INSIDE:

Engineering Design with SolidWorks 2008

and MultiMedia CD

A Step-by-Step Project Based Approach Utilizing 3D Solid Modeling

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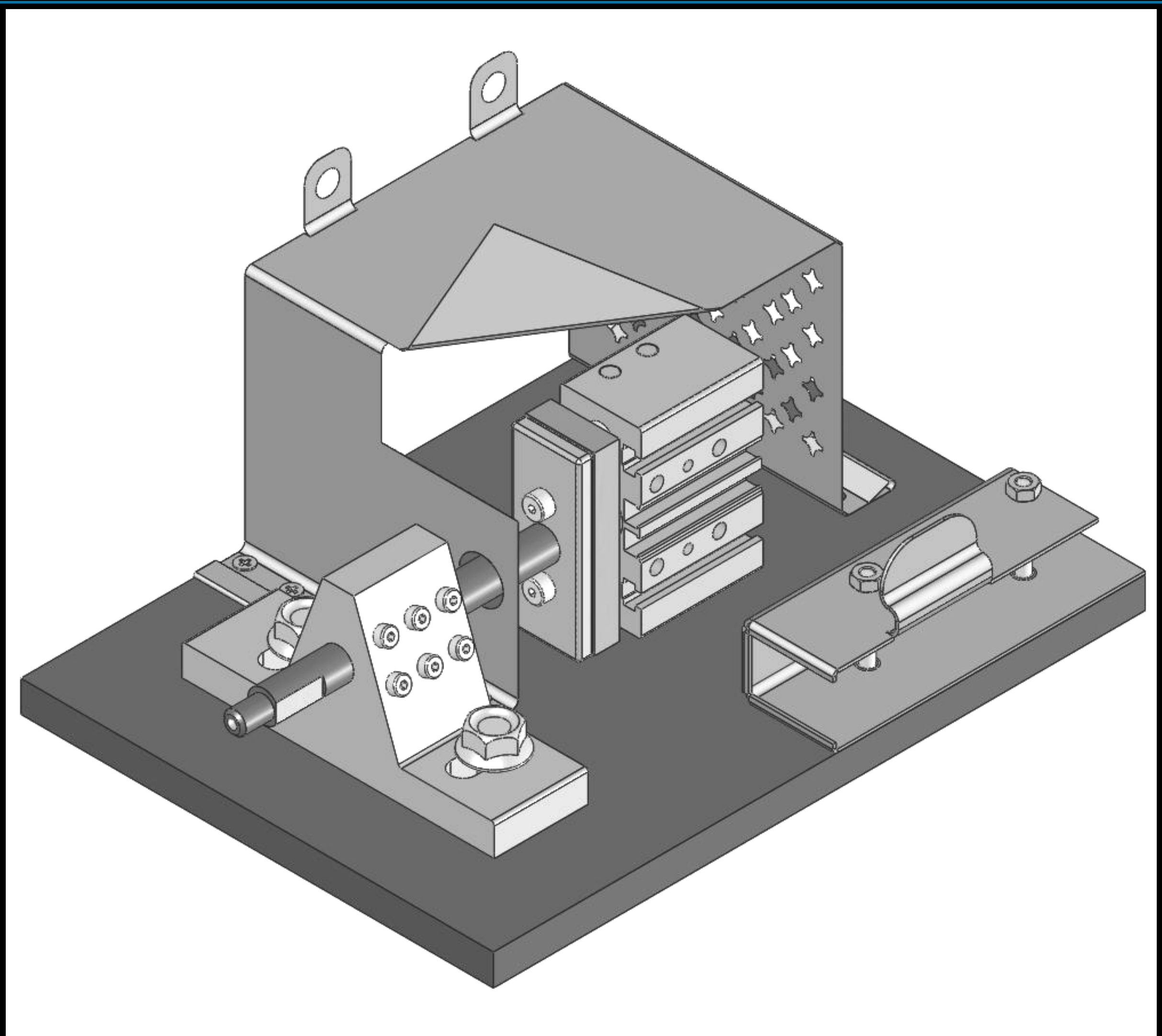

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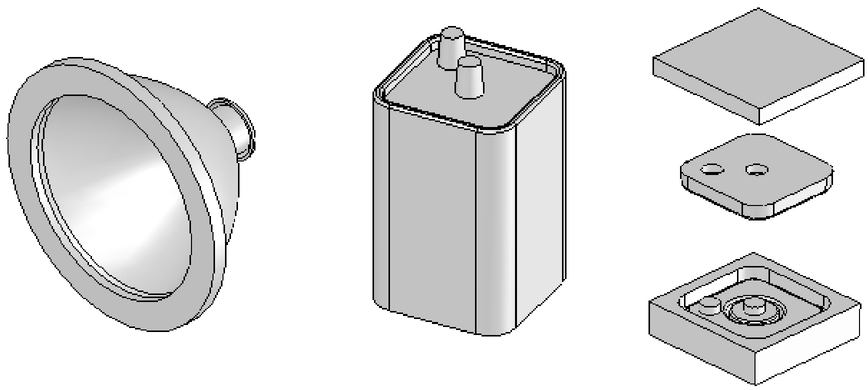
Project 4

Extrude and Revolve Features



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

Project Desired Outcomes:	**Usage Competencies:**
• An understanding of the customer's requirements for the FLASHLIGHT assembly.	• Ability to incorporate Design Intent into Sketches, Features, Parts, and Assemblies.
• Two Part Templates: o PART-IN-ANSI. o PART-MM-ISO.	• Ability to apply Document Properties and to create custom Part Templates.
• Four key parts: o BATTERY. o BATTERYPLATE. o LENS. o BULB.	• Specific knowledge and understanding of the following Features: Extruded Boss/Base, Extruded Cut, Revolved Boss/Base, Revolved Cut, Dome, Shell, Circular Pattern, and Fillet.
• Core and Cavity Tooling for the BATTERYPLATE.	• Understanding of the Mold tools: Scale, Parting Lines, Parting Surfaces, Shut-off Surfaces, Tooling Split, and Draft Analysis to create a simple Core and Cavity part.

Notes:

Project 4-Extrude and Revolve Features

Project Objective

Design a FLASHLIGHT assembly according to the customer's requirements. The FLASHLIGHT assembly will be cost effective, serviceable, and flexible for future manufacturing revisions.

Design intent is the process in which the model is developed to accept future changes. Build design intent into the FLASHLIGHT sketches, features, parts, and assemblies.

Create a custom Part Template. The Part Template is the foundation for the FLASHLIGHT parts.

Create the following parts:

- BATTERY.
- BATTERYPLATE.
- LENS.
- BULB.

The other parts for the FLASHLIGHT assembly are addressed in Project 5.

Create the Core and Cavity mold tooling required for the BATTERYPLATE.

On the completion of this project, you will be able to:

- Apply design intent to sketches, features, parts, and assemblies.
- Select the best profile for a sketch.
- Select the proper Sketch plane.
- Create a template: English and Metric units.
- Set Document Properties.
- Customize the SolidWorks CommandManager toolbar.
- Insert/Edit dimensions.
- Insert/Edit relations.
- Use the following SolidWorks features:
  - Extruded Boss/Base.
  - Extruded Cut.
  - Revolved Boss/Base.

- Revolved Boss Thin.
- Revolved Cut Thin.
- Dome.
- Shell.
- Circular Pattern.
- Fillet.

* Use the following Mold tools:

  - Draft Analysis.
  - Scale.
  - Parting Lines.
  - Shut-off Surfaces.
  - Parting Surfaces.
  - Tooling Split.

Project Overview

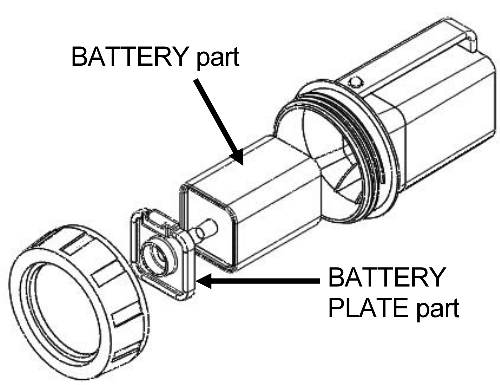
In Project 4, you started the design of the FLASHLIGHT assembly according to the customer's requirements. The FLASHLIGHT assembly will be cost effective, serviceable, and flexible for future manufacturing revisions.

A template is the foundation for a SolidWorks document. A template contains document settings for units, dimensioning standards, and other properties. Create two part templates for the FLASHLIGHT Project:

* PART-IN-ANSI.
* PART-MM-ISO.

Create two parts for the FLASHLIGHT assembly in this Project:

* BATTERY.
* BATTERYPLATE.



FLASHLIGHT Assembly

Parts models consist of 3D features. Features are the building blocks of a part.

A 2D sketch is required to create an Extruded feature. Utilize the sketch geometry and sketch tools to create the following features:

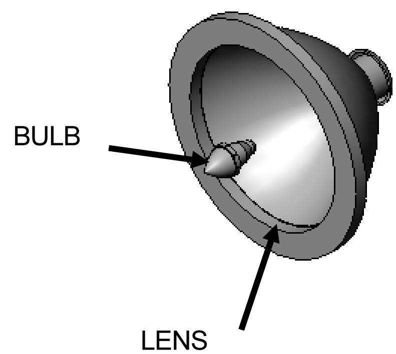
- Extruded Base.
- Extruded Boss.
- Extruded Cut.

Utilize existing faces and edges to create the following features:

- Fillet.
- Chamfer.

This project introduces you to the Revolved feature.

Create two parts for the FLASHLIGHT assembly in this section:



- LENS.
- BULB.

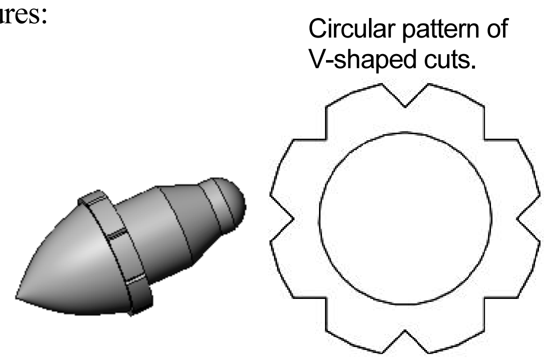
A Revolved feature requires a 2D sketch profile and a centerline. Utilize sketch geometry and sketch tools to create the following features:

- Revolved Base.
- Revolved Boss.
- Revolved Boss-Thin.
- Revolved Cut.

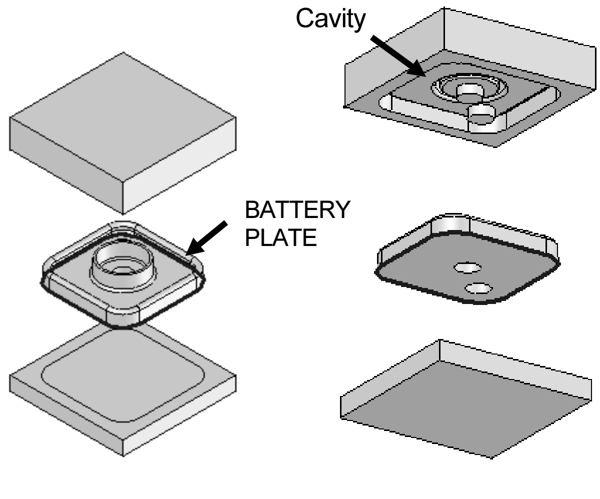
Utilize existing faces to create the following features:

- Shell.
- Dome.
- Hole Wizard.

Utilize the Extruded Cut feature to create a Circular Pattern.



Utilize the Mold tools to create the Cavity tooling plates for the BATTERYPLATE part.



Isometric view Rotated
Mold Tools

Design Intent

The SolidWorks definition of design intent is the process in which the model is developed to accept future changes.

Models behave differently when design changes occur. Design for change. Utilize geometry for symmetry, reuse common features and reuse common parts.

Build change into the following areas:

1. Sketch.

2. Feature.

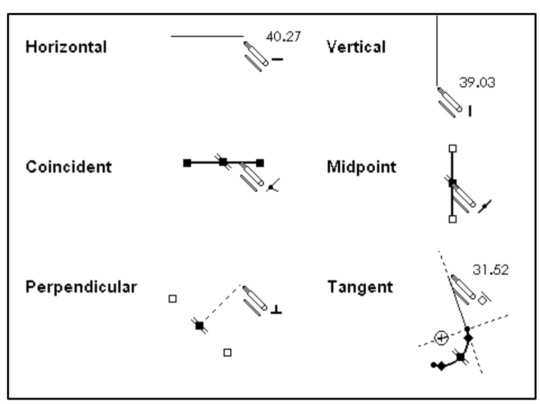
3. Part.

4. Assembly.

5. Drawing.

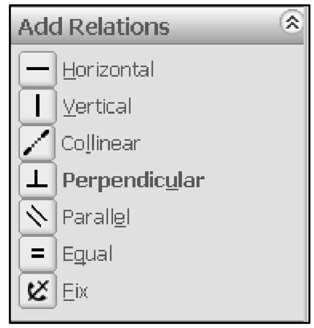
1. Design Intent in the Sketch

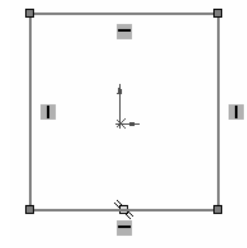
Build design intent in a sketch as the profile is created. A profile is determined from the Sketch Entities. Example: Rectangle, Circle, Arc, Point, etc. Apply symmetry into a profile through a sketch centerline, mirror entity, and position about the reference planes and Origin.



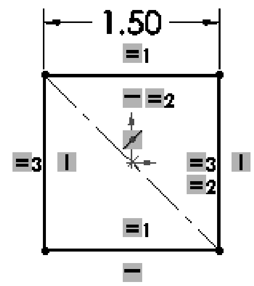
Build design intent as you sketch with automatic geometric relations. Document the decisions made during the up front design process. This is very valuable when you modify the design later.

A rectangle contains Horizontal, Vertical, and Perpendicular automatic geometric relations. Apply design intent using added geometric relations. Example: Horizontal, Vertical, Collinear, Perpendicular, Parallel, etc.

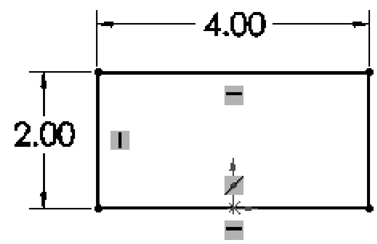




Example A: Apply design intent to create a square profile. Sketch a corner rectangle with the Origin approximately in the center. Insert a construction reference centerline. Add a Midpoint relation. Add an Equal relation between the two vertical and horizontal lines. Insert a dimension to define the square.



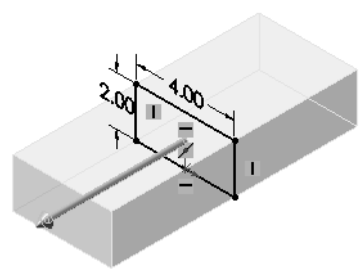
Example B: Develop a corner rectangular profile. The bottom horizontal midpoint of the rectangular profile is located at the Origin. Sketch a rectangle. Add a Midpoint relation between the horizontal edge of the rectangle and the Origin. Insert two dimensions to define the width and height of the rectangle.



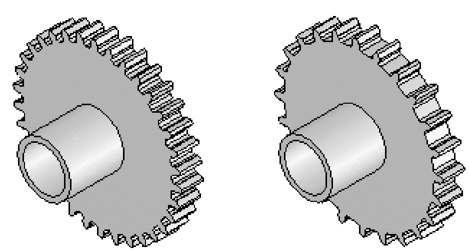
2. Design Intent in the Feature

Build design intent into a feature by addressing symmetry, feature selection, and the order of feature creation.

Example A: The Base Extrude feature remains symmetric about the Front Plane. Utilize the Mid Plane End Condition option in Direction 1. Modify the depth, and the feature remains symmetric about the Front Plane.

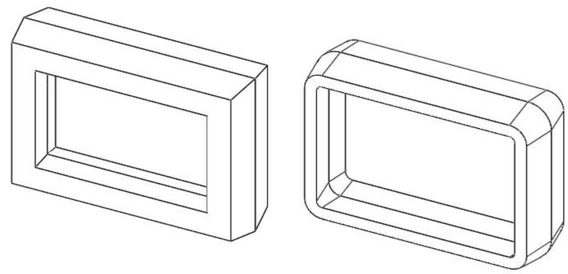


Example B: Do you create each tooth separate using the Extruded Cut feature? No. Create a single tooth and then apply the Circular Pattern feature. Create 34 teeth for a Circular Pattern feature. Modify the number of teeth from 32 to 24.



3. Design Intent in the Part

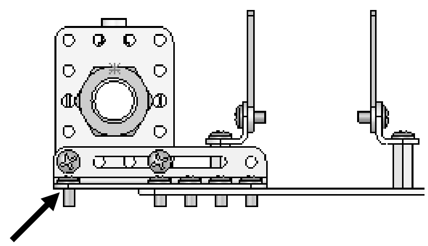
Utilize symmetry, feature order and reusing common features to build design intent into the part.



Example A: Feature order. Is the entire part symmetric? Feature order affects the part. Apply the Shell feature before the Fillet feature and the inside corners remain perpendicular.

4. Design Intent in the Assembly

Utilizing symmetry, reusing common parts and using the Mate relation between parts builds the design intent into an assembly.



Example A: Reuse geometry in an assembly. The assembly contains a linear pattern of holes. Insert one screw into the first hole. Utilize the Component Pattern feature to copy the machine screw to the other holes.

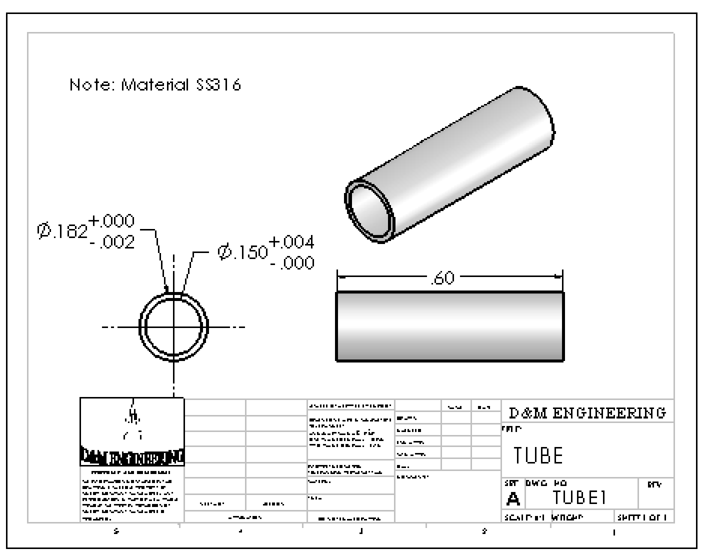
5. Design Intent in the Drawing

Utilize dimensions, tolerance and notes in parts and assemblies to build the design intent into the Drawing.

Example A: Tolerance and material in the drawing.

Insert an outside diameter tolerance +.000/-.002 into the TUBE part. The tolerance propagates to the drawing.

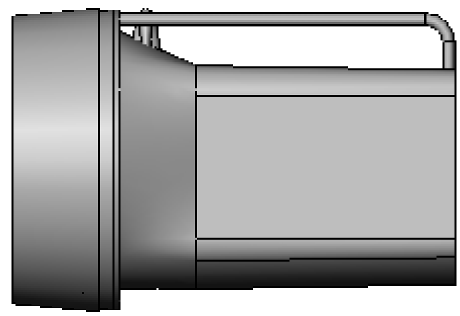
Define the Custom Property MATERIAL in the part. The MATERIAL Custom Property propagates to the drawing.



Project Situation

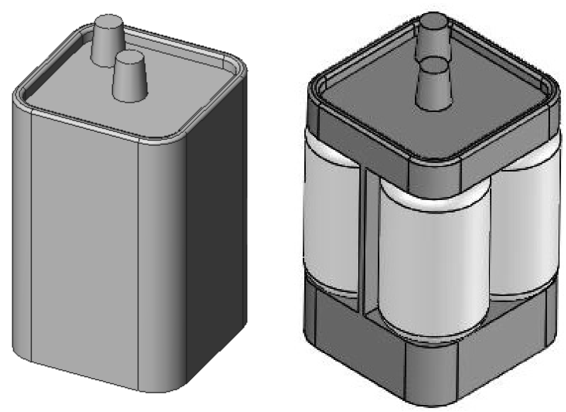
You work for a company that specializes in providing promotional tradeshow products. The company is expecting a sales order for 100,000 flashlights with a potential for 500,000 units next year. Prototype drawings of the flashlight are required in three weeks.

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.
- Lightweight semi indestructible body.
- Self standing with a handle.

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 conceptual sketch. Your colleagues review the sketch. The team's consensus is to proceed with the conceptual design.

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

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 assembly.

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 four D cell battery model for the next product revision.

Design the FLASHLIGHT assembly 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 parts: 6-volt BATTERY, LENS ASSEMBLY, SWITCH, and an O-RING. Model the following purchased parts: BATTERY, LENS assembly, SWITCH and the O-RING. The LENS assembly consists of the LENS and the BULB.

Your company will design, model and manufacture the following parts: BATTERYPLATE, LENSCAP and HOUSING.

Purchased Parts:	**Designed Parts:**
BATTERY	BATTERYPLATE
LENS assembly	MOLD TOOLING
*SWITCH	*LENSCAP
*O-RING	*HOUSING

*Parts addressed in Project 5.

The BATTERYPLATE, LENSCAP, and HOUSING are plastic parts. Review the injection molded manufacturing process and the SolidWorks Mold tools. Modify the part features to eject the part from the mold. Create the MOLD TOOLING for the BATTERYPLATE.

Part Template

Units are the measurement of physical quantities. Millimeter dimensioning and decimal inch dimensioning are the two most common unit types specified for engineering parts and drawings. The FLASHLIGHT project is designed in inch units and manufactured in millimeter units. Inch units are the primary unit and Millimeter units are the secondary unit.

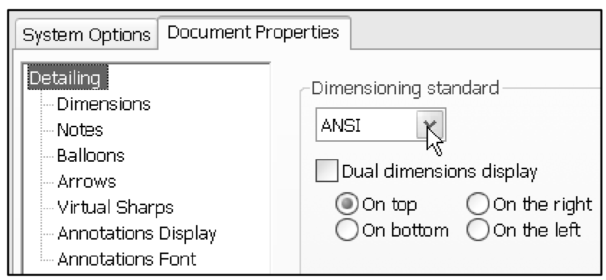
Create two Part templates:

- PART-IN-ANSI.
- PART-MM-ISO.

Save the Part templates in the MY-TEMPLATES folder. System Options, File Locations option controls the file folder location of SolidWorks documents. Utilize the File Locations option to reference your Part templates in the MY-TEMPLATES folder. Add the MY-TEMPLATES folder path name to the Document Templates File Locations list.

Activity: Two Part Templates

Create the PART-IN-ANSI Template.



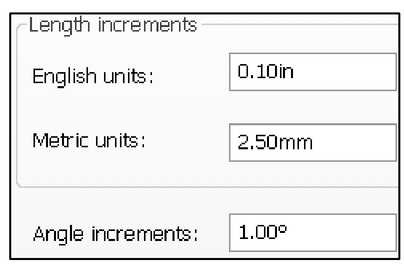
1) Click **New** from the Menu bar.

2) Double-click **Part** from the default Templates tab from the Menu bar.

Set the Dimensioning Standard to ANSI.

3) Click **Options**, **Document Properties** tab.

4) Select **ANSI** from the Dimensioning standard drop-down box.



5) Click the **System Options** tab.

6) Click **Spin Box Increments**. View the default settings.

7) Click **inside** the English units box.

8) Enter **.10**in.

9) Click **inside** the Metric units box.

10) Enter **2.50**mm.

11) Click the **Document Properties** tab.

Set the part units for inch.

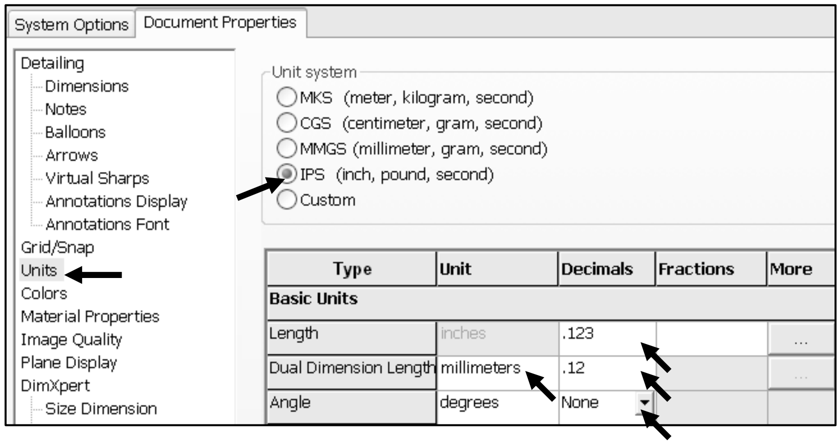
12) Click **Units**.

13) Select **IPS** for Unit system.

14) Select **.123** for Basic unit length decimal place.

15) Select **millimeters** for Dual dimension length unit.

16) Select **.12** for Basic unit decimal place.



17) Select **None** for Basic unit angle decimal place.

18) Click **OK** from the Document Properties - Units dialog box.

Save the part template.

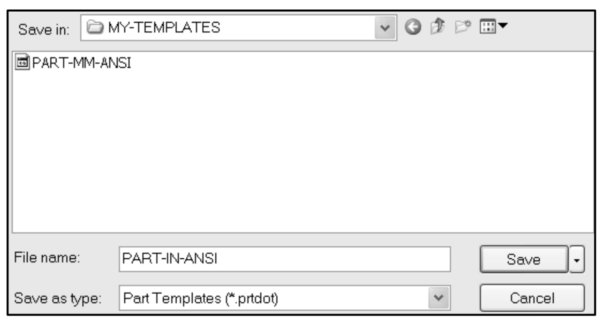
19) Click **Save As** from the Menu bar.

20) Select **Part Templates (*.prtdot)** from the Save As type box.

21) Select **ENGDESIGN-W-SOLIDWORKS\MY-TEMPLATES** for the Save in folder.

22) Enter **PART-IN-ANSI** for File name.

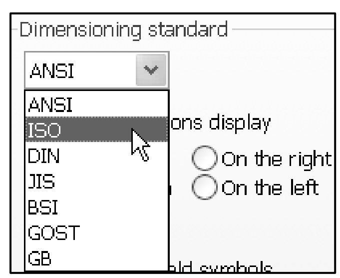
23) Click **Save**.



Utilize the PART-IN-ANSI template to create the PART-MM-ISO template.

24) Click **Options** , **Document Properties** tab.

25) Select **ISO** from the Dimensioning standard drop-down box.



Set the part units for millimeter.

26) Click **Units**.

27) Select **MMGS** for Unit system.

28) Select **.12** for Basic unit length decimal place.

29) Select **None** for Basic unit angle decimal place.

30) Click **OK**.

Save the part template.

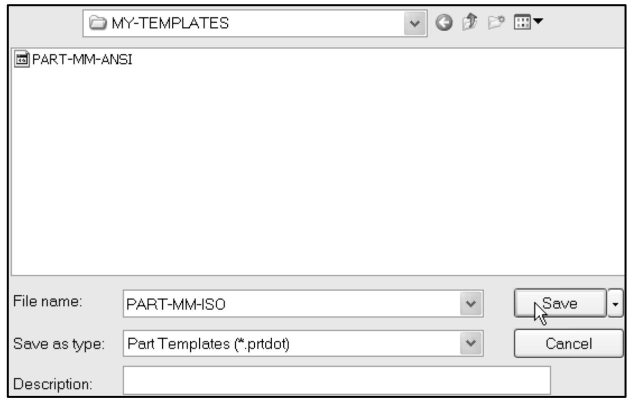
31) Click **Save As** from the Menu bar.

32) Select **Part Templates (*.prtdot)** from the Save As type box.

33) Select **ENGDESIGN-W-SOLIDWORKS\MY-TEMPLATES** for the Save in folder.

34) Enter **PART-MM-ISO** for File name.

35) Click **Save**.



Set the System Options for File Locations to display in the New dialog box if needed.

36) Click **Options** from the Menu bar.

37) Click **File Locations** from the System Options tab.

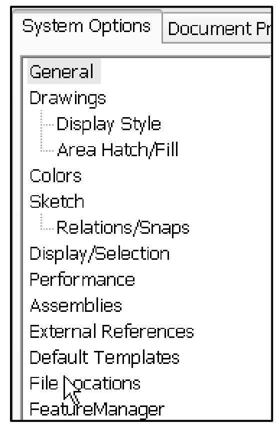
38) Select **Document Templates** from Show folders for.

39) Click the **Add** button.

40) Select the **MY-TEMPLATES** folder.

41) Click **OK** from the Browse for Folder dialog box.

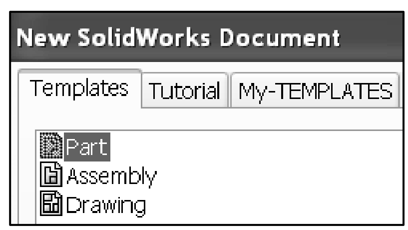
42) Click **OK** from the System Options dialog box.



Close all documents.

43) Click **Windows**, **Close All** from the Menu bar.

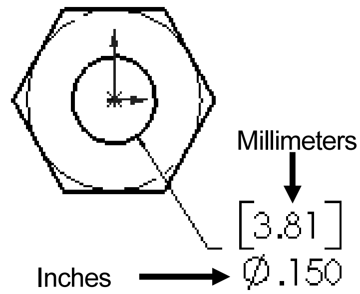
Each folder listed in the System Options, File Locations, Document Templates, Show Folders For option produces a corresponding tab in the New SolidWorks Document dialog box. The order in the Document Templates box corresponds to the tab order in the New dialog box.



The MY-TEMPLATES tab is visible when the folder contains SolidWorks Template documents. Create the PART-MM-ANSI template as an exercise.

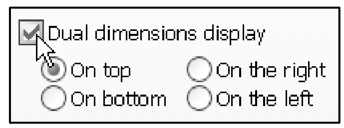
The PART-IN-ANSI template contains Document Properties settings for the parts contained in the FLASHLIGHT assembly. Substitute the PART-MM-ISO or PART-MM-ANSI template to create the identical parts in millimeters.

The primary units in this Project are IPS, (inch, pound, seconds).

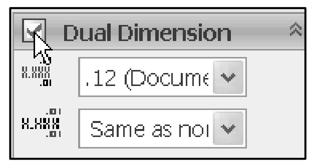


The optional secondary units are MMGS (millimeters, grams, second) and are indicated in brackets [ ].

Illustrations are provided in both inches and millimeters. Utilize inches, millimeters or both.

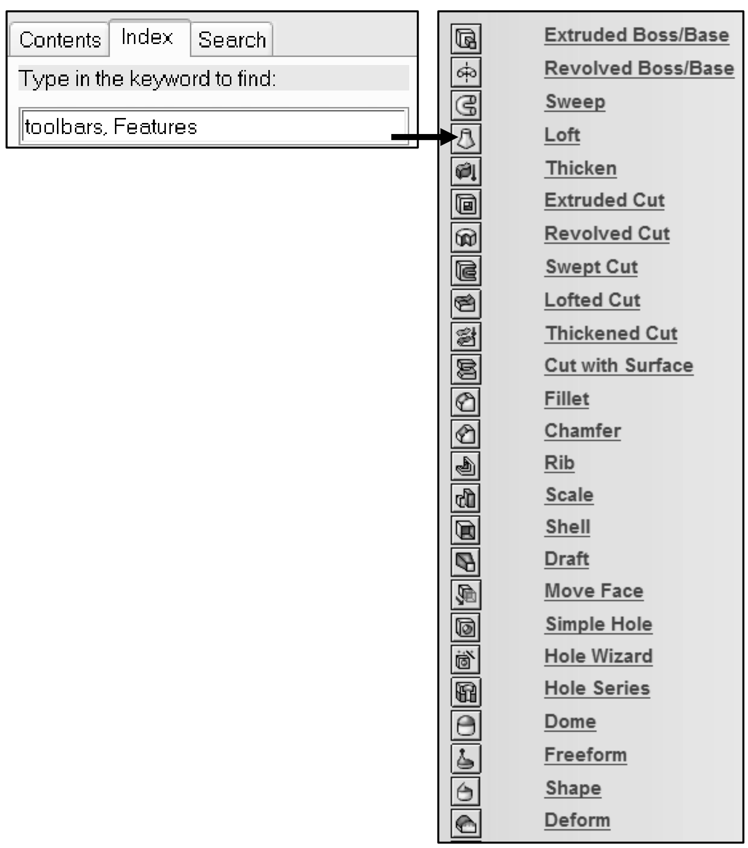


To set dual dimensions, select Options, Document Properties. Check the Dual dimension display box as illustrated.



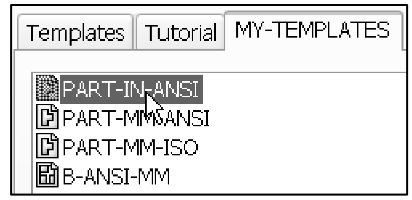
To set dual dimensions for an active model, check the Dual Dimension box in the Dimension PropertyManager.

Select Toolbars, Features in SolidWorks Help Topic to review the function of each tool in the Features toolbar.



Additional information on System Options, Document Properties, File Locations and Templates is found in SolidWorks Help. Keywords: Options (detailing, units), templates, Files (locations), menus and toolbars (features, sketch).

Review of the Part templates



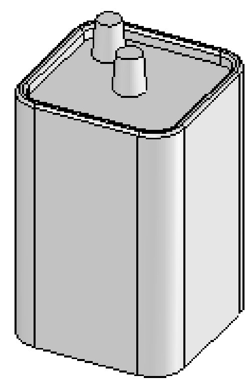
You created two Part templates: PART-MM-ANSI and PART-IN-ISO. The Document Properties Dimensioning Standard, units, and decimal places were stored in the Part Templates.

The File Locations System Option, Document Templates option controls the reference to the MY-TEMPLATES folder.

Note: In some network locations and school environments, the File Locations option must be set to MY-TEMPLATES for each session of SolidWorks.

You can exit SolidWorks at any time during this project. Save your document. Select File, Exit from the Menu bar.

BATTERY Part



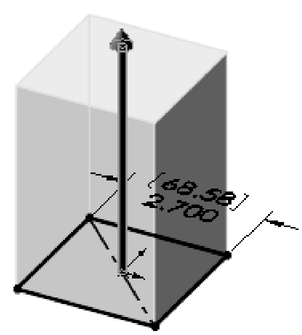
The BATTERY is a simplified representation of a purchased OEM part. Represent the battery terminals as cylindrical extrusions. The battery dimensions are obtained from the ANSI standard 908D.

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

Create the BATTERY part. Use features to create parts. Features are building blocks that add or remove material.

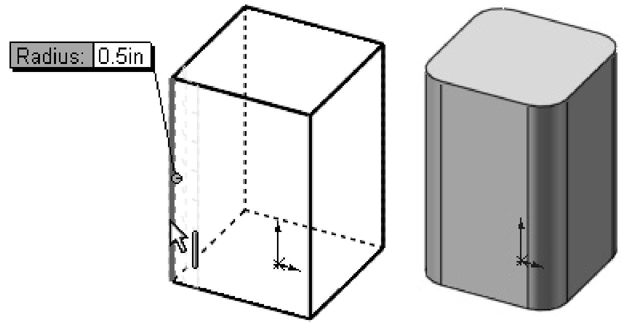
Utilize the Extruded Base feature. The Extrude Base features add material. The Base feature is the first feature of the part.

Apply symmetry. Sketch a rectangle profile on the Top Plane, centered at the Origin.

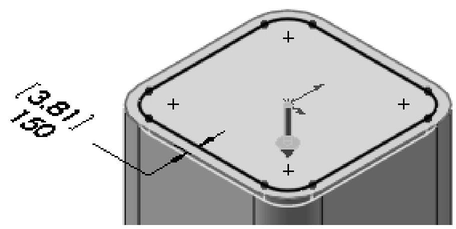


Extend the profile perpendicular (⊥) to the Top Plane.

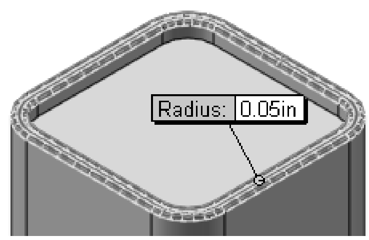
Utilize the Fillet feature to round the four vertical edges.



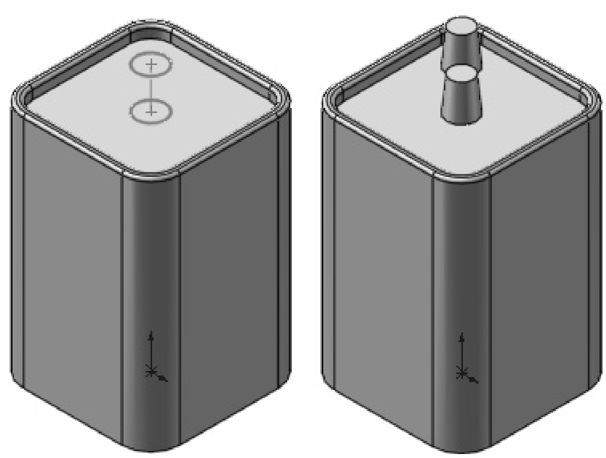
The Extruded Cut feature removes material from the top face. Utilize the top face for the Sketch plane. Utilize the Offset Entity Sketch tool to create the profile.



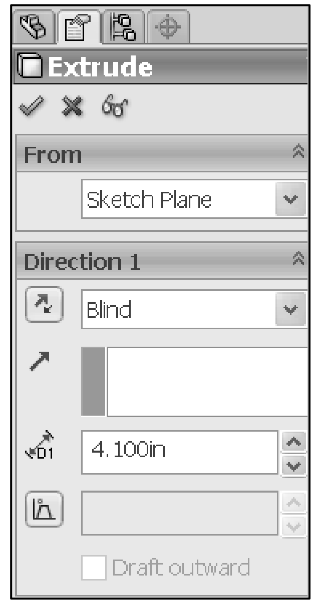
Utilize the Fillet feature to round the top narrow face.



The Extruded Boss feature adds material. Conserve design time. Represent each of the terminals as a cylindrical Extruded Boss feature.



BATTERY Part-Extruded Base Feature



The Extruded Base feature requires:

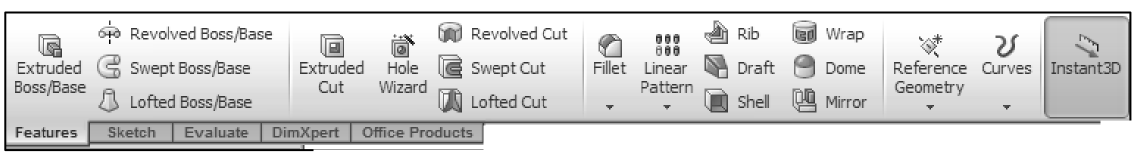
- Sketch plane (Top).
- Sketch profile (Rectangle).
  - o Geometric relations and dimensions.
- End Condition Depth (Blind) in Direction 1.

Create a new part named, BATTERY. Insert an Extruded Base feature. Extruded features require a Sketch plane. The Sketch plane determines the orientation of the Extruded Base feature. The Sketch plane locates the Sketch profile on any plane or face.

The Top Plane is the Sketch plane. The Sketch profile is a rectangle. The rectangle consists of 2 horizontal lines and 2 vertical lines.

Geometric relations and dimensions constrain the sketch in 3D space. The Blind End Condition in Direction 1 requires a depth value to extrude the 2D Sketch profile and to complete the 3D feature.

Note: Alternate between the Features tab and the Sketch tab in the CommandManager to display the available Feature tools and Sketch tools for your model.



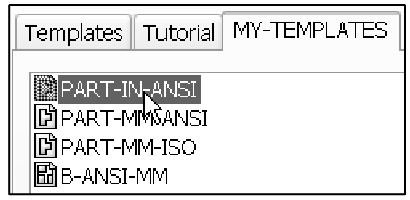
Activity: BATTERY Part-Extruded Base Feature

Create a new part.

44) Click **New** from the Menu bar.

45) Click the **MY-TEMPLATES** tab.

46) Double-click **PART-IN-ANSI**, [**PART-MM-ISO**].



Save the empty part.

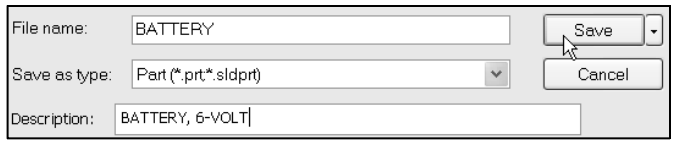
47) Click **Save**.

48) Select **PROJECTS** for Save in folder.

49) Enter **BATTERY** for File name.

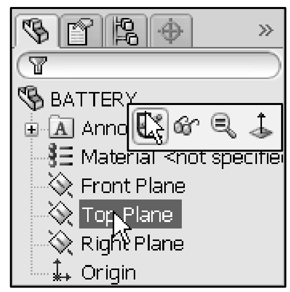
50) Enter **BATTERY**, **6-VOLT** for Description.

51) Click **Save**. The Battery FeatureManager is displayed.



Select the Sketch plane.

52) Right-click **Top Plane** from the FeatureManager.



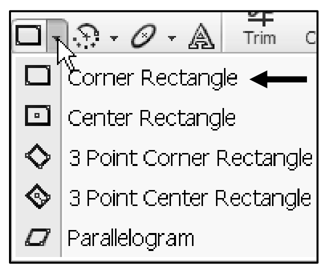
Sketch the profile.

53) Click **Sketch** from the shortcut toolbar. The Sketch toolbar is displayed.

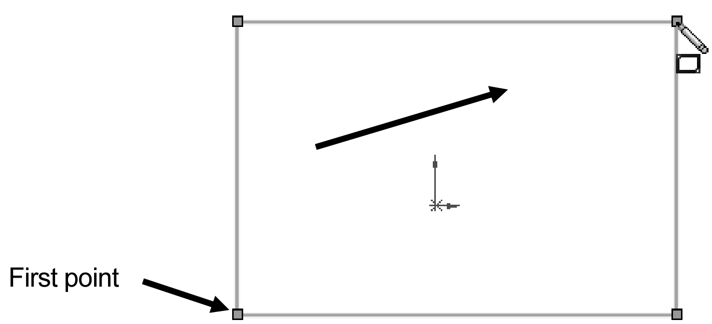
54) Click the **Corner Rectangle** Sketch tool. The Corner Rectangle icon is displayed.

55) Click the **first point** in the lower left quadrant as illustrated.

56) Drag and click the **second point** in the upper right quadrant as illustrated. The Origin is approximately in the middle of the rectangle. Insert a Midpoint relation in the next step.



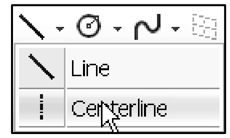
You can apply the Center Rectangle Sketch tool which automatically provides a Midpoint geometric relation. This Sketch tool is new for 2008. Select the Center Rectangle Sketch tool. Click the Origin. Sketch your rectangle.



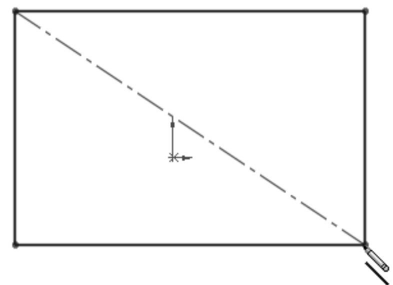
The book is designed to expose the new user to all tools and procedures.

Sketch the centerline.

57) Click the **Centerline** Sketch tool. The Insert Line PropertyManager is displayed.



58) 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.



Insert a Midpoint relation.

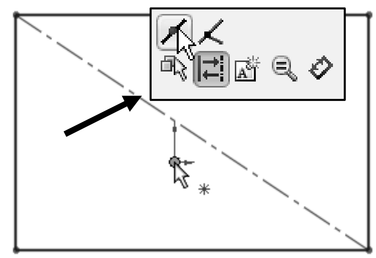
59) Right-click **Select** in the Graphics window.

60) Click the **centerline** in the Graphics window.

61) Hold the **Ctrl** key down.

62) Click the **Origin**. The Properties PropertyManager is displayed. The selected entities are displayed in the Selected Entities box.

63) Release the **Ctrl** key.



64) Right-click **Make Midpoint** from the shortcut toolbar as illustrated.

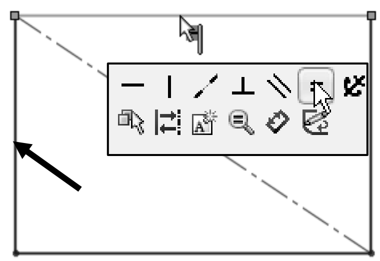
65) Click **OK** from the Properties PropertyManager.

Create a square. Insert an Equal relation.

66) Click the **top horizontal line** in the sketch.

67) Hold the **Ctrl** key down.

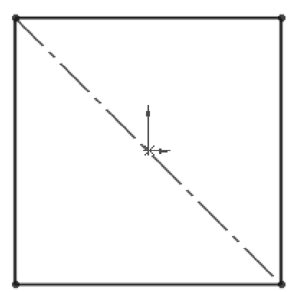
68) Click the **left vertical line**. The Properties PropertyManager is displayed. The selected entities are displayed in the Selected Entities box.



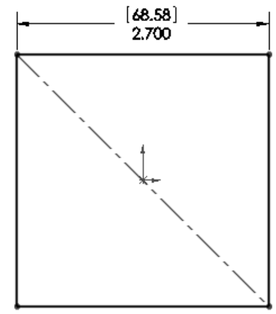
69) Release the **Ctrl** key.

70) Right-click **Make Equal** = from the shortcut toolbar.

71) Click **OK** from the Properties PropertyManager. View the results in the Graphics window.



Add a dimension.



72) Click the **Smart Dimension** Sketch tool.

73) Click the **top horizontal line**.

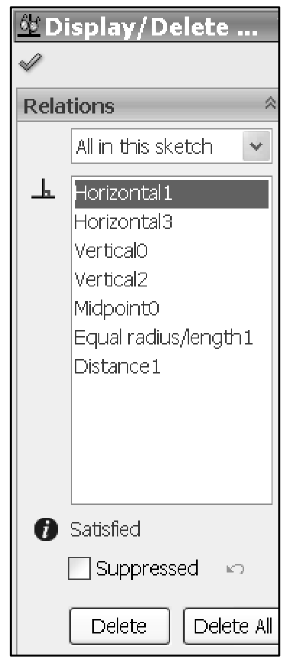
74) Click a **position** above the horizontal line.

75) Enter **2.700**in, [**68.58**] for width.

76) Click the **Green Check mark** in the Modify dialog box. The black Sketch status is fully defined.

77) Click **OK** from the Dimension PropertyManager.

Display the Sketch relations.



78) Click the **Display/Delete Relations** Sketch tool. The Display/Delete Relations PropertyManager is displayed. View the model relations. The Distance1 relation was created from the dimension.

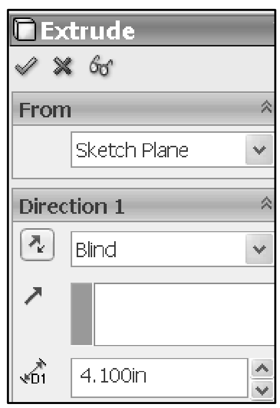
79) Click **OK** from the Display/Delete Relations PropertyManager.

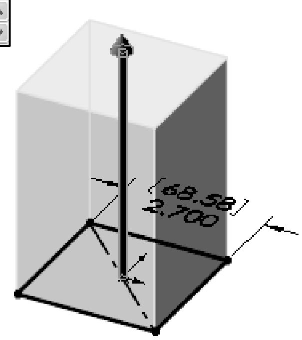
Insert an Extruded Base feature.

80) Click the **Extruded Boss/Base** feature tool. The Extrude PropertyManager is displayed. Blind is the default End Condition in Direction 1.

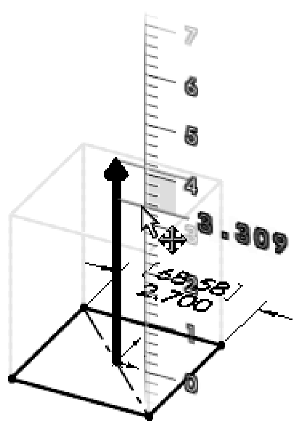
81) Enter **4.100**in, [**104.14**] for Depth in Direction 1. Accept the default settings. The extrude direction is upwards. Note the location of the Origin.

82) Click **OK** from the Extrude PropertyManager. Extrude1 is displayed in the FeatuerManager.



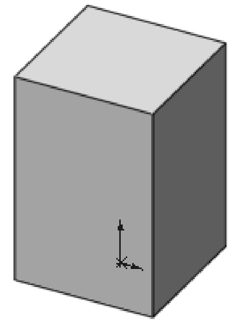


New in 2008 is Instant3D. Instant3D provides the ability to drag geometry and dimension manipulator points to resize features in the Graphics window, and to use on-screen rulers to measure modifications. In this book, you will use the PropertyManager.



Caution should be used when applying the Instant3D tool. Understand the dimension changes to your model.

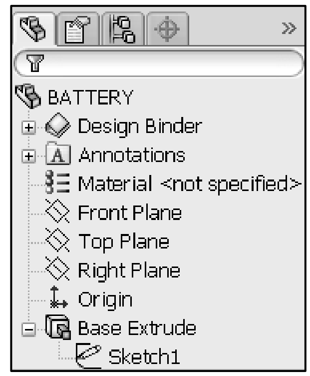
Fit the part to the Graphics window.
83) Press the **f** key.



Rename the Extruded Base feature.
84) Rename **Extrude1** to **Base Extrude**.

Save the BATTERY.
85) Click **Save**.

Utilize an Equal relation versus two linear dimensions when a rectangular profile is square.



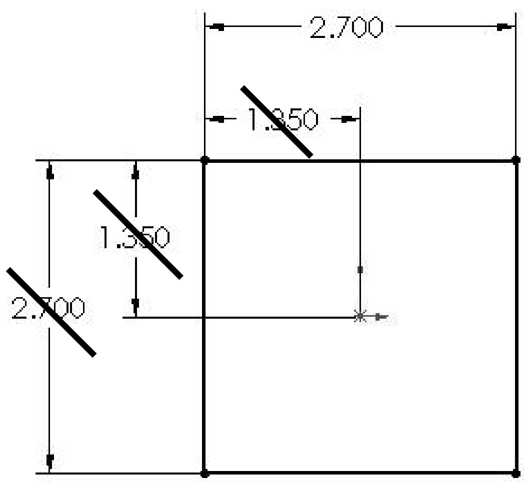
One dimension controls the size. The 6-Volt manufacturing standard determines the square profile.

The Midpoint relation centers the square profile about the Origin. Note: the new Center Rectangle tool would also perform this function.

One relation eliminates two dimensions to locate the profile with respect to the Origin.

The color of the sketch indicates the sketch status.

- Light Blue – Currently selected.
- Blue – Under defined, requires additional geometric relations and dimensions.
- Black – Fully defined.
- Red – Over defined, requires geometric relations or dimensions to be deleted or redefined to solve the sketch.



Short cuts save time. Right-click Select to deselect a tool and to choose geometry and entities.

Click inside the Graphics window to close the Properties PropertyManager or Dimension PropertyManager. Tools are located on the right mouse button and the toolbars. The Select icon is also located in the Standard toolbar.

BATTERY Part-Fillet Feature Edge

Fillets remove sharp edges. Utilize Hidden Lines Visible to display hidden edges.

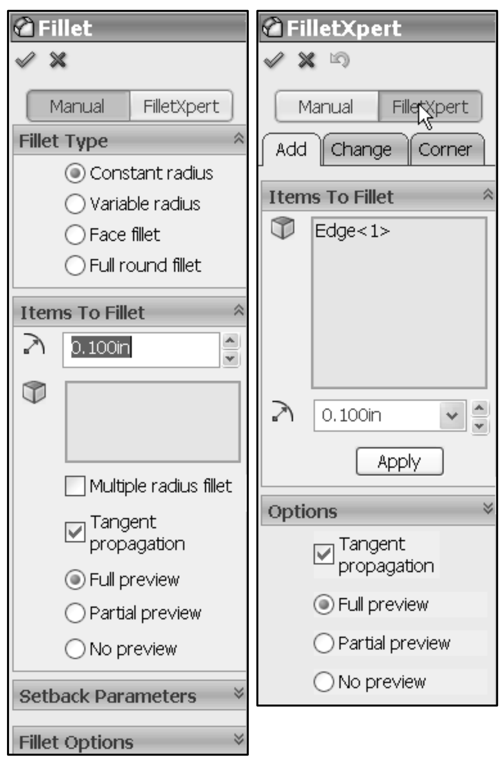
An edge Fillet requires:

- Edge.
- Fillet radius.

Select a vertical edge. Select the Fillet feature from the Features toolbar. Enter the Fillet radius. Add the other vertical edges to the Items To Fillet option.

The order of selection for the Fillet feature is not predetermined. Select edges to produce the correct result.

The Fillet feature uses the Fillet PropertyManager. The Fillet PropertyManager for 2008 has had a major interface enhancement, and provides the ability to select either the Manual or FilletXpert tab. Each tab has a separate menu and PropertyManager. The Fillet PropertyManager and FilletXpert PropertyManager displays the appropriate selections based on the type of fillet you create.



The FilletXpert automatically manages, organizes and reorders your fillets in the FeatureManager design tree. The FilletXpert PropertyManager provides the ability to add, change or corner fillets in your model. The PropertyManager remembers its last used state. View the SolidWorks tutorials for additional information on fillets.

The FilletXpert can ONLY create and edit constant radius fillets.

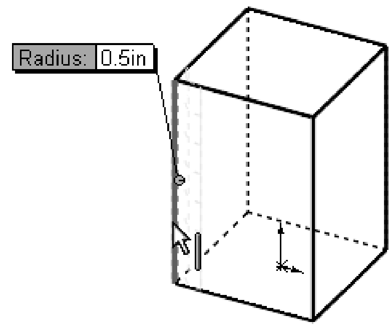
Activity: BATTERY Part-Fillet Feature Edge

Display the hidden edges.

86) Click **Hidden Lines Visible**.

Insert a Fillet feature.

87) Click the **left front vertical edge** as illustrated. Note the mouse pointer edge icon.



88) Click the **Fillet** feature tool. The Fillet PropertyManager is displayed.

89) Click the **Manual** tab. Edge<1> is displayed in the Items To Fillet box.

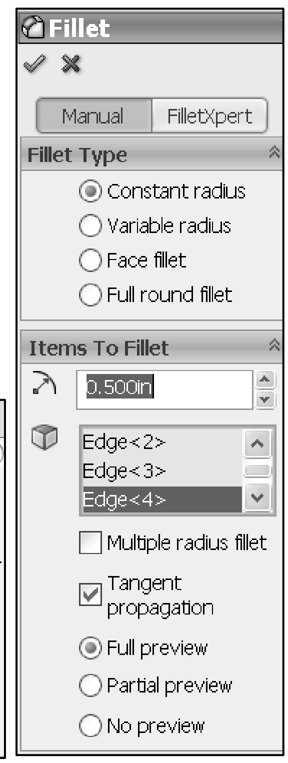
90) Constant radius is the default Fillet Type. Click the remaining **3 vertical edges**. The selected entities are displayed in the Items To Fillet box

91) Enter **.500**in, [**12.7**] for Radius. Accept the default settings.

92) Click **OK** from the Fillet PropertyManager. Fillet1 is displayed in the FeatureManager.

93) Click **Isometric view**.

94) Click **Shaded With Edges**.

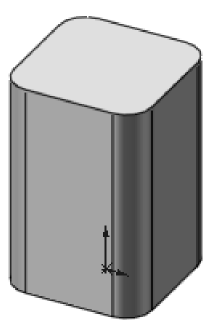


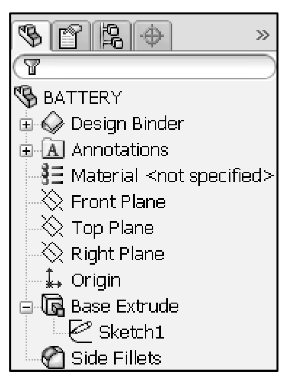
Rename the feature.

95) Rename **Fillet1** to **Side Fillets** in the FeatureManager.

Save the BATTERY.

96) Click **Save**.





BATTERY Part-Extruded Cut Feature

An Extruded Cut feature removes material. An Extruded Cut feature requires:

- Sketch plane (Top face).
- Sketch profile (Offset Entities).
- End Condition depth (Blind) in Direction 1.

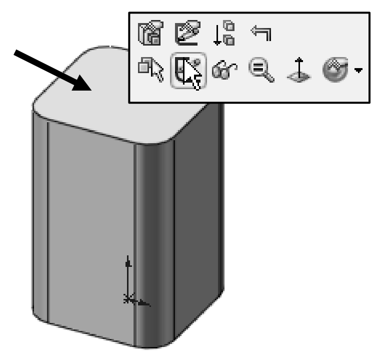
The Offset Entity Sketch tool uses existing geometry, extracts an edge or face and locates the geometry on the current Sketch plane.

Offset the existing Top face for the 2D sketch. Utilize the default Blind End Condition in Direction 1.

Activity: BATTERY Part-Extruded Cut Feature

Select the Sketch plane.

97) Right-click the **Top face** of the BATTERY in the Graphics window. Base Extruded is highlighted in the FeatureManager.



Create a sketch.

98) Click Sketch from the shortcut toolbar. The Sketch toolbar is displayed.

Display the face.

99) Click **Top view**.

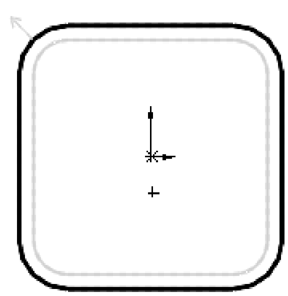
Offset the existing geometry from the boundary of the Sketch plane.

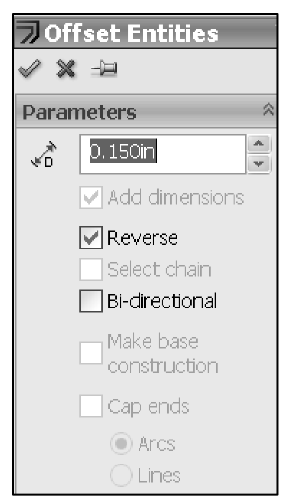
100) Click the **Offset Entities** Sketch tool. The Offset Entities PropertyManager is displayed.

101) Enter **.150**in, [**3.81**] for the Offset Distance.

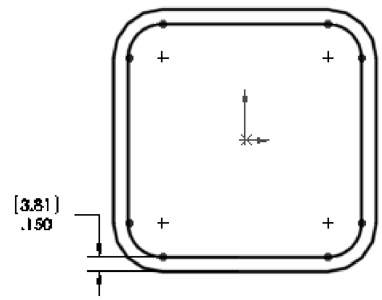
102) Click the **Reverse** box. The new Offset yellow profile displays inside the original profile.

103) Click **OK** from the Offset Entities PropertyManager.





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 in the ANSI standard.



Display the profile.

104) Click **Isometric view**.

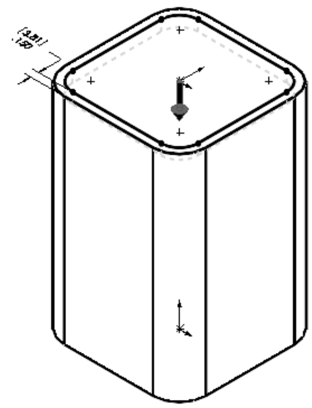
105) Click **Hidden Lines Removed**.

Insert an Extruded Cut feature.

106) Click the **Extruded Cut** feature tool. The Extrude PropertyManager is displayed.

107) Enter **.200**in, [**5.08**] for Depth in Direction 1. Accept the default settings.

108) Click **OK** from the Extrude PropertyManager. Extrude2 is displayed in the FeatureManager.



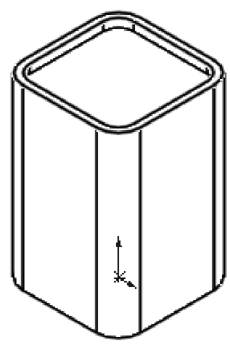
Rename the feature.

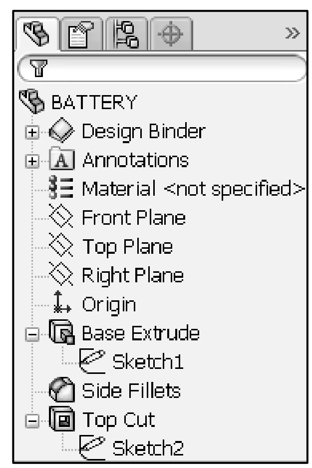
109) Rename **Extrude2** to **Top Cut** in the FeatureManager.

Save the BATTERY

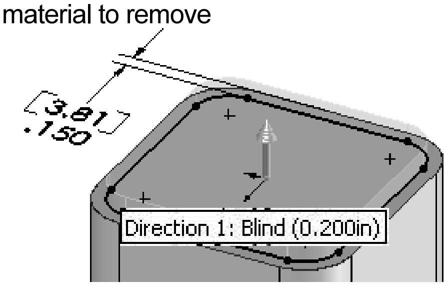
110) Click **Save**.

The Extrude PropertyManager contains numerous options. The Reverse Direction option determines the direction of the Extrude. The Extruded Cut feature is valid only when the direction arrow points into material to be removed.

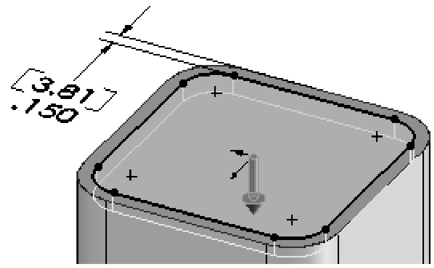


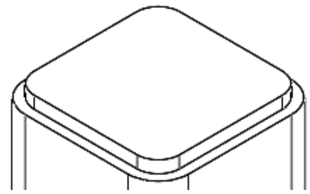


Cut direction not valid, no material to remove



The Flip side to cut option determines if the cut is to the inside or outside of the Sketch profile. The Flip side to cut arrow points outward. The Extruded Cut feature occurs on the outside of the BATTERY.





Extruded Cut with Flip side to cut option checked

BATTERY Part-Fillet Feature

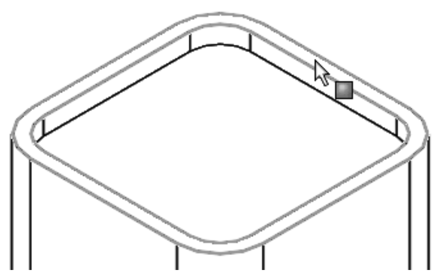
The Fillet feature tool rounds sharp edges with a constant radius by selecting a face. A Fillet requires a:

- Select face.
- Fillet radius.

Activity: BATTERY Part-Fillet Feature Face

Insert a Fillet feature on the top face.

111) Click the **top thin face** as illustrated. Note: The face icon feedback symbol.

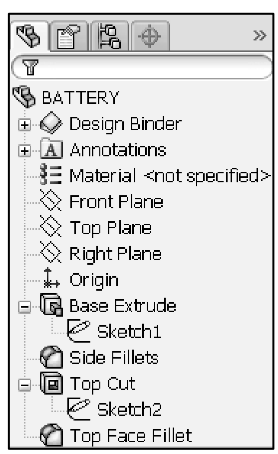


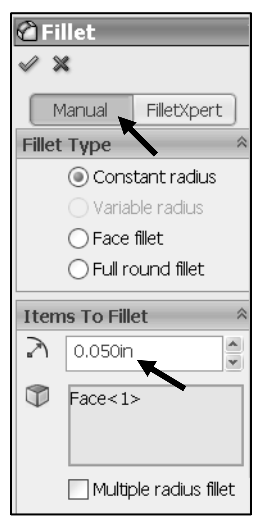
112) Click the **Fillet** feature tool. The Fillet PropertyManager is displayed. Face<1> is displayed in the Items To Fillet box.

113) Click the **Manual** tab. Create a Constant Radius for Fillet Type.

114) Enter **.050**in, [**1.27**] for Radius.

115) Click **OK** from the Fillet PropertyManager. Fillet2 is displayed in the FeatureManager.





Rename the feature.

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

117) Press the **f** key.

Save the BATTERY.

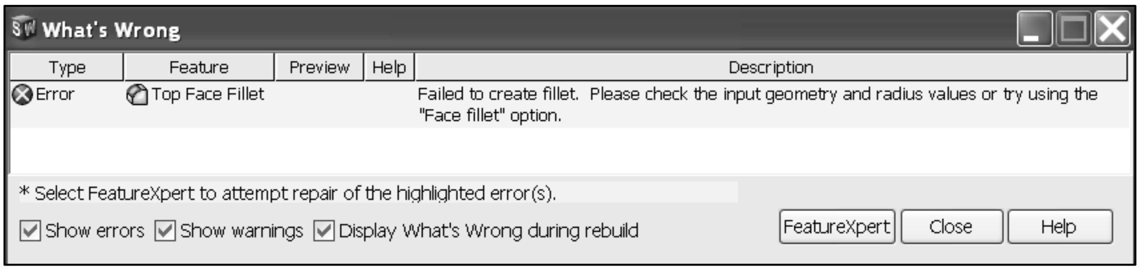
118) Click **Save**.

View the mouse pointer for feedback to select Edges or Faces for the fillet.

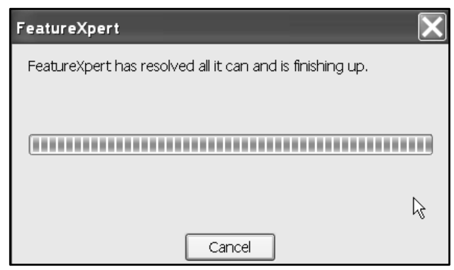
Do not select a fillet radius which is larger then the surrounding geometry.

Example: The top edge face width is .150in, [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 .200in, [5.08] is required for a fillet radius of .100in, [2.54].

The following error occurs when the fillet radius is too large for the existing geometry:



Avoid the fillet rebuild error. Use the FeatureXpert to address a constant radius fillet build error or manually enter a smaller fillet radius size. As an exercise, insert a large Fillet radius and use the FeatureXpert option.



BATTERY Part-Extruded Boss Feature

The Extruded Boss feature requires a truncated cone shape to represent the geometry of the BATTERY terminals. The Draft Angle option creates the tapered shape.

Sketch the first circle on the Top face. Utilize the Ctrl key to copy the first circle.

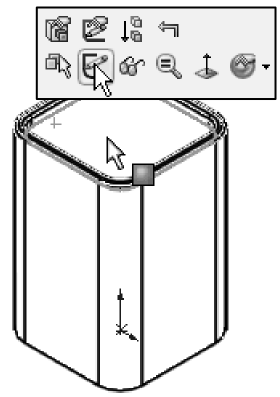
The dimension between the center points is critical. Dimension the distance between the two center points with an aligned dimension. The dimension text toggles between linear and aligned. An aligned dimension is created when the dimension is positioned between the two circles.

An angular dimension is required between the Right Plane and the centerline. 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 4.500in, [114.30].

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.

Activity: BATTERY Part-Extruded Boss Feature



Select the Sketch plane.

119) Right-click the **Top face** of the Top Cut feature in the Graphics window.

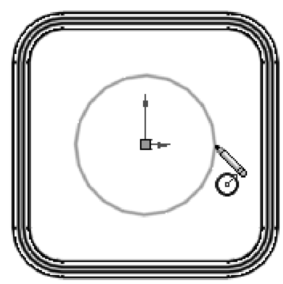
Create the sketch.

120) Click **Sketch** from the shortcut toolbar. The Sketch toolbar is displayed.

121) Click **Top view**.

Sketch the profile.

122) Click the **Circle** Sketch tool. The Circle PropertyManager is displayed.



123) Click the **center point** of the circle coincident to the Origin.

124) Drag and click the **mouse pointer** to the right of the Origin as illustrated.

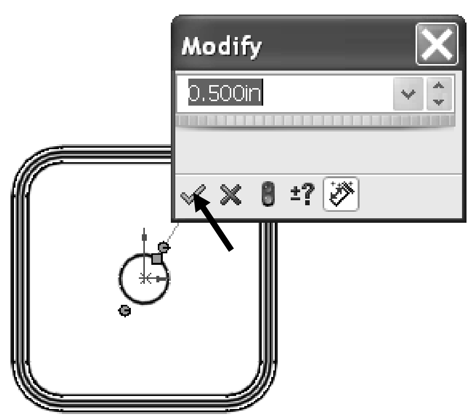
Add a dimension.

125) Click the **Smart Dimension** Sketch tool.

126) Click the **circumference** of the circle.

127) Click a **position** diagonally to the right.

128) Enter **.500**in, [**12.7**].



129) Click the **Green Check mark** in the Modify dialog box. The black sketch is fully defined.

Copy the sketched circle.

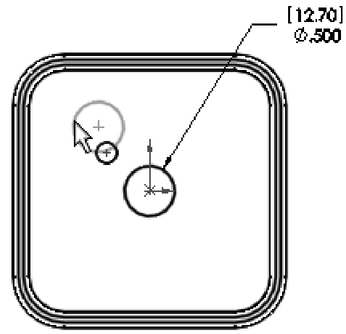
130) Right-click **Select**.

131) Hold the **Ctrl** key down.

132) Click and drag the **circumference** of the circle to the upper left quadrant as illustrated.

133) Release the **mouse button**.

134) Release the **Ctrl** key. The second circle is selected and is displayed in blue.



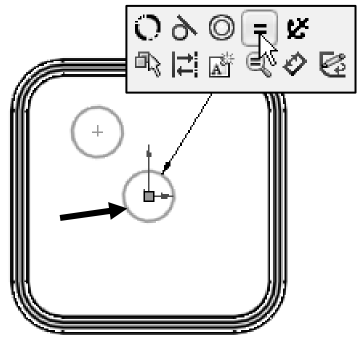
Add an Equal relation.

135) Hold the **Ctrl** key down.

136) Click the **circumference of the first circle**. The Properties PropertyManager is displayed. Both circles are selected and are displayed in green.

137) Release the **Ctrl** key.

138) Right-click **Make Equal** = from the shortcut toolbar.



139) Click **OK** from the Properties PropertyManager. The second circle remains selected.

Show the Right Plane for the dimension reference.

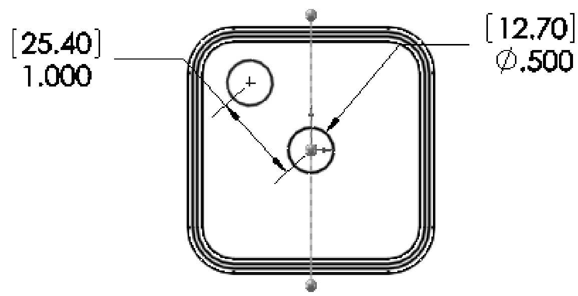
140) Click **Right Plane** from the FeatureManager. Click **Show**. The Right Plane is displayed in the Graphics window.

Add an aligned dimension.

141) Click the **Smart Dimension** Sketch tool.

142) Click the **two center points** of the two circles.

143) Click a **position** off the profile in the upper left corner.

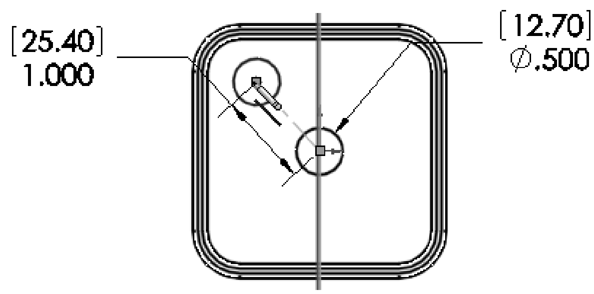


144) Enter **1.000**in, [**25.4**] for the aligned dimension.

145) Click the **Green Check mark** in the Modify dialog box.

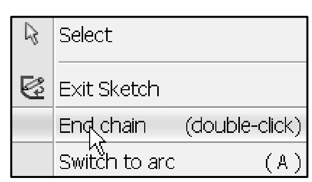
Insert a centerline.

146) Click the **Centerline** ⁞ Sketch tool. The Insert Line PropertyManager is displayed.



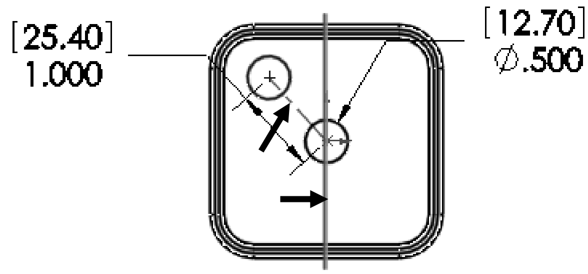
147) Sketch a centerline between the **two circle center points** as illustrated.

148) Right-click **End Chain** to end the line.



Shortcut: Double-click to end the centerline.

Shortcut: Press the Enter key to accept the value in the Modify dialog box. The Enter key replaces the Green Check mark.

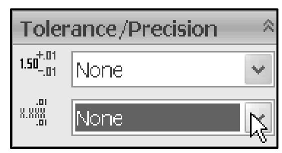


Add an angular dimension.

149) Click the **Smart Dimension** Sketch tool.

150) Click the **centerline** between the two circles.

151) Click the **Right Plane** (vertical line) in the Graphics window. Note: You can also click Right Plane in the FeatureManager.



152) Click a **position** between the centerline and the Right Plane, off the profile.

153) Enter **45**. Click **OK** from the Dimension PropertyManager.

Fit the model to the Graphics window.

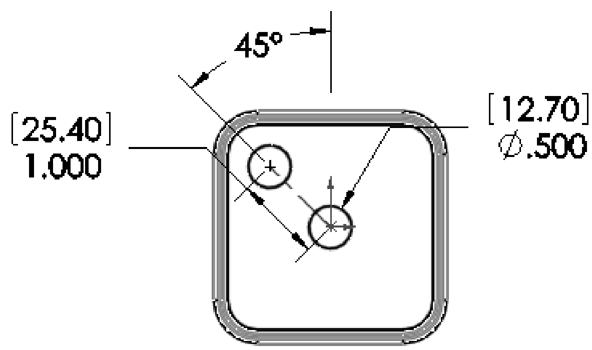
154) Press the **f** key.

Hide the Right Plane.

155) Right-click **Right Plane** in the FeatureManager.

156) Click **Hide**.

157) Click **Save**.



Create an angular dimension between three points or two lines. Sketch a centerline/construction line when an additional point or line is required.

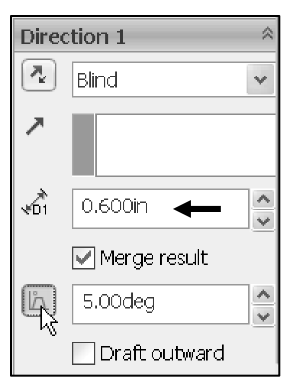
Insert an Extruded Boss feature.

158) Click **Isometric view**.

159) Click the **Extruded Boss/Base** feature tool. The Extrude PropertyManager is displayed. Blind is the default End Condition Type. Enter **.600**in, [**15.24**] for Depth in Direction 1.

160) Click the **Draft ON/OFF** button. Enter **5**deg in the Draft Angle box.

161) Click **OK** from the Extrude PropertyManager. The Extrude feature is displayed in the FeatureManager.



Rename the feature and sketch.

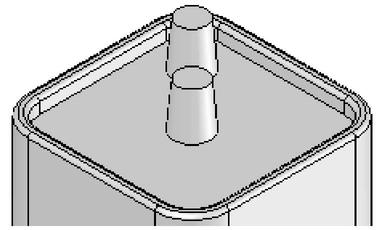
162) Rename **Extrude3** to **Terminals**.

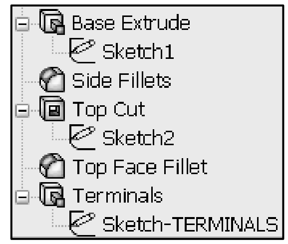
163) **Expand** Terminals.

164) Rename **Sketch3** to **Sketch-TERMINALS**.

165) Click **Shaded With Edges**.

166) Click **Save**.





Each time you create a feature of the same feature type, the feature name is incremented by one. Example: Extrude1 is the first Extrude feature. Extrude2 is the second Extrude feature. If you delete a feature, rename a feature or exit a SolidWorks session, the feature numbers will vary from those illustrated in the text.

Rename your features with descriptive names. Standardize on feature names that are utilized in mating parts. Example: Mounting Holes.

Measure the overall BATTERY height.

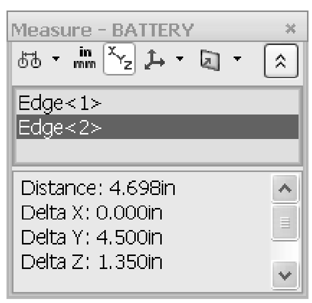
167) Click **Front view**.

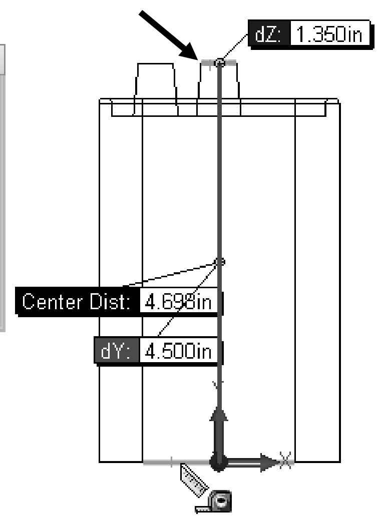
168) Click the **Measure** tool from the Evaluate tab in the CommandManager. The Measure - BATTERY dialog box is displayed.

169) Click the **top edge** of the battery terminal as illustrated.

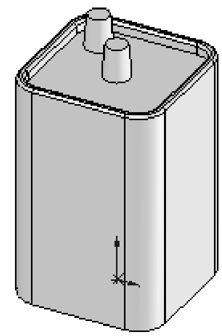
170) Click the **bottom edge** of the battery. The overall height, Delta Y is 4.500, [114.3]. Apply the Measure tool to insure a proper design.

171) Click **Close** from the Measure - BATTERY dialog box.





The Selection Filter option toggles the Selection Filter toolbar. When Selection Filters are activated, the mouse pointer displays the Filter icon . The Clear All Filters tool removes the current Selection Filters. The Help icon displays the SolidWorks Online Users Guide.



Display the Trimetric view.

172) Click **Trimetric view** from the Heads-up View toolbar.

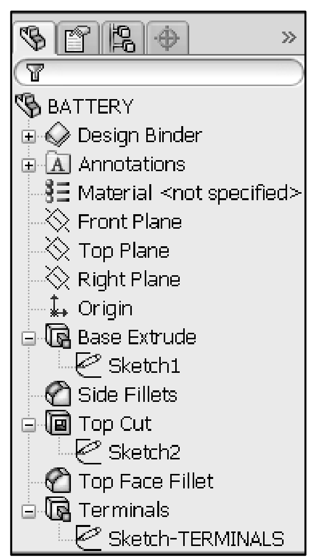
Save the BATTERY.

173) Click **Save** .

Additional information on Extruded Boss/Base Extruded Cut and Fillets is located in SolidWorks Help Topics. Keywords: Extruded (Boss/Base, Cut), Fillet (Constant radius fillet), Geometric relations (sketch, equal, midpoint), Sketch (rectangle, circle), Offset Entities and Dimensions (angular).

Refer to the Help, SolidWorks Tutorials, Fillet exercise for additional information.

Review of the BATTERY Part



The BATTERY utilized an Extruded Base feature sketched on the Top Plane. The rectangle was sketched with a diagonal centerline to build symmetry into the part. A Midpoint geometric relation centered the sketch on the Origin. The Equal relation created a square sketch.

The Fillet feature rounded sharp edges. All four edges were selected to combine common geometry into the same Fillet feature. The Fillet feature also rounded the top face. The Sketch Offset Entity created the profile for the Extruded Cut feature.

The Terminals were created with an Extruded Boss feature. You sketched a circular profile and utilized the Ctrl key to copy the sketched geometry.

A centerline was required to locate the two holes with an angular dimension. The Draft Angle option tapered the Extruded Boss feature. All feature names were renamed.