Engineering Design with SolidWorks 2001Plus

A Competency Project Based Approach Utilizing 3D Solid Modeling

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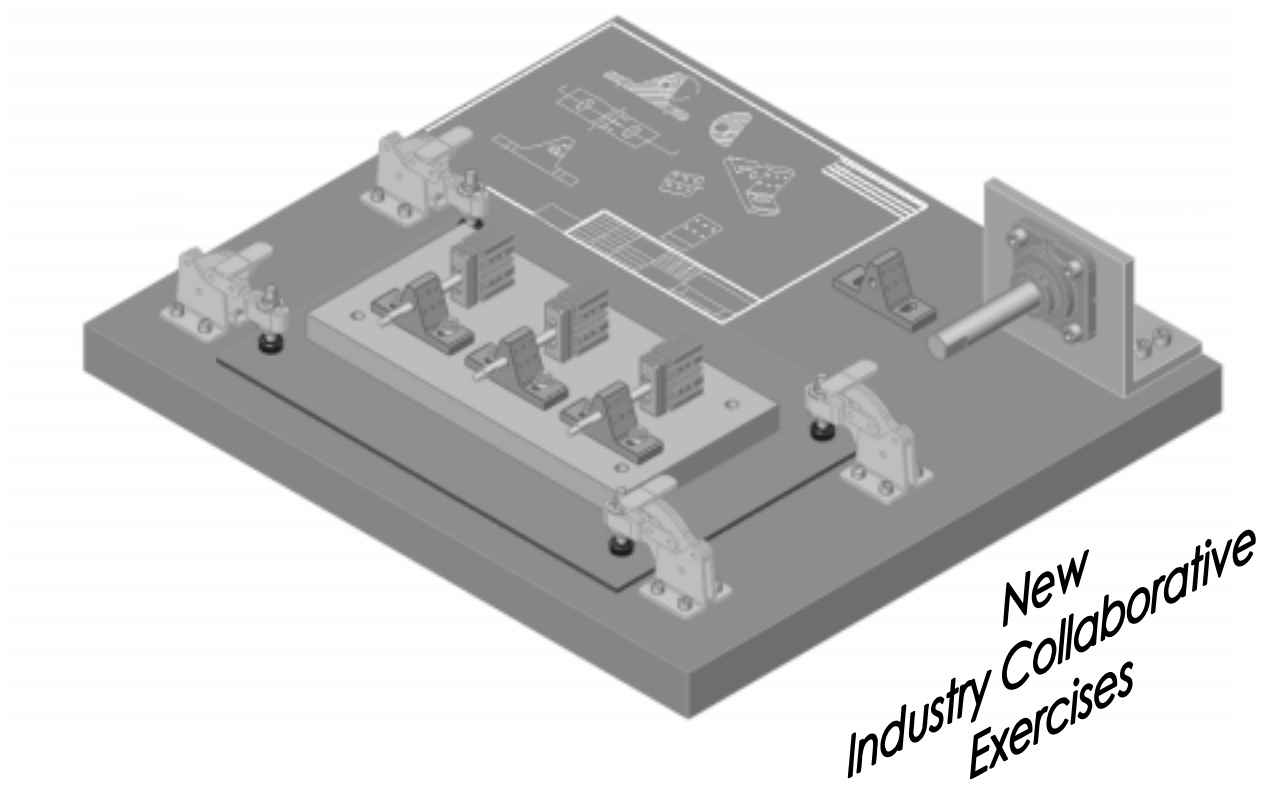

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Project 4

Extrude and Revolve Features

Below are the desired outcomes and usage competencies based upon the completion of Project 4.

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.
A product design that is cost effective, serviceable and flexible for future manufacturing revisions.	Specific knowledge and understanding of the Extrude and Revolve features.
Four key flashlight components: • BATTERY • BATTERY PLATE • LENS • BULB	

NOTES:

Project 4 – Extrude and Revolve Features

Project Objective

Create four components of the flashlight. Create the BATTERY, BATTERY PLATE, LENS and BULB components.

Project Situation

You are employed by a company that specializes in providing promotional trade show products. Your company is expecting a sales order for 100,000 flashlights with a potential for 500,000 units next year. Prototype drawings of the flashlight are required in three weeks.

You are the design engineer responsible for the project. You contact the customer to discuss design options and product specifications. The customer informs you that the flashlights will be used in an international marketing promotional campaign. Key customer requirements:

- Inexpensive reliable flashlight.
- Available advertising space of 10 square inches, 64.5 square centimeters.
- Light weight semi indestructible body.
- Self standing with a handle.

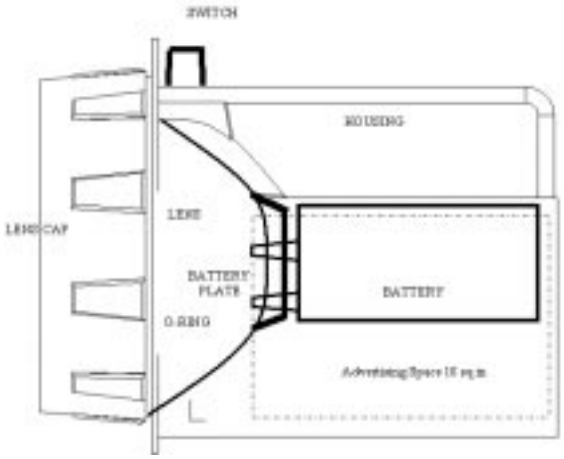


Figure 4.1

Your company's standard product line does not address the above key customer requirements. The customer made it clear that there is no room for negotiation on the key product requirements.

You contact the salesperson and obtain additional information on the customer and product. This is a very valuable customer with a long history of last minute product changes. The job has high visibility with great future potential.

In a design review meeting, you present a conceptional sketch. Your colleagues review the sketch. The team's consensus is to proceed with the conceptual design, Figure 4.1.

The first key design decision is the battery. The battery type will directly affect the flashlight body size, bulb intensity, case structure integrity, weight, manufacturing complexity and cost.

You review two potential battery options:

- A single 6-volt lantern battery.
- Four 1.5 volt D cell batteries.

The two options affect the product design and specification. Think about it.

A single 6-volt lantern battery is approximately 25% higher in cost and 35% more in weight. The 6-volt lantern battery does provide higher current capabilities and longer battery life.

A special battery holder is required to incorporate the four 1.5 volt D cell configuration. This would directly add to the cost and design time of the flashlight, Figure 4.2.

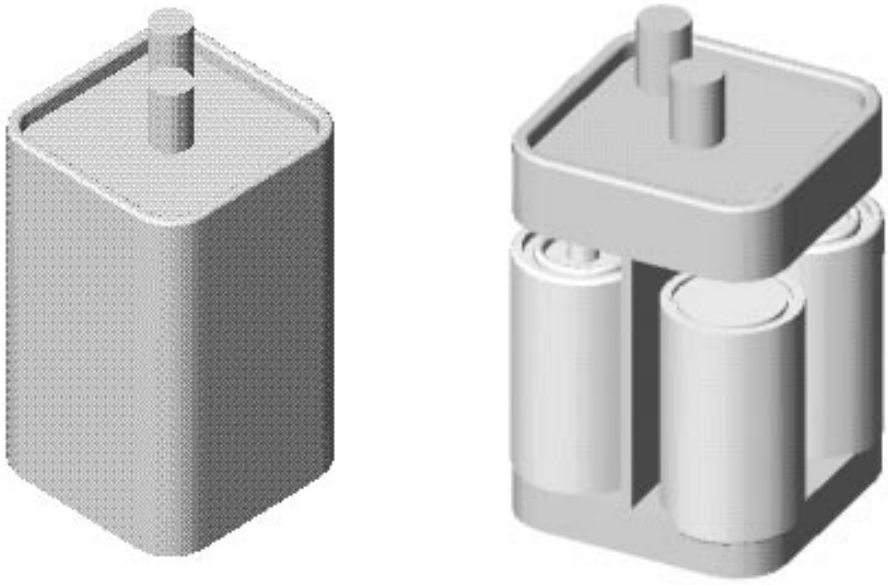


Figure 4.2

Time is critical. For the prototype, you decide to use a standard 6-volt lantern battery. This eliminates the requirement to design and procure a special battery holder. However, you envision the 4-D cell battery model for the next product revision. You design the flashlight to accommodate both battery design options.

Battery dimensional information is required for the design. Where do you go? Potential sources: product catalogs, company web sites, professional standards organizations, design handbooks and colleagues.

The team decides to purchase the following components: 6-volt BATTERY, LENS ASSEMBLY, SWITCH and an O-RING. Your company will design and manufacture the following components: BATTERY PLATE, LENSCAP, HOUSING and SWITCH PLATE.

Purchased Parts	Designed Parts
BATTERY	BATTERY PLATE
LENS ASSEMBLY	LENS CAP
SWITCH	HOUSING
O-RING	SWITCH PLATE

Project Overview

Create four parts in this section, Figure 4.3a:

- BATTERY
- BATTERY PLATE
- LENS
- BULB

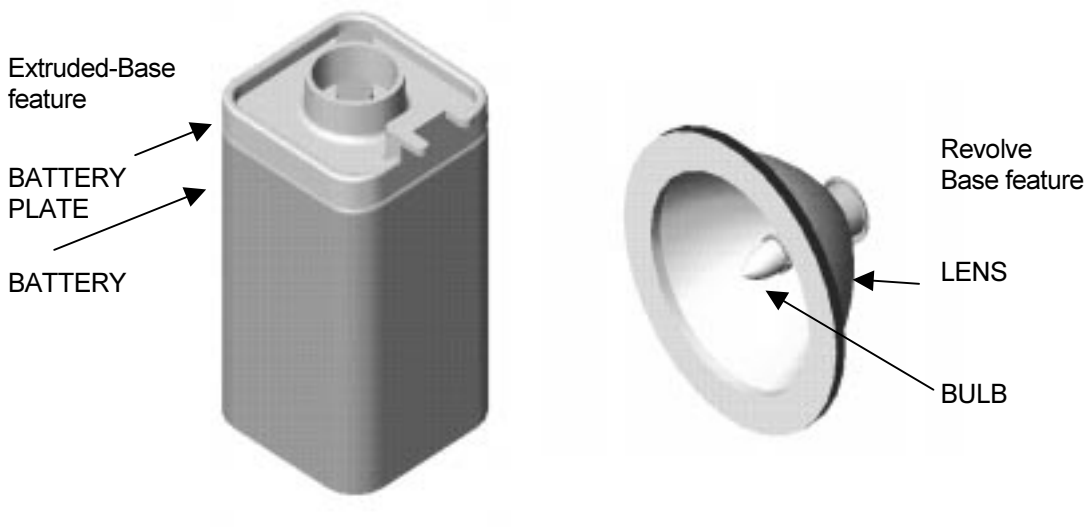


Figure 4.3a

Two major Base features are discussed in this project:

- Extrude – BATTERY and BATTERY PLATE.
- Revolve – LENS and BULB.

Note: Dimensions and features are used to illustrate the SolidWorks functionality in a design situation. Wall thickness and thread size have been increased for improved picture illustration. Parts have been simplified.

You will create four additional parts in Project 5 for a final flashlight assembly, Figure 4.3b.

- O-RING
- LENSCAP
- SWITCH
- HOUSING

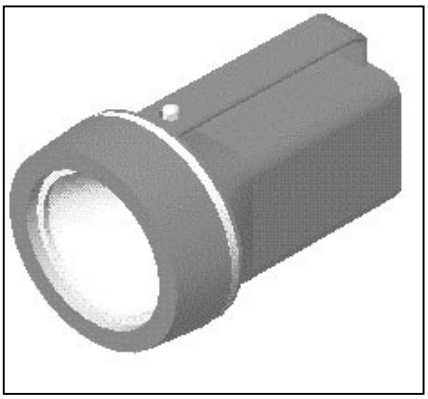


Figure 4.3b

BATTERY

The BATTERY is a simplified representation of an OEM component. The BATTERY consists of the following features:

- Extruded Base
- Extruded Cut
- Edge Fillets
- Face Fillets

The battery terminals are represented as cylindrical extrusions. The battery dimension is obtained from the ANSI standard 908D.

Note: A 6-volt lantern battery weighs approximately 1.38 pounds, (0.62kg). Locate the center of gravity closest to the center of the battery.

BATTERY Feature Overview

Create the BATTERY, Figure 4.4a. Identify the required BATTERY features.

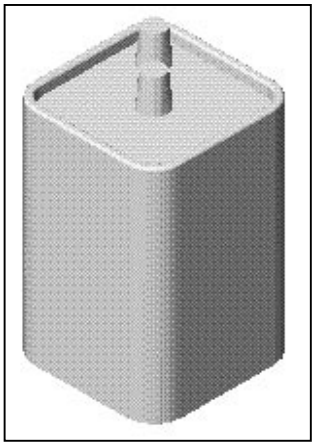


Figure 4.4a

- Extruded Base: The Extruded Base feature is created from a symmetrical square sketch, Figure 4.4b.
- Fillet: The Fillet feature is created by selecting the vertical edges and the top face, Figure 4.4c and Figure 4.4e.
- Extruded Cut: The Extruded Cut feature is created from the top face offset, Figure 4.4d.
- Extruded Boss: The Extruded Boss feature is created to represent the battery terminals, Figure 4.4f.

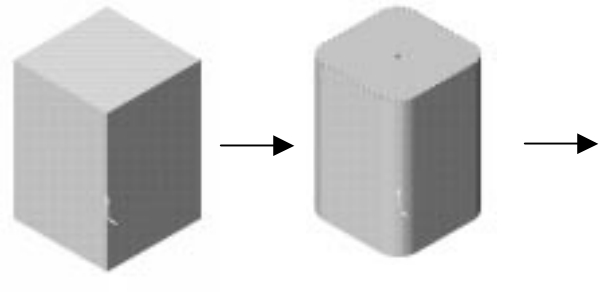


Figure 4.4b Figure 4.4c

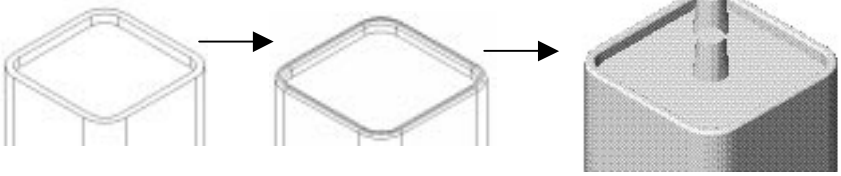


Figure 4.4d Figure 4.4e Figure 4.4f

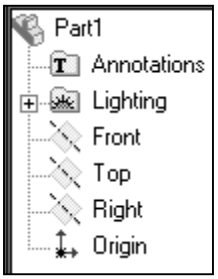
Let's create the BATTERY.

Create the Template

Dimensions for the FLASHLIGHT ASSEMBLY are provided both in English and Metric units. The Primary units are in inches. Three decimal places are displayed to the right of the decimal point. The Secondary units are in millimeters. Secondary units are displayed in brackets [x]. Two decimal places are displayed to the right of the decimal point. The PARTENGLISH TEMPLATE contains System Options and Document Properties settings for the parts contained in the FLASHLIGHT ASSEMBLY. Substitute the PARTMETRIC TEMPLATE to create the same parts in millimeters.

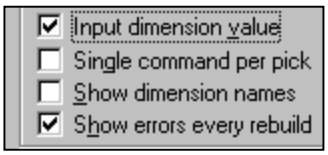
Create an English document template.

1) Click **New**. Click the **Part** template. Click **OK**. The Front, Top and Right reference planes are displayed in the Part1 Feature Manager.



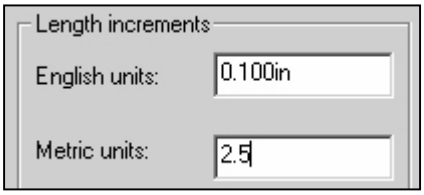
Set System Options.

2) Click **Tools**, **Options**, from the Main menu. The System Options - General dialog box is displayed. Insure that the check box Input dimension value and Show errors every rebuild in the General box are checked. These are the default settings.



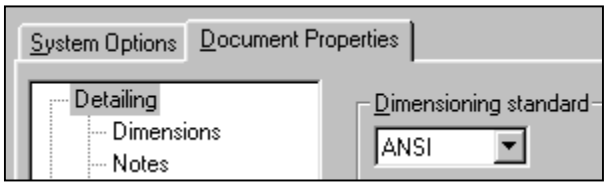
Set the Length increment.

3) Click the **Spin Box Increments** option. Click the English units **text box**. Enter **.100**. Click the Metric units **text box**. Enter **2.5**.



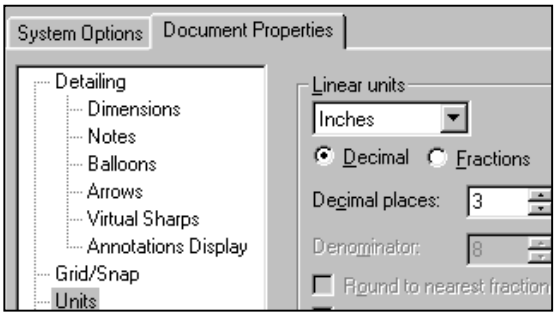
Set the Dimension Standard to ANSI.

4) Click the **Document Properties** tab. Select **ANSI** from the Dimensioning standard drop down list.



Set the Document Properties.

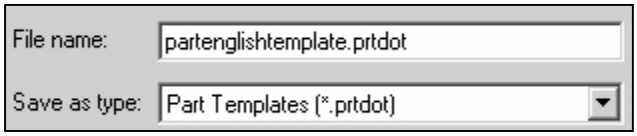
5) Click the **Units** option. Enter **inches**, [**millimeters**] from the Linear units list box. Click the **Decimal** button. Enter **3**, [**2**] in the Decimal places spin box.



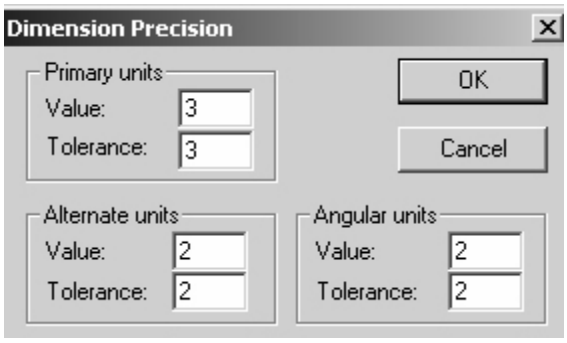
Save the Settings and Template.

6) Click **OK** from the Document Properties dialog box.

7) Click **File** from the Main menu. Click **Save As**. Click ***.prtdot** from the Save As type list box. The default Templates file folder is displayed. Enter **PARTENGLISH TEMPLATE**, [**PARTMETRIC TEMPLATE**] in the File name text box. Click **Save**.



ASMEY14.5M defines the types of decimal dimension display for inches and millimeters. The Primary units are in inches. Three decimal places are displayed to the right of the decimal point. The Secondary units are in millimeters. Secondary units are displayed in brackets [x]. Two decimal places are displayed to the right of the decimal point.



The precision is set to 3 decimal places for inches. Example: 2.700 is displayed. If you enter 2.7, the value 2.700 is displayed. The precision is set to 2 decimal places for millimeters. Example: [68.58] is displayed. For consistency, the inch part dimension values for the text include the number of decimal places required. The drawings utilizes the decimal dimension display as follows:

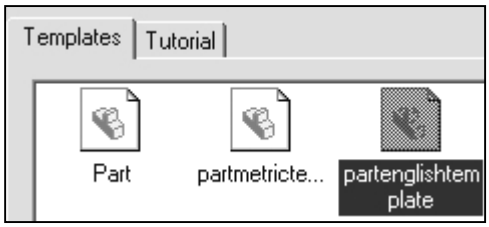
TYPES of DECIMAL DIMENSIONS (ASME Y14.5M)			
Description	**Example MM**	**Description**	**Example INCH**
Dimension is less than 1mm. Zero precedes the decimal point.	0.9 0.95	Dimension is less than 1 inch. Zero is not used before the decimal point.	.5 .56
Dimension is a whole number. No decimal point. Display no zero after decimal point.	19	Express dimension to the same number of decimal places as its tolerance. Add zeros to the right of the decimal point. If the tolerance is expressed to 3 places, the dimension contains 3 places to the right of the decimal point.	1.750
Dimension exceeds a whole number by a decimal fraction of a millimeter. Display no zero to the right of the decimal.	11.5 11.51		

Create the BATTERY

Create the BATTERY with an Extruded Base feature. The Extruded Base feature uses a square sketch drawn centered about the Origin on the Top plane. Build parts with symmetric relationships. Use a line of symmetry in a sketch. Add geometric relationships.

Create a New part.

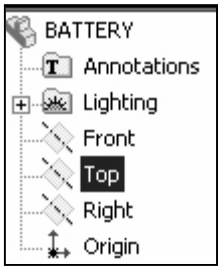
8) Click **New** . Click **PARTENGLISHTEMPLATE** from the Template dialog box. Click **OK**.



9) Save the empty part. Click **Save** . Enter the name of the part. Enter **BATTERY**. Click the **Save** button.

Create the Extruded Base feature.

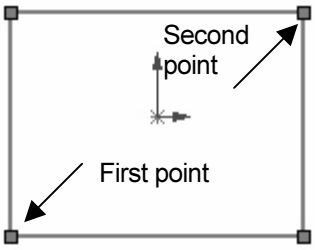
10) Select the Sketch plane. Click the **Top** plane from the Feature Manager.



11) Create a new Sketch. Click **Sketch** from the Sketch toolbar.

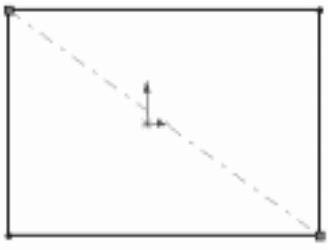
12) Display the Top view. Click **Top** from the Standards View toolbar.

13) Sketch the profile. Click **Rectangle** . Click the **first point** in the lower left quadrant. Click the **second point** in the upper right quadrant. The Origin is approximately in the middle of the Rectangle.

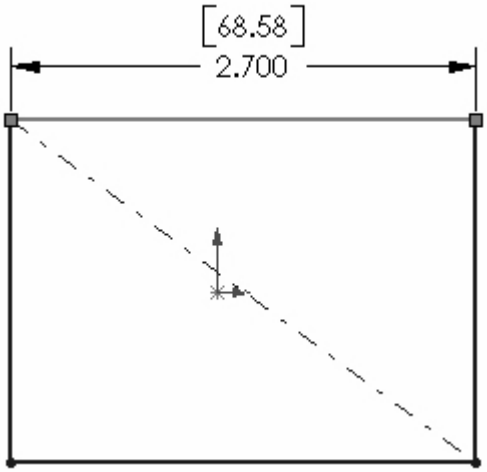


14) Sketch the Centerline. Click **Centerline** from the Sketch Tools toolbar. Sketch a diagonal centerline from the **upper left corner** to the **lower right corner**.

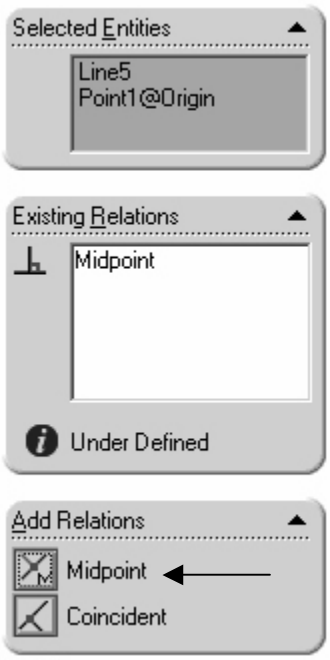
The endpoints of the centerline are coincident with the corner points of the Rectangle.

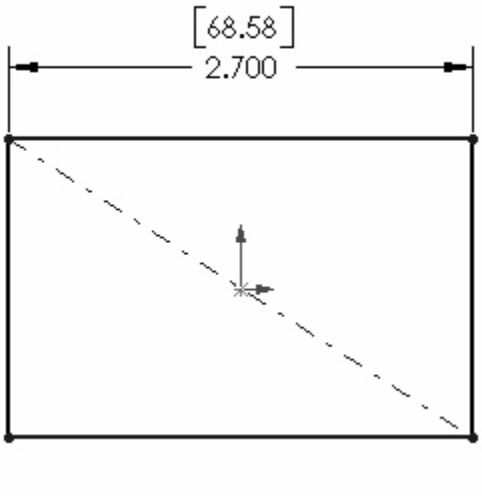


15) Add a dimension. Click **Dimension** from the Sketch toolbar. Select the **top horizontal line**. Drag the **mouse pointer** off the Sketch. Position the dimension text. Click the **text location** above the horizontal line. Enter **2.700**, [**68.58**] for width.



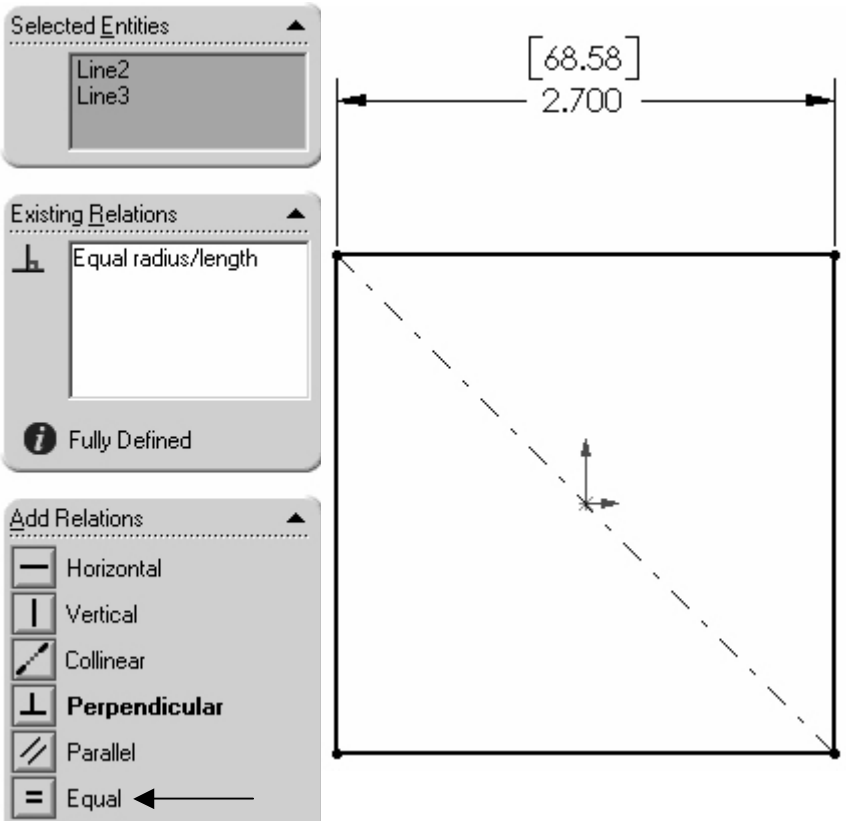
16) Add Geometric Relations. Click **Select**. Add a midpoint relation. Hold down the **Ctrl** key. Click the diagonal centerline, **Line5**. Click the **Origin**. Release the **Ctrl** key. Click the **Midpoint** button. Click **OK.**



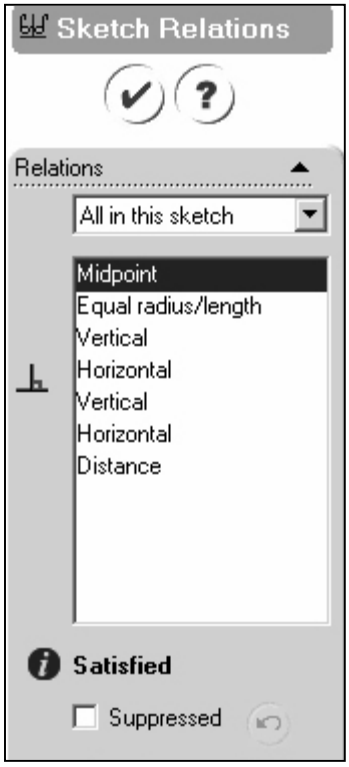


Note: The Line# may be different than the numbers above. The Line# is dependent on the Line# order creation.

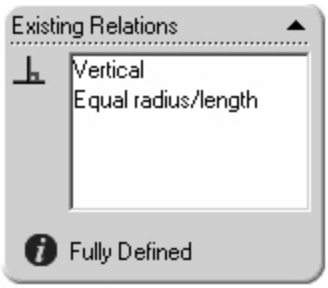
17) Add an equal relation. Click the top horizontal profile line, **Line1**. Click the left vertical profile line, **Line2**. Click the **Equal** button. Click **OK**. The black Sketch is fully defined.



18) Display the sketch relations. Click **Display/Delete Relations** from the Sketch Relations toolbar. The Distance relation is created from a dimension. The Vertical and Horizontal relations are created from the Rectangle Sketch tool. Click **OK**.



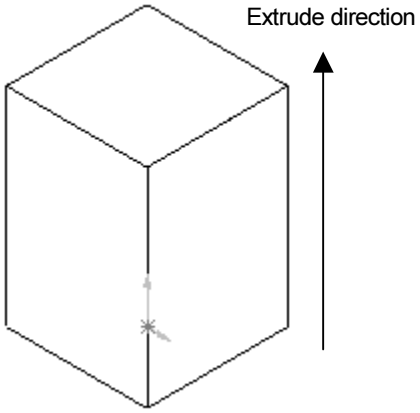
19) Click **Select** . Click a **vertical line**. Individual geometric relations are displayed in the Existing Relations text box.



20) Extrude the Sketch. Click **Extruded Boss/Base**. Blind is the default Type option. Enter **4.100**, [**104.14**] for Depth. Display the Base-Extrude feature. Click **OK**.

21) Fit the part to the Graphics window. Click **Zoom to Fit**.

22) Save the BATTERY. Click **Save**.



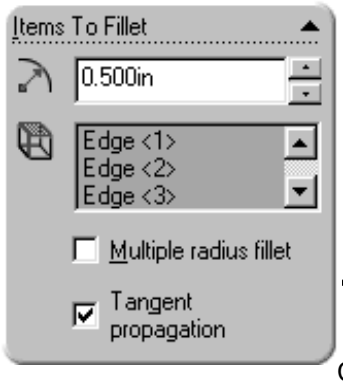
Create the BATTERY - Fillet Feature

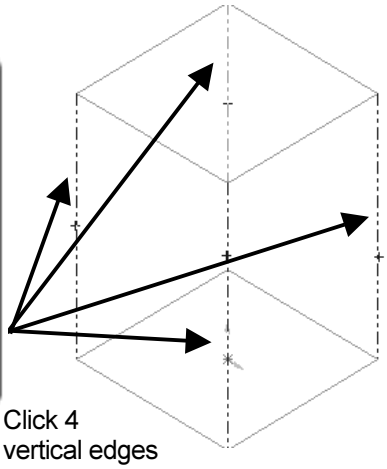
The vertical sides on the BATTERY are rounded. Use the Fillet feature to round the 4 side edges.

Create a Fillet feature.

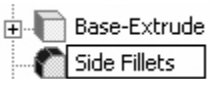
23) Display the part's hidden edges in gray. Click **Hidden In Gray** from the View toolbar.

24) Create a Fillet feature. Click **Fillet** from the Feature toolbar. Click the **4 vertical edges**. Enter **.500**, [**12.7**] for Radius. Display the Fillet feature. Click **OK**.





25) Rename **Fillet1** to **Side-Fillets** in the Feature Manager.

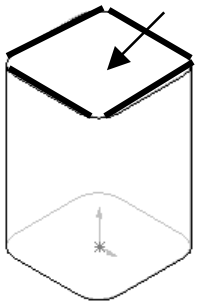


26) Save the BATTERY. Click **Save**.

Create the BATTERY - Extruded Cut Feature

The Extruded Cut feature removes material. An Offset Edge takes existing geometry, extracts it from an edge or face and locates it on the current sketch plane. Offset the existing Top face. Create a Cut feature.

Create the Extruded Cut feature.



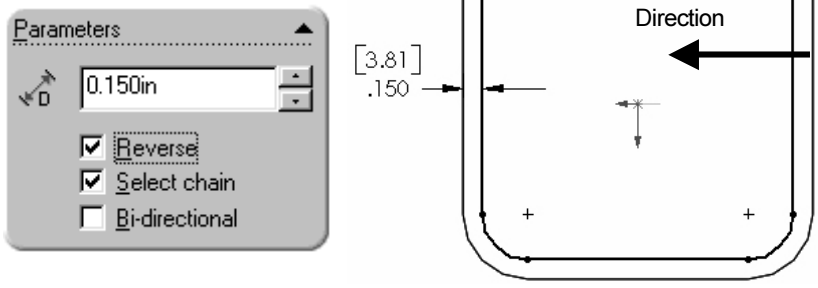
27) Select the Sketch plane. Click the **Top** face.

28) Create the Sketch. Click **Sketch** .

29) Display the face. Click **Top** from the Standards View toolbar.

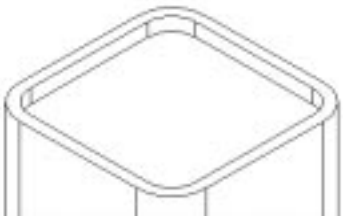
30) Offset the existing geometry from the boundary of the Sketch plane. Click **Offset** from the Sketch Tools toolbar. Enter **.150**, [**3.81**] for the Offset distance. Click the **Reverse** check box. The new Offset profile displays inside the original profile. Click **OK**.

Note: A leading zero is displayed in the spin box. For inch dimensions less than 1, the leading zero is not displayed in the part dimension.



31) Display the profile. Click **Isometric** from the Standards View toolbar.

32) Extrude the Offset profile. Click **Extruded Cut** from the Feature toolbar. Enter **.200**, [**5.08**] for Depth of the Cut. Display Cut-Extrude1. Click **OK**.



33) Rename **Cut-Extrude1** to **Top-Cut**.

34) Save the BATTERY. Click **Save** .

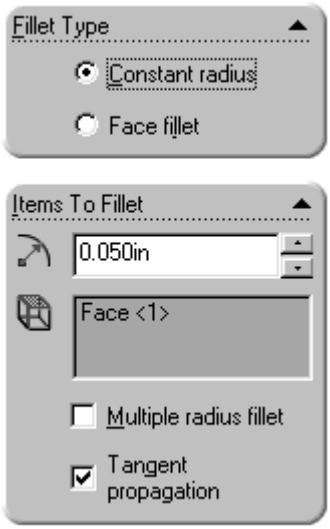
Create the Battery - Fillet Feature on the Top Face

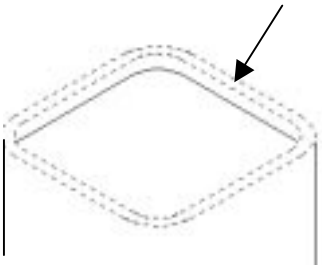
Top outside edges require fillets. Use the top face to create a constant radius Fillet feature. The top narrow face is small. Use the Face Selection Filter to select faces. Turn off the filters to select all geometry.

Create the Fillet feature on the top face.

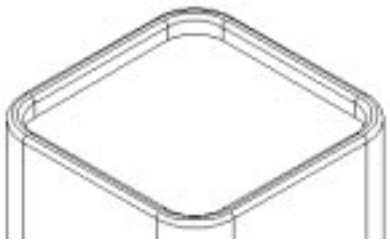
35) Display the Selection Filter toolbar. Click **View** from the Main menu. Click **Tools**, **Selection Filter**.

36) Create the Fillet. Click **Face Filter** from the Selection Filter toolbar. Click the **top thin face**. Select **Fillet** from the Feature toolbar. Face<1> is displayed in the Edge fillet items box. Click **Constant Radius** for Fillet Type. Enter **.050**, [**1.27**] for Fillet Radius.





37) Display the Fillet on the inside and outside top edges. Click **OK**.



38) Turn the Face Filter off. Click **Face Filter**.

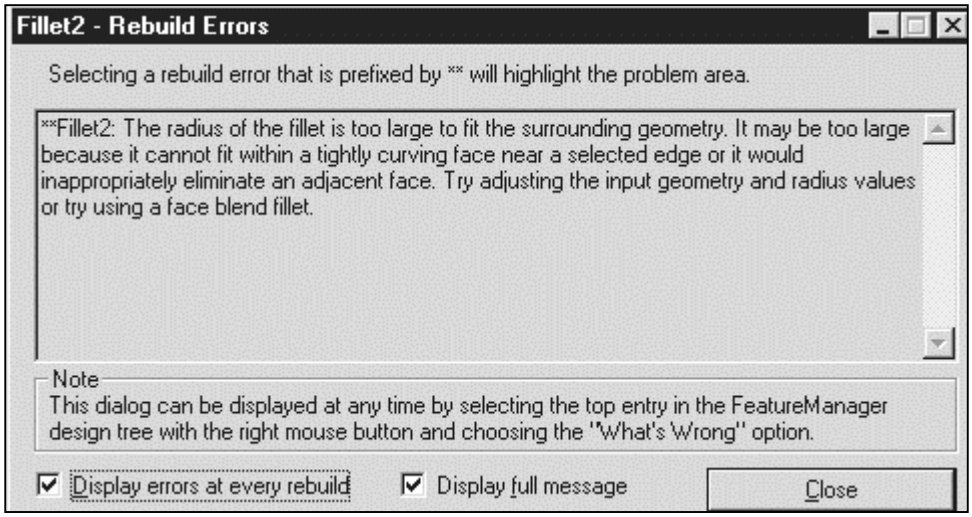
39) Rename **Fillet2** to **Top Face Fillet**.

40) Save the BATTERY. Click **Save**.

Note: Do not select a Fillet radius which is larger that the surrounding geometry.

Example: The top edge face width is .150, [3.81]. The Fillet is created on both sides of the face. A common error is to enter a Fillet too large for the existing geometry. A minimum face width of .200, [5.08] is required for a Fillet radius of .100, [2.54].

The following error occurs went the Fillet radius is too large for the existing geometry:



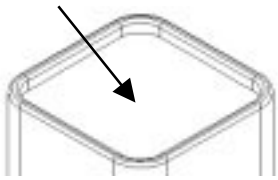
Avoid the Fillet Rebuild error. To avoid this error, reduce the Fillet size or increase the face width.

Create the BATTERY - Extruded Boss Feature

Two Battery Terminals are required. To conserve design time, represent the terminals as cylindrical Extruded Boss feature.

Create the Extruded Boss feature.

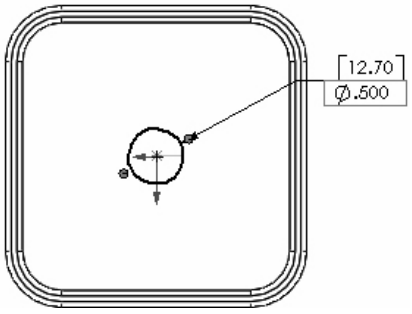
41) Select the Sketch plane. Click the **face** of the Top-Cut feature.



42) Create the Sketch. Click **Sketch** .

43) Display the Sketch plane. Click **Top** from the Standards View toolbar.

44) Sketch the Profile. Click **Circle** from the Sketch Tools toolbar. Create the first point. Click the **center point** of the circle coincident to the Origin . Create the second point. Drag the **mouse pointer** to the right. Release the **left mouse button**.

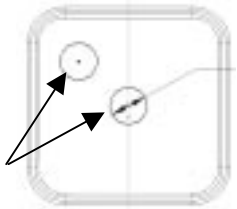


45) Click **Dimension** . Select the **circumference** of the circle. Click the **text location**. Enter **.500**, [**12.7**].

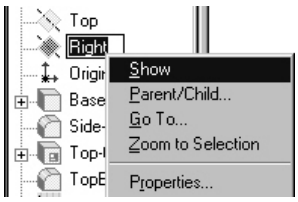
46) Copy the sketched circle. Click **Select** . Hold the **Ctrl** key down. Click the **circumference** of the circle. Drag the **circle** to the upper left quadrant. Create the second circle. Release the **mouse button**. Release the **Ctrl** key.

47) Add an equal relation. Click **Select** . Hold down the **Ctrl** key. Click the **circumference of the first circle**. Both circles are selected. Click **Equal**. Release the **Ctrl** key. Click **OK**.

The dimension between the center points is critical. Dimension the distance between the two center points with an aligned dimension.

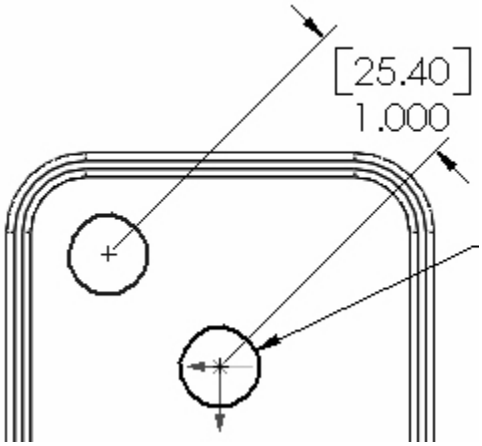


48) The Right plane is the dimension reference. Right-click the **Right** plane from the FeatureManager. View the plane. Click **Show**.

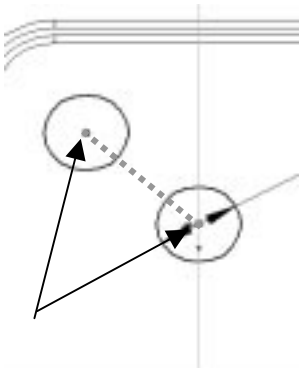


49) Add a dimension. Click **Dimension** . Click the **two center points** of the circles. Drag the **dimension text** off the profile. Release the **mouse button**. Enter **1.000**, [**25.4**] for the aligned dimension.

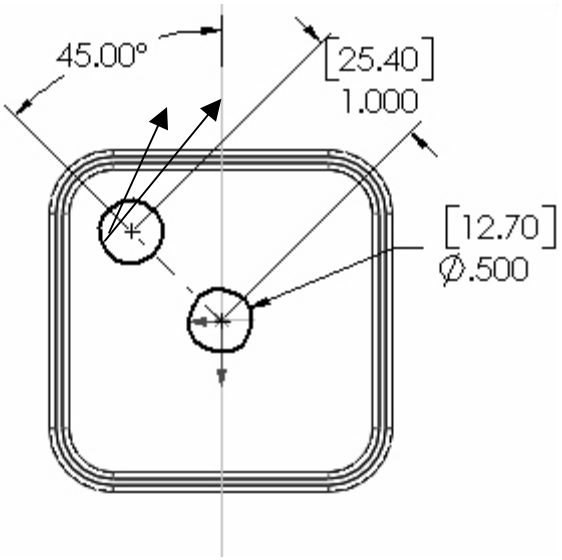
The dimension text toggles between linear and aligned. An aligned dimension is created when the dimension is positioned between the two circles.



50) Create an angular dimension. Click **Centerline** . Sketch a centerline between the **two circle center points**.



51) Create an acute angular dimension. Click **Dimension** . Click the **centerline** between the two circles. Click the **Right** plane. Drag the **dimension text** between the centerline and the Right plane off the profile. Release the **mouse button**. Enter **45**.



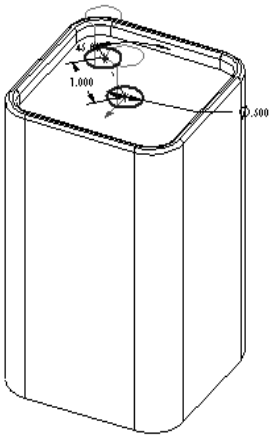
Note: Acute angles are less than 90°. Acute angles are the preferred dimension standard.

The overall battery height is a critical dimension. The battery height is 4.500 inch, [114.30mm]. Calculate the depth of the extrusion:

For Inches: 4.500in. – (4.100in. Base-Extrude height – .200in. Offset cut depth) = .600in. The depth of the extrusion is .600in.

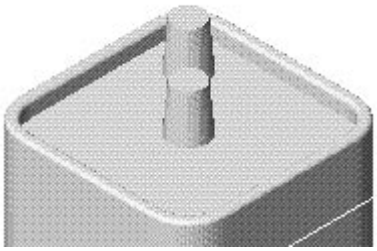
For Millimeters: 114.3mm – (104.14mm Base-Extrude height – 5.08mm Offset cut depth) = 15.24mm. The depth of the extrusion is 15.24mm.

52) Extrude the Sketch. Click **Extruded Boss/Base** from the Feature toolbar. Blind is the default Type option. Enter **.600**, [**15.24**] for Depth. Create a truncated cone shape for the battery terminals. Click the **Draft ON/OFF** button. A draft angle is a taper. Enter **5** in the Draft Angle text box.



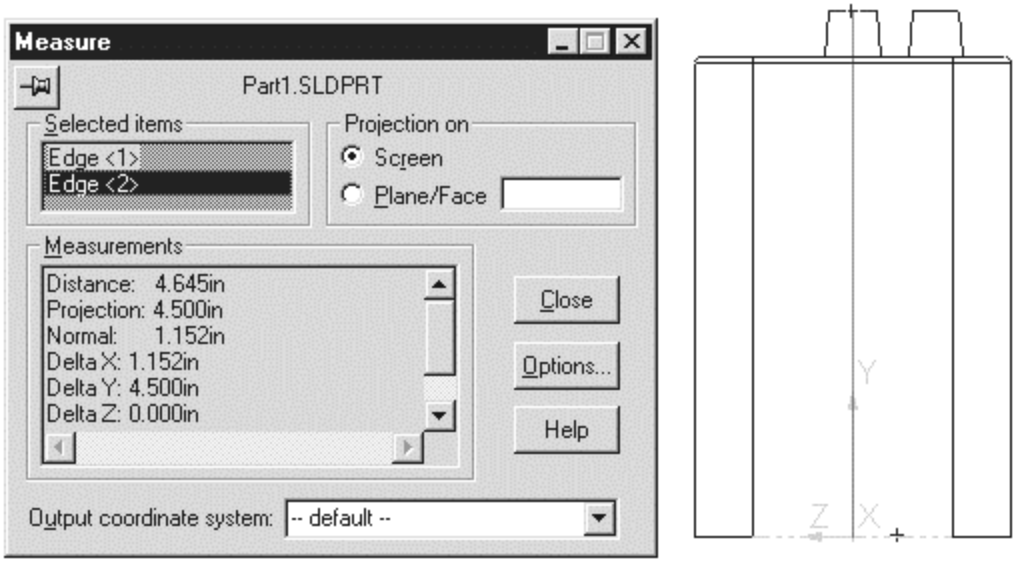
53) Display the Boss-Extrude1 feature. Click **OK**.

54) Rename **Boss-Extrude1** to **Terminals**. Rename **Sketch3** to **Sketch-TERMINALS**.



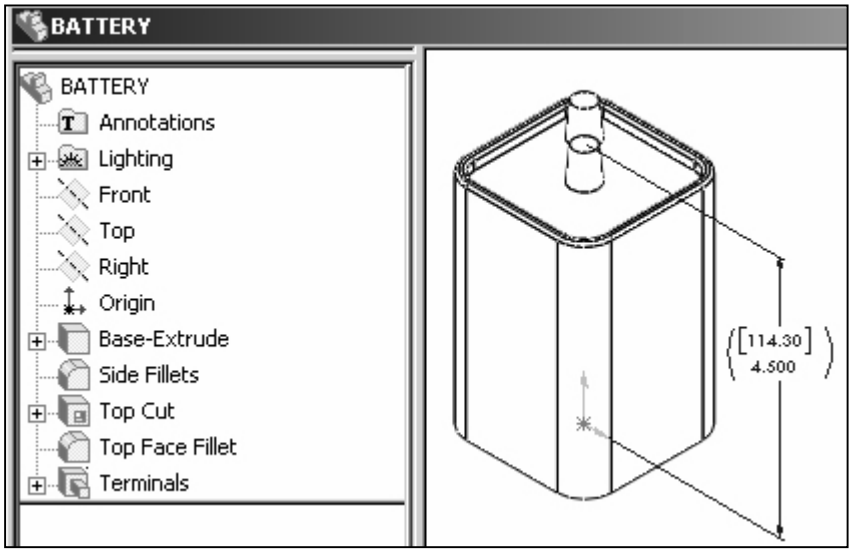
Measure the overall height.

55) Verify the overall height. Click **Tools**, **Measure** from the Main menu. Click **Right** from the Standard Views toolbar. Click the **top edge** of the battery terminal. Click the **bottom edge** of the battery. The overall height, Y is 4.500inch, [114.3mm]. Click **Close**.



56) Hide all planes. Click **View** from the Main menu. Click **Planes**.

57) Display the Trimetric view. Click **View Orientation**. Double-click **Trimetric**.



58) Save the BATTERY. Click **Save**.

BATTERY PLATE

The BATTERY PLATE has a variety of design functions. The BATTERY PLATE:

- Aligns the LENS assembly.
- Creates an electrical connection between the SWITCH assembly, BATTERY and LENS.

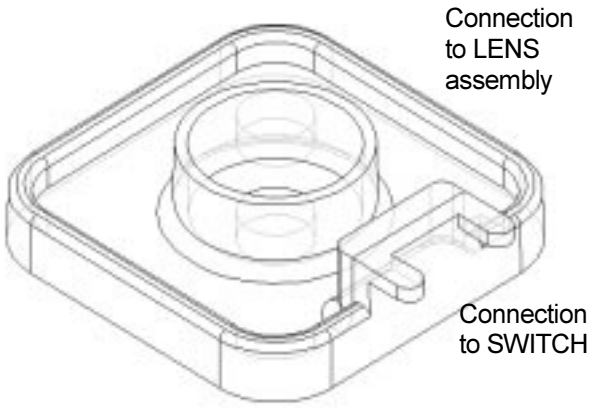


Figure 4.5

Design the BATTERY PLATE, Figure 4.5.

Utilize features from the BATTERY to develop the BATTERY PLATE.

BATTERY PLATE Feature Overview

Create the BATTERY PLATE. Modify the BATTERY features. Create two holes from the original sketched circles. Use the Extruded Cut feature, Figure 4.6.

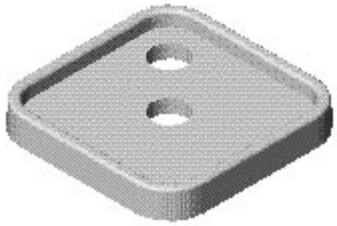


Figure 4.6

Modify the dimensions of the Base feature. Add a 1-degree draft angle.

Note: A sand pail contains a draft angle. The draft angle assists the sand to leave the pail when the pail is flipped upside down.

Create a new Extruded Boss Thin feature. Offset the center circular sketch, Figure 4.7.

The Extruded Boss Thin feature contains the LENS. Create an inside draft angle. The draft angle assists the LENS into the Holder.

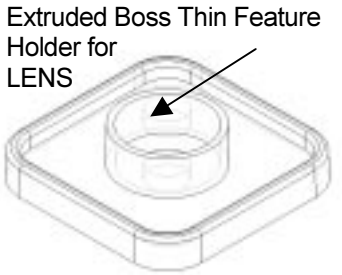


Figure 4.7

Create the first Extruded Boss feature using two depth directions, Figure 4.8. Create the second Extruded Boss feature using sketched mirror geometry, Figure 4.9.

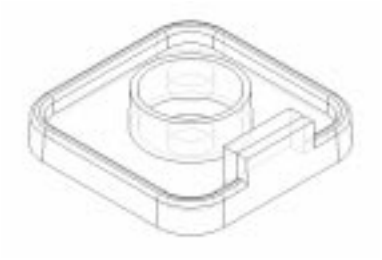


Figure 4.8

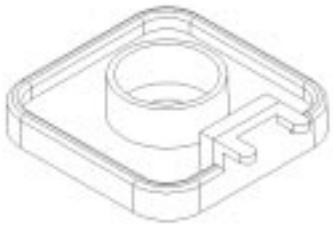


Figure 4.9

Create Face and Edge Fillet features to remove sharp edges, Figure 4.10.

Let's create the BATTERYPLATE.

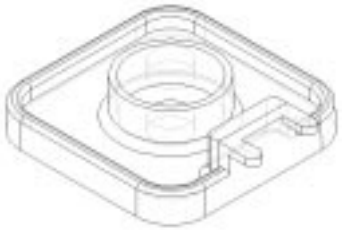


Figure 4.10

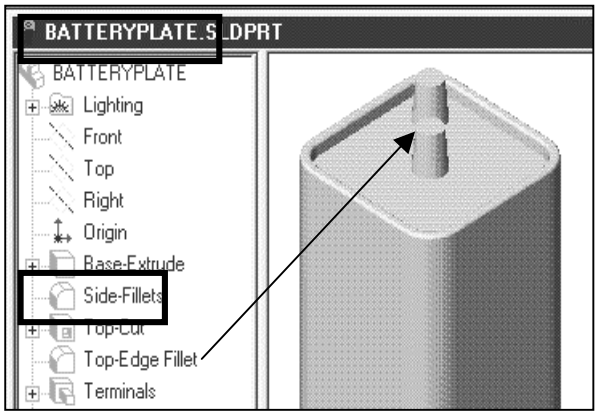
Create the BATTERYPLATE

Create the BATTERYPLATE from the BATTERY.

Create a New part from an existing part.

59) Create the BATTERYPLATE from the BATTERY. Click **File**, **SaveAs**. Enter the name of the part. Enter **BATTERYPLATE**. Click **Save**.

The BATTERYPLATE part icon is displayed at the top of the FeatureManager



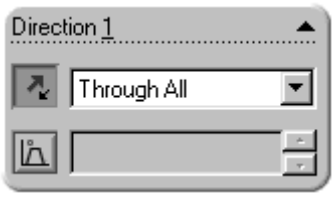
Create the BATTERYPLATE - Delete and Edit Features

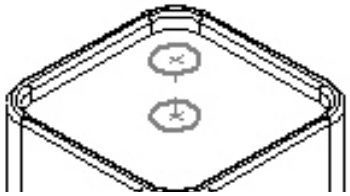
Create two holes. Delete the Terminals feature and reuse the circle sketch.

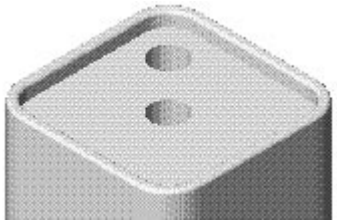
Delete and Edit Features.

60) Remove the Terminals (Extruded Boss) feature. Click **Edit** from the Main menu. Click **Delete**. Click **Yes** from the Confirm Delete dialog box. Do not delete the two-circle sketch, Sketch-TERMINALS.

61) Create an Extruded Cut feature from the two circles. Click **Sketch-TERMINALS** from the FeatureManager. Click **Extruded-Cut**. Click **Through All** for the Depth. Create the cut holes. Click **OK**.



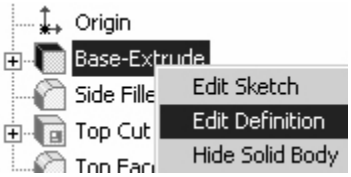


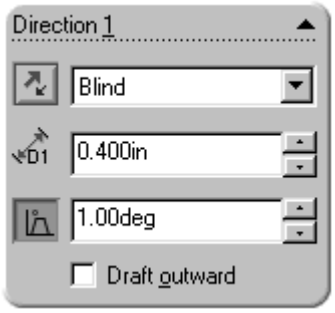


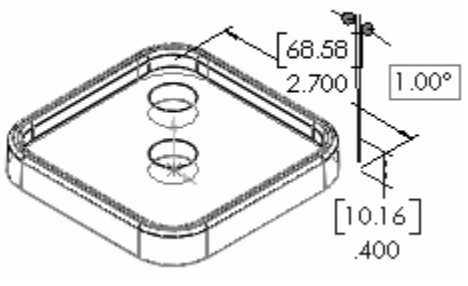
62) Rename **Cut-Extrude** to **Holes**.

63) Save the BATTERYPLATE. Click **Save**.

64) Edit the Base-Extrude feature. Right-click the **Base-Extrude** feature. Click **Edit Definition** from the Pop-up menu. Change the overall Depth to **.400**, [**10.16**]. Click the **Draft ON/OFF** button. Enter **1.00** in the Angle text box.



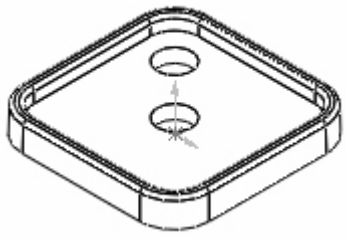




65) Display the modified Base feature. Click **OK**.

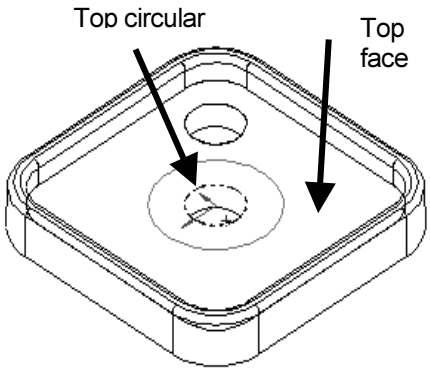
66) Save the BATTERYPLATE.

Click **Save**.



Create the BATTERYPLATE - Extruded Boss Feature

The Holder is created with a circular Extruded Boss feature.

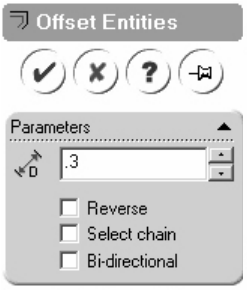


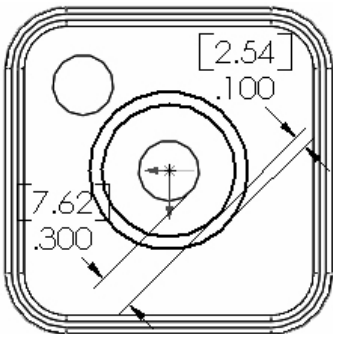
Create the Extruded Boss feature.

67) Select the Sketch plane. Click the **top face**.

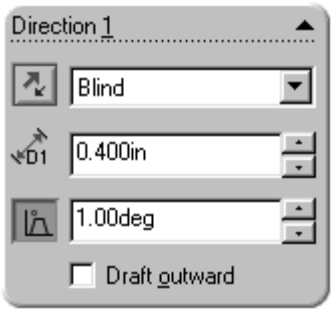
68) Create the Sketch. Click **Sketch**. Offset the center circular edge. Click the **top circular edge** of the center Hole feature.

69) Click **Offset Entities**. Enter **.300**, [**7.62**]. Click **OK**. Create the second offset circle. Select the first **offset circle**. Click **Offset Entities**. Enter **.100**, [**2.54**]. Click **OK**.





70) Extrude the Sketch. Click **Extruded Boss/Base**. Blind is the default Type option. Enter **.400**, [**10.16**] for Depth. Click the **Draft ON/OFF** button. Enter **1** in the Angle text box. Display the Extrude Boss feature. Click **OK**.

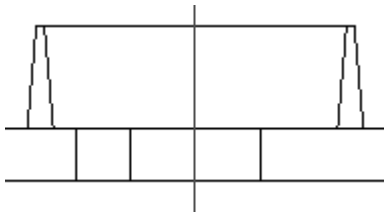


71) Rename **Boss-Extrude** to **Holder**.

72) Save the BATTERYPLATE. Click **Save**.

The outside face tapers inward and the inside face tapers outward when applying the Draft Angle to the two concentric circles.

Draft Angle displayed at 5 degrees



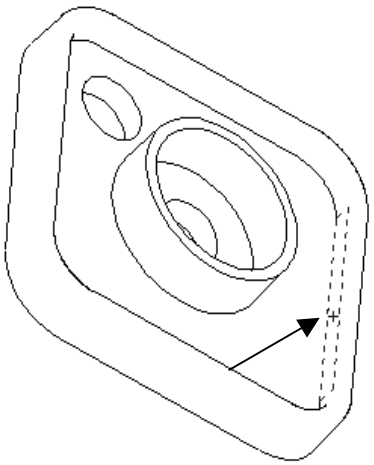
Create the BATTERYPLATE - Extruded Boss Feature

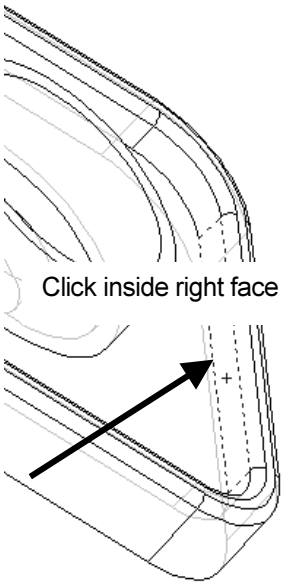
The next two Extruded Boss features are used to connect the BATTERY to the SWITCH. The first Sketch is extruded in two directions. The second Sketch is extruded in one direction. Both sketches utilize symmetry with the Origin and the Mirror Sketch Tool. The sketches utilize smaller dimensions than the current Grid Snap settings. Turn off the Snap to Points setting before you sketch the profiles.

Create the first Extruded Boss feature.
73) Zoom and **Rotate** the view to clearly display the inside right face.

Note: Press the arrow keys to rotate in 15-degree increments.

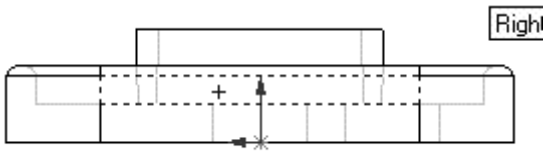
74) Select the Sketch plane. Click the **inside right face**.





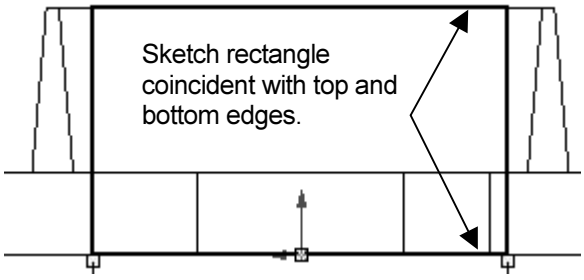
75) Create the Sketch. Click **Sketch** .

76) Turn the grid and snap off. Click **Grid** . Uncheck the **Display Grid** and **Snap to points** check box. Click **OK**.



77) Display the Right view. Click **Right** .

78) Sketch the profile. Click **Rectangle** . Sketch a **rectangle** coincident with the bottom and top edges. The Origin is approximately in the middle of the bottom line.



Geometric relationships are captured as you sketch.

The mouse pointer icon displays the following relationships: Horizontal , vertical , coincident , midpoint , intersection , tangent and perpendicular .

Note: If Automatic Relations are not displayed, Click Tools from the Main menu. Click Options, General, Automatic Relations in the Sketch box.

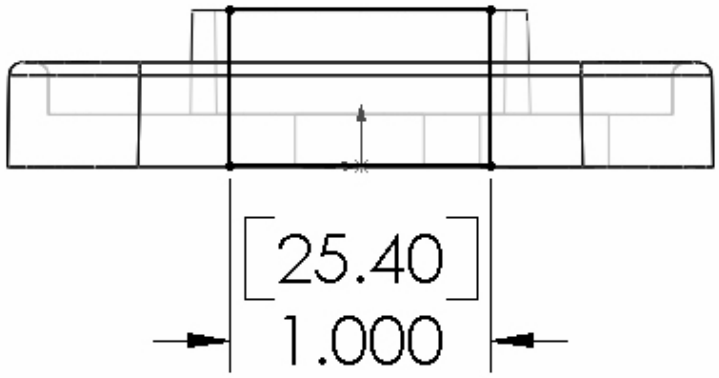
79) Add geometric relations. Click **Select** . Click **Origin** from the FeatureManager. Hold down the **Ctrl** key. Click the **bottom line** of the rectangle. Click **Midpoint** from the Relations dialog box. Release the **Ctrl** key. Click **Apply**.

The sketch is symmetric about the Origin.

80) Dimension the Sketch.

Click **Dimension** . Click the **bottom horizontal line**. Click the **text location**. Enter **1.000**, [**2.54**].

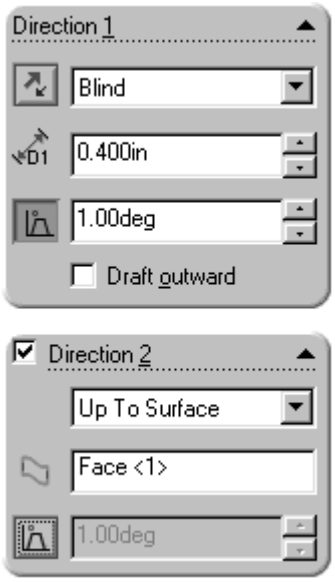
The black Sketch is fully defined.

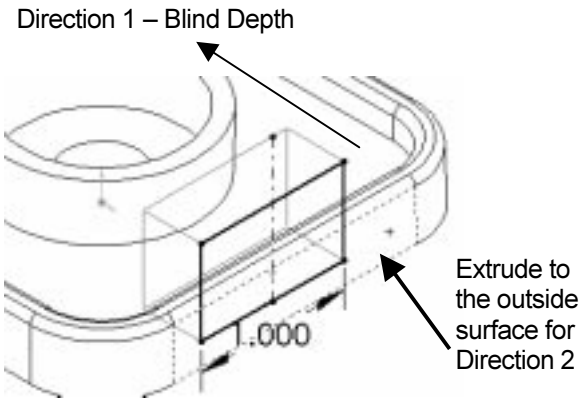


81) Display the Isometric view. Click **Isometric** .

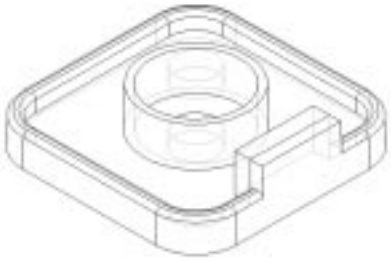
82) Extrude the Sketch. Click **Extruded Boss/Base** . Create the first depth direction, Direction 1. Blind is the Type option. Enter **.400**, [**10.16**] for Depth. Click the **Draft IN/OUT** button. Enter **1.00** for Draft Angle. The sketch is extruded towards the Holes.

83) Create the second depth direction. Click the **Direction 2** check box. Select **Up to Surface** for Type. Select the outside **right face** for the second extruded depth. The Selected Items text box displays Face<1>.





84) Display the Boss-Extrude2 feature. Click **OK**.



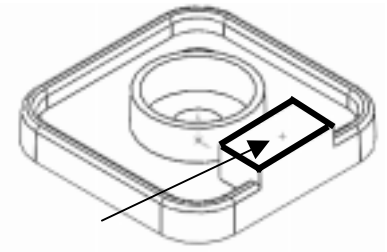
85) Rename **Boss-Extrude2** to **Connector Base**.

86) Show the Connector Base sketch. Click **Plus** to expand Connector Base in the FeatureManager. Right-click **Sketch5**. Click **Show Sketch**.

87) Save the BATTERYPLATE. Click **Save** .

Create the second Extruded Boss feature.

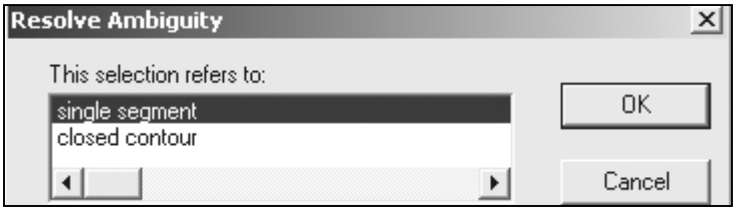
88) Select the Sketch plane. Click the **top narrow face** of the first Extruded Boss feature.



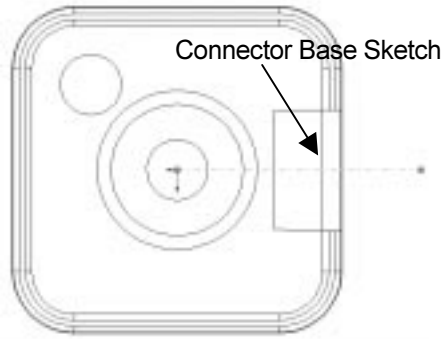
89) Create the Sketch. Click **Sketch** . Display the Top view. Click **Top** .

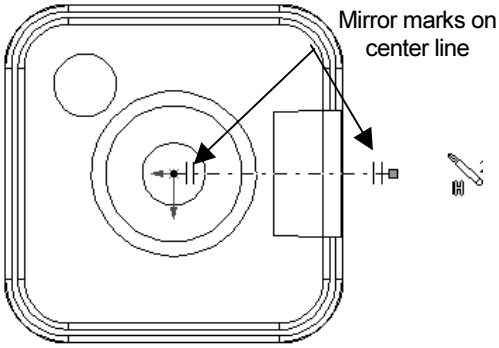
Convert a line segment from the Connector Base sketch to the current sketch plane.

90) Click the **vertical line** of the Connector Base Sketch. Click **Convert Entities** . Select **single segment**. Click **OK**.

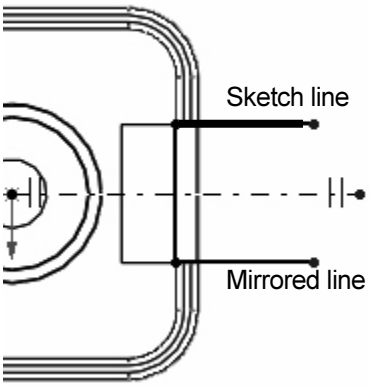


91) Sketch the centerline. Click **Centerline** . Sketch a **horizontal centerline** with the first point coincident to the Origin . Create the mirrored centerline. Click **Mirror** . The centerline displays two parallel mirror marks.

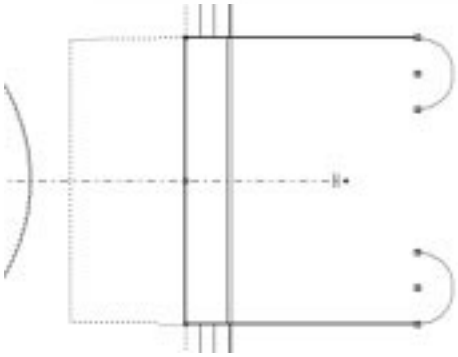




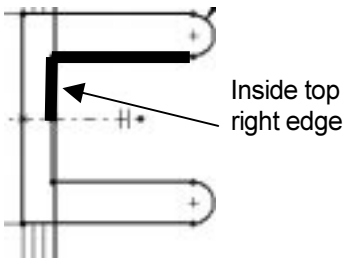
92) Sketch the profile. Create the Sketch on one side of the mirror centerline. Click **Line** . Create a **horizontal line** coincident with the endpoint of the converted Connector Base sketch. The line is automatically mirrored.



93) Create a Tangent Arc. Click **Tangent Arc** . Create the first arc point. Click the **endpoint** of the horizontal line. Create a 180° arc. Drag the **mouse pointer** downward until the start point, center point and end point are vertically aligned. Release the **mouse button**.



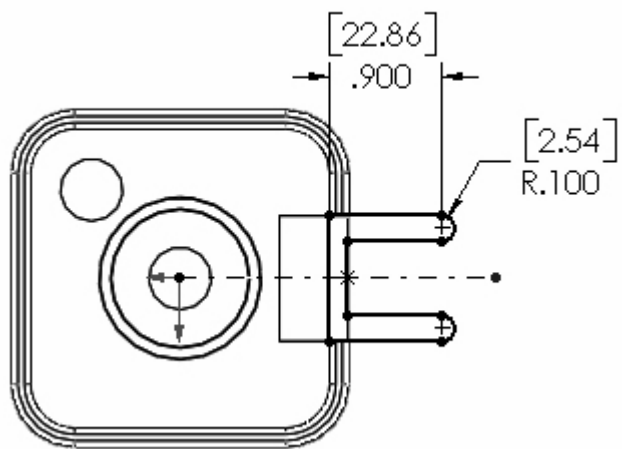
94) Complete the Sketch. Click **Line** . Create a **horizontal line**. Create a **vertical line** coincident with the inside top right edge.



95) Turn the Mirror function off. Click **Mirror** .

96) Dimension the Sketch.
Create a radial dimension.

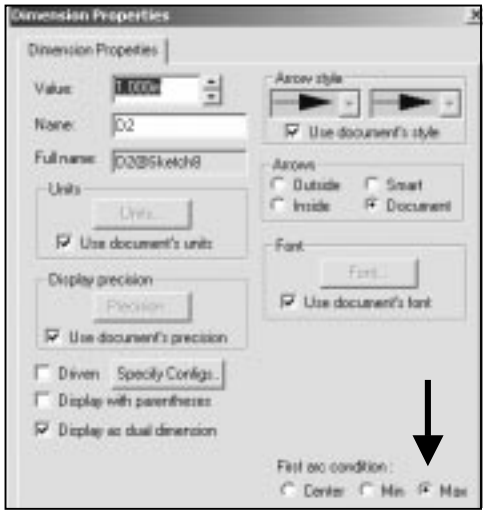
Click **Dimension** . Click the **arc edge**. Click the **text location**. Enter **.100**, [**2.54**] for the Radius.

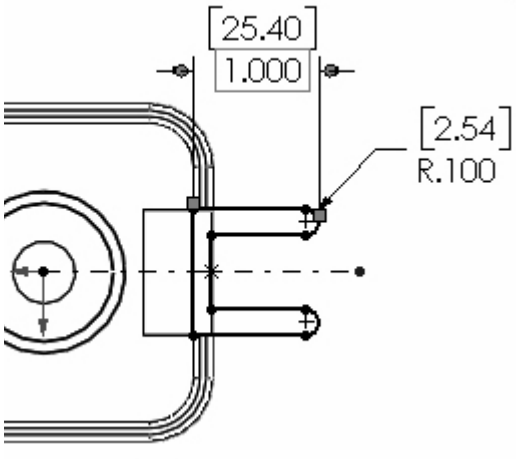


97) Create a linear dimension. Click the left most **vertical line**. Click the **arc edge**. The arc edge displays red. Click the **text location**.

Note: Click the arc edge, not the arc center point to create a max. dimension. The linear dimension uses the arc center point as a reference. Modify the Properties of the dimension. The Maximize option references the outside tangent edge of the arc.

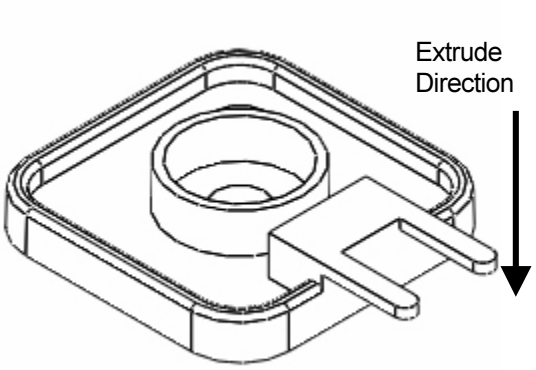
98) Right-click on the **dimension text**. Click **Properties** from the Pop-up menu. Click the **Max** button from the First arc condition option. Enter **1.000**, [**25.4**] in the Value list box. Display the dimension. Click **OK**. The black Sketch is fully defined.





99) Display the Isometric view. Click **Isometric** .

100) Extrude the Sketch. Click **Extruded Boss/Base** . Blind is the default Type option. Enter **.100**, [**2.54**] for Depth. Click the **Reverse** check box. Display the Boss-Extrude3 feature. Click **OK**.



101) Rename **Boss-Extrude3** to **ConnectorSwitch**.

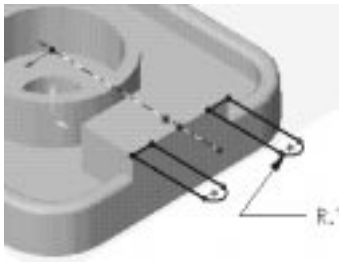
102) Save the BATTERYPLATE. Click **Save** .

Disjoint Geometry

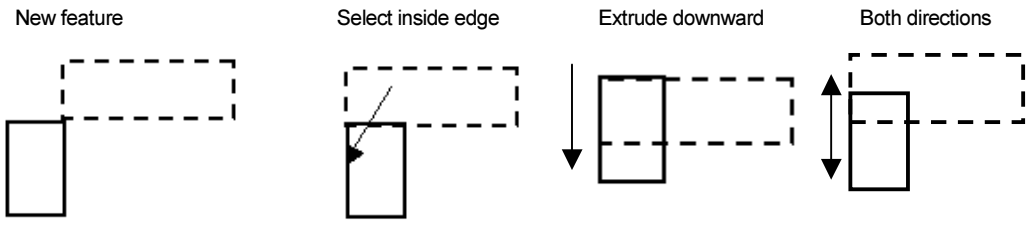
Incorrect selection of edges and faces leads to disjointed bodies. Disjointed bodies occur when the geometry contains gaps.

Example: Create a Sketch from the outside edge. Reverse the extrusion direction to create disjoint geometry.

The feature is not created and a Rebuild error is displayed.



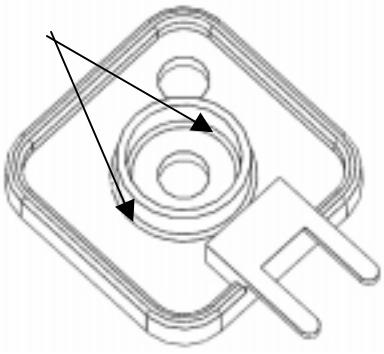
Profiles of disjointed and joined geometry are displayed.



Disjoint profile geometry Joined profile geometry Joined profile geometry

Create the BATTERYPLATE - Edge and Face Fillets

Both edge and face options for Fillet features are used to smooth rough edges.



Create a Fillet feature.

103) Create a fillet on the inside and outside edge of the Holder. Create a fillet on all inside tangent edges of the Top-Cut. Click **Fillet** . Enter **.050**, [**1.27**] for Radius. Click the **outside circular edge** of the Holder. Click the **inside circular edge**. Display the Fillet. Click **OK**.

104) Rename **Fillet3** to **HolderFillet**.

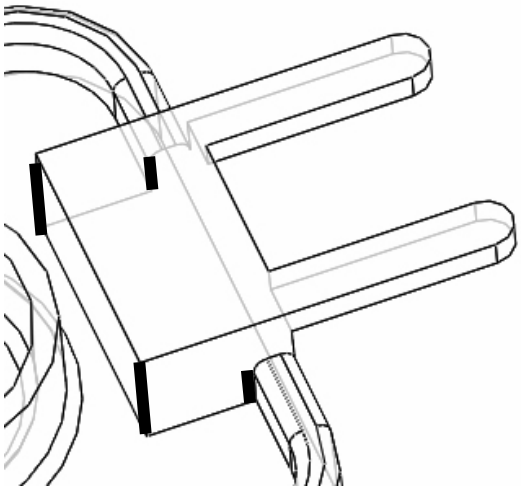
Create a Fillet on the outside bottom edge of the Connector. This is a two-step process:

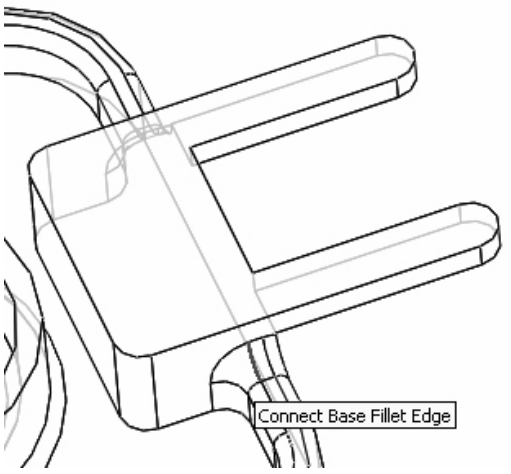
- Create an edge Fillet
- Create a face Fillet

Create an edge Fillet on the four vertical edges of the Connector. Create a face blend between two sets of faces.

Create the edge Fillet feature.

105) Click **Fillet** . Click the four **vertical edges**. Enter **.100**, [**2.54**] for Radius. Display the Fillet. Check **Tangent Propagation**. Click **OK**.

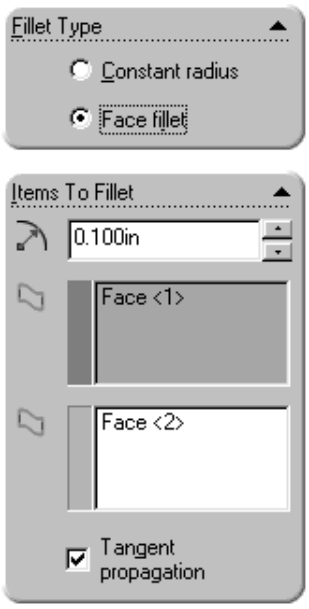


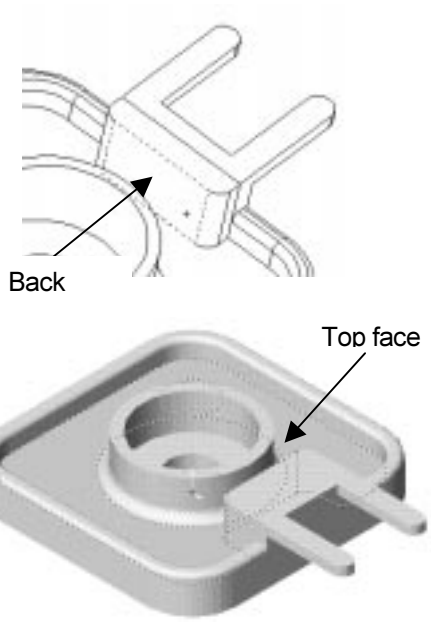


106) Rename **Fillet4** to **Connect Base Fillet Edge**.

Create the face Fillet feature.

107) Click **Fillet** . Click **Face fillet** button from the Fillet type list box. Enter **.100**, [**2.54**] for Radius. Select the first Face set. Click the **back face**. Select the second Face set. Click **inside** the Face Set 2 list box. Click the **top face** of the Base-Extrude feature. Click **OK**. The Radius is too large. Enter **.050**, [**1.27**] for Radius. Display the Fillet. Click **OK**.

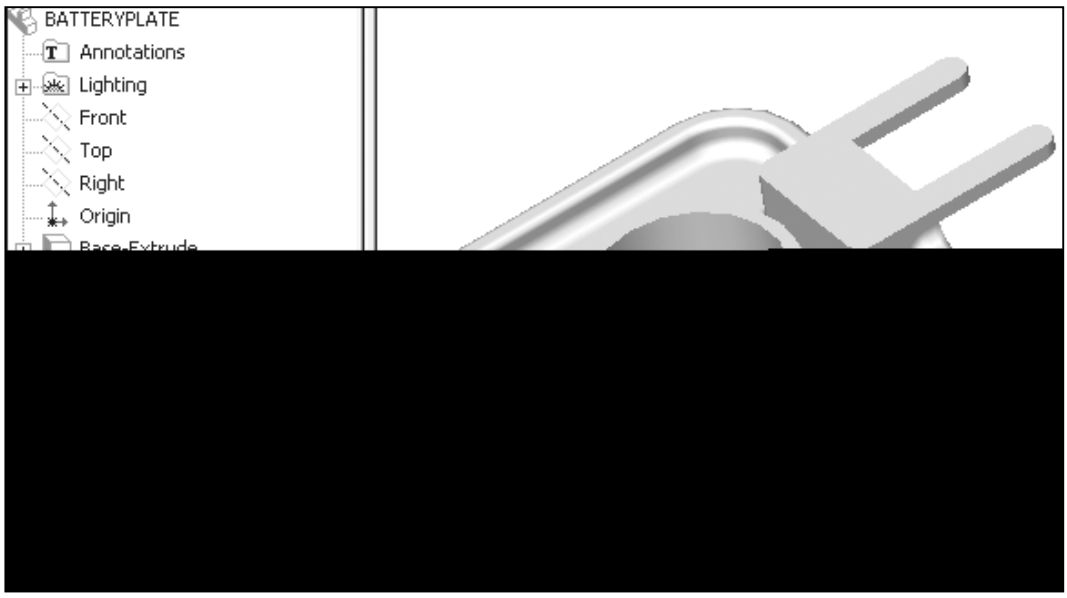




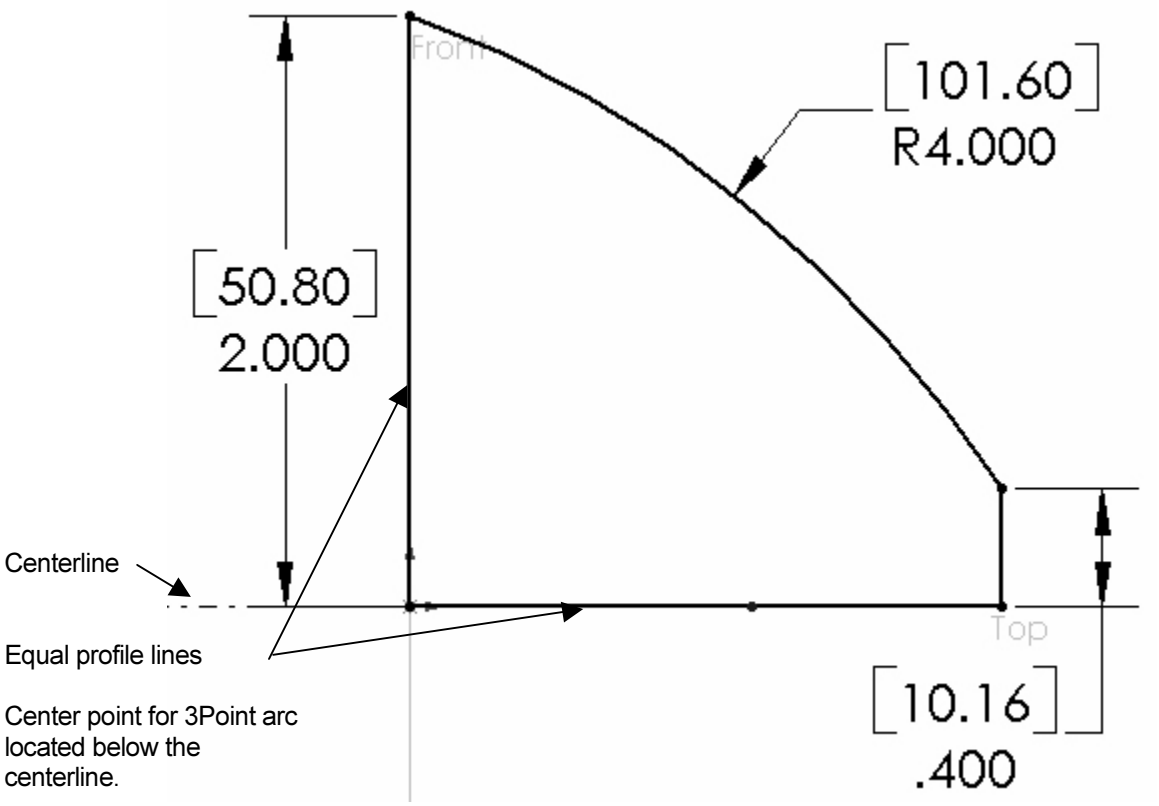
108) Rename **Fillet5** to **Connect Base Fillet Face**.

109) Save the BATTERYPLATE. Click **Save** .

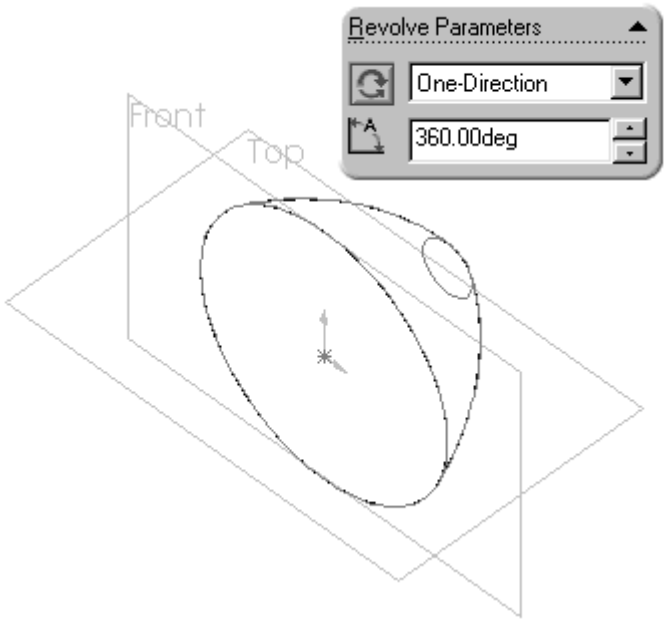
The FeatureManager displays all successful feature name icons in yellow. The rotation of the BATTERYPLATE is completed.



119) Add dimensions. Click **Dimension**. Create a vertical linear dimension for the **left line**. Enter **2.000**, [50.8]. Create a vertical linear dimension for the **right line**. Enter **.400**, [**10.16**]. Create a radial dimension for the **arc**. Enter **4.000**, [**101.6**]. The black Sketch is fully defined.



120) 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**.



121) Save the LENS. Click **Save**.

Revolve features contain an axis of revolution.

The axis is critical to align other features.

122) 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.

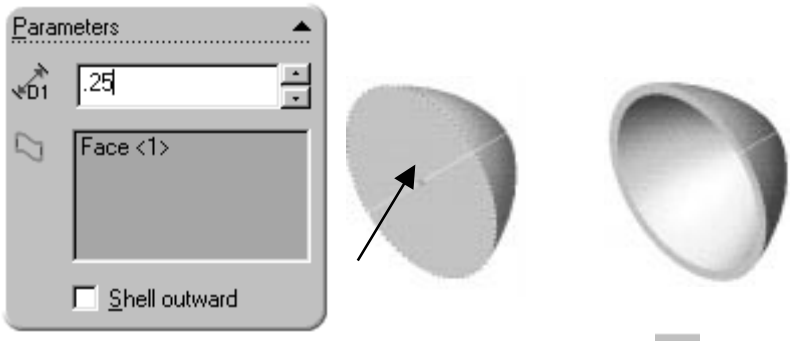
Solid Revolve features must contain a closed profile. Each revolved profile requires an individual sketched centerline.

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.

123) Select the face. Click the **front face** of the Base-Revolve feature. Click **Shell** from the Feature toolbar. Enter **.250**, [**6.35**] in the Thickness text box. Display the Shell feature. Click **OK**.



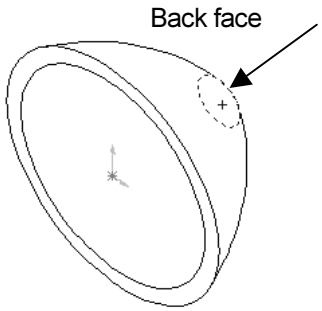
124) Rename **Shell1** to **LensShell**. Save the LENS. Click **Save**.

Create the LENS - Extruded Boss Feature

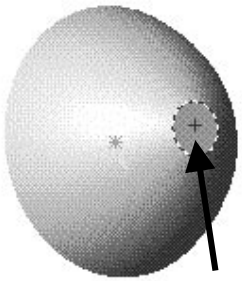
Create the LensNeck. Use the Extruded-Boss feature. The LensNeck houses the BULB base and is connected to the BATTERY PLATE. The feature extracts the back circular edge from the Base-Revolve feature.

Create the Extruded Boss feature.

125) 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).



126)Rotate the part to view the back face.



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

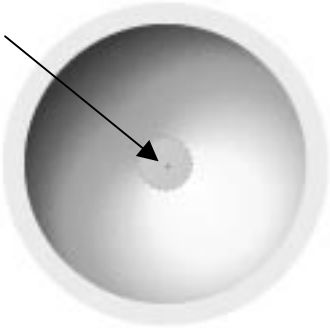
128) Extrude the Sketch. Click **Extrude Boss/Base** . Enter **.400**, [**10.16**] for Depth. Display the Boss-Extrude1 feature. Click **OK**.

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

130) 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. Dimensions for the Counterbore Hole are provided both in inches and millimeters.

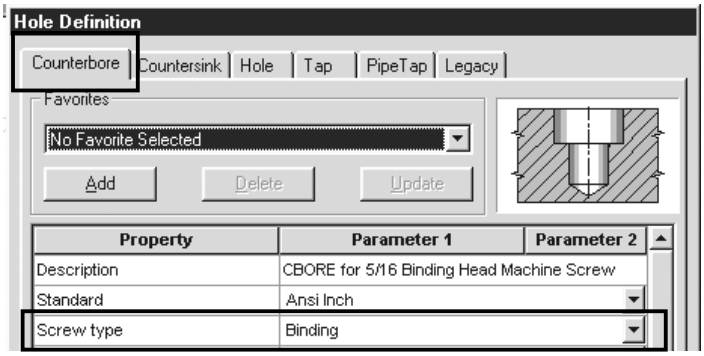


Create the Counterbore Hole.

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

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

133) 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.



Note: For a metric hole, skip the next step.

For Inch Cbore Hole:

134) Select **Ansi Inch** for Standard. Enter **Hex Bolt** from the drop down list for Screw type. Select **½** 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 **.600**. Click the **C-Bore Depth** value. Enter **.200**.

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	.2

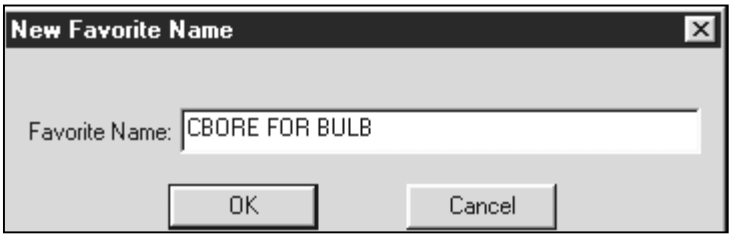
Note: For an inch hole, skip the next step.

For Millimeter Cbore Hole:

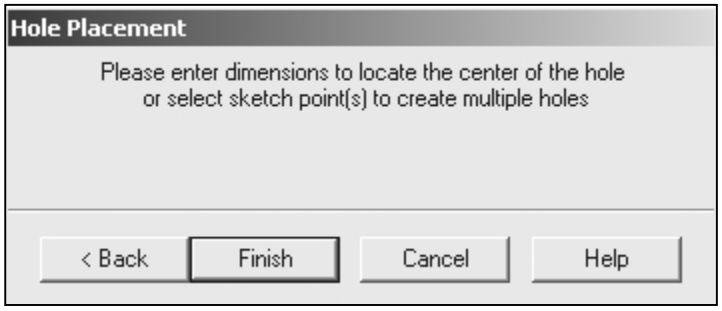
135) Select **Ansi Metric** for Standard. Enter **Hex Bolt** from the drop down list for Screw type. Select **M5** from the drop down list for Size. Click **Through All** from the drop down list for End Condition & Depth. Click the Hole Diameter value. Enter **13.5**. Click the **C-Bore Diameter** value. Enter **15.24**. Click the **C-Bore Depth** value. Enter **5**.

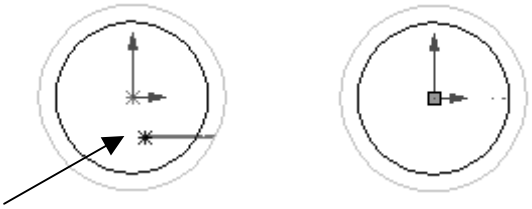
Property	Parameter 1	Parameter 2
Description	CBORE for M5 Hex Head Bolt	
Standard	Ansi Metric	
Screw type	Hex Bolt	
Size	M5	
End Condition & Depth	Through All	16.510mm
Selected Item & Offset		16.510mm
Hole Fit & Diameter	Normal	13.500mm
Angle at Bottom	118deg	
C'Bore Diameter & Depth	15.240mm	5.000mm

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

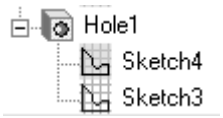


137) Click **Next** from the Hole Definition dialog box. The Hole Placement dialog box is displayed. Position the hole coincident with the Origin. Click **Add Relations**. Click the **center point** of the Counterbore hole. Click the **Origin**. Click **Coincident**. Complete the hole. Click **Finish** from the Hole Wizard.

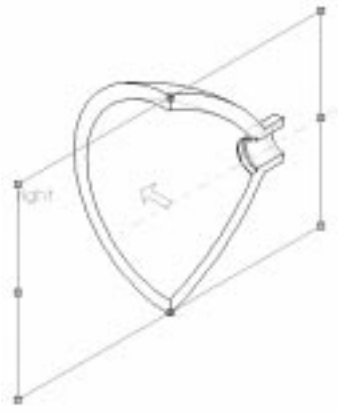




138) Expand the Hole. Click the **Plus Sign** to the left of the Hole feature. Sketch3 and Sketch4 are used to create the Hole feature.



139) 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**.



140) Display the Full view. Click **View**, **Display**, **SectionView**.

141) Display the Temporary Axis. Click **View**, **Temporary Axis**.

142) Rename **Cbore Hex Head Bolt** to **BulbHole**.

143) Save the LENS. Click **Save**.

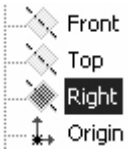
Create the LENS - Boss Revolve Thin Feature

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.

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

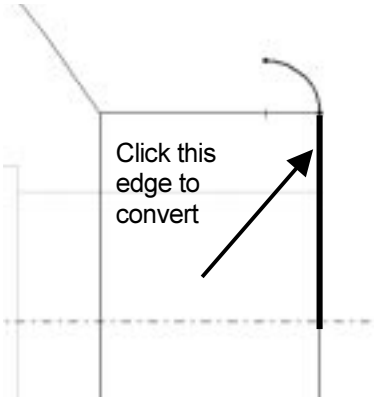
Create the Boss Revolve Thin feature.

144) Select the Sketch plane. Click the **Right** plane. Create the Sketch. Click **Sketch**.



145) Display the Right view. Click **Right**.

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

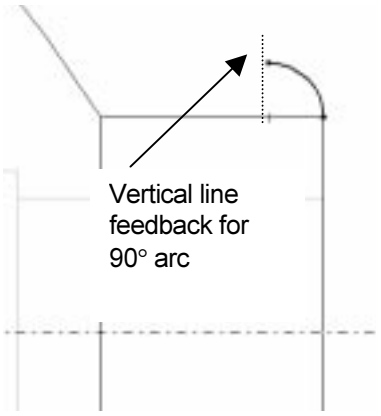


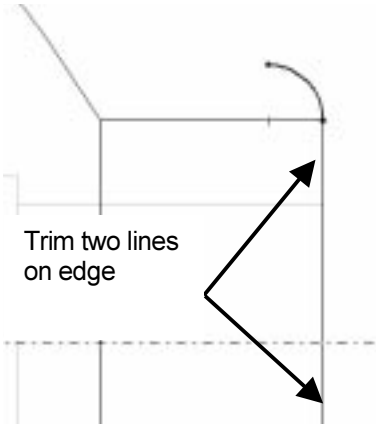
147) Select the right edge. **Right-click** in the Graphics window. Click **Select** from the Pop-up menu. Click the **right edge** of the Base feature.

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

149) 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 **left mouse button**.

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



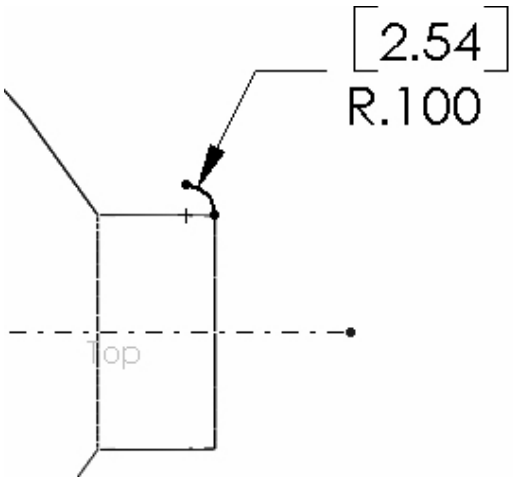


150) The vertical line segments are required to create the Tangent Arc. Remove the two line segments.

Click **Trim** . Click **both vertical edges**. The Sketch consists of an arc and a centerline.

151) Add a dimension. Click **Dimension** . Create a radial dimension. Enter **.100**, [**2.54**].

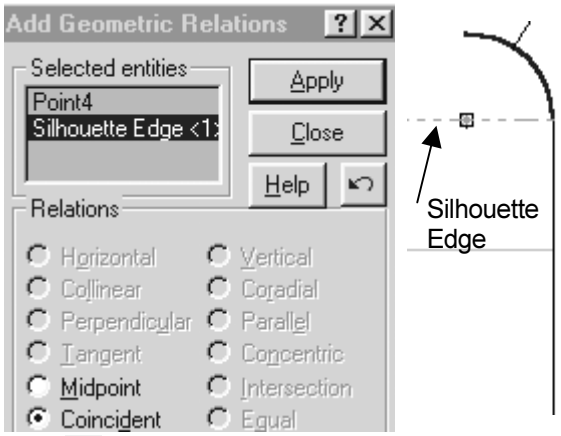
The sketch arc requires a coincident relationship. This insures that the center point of the arc is coincident with the horizontal silhouette edge of the Base-Revolve feature.



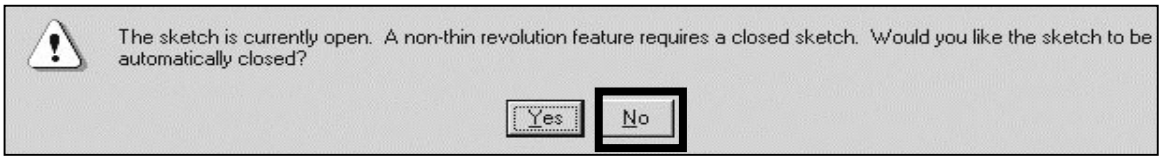
152) Add geometric relations.

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.



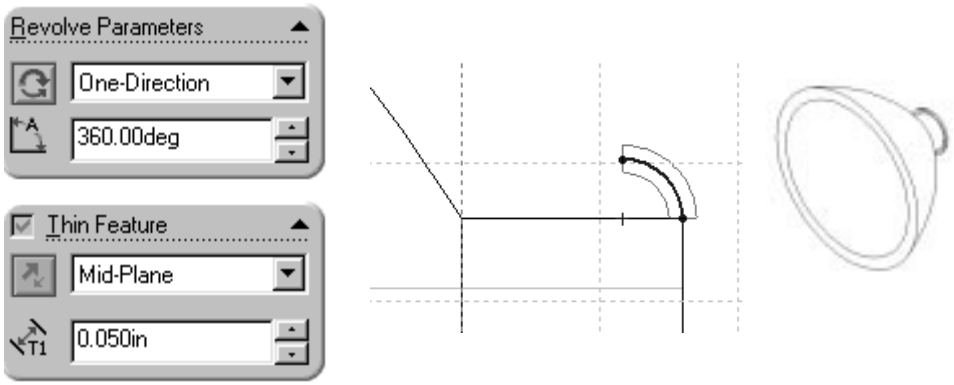
153) Revolve the Sketch. Click **Revolve** . A warning message appears:



154) Keep the Sketch open. Click **No**. A second warning message appears:

155) Click **OK**. The Thin Feature check box is active.

156) Create the Thin-Revolved feature on both sides of the Sketch. Select **Mid-Plane** from the Type list box. Enter **.050**, [**1.27**] for Wall Thickness. Display the Boss-Revolve-Thin1 feature. Click **OK**.



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

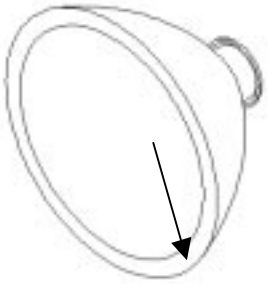
158) Save the LENS. Click **Save** .

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 LENSCAP.

Create the Extruded Boss feature.

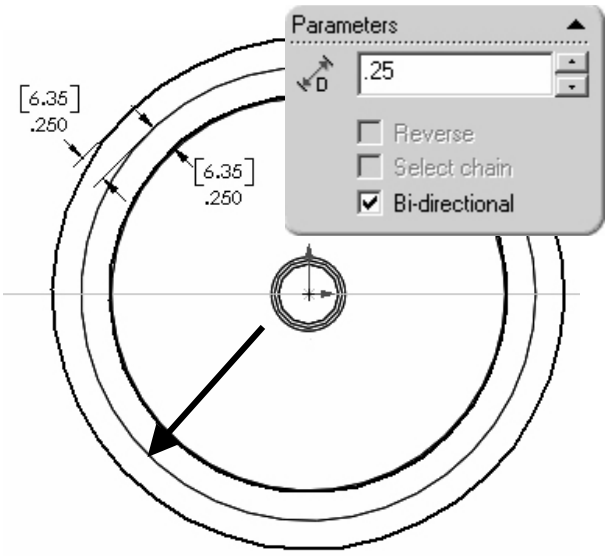
159) Select the Sketch plane. Click the **front circular face**.



160) Create the Sketch. Click **Sketch** .

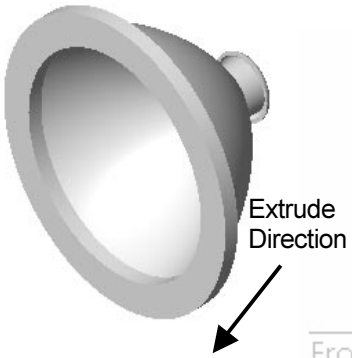
161) Display the Front view. Click **Front** .

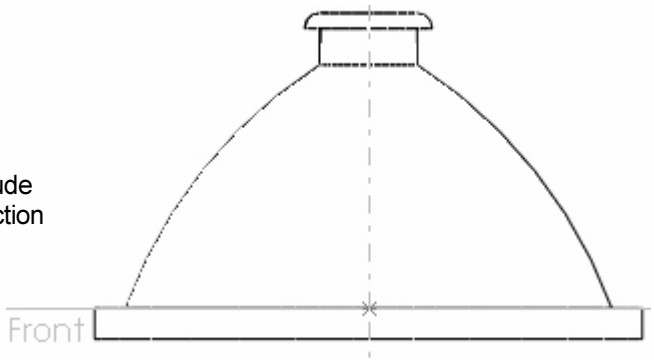
162) Click the **outside circular edge**. Click **Offset Entities** . Click the **Bi-directional** check box. Enter **.250**, [**6.35**].



163) Extrude the Sketch. Click **Extrude Boss/Base** . Enter **.250**, [**6.35**] for Depth. Display the Boss-Extrude feature. Click **OK**.

164) Verify the position of the Boss Extrude. Click the **Top** view.





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

166) 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.

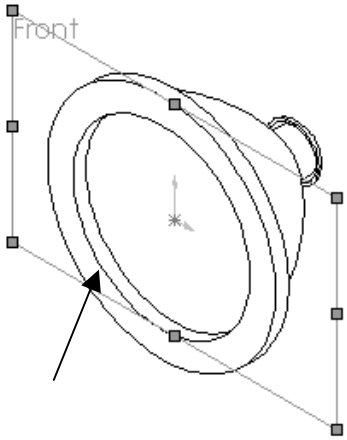
Create the Extruded Boss feature.

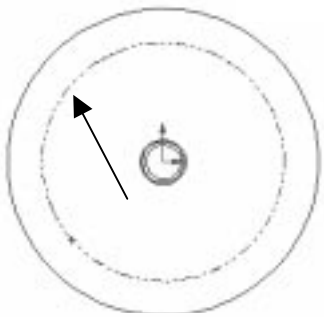
167) Select the Sketch plane. Click the **Front** plane.

168) Create the Sketch. Click **Sketch** .

169) Display the Front view. Click **Front** .

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

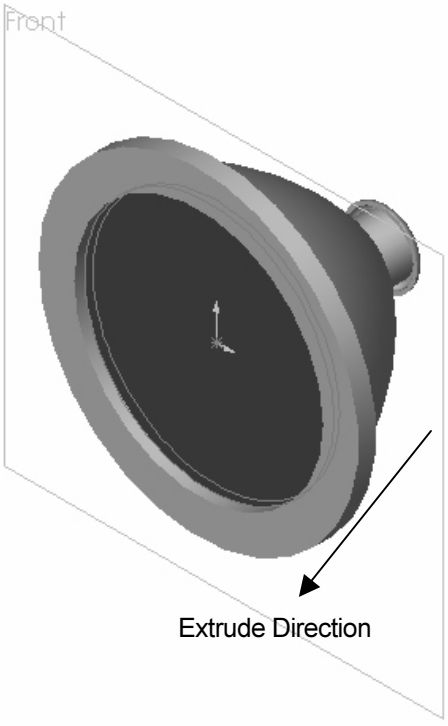




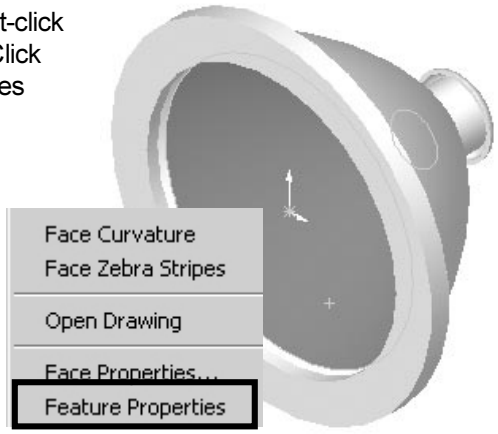
171) Extrude the Sketch. Click **Extruded Boss/Base** . Enter **.100**, [**2.54**] for Depth. Click **OK**.

Note: If you select the inside circular edge on Plane1, you will create a disjoint feature.

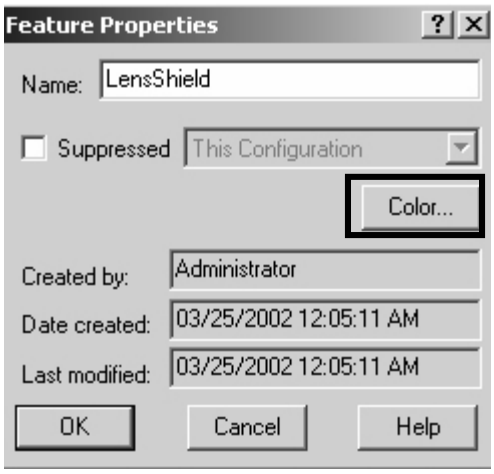
172) Rename **Boss-Extrude3** to **LensShield**.

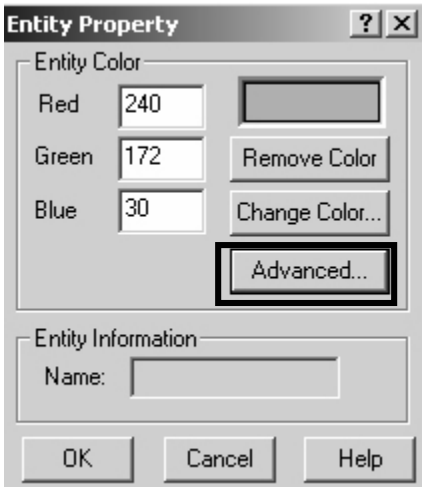


173) Add transparency to the LensShield. Right-click the **LensShield** in the Graphics window. Click **Feature Properties**. The Feature Properties dialog box is displayed.

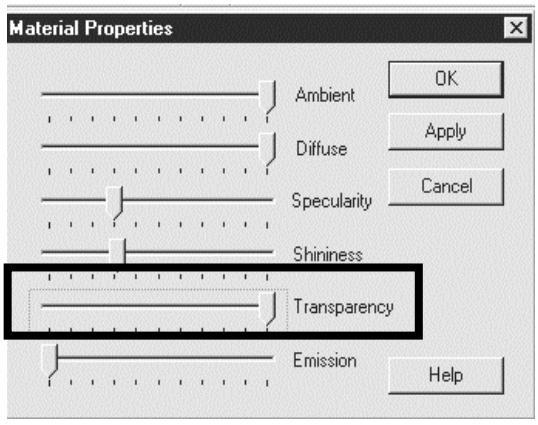


174) Click the **Color** button. The Entity Property dialog box is displayed. Click the **Advanced** button.





175) 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.

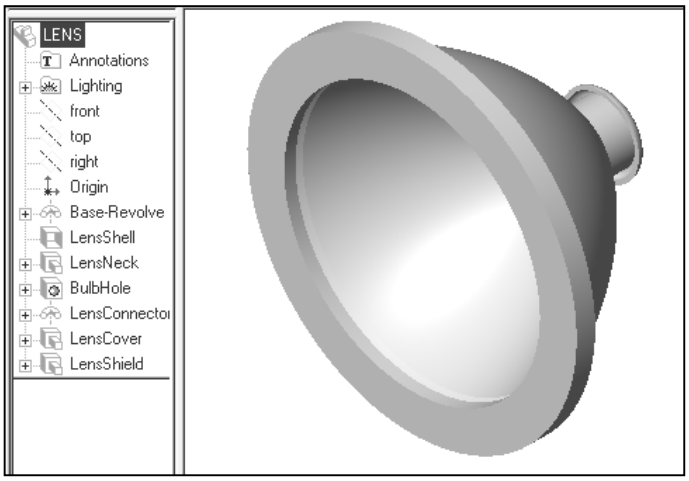


176) Display the transparent faces.

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

177) 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 4.20a.

Create a Revolved Boss feature using a B-Spline sketched profile. A B-Spline is a complex curve, Figure 4.20b.

Create a Revolved Cut Thin feature at the base of the BULB, Figure 4.20c.

Create a Dome feature at the base of the BULB, Figure 4.20d.

Create a Circular Pattern feature from an Extruded Cut, Figure 4.20e.

Figure 4.20a 4.20b 4.20c 4.20d 4.20e

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.

178) Create the BULB. Click **New**. Click **PartEnglishTemplate**, [**PARTMETRICTEMPLATE**]. Click **OK**. Click **Save**. Enter the name of the part. Enter **BULB**. Click **Save**.

179) Select the Sketch plane. Click the **Right** plane. Create the Sketch. Click **Sketch**.

180) Show the three planes. Hold down the **Ctrl** key. Click **Front**, **Top** and **Right** from the FeatureManager. Right-click **Show**. Release the **Ctrl** key.

181) Display the Right view. Click **Right**.

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

183) Sketch the profile. Create six lines. Click **Line**.

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

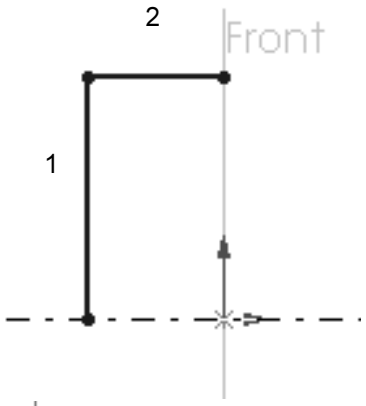
Create the second line. Sketch a **horizontal line** with the endpoint coincident to the Front plane.

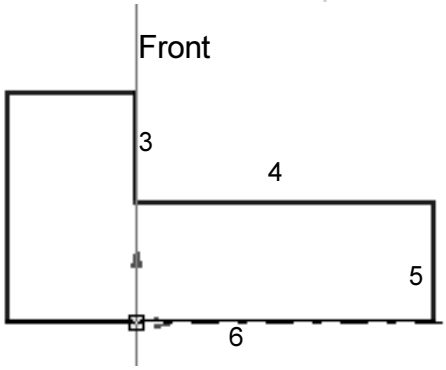
Create the third line. Sketch a short **vertical line** towards the centerline, collinear with the Front plane.

Create the forth 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**.





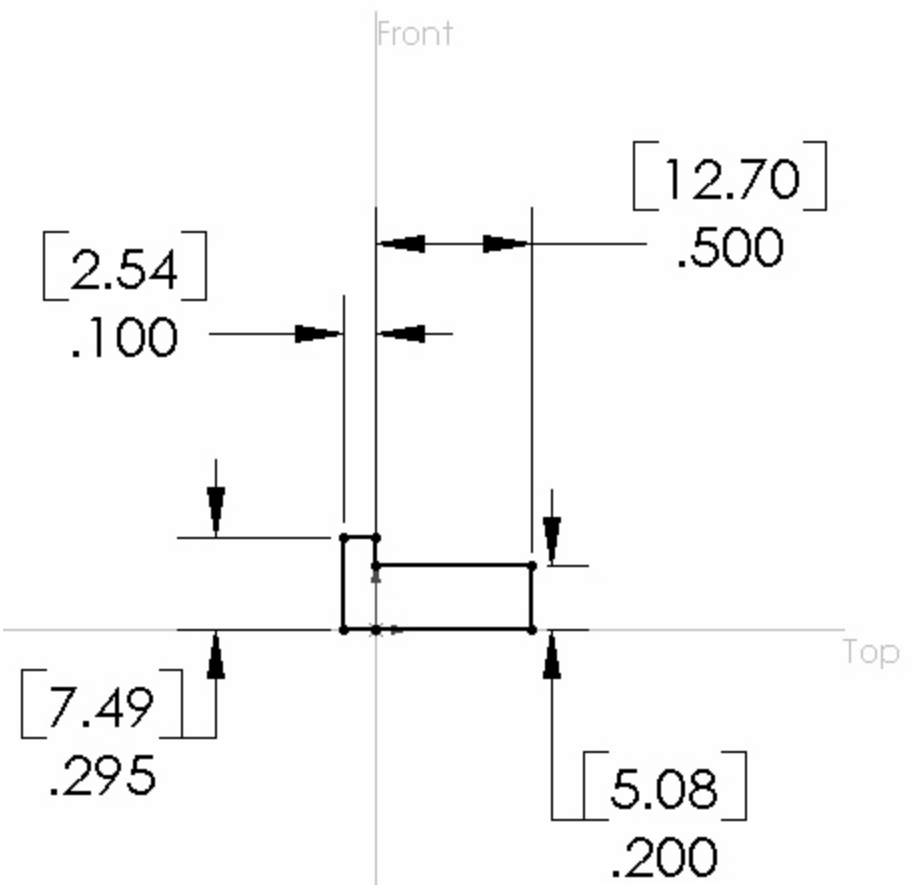
184) Add dimensions. Click **Dimension** .

Create a vertical linear dimension. Click the **right line**. Enter **.200**, [**5.08**].

Create a vertical linear dimension. Click the **left line**. Enter **.295**, [**7.49**].

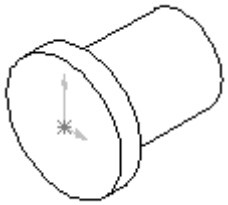
Create a horizontal linear dimension. Click the **top left line**. Enter **.100**, [**2.54**].

Create a horizontal linear dimension. Click the **top right line**. Enter **.500**, [**12.7**].



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

186) Save the BULB. Click **Save**.



Create the BULB - Revolved Boss Feature

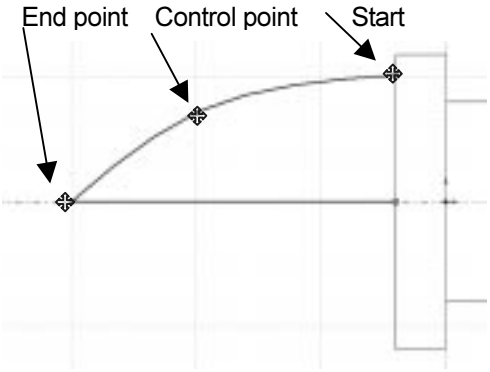
The bulb requires a second solid Revolve feature. The profile utilizes a complex curve called a B-Spline (Non-Uniform Rational B-Spline or NURB). B-Splines are drawn with control points. Adjust the shape of the curve by dragging the control points.

Create the Revolved Boss feature.

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

188) Select the Sketch plane. Click the **Right** plane. Create the Sketch. Click **Sketch** . Display the Right view. Click **Right** .

189) 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.

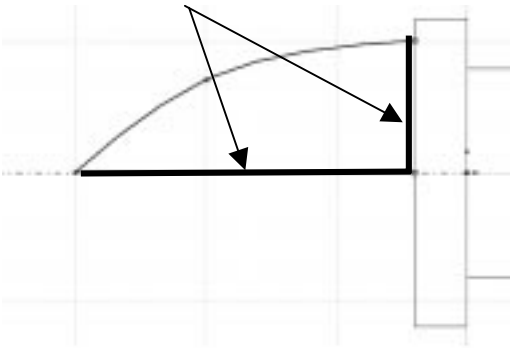
Sketch the control point. Drag the **mouse pointer** to the left of the Base feature and below the first point. Release the **left mouse** button.

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

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

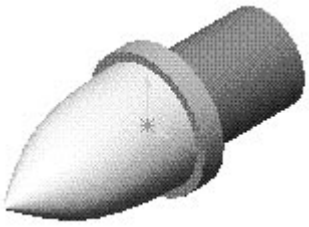
Note: SolidWorks does not require dimensions to create a feature.

Horizontal and Vertical lines



191) 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.

192) 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**.



193) Save the BULB. Click **Save**.

Create the BULB - Revolved Cut Thin Feature

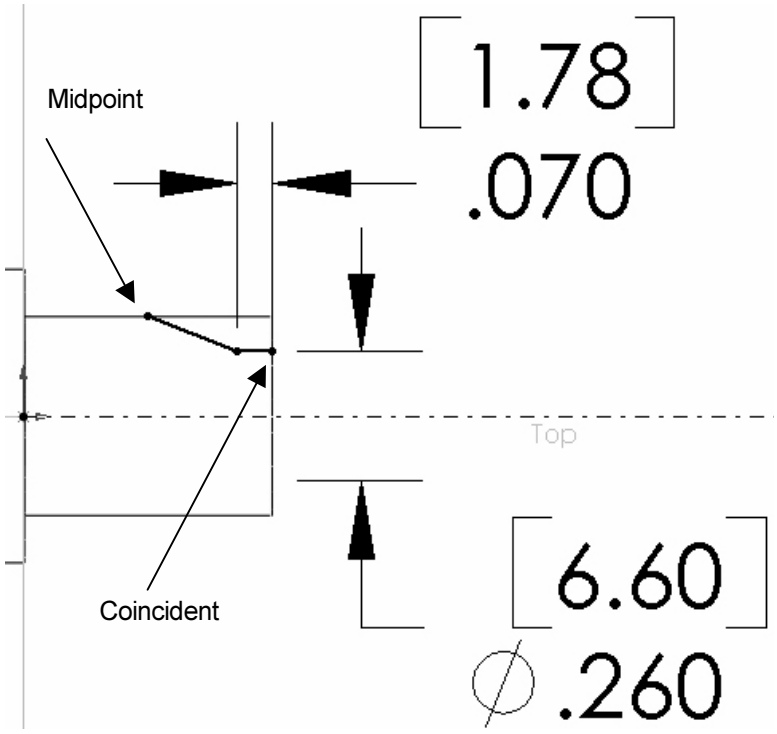
A Revolved Cut Thin feature removes material by rotating an open sketch profile around a centerline.

Create the Revolved Cut Thin feature.

194) Select the Sketch plane. Click the **Right** plane. Create the profile. Click **Sketch** . Display the Right view. Click **Right** .

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

196) 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 .260, [6.6] end point coincident with the vertical **right edge**.



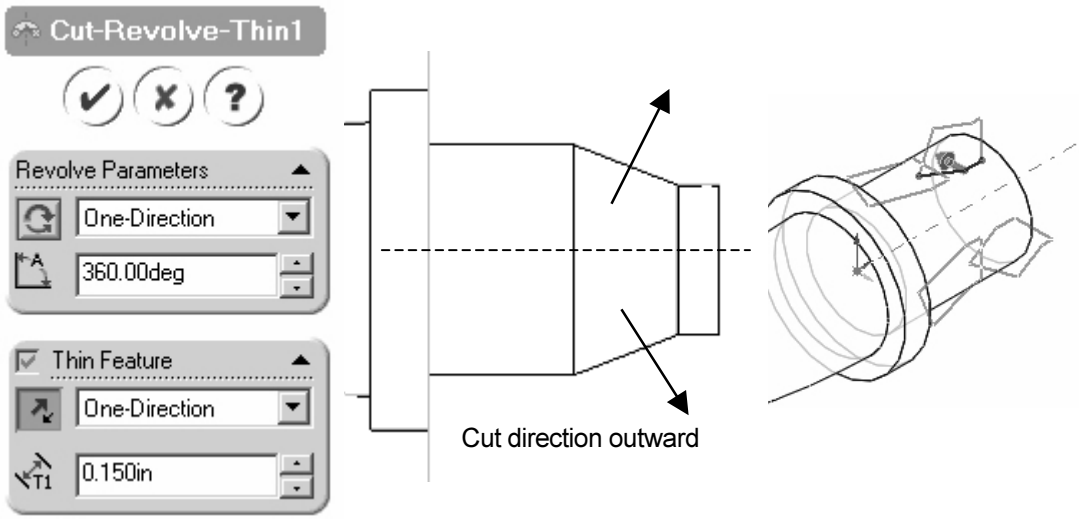
197) Add relations. Hold down the **Ctrl** key. Click the **start point** of the line. Click the top **Silhouette edge**. Release the **Ctrl** key. Click the **Midpoint** button. Click **OK**. Hold down the **Ctrl** key. Click the **end point** of the line. Click the right **vertical edge**. Release the **Ctrl** key. Click the **Coincident** button. Click **OK**.

198) Add dimensions. Click **Dimension** . Create the diameter dimension. Click the **centerline**. Click the **short horizontal line**. Enter **.260**, [**6.6**]. Add a horizontal dimension. Click the **short horizontal line**. Enter **.070**, [**1.78**]. The black Sketch is fully defined.

Note: The ∅.260 is displayed as a diameter dimension. Right-click Properties, uncheck the Display diameter check box to display a radius value.

199) Revolve the Sketch. Click **Revolved Cut** from the Feature toolbar. Click **No** to the Warning Message, "Would you like the sketch to be automatically closed?" Click **OK** to the Warning Message, "The profile is only suitable for a thin feature".

200) The Cut Revolve Thin Feature dialog box is displayed. The direction arrow points away from the centerline. Click the **Direction** button. Enter **.150**, [**3.81**] for Thickness. Display the Revolved Cut Thin feature. Click **OK**.



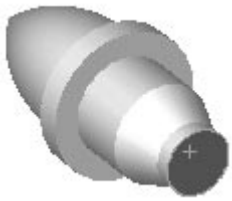
201) Save the BULB. Click **Save**.

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.

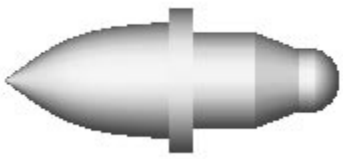
Create the Dome feature.

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



203) Click **Insert** from the Main menu. Click **Features**, **Dome**. The Dome dialog box is displayed. Enter **.100**, [**2.54**] for Height. Display the Dome. Click **OK**.

204) Save the BULB. Click **Save**.



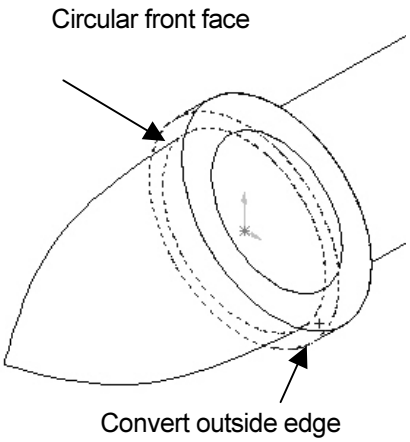
Create the BULB - Circular Pattern

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.

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

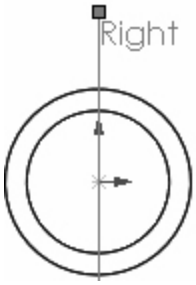


206) Create the Sketch. Click **Sketch** .

207) Extract the outside circular edge. Click **Select** . Click the **outside circular edge**. Click **Convert Entities** .

208) Display the Front view. Click **Front** .

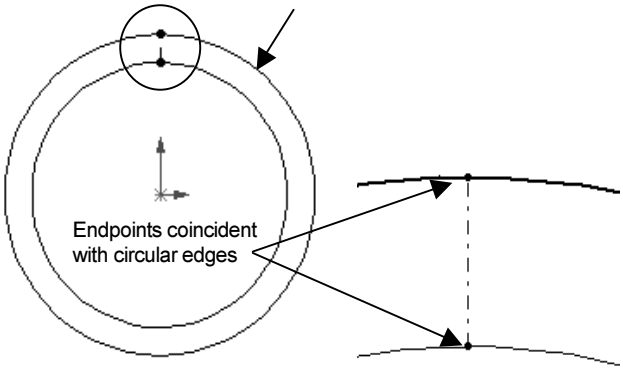
209) Show the Right plane. Click the **Right** plane in the FeatureManager. Right-click **Show**.



210) Sketch the centerline.

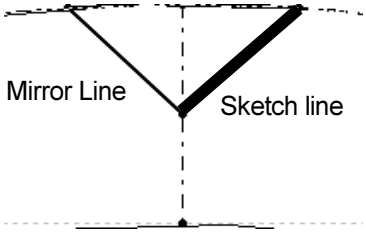
Click **Centerline** . Sketch a **vertical centerline** coincident with the top and bottom circular circles and coincident with the Right plane.

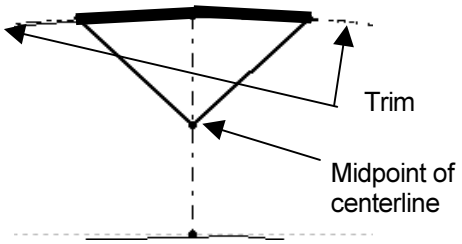
211) **Zoom** to display the centerline and the outside circular edge.



212) 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 the Mirror off.



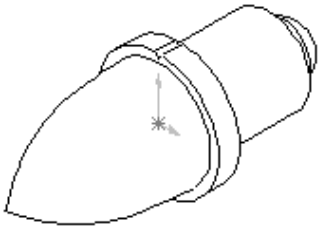


Click **Mirror**.

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

214) Add the geometry relations. Hold down the **Ctrl** key. Click the **two lines**. Click the **Perpendicular** button. Release the **Ctrl** key. The black Sketch is fully defined.

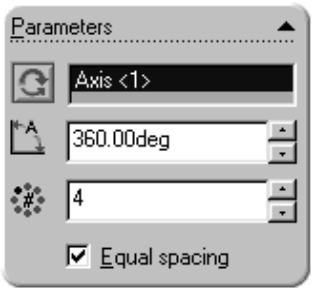
215) Extrude the Sketch. Click **Extruded Cut**. Click **Up to Next** from the Type list box. Display the Extruded Cut. Click **OK**.

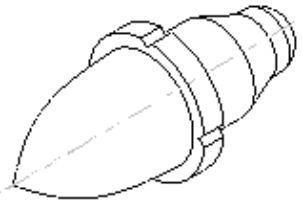


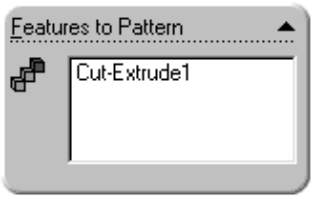
216) Display the Temporary axis. Click **View**, **Temporary Axis** from the Main menu.

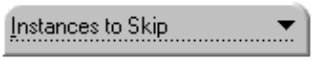
The Cut-Extrude is the seed feature for the Pattern.

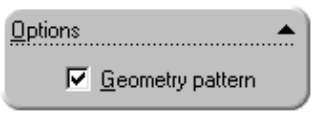
217) Create the Pattern. Click the **Cut-Extrude** feature. Click **Circular Pattern**. The Circular Pattern dialog box is displayed. Click the **Direction selected** text box. Click **Temporary Axis**. Create 4 copies of the Cut. Enter **4** in the Total Instances spin box. Click the **Equal spacing** check box. Click the **Geometry pattern** check box. Display the Pattern feature. Click **OK**.











218) 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**.

219) Hide the Temporary axis. Click **View** from the Main menu. Click **Temporary Axis**. Hide the Planes. Click **Planes** from the View menu.

220) Save the BULB. Click **Save**.

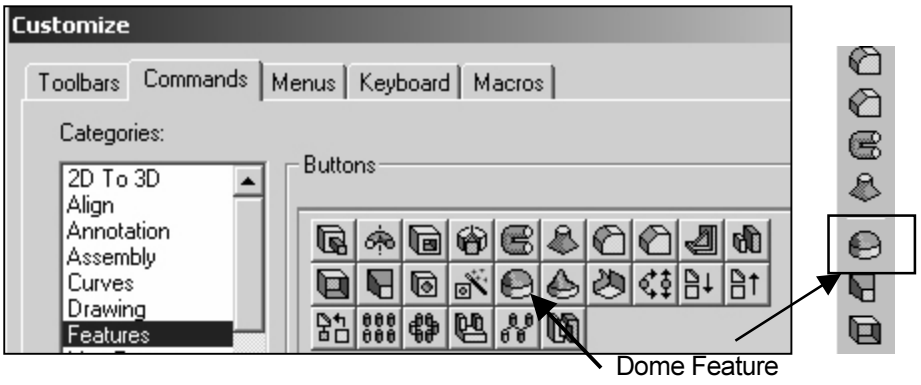
Customizing Toolbars

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.

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

222) Click the **Commands** tab. Click **Features** from the category text box. Drag the **Dome** icon into the Features Toolbar. Update the Toolbar. Click **OK** from the Customize dialog box.



You have just created four parts:

- BATTERY
- BATTERY PLATE
- LENS
- BULB

Practice the exercises before moving onto the next section.

Questions

1. Identify the function of the following features:

   Fillet

   Extruded Cut

   Extruded Boss

   Revolved Base

   Revolved Cut Thin

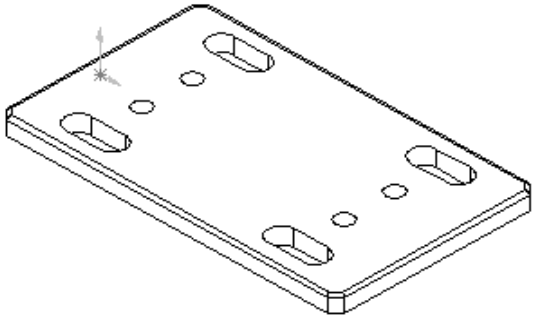
2. How do you add symmetric relations?
3. How do you avoid the Fillet Rebuild error message?
4. How do you create an angular dimension?
5. What is a draft angle?
6. When do you use a draft angle?
7. When do you use the Mirror command?
8. Describe disjointed geometry.
9. What is the function of the Shell feature?
10. An arc requires _____ points?
11. Name the required points of an arc?
12. When do you use the Hole Wizard feature?
13. What is a B-Spline?
14. Identify the required information for a Circular Pattern?
15. How do you add the Dome feature icon to the Feature Toolbar?

Exercises

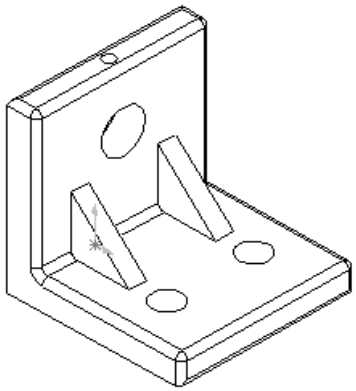
Create the following Extruded Parts:

Exercise 4.1: MOUNTING PLATE.

Exercise 4.2: L-BRACKET WITH ANGLE SUPPORT.



Exercise 4.1



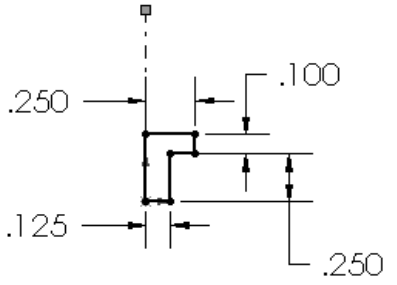
Exercise 4.2

Create the following Revolved Parts:

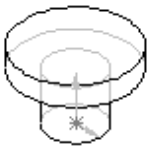
Exercise 4.3: SIMPLE SCREW.

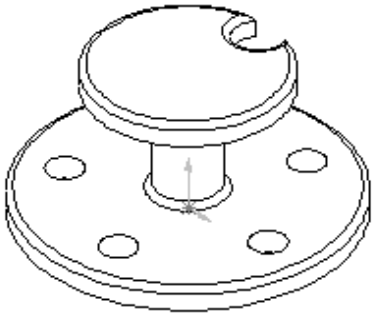
Exercise 4.4: SIMPLE CAP SCREW.

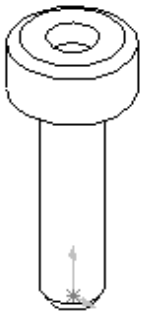
Exercise 4.5: SPOOL.



Exercise 4.3







Exercise 4.4

Exercise 4.5

Design Projects.

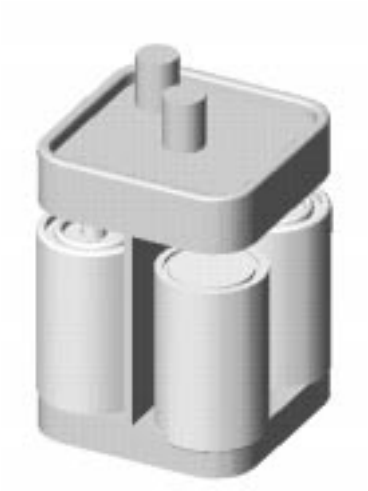
Exercise 4.6a:

Create a D-size battery.

Exercise 4.6b:

Create a battery HOLDER to hold 4-D size batteries.

Exercise 4.6a



Exercise 4.6b

Exercise 4.7:

Create a WHEEL assembly. A SHAFT supports the WHEEL. The SHAFT connects two L-BRACKETS. The L-BRACKETS are mounted to a BASE PLATE. Use purchased parts to save time and cost. The only dimension provided is the WHEEL.

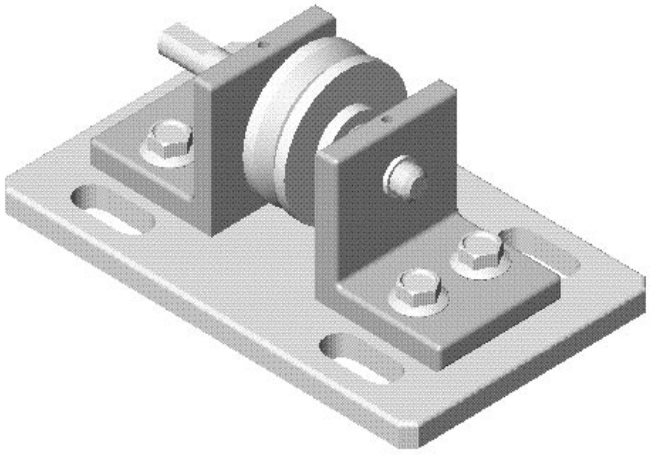
Select a WHEEL diameter:

- 3in.
- 4in.
- 100mm

Find a material supplier using the WWW. See **Exercise 4.11**: Globspec.com.

WHEEL Assembly Parts:

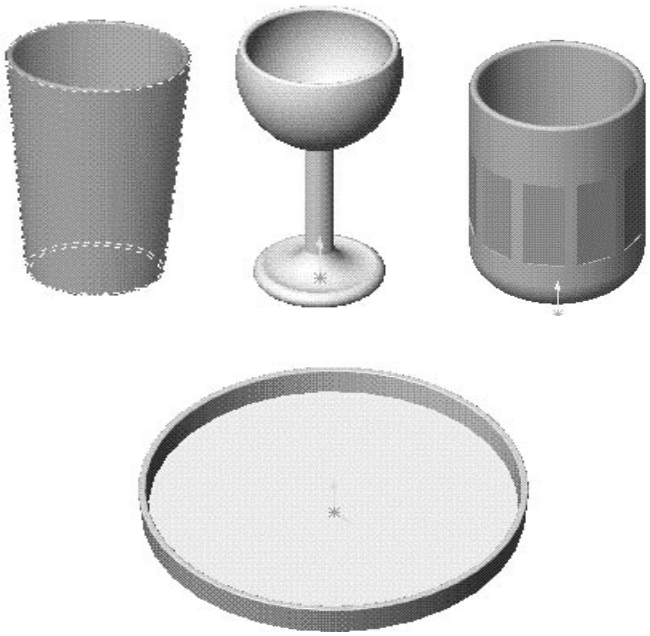
- BASE PLATE
- BUSHINGS
- L-BRACKET
- BOLTS
- SHAFT



Exercise 4.7

Exercise 4.8:

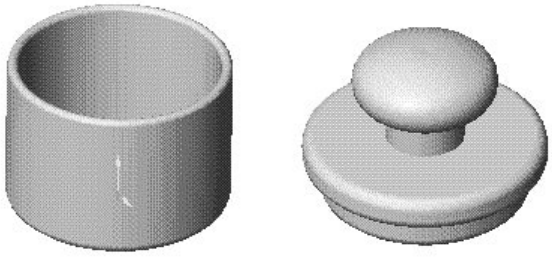
Create a TRAY and GLASS. Use real objects to determine the overall size and shape of the Base feature. Below are a few examples.



Exercise 4.8

Exercise 4.9:

Create a JAR-BASE. Save the JAR-BASE as a new part, JAR COVER. Use the dimensions from the JAR-BASE to determine the size of the JAR-COVER.

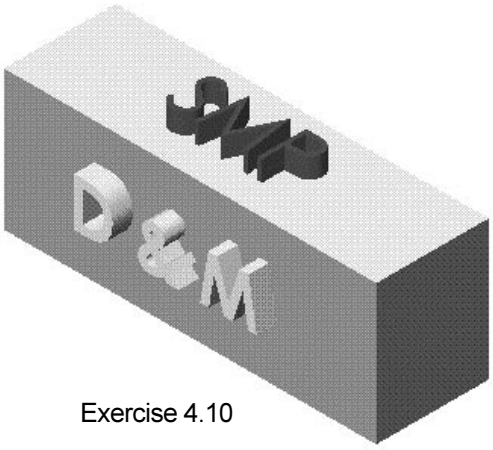


Exercise 4.9

Exercise 4.10:

Create an EMBOSSED-STAMP with your initials.

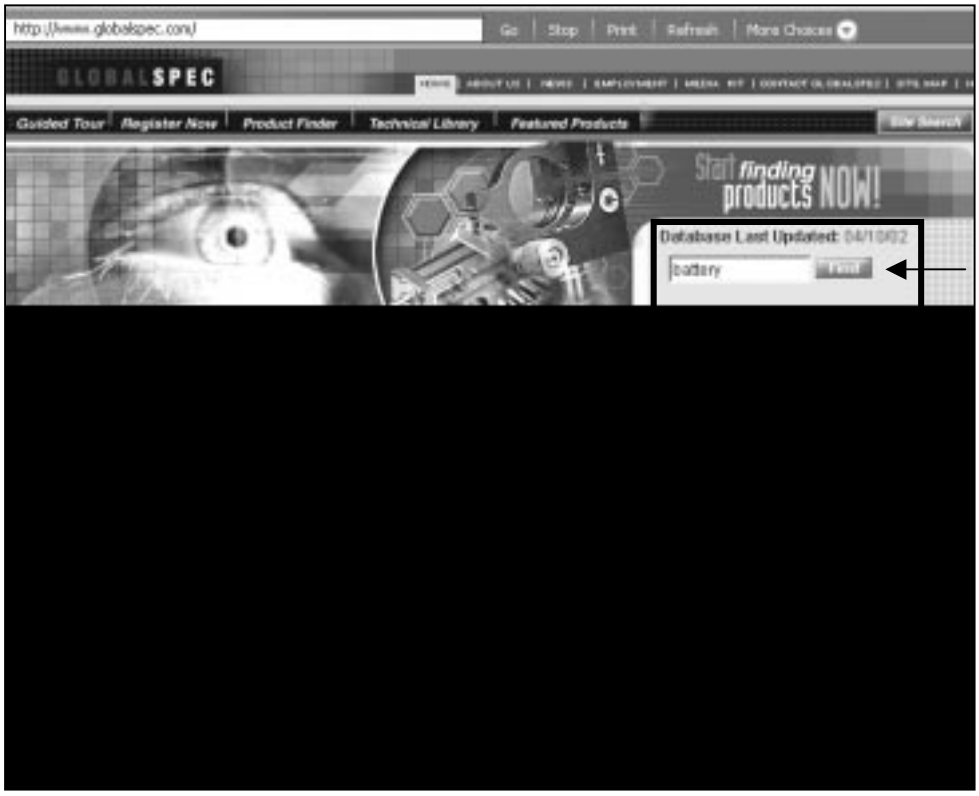
The initials are created with Extruded Sketched text. How do you create the text? Answer: Explore the command with SolidWorks on-line Help. Click Help. Click Index. Enter text. Click extruded text on model. Follow the instructions.



Exercise 4.10

Exercise 4.11: Industry Collaborative Exercise.

Engineers and designers spend a great deal of time searching for product suppliers and part specifications. How do you obtain a supplier for the batteries used in this project? What are the overall dimensions and voltage of a D size battery compared to the current 6-volt battery design? Research suppliers and part information utilizing the URL: http:// www.globalspec.com. Enter Battery. Click the Find button. Select D for battery size.



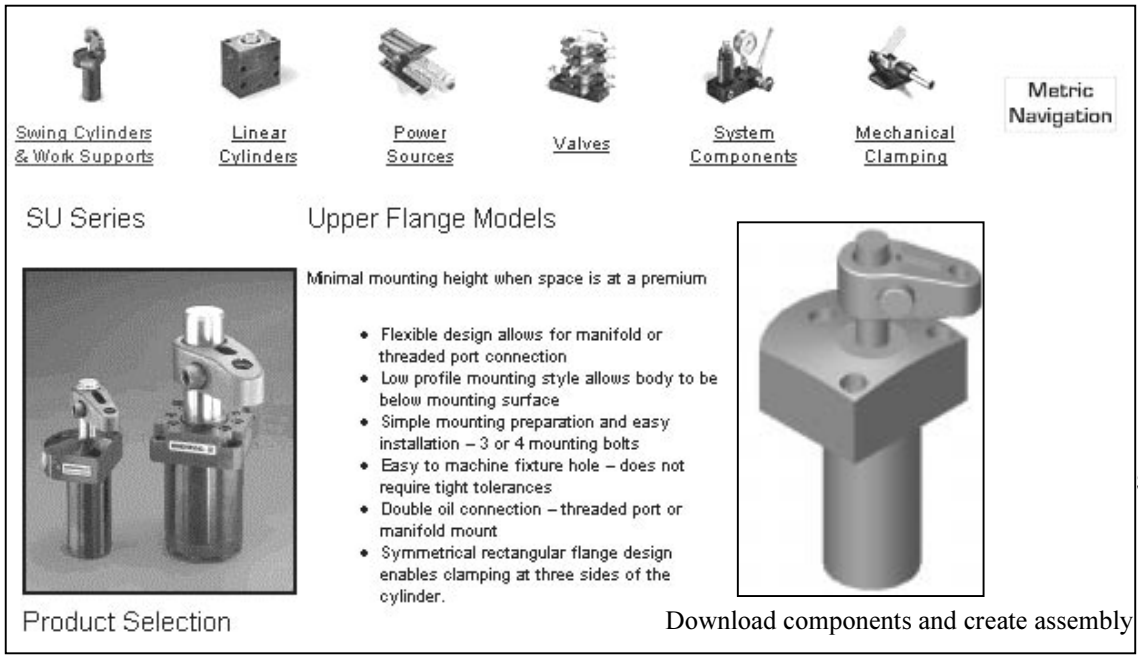
Gold Peak Industries of North America is the supplier. Select Search Results. Record the overall dimensions for the D size battery and voltage requirements.

Add to My Part List | Compare/Print Selected Parts

Select Part	Product Names (☆ indicates an exact match)	Send RFQ	Supplier's Product Info	Print-Friendly Version	Battery Voltage & Cell Size		Performance Specifications			Primary Batteries	
					Battery Size	Dimensions	Voltage (volts)	Capacity (ampere-hour)	Internal Resistance (ohms)	Standard	Lithium
☐	★GP13A Super Alkaline	RFQ	Web	Print	D	34.2mm(D) x 61.5mm (H)	1.50			Manganese	

The second design option for the FLASHLIGHT assembly requires a battery holder and 4-D size batteries. Does a supplier for a 4-D battery holder exist? If so, list the name of the supplier, material and the overall size of the battery holder.

b) Your second task is to find a left turning swing cylinder with a maximum clamping force of 2.1 kN. The swing cylinder utilizes a standard clamp arm. Download the Swing Cylinder and the Clamp. Create the new assembly that mates the Clamp to the Swing Cylinder. Create an assembly drawing with a Bill of Materials listing the two components with part number and description.



Clamping Force 1)	Stroke		Left Turning	Right Turning	Cylinder Effective Area		Oil Capacity		Max Oil Flow 1)	Standard Clamp Arm	Long Clamp Arm
kN	mm				cm^2		cm^3		l/min		
	Clamp	Total			Clamp	Unclamp	Clamp	Unclamp		Sold Seperately	
Single Acting			Model Number 2)								
2,1	8,1	16,5	SULS-22	SURS-22	0,77		1,31		0,2	CAS-22	CAL-22
4,9	9,9	22,6	SULS-52	SURS-52	1,81		4,10		0,4	CAS-52	CAL-52
8,0	11,9	22,1	SULS-92	SURS-92	3,16		6,88		1,0	CAS-92	CAL-92
10,7	12,7	28,4	SULS-121	SURS-121	4,06		11,47		1,6	CAS-121	CAL-122
17,4	14,0	27,9	SULS-202	SURS-202	7,10		19,99		2,3	CAS-202	CAL-202
33,1	16,0	30,0	SULS-352	SURS-352	12,39		37,20		3,9	CAS-352	CAL-352

c) Create a new Web Page document using Microsoft Word 2000. Add text and a jpeg image file of the PUMP assembly to the document. View the web page using File, Web Page Preview.

Note: Other web creation software tools can be utilized to create this web page.

NOTES: