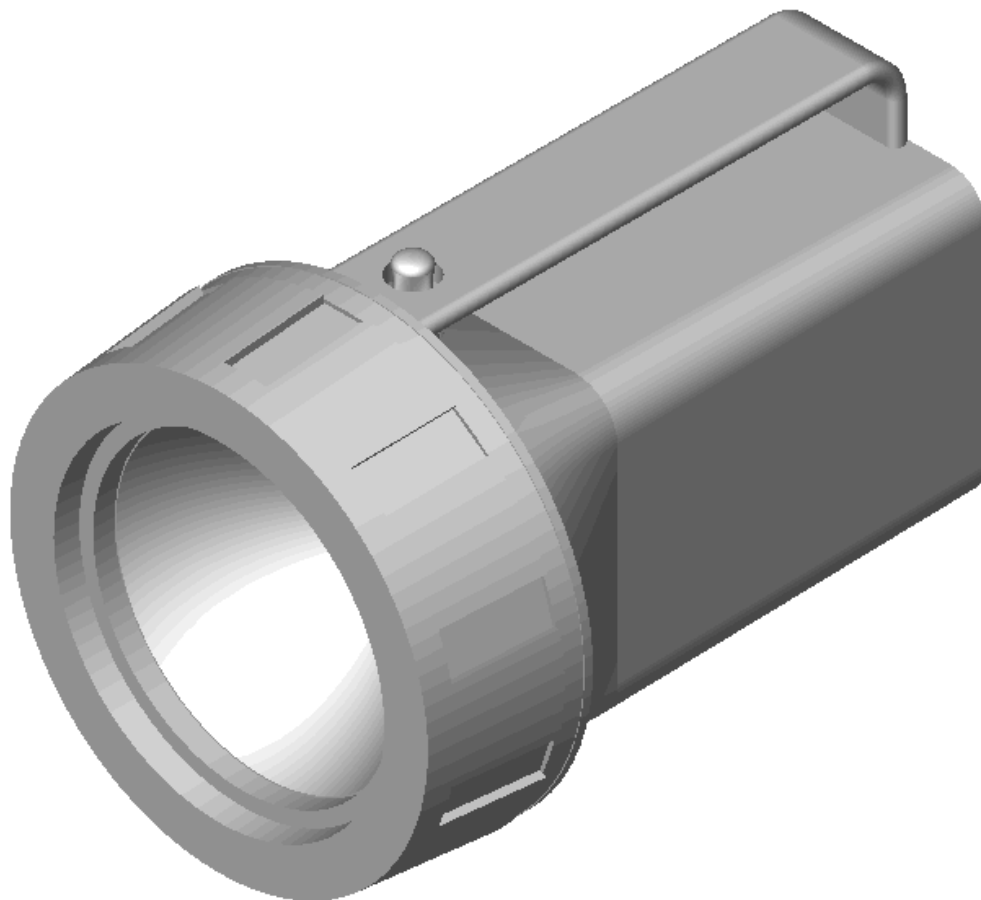

SolidWorks 2001 Tutorial

A Basic Introduction

Marie P. Planchard & David C. Planchard



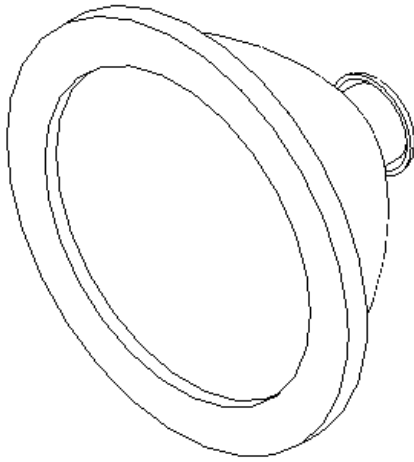
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Project 2

Revolve Features



Below are the desired outcomes and usage competencies based upon the completion of Project 2.

Project Desired Outcomes	Usage Competencies
A comprehensive understanding of the customer's design requirements and desires.	To comprehend the fundamental definitions and process of Feature-Based 3D Solid Modeling.
Two key flashlight components: <ul style="list-style-type: none"> • LENS • BULB 	Specific knowledge of Revolve base features.
	Understanding of the Shell feature, Hole Wizard, Dome feature and Circular Pattern.
	Ability to apply Extrude and Fillet features.

Notes:



Project 2 – Revolve Features

Project Objective

Create two components of the flashlight. Create the LENS and BULB.

Project Situation

The LENS is a purchase part utilized in the FLASHLIGHT assembly, Figure 2.1. Obtain dimensional information on the LENS. Review the size, material and construction.

The BULB is a purchased part, Figure 2.2. The BULB is a replacement part and requires a separate part number and order number.



Figure 2.1



Figure 2.2

Project Overview

Create two parts in this section:

- LENS
- BULB

The LENS and the BULB utilize a Revolve Base feature.

LENS

Determine the key features of the LENS. The Base feature for the LENS is a solid Revolved feature. A solid Revolved feature adds material. The Revolved Base feature is the foundation for the LENS.

A Revolved feature is geometry created by rotating a sketched profile around a centerline, Figure 2.3. Close the Sketch profile for a solid Revolved feature, Figure 2.4. Do not cross the centerline.

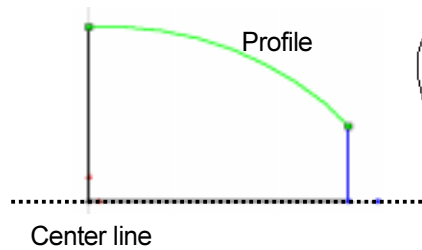


Figure 2.3



Figure 2.4

LENS Feature Overview

Create the LENS. Use the solid Revolved Base feature, Figure 2.5.

Create uniform wall thickness. Create the Shell feature, Figure 2.6.

Create an Extruded-Boss feature from the back of the LENS, Figure 2.7.

Create a Thin-Revolved feature to connect the LENS to the BATTERYPLATE, Figure 2.8.



Figure 2.5



Figure 2.6



Figure 2.7



Figure 2.8

Create a Counterbore Hole feature with the HoleWizard, Figure 2.9. The BULB is located inside the Counterbore Hole.

Create the front LensFlange feature. Add a transparent LensShield feature, Figure 2.10.

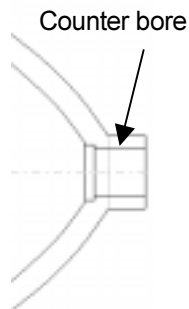


Figure 2.9




Figure 2.10

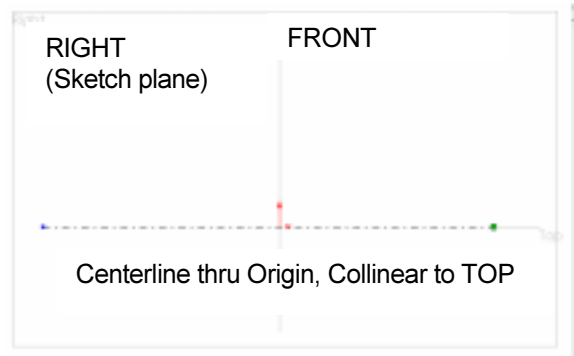
Create the LENS




Create the LENS with a Revolved Base feature. The solid Revolved Base feature requires a sketched profile and a centerline. The profile is located on the Right plane with the centerline collinear to the Top plane. The profile lines reference the Top and Front planes. The curve of the LENS is created with a 3-point arc.


Create the LENS.

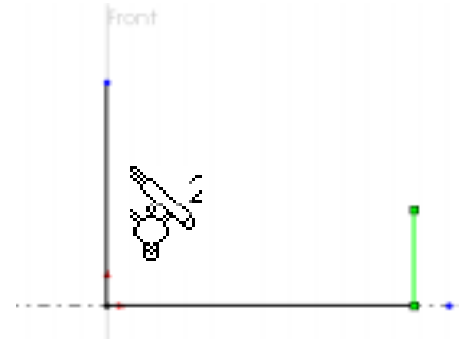
- 1) Click **New** . Click **PartEnglishTemplate**. Click **OK**. Click **Save** . Enter **LENS**. Click the **Save** button.
- 2) View the planes. Right-click on the **Front** plane in the FeatureManager. Click **Show**. Right-click **Top** plane in the FeatureManager. Click **Show**.

- 3) Select the Sketch plane. Click the **Right** plane. Display the view. Click **Right** .



- 4) Sketch the centerline. Click **Sketch** . Click **Centerline** . Sketch a horizontal **centerline** collinear to the Top plane, through the Origin .

- 5) Sketch the profile. Create three lines. Click **Line** . Create the first line. Sketch a **vertical line** collinear to the Front plane coincident with the Origin. Create the second line. Sketch a **horizontal line** coincident with the Top plane. Create the third line. Sketch a **vertical line** approximately 1/3 the length of the first line.



Create an arc. Determine the curvature of the LENS.

A 3 POINT Arc requires three points:


- Start point
- End point
- Center point

The arc midpoint is aligned with the center point. The arc position is determined by dragging the arc midpoint or center point above or below the arc.

On-line help contains an animation file to create a 3-point arc. Click **Help, Index, Arc,**

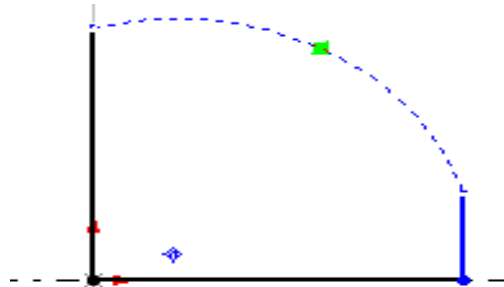
3Point. Run the animation. Click the **AVI** icon  **Show Me**.

6) Create a 3 Point Arc. Click **3Pt Arc**


 Create the arc start point. Click the **top point** on the left vertical line. Hold the **left mouse button** down. Drag the mouse pointer to the **top point** on the right vertical line.

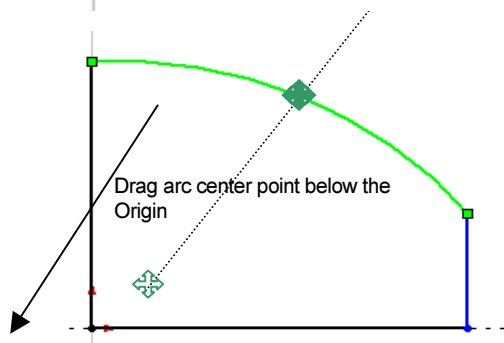
Create the arc end point. Release the **mouse button**.


Click and drag the **arc center point** below the Origin. Release the **left mouse button**.



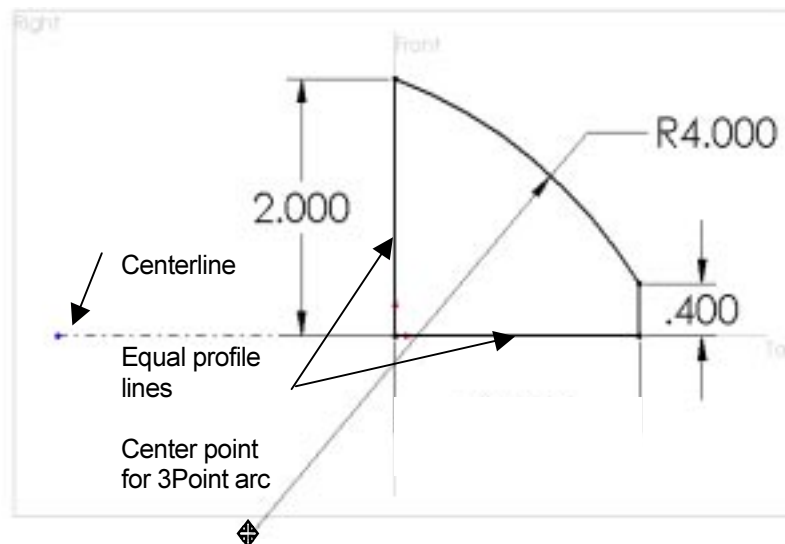
7) Add geometric relationships. Click **Add**

Relations . The arc is currently selected. Click the **arc** to remove it from the Select Entities text box. Create an equal relationship. Click the **left vertical line**. Click the **horizontal line**. Click the **Equal** button. Click **Apply**. Click **Close**.




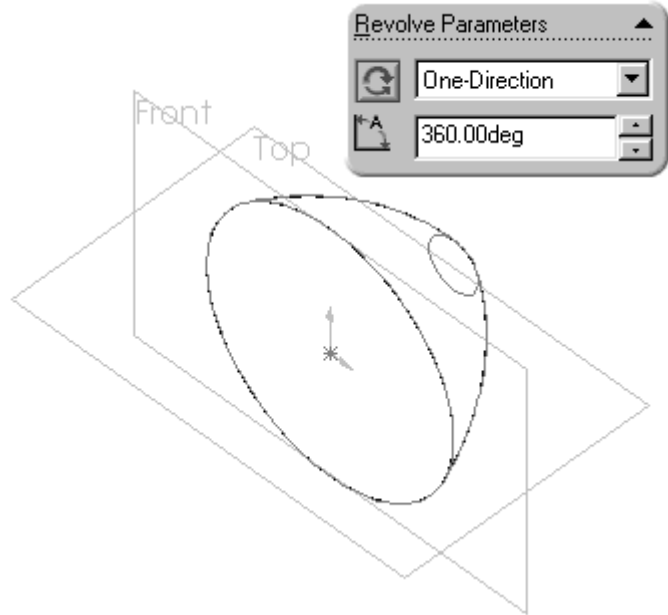
8) Add dimensions. Click **Dimension** 

Create a vertical linear dimension for the **left line**. Enter **2.000**. Create a vertical linear dimension for the **right line**. Enter **0.400**. Create a radial dimension for the **arc**. Enter **4.000**. The black Sketch is fully defined.



- 9) Revolve the Sketch.

Click **Revolve**  from the Feature toolbar. The Revolve Feature dialog box is displayed. Accept the default option values. Create the solid Revolve feature. Click **OK**.



- 10) Save the LENS. Click

Save .

Revolve features contain an axis of revolution. The axis is critical to align other features.


- 11) Display the axis of revolution. Click **View** from the Main menu. Click **Temporary Axis**. A check mark is displayed next to the option. Hide the Temporary axis. Click **Temporary Axis** to remove the check mark. Hide the Planes. Click **Planes** to remove the check mark.

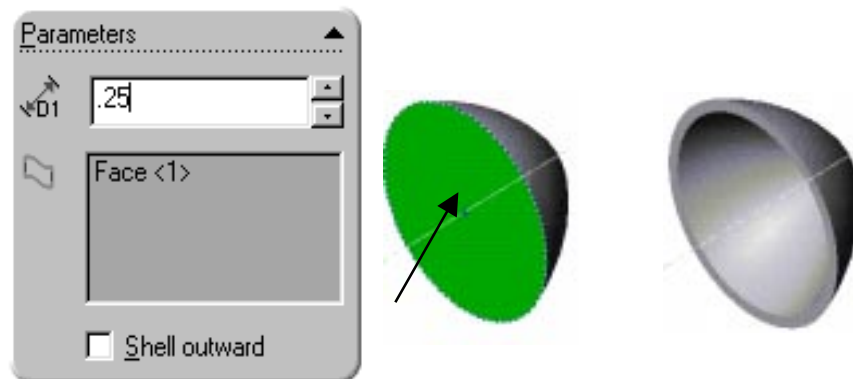
Solid Revolve features must contain a closed profile. Each revolved profile requires an individual sketched centerline.


Create the LENS - Shell Feature

The Shell feature removes face material from a solid. The Shell feature requires a face and thickness. Use the Shell feature to create thin-walled parts.

Create the Shell feature.

- 12) Select the face. Click the **front face** of the Base-Revolve feature. Click **Shell**  from the Feature toolbar. Enter **0.250** in the Thickness text box. Display the Shell feature. Click **OK**.




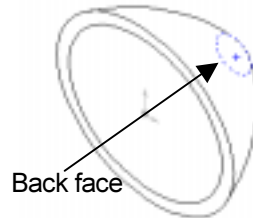
13) Rename **Shell1** to **LensShell**. Save the LENS. Click **Save** .

Create the LENS - Extruded Boss Feature



Create the LensNeck. Use the Extruded-Boss feature. The LensNeck houses the BULB base and is connected to the BATTERY PLATE. The feature extracts the back circular edge from the Base-Revolve feature. The Convert Entities sketch tool creates the sketch by projecting an existing edge onto the current sketch plane.

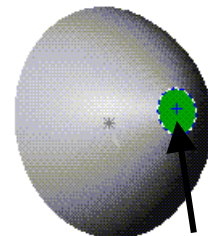
Create the Extruded Boss feature.


14) Select the Sketch plane. Right-click near the small hidden **back face**. Click **Select Other**  from the Pop-up menu. Click the **right mouse button (N)** until the back face is displayed. Accept the back face. Click the **left mouse button (Y)**.



15) **Rotate** the part to view the back face.

16) Create the profile. Click **Sketch** . Extract the **back face** to the Sketch plane. Click **Convert Entities** .



17) Extrude the Sketch. Click **Extrude Boss/Base** . Enter **0.400** for Depth. Display the Boss-Extrude1 feature. Click **OK**.




18) Rename **Boss-Extrude1** to **LensNeck**.

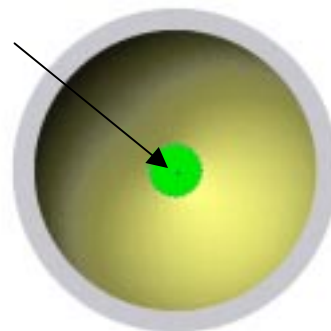
19) Save the LENS. Click **Save** .

Create the LENS – Hole Wizard Counterbore Hole Feature

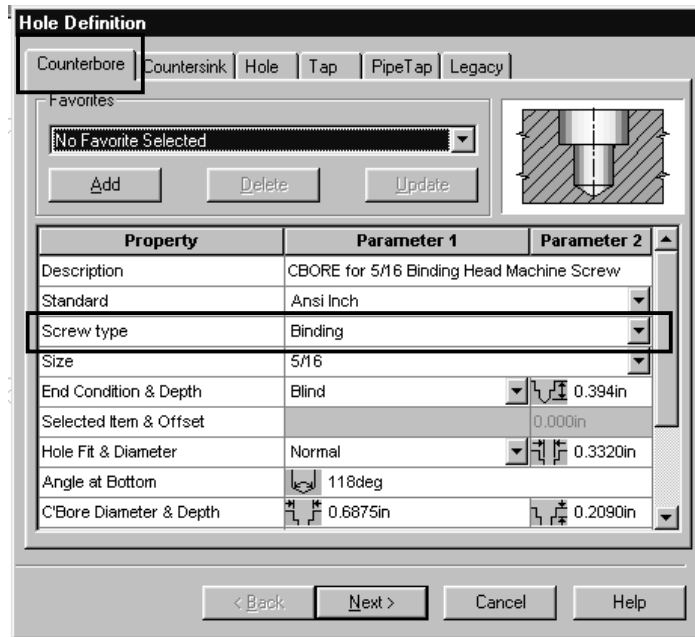
The LENS requires a Counterbore Hole feature. Use the HoleWizard. HoleWizard assists in creating complex and simple Hole features. Specify the user parameters for the custom Counterbore Hole.

Create the Counterbore Hole.

20) Select the Sketch plane. Click **Front** . Click the small **inside back face** of the Base-Revolve feature.



Create the Counterbore Hole. Click **HoleWizard** . The Hole Definition dialog box is displayed. Click the **Counterbore** tab.

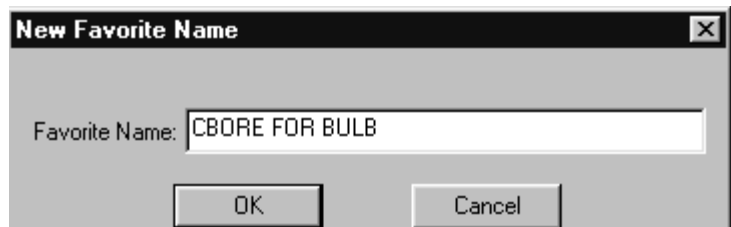




21) Define the parameters. Click the Parameter 1 **Binding** in the Screw type property text box. The Parameter 1 and Parameter 2 text boxes are displayed.

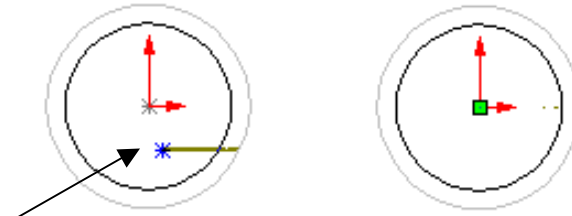
Property	Parameter 1	Parameter 2
Description	CBORE for 1/2 Hex Head Bolt	
Standard	Ansi Inch	
Screw type	Hex Bolt	
Size	1/2	
End Condition & Depth	Through All	0.394in
Selected Item & Offset	0.000in	
Hole Fit & Diameter	Normal	0.5312in
Angle at Bottom	118deg	
C'Bore Diameter & Depth	.6	0.2

22) Enter **Hex Bolt** from the drop down list for Screw type. Select $\frac{1}{2}$ from the drop down list for Size. Click **Through All** from the drop down list for End Condition & Depth. Accept the Hole Fit and Diameter value. Click the **C-Bore Diameter** value. Enter **0.600**. Click the **C-Bore Depth** value. Enter **0.200**.

23) Add the new hole type to the favorites list. Click the **Add** button. Enter **CBORE FOR BULB**. Click **OK**.



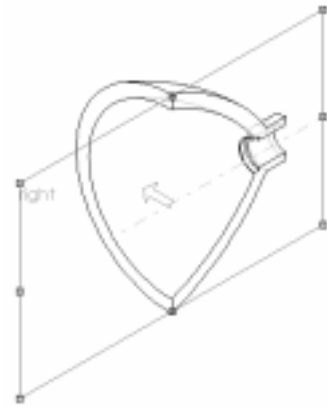
- 24) Click **Next** from the Hole Definition dialog box. Position the hole coincident with the Origin. Click **Add Relations** . Click the **center point** of the Counterbore hole. Click the **Origin** . Click **Coincident**. Complete the hole. Click **Finish** from the Hole Wizard.



- 25) Expand the Hole. Click **Plus Sign**  to the left of the Hole feature. Sketch3 and Sketch4 are used to create the Hole feature.




- 26) Display the Section view of BulbHole through the Right plane. Click the **Right** plane from the FeatureManager. Click **View** from the Main menu. Click **Display, SectionView**. Click the **Flip Side to View** check box. Click **OK**. Display the Isometric View. Click **Isometric**.



- 27) Display the Full view. Click **View, Display, SectionView**.

- 28) Rename **CboreHole1** to **BulbHole**.

- 29) Save the LENS. Click **Save** .




Create the LENS - Boss Revolve Thin Feature

Create a Boss Revolve Thin feature. Rotate an open sketched profile around a centerline. The sketch profile must be open and cannot cross the centerline. The Tangent Arc sketch tool utilizes extracted geometry created with the Convert Entities sketch tool. Delete the extracted geometry after the Tangent Arc is created.

Use the Boss Revolve Thin feature to physically connect the LENS to the BATTERYPLATE in the FLASHLIGHT.


Create the Boss Revolve Thin feature.

- 30) Select the Sketch plane. Click the **Right** plane. Display the Right view. Click **Right** .


- 31) Create the Sketch. Click **Sketch** . Sketch the centerline. Click **Centerline** . Sketch a **horizontal centerline** collinear to the Top plane through the Origin . The new centerline is collinear with the Temporary axis.

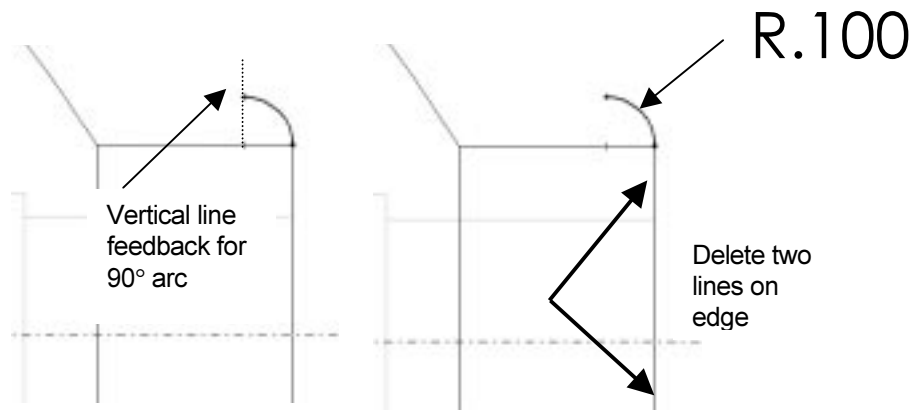
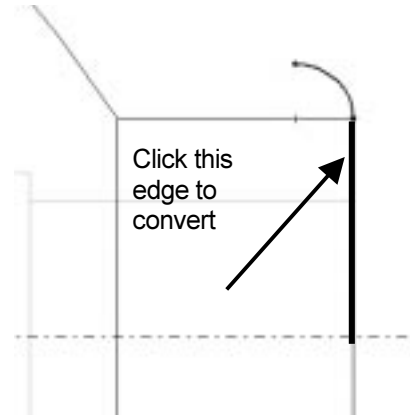
32) Select the right edge. Right-click in the **Graphics window**. Click **Select** from the Pop-up menu. Click the **right edge** of the Base feature.

33) Click **Convert Entities** . Select the **edge**. Create an arc tangent to the extracted edge.

34) Click **TangentArc** . Click the **top point** of the vertical line. Drag the **mouse pointer** to the left. The mouse pointer displays a vertical line when the endpoint aligns with the arc center point. Create the 90° arc. Release the **mouse button**.

Note: To create the 90° arc, the Snap to points in the Grid/Units must be unchecked.


35) The vertical line segments are required to create the Tangent Arc. Remove the two line segments. Click **Trim** . Click **both vertical edges**. The Sketch consists of an arc and a centerline.



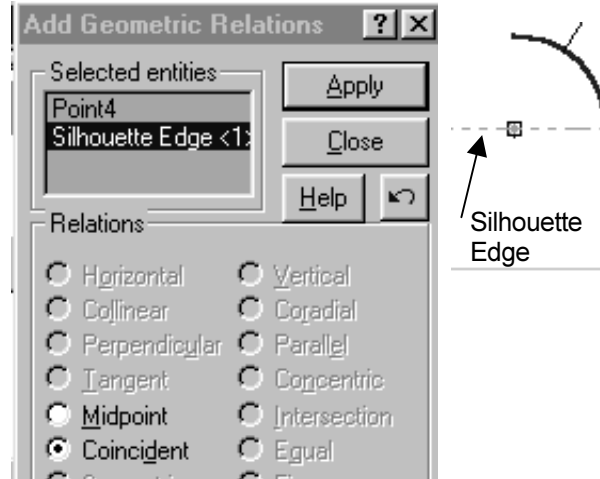
36) Add a dimension. Click **Dimension** . Create a radial dimension. Enter **0.100**.

The sketch arc requires a coincident relationship. The center point of the arc is coincident with the horizontal silhouette edge of the Base-Revolve feature.

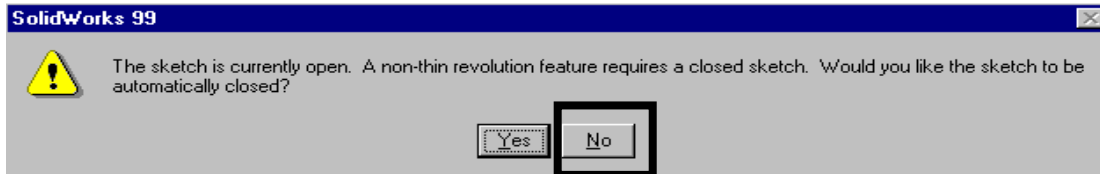
37) Add geometric relations.

Click **Add Relations** 
 Click the **arc center point**.
 Click the **horizontal line**
 (silhouette edge) of the
 Base-Revolve feature.
 Click the **Coincident** button.
 Click **Apply**. Click **Close**.

The black Sketch is fully defined.

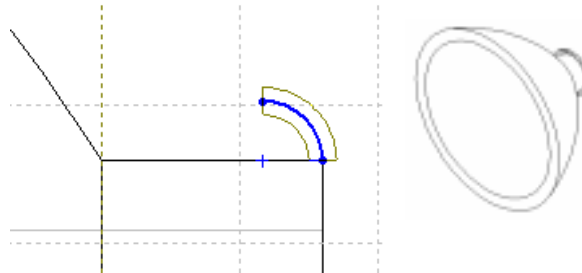


38) Revolve the Sketch. Click **Revolve** . A warning message appears:



39) Keep the Sketch open. Click **No**. The Thin Feature check box is active.

40) Create the Thin-Revolved feature on both sides of the Sketch. Select **Mid-Plane** from the Type list box. Enter **0.050** for Wall Thickness. Display the Boss-Revolve-Thin1 feature. Click **OK**.



41) Rename **Boss-Revolve-Thin1** to **LensConnector**.

42) Save the LENS. Click **Save** .

43) Turn off the Temporary Axis. Click **Views**. Click **Temporary Axis** to uncheck.



Create the LENS - Extruded Boss Feature


Use the Extruded-Boss feature to create the front LensCover. The feature extracts the front outside circular edge from the Base-Revolve feature. The front LensCover is a key feature for designing the mating component. The mating component is the LENS CAP.

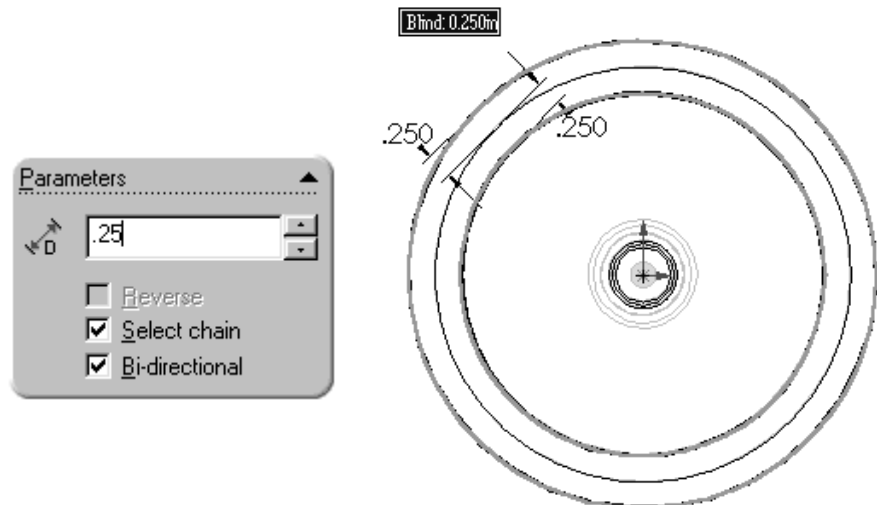
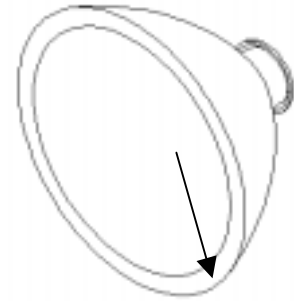
Create the Extruded Boss feature.

- 44)** Select the Sketch plane. Click the **front circular face**.

Display the Front view. Click **Front** .

- 45)** Create the Sketch. Click **Sketch** . Click the **outside circular edge**. Click the **Offset Entities** . Click the **Bi-directional** check box. Enter **0.250**.

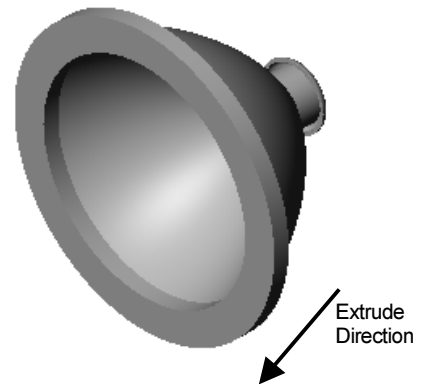
- 46)** Extrude the Sketch. Click **Extrude Boss/Base** . Enter **0.250** for Depth.



- 47)** Display the Boss-Extrude feature. Click **OK**.

- 48)** Rename **Boss-Extrude** to **LensCover**.

- 49)** Save the LENS. Click **Save** .



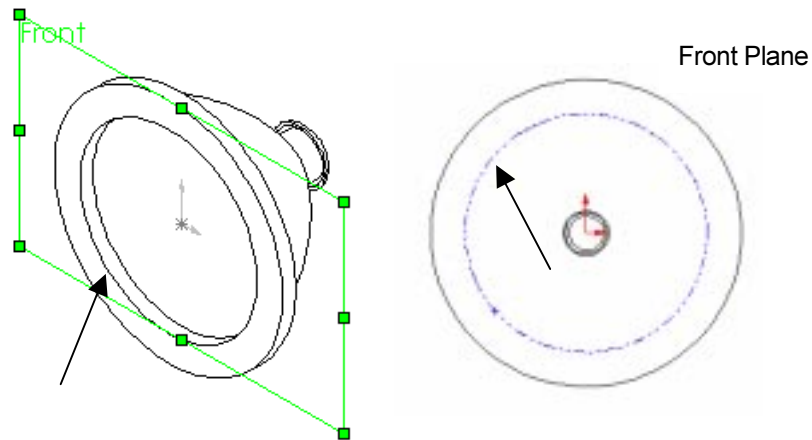
Create the LENS - Extruded Boss Feature

An Extruded Boss feature is used to create the LensShield. The feature extracts the inside circular edge of the LensCover and places it on the Front plane. The LensShield feature is transparent in order to view the BULB and simulate clear plastic.

Create the Extruded Boss feature.

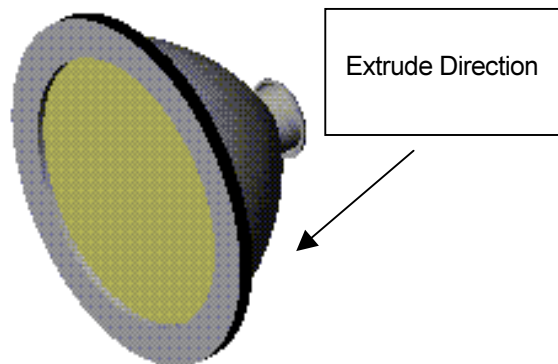
50) Select the Sketch plane. Click the **Front** plane from the FeatureManager. Display the Front view. Click **Front** .

51) Sketch the profile. Click **Sketch** . Click the **front inner circular edge** of the LensShield (Boss-Extrude2). Click **Convert Entities** . The circle is projected onto the Front Plane.

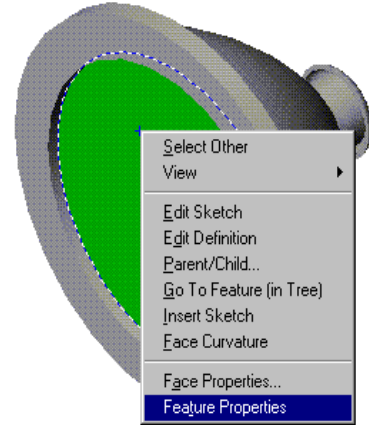


52) Extrude the Sketch. Click **Extruded Boss/Base** . Enter **0.100** for Depth. Click **OK**.

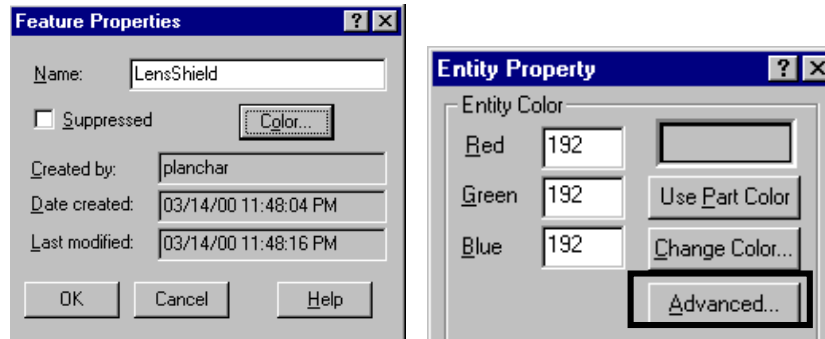
53) Rename **Boss-Extrude3** to **LensShield**.



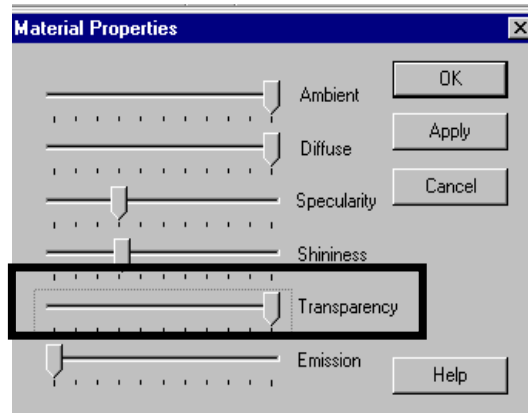
54) Add transparency to the LensShield. Right-click the **LensShield** in the Graphics window. Click **Feature Properties**. The Feature Properties dialog box is displayed.





55) Click the **Color** button. The Entity Property dialog box is displayed. Click the **Advanced** button.

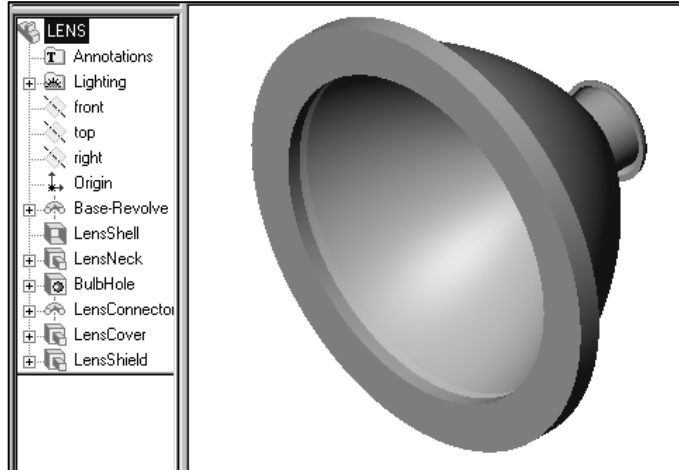


56) Set the transparency for the feature. Drag the **Transparency slider** to the far right side. Click **OK** from the Material Properties dialog box. Click **OK** from the Entity Property dialog box. Click **OK** from the Feature Properties box.



57) Display the transparent faces.
 Click **Shaded** 
 When the LensShield is selected, the faces are not transparent.
 Click in the **Graphics window** to display the face transparency.

58) Save the LENS. Click **Save** .



BULB

The BULB is contained within the LENS assembly. The BULB is a purchased part. The BULB utilizes the Revolved feature as the Base feature.



BULB Feature Overview

Create the Revolved Base feature from a sketched profile on the Right plane, Figure 2.11a.

Create a Revolved Boss feature using a B-Spline sketched profile. A B-Spline is a complex curve, Figure 2.11b.

Create a Revolved Cut Thin feature at the base of the BULB, Figure 2.11c.

Create a Dome feature at the base of the BULB, Figure 2.11d.

Create a Circular Pattern feature from an Extruded Cut, Figure 2.11e.



Figure 2.11a

2.11b

2.11c

2.11d

2.11e

Modify the BULB to practice Edit Definition and Edit Sketch after a design change.




Create the BULB - Revolved Base Feature


The solid Revolved Base feature requires a centerline and a sketched profile. The flange of the BULB is located inside the Counterbore Hole of the LENS. Align the bottom of the flange with the Front plane. The Front plane mates against the Counterbore face.

Create a Revolved Base feature.

59) Create the BULB. Click **New** . Click **PartEnglishTemplate**. Click **OK**. Click **Save** . Enter part name. Enter **BULB**. Click **Save**.

60) Select the Sketch plane. Click the **Right** plane. Display the Right view. Click **Right** .

61) Sketch the centerline. Click **Sketch** . Click **Centerline** . Sketch a horizontal **centerline** collinear to the Top plane through the Origin .

62) Right-click the **Front** plane in the FeatureManager. Click **Show**. Sketch the profile. Create six lines. Click **Line** .

Create the first line. Sketch a **vertical line** to the left of the Front plane.

Create the second line. Sketch a **horizontal line** with the endpoint coincident to the Front plane.

Create the third line. Sketch a short **vertical line** towards the centerline, collinear with the Front plane.

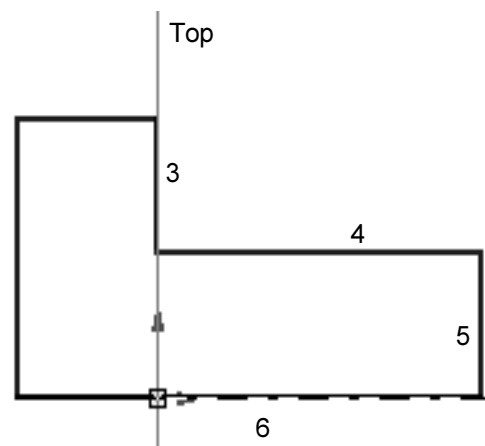
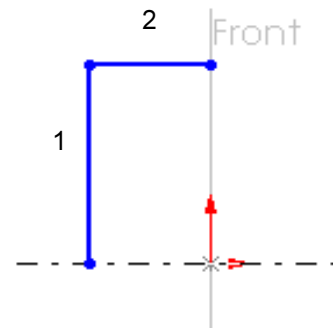
Create the fourth line. Sketch a **horizontal line** to the right.


Create the fifth line. Sketch a **vertical line** with the endpoint collinear with the centerline.

Create the sixth line. Close the Sketch. Sketch a **horizontal line**.

Verify that vertical Line 3 is collinear with the Top plane.

A solid Revolved feature requires a closed profile.



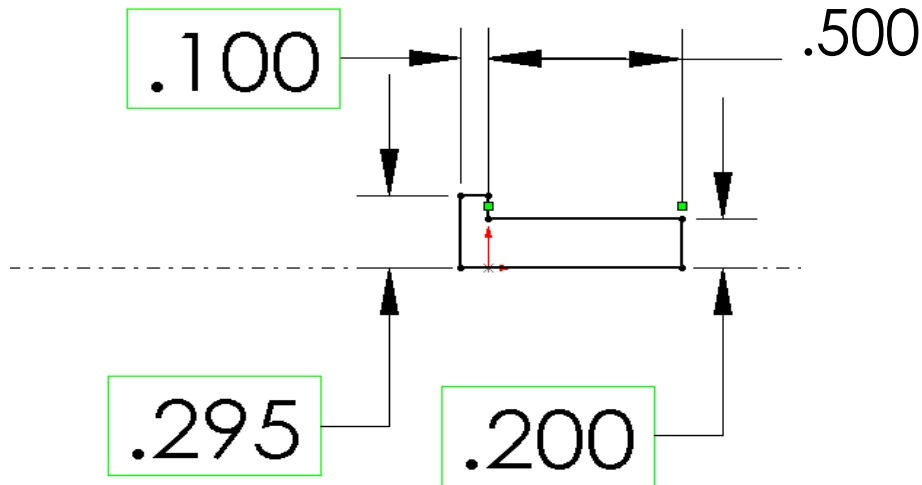
63) Add dimensions. Click **Dimension** .

Create a vertical linear dimension. Click the **right line**. Enter **.200**.

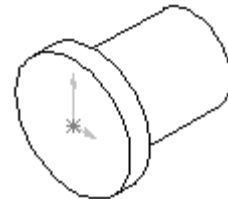
Create a vertical linear dimension. Click the **left line**. Enter **.295**.

Create a horizontal linear dimension. Click the **top left line**. Enter **0.100**.

Create a horizontal linear dimension. Click the **top right line**. Enter **0.500**.



64) Revolve the Sketch. Click **Revolve**  from the Feature toolbar. The Revolve Feature dialog box is displayed. Accept the default option values. Click **OK**.



65) Save the BULB. Click **Save**.

Create the BULB - Revolved Boss Feature




The bulb requires a second solid Revolve feature. The profile utilizes a complex curve called a B-Spline (Non-Uniform Rational B-Spline or NURB). B-Splines are drawn with control points. Adjust the shape of the curve by dragging the control points.

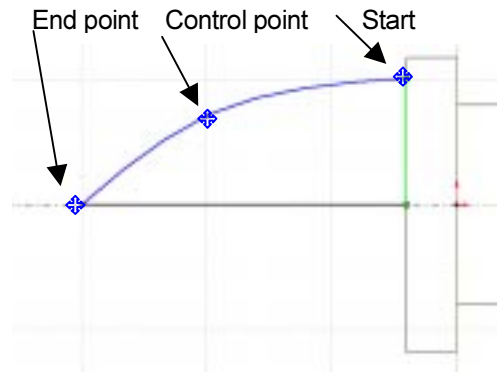
Create the Revolved Boss feature.


66) Turn the Grid Snap off. Click **Grid** . Uncheck the **Snap to points** check box.

67) Select the Sketch plane. Click the **Right** plane. Display the Right view. Click **Right**



- 68)** Create the Sketch. Click **Sketch** . Sketch the centerline. Click **Centerline** . Sketch a **horizontal centerline** collinear to the Top plane, coincident to the Origin .




- Sketch the profile. Click **B-Spline** . Sketch the start point. Click the **left vertical edge** of the Base feature.

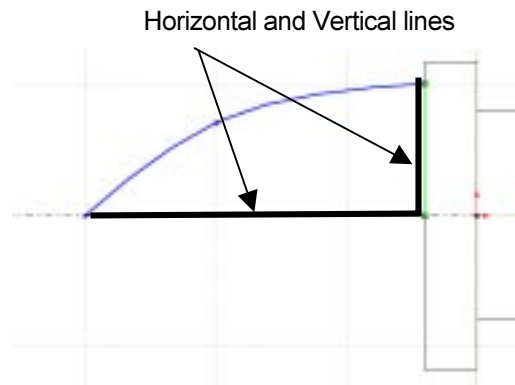
Sketch the control point. Drag the **mouse pointer** to the left of the Base feature and below the first point. Release the **mouse** button.


Sketch the end point. Click the **control point**. **Drag** the mouse pointer to the centerline. Release the **mouse** button.

- 69)** Adjust the B-Spline. Click **Select** . Position the **mouse pointer** over the B-Spline control point. Drag the **mouse pointer** upward. Release the **mouse** button.

Note: SolidWorks does not require dimensions to create a feature.

- 70)** Complete the profile. Sketch two lines. Click **Line** . Create a horizontal line. Sketch a **horizontal line** from the B-Spline endpoint to the left edge of the Base-Revolved feature. Create a vertical line. Sketch a **vertical line** to the B-Spline start point, collinear with the left edge of the Base-Revolved feature.



- 71)** Revolve the Sketch. Click **Revolve**  from the Feature toolbar. The Revolve Feature dialog box is displayed. Accept the default options. Display the Revolve feature. Click **OK**.

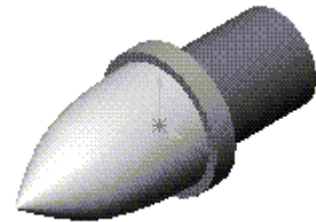
- 72)** Save the BULB. Click **Save**.

Create the BULB - Revolved Cut Thin Feature

A Revolved Cut Thin feature removes material by rotating an open sketch profile around a centerline.



Create the Revolved Cut Thin feature.

- 73)** Select the Sketch plane. Click the **Right** plane.




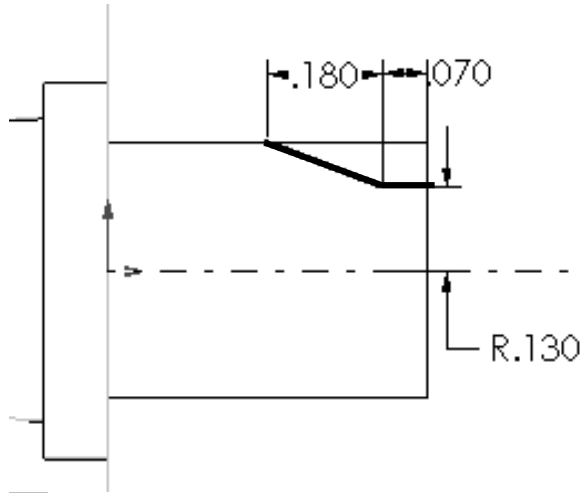
74) Create the profile. Click **Sketch** . Display the Right view. Click **Right** .


75) Sketch the centerline. Click



Centerline . Sketch a **horizontal centerline** collinear to the Top plane, coincident to the Origin .

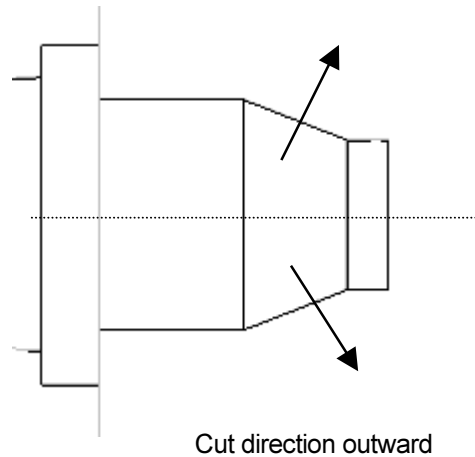
76) Sketch the profile. Click **Line**

. Sketch a line from the **midpoint** of the top silhouette edge downward and to the right. Sketch a horizontal line with the end point coincident with the **right edge**.



77) Add dimensions. Click **Dimension** . Create the right vertical dimension. Enter **0.130**. Add two horizontal dimensions. Enter **0.180** for the first line. Enter **0.070** for the second line. The black Sketch is fully defined.

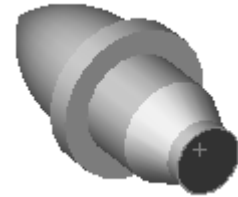
78) Revolve the Sketch. Click **Revolved Cut**  from the Feature toolbar. The Revolve Feature dialog box is displayed. Material to be removed points away from the centerline. Click **No** to the question, "Would you like the sketch to be automatically closed?" Click the **Direction**  button. Enter **0.150** for Thickness. Display the Revolved Cut Thin feature. Click **OK**.



79) Save the BULB. Click **Save**.

Create the BULB - Dome Feature

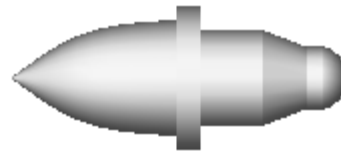
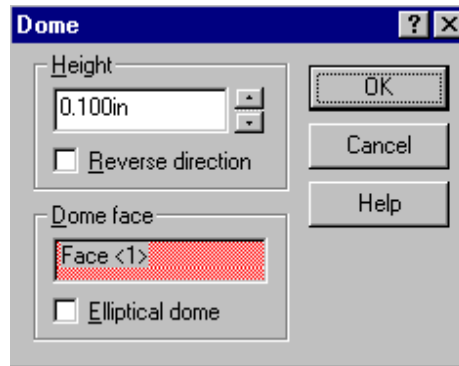
A Dome feature creates spherical or elliptical shaped geometry. Use the Dome feature to create the Connector feature of the BULB.



Create the Dome feature.

80) Select the Sketch plane. Click the **back circular face** of the Revolve Cut Thin.

81) Click **Insert** from the Main menu. Click **Features, Dome**. The Dome dialog box is displayed. Enter **0.100** for Height. Display the Dome. Click **OK**.



82) Save the BULB. Click **Save**.

Create the BULB - Circular Pattern

The Pattern feature creates one or more instances of a feature or a group of features. The Circular Pattern feature places the instances around an axis of revolution.



The Pattern feature requires a seed feature. The seed feature is the first feature in the Pattern. The seed feature in this section is an Extruded-Cut.

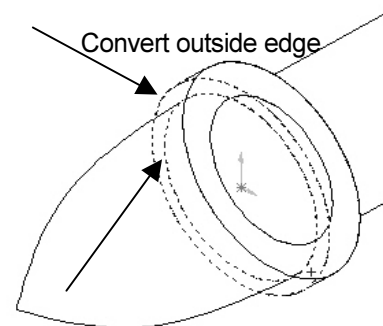
Create the Circular Pattern.

83) Select the Sketch plane. Click the **front circular face** of the Base feature.


84) Create the Sketch. Click **Sketch** .

Display the Front view. Click **Front** .

85) Extract the outside circular edge of the Sketch plane. Click **Select** . Click the **outside circular edge**. Click **Convert Entities** .

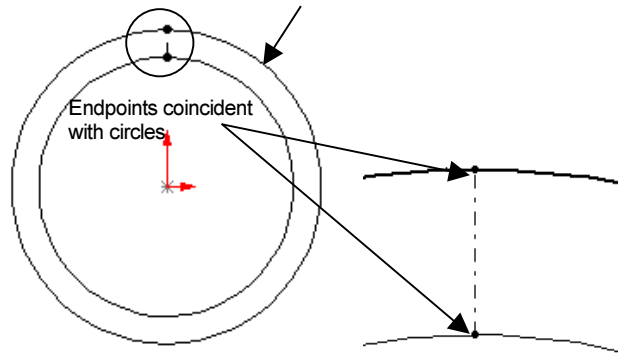




86) **Zoom in** to display the top area of the sketch plane.

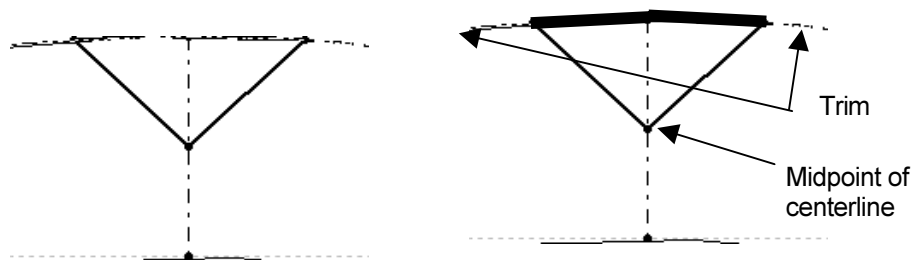
87) Sketch the centerline. Click **Centerline** . Sketch a **vertical centerline** coincident with the top and bottom circular circles and coincident with the Right plane.

88) **Zoom** to display the centerline and the outside circular edge.

89) Sketch a V-shaped line. Click **Mirror** . Select the **centerline**. Click **Line** .




Create the first point. Click the **midpoint**  of the centerline. Create the second point. Click the coincident **outside circle edge**. Turn off Mirror. Click **Mirror** .

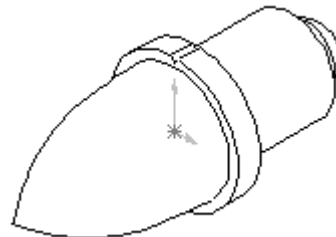


90) Trim the lines. Click **Trim** . Click the **circle** outside the V shape.

91) Add the geometry relations. Click **Add Relations** . Add the perpendicular relations. Click the **two lines**. Click the **Perpendicular** button.


The black Sketch is fully defined.

92) Extrude the Sketch. Click **Extruded Cut** . Click **Up to Next** from the Type list box. Display the Extruded Cut. Click **OK**.

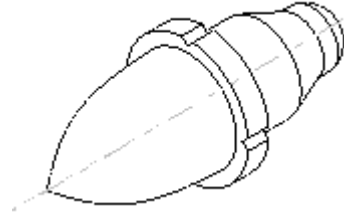
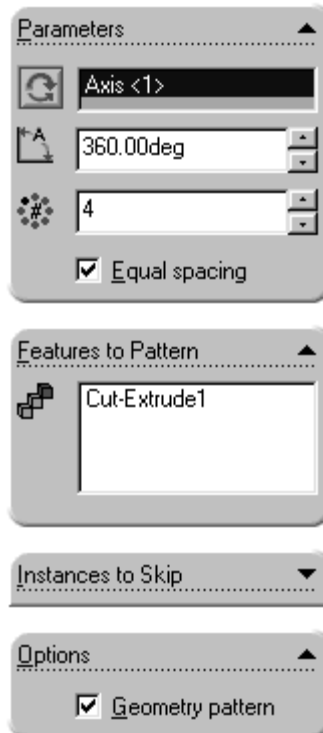


- 93)** Display the Temporary axis. Click **View, Temporary Axis** from the Main menu. Click the **Direction selected** text box. Click **Temporary Axis**.

The Cut-Extrude is the seed feature for the Pattern.

- 94)** Create the Pattern. Click the **Cut-Extrude** feature. Click **Circular Pattern** . The Circular Pattern dialog box is displayed.

- 95)** Create 4 copies of the Cut. Enter **360** in the Total angle spin box. Enter **4** in the Total Instances spin box. Click the **Equal spacing** check box. Click the **Geometry pattern** check box. Display the Pattern feature. Click **OK**.



- 96)** Edit the Pattern feature. Right-click on the **Circular Pattern** from the Feature Manager. Click **Edit Definition**. Enter **8** in the Total instances spin box. Display the updated Pattern. Click **OK**.

- 97)** Hide the Temporary axis. Click **View** from the Main menu. Click **Temporary Axis**.

- 98)** Save the BULB. Click **Save**.



Design Change with Rollback

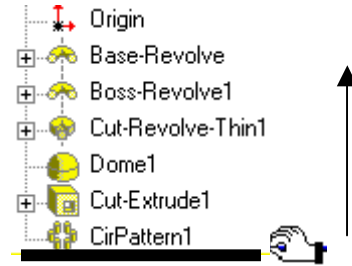
You are required to implement a design change for the BULB. The BULB requires a small fillet on the front outside circular face. The Fillet feature is created before the v-shaped Extruded Cut and Circular Pattern.

The Rollback and Edit Definition functions are used to implement the design change.

The Rollback function allows a feature to be redefined in any state or order. Implement the design change. Add the new Fillet feature before the Extruded Cut feature. Reorder the Fillet feature in the FeatureManager and view the results.

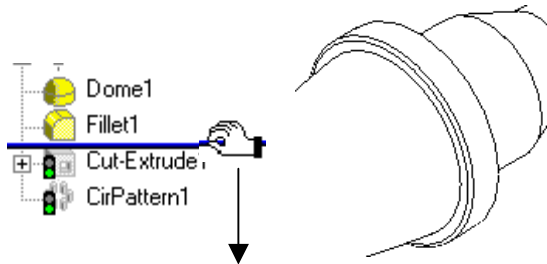
Create the Fillet.

99) Position the Rollback bar. Place the **mouse pointer** over the yellow Rollback bar at the bottom of the FeatureManager design tree. The mouse pointer displays a symbol of a hand. Drag the **Rollback** bar upward to below the Dome feature.

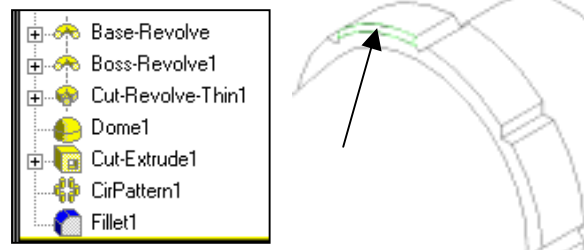


100) Click the outside **front circular edge** of the BULB. Enter **0.01** for Fillet Radius. Click **OK**.

101) Position the Rollback bar. Drag the **Rollback** bar to the bottom of the FeatureManager.



102) Reorder Fillet1. Drag **Fillet1 text** to the bottom of the FeatureManager.



103) One Fillet edge is selected between 2 v-shape Extruded Cuts. This is not the design intent. Drag **Fillet1 text** before the Cut-Extrude1 text.

104) Save the BULB. Click **Save**.

The v-shape Extruded Cut requires a 2D sketch plane. The Extruded Cut fails when the Fillet radius becomes too large and removes the sketch plane. Creating features on curves surfaces with reference planes is discussed in the next project.




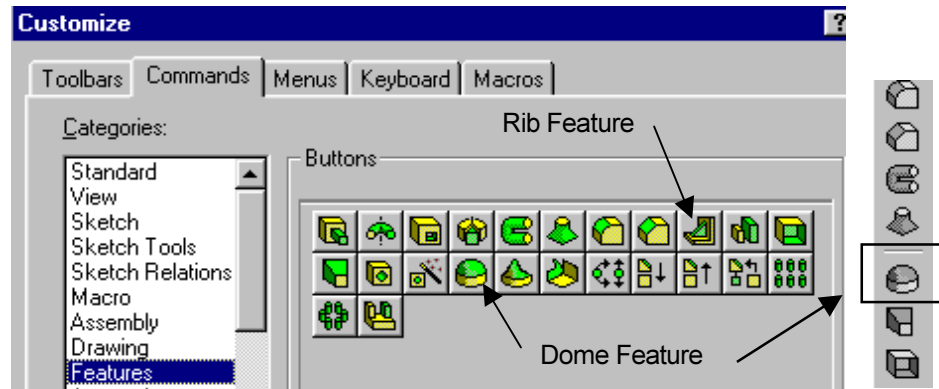
Customizing Toolbars

The default Toolbars contains numerous icons that represent basic functions. SolidWorks contains additional features and functions not displayed on the default Toolbars.

Customize the Toolbar.

105) Place the Dome icon and the Rib icon on the Features Toolbar. Click **Tools** from the Main menu. Click **Customize**. The Customize dialog box is displayed.

106) Click the **Commands** tab. Click **Features** from the category text box. Drag the **Dome**  icon into the Features Toolbar.



Drag Dome icon into Feature toolbar

107) Drag the **Rib**  icon into the Features Toolbar. Update the Toolbars. Click **OK** from the Customize dialog box. The Rib feature is required for the next Project.

You have just created two parts:

- LENS
- BULB

Practice the exercises before moving onto the next section.

Questions

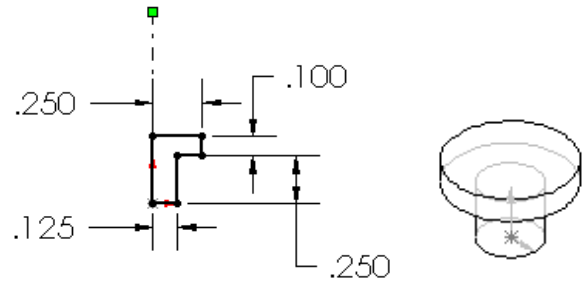
1. Identify the function of the following features:
Revolved Base
Revolved Cut Thin
2. Name the two line types required in the sketch of a Revolved feature.
3. What is the function of the Shell feature?
4. An arc requires _____ points?
5. Name the required points of an arc?
6. When do you use the Hole Wizard feature?
7. Describe the Mid Plane option for a Revolved Thin feature.
8. What is a B-Spline?
9. Identify the required information for a Circular Pattern?
10. How do you control the display of the Temporary Axis?
11. Define Rollback in the FeatureManager.
12. How do you add the Dome feature icon to the Feature Toolbar?

Exercises

I. Create the following Revolved Parts:

Exercise 2.1

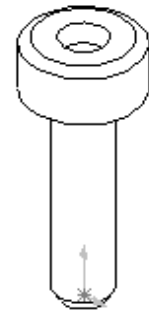
SIMPLE SCREW.



Exercise 2.1

Exercise 2.2

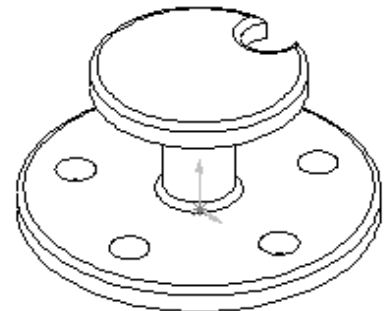
SIMPLE CAP SCREW.



Exercise 2.2

Exercise 2.3

SPOOL.

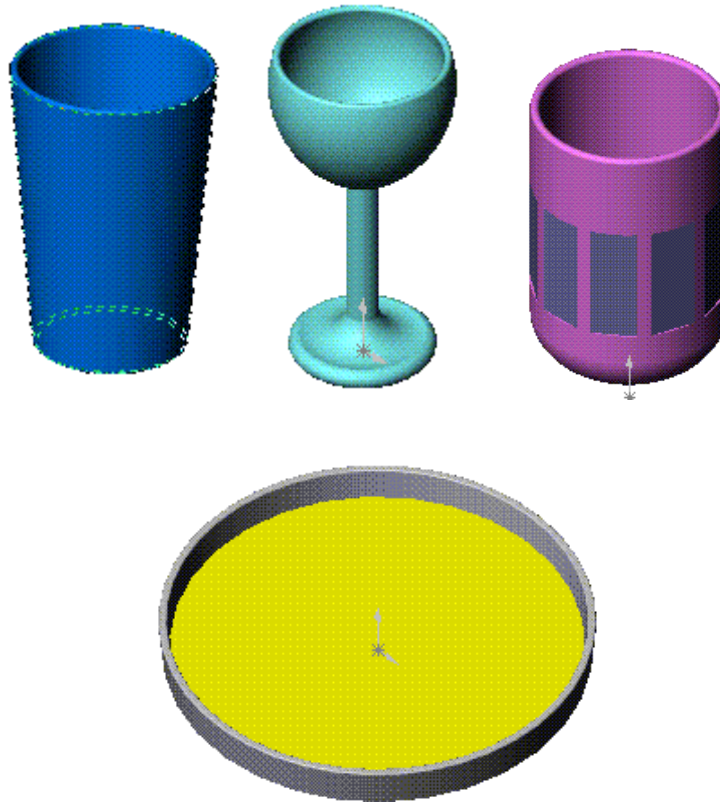


Exercise 2.3

II. Design Projects.

Exercise 2.4

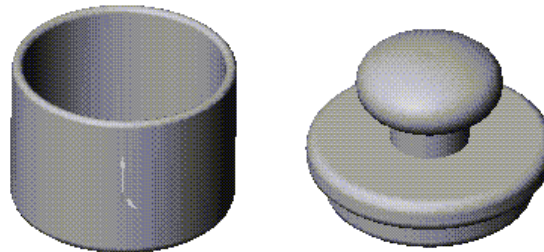
Create a TRAY and GLASS. Use real objects to determine the overall size and shape of the Base feature. Below are a few examples.



Exercise 2.4

Exercise 2.5

Create a JAR-BASE. Use the dimensions from the JAR-BASE to determine the size of the JAR-COVER.



Exercise 2.5