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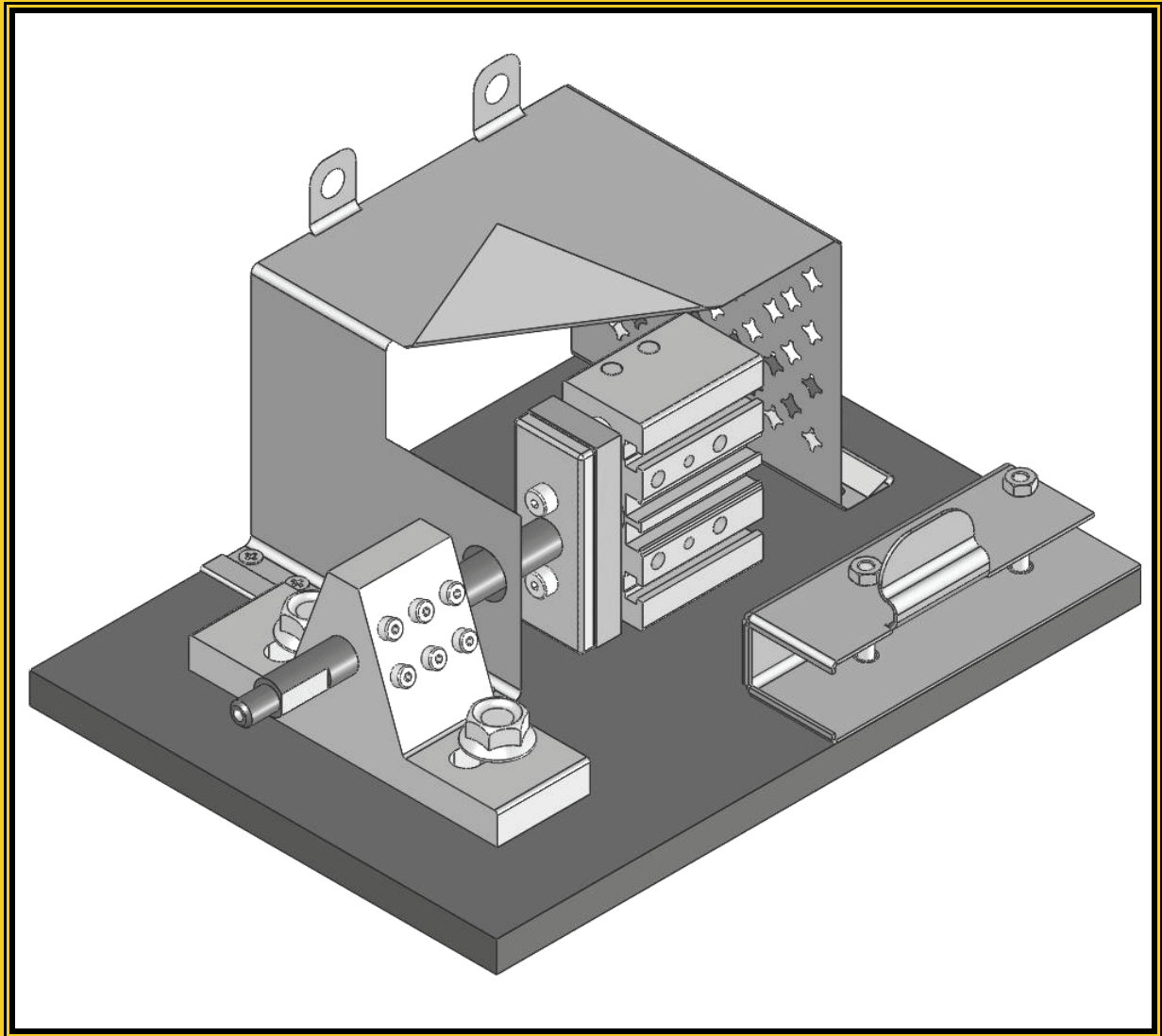
MultiMedia CD



# Engineering Design with SolidWorks 2010

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

Introductory Level ♦ Tutorial Style ♦ Video Instruction



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**SDC**  
PUBLICATIONS

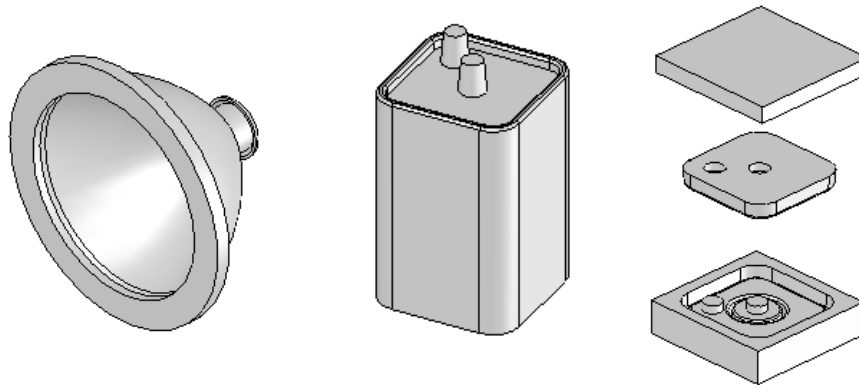
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# Project 4

## Extrude and Revolve Features



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

<b>Project Desired Outcomes:</b>	<b>Usage Competencies:</b>
<ul style="list-style-type: none"> <li>• Obtain your customer's requirements for the FLASHLIGHT assembly.</li> </ul>	<ul style="list-style-type: none"> <li>• Ability to incorporate Design Intent into sketches, features, parts and assemblies.</li> </ul>
<ul style="list-style-type: none"> <li>• Two Part Templates:                             <ul style="list-style-type: none"> <li>○ PART-IN-ANSI</li> <li>○ PART-MM-ISO</li> </ul> </li> </ul>	<ul style="list-style-type: none"> <li>• Aptitude to apply Document Properties in a custom Part Template.</li> </ul>
<ul style="list-style-type: none"> <li>• Four key parts:                             <ul style="list-style-type: none"> <li>○ BATTERY</li> <li>○ BATTERYPLATE</li> <li>○ LENS</li> <li>○ BULB</li> </ul> </li> </ul>	<ul style="list-style-type: none"> <li>• Specific knowledge of the following features: Extruded Boss/Base, Instant3D, Extruded Cut, Revolved Boss/Base, Revolved Cut, Dome, Shell, Circular Pattern and Fillet.</li> </ul>
<ul style="list-style-type: none"> <li>• Core and Cavity Tooling for the BATTERYPLATE.</li> </ul>	<ul style="list-style-type: none"> <li>• Understanding of the Mold tools: Scale, Parting Lines, Parting Surfaces, Shut-off Surfaces, Tooling Split and Draft.</li> </ul>

**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 two custom Part Templates. 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:
  - Instant3D
  - 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
  - Scale
  - Parting Lines
  - Shut-off Surfaces
  - Parting Surfaces
  - Tooling Split

### **Project Overview**

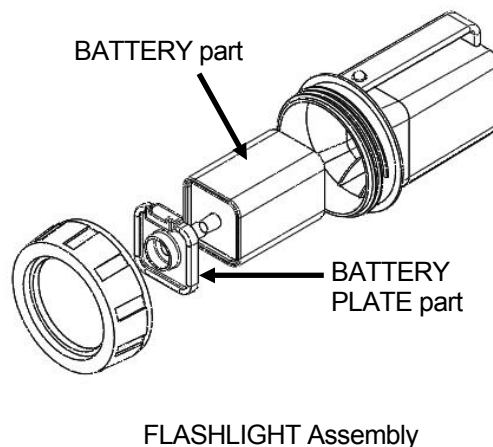
Start the design of the FLASHLIGHT assembly according to the customer's requirements. The FLASHLIGHT assembly will be cost effective, serviceable and flexible for future manufacturing revisions.

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 four parts for the FLASHLIGHT assembly in this Project:

- BATTERY
- BATTERYPLATE
- LENS
- BULB



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

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

- Extruded Boss/Base
- 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

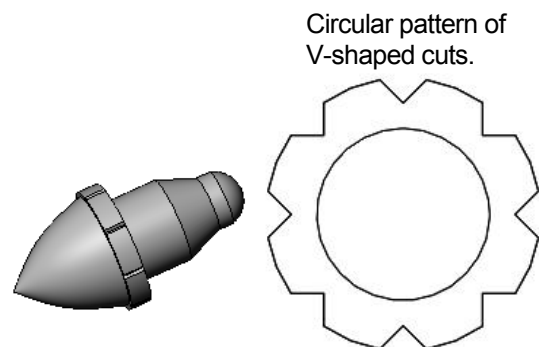
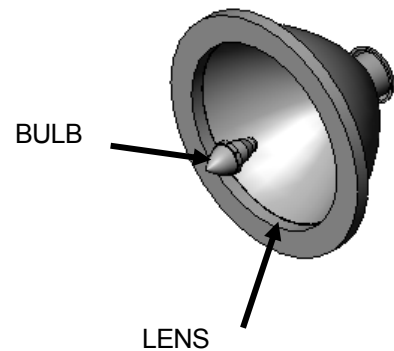
A Revolved feature requires a 2D sketch profile and a centerline. Utilize sketch geometry and sketch tools to create the following features:

- Revolved Boss/Base
- 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.

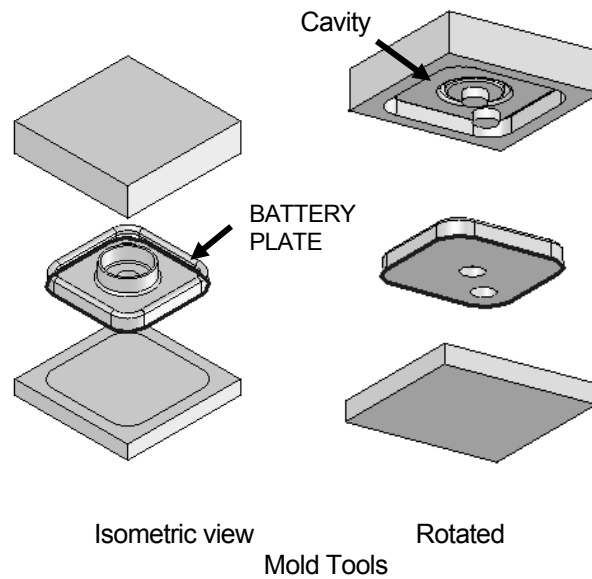


Utilize the Mold tools to create the Cavity tooling plates for the BATTERYPLATE part.

**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:

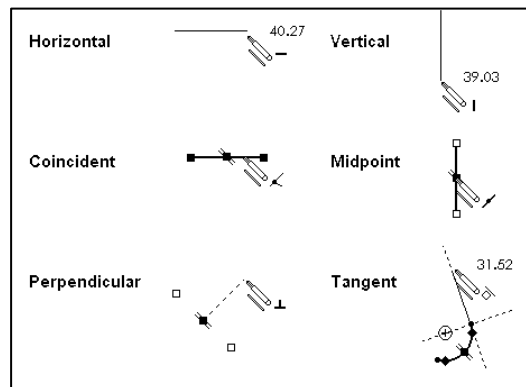
1. Sketch
2. Feature
3. Part
4. Assembly
5. Drawing

**1. Design Intent in the Sketch**

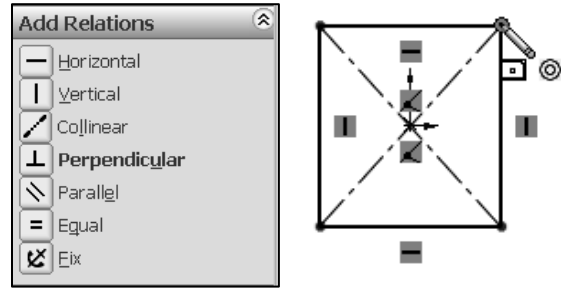
Build design intent in a sketch as the profile is created. A profile is determined from the selected Sketch Entity. Example: Corner Rectangle, Circle, Arc, Point, etc.

Apply symmetry into a profile through a sketch centerline, mirror entity and position about the reference planes and Origin.

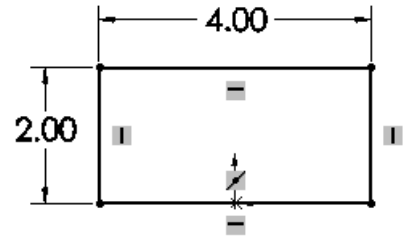
Build design intent as you sketch with automatic Geometric relations. Document the decisions made during the up-front design process. This is very valuable when you modify the design later.



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



Example A: Apply design intent to create a square profile. Sketch a rectangle. Apply the Center Rectangle tool. Note: No construction reference centerline or Midpoint relation is required with the Center Rectangle tool. Insert dimensions to define the square.

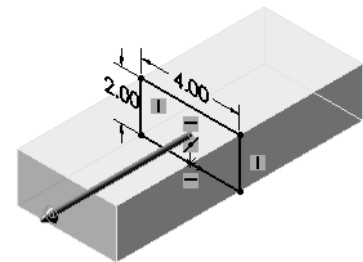


Example B: Develop a rectangular profile. Apply the Corner Rectangle tool. The bottom horizontal midpoint of the rectangular profile is located at the Origin. Add a Midpoint relation between the horizontal edge of the rectangle and the Origin. Insert two dimensions to define the width and height of the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

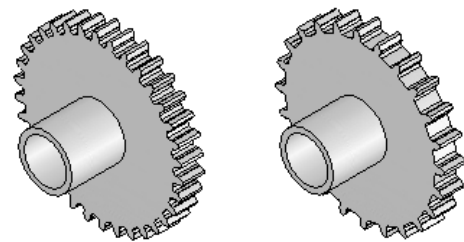
**2. Design Intent in the Feature**

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

Example A: The Boss-Extrude1 feature (Base 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.

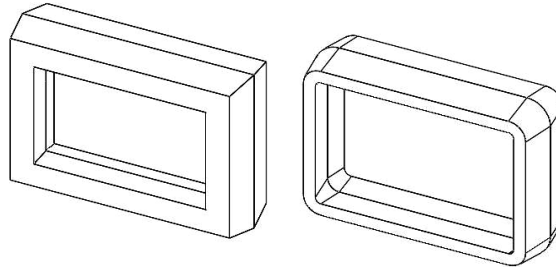




### 3. Design Intent in the Part

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

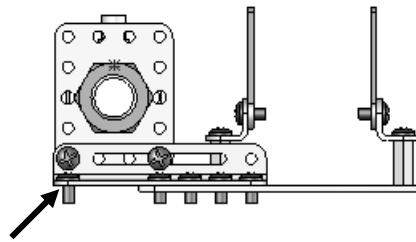
Example A: Feature order. Is the entire part symmetric? Feature order affects the part. Apply the Shell feature before the Fillet feature and the inside corners remain perpendicular.



### 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.



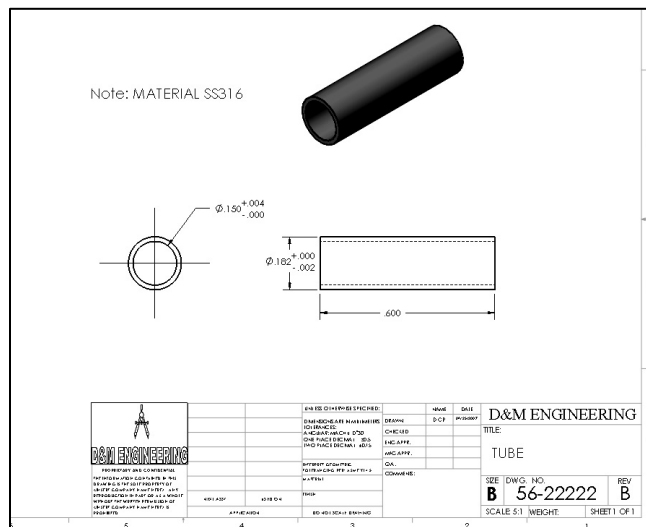
### 5. Design Intent in the Drawing

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  $+.000/-0.002$  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.



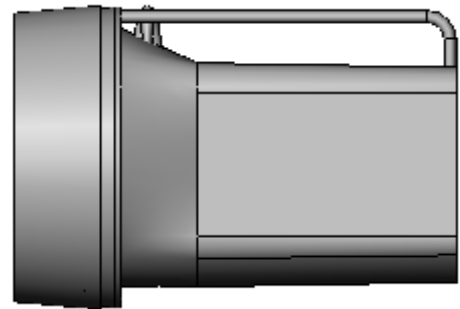
## Project Situation

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.

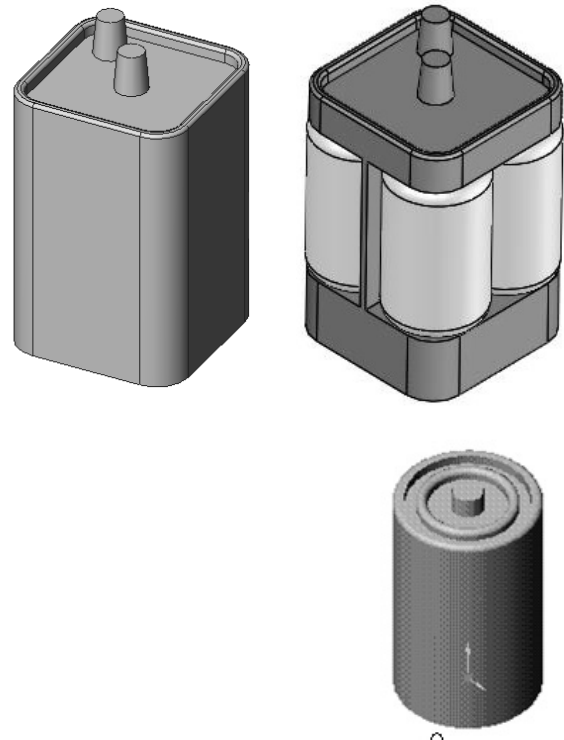


Your company's standard product line does not address the above key customer requirements. The customer made it clear that there is no room for negotiation on the key product requirements.

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, LENSAP and HOUSING.

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

\*Parts addressed in Project 5.

The BATTERYPLATE, LENSAP 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.

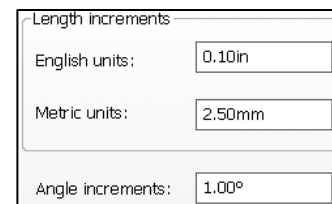
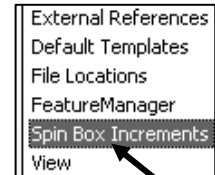
### Activity: Create Two Part Templates

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.

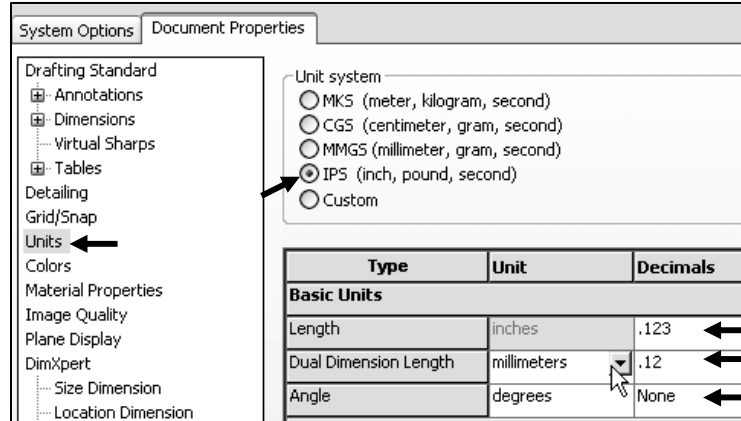
Set the Dimensioning Standard to ANSI.

- 3) Click **Options** , **Document Properties** tab.
- 4) Click the **System Options** tab.
- 5) Click **Spin Box Increments**. View the default settings.
- 6) Click **inside** the English units box.
- 7) Enter **.10in**.
- 8) Click **inside** the Metric units box.
- 9) Enter **2.50mm**.
- 10) Click the **Document Properties** tab.
- 11) Select **ANSI** from the Overall drafting standard drop-down menu.



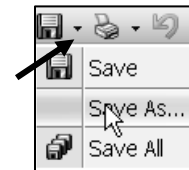
Set the part units for inch.

- 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.



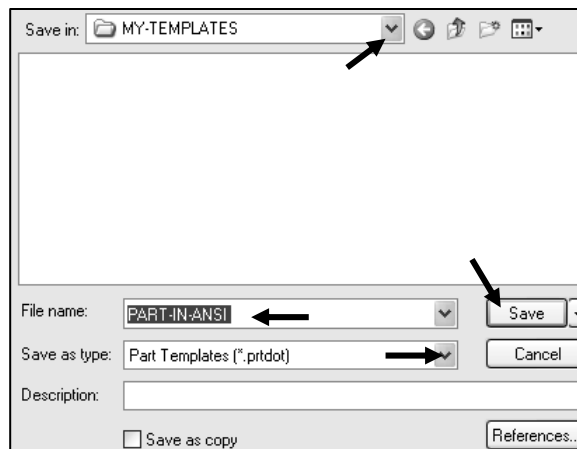
Save the part template.

- 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 Overall drafting standard drop-down menu.

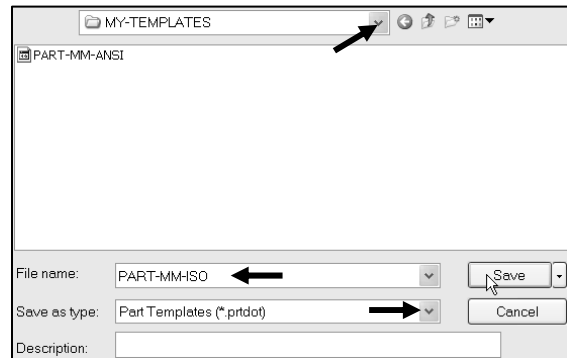


Set the part units for millimeter.

- 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** from the Document Properties - Units dialog box.

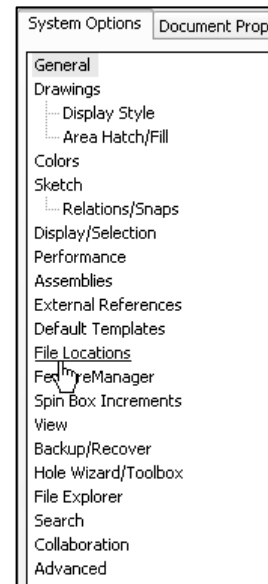
Save the part template.

- 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*.

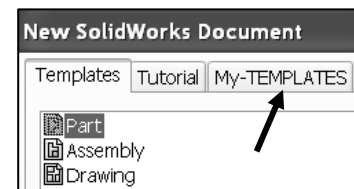
- 36) Click **Options** from the Menu bar.
- 37) Click **File Locations** from the System Options tab.
- 38) Select **Document Templates** from Show folders for.
- 39) Click the **Add** button.
- 40) Select the **MY-TEMPLATES** folder.
- 41) Click **OK** from the Browse for Folder dialog box.
- 42) Click **OK** from the System Options dialog box.




Close all documents.

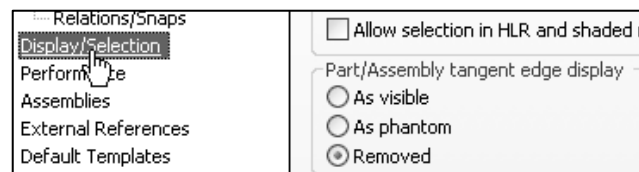
- 43) Click **Windows, Close All** from the Menu bar.

 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.

 To remove Tangent edges on a model, click **Display/Selections** from the Options menu, check the **Removed** box.



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.

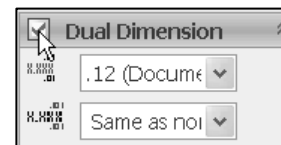
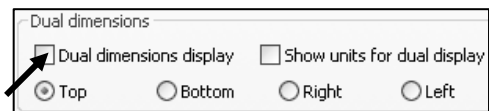
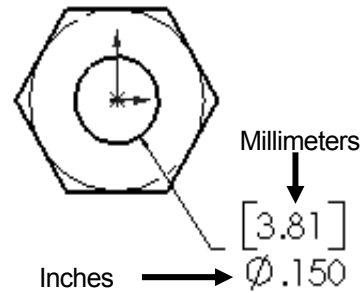
The primary units in this Project are IPS, (inch, pound, seconds).

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, Dimensions. Check the Dual dimensions display box as illustrated.

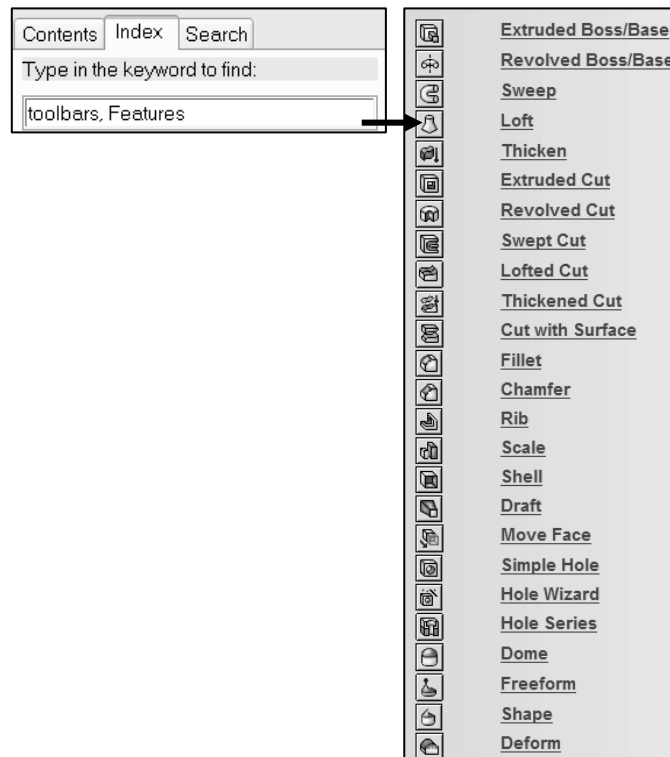
To set dual dimensions for an active document, check the Dual Dimension box in the Dimension PropertyManager.



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. Note: Other templates were created in the previous project. The Document Properties Dimensioning Standard, units and decimal places are stored in the Part Templates.



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

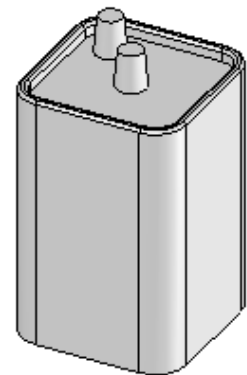
Note: In some network locations and school environments, the File Locations option must be set to MY-TEMPLATES for each session of SolidWorks.

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.

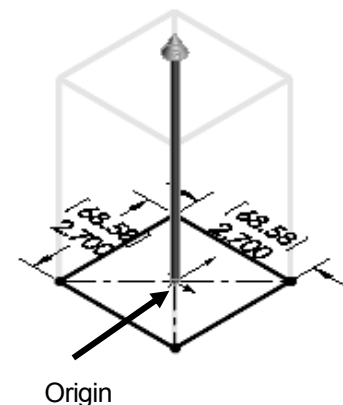
A 6-Volt lantern battery weighs approximately 1.38 pounds, (0.62kg). Locate the center of gravity closest to the center of the battery.



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


Utilize the Instant3D tool to create the Extruded Boss/Base  feature vs. using the Boss-Extrude PropertyManager. The Extrude Boss/Base features add material. The Base feature (Boss-Extrude1) is the first feature of the part. Note: The default End Condition for Instant3D is Blind!

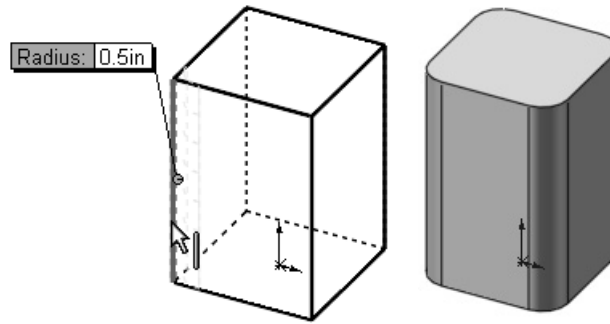
Apply symmetry. Use the Center Rectangle Sketch tool on the Top Plane. The 2D Sketch profile is centered at the Origin.




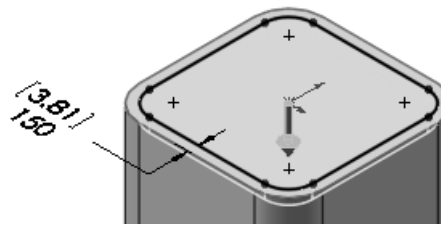


Extend the profile perpendicular ( $\perp$ ) to the Top Plane.

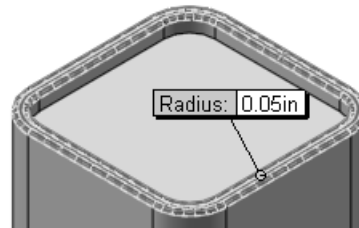
Utilize the Fillet  feature to round the four vertical edges.




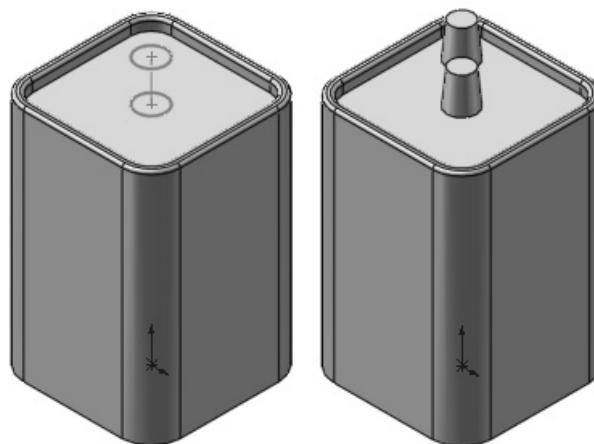
The Extruded Cut  feature removes material from the top face. Utilize the top face for the Sketch plane. Utilize the Offset Entity Sketch tool to create the profile.



Utilize the Fillet feature  to round the top narrow face.



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



## BATTERY Part-Extruded Boss/Base Feature


The Extruded Boss/Base feature requires:

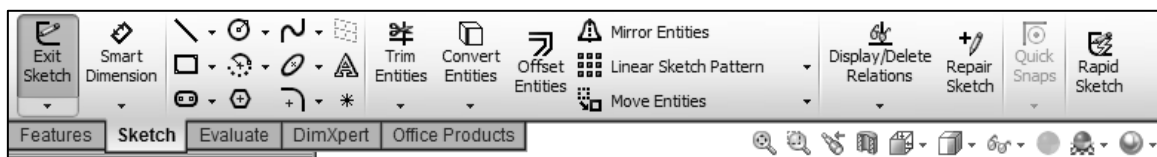
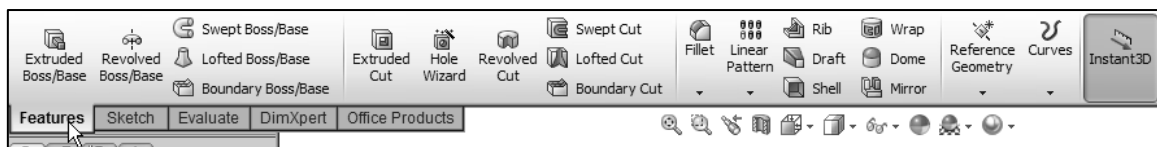
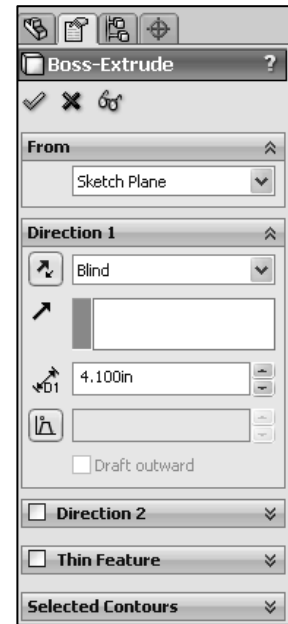
- Sketch plane (Top)
- Sketch profile (Rectangle)
  - Geometric relations and dimensions
- End Condition Depth (Blind) in Direction 1

Create a new part named, BATTERY. Insert an Extruded Boss/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 two horizontal lines and two 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.

 Alternate between the Features tab and the Sketch tab in the CommandManager to display the available Feature and Sketch tools for the Part document.




**Activity: BATTERY Part-Create the Extruded Base Feature**

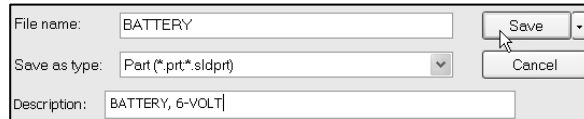
Create a new part.

- 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.






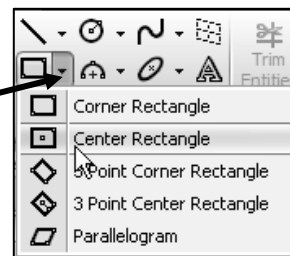
Select the Sketch plane.


- 52) Right-click **Top Plane** from the FeatureManager. This is your Sketch plane.




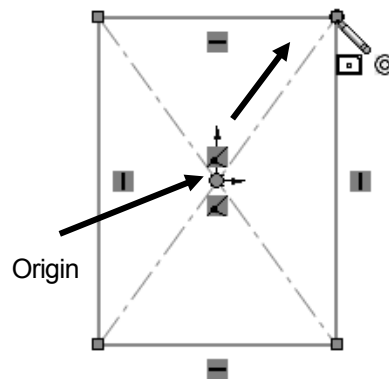
Sketch the 2D Sketch profile centered at the Origin.

- 53) Click **Sketch**  from the Context toolbar. The Sketch toolbar is displayed.
- 54) Click the **Center Rectangle**  Sketch tool. The Center Rectangle  icon is displayed.
- 55) Click the **Origin**. This is your first point.
- 56) Drag and click the **second point** in the upper right quadrant as illustrated. The Origin is located in the center of the sketch profile. The Center Rectangle Sketch tool automatically applies equal relations to the two horizontal and two vertical lines. A midpoint relation is automatically applied to the Origin.







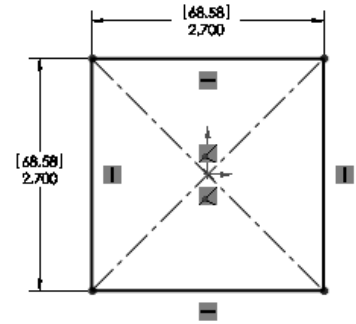
 A key goal of this book is to expose the new user to various tools, techniques and procedures. It may not always use the most direct tool or process.

 Click **View, Sketch Relations** from the Main menu to view sketch relations in the Graphics area.



Dimension the sketch.

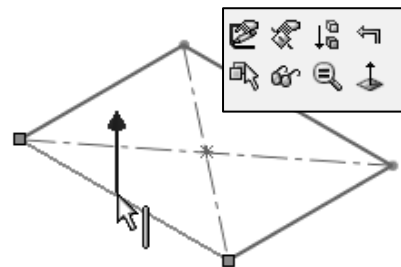
- 57) Click the **Smart Dimension**  Sketch tool.
- 58) Click the **top horizontal line**.
- 59) Click a **position** above the horizontal line.
- 60) Enter **2.700in**, [68.58] for width.
- 61) Click the **Green Check mark**  in the Modify dialog box.
- 62) Enter **2.700in**, [68.58] for height as illustrated.
- 63) Click the **Green Check mark**  in the Modify dialog box. The black Sketch status is fully defined.
- 64) Click **OK**  from the Dimension PropertyManager.



Exit the Sketch.

- 65) Click **Exit Sketch**.



Insert an Extruded Boss/Base feature. Apply the Instant3D tool. The Instant3D tool provides the ability to drag geometry and dimension manipulator points to resize or to create features directly in the Graphics window.

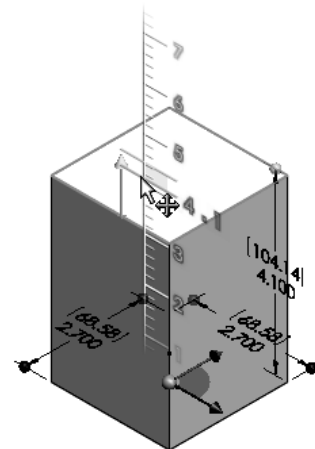
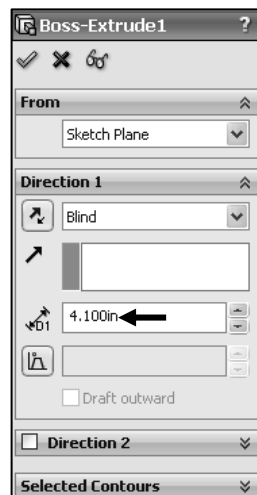


Use the on-screen ruler.

- 66) Click **Isometric view** from the Heads-up View toolbar.
- 67) Click the **front horizontal line** as illustrated. A green arrow is displayed.
- 68) Click and drag the **green/red arrow** upward.
- 69) Click the on-screen ruler at **4.1in**, [104.14] as illustrated. This is the depth in direction 1. The extrude direction is upwards. Boss-Extrude1 is displayed in the FeatureManager.

Check the Boss-Extrude1 feature depth dimension.

- 70) Right-click **Boss-Extrude1** from the FeatureManager.
- 71) Click **Edit Feature**  from the Context toolbar. 4.100in is displayed for depth. Blind is the default End Condition. Note: If you did not select the correct depth, input the depth in the Boss-Extrude1 PropertyManager.
- 72) Click **OK**  from the Boss-Extrude1 PropertyManager.





Modify the **Spin Box Increments** in System Options to display different increments in the on-screen ruler.

Fit the part to the Graphics window.

**73)** Press the **f** key.

Rename the Boss-Extrude1 feature.

**74)** Rename **Boss-Extrude1** to **Base Extrude**.

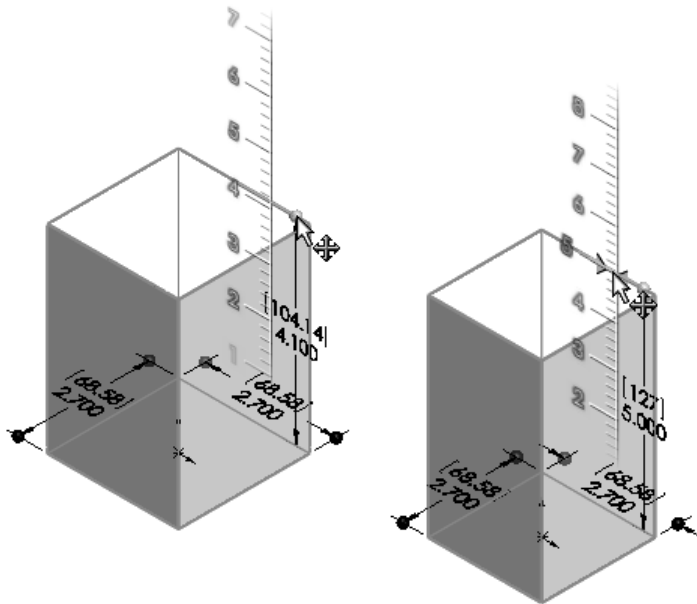
Save the BATTERY.

**75)** Click **Save** .


Modify the BATTERY.

**76)** Click **Base Extrude** from the FeatureManager. Note: Instant3D is activated by default.

**77)** Drag the **manipulator point** upward and click the on-screen ruler to create a **5.000in, [127]** depth as illustrated. Blind is the default End Condition.



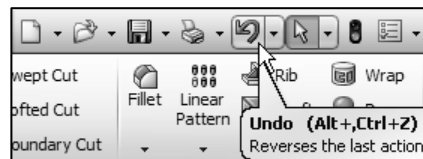
Return to the 4.100 depth.

**78)** Click the **Undo**  button from the Menu bar. The depth of the model is 4.100in, [104.14]. Blind is the default End Condition. Practice may be needed to select the correct on-screen ruler dimension.

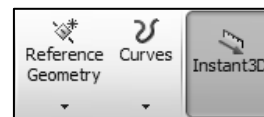


The color of the sketch indicates the sketch status.


- 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



The Instant3D tool is active by default in the Features toolbar located in the CommandManager.



## BATTERY Part-Fillet Feature Edge

Fillet features remove sharp edges. Utilize Hidden Lines Visible  from the Heads-up View toolbar to display hidden edges.

An edge Fillet feature requires:


- A selected 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.

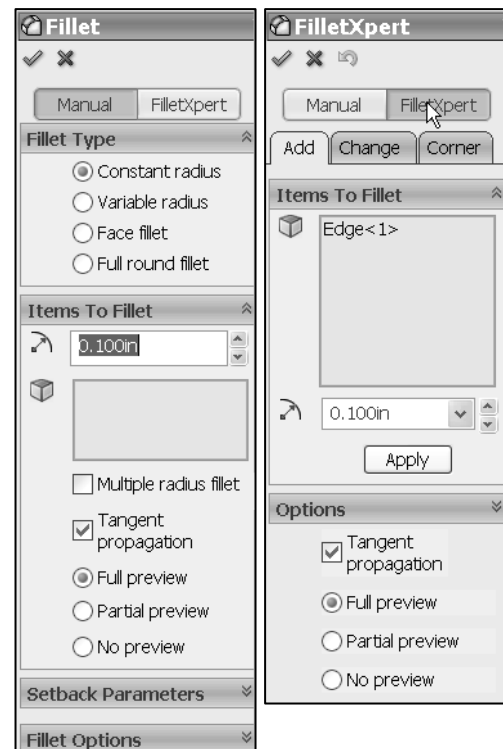
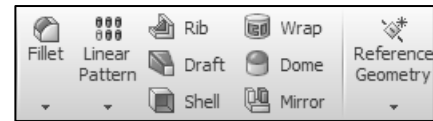
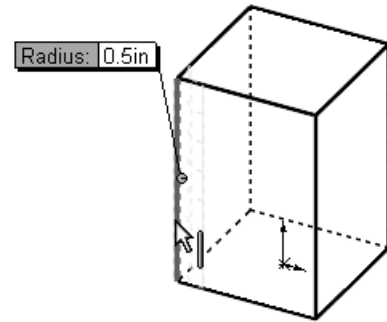
The order of selection for the Fillet feature is not predetermined. Select edges to produce the correct result.

The Fillet feature uses the Fillet PropertyManager. The Fillet PropertyManager 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.




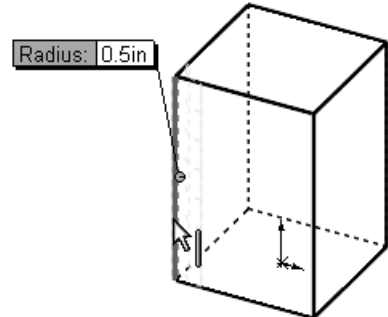
**Activity: BATTERY Part-Fillet Feature Edge**





Display the hidden edges.

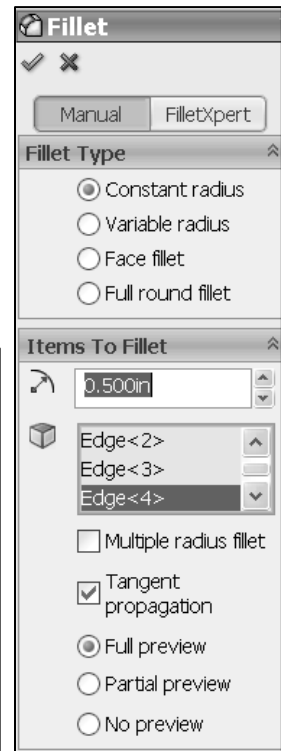
- 79) Click **Hidden Lines Visible**  from the Heads-up View toolbar.

Insert a Fillet feature.

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




- 81) Click the **Fillet**  feature tool. The Fillet PropertyManager is displayed.
- 82) Click the **Manual** tab. Edge<1> is displayed in the Items To Fillet box.
- 83) Constant radius is the default Fillet Type. Click the remaining **3 vertical edges**. The selected entities are displayed in the Items To Fillet box.
- 84) Enter **.500in, [12.7]** for Radius. Accept the default settings.
- 85) Click **OK**  from the Fillet PropertyManager. Fillet1 is displayed in the FeatureManager.
- 86) Click **Isometric view**  from the Heads-up View toolbar.
- 87) Click **Shaded With Edges**  from the Heads-up View toolbar.

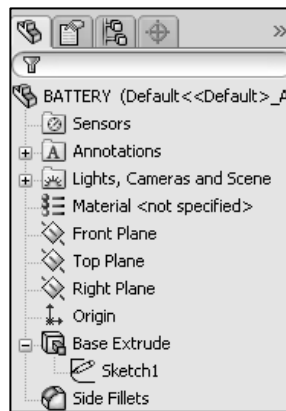
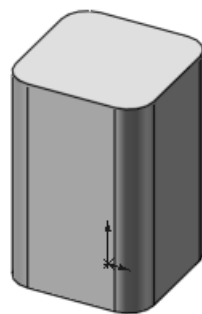


Rename the feature.

- 88) Rename **Fillet1** to **Side Fillets** in the FeatureManager.

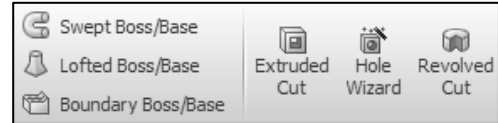
Save the BATTERY.

- 89) 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.

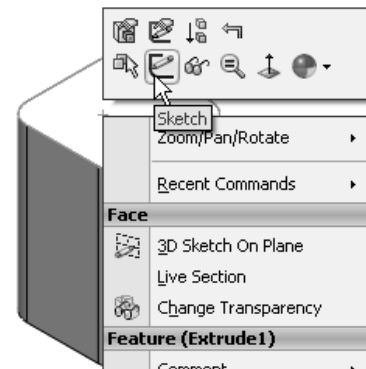
### Activity: BATTERY Part-Extruded Cut Feature

Select the Sketch plane.

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

Create a sketch.

- 91)** Click Sketch  from the Context toolbar. The Sketch toolbar is displayed.

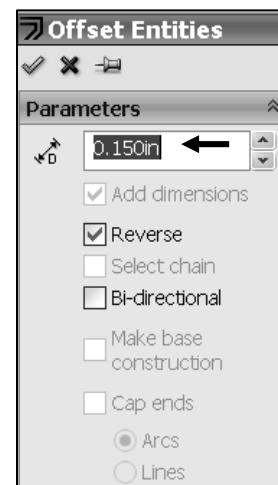
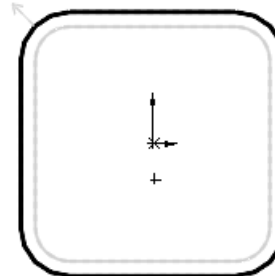


Display the face.

- 92)** Click **Top view**  from the Heads-up View toolbar.

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

- 93)** Click the **Offset Entities**  Sketch tool. The Offset Entities PropertyManager is displayed.
- 94)** Enter **.150in**, [3.81] for the Offset Distance.
- 95)** Click the **Reverse** box. The new Offset yellow profile displays inside the original profile.
- 96)** 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.


97) Click **Isometric view**  from the Heads-up View toolbar.

98) Click **Hidden Lines Removed**  from the Heads-up View toolbar.

Insert an Extruded Cut feature. As an exercise, use the Instant3D tool to create the Extruded Cut feature. In this section, the Extruded-Cut PropertyManager is used. Note: With the Instant3D tool, you may lose the design intent of the model.

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

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

101) Click **OK**  from the Cut-Extrude PropertyManager. Cut-Extrude1 is displayed in the FeatureManager.

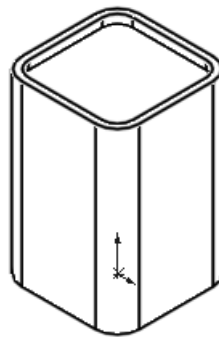
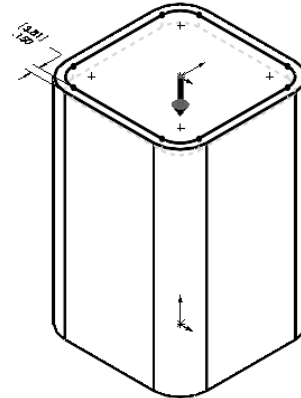
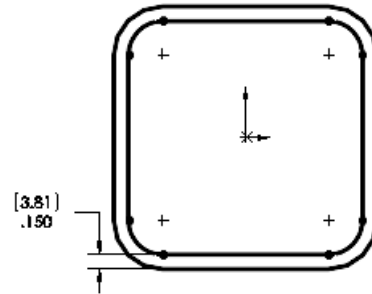
Rename the feature.

102) Rename **Cut-Extrude1** to **Top Cut** in the FeatureManager.

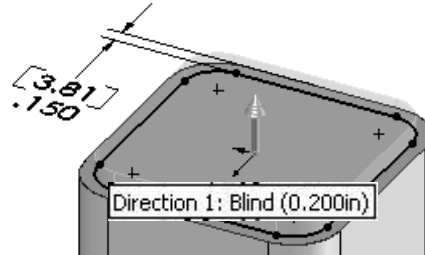
Save the BATTERY

103) Click **Save** .

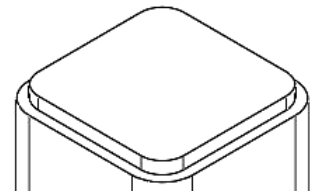
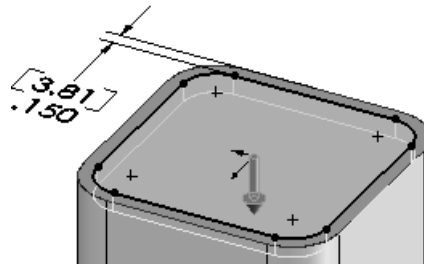
The Cut-Extrude PropertyManager contains numerous options. The Reverse Direction option determines the direction of the Extrude. The Extruded Cut feature is valid only when the direction arrow points into material to be removed.



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

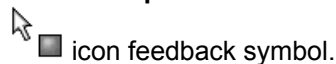
The Fillet  feature tool rounds sharp edges with a constant radius by selecting a face. A Fillet requires a:


- A selected face
- Fillet radius

### Activity: BATTERY Part-Fillet Feature Face

Insert a Fillet feature on the top face.


**104)** Click the **top thin face** as illustrated. Note: The face

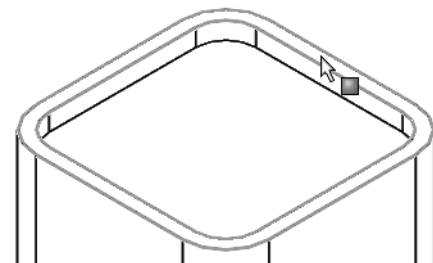


**105)** Click the **Fillet**  feature tool. The Fillet PropertyManager is displayed. Face<1> is displayed in the Items To Fillet box.

**106)** Click the **Manual** tab. Create a Constant radius for Fillet Type.

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

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



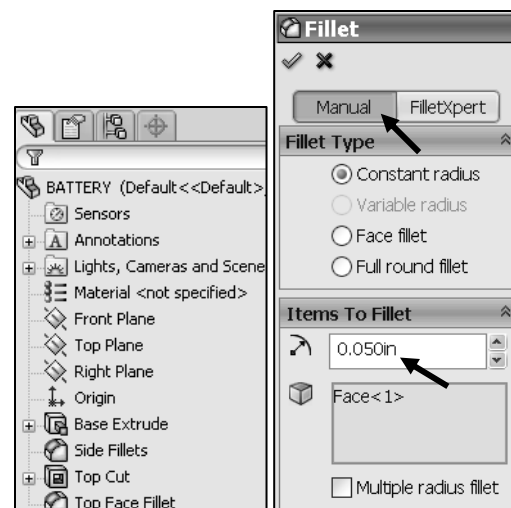
Rename the feature.

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

**110)** Press the **f** key.

Save the BATTERY.

**111)** Click **Save** .



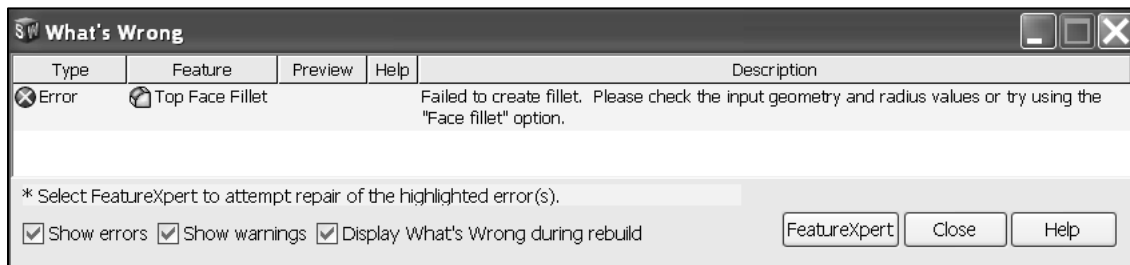
View the mouse pointer for feedback to select Edges or Faces for the fillet.



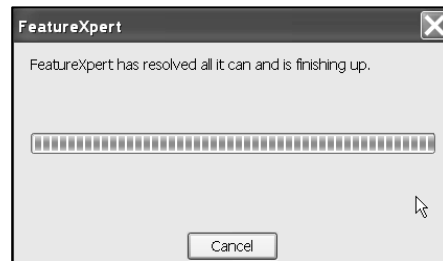
Do not select a fillet radius which is larger than the surrounding geometry.

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/Base 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, [114.3].

Calculate the depth of the extrusion: For inches:  $4.500\text{in} - (4.100\text{in Base-Extrude height} - .200\text{in Offset cut depth}) = .600\text{in}$ . The depth of the extrusion is .600in.

For millimeters:  $114.3\text{mm} - (104.14\text{mm Base-Extrude height} - 5.08\text{mm Offset cut depth}) = 15.24\text{mm}$ . The depth of the extrusion is 15.24mm.

**Activity: BATTERY Part-Extruded Boss Feature**

Select the Sketch plane.

**112)** Right-click the **Top face** of the Top Cut feature in the Graphics window. This is your Sketch plane.

Create the sketch.

**113)** Click **Sketch** from the Context toolbar. The Sketch toolbar is displayed.

**114)** Click **Top view** from the Heads-up View toolbar.

Sketch the profile.

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

**116)** Click the **centerpoint** of the circle coincident to the



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

Add a dimension.

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

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

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

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

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

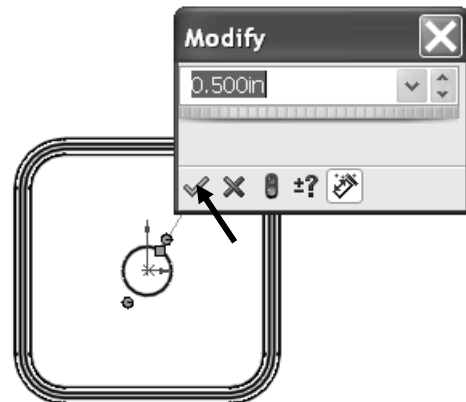
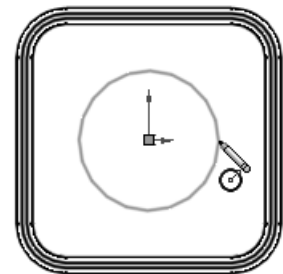
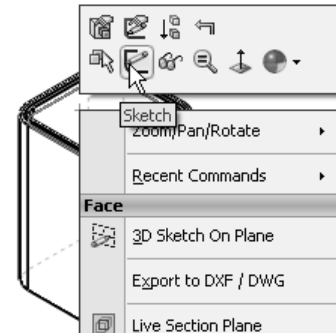
Copy the sketched circle.

**123)** Right-click **Select** to de-select the Smart Dimension Sketch tool.

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

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

**126)** Release the **mouse button**.



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


Add an Equal relation.

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

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

**130)** Release the **Ctrl** key.

**131)** Right-click **Make Equal**  from the Context toolbar.

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

Show the Right Plane for the dimension reference.

**133)** Click **Right Plane** from the FeatureManager.

**134)** Click **Show**. The Right Plane is displayed in the Graphics window.


Add an aligned dimension.

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


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

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

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

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

Insert a centerline.

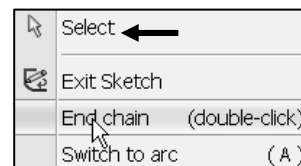
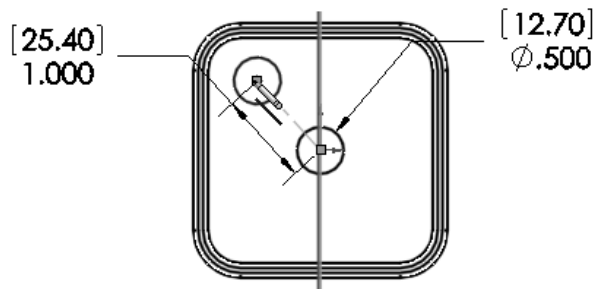
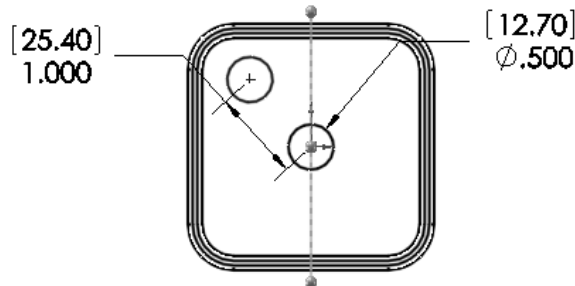
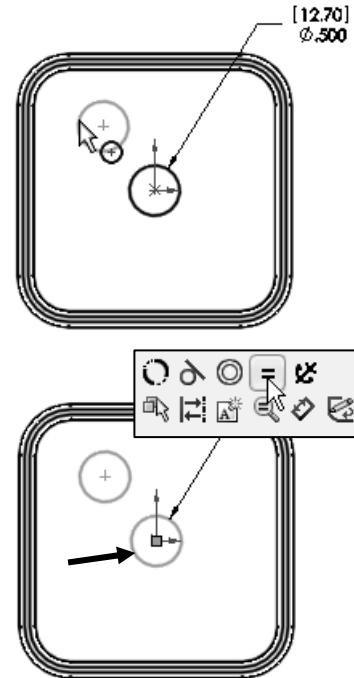
**140)** Click the **Centerline**  Sketch tool. The Insert Line PropertyManager is displayed.


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

**142)** Right-click **Select** to end the line.





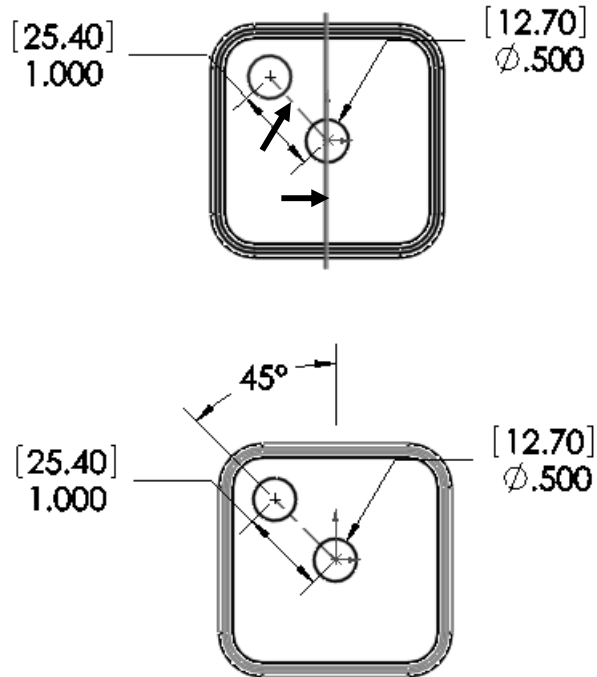
Double-click to end the centerline.




 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.

- 143)** Click the **Smart Dimension**  Sketch tool. Click the **centerline** between the two circles.
- 144)** Click the **Right Plane** (vertical line) in the Graphics window. Note: You can also click Right Plane in the FeatureManager.
- 145)** Click a **position** between the centerline and the Right Plane, off the profile.
- 146)** Enter **45**. Click **OK**  from the Dimension PropertyManager.






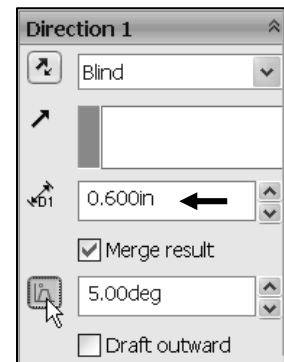
Fit the model to the Graphics window.

- 147)** Press the **f** key.
- Hide the Right Plane.
- 148)** Right-click **Right Plane** in the FeatureManager.
- 149)** Click **Hide** from the Context toolbar. 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.

- 150)** Click **Isometric view**  from the Heads-up View toolbar.
- 151)** Click the **Extruded Boss/Base**  feature tool. The Boss-Extrude PropertyManager is displayed. Blind is the default End Condition Type.
- 152)** Enter **.600in**, **[15.24]** for Depth in Direction 1.
- 153)** Click the **Draft ON/OFF** button.
- 154)** Enter **5deg** in the Draft Angle box.
- 155)** Click **OK**  from the Boss-Extrude PropertyManager. The Boss-Extrude2 feature is displayed in the FeatureManager.



Rename the feature and sketch.

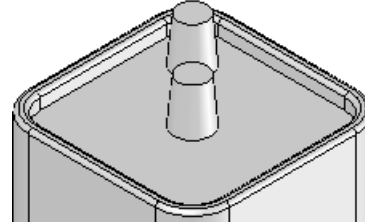
**156)** Rename **Boss-Extrude2** to **Terminals**.

**157)** **Expand** Terminals.

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

**159)** Click **Shaded With Edges**  from the Heads-up View toolbar.

**160)** Click **Save** .




Each time you create a feature of the same feature type, the feature name is incremented by one. Example: Boss-Extrude1 is the first Extrude feature. Boss-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.

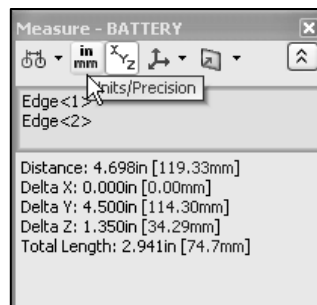


Rename your features with descriptive names. Standardize on feature names that are utilized in mating parts. Example: Mounting Holes.

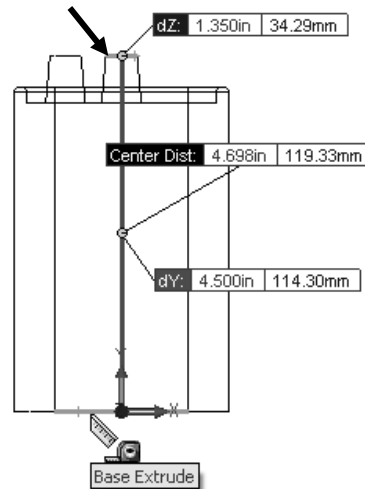
Measure the overall BATTERY height.

**161)** Click **Front view**  from the Heads-up View toolbar.


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



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

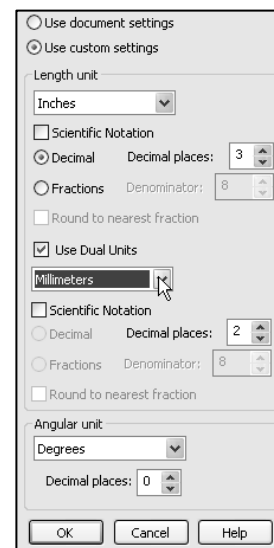






**164)** Click the **bottom edge** of the battery. The overall height, Delta Y is 4.500, [114.3]. Apply the Measure tool to ensure a proper design.

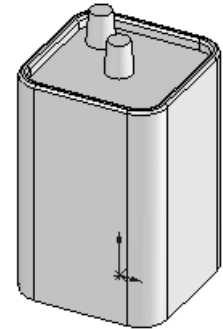
**165)** **Close**  the Measure - BATTERY dialog box.



The Measure tool provides the ability to display dual dimensions. Click **Units/Precision** from the Measure dialog box. Check the **Use custom settings** box. Select **dual unit** type. Select **decimal places**. Click **OK**.



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.

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

Save the BATTERY.

**167)** 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 a 2D Sketch profile located on the Top Plane. The 2D Sketch profile utilized the Center Rectangle Sketch tool. The Center Rectangle Sketch tool applied equal geometric relations to the two horizontal and two vertical lines. A midpoint relation was added to the Origin.

The Extruded Boss/Base feature was created using the Instant3D tool. Blind was the default End Condition. 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.

