

INSIDE:

MultiMedia CD

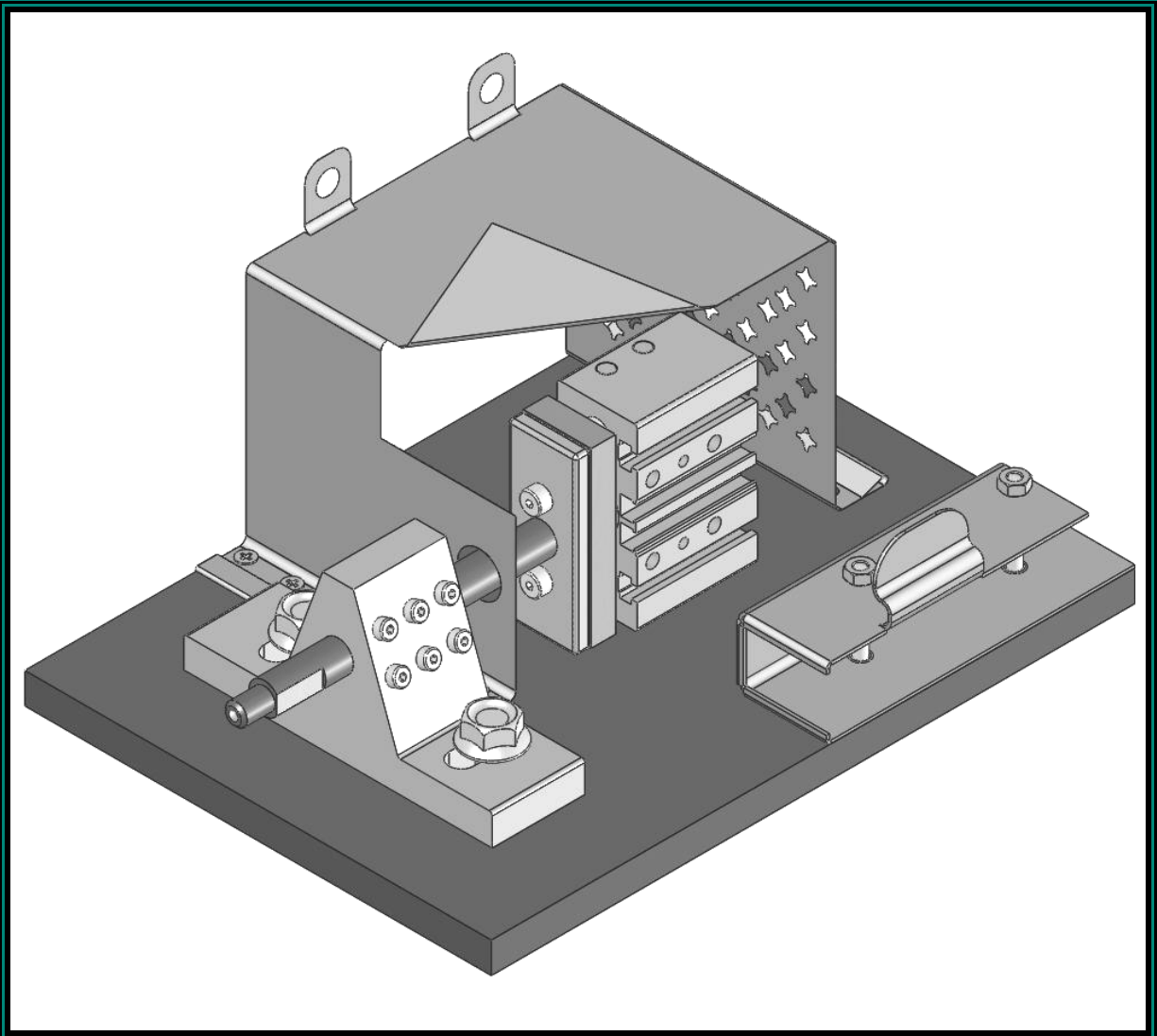
An audio/visual
presentation of the
tutorial projects

Engineering Design with SolidWorks 2007

and MultiMedia CD

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

David C. Planchard & Marie P. Planchard



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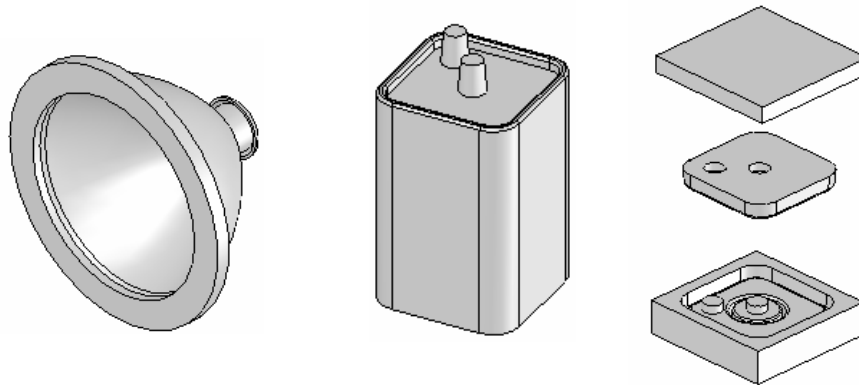
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3D
SolidWorks

Solution
Partner

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"> • An understanding of the 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"> • Ability to apply Document Properties and to create custom Part Templates.
<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 and understanding of the following Features: Extruded Boss/Base, Extruded Cut, Revolve 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 Analysis to create a simple Core and Cavity part.

Notes:



Project 4-Extrude and Revolve Features

Project Objective

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

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

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

Create the following parts:

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

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

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

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

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

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

- Use the following Mold tools:
 - Draft Analysis.
 - Scale.
 - Parting Lines.
 - Shut-off Surfaces.
 - Parting Surfaces.
 - Tooling Split.

Project Overview

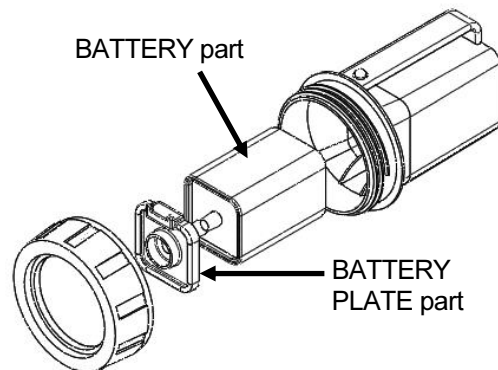
In Project 4, you started the design of the FLASHLIGHT assembly according to the customer's requirements. The FLASHLIGHT assembly will be cost effective, serviceable, and flexible for future manufacturing revisions.

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

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

Create two parts for the FLASHLIGHT assembly in this Project:

- BATTERY.
- BATTERYPLATE.



FLASHLIGHT Assembly

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

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

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

Utilize existing faces and edges to create the following features:

- Fillet.
- Chamfer.

This project introduces you to the Revolved feature.

Create two parts for the FLASHLIGHT assembly in this section:

- LENS.
- BULB.

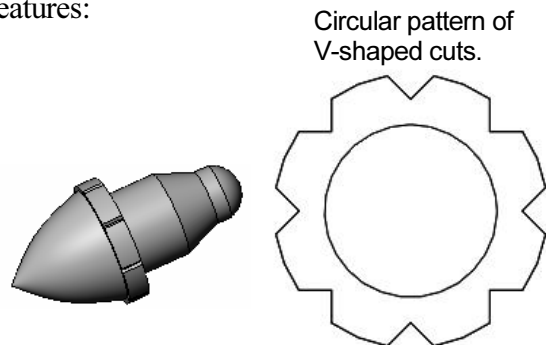
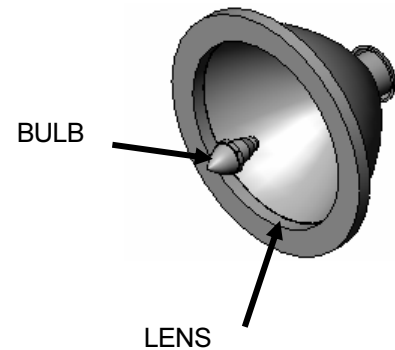
A Revolved feature requires a 2D sketch profile and a centerline. Utilize sketch geometry and sketch tools to create the following features:

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

Utilize existing faces to create the following features:

- Shell.
- Dome.
- Hole Wizard.

Utilize the Extruded Cut feature to create a Circular Pattern.



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

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:

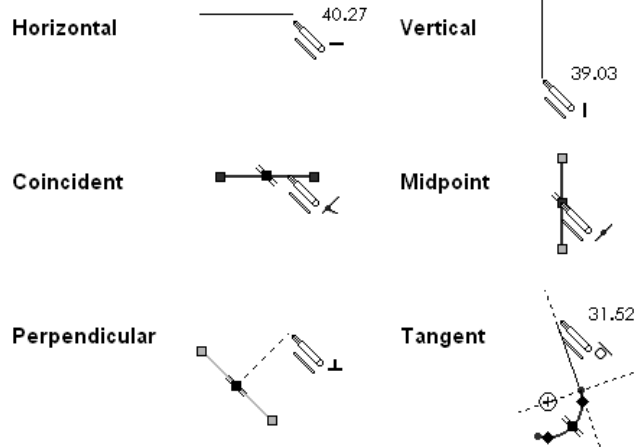
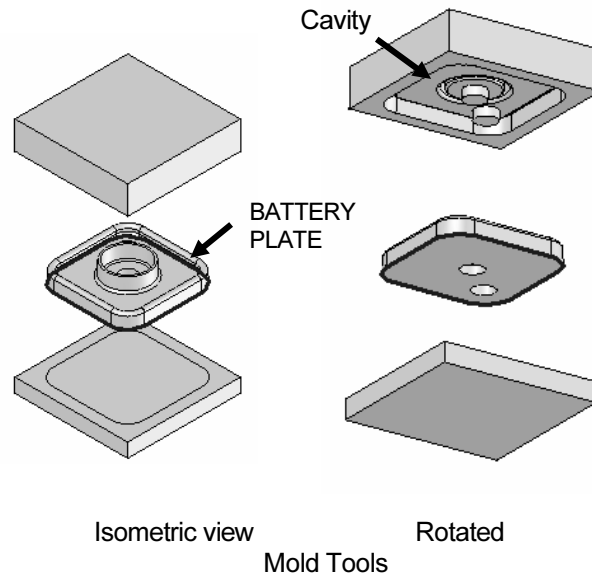
1. Sketch.
2. Feature.
3. Part.
4. Assembly.
5. Drawing.

1. Design Intent in the Sketch.

Build the design intent in the sketch as the profile is created. A profile is determined from the sketch tools, Example: Rectangle, Circle, and Arc.

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

Build design intent as you sketch with automatic relationships.



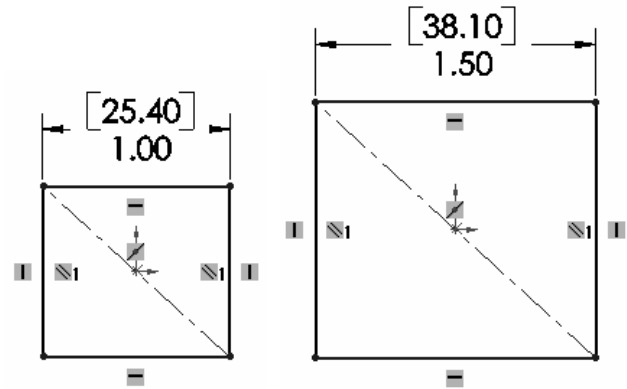
A rectangle contains horizontal, vertical and perpendicular relations.

Build design intent using added Geometric relations. Example: Horizontal, Vertical, Coincident, Midpoint, Intersection, Tangent, and Perpendicular.

Example A: Develop a square profile.

Build the design intent to create a square profile.

Sketch a rectangle. Insert a centerline. Add a Midpoint relation. Add an Equal relation between the two perpendicular lines. Insert a dimension to define the width of the square.



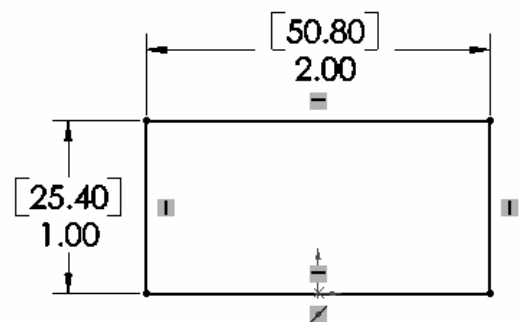
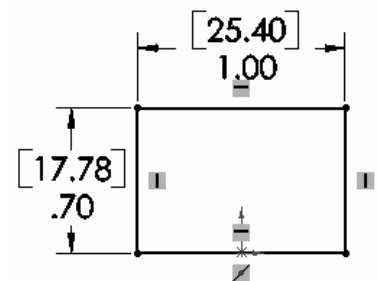
Example B: Develop a rectangular profile.

The bottom horizontal midpoint of the rectangular profile is located at the Origin.

Sketch a rectangle.

Add a Midpoint relation between the horizontal edge of the rectangle and the Origin.

Insert two dimensions to define the width and height of the rectangle.



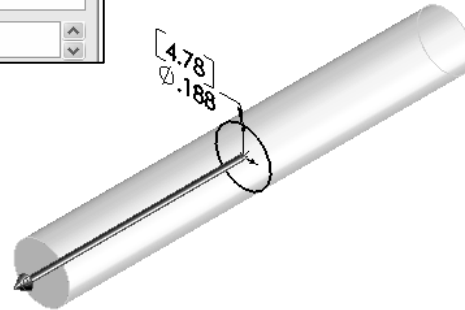
2. Design Intent in the Feature.

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

Example A: Extruded feature remains symmetric about a plane.



Utilize the Mid Plane End Condition type in Direction 1. Modify the depth and the feature remains symmetric about the Front plane.

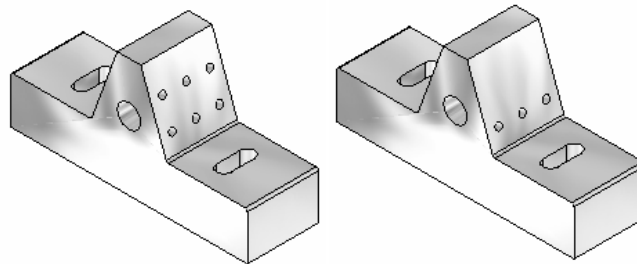


Example B: Create six holes for a pattern.

Do you create six separate Extruded Cuts? No. Create a single hole with the Hole Wizard feature.

Insert a Linear Pattern feature.

Modify the number of holes from six to three.



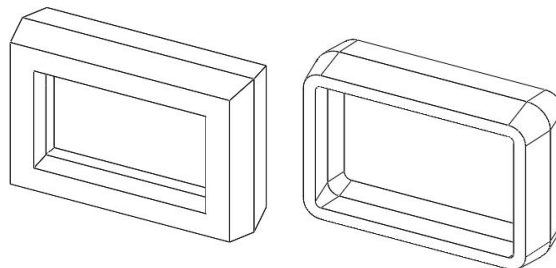
3. Design Intent in the Part.

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.

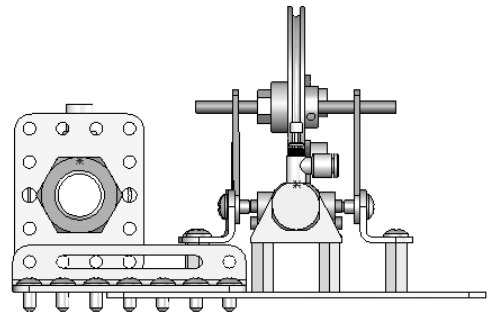
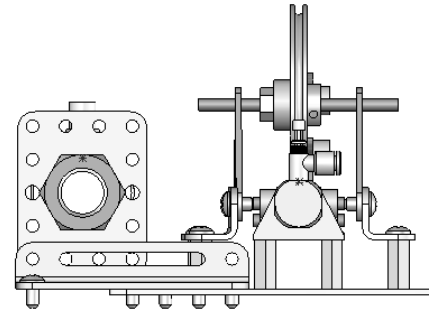


4. Design Intent in the Assembly.

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

Example A: Reuse geometry in an assembly.

The PNEUMATIC-TEST-MODULE assembly contains a Linear Pattern of holes. Insert a SCREW into the first hole. Utilize the Linear Pattern feature to copy the single SCREW to the other locations.



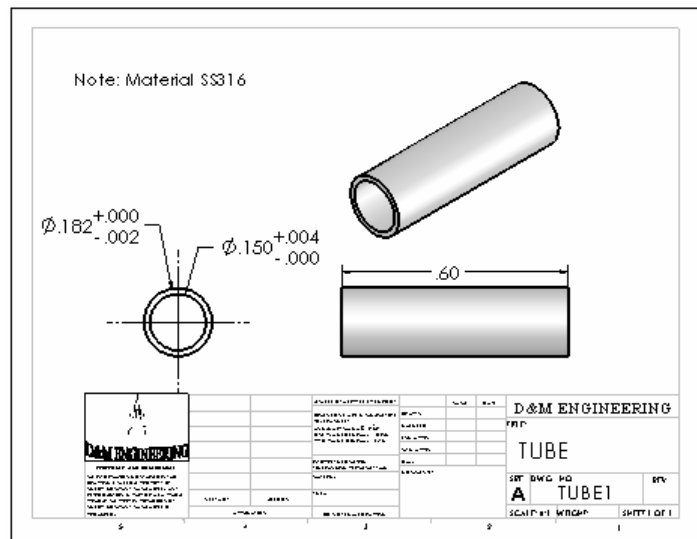
5. Design Intent in the Drawing.


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.



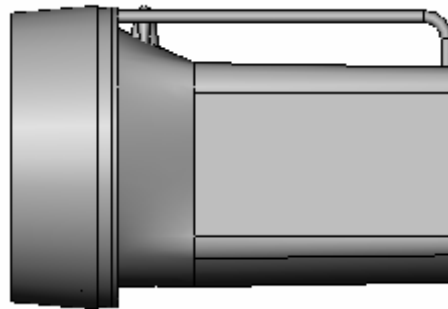
 Additional information on the design process and design intent is available in SolidWorks Help.

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.

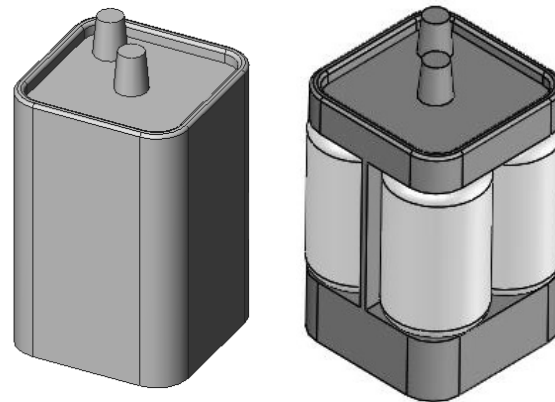
You are the design engineer responsible for the project. You contact the customer to discuss design options and product specifications. The customer informs you that the flashlights will be used in an international marketing promotional campaign. Key customer requirements:

- 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.

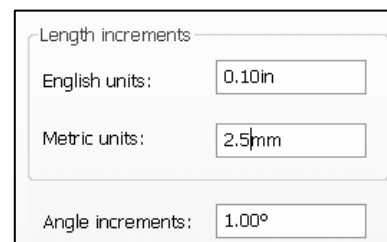
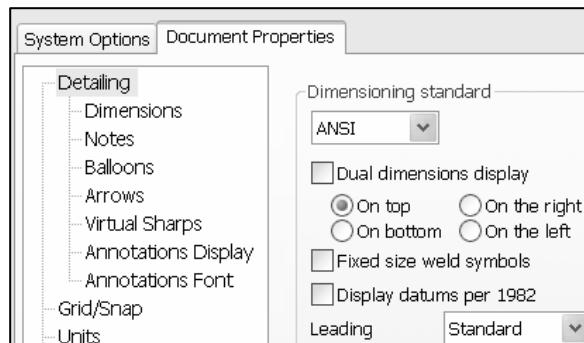
Activity: Part Template

Create the PART-IN-ANSI Template.

- 1) Click **File, New** from the Main menu.
- 2) Double-click **Part** from the default Templates tab.

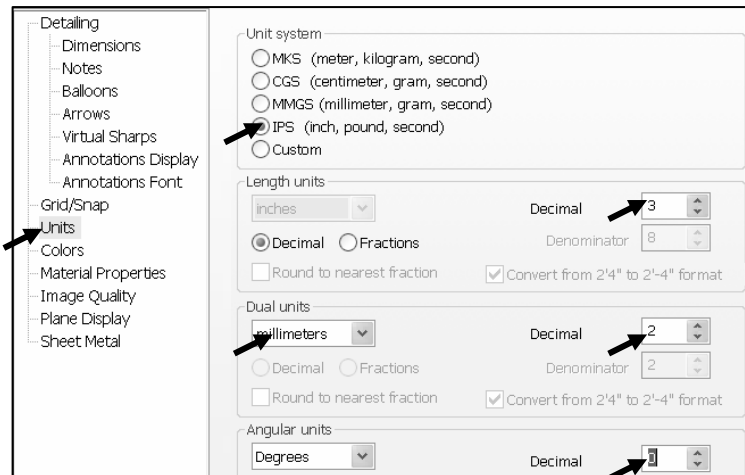
Set the Dimensioning Standard to ANSI.

- 3) Click **Tools, Options, Document Properties** tab from the Main menu.
- 4) Select **ANSI** from the Dimensioning Standard box.
- 5) Click the **System Options** tab.
- 6) Click **Spin Box Increments**.
- 7) Click **inside** the English units box.
- 8) Enter **.100in**.
- 9) Click **inside** the Metric units box.
- 10) Enter **2.5mm**.
- 11) Click the **Document Properties** tab.



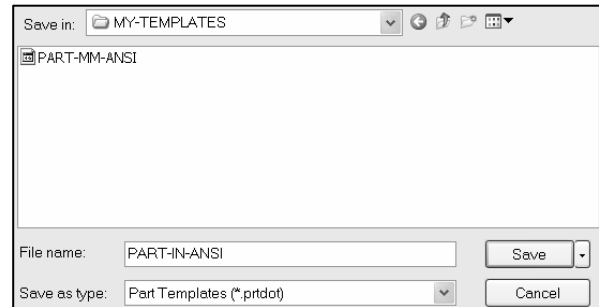
Set the part units for inch.

- 12) Click **Units**.
- 13) Select **IPS** for Unit system.
- 14) Select **3** for Length units Decimal places.
- 15) Select **millimeters** for Dual units.
- 16) Select **2** for Decimal places.
- 17) Select **0** for Angular units Decimal places.
- 18) Click **OK**.



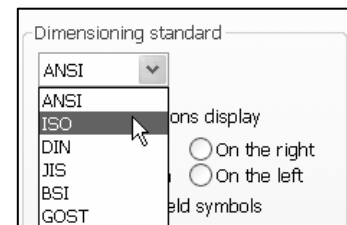
Save the part template.

- 19) Click **File, Save As** from the Main menu.
- 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 **Tools, Options, Document Properties** tab from the Main menu.
- 25) Select **ISO** from the Dimensioning standard box.

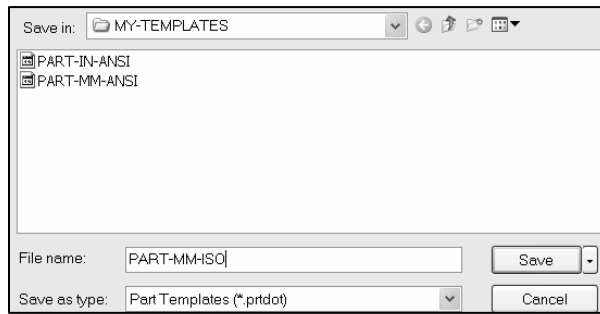


Set the part units for millimeter.

- 26) Click **Units**.
- 27) Select **MMGS** for Unit system.
- 28) Select **2** for Length units Decimal places.
- 29) Select **0** for Angular units Decimal places.
- 30) Click **OK**.

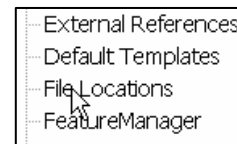
Save the part template.

- 31) Click **File, Save As** from the Main menu.
- 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.

- 36) Click **Tools, Options** from the Main menu.
- 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.

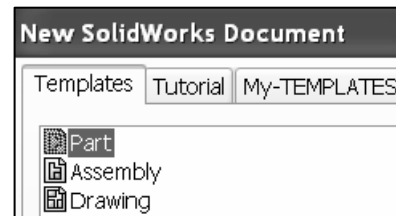


Close all documents.

- 43) Click **Windows, Close All** from the Main menu.



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



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

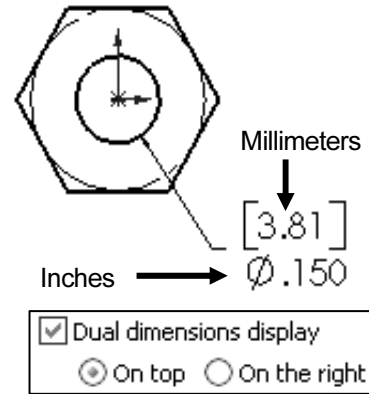
The PART-IN-ANSI Template contains Document Properties settings for the parts contained in the FLASHLIGHT assembly. Substitute the PART-MM-ISO or PART-MM-ANSI Template to create the identical parts in millimeters.


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

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


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

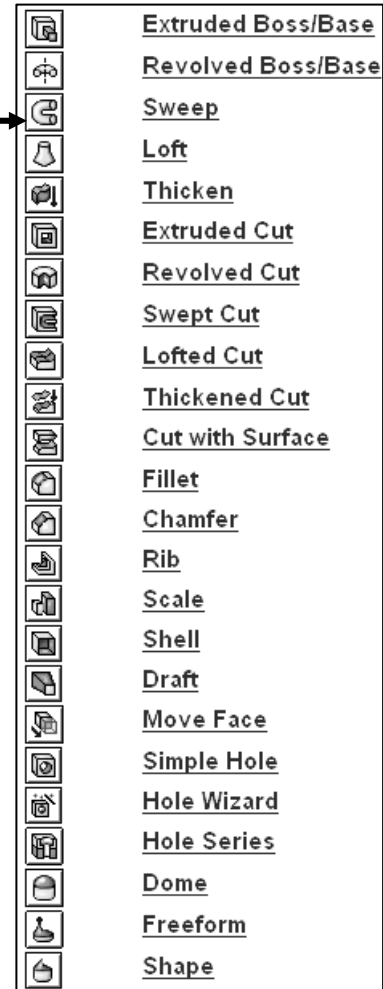
To set dual dimensions, select Tools, Options, Document Properties. Check the Dual dimension display box.



 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-ISO and PART-IN-ANSI. The Document Properties Dimensioning Standard, Units, and Decimal Places were stored in the Part Templates.

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

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

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

BATTERY Part

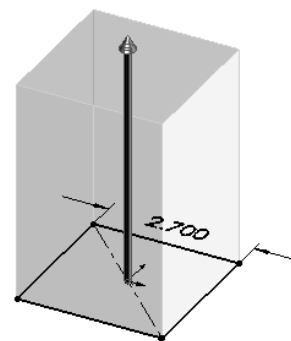
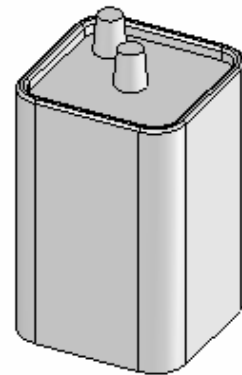
The BATTERY is a simplified representation of a purchased OEM part. Represent the battery terminals as cylindrical extrusions. The battery dimensions are obtained from the ANSI standard 908D.

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

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

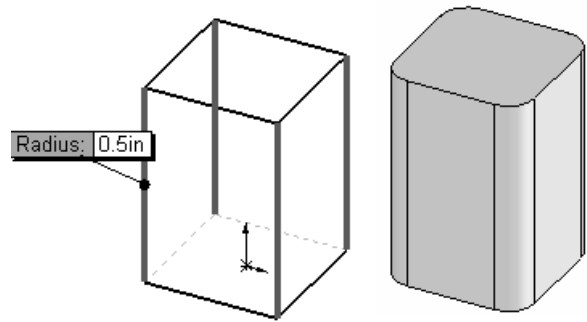
Utilize the Extruded Base feature. The Extrude Base features add material. The Base feature is the first feature of the part.

Utilize symmetry. Sketch a rectangle profile on the Top plane, centered at the Origin.

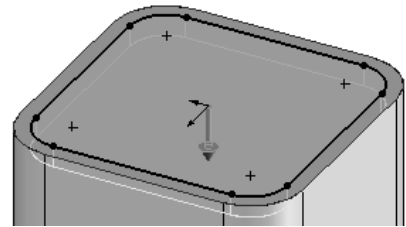


Extend the profile perpendicular (\perp) to the Top plane.

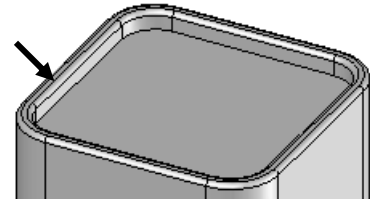
Utilize the Fillet feature to round the four vertical edges.



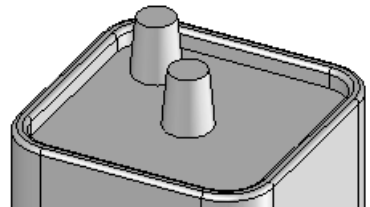
The Extruded Cut feature removes material from the top face. Utilize the top face for the Sketch plane. Utilize the Offset Entity Sketch tool to create the profile.



Utilize the Fillet feature to round the top narrow face.



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



BATTERY Part-Extruded Base Feature

The Extruded Base feature requires:

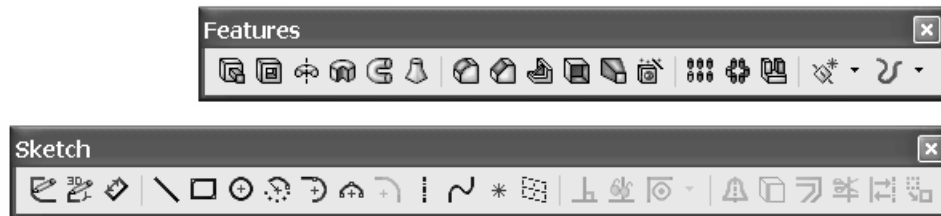
- Sketch plane (Top).
- Sketch profile (Rectangle).
 - Geometric relations and dimensions.
- End Condition depth (Blind) in Direction 1.

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

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

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

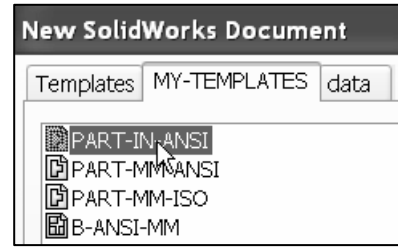
Note: Alternate between Feature and Sketch in the Control Area to display the Features toolbar and Sketch toolbar or display the individual toolbars outside the Graphics window.




Activity: BATTERY Part-Extruded Base Feature

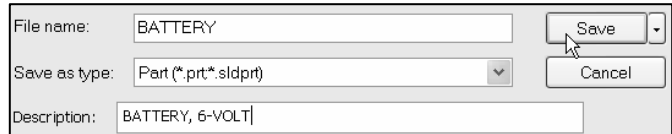
Create a new part.

- 44) Click **File, New** from the Main menu.
- 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.






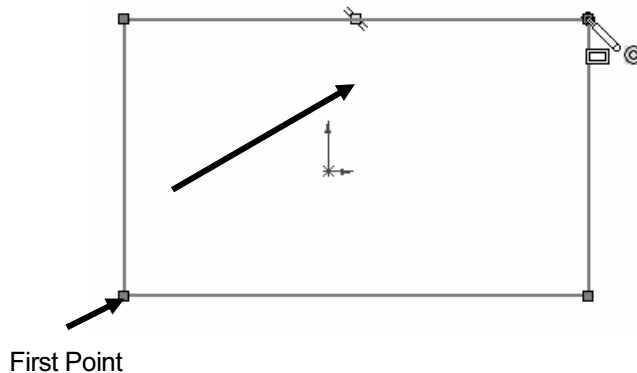
Select the Sketch plane.

- 52) Click **Top Plane** from the FeatureManager.




Sketch the profile.

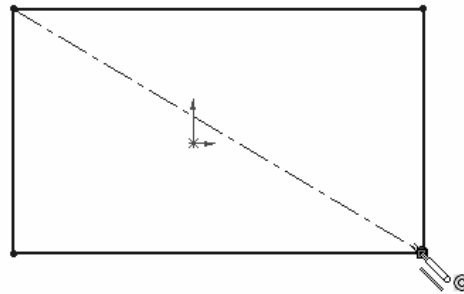
- 53) Click **Sketch**  from the Sketch toolbar.
- 54) Click **Top view** .
- 55) Click **Rectangle**  from the Sketch toolbar.
- 56) Click the **first point** in the lower left quadrant.
- 57) Drag and click the **second point** in the upper right quadrant as illustrated. The Origin is approximately in the middle of the rectangle. Insert a Midpoint relation in the next step.



Sketch the Centerline.

58) Click **Centerline**  from the Sketch toolbar. The Insert Line PropertyManager is displayed.

59) Sketch a diagonal centerline from the **upper left corner** to the **lower right corner**. The endpoints of the centerline are Coincident with the corner points of the rectangle.



Add a Midpoint relation.


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


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

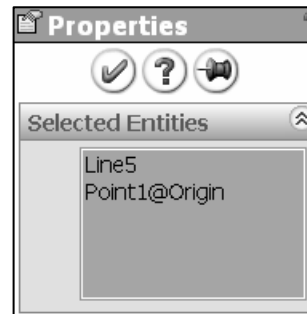
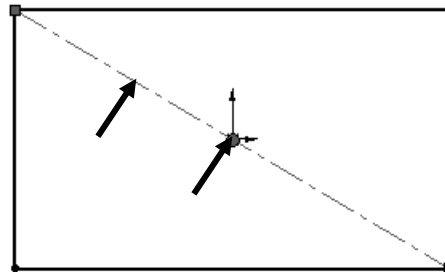
62) Hold the **Ctrl** key down.

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

64) Release the **Ctrl** key.

65) Click **Midpoint**  from the Add Relations box.

66) Click **OK**  from the Properties PropertyManager. Note: The Line# may be different than illustrated. The Line# is dependent on the line number order creation.



To clear entities from the Selected Entities box, right-click Clear Selections.


Create a square. Add an Equal relation.

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

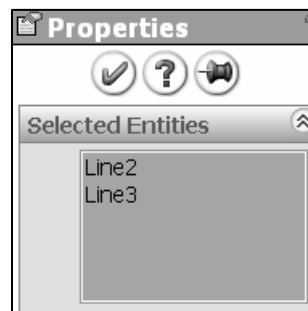
68) Hold the **Ctrl** key down.

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

70) Release the **Ctrl** key.

71) Click **Equal**  from the Add Relations box.

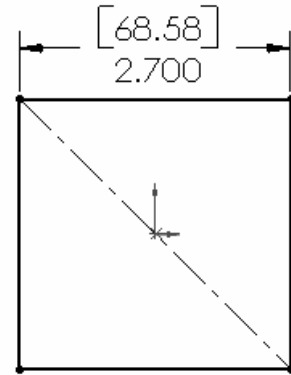
72) Click **OK**  from the Properties PropertyManager.



Add a dimension.



- 73) Click **Smart Dimension** Dimens... from the Sketch toolbar.
- 74) Click the **top horizontal line**.
- 75) Click a **position** above the horizontal line.
- 76) Enter **2.700in**, **[68.58]** for width.
- 77) Click the **Green Check mark** ✓. The black Sketch status is fully defined.

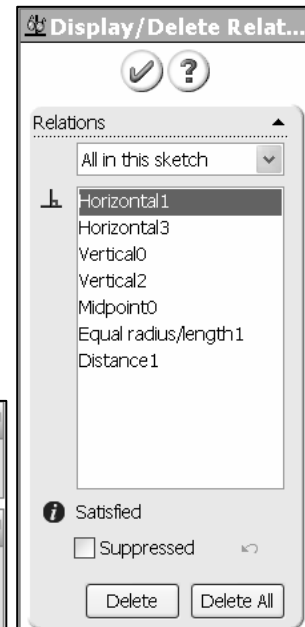


- 78) Click **OK** ✓ from the Dimension PropertyManager.

Display the Sketch Relations.



- 79) Click **Display/Delete Relations** Relations from the Sketch toolbar. The Distance relation was created from the dimension. The Display/Delete Relations PropertyManager is displayed.

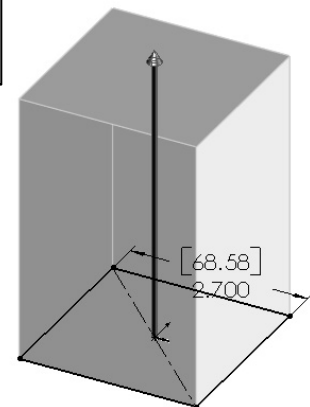
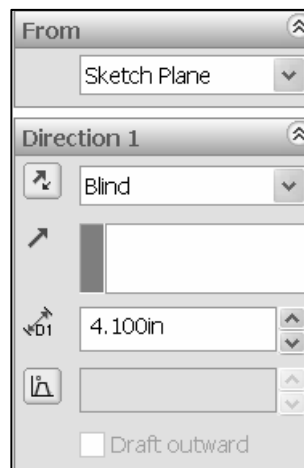


- 80) Click **OK** ✓ from the Display/Delete Relations PropertyManager.

Insert an Extruded Base feature.



- 81) Click **Extruded Boss/Base** Boss/B... from the Features toolbar. The Extrude PropertyManager is displayed. Blind is the default End Condition.
- 82) Enter **4.100in**, **[104.14]** for Depth in Direction 1. Accept the default settings.
- 83) Click **OK** ✓ from the Extrude PropertyManager. Extrude1 is displayed in the FeatureManager.



Fit the part to the Graphics window.


84) Press the **f** key.

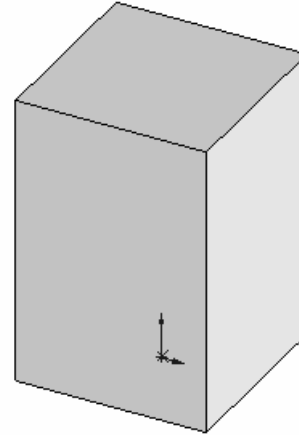
Rename the Extruded Base feature.

85) Rename **Extrude1** to **Base Extrude**.

Save the BATTERY.

86) Click **Save** .

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



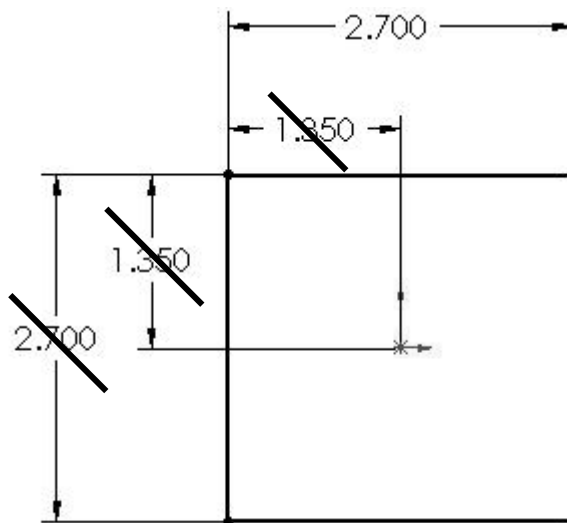
One dimension controls the size.
The 6-Volt manufacturing standard determines the square profile.

The Midpoint relation centers the square profile about the Origin.


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

The color of the sketch indicates the sketch status.

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



 Short Cuts save time. Right-click Select  Select to choose geometry.

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

BATTERY Part-Fillet Feature Edge

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


An edge Fillet requires:

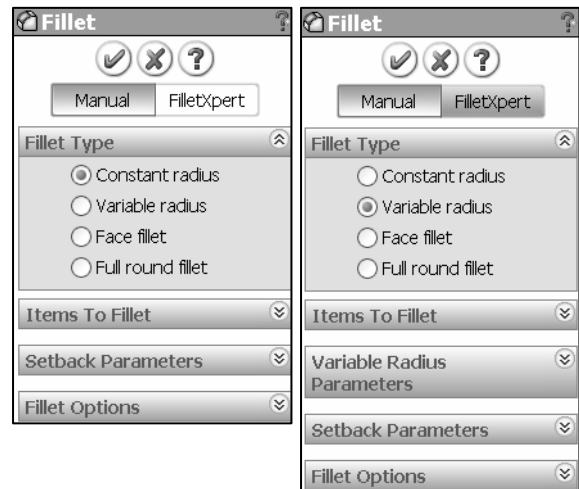
- Edge.
- Fillet radius.

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

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

The Fillet feature uses the Fillet PropertyManager. The Fillet PropertyManager for 2007 has had a major interface enhancement, and provides the ability to select either the Manual or FilletXpert tab. Each tab has a separate menu and option selections. The Fillet 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 or change fillets using the Add or Change tab. Use the Add tab to create new constant radius fillets. The PropertyManager remembers its last used state.





The FilletXpert can ONLY create and edit constant radius fillets.

Activity: BATTERY Part-Fillet Feature Edge

Display the hidden edges.

87) Click **Hidden Lines Visible**  from the View toolbar.

Insert a Fillet feature.

88) Click the **left vertical edge** as illustrated.

89) Click **Fillet**  from the Features toolbar. The Fillet PropertyManager is displayed.

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

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

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

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

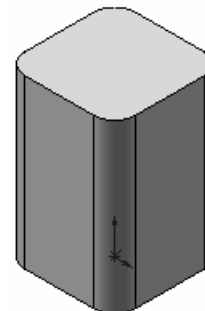
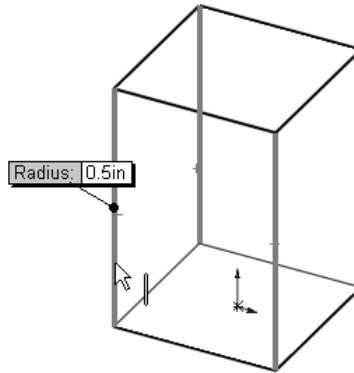
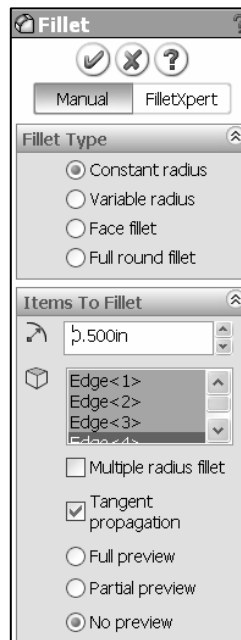
94) Click **Isometric view** .

95) Click **Shaded With Edges** .

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

Save the BATTERY.

97) 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.

- 98)** Click the **Top face** of the BATTERY in the Graphics window. Base Extruded is highlighted in the FeatureManager.

Create a Sketch.


- 99)** Click **Sketch**  from the Sketch toolbar.

Display the face.

- 100)** Click **Top view** .



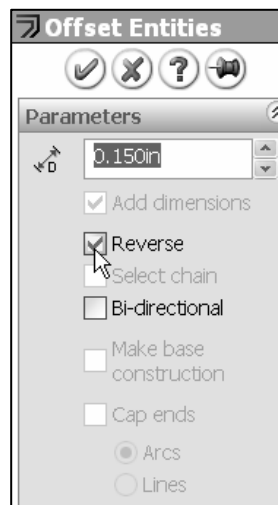
Offset the existing geometry from the boundary of the Sketch plane.

- 101)** Click **Offset Entities**  from the Sketch toolbar. The Offset Entities PropertyManager is displayed.

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

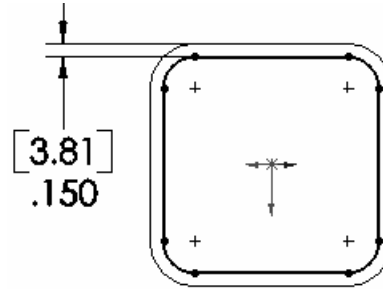
- 103)** Click the **Reverse** box. The new Offset yellow profile displays inside the original profile.

- 104)** 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.


105) Click **Isometric view** .

Insert an Extruded Cut feature.

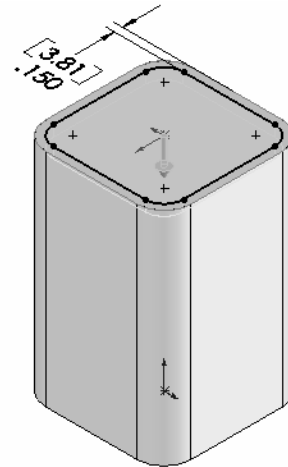


106) Click **Extruded Cut** from the Features toolbar. The Cut-Extrude PropertyManager is displayed.

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

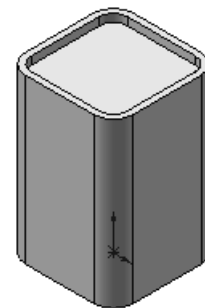
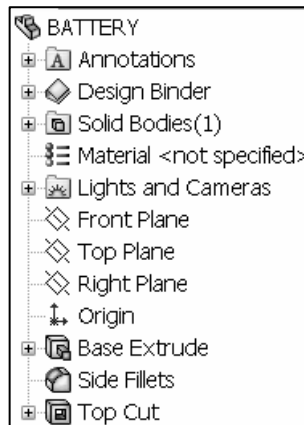
108) Click **OK**  from the Cut-Extrude PropertyManager. Cut-Extrude1 is displayed in the FeatureManager.

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



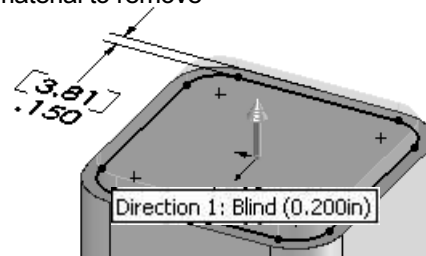
Save the BATTERY

110) Click **Save** .

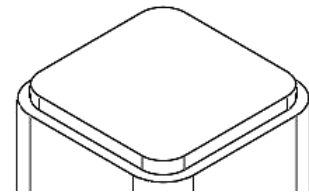
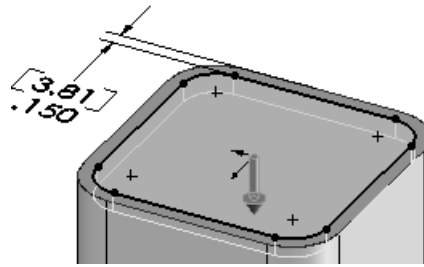


The Cut-Extrude PropertyManager contains numerous options. The Reverse Direction option determines the direction of the Extruded Cut. The Extruded Cut is valid only when the direction arrow points into material to be removed.

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

- Face.
- Fillet radius.

Activity: BATTERY Part-Fillet Feature Face


Insert a Fillet feature on the top face.

111) Click the **top thin face** as illustrated.

112) Click **Fillet**  from the Features toolbar. The Fillet PropertyManager is displayed. Face<1> is displayed in the Items To Fillet box.

113) Click **Constant Radius** for Fillet Type.

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

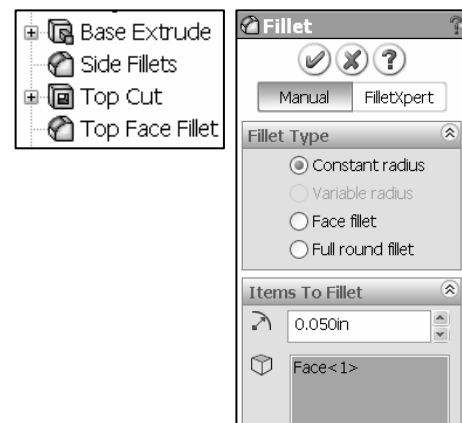
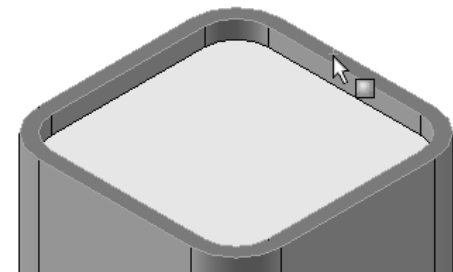
115) Click **OK**  from the Fillet PropertyManager. Fillet2 is displayed in the FeatureManager.

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

117) Press the **f** key.

Save the BATTERY.

118) Click **Save** .



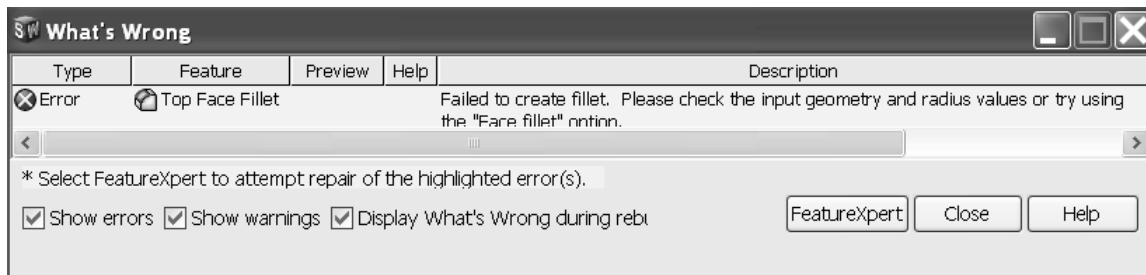
View the mouse pointer for feedback to select Edges or Faces for the fillet.



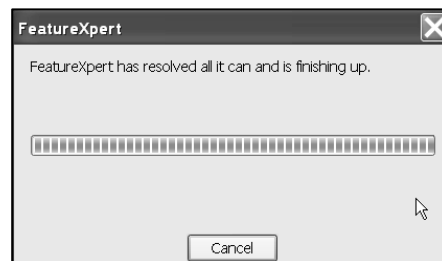
Do not select a fillet radius which is larger than the surrounding geometry.

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 Feature

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

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

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

An angular dimension is required between the Right plane and the centerline. Acute angles are less than 90°. Acute angles are the preferred dimension standard. The overall BATTERY height is a critical dimension. The BATTERY height is 4.500in, [114.30].

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


119) Click the **Top face** of the Top Cut feature in the Graphics window.

Create the Sketch.

120) Click **Sketch**  from the Sketch toolbar.

121) Click **Top view** .

Sketch the profile.

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

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



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

Add a dimension.

125) Click **Smart Dimension**  from the Sketch toolbar.

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

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

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

129) Click the **Green Check mark** . The black sketch is fully defined.

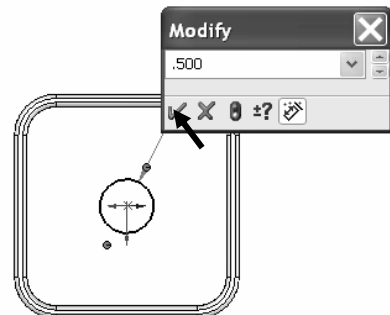
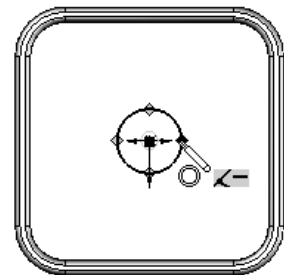
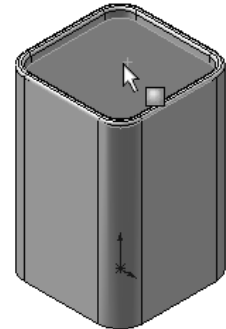
Copy the sketched circle.

130) Right-click **Select** .

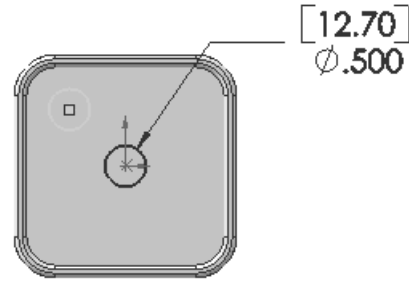
131) Hold the **Ctrl** key down.

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

133) Release the **mouse button**.

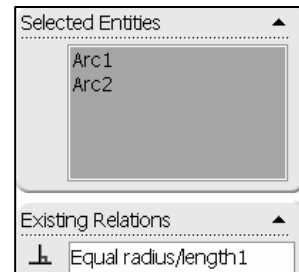




- 134)** Release the **Ctrl** key. The second circle is selected and is displayed in green.



Add an Equal relation.

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




- 137)** Release the **Ctrl** key.
- 138)** Click **Equal**  from the Add Relations box.
- 139)** Click **OK**  from the Properties PropertyManager.

Show the Right Plane for the dimension reference.

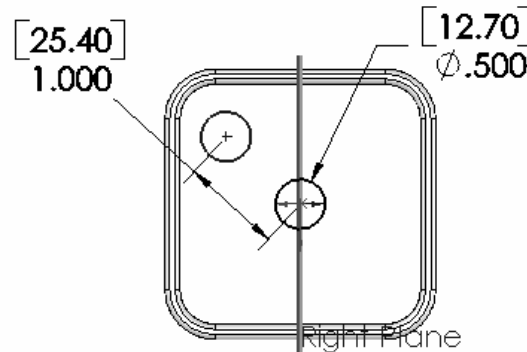
- 140)** Right-click **Right Plane** from the FeatureManager.

- 141)** Click **Show**.

Add an aligned dimension.


- 142)** Click **Smart Dimension**  **Dimens...** from the Sketch toolbar.

- 143)** Click the **two center points** of the two circles.
- 144)** Click a **position** off the profile in the upper left corner.
- 145)** Enter **1.000in**, [**25.4**] for the aligned dimension.



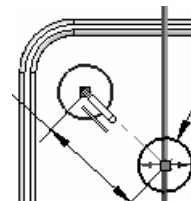
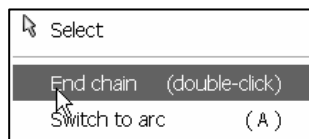
- 146)** Click the **Green Check mark** .

Insert a centerline.


- 147)** Click **Centerline**  **Centerl...** from the Sketch toolbar. The Insert Line PropertyManager is displayed.

- 148)** Sketch a centerline between the **two circle center points**.

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



 Shortcut: Double-click to end the centerline.


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

Add an angular dimension.

 Smart Dimens...
150) Click **Smart Dimension** from the Sketch toolbar.

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

152) Click the **Right Plane** (vertical line) in the Graphics window. Click a **position** between the centerline and the Right plane, off the profile.


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


Fit the Model to the Graphics window.

154) Press the **f** key.

Hide the Right Plane.


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

156) Click **Hide**. 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.

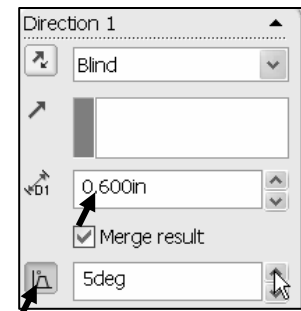
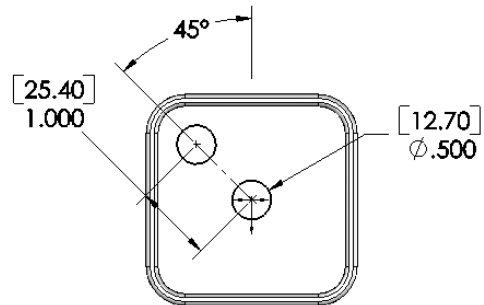
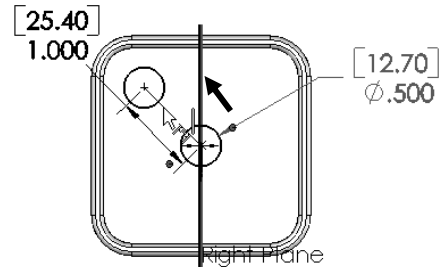
157) Click **Isometric view** .

 Extruded Boss/B...
158) Click **Extruded Boss/Base** from the Features toolbar. The Extrude PropertyManager is displayed. Blind is the default End Condition Type.

159) Enter **.600in**, **[15.24]** for Depth in Direction 1.

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

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

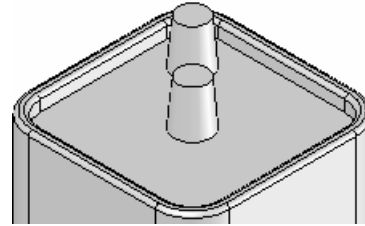
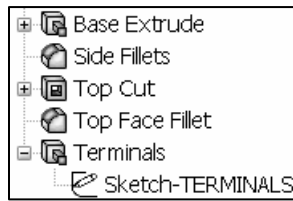


Rename the Feature and Sketch.

162) Rename **Extrude2** to **Terminals**.

163) **Expand** Terminals.

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



165) Click **Save**  .


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



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

Measure the overall BATTERY height.

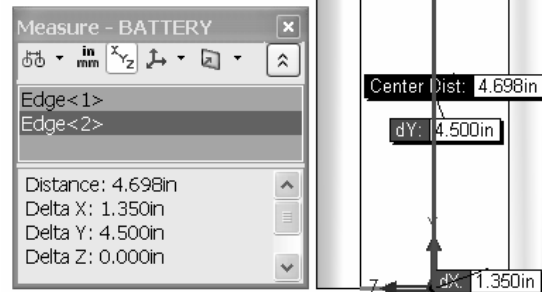
166) Click **Right view**  .


167) Click **Tools, Measure**  Measure... from Main menu. The Measure - BATTERY dialog box is displayed.

168) Click the **top edge** of the battery terminal.



169) Click the **bottom edge** of the battery. The overall height, Delta Y is 4.500, [114.3].





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

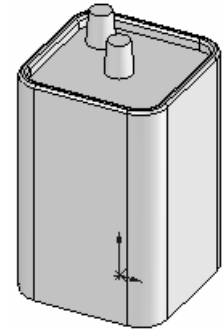


To measure a line or diameter quickly, click an entity. Read the measured value in the Status bar  Editing Part in the right corner of the Graphics window.



Save time. The Standard toolbar contains quick access to the icons: Tools, Selection Filters and Help  . The Tools  icon displays the Measure, Equations, Analysis, and other commands.

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.

171) Click **Trimetric view**  from the View toolbar.

Save the BATTERY.

172) Click **Save** .

 Additional information on Extrude Boss/Base Extrude Cut and Fillets is located in SolidWorks Help Topics. Keywords: Extrude (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, Fillets exercise for additional information.



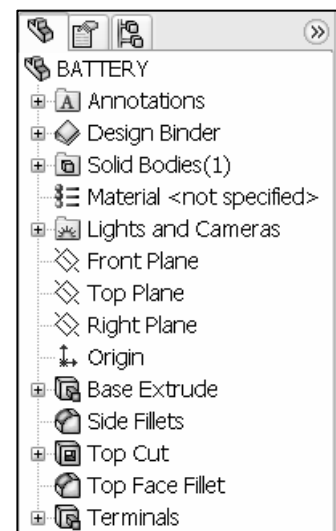
Review of the BATTERY Part.

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

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

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

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



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.

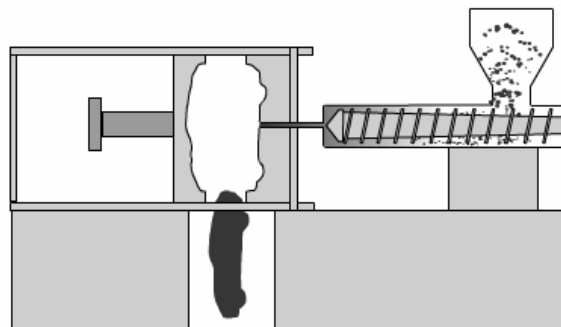
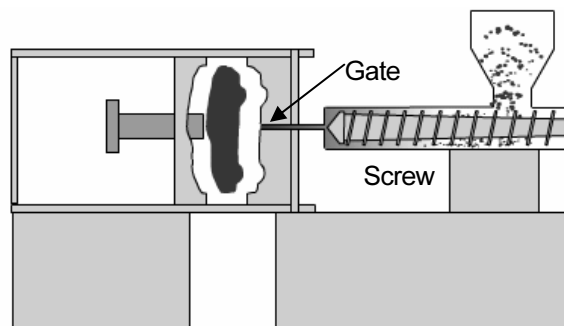
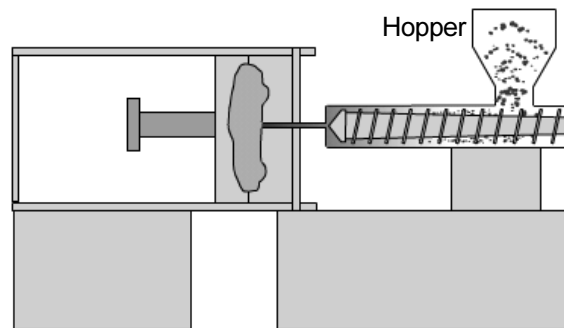
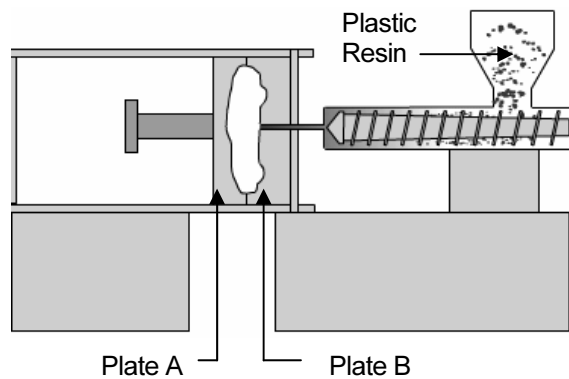
The injection molding process is as follows:

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 Base\Boss features.

Edit the BATTERY features. Create two holes from the original sketched circles. Use the Extruded Cut feature.

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


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

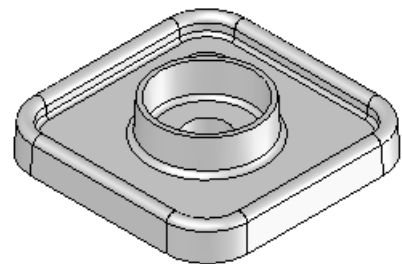
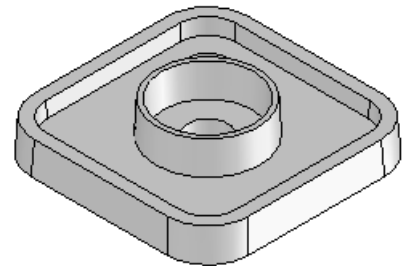
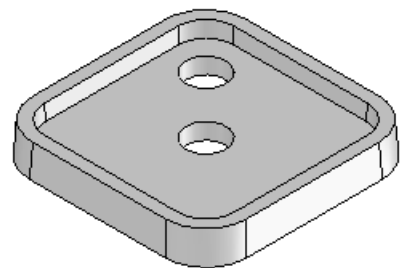
Insert an Extruded Boss feature. Offset the center circular sketch.

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

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

In this project you will perform a Draft Analysis on this 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, Modify, and Edit Feature

Create the BATTERYPLATE from the BATTERY. Utilize the File, Save As option to copy the BATTERY to the BATTERYPLATE.

Reuse existing geometry. Create two holes. Delete the Terminals feature and reuse the circle sketch. Select the sketch in the FeatureManager. Insert an Extruded Cut feature. The Through All Depth option creates two holes that cut through the entire Extruded Base feature.

Right-click the Cut-Extrude1 feature in the FeatureManager. Select Edit Feature. The Edit Feature option returns to the Cut-Extrude1 PropertyManager. Modify the End Condition from Blind to Through All in Direction 1.

Modify the depth dimension of the Extruded Base feature. Sketch dimensions are displayed in black. Feature dimensions are displayed in blue. Select Rebuild to update the part.

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

Create a new part.

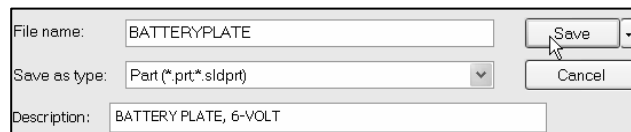
173) Click **File, Save As** from the Main menu.

174) Select **PROJECTS** for Save In folder.

175) Enter **BATTERYPLATE** for File name.

176) Enter **BATTERY PLATE, FOR 6-VOLT** for Description.

177) Click **Save**. The BATTERYPLATE FeatureManager is displayed. The BATTERY part is closed.

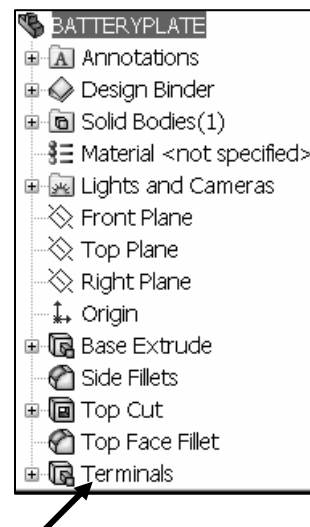


Delete the Terminals feature.

178) Right-click **Terminals** from the FeatureManager.

179) Click **Delete**.

180) 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.

181) Click **Sketch-TERMINALS** from the FeatureManager.

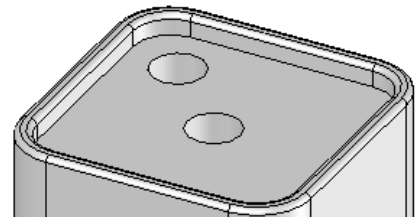
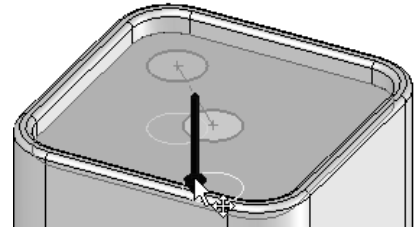


182) Click **Extruded Cut** from the Features toolbar. The Cut-Extrude PropertyManager is displayed.

183) Select **Through All** for End Condition in Direction 1.

184) Click **OK** from the Cut-Extrude PropertyManager. Cut-Extrude2 is displayed in the FeatureManager.

185) Rename **Cut-Extrude2** to **Holes** in the FeatureManager.



Edit the Base Extrude feature.

186) Right-click **Base Extrude** from the FeatureManager.

187) Click **Edit Feature** from the Pop-up menu.

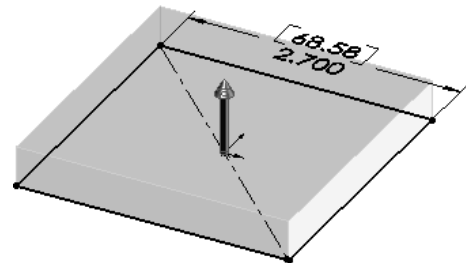
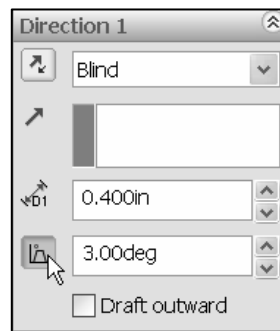
Change the overall Depth.

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.

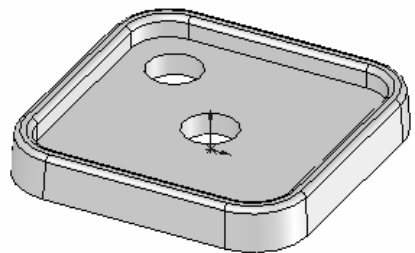


Fit the model to the Graphics window.

192) Press the **f** key.

Save the BATTERYPLATE.

193) Click **Save**.

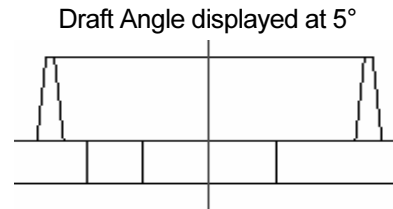


To delete both the feature and the sketch at the same time, select the Also Delete Absorbed Feature check box from the Confirm Delete dialog box.

BATTERYPLATE Part-Extruded Boss Feature

The Holder is created with a circular Extruded Boss feature. Utilize Offset Sketch Entity to create the second circle. Utilize 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. A 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) Click the **top face** of Top Cut.

Create the Sketch.



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

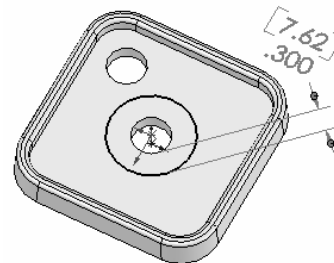
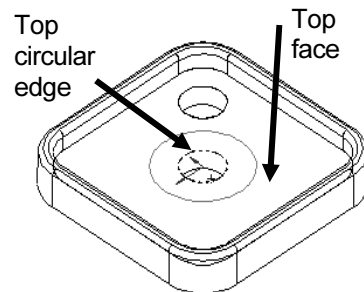
196) Click the **top circular edge** of the center Hole. Note: Use the keyboard arrow keys to rotate the sketch.



197) Click **Offset Entities** from the Sketch toolbar. 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.



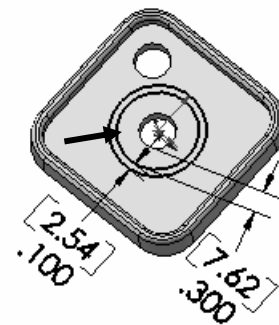
Create the second offset circle.


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



201) Click **Offset Entities** from the Sketch toolbar. The Offset Entities PropertyManager is displayed.


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

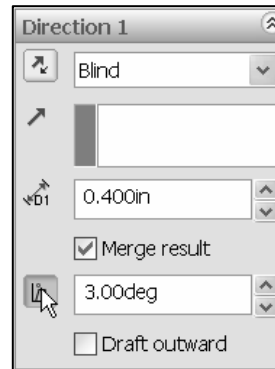


203) Click **OK**  from the Offset Entities PropertyManager.

Two offset concentric circles define the sketch.

Insert the Extruded Boss feature.


204) Click **Extruded Boss/Base**  from the Features toolbar. The Extrude PropertyManager is displayed.



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

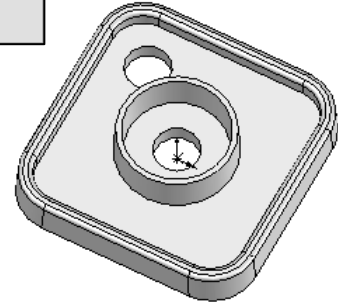
206) Click the **Draft ON/OFF** button.

207) Enter 3deg in the Angle box.

208) Click **OK**  from the Extrude PropertyManager. Extrude3 is displayed in the FeatureManager.

209) Rename **Extrude3** to **Holder** in the FeatureManager.


210) Click **Save** .



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

Fillet features are used to smooth rough edges. Plastic parts require fillets 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 is a Full Round Fillet. Insert a Full Round Fillet on the top face for a smooth rounded transition.

The second Fillet 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 Fillets into a Fillet folder. Placing Fillets into a folder reduces the time spent for your mold designer or toolmaker to look for each Fillet in the FeatureManager.

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

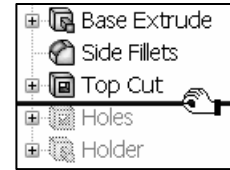
Delete the Top Edge Fillet.

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

212) Click **Delete**.


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

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



Insert the Full Round Fillet feature.

215) Click **Hidden Lines Visible** .

216) Click **Fillet**  from the Features toolbar. The Fillet PropertyManager is displayed.

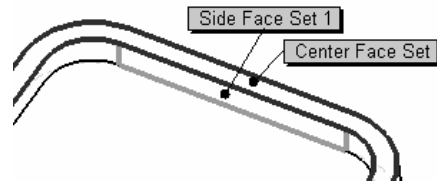
217) Click the **Manual** tab.

218) Click **Full round fillet** for Fillet Type.

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

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

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

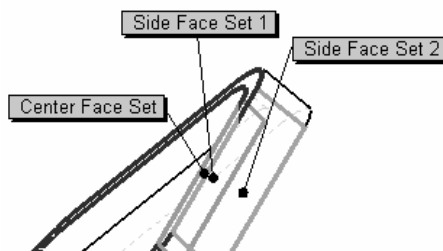



Rotate the part.

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

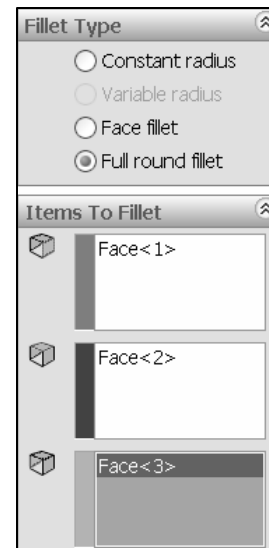
223) Click **inside** the Side Face Set 2 box.

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




225) Click **OK**  from the Fillet PropertyManager. Fillet3 is displayed in the FeatureManager.

226) Rename **Fillet3** to **TopFillet**.



Save the BATTERYPLATE.

227) Click **Isometric view** .

228) Click **Hidden Lines Removed** .


229) Click **Save** .

230) Click **Yes** to rebuild the part.

Note: The Rollback bar is placed at the bottom of the FeatureManager during a Rebuild.

Insert a Multiple Radius Fillet feature.

231) Click the **bottom outside circular edge** of the Holder.

232) Click **Fillet**  from the Features toolbar. The Fillet PropertyManager is displayed.

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

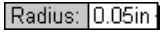
234) Click the **bottom inside circular edge** of the Holder.

235) Click the **inside edge** of the Top Cut.

236) Check the **Tangent propagation** box.

237) Check the **Multiple radius fillet** box.

Modify the Fillet values.

238) Click the **Radius** box  for the Holder outside edge.

239) Enter **0.060in, [1.52]**.

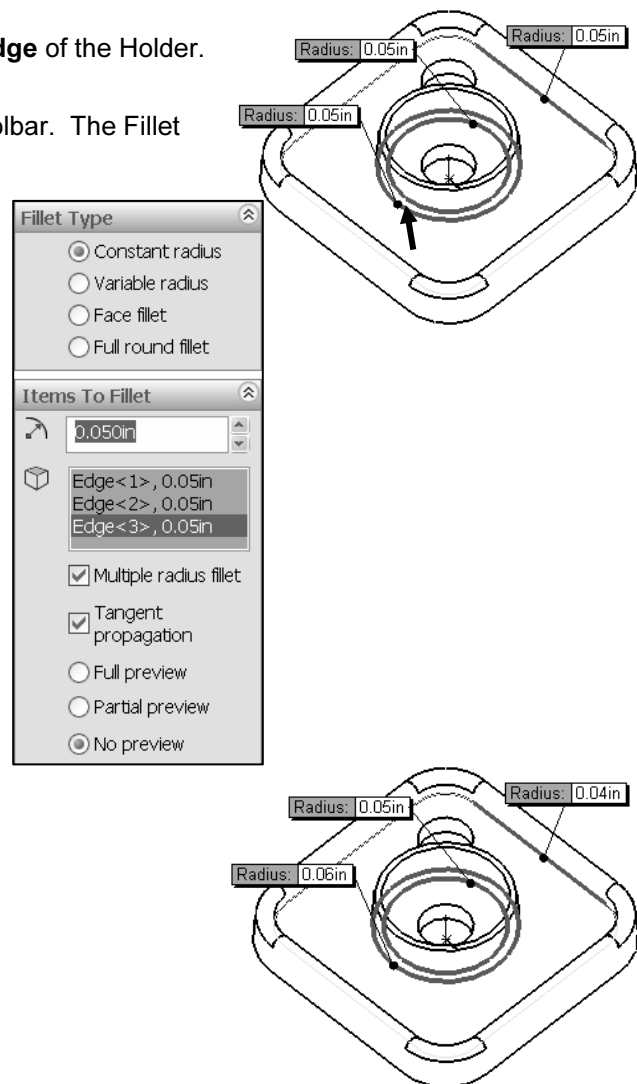
240) Click the **Radius** box for the Top Cut inside edge.

241) Enter **0.040in, [1.02]**.

242) Click **OK**  from the Fillet PropertyManager. Fillet4 is displayed in the FeatureManager.

243) Rename **Fillet4** to **HolderFillet**.

244) Click **Shaded With Edges** .



Group the Fillets into a Folder.

245) Click **TopFillet** from the FeatureManager.

246) Drag the **TopFillet** feature directly above the HolderFillet feature in the FeatureManager.

247) Click **HolderFillet** in the FeatureManager.

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

249) Click **TopFillet** in the FeatureManager.

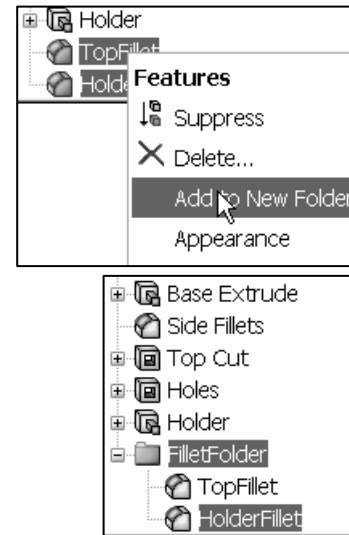
250) Right-click **Add to New Folder**.

251) Release the **Ctrl** key.

252) Rename **Folder1** to **FilletFolder**.

Save the BATTERYPLATE.

253) Click **Save** .

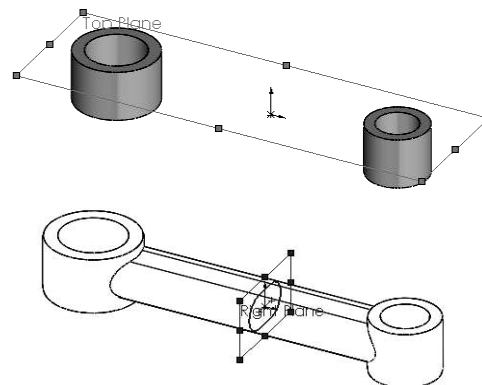


Multibody Parts and the Extruded Boss Feature


A Multibody 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 Topics.
Keywords: Save (save as copy), Extruded (Boss/Base, Cut), Fillet (face blends, variable radius), Chamfer, Geometric relations (sketch), Copy (sketch entities), Multibody (Extrude, Modeling techniques).

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

30 Minute Lesson	Tutorials for Consumer Product Design
	Tutorials for Machine Design
	Tutorials for Mold Design
Lesson 1 - Parts	Lofts
Lesson 2 - Assemblies	Mold Design
Lesson 3 - Drawings	Molded Product Design - Advanced
AutoCAD and SolidWorks	MoldflowXpress
3D Sketching	Multibody Parts
3D Sketching with Planes	Pattern Features
Advanced Design	PDMWorks Workgroup
Advanced Drawings	PhotoWorks
Assembly Mates	Revolves and Sweeps
Blocks	Sheet Metal
COSMOSXpress	Smart Components
Customizing SolidWorks	SolidWorks Animator
Design Checker	SolidWorks API
Design Tables	SolidWorks Utilities
eDrawings	Surfaces
FeatureWorks	Toolbox
Fillets	Weldments
Import/Export	



Review of the BATTERYPLATE Part.

The File, Save As option was utilized to copy the BATTERY part to the BATTERYPLATE part. You modified and deleted features in the BATTERYPLATE.

The BATTERYPLATE is a plastic part. The Draft Angle option was added in the Extruded Base 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 Fillets were grouped together into a Folder. All features were renamed in the FeatureManager. The BATTERY and BATTERYPLATE utilized an Extruded Base feature.

