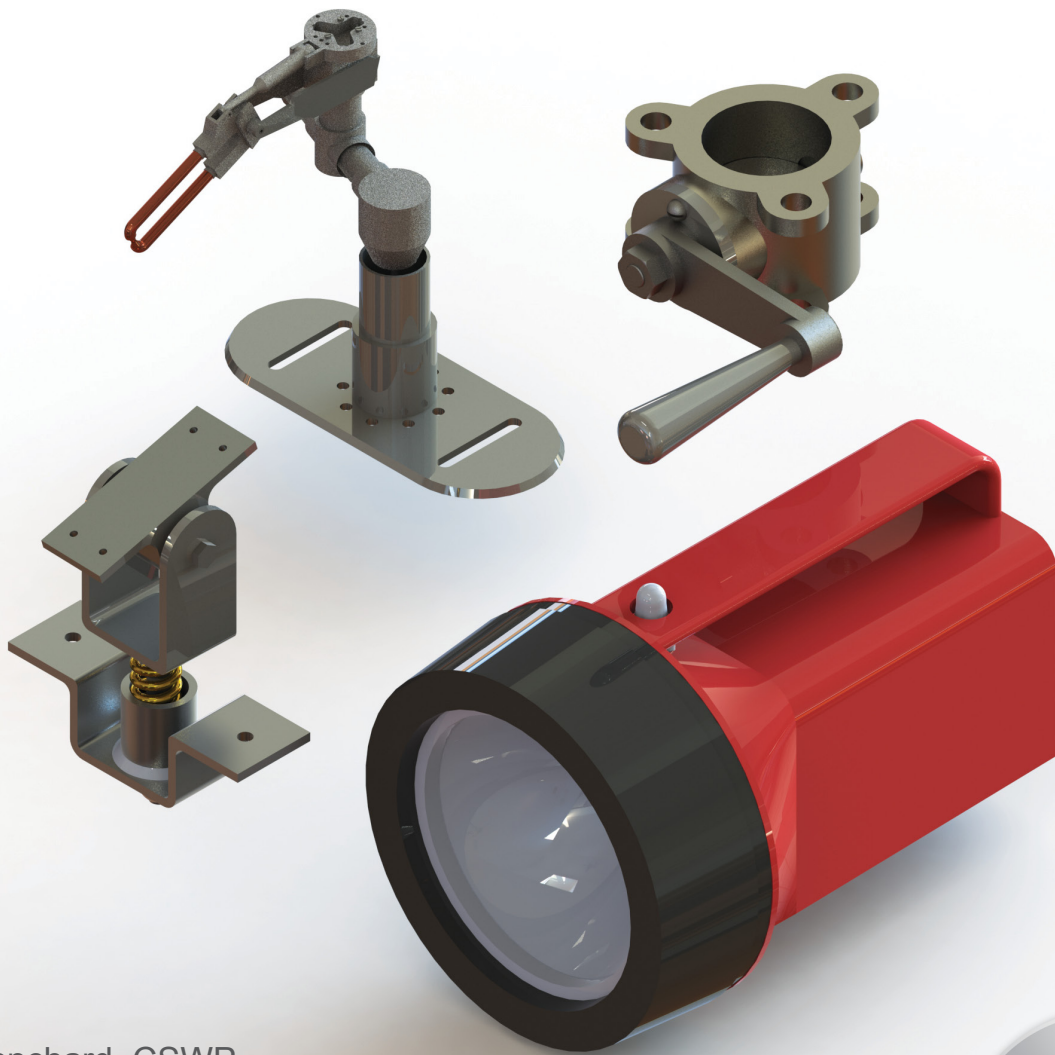


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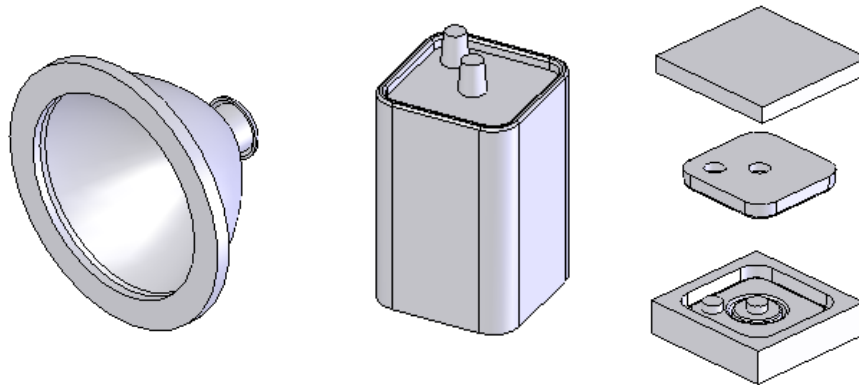
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Project 4

Extrude and Revolve Features



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

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

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

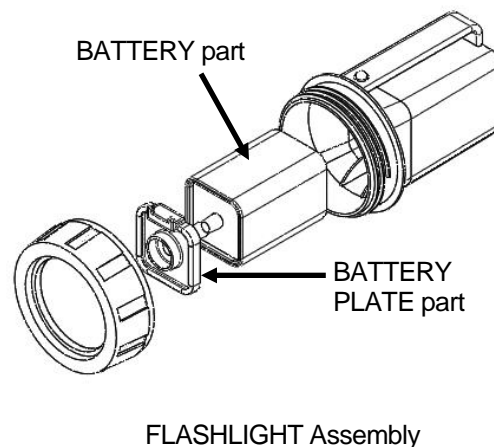
Start the design of the FLASHLIGHT assembly according to the customer's requirements. The FLASHLIGHT assembly will be cost effective, serviceable and flexible for future manufacturing revisions.

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

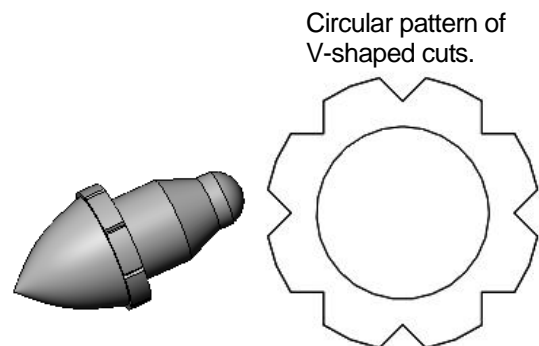
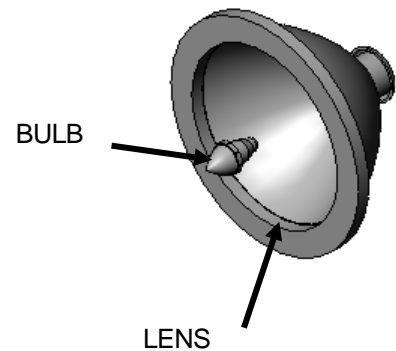
A Revolved feature requires a 2D sketch profile and a centerline. Utilize sketch geometry and sketch tools to create the following features:

- 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.

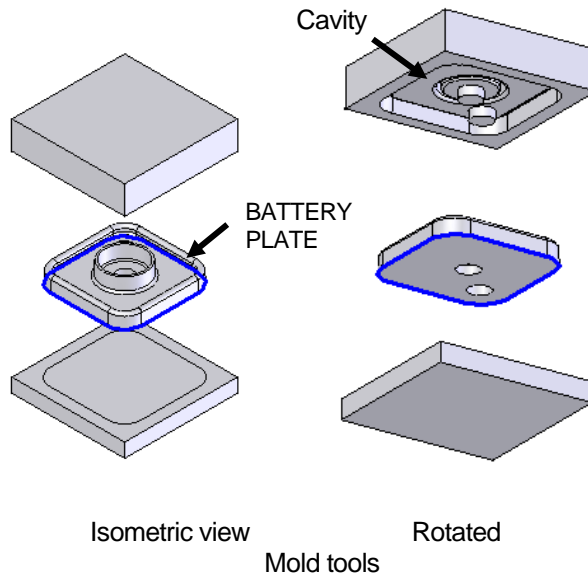


Utilize the Mold tools to create the Cavity tooling plates for the BATTERYPLATE part.

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

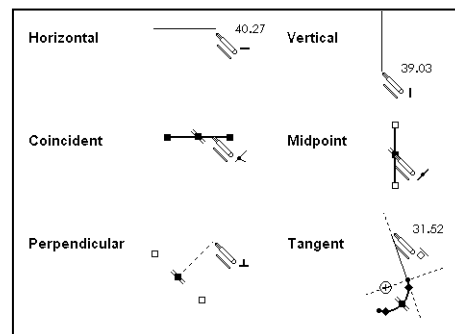


See Project 8 (Intelligent modeling techniques) for additional information.

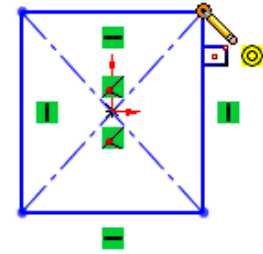
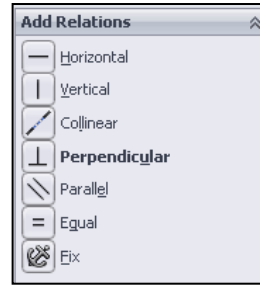
1. Design Intent in the Sketch

In SolidWorks, relations between sketch entities and model geometry, in either 2D or 3D sketches, are an important means of building in design intent. In this chapter - we will only address 2D sketches.

Apply design intent in a sketch as the profile is created. A profile is determined from the Sketch Entities. Example: Rectangle, Circle, Arc, Point, Slot etc.

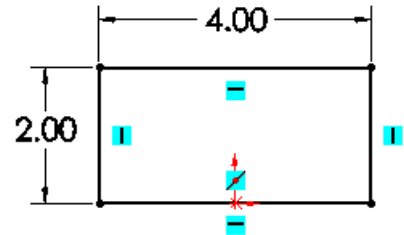


Develop design intent as you sketch with Geometric relations. Sketch relations are geometric constraints between sketch entities or between a sketch entity and a plane, axis, edge, or vertex. Relations can be added automatically or manually.

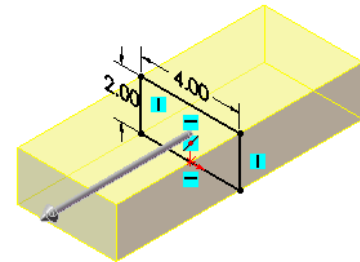


A rectangle contains Horizontal, Vertical, and Perpendicular automatic Geometric relations. Apply design intent using added Geometric relations. Example: Horizontal, Vertical, Collinear, Perpendicular, Parallel etc.

Example A: Apply design intent to create a square profile. Sketch a 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.



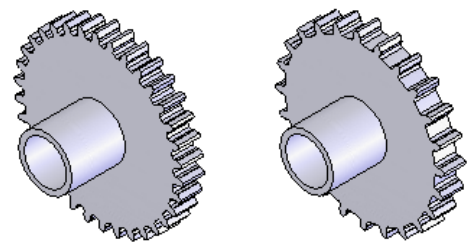
Example B: Develop a rectangular profile. Apply the Corner Rectangle tool. The bottom horizontal midpoint of the rectangular profile is located at the Origin. Add a Midpoint relation between the horizontal edge of the rectangle and the Origin. Insert two dimensions to define the width and height of the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

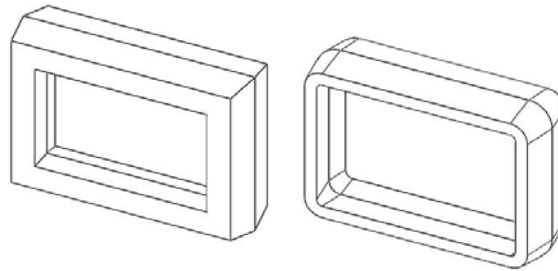


Example B: Do you create each tooth separate using the Extruded Cut feature? No. Create a single tooth and then apply the Circular Pattern feature. Create 34 teeth for a Circular Pattern feature. Modify the number of teeth from 32 to 24.

3. Design Intent in the Part

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

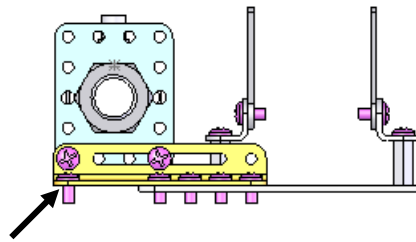
Example A: Feature order. Is the entire part symmetric? Feature order affects the part. Apply the Shell feature before the Fillet feature and the inside corners remain perpendicular.



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.



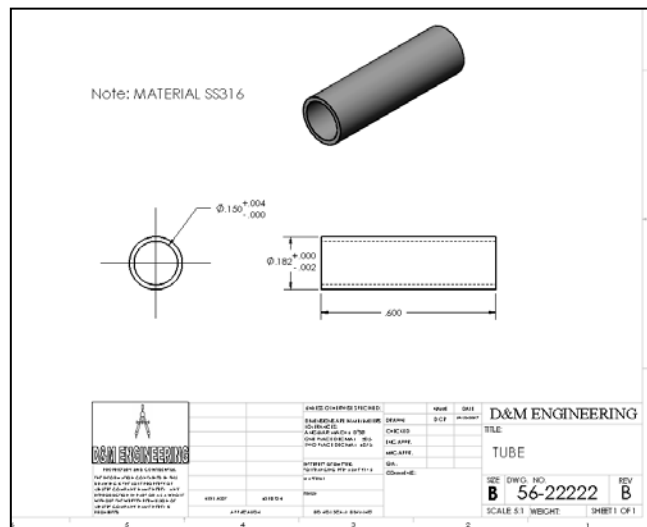
5. Design Intent in the Drawing

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 $+0.000/-0.002$ 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.



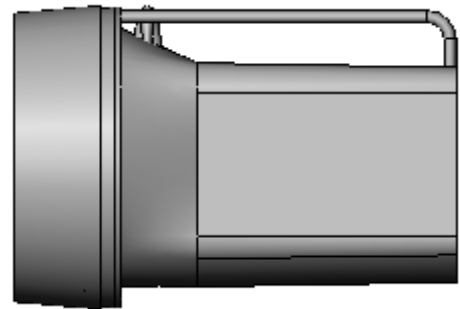
Project Situation

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.

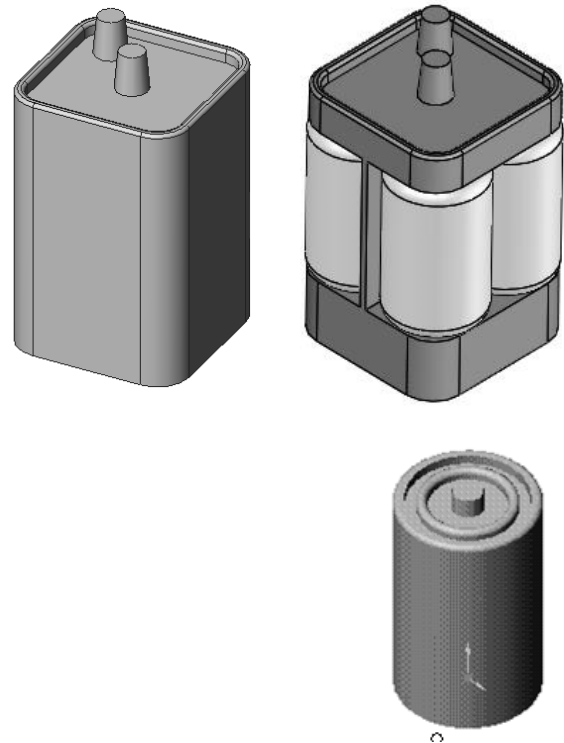


Your company's standard product line does not address the above key customer requirements. The customer made it clear that there is no room for negotiation on the key product requirements.

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.

Purchased Parts:	Designed Parts:
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:

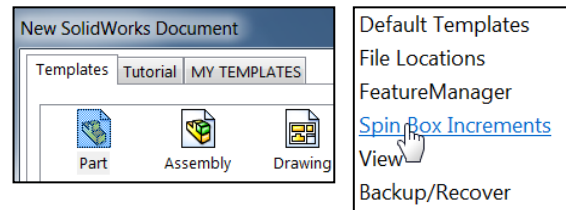
- PART-IN-ANSI
- PART-MM-ISO

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

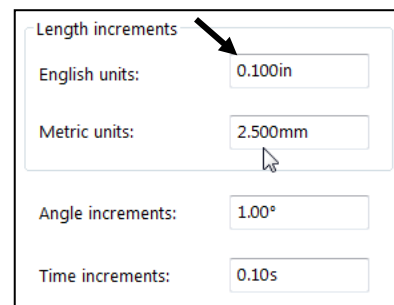
Create a PART-IN-ANSI Template.

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



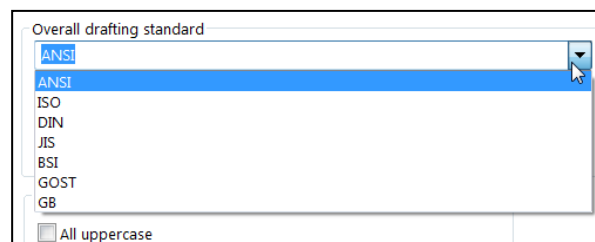
Set the Dimensioning Standard to ANSI.

- 3) Click **Options** .
- 4) Click the **System Options** tab.
- 5) Click **Spin Box Increments**. View the default settings.
- 6) Click **inside** the English units box.
- 7) Enter **.100in**.
- 8) Click **inside** the Metric units box.
- 9) Enter **2.500mm**.



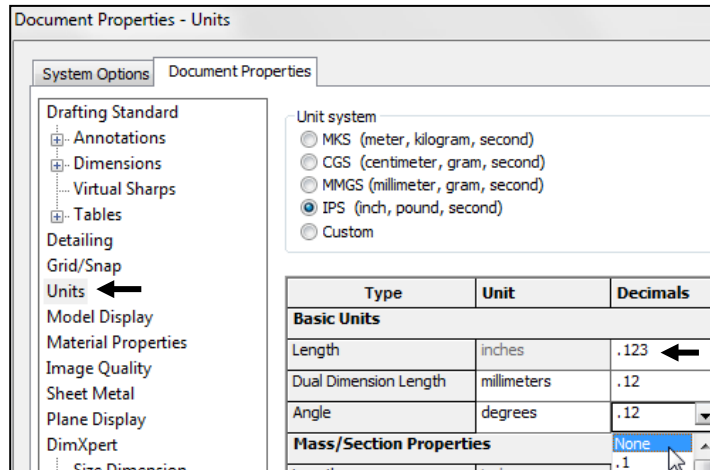
Set Document Properties.

- 10) Click the **Document Properties** tab.
- 11) Select **ANSI** from the Overall drafting standard drop-down menu.



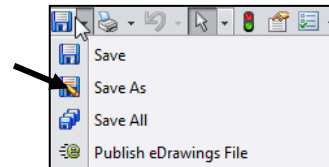
Set the part units for inch.

- 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 and name the part template.

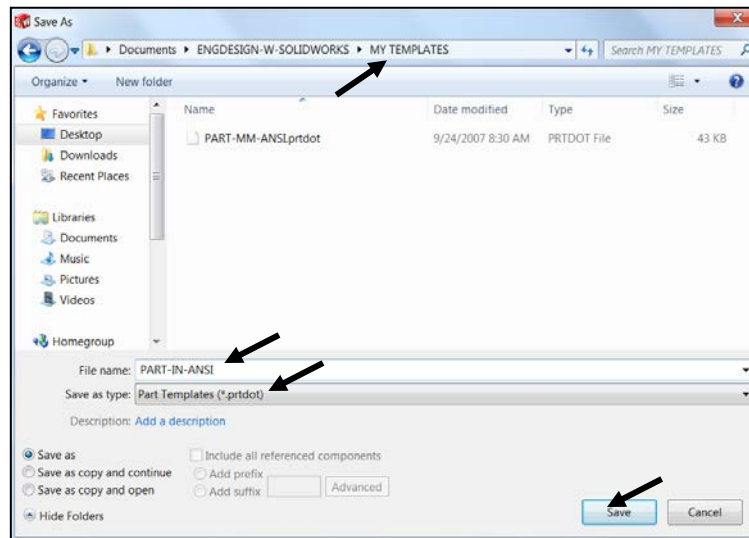
- 19) Click **Save As** from the drop-down Menu bar.
- 20) Select **Part Templates (*.prtdot)** from the Save as type box.
- 21) Select **ENGDESIGN-W-SOLIDWORKSMY-TEMPLATES** for the Save in folder.



- 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.

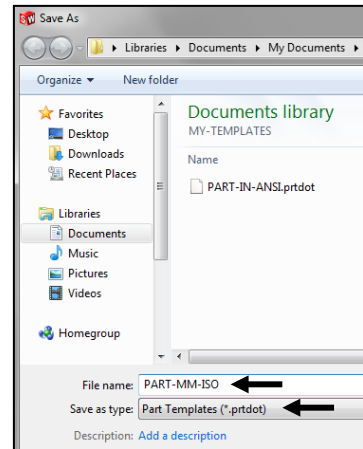


Set the part units for millimeter.

- 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.

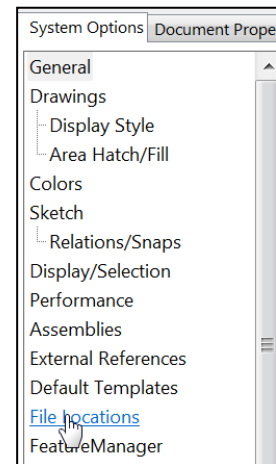
Save and name the part template.

- 31) Click **Save As** from the drop-down Menu bar.
- 32) Select **Part Templates (*.prtdot)** from the Save as type box.
- 33) Select **ENGDESIGN-W-SOLIDWORKSMY-TEMPLATES** for the Save in folder.
- 34) Enter **PART-MM-ISO** for File name.
- 35) Click **Save**.



If needed, set the System Options for File Locations to display in the New dialog box.

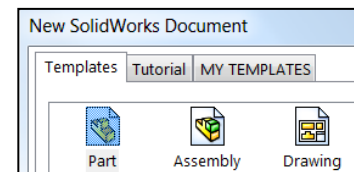
- 36) Click **Options** from the Menu bar.
- 37) Click **File Locations** from the System Options tab.
- 38) Select **Document Templates** from Show folders for.
- 39) Click the **Add** button.
- 40) Select the **MY-TEMPLATES** folder.
- 41) Click **OK** from the Browse for Folder dialog box.
- 42) Click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



Close all documents.

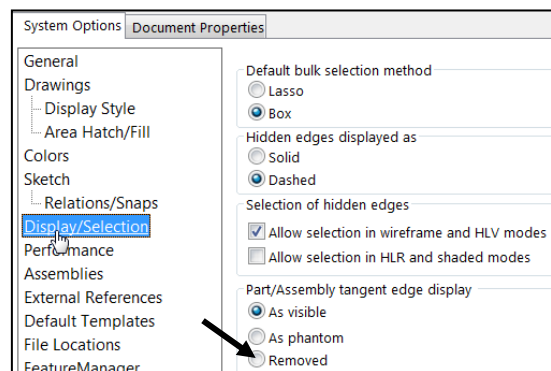
- 43) Click **Windows, Close All** from the Menu bar.

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.

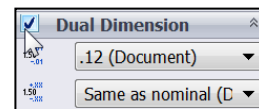
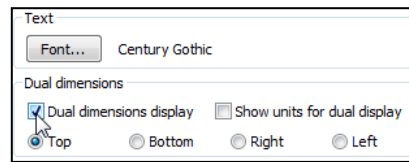
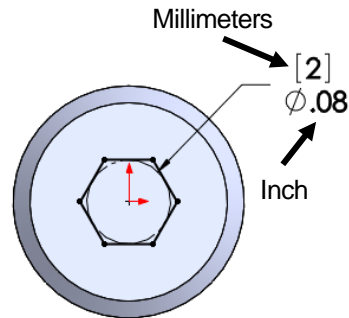
The primary units in this Project are IPS, (inch, pound, seconds).


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.

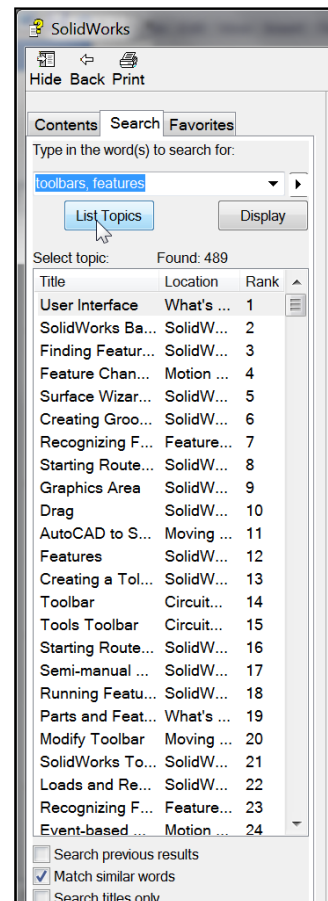
To set dual dimensions for an active document, check the Dual Dimension box in the Dimension PropertyManager.

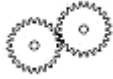


 Enter toolbars, features in SolidWorks Help Search category to review the function of each Features toolbar.

 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).

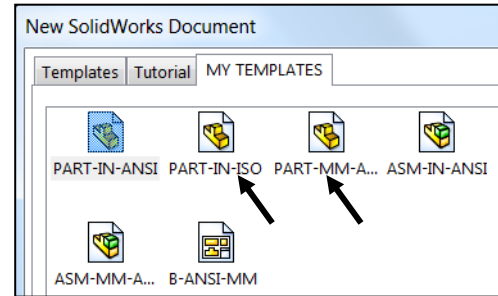
 **Model about the Origin:** This provides a point of reference for your dimensions to fully define the sketch.





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.

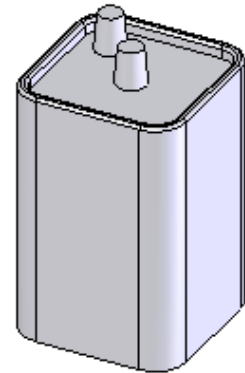
Note: In some network locations and school environments, the File Locations option must be set to MY-TEMPLATES for each session of SolidWorks.

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.

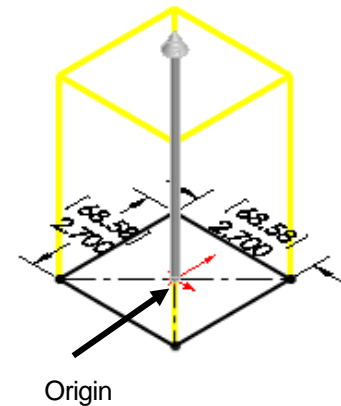


Create the BATTERY part. Use features to create parts. Features are building blocks that add or remove material.


Utilize the Instant3D tool to create the Extruded Boss/Base feature vs. using the Boss-Extrude PropertyManager.

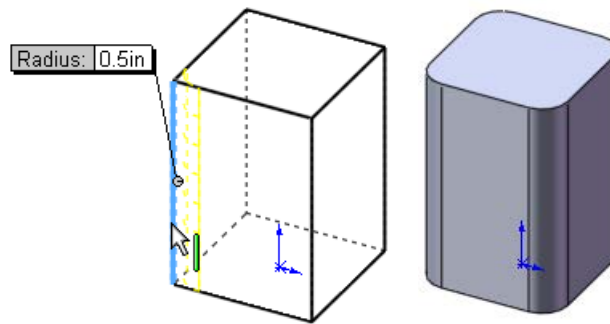
The Extrude Boss/Base features add material. The Base feature (Boss-Extrude1) is the first feature of the part. Note: The default End Condition for Instant3D is Blind.


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

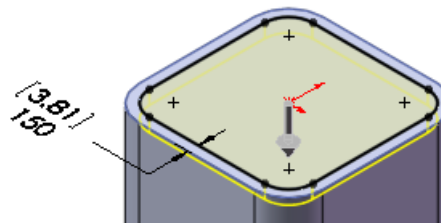


Extend the profile perpendicular (\perp) to the Top Plane.

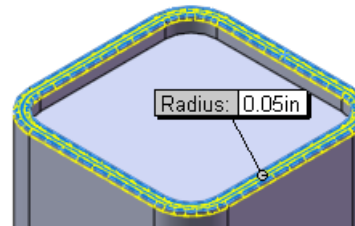
Utilize the Fillet  feature to round the four vertical edges.





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.

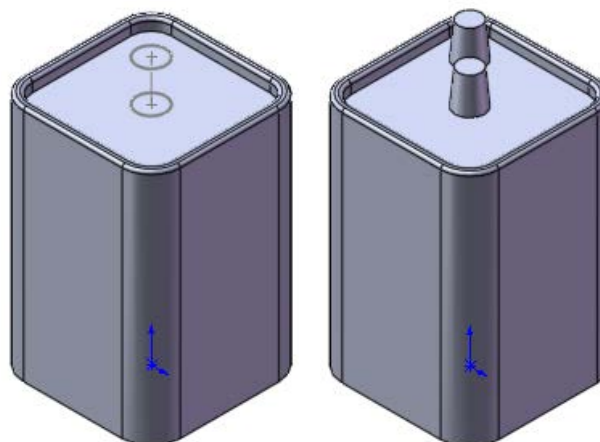


Utilize the Fillet feature  to round the top narrow face.



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

 Think design intent. When do you use the various End Conditions and Geometric sketch relations? What are you trying to do with the design? How does the component fit into the assembly? Design for change and flexibility.



BATTERY Part-Extruded Boss/Base Feature


The Extruded Boss/Base feature requires:

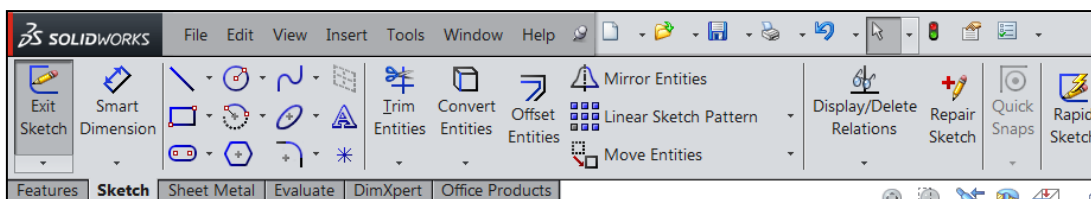
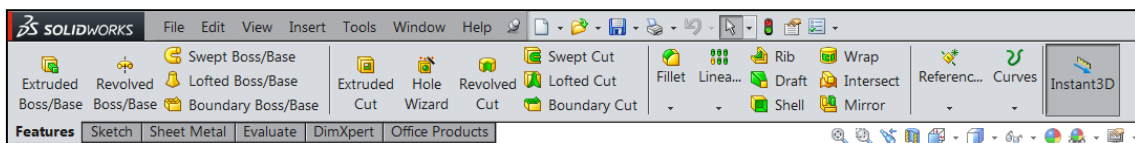
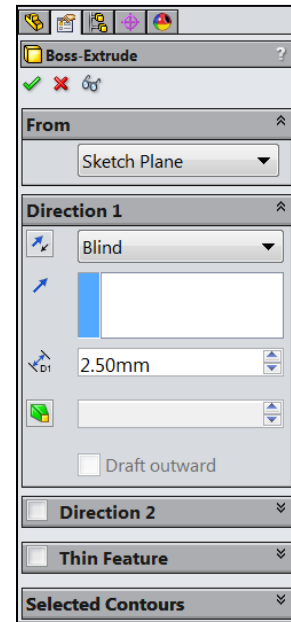
- Sketch plane (Top)
- Sketch profile (Rectangle)
 - Geometric relations and dimensions
- End Condition Depth (Blind) in Direction 1

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 and name 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.

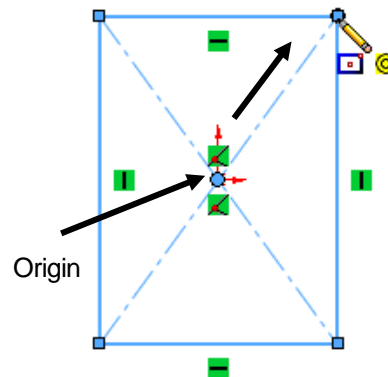
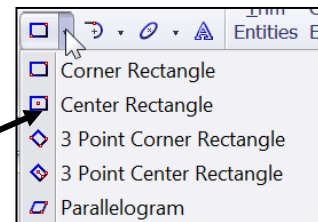
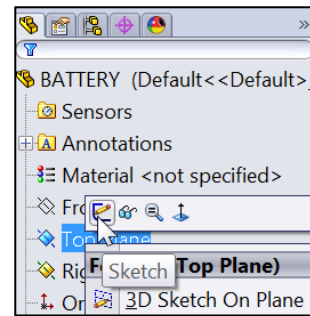
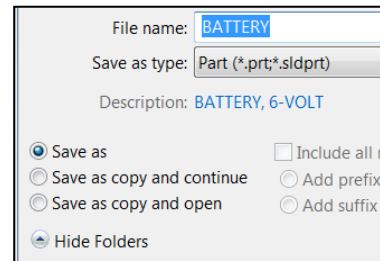
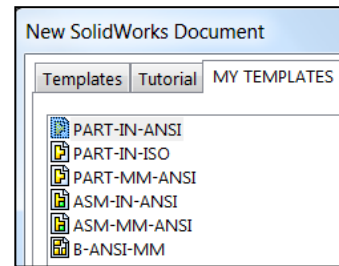
- 53) Click **Sketch**  from the Context toolbar. The Sketch toolbar is displayed.
- 54) Click the **Center Rectangle**  Sketch tool. The Center Rectangle  icon is displayed.
- 55) Click the **Origin**. This is your first point.
- 56) Drag and click the **second point** in the upper right quadrant as 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 goal of this book is to expose the new SolidWorks user to various tools, techniques and procedures. The text may not always use the most direct tool or process.

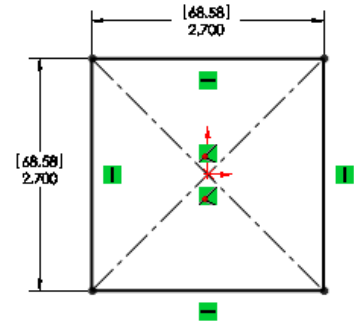


Click **View, Sketch Relations** from the Main menu to view sketch relations in the Graphics area.



Dimension the sketch.

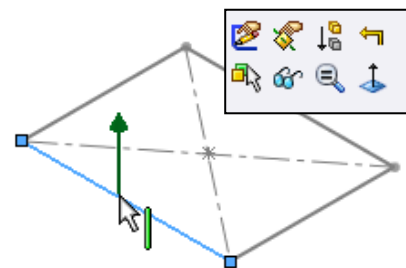
- 57) Click the **Smart Dimension**  Sketch tool.
- 58) Click the **top horizontal line**.
- 59) Click a **position** above the horizontal line.
- 60) Enter **2.700in**, [68.58] for width.
- 61) Click the **Green Check mark**  in the Modify dialog box.
- 62) Enter **2.700in**, [68.58] for height as illustrated.
- 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.

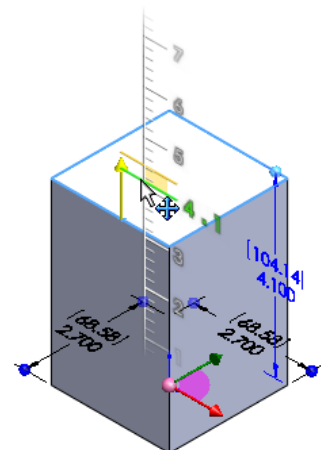
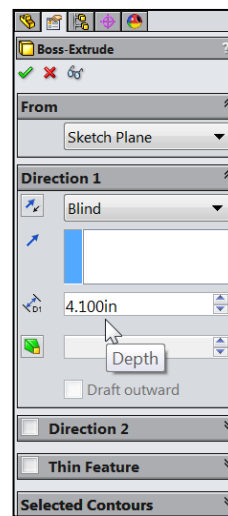


Display an Isometric view. Use the on-screen ruler.

- 66) Press the **space bar** to display the Orientation dialog box. Click **Isometric view** .
- 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**.

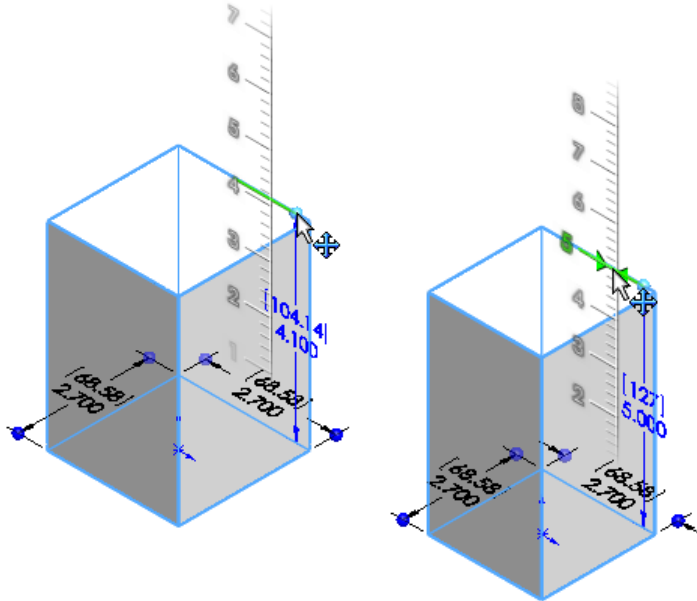
Save the BATTERY.

75) Click **Save** .


Modify the BATTERY.

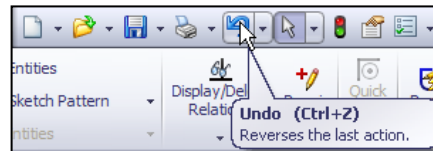
76) Click **Base Extrude** from the FeatureManager. Note: Instant3D is activated by default.

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.

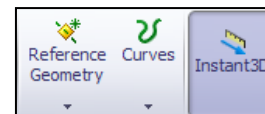


The color of the sketch indicates the sketch status.


- Light Blue - Currently selected.
- Blue - Under defined, requires additional geometric relations and or dimensions.
- Black - Fully defined.
- Red - Over defined, requires geometric relations and 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

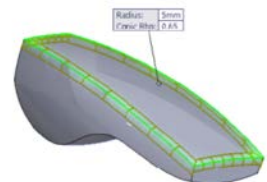
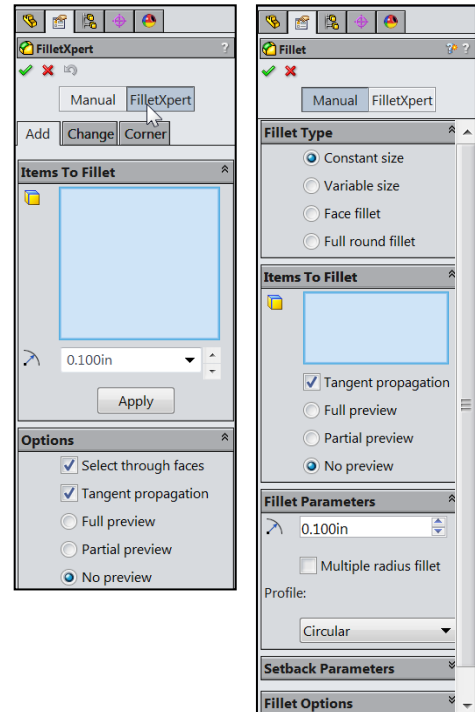
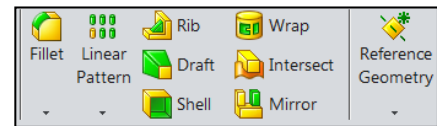
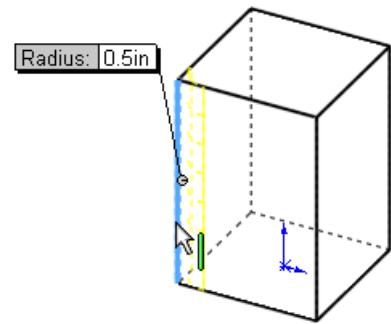
Select a vertical edge. Select the Fillet feature from the Features toolbar. Enter the Fillet radius. Add the other vertical edges to the Items To Fillet option.

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.

 Use the Fillet tool to create symmetrical conic shaped fillets for parts, assemblies, and surfaces. You can apply conic shapes to *Constant Size*, *Variable Size*, and *Face* fillets.






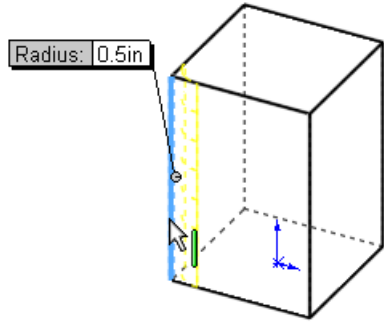
Activity: BATTERY Part-Fillet Feature Edge

Display hidden edges.



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

Insert a Fillet feature.

- 80) Click the **Fillet**  feature tool. The Fillet PropertyManager is displayed.
- 81) Click the **Manual** tab. Constant radius is the default Fillet Type.
- 82) Click the **left front vertical edge** as illustrated. Note the mouse pointer edge  icon. Edge<1> is displayed in the Items To Fillet box. The fillet option pop-up toolbar is displayed. Options are model dependent.
- 83) Select the **Connected to start face, 3 Edges icon**. The four selected edges are displayed in the Edges, Faces, Features, and Loop 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.



Display an Isometric, Shaded with Edges view.

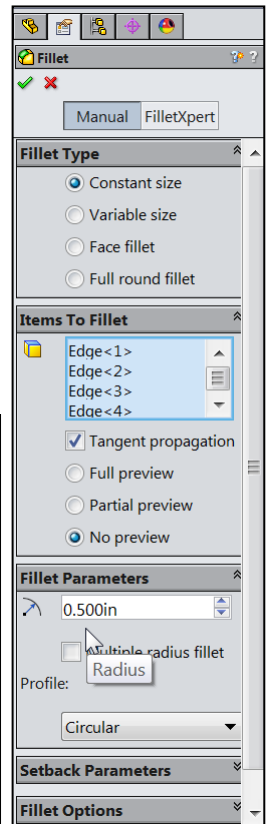
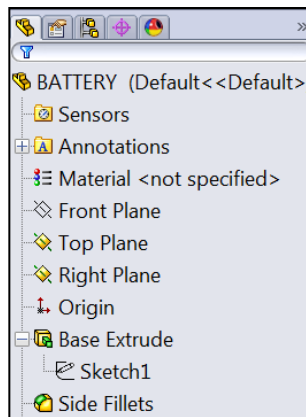
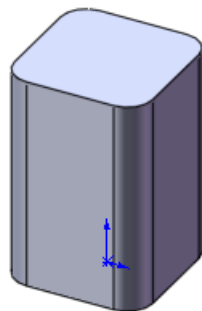
- 86) Click **Isometric view** .
- 87) Click **Shaded With Edges**  from the Heads-up View toolbar.

Rename the feature.

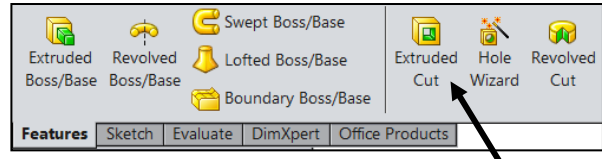
- 88) Rename **Fillet1** to **Side Fillets** in the FeatureManager.

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.

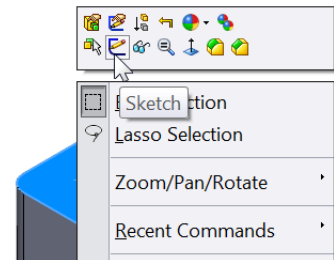
Activity: BATTERY Part-Extruded Cut Feature

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 Top face.

- 92) Press the **space bar** to display the Orientation dialog box.

- 93) Click **Top view** .

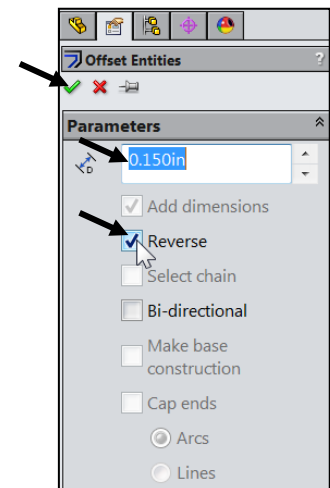
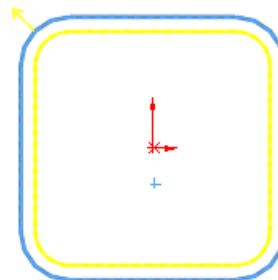
Offset the existing geometry from the boundary of the Sketch plane.

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

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

- 96) If needed check the **Reverse** box. The new Offset yellow profile displays inside the original profile.

- 97) 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 an Isometric view, with Hidden Lines Removed.

98) Press the **space bar** to display the Orientation dialog box.


99) Click **Isometric view** .

100) 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.

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

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

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

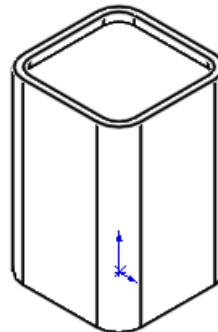
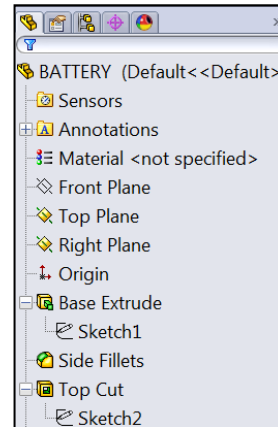
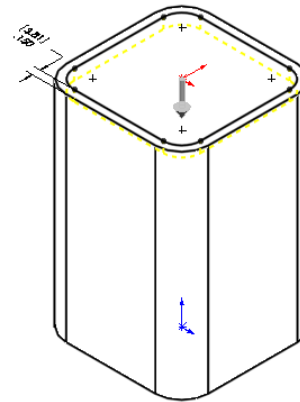
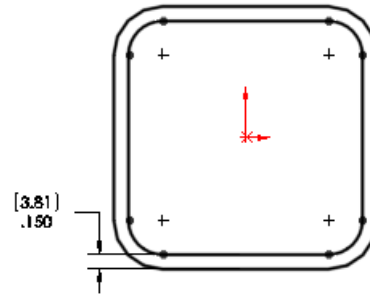
Rename the feature.

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

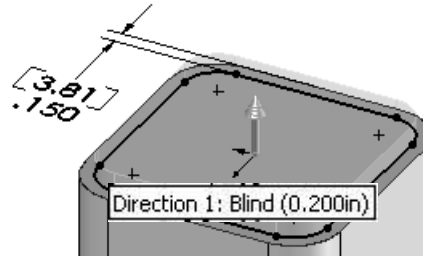
Save the BATTERY

105) Click **Save** .

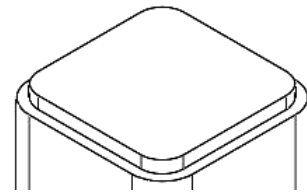
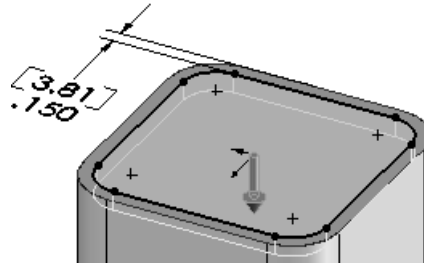
The Cut-Extrude PropertyManager contains numerous options. The Reverse Direction option determines the direction of the Extrude. The Extruded Cut feature is valid only when the direction arrow points into material to be removed.



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

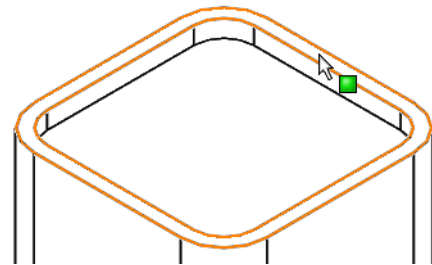
The Fillet  feature tool rounds sharp edges with a constant radius by selecting a face. A Fillet requires a:

- A selected face
- Fillet radius

Activity: BATTERY Part-Fillet Feature Face

Insert a Fillet feature on the top face.

- 106) Click the **Fillet**  feature tool. The Fillet PropertyManager is displayed.
- 107) Click the **top thin face** as illustrated. Note: The face  icon feedback symbol. Face<1> is displayed in the Items To Fillet box.
- 108) Click the **Manual** tab. Create a Constant radius for Fillet Type.
- 109) Enter **.050in**, [1.27] for Radius.
- 110) Click **OK**  from the Fillet PropertyManager. Fillet2 is displayed in the FeatureManager.



Rename the feature.

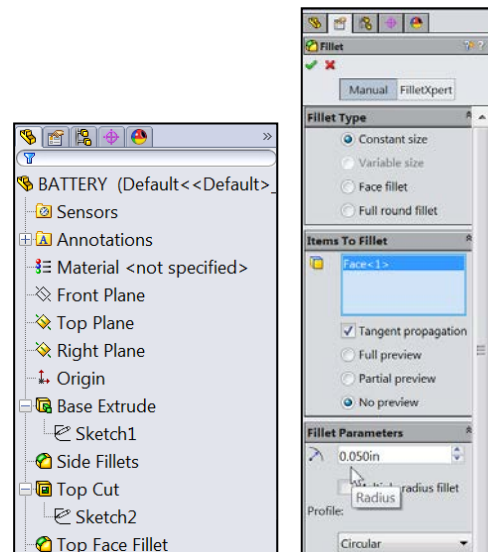
- 111) Rename **Fillet2** to **Top Face Fillet**.

Fit the model to the Graphics window.

- 112) Press the **f** key.

Save the BATTERY.

- 113) Click **Save** .



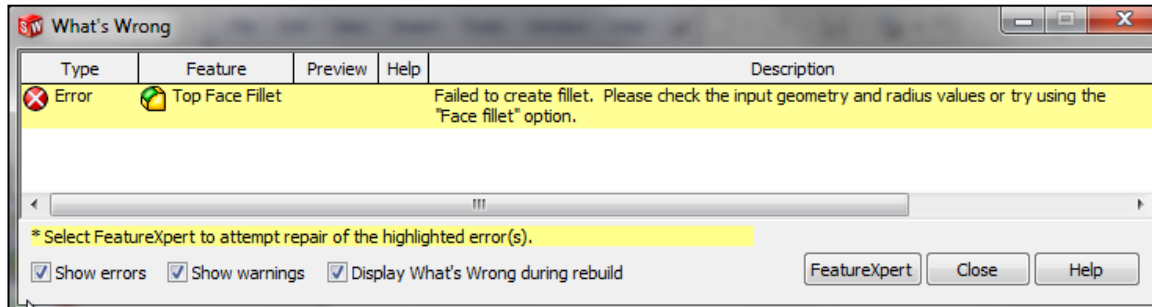
View the mouse pointer for feedback to select Edges or Faces for the fillet.



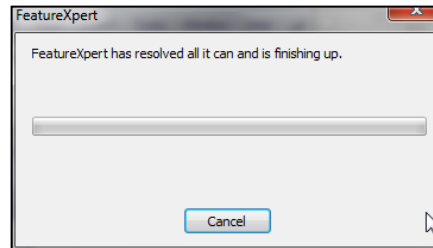
Do not select a fillet radius which is larger than the surrounding geometry.

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.



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 $.600\text{in}$.

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.

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

Create the sketch.

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

- 116) Click **Top view**.

Sketch the Close profile.

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

- 118) Click the **center point** of the circle coincident to the Origin

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

Add a dimension.

- 120) Click the **Smart Dimension** Sketch tool.

- 121) Click the **circumference** of the circle.

- 122) Click a **position** diagonally to the right.

- 123) Enter **.500in, [12.7]**.

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

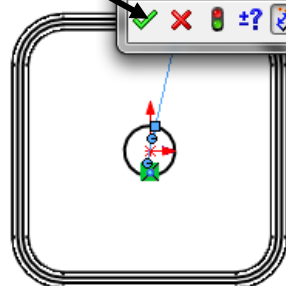
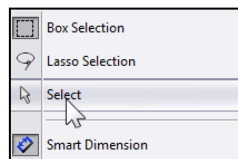
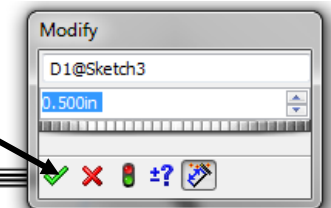
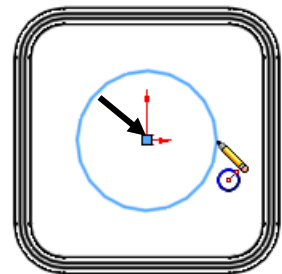
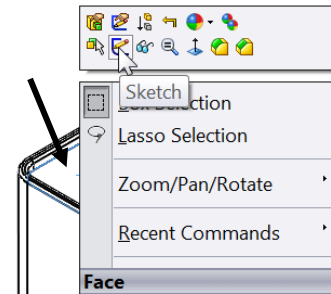
Copy the sketched circle.

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

- 126) Hold the **Ctrl** key down.

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

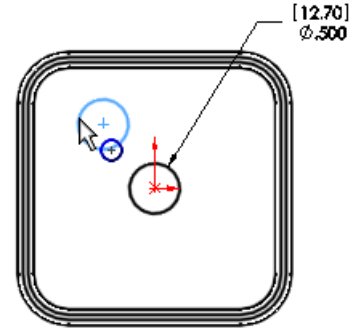
- 128) Release the **mouse button**.



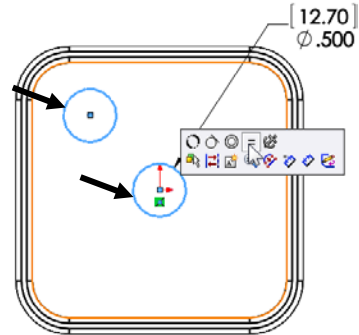
- 129) Release the **Ctrl** key. The second circle is selected and is displayed in blue. If needed click **OK** ✓ from the Circle PropertyManager.

Add an Equal relation.

- 130) Hold the **Ctrl** key down.
 131) Click the **circumference of the first circle**. The Properties PropertyManager is displayed. Both circles are selected and are displayed in green.



- 132) Release the **Ctrl** key.
 133) Right-click **Make Equal** = from the Context toolbar.
 134) Click **OK** ✓ from the Properties PropertyManager. The second circle remains selected.

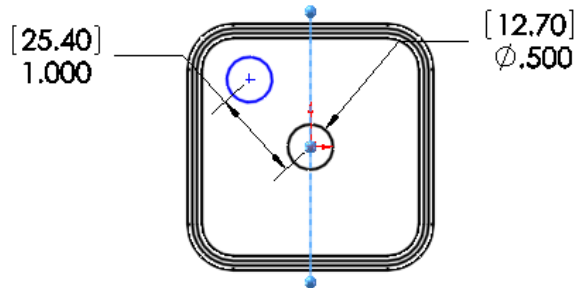


Show the Right Plane for the dimension reference.

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

Add an aligned dimension.

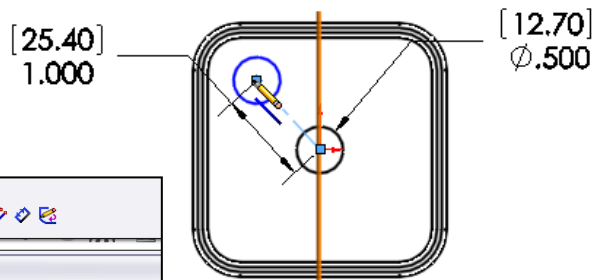
- 137) Click the **Smart Dimension** Sketch tool.
 138) Click the **two center points** of the two circles.
 139) Click a **position** off the profile in the upper left corner.
 140) Enter **1.000in, [25.4]** for the aligned dimension.



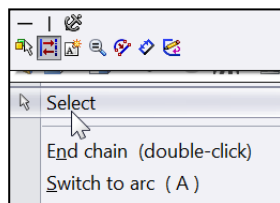
- 141) Click the **Green Check mark** ✓ in the Modify dialog box.

Insert a centerline.


- 142) Click the **Centerline** Sketch tool. The Insert Line PropertyManager is displayed.
 143) Sketch a centerline between the **two circle center points** as illustrated.





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

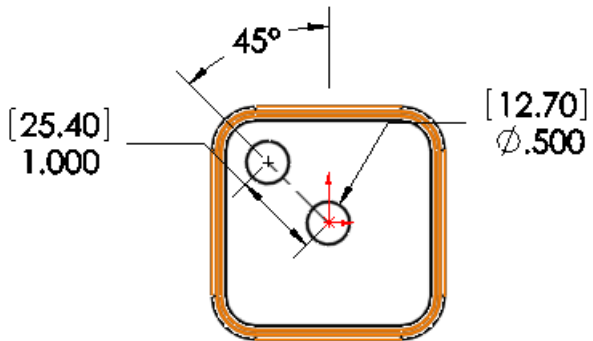
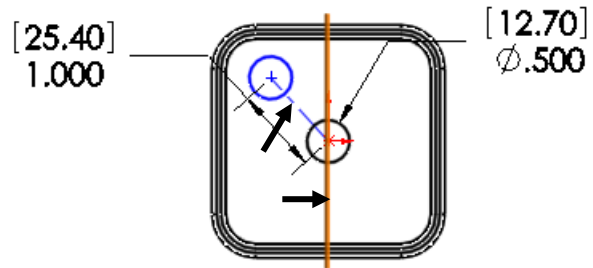


Double-click to end the centerline.

 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.


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




Fit the model to the Graphics window.




- 150) Press the **f** key.

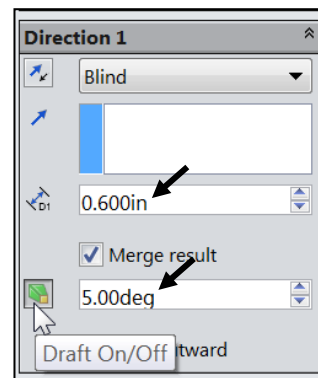
Hide the Right Plane. Save the model.

- 151) Right-click **Right Plane** in the FeatureManager.
- 152) 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.

Display an Isometric view. Insert an Extruded Boss feature.

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

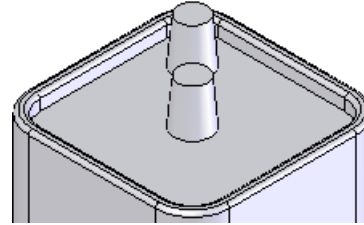


Rename the feature and sketch.

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

160) **Expand** Terminals.

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



Display Shaded With Edges. Save the model.

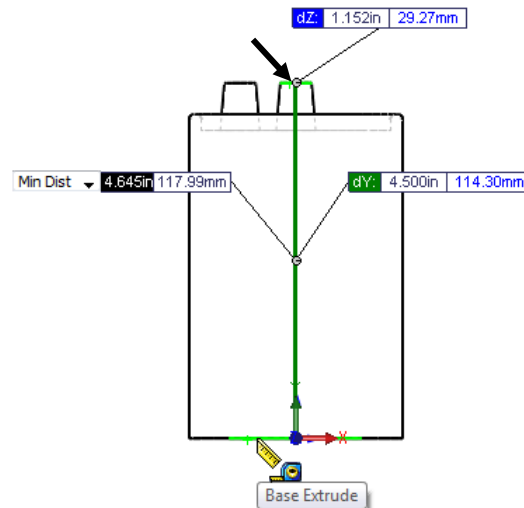
162) Click **Shaded With Edges**  from the Heads-up View toolbar.

163) 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.



Utilize the Measure tool to measure distances and add reference dimensions between the COM point and entities such as vertices, edges, and faces.



Measure the overall BATTERY height.


164) Click **Front view** .

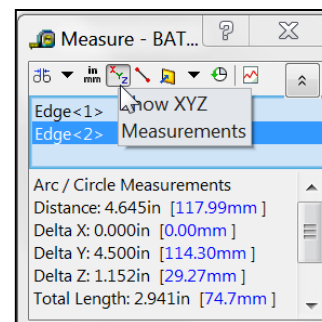
165) Click the **Measure**  tool from the Evaluate tab in the CommandManager. The Measure - BATTERY dialog box is displayed.

166) Click the **Show XYZ Measurements** option. This should be the *only active* option.

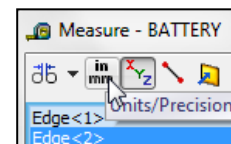
167) Click the **top edge** of the battery terminal as illustrated.





168) 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.

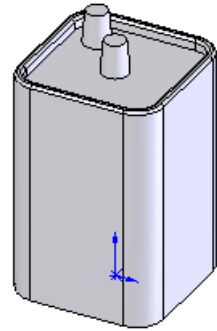
169) **Close**  the Measure - BATTERY dialog box.



The Measure tool provides the ability to display custom settings. Click **Units/Precision** from the Measure dialog box. View your options. 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 a Trimetric view.

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

Save the BATTERY.

171) 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).



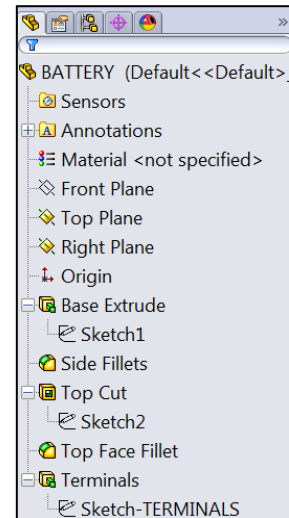
Review of the BATTERY Part

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.



Injection Molded Process

Lee Plastics of Sterling, MA is a precision injection molding company. Through the World Wide Web (www.leeplastics.com), review the injection molded manufacturing process.

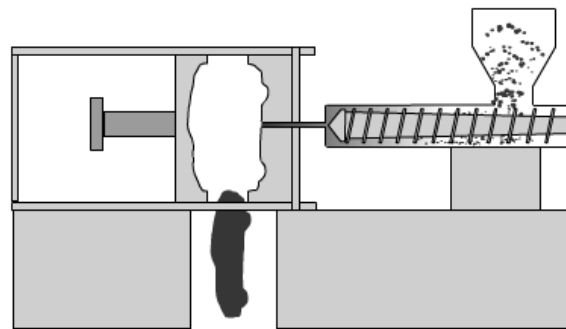
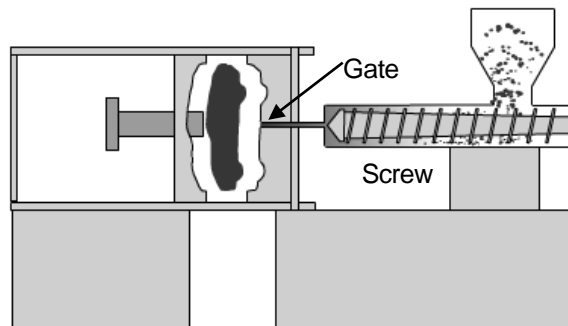
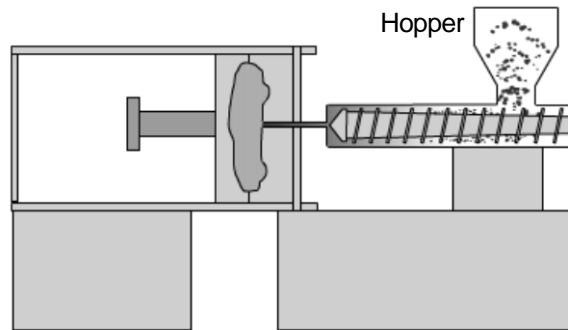
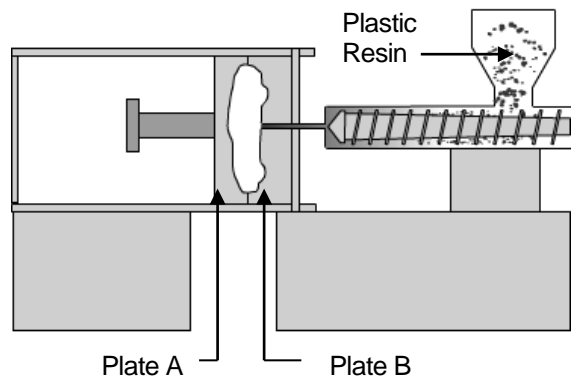
The injection molding process is as follows:

An operator pours the plastic resin in the form of small dry pellets, into a hopper. The hopper feeds a large auger screw. The screw pushes the pellets forward into a heated chamber. The resin melts and accumulates into the front of the screw.

At high pressure, the screw pushes the molten plastic through a nozzle, to the gate and into a closed mold, (Plates A & B). Plates A and B are the machined plates that you will design in this project.

The plastic fills the part cavities through a narrow channel called a gate.

The plastic cools and forms a solid in the mold cavity. The mold opens, (along the parting line) and an ejection pin pushes the plastic part out of the mold into a slide.



Injection Molded Process
(Courtesy of Lee Plastics, Inc.)

BATTERYPLATE Part


The BATTERYPLATE is a critical plastic part. The BATTERYPLATE:

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

Design the BATTERYPLATE. Utilize features from the BATTERY to develop the BATTERYPLATE. The BATTERYPLATE is manufactured as an injection molded plastic part. Build Draft into the Extruded Boss/Base features.

Edit the BATTERY features. Create two holes from the original sketched circles. Apply the Instant3D tool to create an Extruded Cut feature.

Modify the dimensions of the Base feature. Add a 3° draft angle.


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

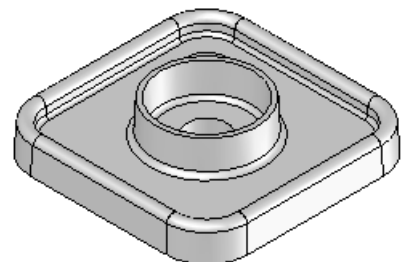
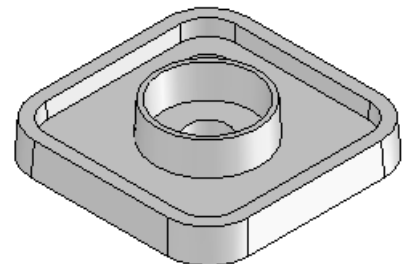
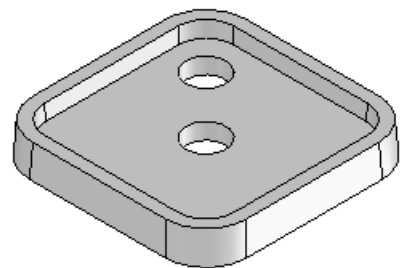
Insert an Extruded Boss/Base feature. Offset the center circular sketch.

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

Insert a Face Fillet and a Multi-radius Edge Fillet to remove sharp edges. Plastic parts require smooth edges. Group Fillet features together into a folder.

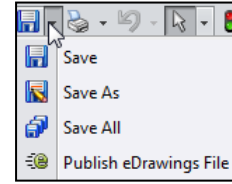
Perform a Draft Analysis on the part and create the Core and Cavity mold tooling.

 Group fillets together into a folder to locate them quickly. Features listed in the FeatureManager must be continuous in order to be placed as a group into a folder.

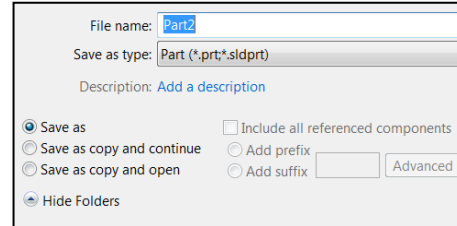


Save As, Delete, Edit Feature and Modify

Create the BATTERYPLATE part from the BATTERY part. Utilize the Save As tool from the Menu bar to copy the BATTERY part to the BATTERYPLATE part.



- Utilize the Save As / Save as command to save the file in another file format.
- Utilize the Save as copy and continue command to save the document to a new file name without replacing the active document.
- Utilize the Save as copy and open command to save the document to a new file name that becomes the active document. The original document remains open. References to the original document are not automatically assigned to the copy.

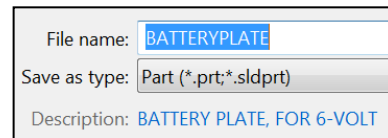
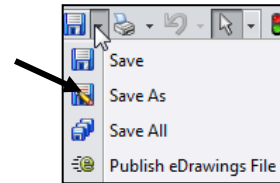


Reuse existing geometry. Create two holes. Delete the Terminals feature and reuse the circle sketch. Select the sketch in the FeatureManager. Create an Extruded Cut feature from the Sketch-TERMINALS using the Instant3D tool. Blind is the default End Condition. Edit the Bass-Extrude feature. Modify the overall depth. Rebuild the model.

Activity: BATTERYPLATE Part-Save As, Delete, Modify and Edit Feature

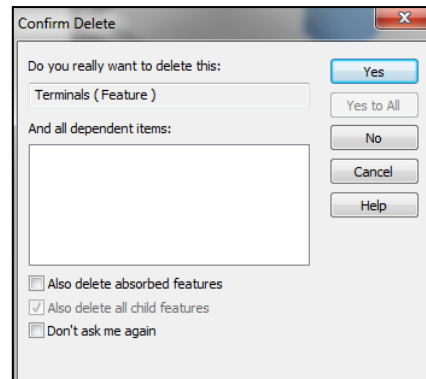
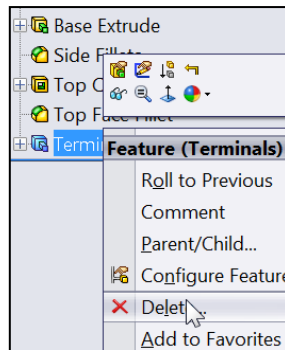
Apply the Save As tool. Create and save a new part.

- 172) Click **Save As** from the drop-down Menu bar.
- 173) Select **PROJECTS** for Save In folder.
- 174) Enter **BATTERYPLATE** for File name.
- 175) Enter **BATTERY PLATE, FOR 6-VOLT** for Description.
- 176) Click **Save** from the Save As dialog box. The BATTERYPLATE FeatureManager is displayed. The BATTERY part is closed.



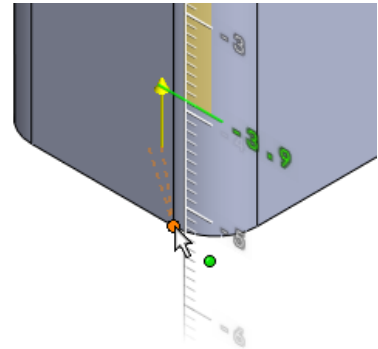
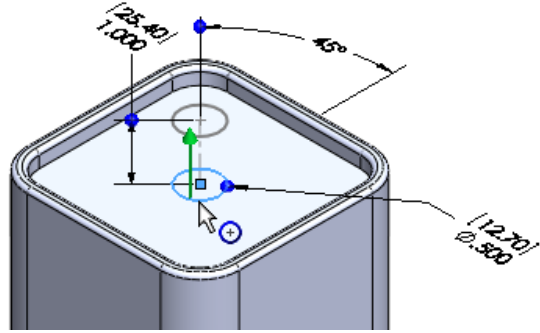
Delete the Terminals feature.

- 177) Right-click **Terminals** from the FeatureManager.
- 178) Click **Delete**.
- 179) Click **Yes** from the Confirm Delete dialog box. Do not delete the two-circle sketch, Sketch-TERMINALS.




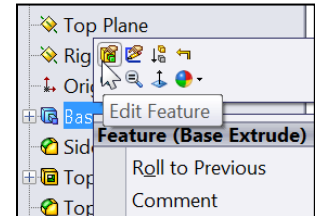
Create an Extruded Cut feature from the Sketch-TERMINALS using Instant3D.

- 180) Click **Sketch-TERMINALS** from the FeatureManager.
- 181) Click the **circumference** of the center circle as illustrated. A green arrow is display.
- 182) Hold the **Alt** key down. Drag the **green arrow** downward below the model to create a hole in Direction 1.
- 183) Release the mouse button on the **vertex** as illustrated. This ensures a Through All End Condition with model dimension changes.
- 184) Release the **Alt** key. Boss-Extrude1 is displayed in the FeatureManager.
- 185) Rename the **Boss-Extrude1** feature to **Holes** in the FeatureManager.




Edit the Base Extrude feature.

- 186) Right-click **Base Extrude** from the FeatureManager.
- 187) Click **Edit Feature**  from the Context toolbar. The Base Extrude PropertyManager is displayed.



Modify the overall depth.

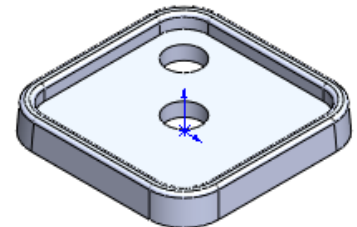
- 188) Enter **.400in, [10.16]** for Depth in Direction 1.
- 189) Click the **Draft ON/OFF** button.
- 190) Enter **3.00deg** in the Angle box.
- 191) Click **OK**  from the Base Extrude PropertyManager.


Fit the model to the Graphics window.


- 192) Press the **f** key.

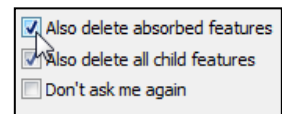
Save the BATTERYPLATE.

- 193) Click **Save** .




 Modify the **Spin Box Increments** in System Options to display different increments for the Instant3D on-screen ruler.

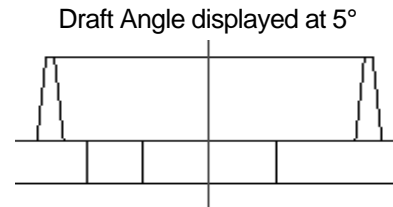
 To delete both the feature and the sketch at the same time, select the Also delete absorbed features check box from the Confirm Delete dialog box.



BATTERYPLATE Part-Extruded Boss Feature

The Holder is created with a circular Extruded Boss/Base feature. Utilize the Offset Entities  Sketch tool to create the second circle. Apply a draft angle of 3° in the Extruded Boss feature.

When applying the draft angle to the two concentric circles, the outside face tapers inwards and the inside face tapers outwards.

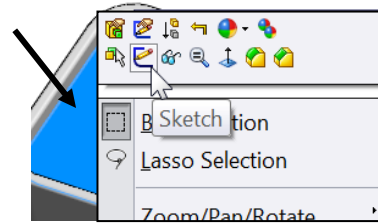


Plastic parts require a draft angle. Rule of thumb; 1° to 5° is the draft angle. The draft angle is created in the direction of pull from the mold. This is defined by geometry, material selection, mold production and cosmetics. Always verify the draft with the mold designer and manufacturer.

Activity BATTERYPLATE Part-Extruded Boss Feature

Select the Sketch plane.

194) Right-click the **top face** of Top Cut. This is your Sketch plane.



Create the sketch.

195) Click **Sketch**  from the Context toolbar.

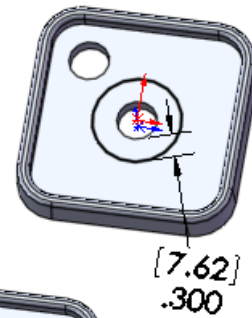
196) Click the **top circular edge** of the center hole. Note: Use the keyboard arrow keys or the middle mouse button to rotate the sketch if needed.

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

198) Enter **.300in, [7.62]** for Offset Distance. Accept the default settings.


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

200) Drag the **dimension** off the model.

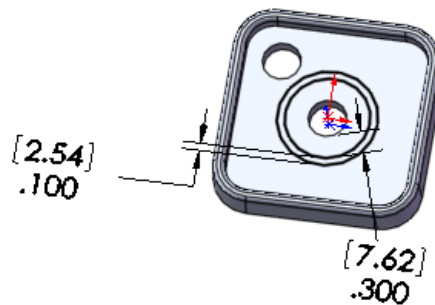


Create the second offset circle.

201) Click the **offset circle** in the Graphics window.

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

203) Enter **.100in, [2.54]** for Offset Distance.



204) Click **OK** ✓ from the Offset Entities PropertyManager. Drag the dimension off the model. Two offset concentric circles define the sketch.

Insert an Extruded Boss/Base feature.

205) Click the **Extruded Boss/Base** feature tool. The Boss-Extrude PropertyManager is displayed.

206) Enter **.400in**, [10.16] for Depth in Direction 1.

207) Click the **Draft ON/OFF** button.

208) Enter **3deg** in the Angle box.

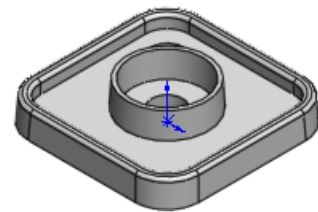
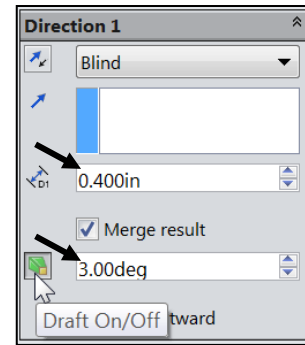
209) Click **OK** ✓ from the Boss-Extrude PropertyManager. The Boss-Extrude1 feature is displayed in the FeatureManager.

Rename the feature.

210) Rename the **Boss-Extrude1** feature to **Holder** in the FeatureManager.

Save the model.


211) Click **Save** .

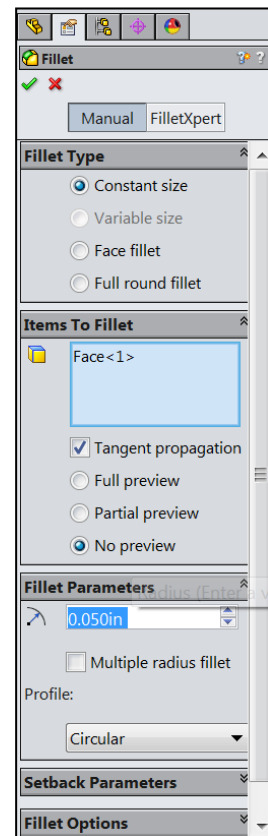


BATTERYPLATE Part-Fillet Features: Full Round and Multiple Radius Options

Use the **Fillet** feature tool to smooth rough edges in a model. Plastic parts require fillet features on sharp edges. Create two Fillets. Utilize different techniques. The current Top Face Fillet produced a flat face. Delete the Top Face Fillet. The first Fillet feature is a Full round fillet. Insert a Full round fillet feature on the top face for a smooth rounded transition.

The second Fillet feature is a Multiple radius fillet. Select a different radius value for each edge in the set. Select the inside and outside edge of the Holder. Select all inside tangent edges of the Top Cut. A Multiple radius fillet is utilized next as an exercise. There are machining instances where radius must be reduced or enlarged to accommodate tooling. Note: There are other ways to create Fillets.

 Group Fillet features into a Fillet folder. Placing Fillet features into a folder reduces the time spent for your mold designer or toolmaker to look for each Fillet feature in the FeatureManager.



Activity: BATTERYPLATE Part-Fillet Features: Full Round, Multiple Radius Options

Delete the Top Edge Fillet.

212) Right-click **Top Face Fillet** from the FeatureManager.

213) Click **Delete**.


214) Click **Yes** to confirm delete.

215) Drag the **Rollback** bar below Top Cut in the FeatureManager.



Create a Full round fillet feature.

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

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

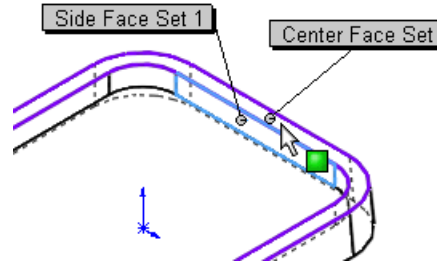
218) Click the **Manual** tab.

219) Click the **Full round fillet** box for Fillet Type.

220) Click the **inside Top Cut face** for Side Face Set 1 as illustrated.

221) Click **inside** the Center Face Set box.

222) Click the **top face** for Center Face Set as illustrated.




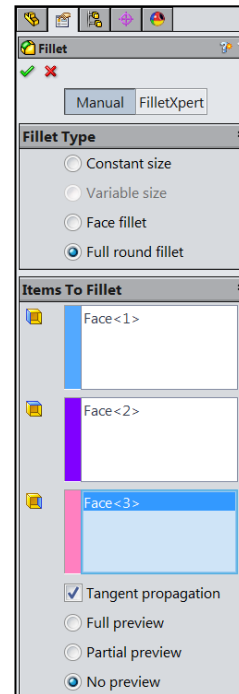
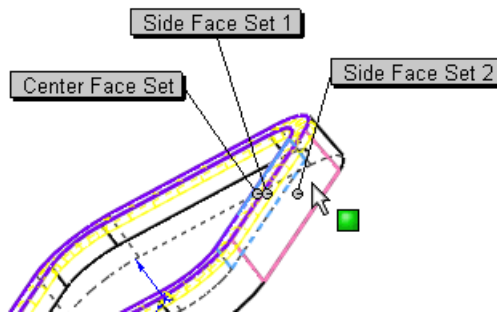
Rotate the part.

223) Press the **Left Arrow** key until you can select the outside Base Extrude face.

224) Click **inside** the Side Face Set 2 box.

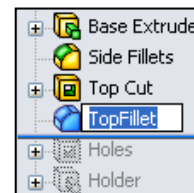
225) Click the **outside Base Extrude face** for Side Face Set 2 as illustrated. Accept the default settings.

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






Rename Fillet1.


227) Rename **Fillet1** to **TopFillet**.

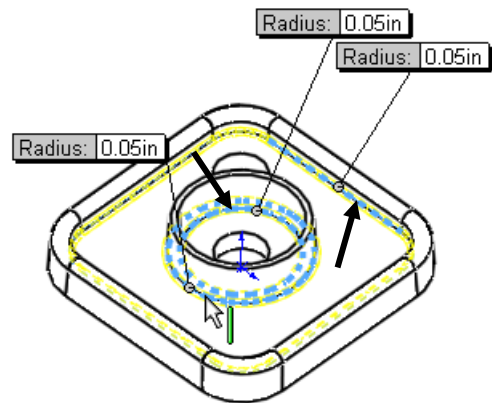


Display an Isometric view with Hidden Lines Removed. Save the BATTERYPLATE.


- 228) Click **Isometric view** .
- 229) Click **Hidden Lines Removed**  from the Heads-up View toolbar.
- 230) Drag the **Rollback bar** to the bottom of the FeatureManager.
- 231) Click **Save** .

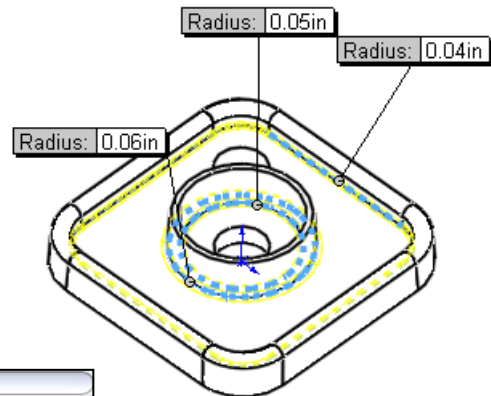
Create a Multiple radius fillet feature.

- 232) Click the **bottom outside circular edge** of the Holder as illustrated.
- 233) Click the **Fillet**  feature tool. The Fillet PropertyManager is displayed.
- 234) Click the **Constant radius** box.
- 235) Enter **.050in**, [1.27] for Radius.
- 236) Click the **bottom inside circular edge** of the Top Cut as illustrated.
- 237) Click the **inside edge** of the Top Cut.
- 238) Check the **Tangent propagation** box.
- 239) Check the **Multiple radius fillet** box.



Modify the Fillet values.


- 240) Click the **Radius** box for the Holder outside edge.
- 241) Enter **0.060in**, [1.52].
- 242) Click the **Radius** box for the Top Cut inside edge.
- 243) Enter **0.040in**, [1.02].
- 244) Click **OK**  from the Fillet PropertyManager. Fillet2 is displayed in the FeatureManager.

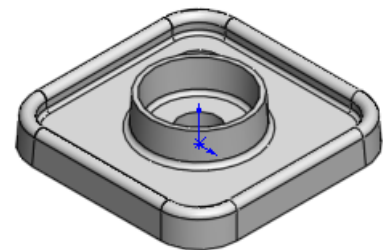
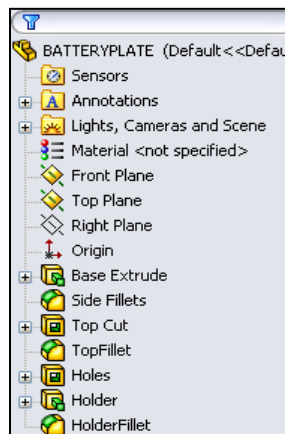


Rename the Fillet2 folder.

- 245) Rename **Fillet2** to **HolderFillet**.

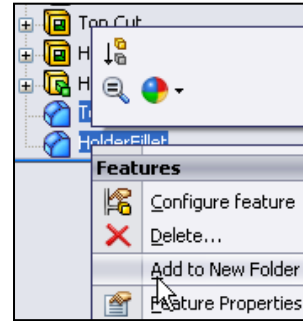
Display Shaded With Edges.

- 246) Click **Shaded With Edges**  from the Heads-up View toolbar. View the results in the Graphics window.



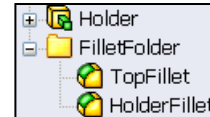
Group the Fillet features into a new folder.

- 247) Click **TopFillet** from the FeatureManager.
- 248) Drag the **TopFillet** feature directly above the HolderFillet feature in the FeatureManager.
- 249) Click **HolderFillet** in the FeatureManager.
- 250) Hold the **Ctrl** key down.
- 251) Click **TopFillet** in the FeatureManager.
- 252) Right-click **Add to New Folder**.
- 253) Release the **Ctrl** key.



Rename Folder1.

- 254) Rename **Folder1** to **FilletFolder**.



Save the BATTERYPLATE.

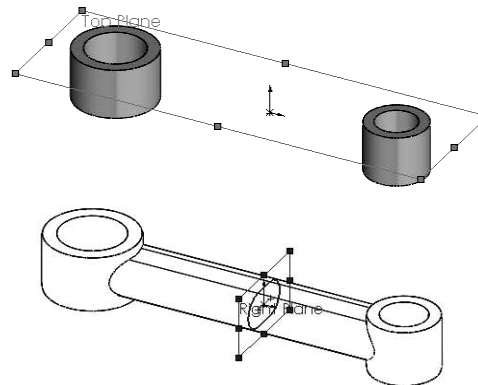
- 255) Click **Save** .

Multi-body Parts and the Extruded Boss/Base Feature


A Multi-body part has separate solid bodies within the same part document.

A WRENCH consists of two cylindrical bodies. Each extrusion is a separate body. The oval profile is sketched on the right plane and extruded with the Up to Body End Condition option.

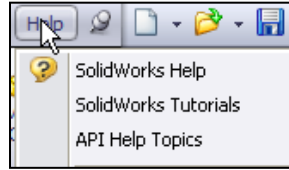
The BATTERY and BATTERYPLATE parts consisted of a solid body with one sketched profile. Each part is a single body part.



Multi-body part
Wrench

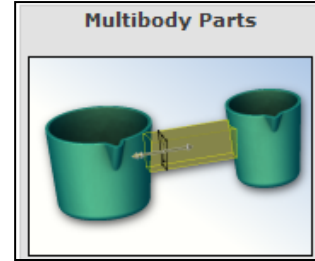
 Additional information on Save, Extruded Boss/Base, Extruded Cut, Fillets, Copy Sketched Geometry, and Multi-body are located in SolidWorks Help.

Refer to **Help, SolidWorks Tutorials, Advanced Techniques, Multibody Parts** for additional information.



Review of the BATTERYPLATE Part

The Save As option was utilized to copy the BATTERY part to the BATTERYPLATE part. You created a hole in the BATTERYPLATE using Instant3D and modified features using the PropertyManager.

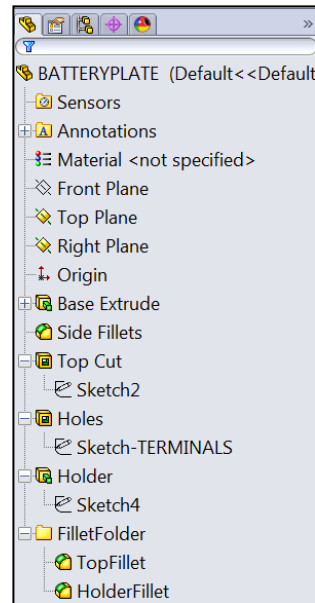


The BATTERYPLATE is a plastic part. The Draft Angle option was added in the Extruded Base (Boss-Extrude1) feature.

The Holder Extruded Boss utilized a circular sketch and the Draft Angle option. The Sketch Offset tool created the circular ring profile. Multi radius Edge Fillets and Face Fillets removed sharp edges.

Similar Fillet features were grouped together into a folder. Features were renamed in the FeatureManager.

The BATTERY and BATTERYPLATE utilized an Extruded Boss/Base feature.



It is considered best practice to fully define all sketches in the model. However; there are times when this is not practical. Generally when using the spline tool to create a complex freeform shape.



Sketch dimensions are displayed in black.



Feature dimensions are displayed in blue.