INSIDE:

MultiMedia CD

An audio/visual presentation of the tutorial projects

SolidWorks 2006: The Basics

with MultiMedia CD

A Working Knowledge of SolidWorks using a Step-by-Step Project Based Approach

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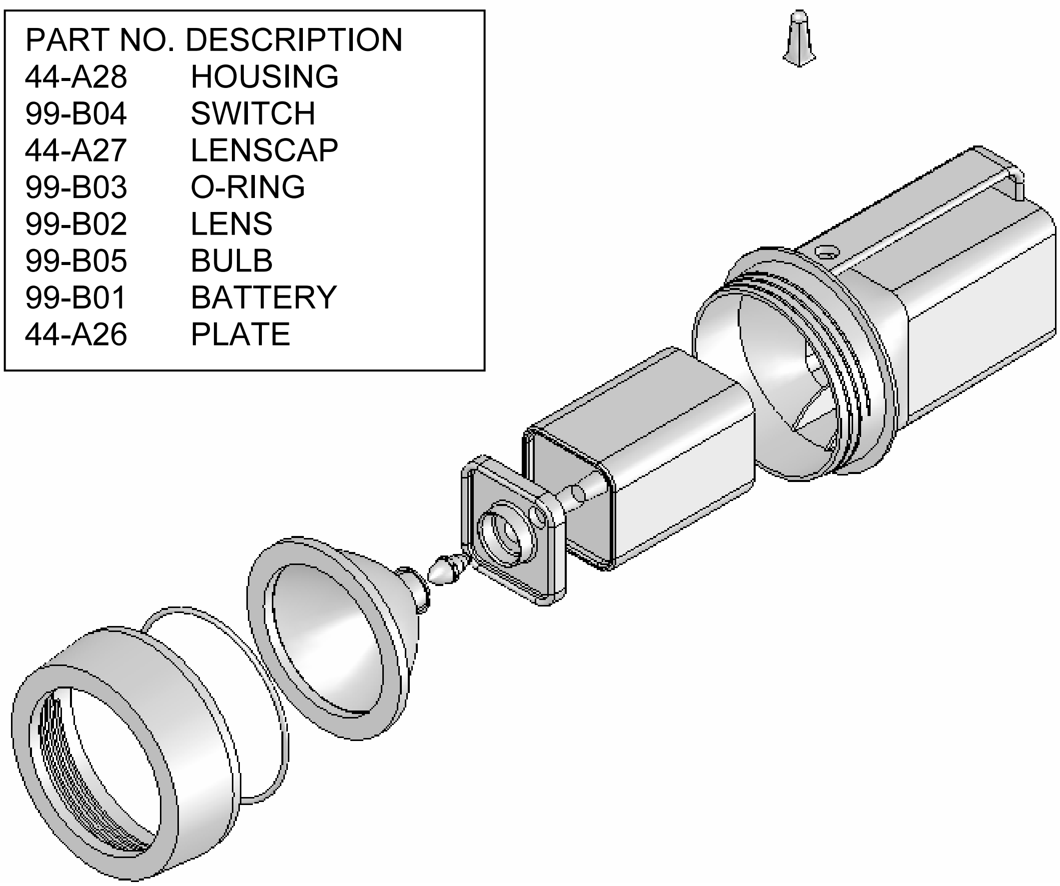
SolidWorks Solution Partner



PART NO.	DESCRIPTION
44-A28	HOUSING
99-B04	SWITCH
44-A27	LENSCAP
99-B03	O-RING
99-B02	LENS
99-B05	BULB
99-B01	BATTERY
44-A26	PLATE

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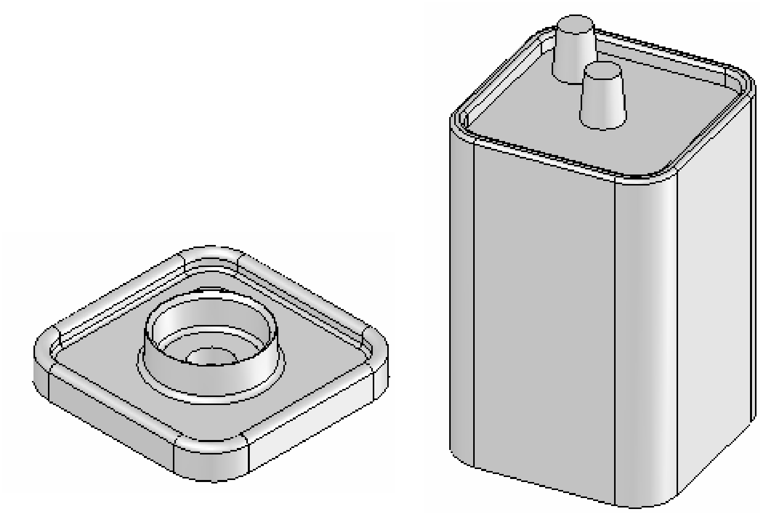
Project 1

Introduction to Part Modeling



Below are the desired outcomes and usage competencies based on the completion of Project 1.

Project Desired Outcomes:	Usage Competencies:
• A comprehensive understanding of the SolidWorks 2006 User Interface.	• Ability to establish and setup a SolidWorks session.
• Address File Management with folders.	• Aptitude to create folders for various projects and templates.
• Two Part Templates: ○ PART-IN-ANSI. ○ PART-MM-ISO.	• Proficiency to apply Document Properties and create custom Part Templates.
• Two FLASHLIGH Parts: ○ BATTERY. ○ BATTERYPLATE.	• Specific knowledge and understanding of the following features: Extruded Boss, Extruded Base, Extruded Cut, Fillet and Chamfer.

Notes:

Project 1-Introduction to Part Modeling

Project Overview

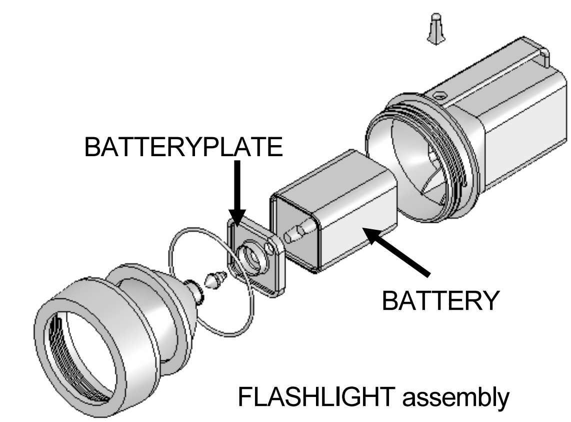
SolidWorks is a 3D design software application used to model and produce parts, assemblies and drawings. Project 1 introduces you to the SolidWorks 2006 software application and user interface.

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

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

Create two parts for the FLASHLIGHT assembly in this project:

- BATTERY.
- BATTERYPLATE.



FLASHLIGHT assembly

Part 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.

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

- Establish a SolidWorks Session.
- Comprehend the SolidWorks 2006 User Interface.
- Recognize default Reference Planes.
- Insert a new sketch and add sketch geometry with the following tools: Line, Circle, Rectangle, Tangent Arc and Centerline.
- Establish Geometric Relations, dimensions and determine the status of the sketch.

- Manipulate existing geometry with the following Sketch tools: Convert, Offset and Mirror Entities.
- Use the following features: Extruded Boss/Base, Extruded Cut, Fillet and Chamfer.
- Create two part templates: PART-IN-ANSI and PART-MM-ISO.
- Create two parts for the FLASHLIGHT assembly: BATTERY and BATTERPLATE

File Management

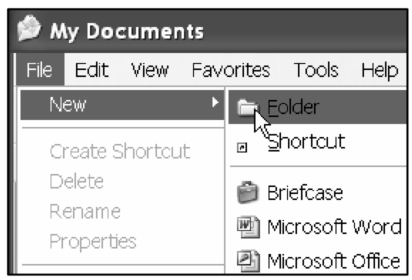
File management organizes parts, assemblies, drawings and templates. Why do you require file management? Answer: A top level assembly has hundreds or even thousands of documents that requires organization. Utilize folders to organize projects, vendor components, templates and libraries. Create the folders. The first folder is named SOLIDWORKS-MODELS. Create two sub-folders named MY-TEMPLATES and PROJECTS.

Activity: File Management

Create a new folder in Windows.

1) Click **Start** from the Windows Taskbar.

2) Click **My Documents** in Windows.



3) Click **File**, **New**, **Folder**  from the Main menu.

Enter the new folder name.

4) Enter **SOLIDWORKS-MODELS**.

Note: Select the Microsoft Windows commands from the Main menu, toolbar icons and with the right mouse button.

Create the first sub-folder.

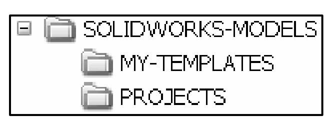
5) Double-click the **SOLIDWORKS-MODELS** folder.

6) Click **File**, **New**, **Folder** from the Main menu. A New Folder icon is displayed.

7) Enter **MY-TEMPLATES** for the folder name.

Create the second sub-folder.

8) Click the **SOLIDWORKS-MODELS** folder.



9) Click **File**, **New**, **Folder** from the Main menu.

10) Enter **PROJECTS** for the second sub-folder name.

Return to the SOLIDWORKS-MODELS folder.

11) Click the **SOLIDWORKS-MODELS** folder.

Utilize the MY-TEMPLATES folder and the PROJECTS folder throughout the text.

Start a SolidWorks session

The SolidWorks application is located in the Programs folder. By default, SolidWorks displays a Tip of the Day box. Read the Tip of the Day every day to obtain additional information on using SolidWorks.

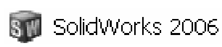
Open a new part. Select File, New from the Main pull down menu. There are two options for new documents: Novice and Advanced. Select the Advanced option. Select the Part document.

Activity: Start SolidWorks

Start a SolidWorks 2006 session.

12) Click **Start** on the Windows Taskbar.

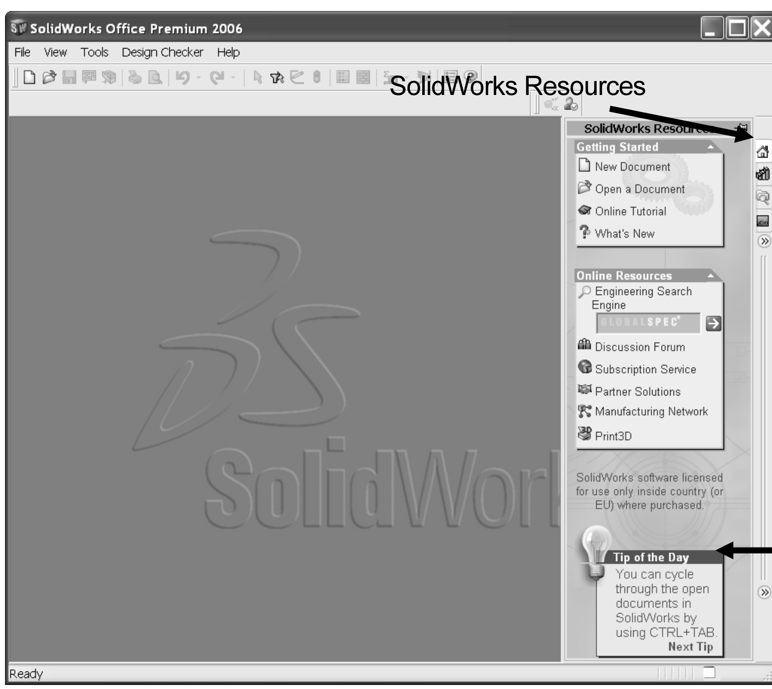
13) Click **All Programs**. Click the **SolidWorks 2006** folder.

14) Click **SolidWorks 2006** 

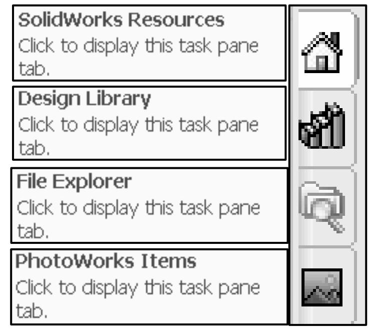
application. The SolidWorks program window opens. Note: Do not open a document.

Read the Tip of the Day dialog box.

15) Click the **Collapse arrow** in the Task Pane to close the Tip of the Day. Note: If you do not see this screen, click the SolidWorks **Resources** icon on the right side of the Graphics window.



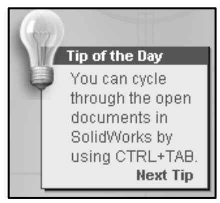
The SolidWorks 2006 Task Pane contains four options:



- SolidWorks Resources.
- Design Library.
- File Explorer.
- PhotoWorks Items.

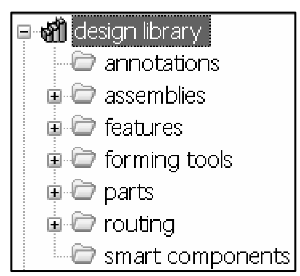
Utilize the left/right arrows to Expand or Collapse the Task Pane options.

SolidWorks Resources contains the Getting Started menu, the Online Resources menu and the Tip of the Day.



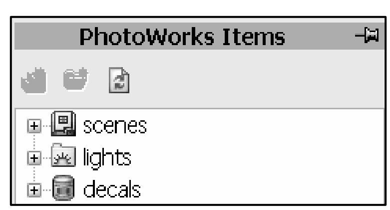
The Design Library includes entries for the design library, Toolbox and 3D ContentCentral. The Design Library contains the following folders: annotations, assemblies, features, forming tools parts routing and smart components.

To access the Design Library folders, click Add File Location , enter: C\Programs Files\SolidWorks\ data\design library. Click OK.

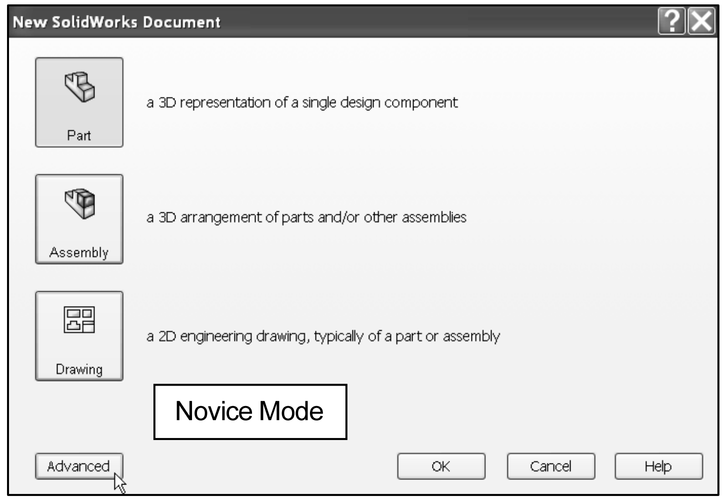


File Explorer duplicates Windows Explorer in functionality.

PhotoWorks Items create photo-realistic images of SolidWorks models. PhotoWorks provides many professional rendering effects.



There are two modes in the New SolidWorks Document dialog box: Novice and Advanced. The Novice option is the default option with three templates. The Advanced option contains access to additional templates.



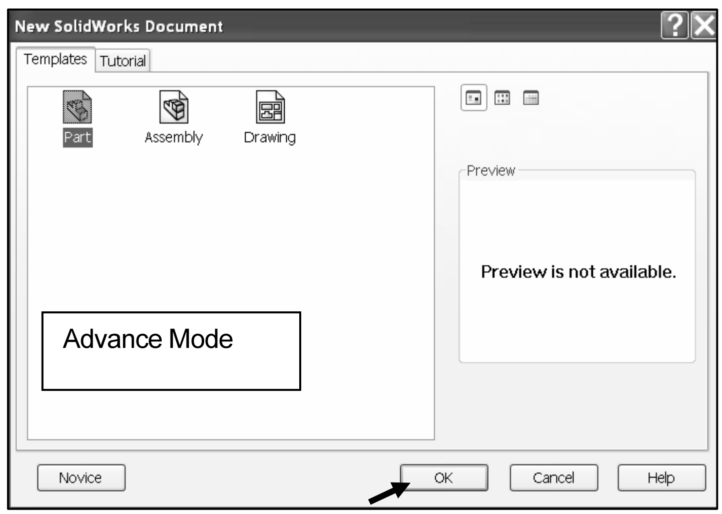
Create a new part.

16) Click **File**, **New** from the Main menu.

Select Advanced Mode.

17) Click the **Advanced** button to display the New SolidWorks Document dialog box in Advanced mode.

18) The Templates tab is the default tab. Part is the default template from the New SolidWorks Document dialog box. Click **OK**.



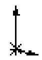
The Advanced mode remains selected for all new documents in the current SolidWorks session. When you exit SolidWorks, the Advanced mode setting is saved.

The default SolidWorks installation contains two tabs in the New SolidWorks Document dialog box, Templates and Tutorial. The Templates tab corresponds to the default SolidWorks templates. The Tutorial tab corresponds to the templates utilized in the Online Tutorials.

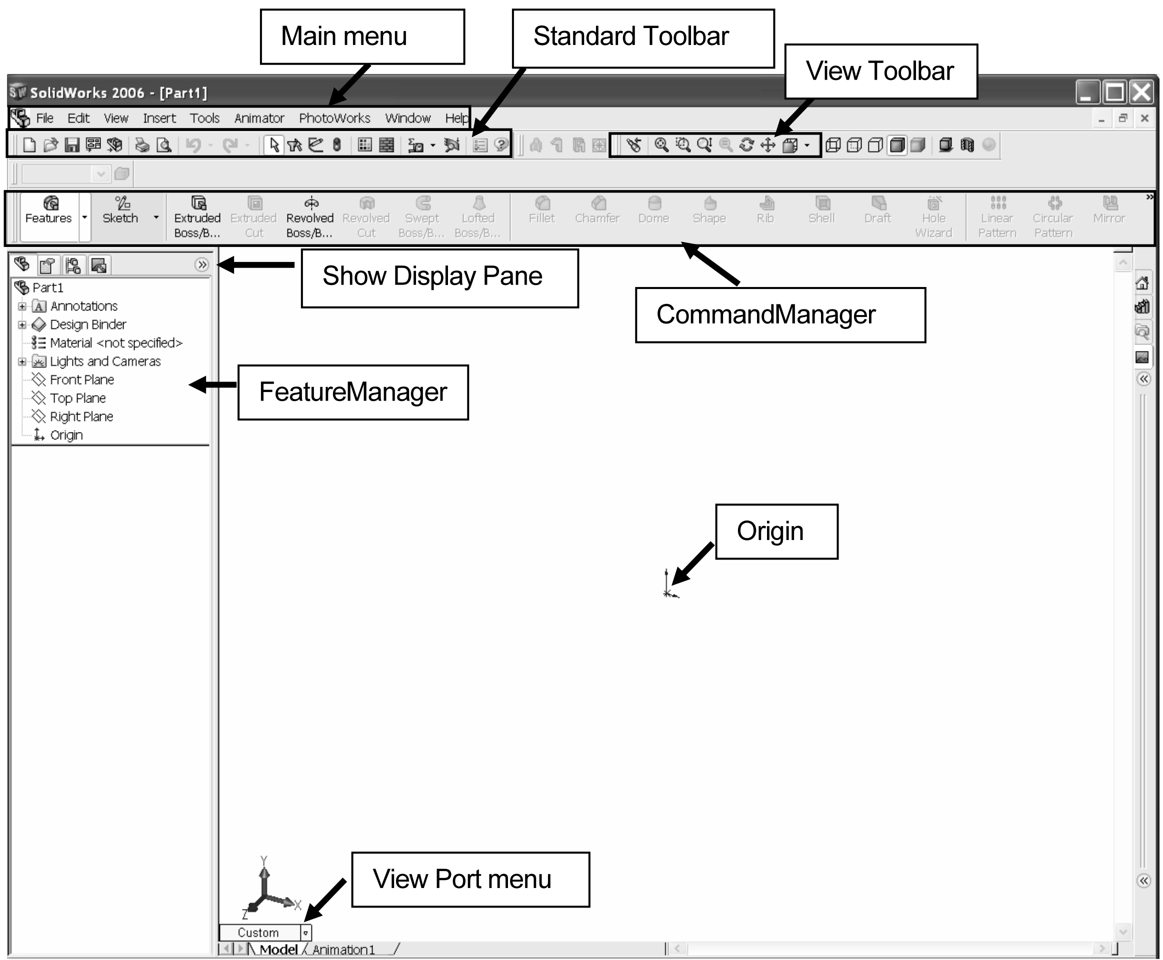
User Interface and CommandManager

The user interface combines the menus, toolbars and commands with graphic display and Microsoft Windows properties.

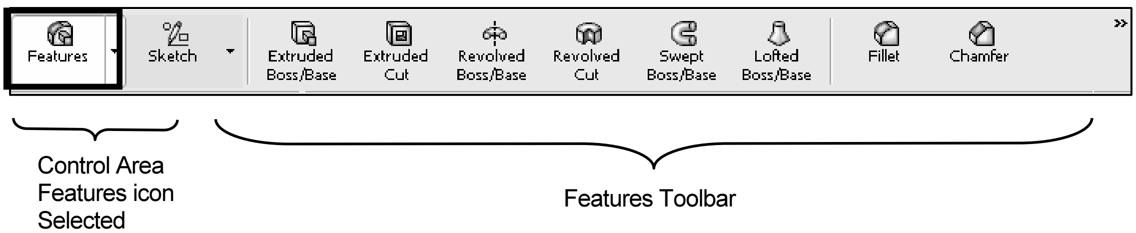
Part1 is displayed. Part1 is the new default part window name. The Main menu, Standard Toolbar, View Toolbar and CommandManager are displayed above the Graphics window.

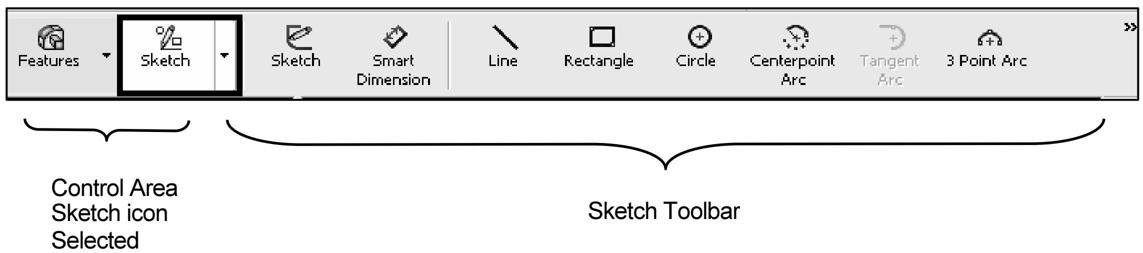
The part Origin  is displayed in blue in the center of the Graphics window. The Origin represents the intersection of the three default reference planes: Front Plane, Top Plane, and Right Plane. The positive X-axis is horizontal and points to the right of the Origin in the Front view. The positive Y-axis is vertical and point upward in the Front view.

The FeatureManager contains a list of features, reference geometry, and settings utilized in the part.



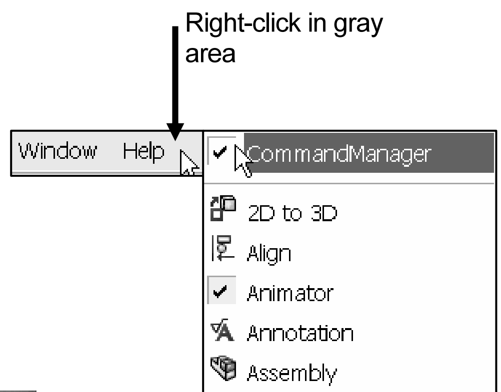
The CommandManager is divided into the Control Area and an expanded Toolbar. Select a Control Area icon to display the corresponding toolbar. The Features icon and Features Toolbar are selected by default in Part mode.



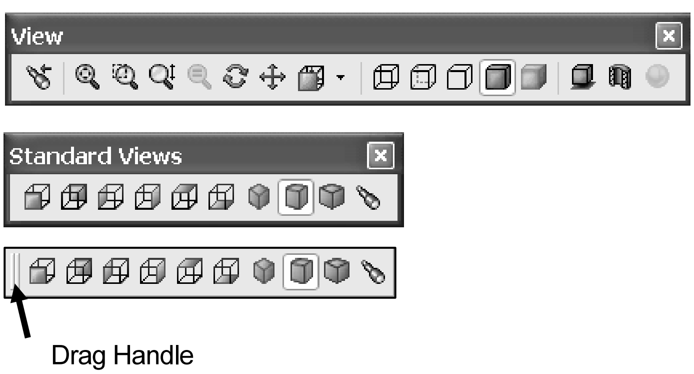


The CommandManager is utilized in this text.

To display the CommandManager, right-click in the gray area to the right of the Help menu. A complete list of toolbars is displayed. Check CommandManager if required.



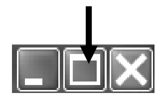
Select individual toolbars from the toolbar list to be displayed in the Graphics window. Reposition toolbars by moving their drag handle.



Activity: User Interface and CommandManager

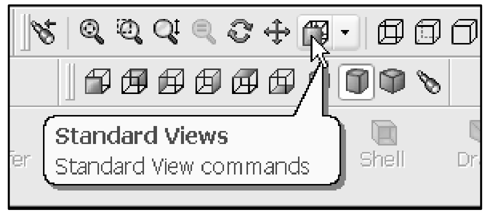
Maximize the Graphics window.

19) Click the **Maximize** button in the top right corner of the SolidWorks window.

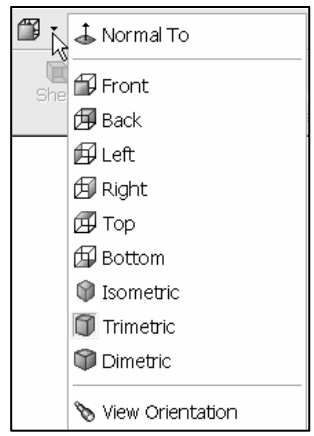


Display Tools and Toolbars.

20) Position the **mouse pointer** on the Standard Views icon. View the Large Tool tip.



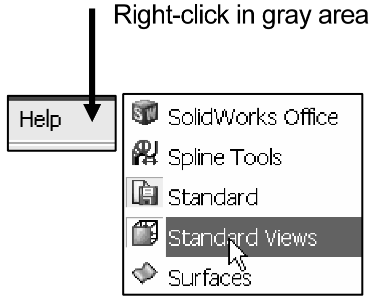
21) Click **Standard Views** from the View toolbar to list the default views. The small down arrow icon indicates additional information.



Display the Standard Views toolbar.

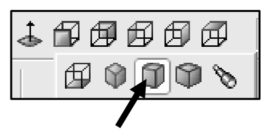
22) Right-click in the **gray area** of the Main menu to the right of Help. The displayed toolbars are listed.

Right-click in gray area



23) Activate **Standard Views**. The Standard Views toolbar is displayed in the SolidWorks window.

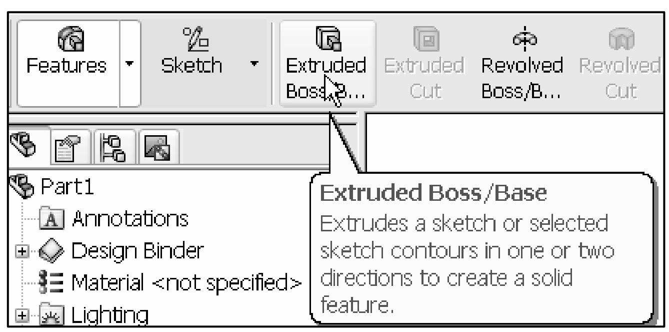
24) Position the **mouse pointer** on the Front view icon to display the Large Tool tip.

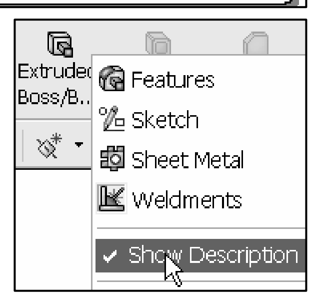


Display the Features tools.

25) Click **Features** Features from the Control Area of the Command Manager.

26) Position the mouse pointer over the **Extruded** Extruded **Boss/Base** Boss/B... tool in the Features toolbar. Do not select at this time. A Tool tip displays the Extruded Boss/Base feature name and a short description.

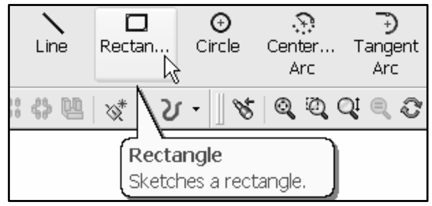




Display the Sketch tools.

27) Click **Sketch** Sketch from the Control Area of the Command Manager.

28) Position the mouse pointer over the **Rectangle** Rectan... tool in the Sketch toolbar. Do not select.



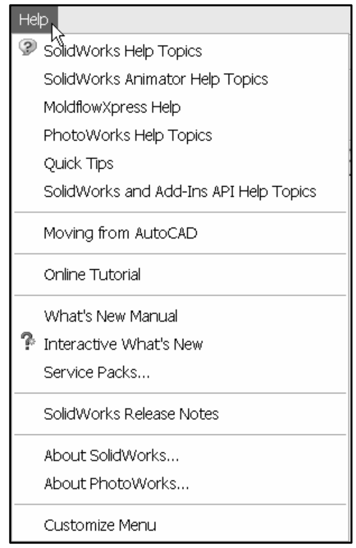
Display Help for a rectangle.

29) Click **Help** from the Main menu.

30) Click **SolidWorks Help Topics** SolidWorks Help Topics.

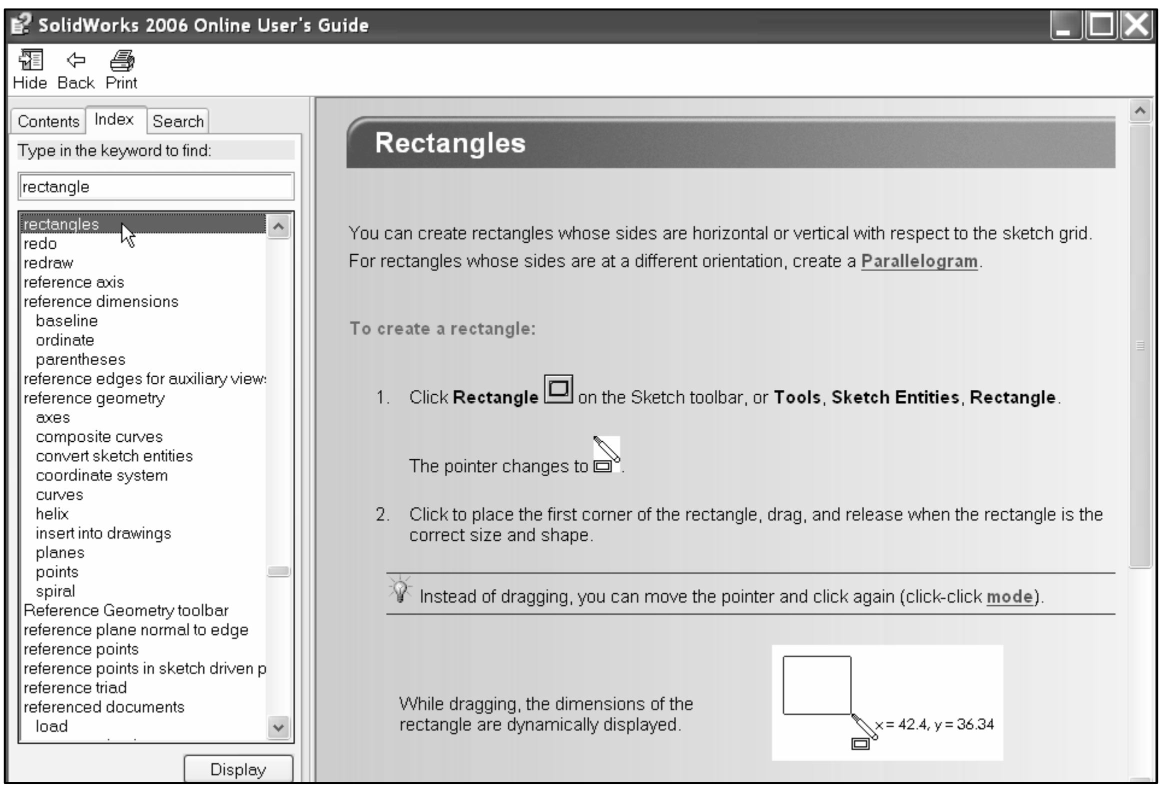
31) Click the **Index** tab.

32) Enter **rectangle**. The description appears in the right window. Review the provided information.



Close the Help window.

33) Click **Close**.



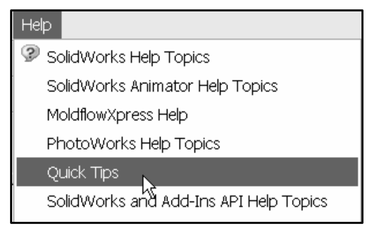
The Help menu contains the SolidWorks Online Tutorial, Introducing SolidWorks and Design Portfolio documents. These documents include additional information on using SolidWorks.

The Closer Look symbol indicates additional information about a tool or command is available from Help.

The Help option contains tools to assist the user.

The SolidWorks Help Topics contains:

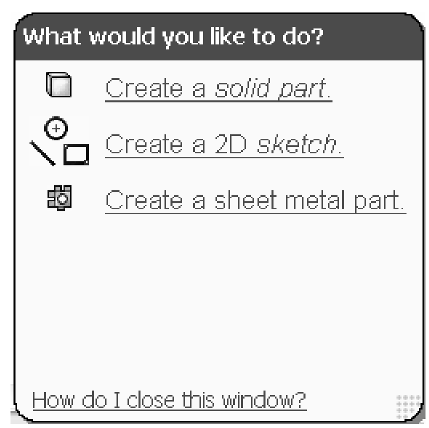
- Contents tab containing the SolidWorks User's Guide documents.
- Index tab containing additional information on key words.
- Search tab for finding information.



Quick Tips are a set of pop-up hints that appear as you create parts, assemblies and drawings. The messages are based on what tool or function is selected. The messages contain hyperlinks to associated areas in the Graphics window or additional files.

Activate Quick Tips by checking the Quick Tips entry in the Help Main menu.

The Pop-up Quick Tip windows contain a blue header with the question, "What would you like to do?" Read the options and select a statement for additional instructions.



Introducing SolidWorks and the Design Portfolio are great electronic documents for the new SolidWorks user.

The Online Tutorial contains step-by-step examples.

The <u>What's New Manual</u> contains descriptions of the new functionality in SolidWorks since the last major revision.

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:

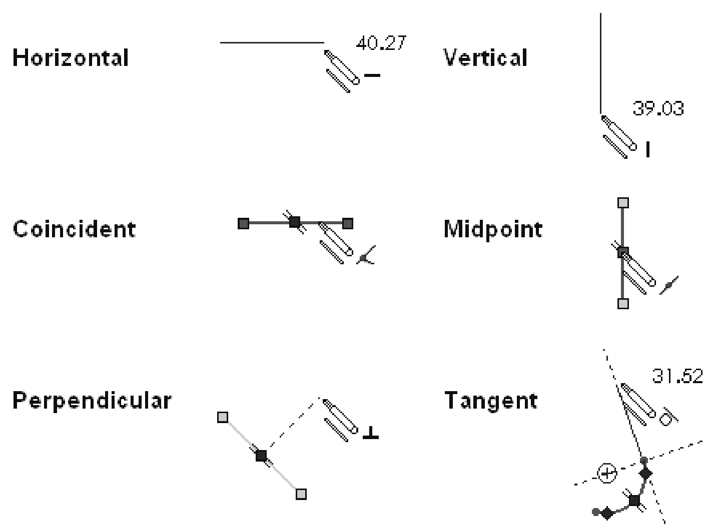
1. Sketch.
2. Feature.
3. Part.
4. Assembly.
5. Drawing.

1. Design Intent in the Sketch.

Build the design intent in the sketch as the profile is created.

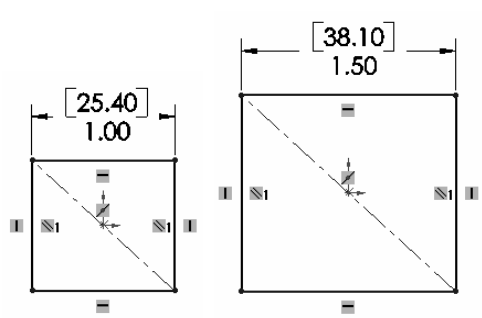
A profile is determined from the sketch tools, Example: rectangle, circle and arc.

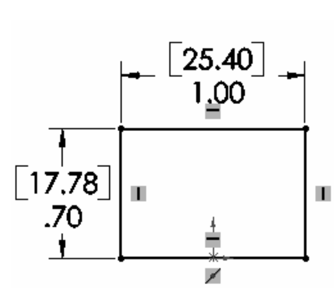
Build symmetry into the profile through a sketch centerline, mirror entity and position about the reference planes and Origin. Build design intent as you sketch with automatic relationships.



A rectangle contains horizontal, vertical and perpendicular automatic 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.

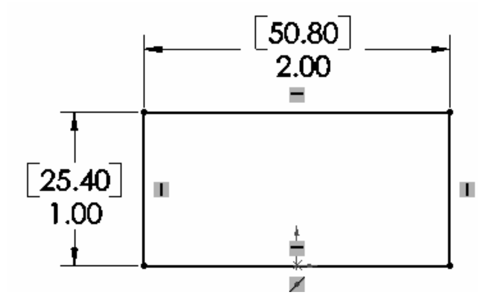




Sketch a rectangle with the Origin approximately in the center. Insert a centerline. Add a Midpoint relation. Add an Equal relation between the two perpendicular lines. Insert a dimension to define the width of the square.

Example B: Develop a rectangular profile.

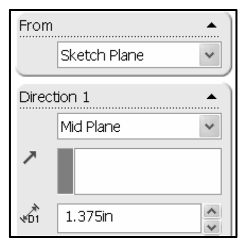
The bottom horizontal midpoint of the rectangular profile is located at the Origin. Sketch a rectangle. Add a Midpoint relation between the horizontal edge of the rectangle and the Origin.



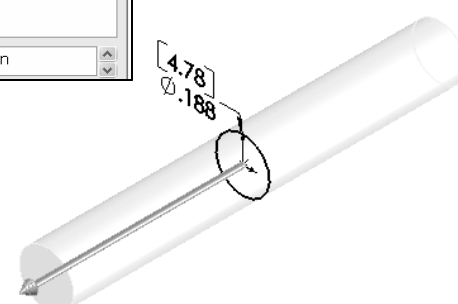
Insert two dimensions to define the width and height of the rectangle.

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.

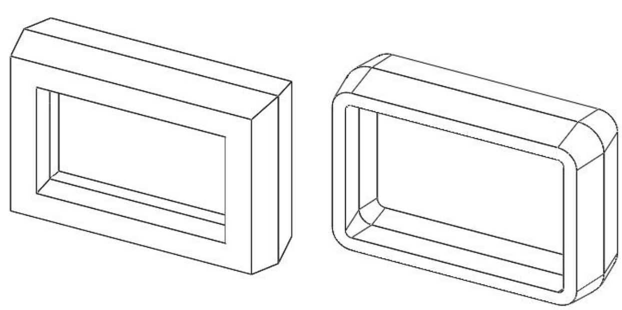


Utilize the Mid Plane Depth option. Change the depth and the feature remains symmetric about the Front Plane.

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.

For Design Intent of an Assembly or Drawing, see page I-12.

Part Template

The Part Template is the foundation for a SolidWorks part. Part1 was created with the default Part Template in the New dialog box.

Document Properties contain the default settings for the Part Template. The Document Properties include the dimensioning standard, units, dimension decimal display, grids, note font and line styles. There are hundreds of document properties. Modify the Document Properties: Dimensioning Standard, Unit and Decimal Places.

The Dimensioning Standard determines the display of dimension text, arrows, symbols and spacing. 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.

Document Properties are stored with the document. Apply the Document Properties to the Part Template. Create two Part Templates: PART-IN-ANSI and PART-MM-ISO. Save the Part Templates in the MY-TEMPLATE folder.

System Options are stored in the registry of your computer. The 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 SOLIDWORKS-MODELS\MY-TEMPLATES folder path name to the Document Templates File Locations list.

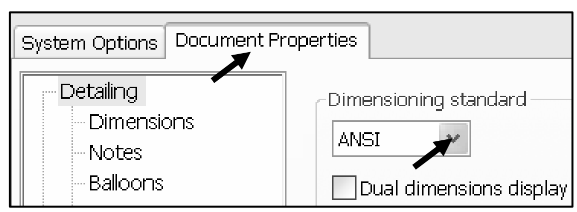
Activity: Part Template

Set the Dimensioning Standard to ANSI.

34) Click **Tools**, **Options** Options... from the Main menu.

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

36) Select **ANSI** from the Dimensioning standard box.



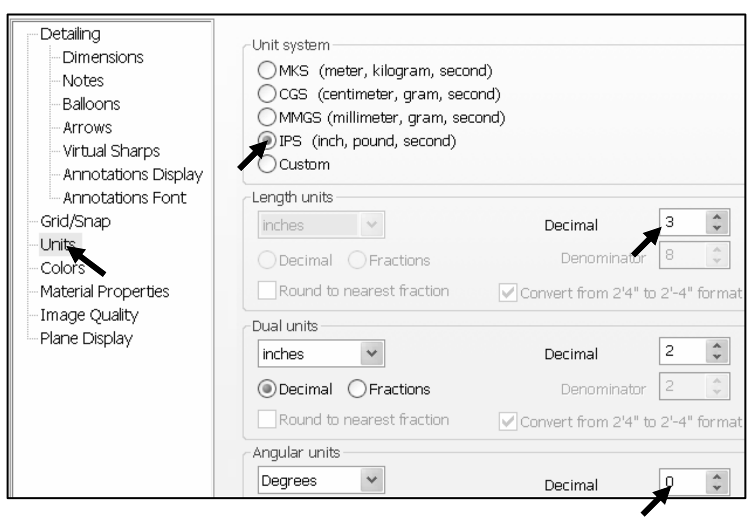
Set part units.

37) Click **Units**.

38) Select **IPS, (inch, pound, second)** for Unit system.

39) Select **3** for Length units Decimal places.

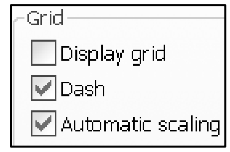
40) Select **0** for Angular units Decimal places.



Set the Grid/Snap option.

41) Click **Grid/Snap**.

42) Uncheck the **Display grid** option.



Return to the SolidWorks Graphics window.

43) Click **OK** from the Document Properties box.

Save the Part Template.

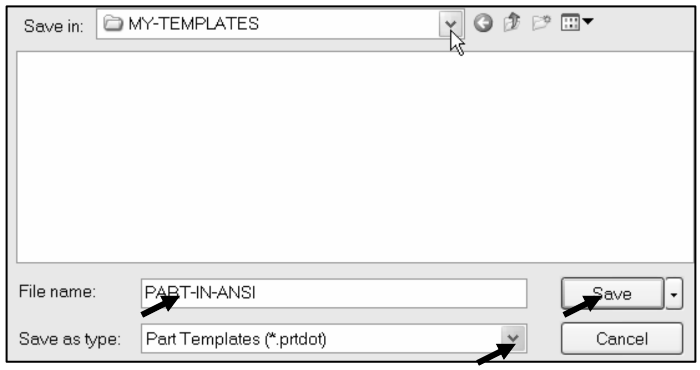
44) Click **File, Save As** from the Main menu.

45) Select **My Documents/ SOLIDWORKS-MODELS/ MY-TEMPLATES** from the Save in list.

46) Click **Part Templates (*.prtdot)** from the Save As type list box.

47) Enter **PART-IN-ANSI** in the File name text box.

48) Click **Save**.

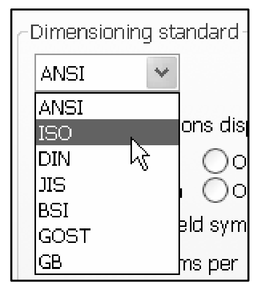


Set the Dimensioning Standard to ISO.

49) Click **Tools**, **Options** from the Main menu.

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

51) Select **ISO** from the Dimensioning standard list box.



Set the part units to millimeter.

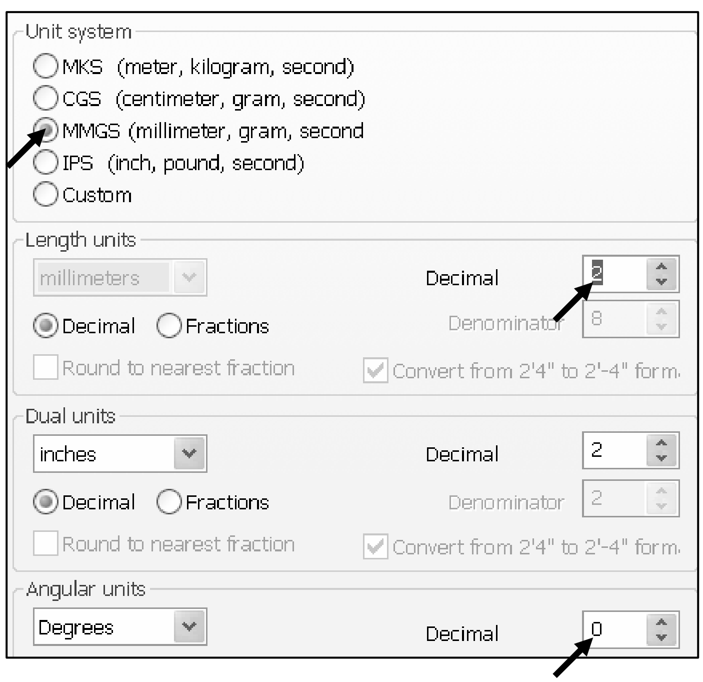
52) Click **Units**.

53) Select **MMGS, (millimeter, gram, second)** for Unit system.

54) Select **2** for Length units Decimal places.

55) Select **0** for Angular units Decimal places.

56) Click **OK** to set the document units.



Save the Part Template.

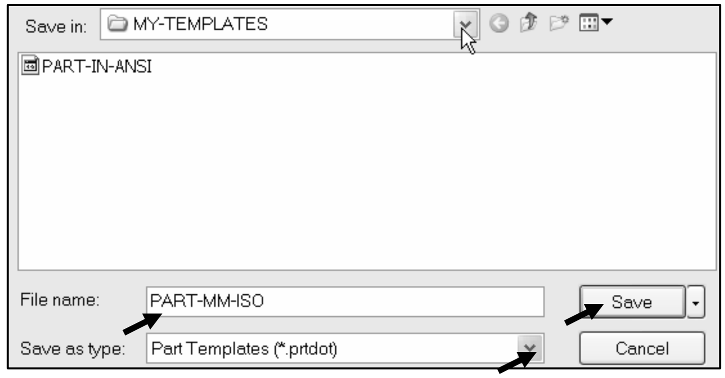
57) Click **File**, **Save As** from the Main menu.

58) Select **Part Templates (*.prtdot)** from the Save as type box.

59) Select **My Documents/ SOLIDWORKS-MODELS/ MY-TEMPLATES** from the Save in list.

60) Enter **PART-MM-ISO** in the File name text box.

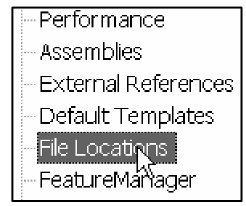
61) Click **Save**.



Set the System Options.

62) Click **Tools**, **Options** from the Main menu.

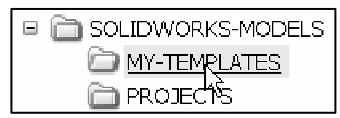
63) Click **File Locations** from the System Options tab.



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

65) Click the **Add** button.

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

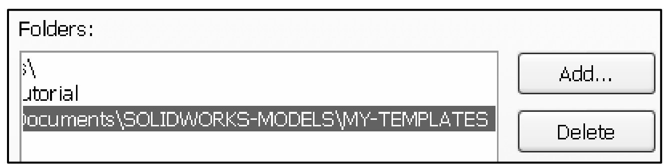


67) Click **OK** from Browse for Folder.

68) Click **OK** from System Options.

Close All documents.

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

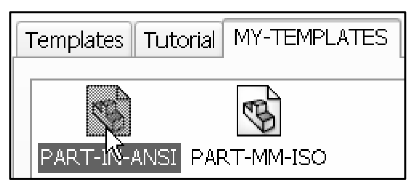


Display the MY-TEMPLATES folder and templates.

70) Click **File**, **New** from the Main menu.

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

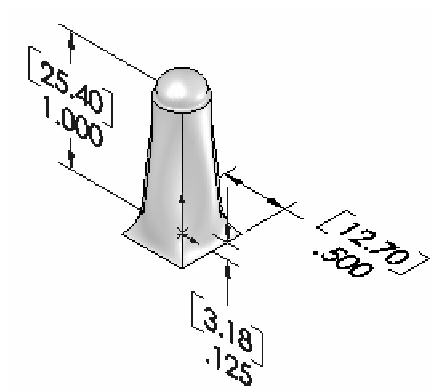
72) Click **Cancel**.



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.

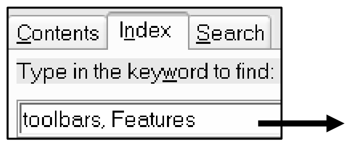
The MY-TEMPLATES Tab is visible when the folder contains SolidWorks Template documents. Create the PART-MM-ANSI template as an exercise.

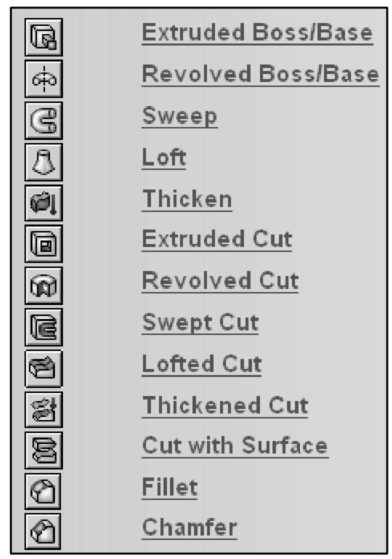
The PART-IN-ANSI Template contains Document Properties settings for the parts contained in the FLASHLIGHT assembly. Substitute the PART-MM-ISO or PART-MM-ANSI Template to create the identical parts in millimeters.



The primary units in this book are IPS (inch, pound, second). The optional secondary units are MMGS (millimeter, gram, second) and are indicated in brackets [ ]. Illustrations are provided in both inches and millimeters.

Input toolbars, click Features in SolidWorks Help Topics 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 Topics. Keywords: Options (detailing, units), templates, Files (locations), menus and toolbars (features, sketch).

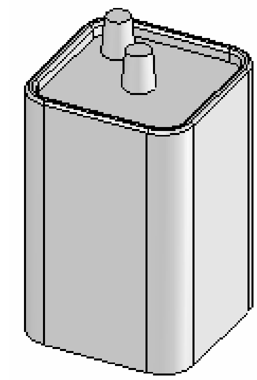
Review of the User Interface and Part Templates

The SolidWorks user interface consists of the following: Pull down menus, toolbars, Command Manager, FeatureManager and Graphics area. The CommandManager controls the display of the Sketch toolbar and Features toolbar.

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 Maim menu.

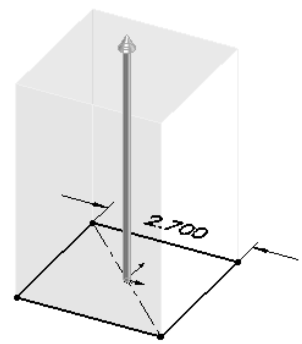
BATTERY Part



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 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.

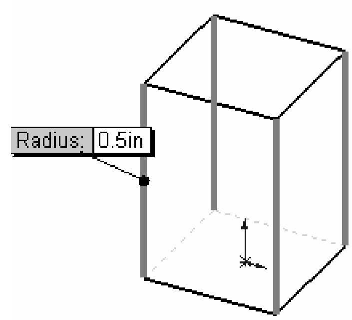


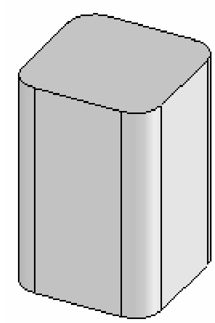
Utilize the Extruded Base feature. The Extrude Base features add material. The Base feature is the first feature of the part.

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

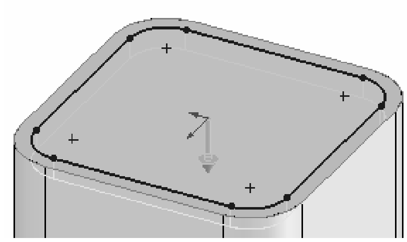
Extend the profile perpendicular (⊥) to the Top Plane.

Utilize the Fillet feature to round four vertical edges.

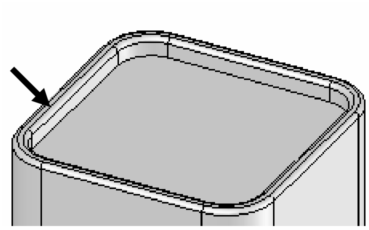




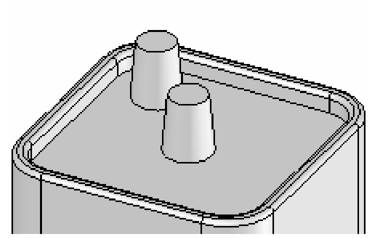
The Extruded Cut feature removes material from the top face. Utilize the top face for the Sketch plane. Utilize the Offset Entity Sketch tool to create the profile.



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 (Blind Depth).

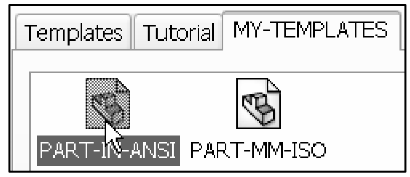
Create a new part named, BATTERY. Insert an Extruded Base feature. Extruded features require a Sketch Plane. The Sketch Plane determines the orientation of the Extruded Base feature. The Sketch Plane locates the Sketch Profile on any plane or face.

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 requires a Depth value to extrude the 2D Sketch Profile and 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

Create a new part.

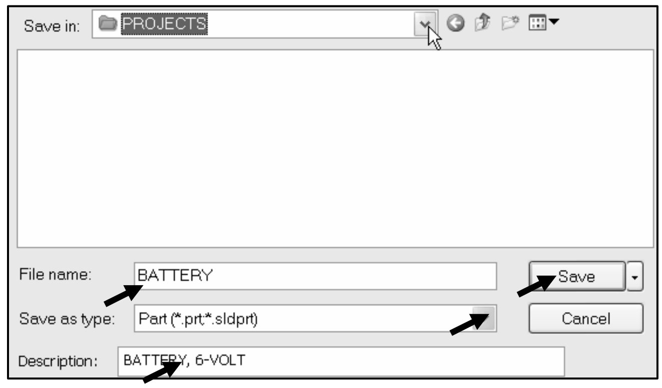


73) Click **File**, **New** from the Main menu.

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

75) Double-click **PART-IN-ANSI**, [**PART-MM-ISO**] from the Template dialog box.

Save the part.



76) Click **Save** .

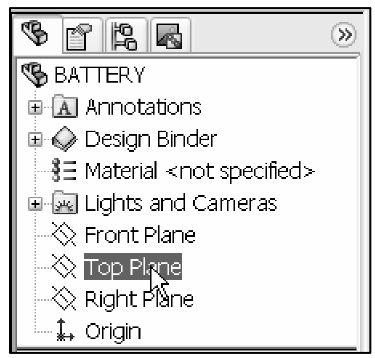
77) Select **SOLIDWORKS-MODELS\PROJECTS** for Save in folder.

78) Enter **BATTERY** for file name.

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

80) Click **Save**.

Select the Sketch plane.



81) Click **Top Plane** from the FeatureManager.

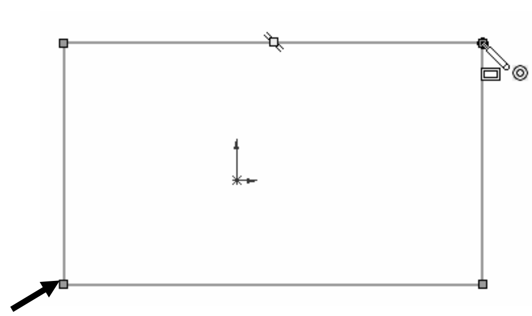
Sketch the profile.

82) Click **Sketch** from the CommandManager.

83) Click **Rectangle** from the Sketch toolbar.

84) Click the **first point** in the lower left quadrant.

85) Drag and click the **second point** in the upper right quadrant. The Origin is approximately in the middle of the Rectangle.



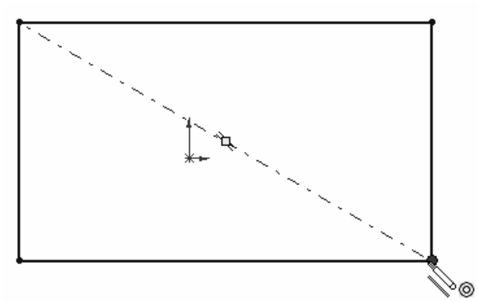
First Point

Sketch the Centerline.

86) Click **Centerline** Centerl... from the Sketch toolbar.

87) 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.

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



Add a Midpoint relation.

89) Click the **centerline**.

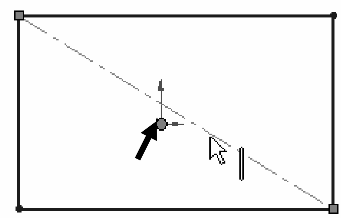
90) Hold the **Ctrl** key down.

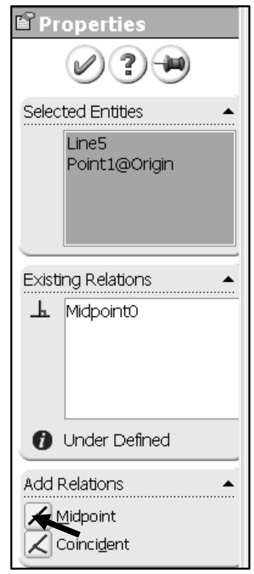
91) Click the **Origin**.

92) Release the **Ctrl** key.

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

94) Click **OK** from the Properites PropertyManager.





Note: Your Line# may be different than the line numbers displayed. 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.

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

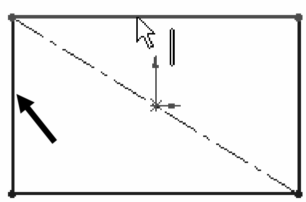
96) Hold the **Ctrl** key down.

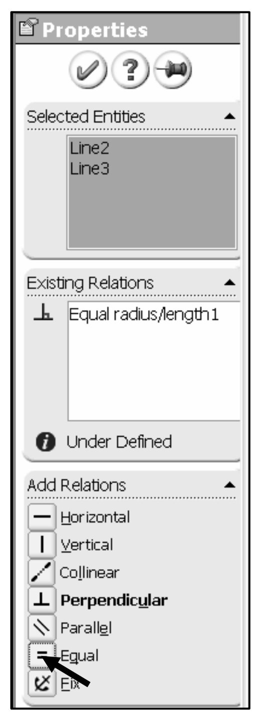
97) Click the **left vertical line**.

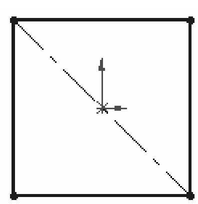
98) Release the **Ctrl** key.

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

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







Add a dimension.

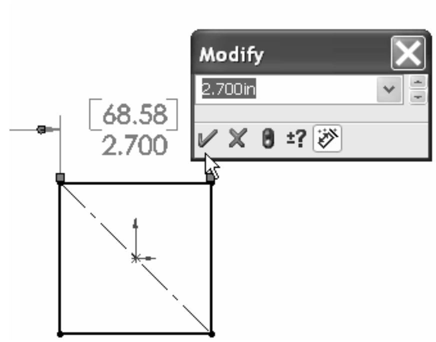
101) Click **Smart Dimension** Smart Dimens... from the Sketch toolbar.

102) Select the **top horizontal line**.

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

104) Enter **2.700** [**68.58**] for width.

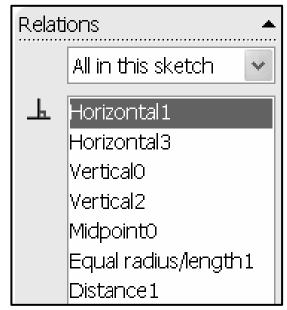
105) Click the **Green Check mark** in the Modify pop-up box. The black Sketch status is fully defined.



View the sketch relations.

106) Click **Display/Delete Relations** Display/... from the Sketch Relations toolbar. The Distance1 relation was created from the dimension.

107) Click **OK** from the Display/Delete Releaions PropertyManager.



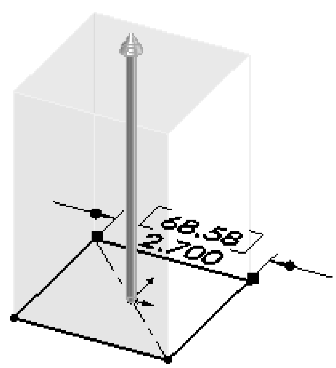
Activity: BATTERY Part-Extruded Base Feature

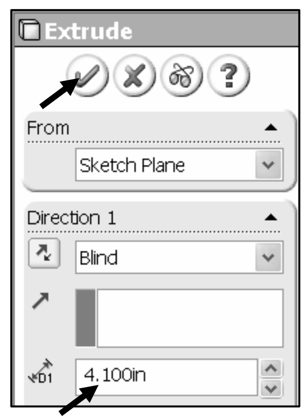
Insert the Extruded Base feature.

108) Click **Features** Features, **Extruded Boss/Base** Extruded Boss/B... from the Features toolbar. Blind is the default option.

109) Enter **4.100** [**104.14**] for Depth.

110) Click **OK** from the Extrude PropertyManager.





Fit the part to the Graphics window.

111) Press the **f** key.

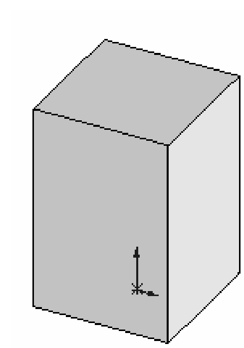
Rename the Extruded Base feature.

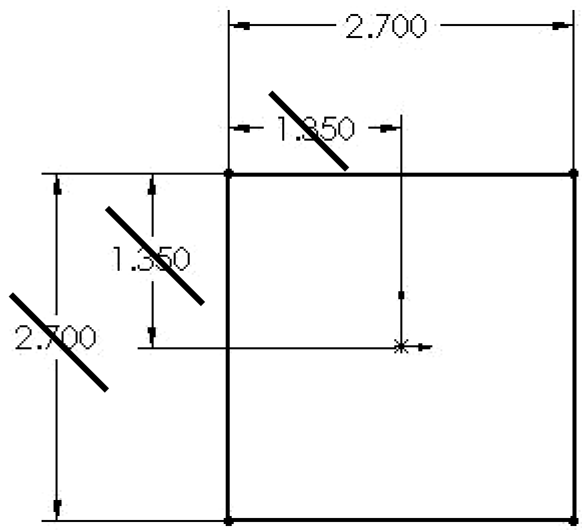
112) Click **Extrude1** in the FeatureManager.

113) Enter **Base Extrude**.

Save the BATTERY part.

114) 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 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.

Fillet Feature

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

An edge Fillet requires:

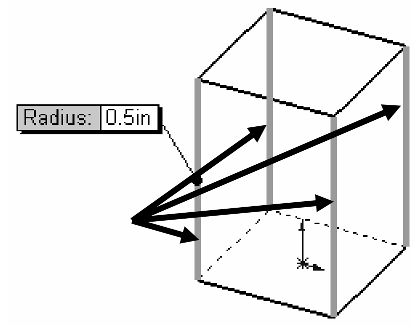
- Edge.
- Fillet Radius.

Select a vertical edge. Select the Fillet feature from the Features toolbar. Enter the Fillet radius. Add the other vertical edges to the Items to Fillet option. The order of selection for the Fillet feature is not predetermined.

Activity: BATTERY Part-Fillet Feature

Display the hidden edges.

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



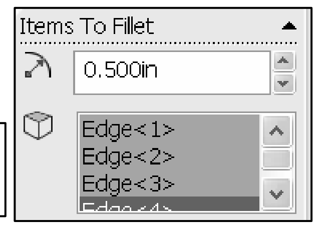
Insert the Fillet feature.

116) Click the **left vertical edge** of the Base Extrude feature.

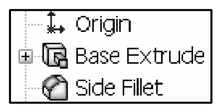
117) Click **Fillet** Fillet from the Features toolbar. Edge<1> is displayed in the Items To Fillet box.

118) Click the remaining **3 vertical edges**. Enter **.500** [**12.7**] for Radius.

119) Click **OK** from the Fillet PropertyManager.

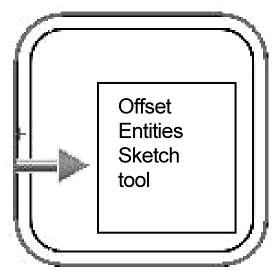


120) Rename **Fillet1** to **Side Fillet**. Click **Shaded With Edges**. Click **Save**.



Note: Select edges to produce the correct result.

Extruded Cut Feature



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

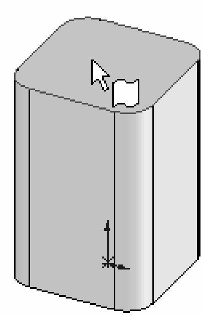
- Sketch Plane, (top face).
- Sketch Profile, (Offset Entities).
- End Condition, (Blind Depth).

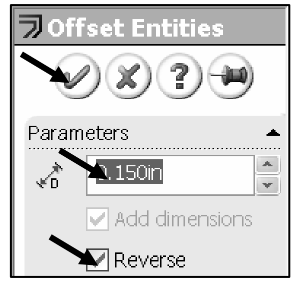
The Offset Entity Sketch tool uses existing geometry, extracts an edge or face and locates the geometry on the current sketch plane. Offset the existing Top face for the 2D sketch. Utilize the Blind Depth for End Condition.

Activity: Battery Part-Extruded Cut Feature-Edge

Select the Sketch plane.

121) Click the **Top face** of the BATTERY.





Create the Sketch.

122) Click **Sketch** Sketch .

Display the face.

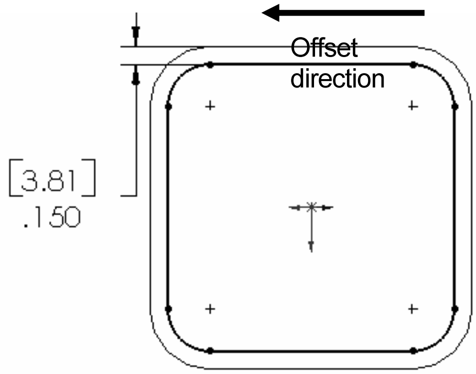
123) Click **Top view** from the Standards View toolbar.

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

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

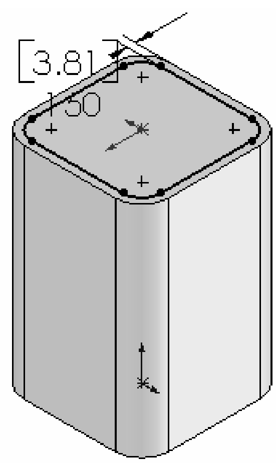
125) Enter **.150** [**3.81**] for the Offset Distance.

126) Click the **Reverse** check box. The new Offset yellow profile is displayed inside the original profile.



127) 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 under the ANSI standard.

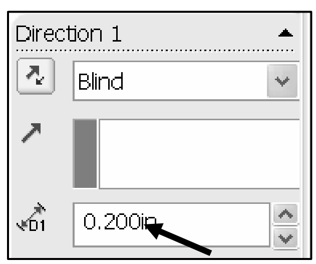


Display the profile.

128) Click **Isometric view** .

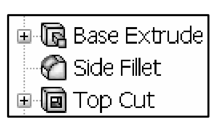
Insert an Extruded Cut feature.

129) Click **Features** Features , **Extruded Cut** Extruded Cut from the Features toolbar.

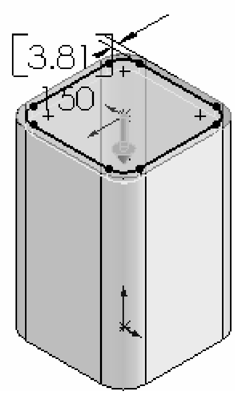


130) Enter **.200** [**5.08**] for Depth.

131) Click **OK** from the Cut-Extrude PropertyManager.



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

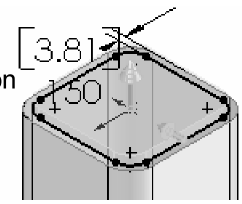


Save the BATTERY part.

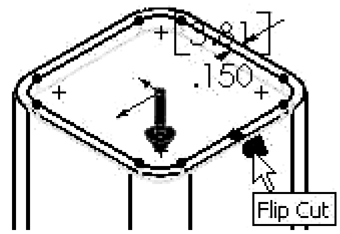
133) Click **Save** .

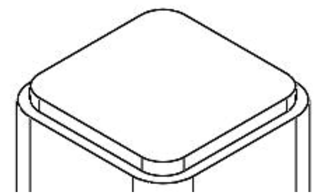
The Extruded Cut PropertyManager contains numerous options. The Reverse Direction option determines the direction of the Extruded Cut. The Extruded Cut is valid when the Direction arrow points into material to be removed.

Cut direction not valid.



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 occurs on the outside.





Extruded Cut with Flip side to cut option checked.

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

Insert the Fillet feature on the top face.

134) **Zoom in** on the Top face of the BATTERY.

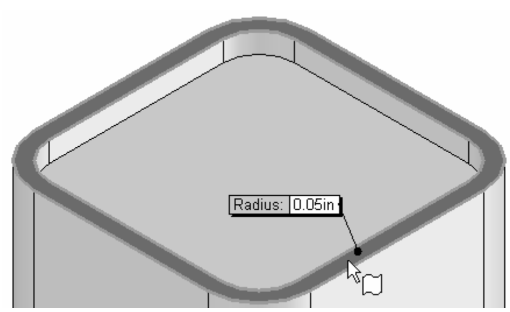
135) Click the **top thin face** of the BATTERY.

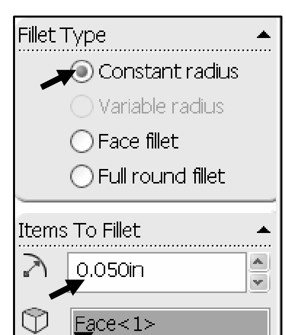
136) Click **Fillet** Fillet from the Features toolbar. Face<1> is displayed in the Items To Fillet box.

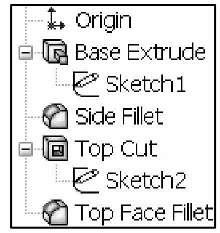
137) Click **Constant radius** for Fillet Type.

138) Enter **.050** [**1.27**] for Radius.

139) Click **OK** from the Fillet PropertyManager.







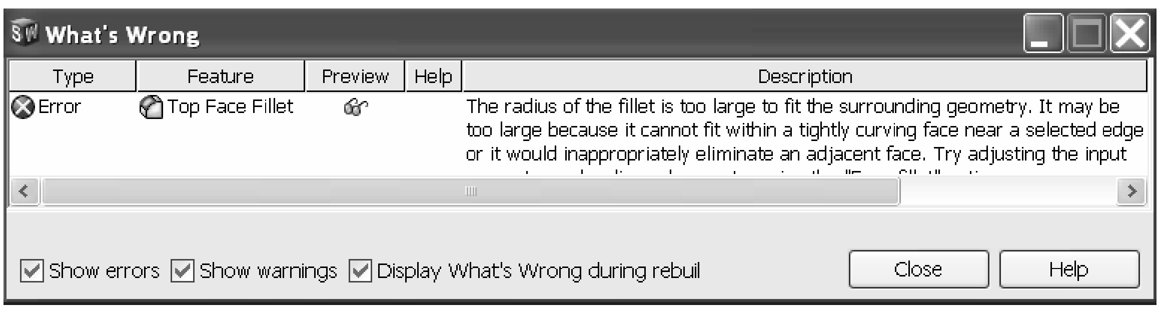
140) Rename **Fillet2** to **Top Face Fillet**.

141) Click **Save** .

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

Do not select a Fillet radius which is larger then the surrounding geometry. Example: The top edge face width is .150, [3.81]. The Fillet is created on both sides of the face. A common error is to enter a Fillet too large for the existing geometry. A minimum face width of .200, [5.08] is required for a Fillet radius of .100, [2.54].

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



Avoid the Fillet Rebuild error. Reduce the Fillet size or increase the face width.

Extruded Boss Feature

The Extruded Boss 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, [114.30mm]. Calculate the depth of the extrusion:

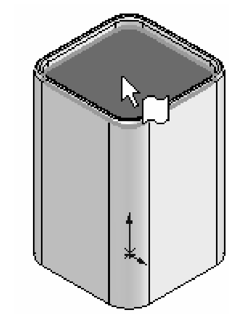
For inches: 4.500in – (4.100in Base-Extrude height – .200in Offset cut depth) = .600in. The depth of the extrusion is .600in.

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

Activity: BATTERY Part-Extruded Boss Feature

Select the Sketch plane.

142) Click the **top face** of the Top Cut feature.



Create the Sketch.

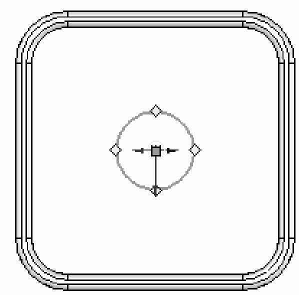
143) Click **Sketch** from the Sketch toolbar.

Display the Sketch Plane.

144) Click **Top view** from the Standards View toolbar.

Sketch the profile.

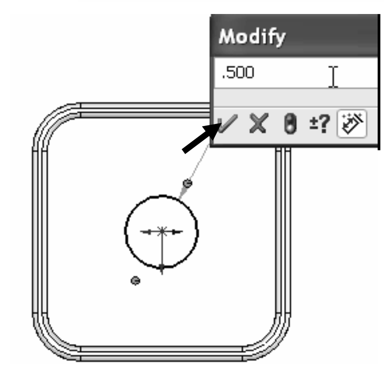
145) Click **Circle** from the Sketch toolbar.



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

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

Add dimensions.



148) Click **Smart Dimension**.

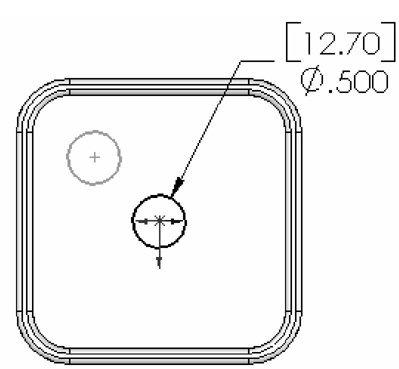
149) Click the **circumference** of the circle.

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

151) Enter **.500** [**12.7**].

152) Click the **Green Check mark**. The black Sketch is fully defined.

Copy the sketched circle.



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

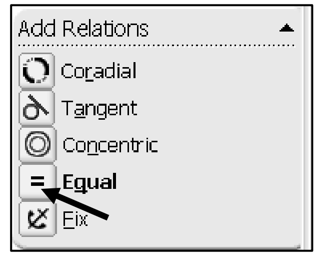
154) Hold the **Ctrl** key down. Click the **circumference** of the circle.

155) Drag the **circle** to the upper left quadrant.

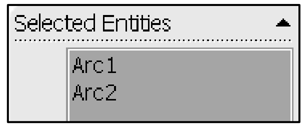
156) Release the **mouse button**. Release the **Ctrl** key. The second circle is selected and is displayed in green.

Add an Equal relation.

157) Hold the **Ctrl** key down. Click the **circumference of the first circle**. Both circles are selected. Release the **Ctrl** key.



158) Click **Equal** from the Add Relations text box

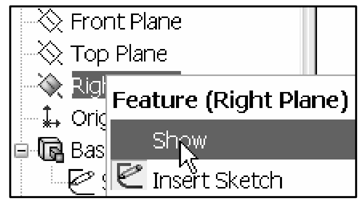


159) Click **OK** from the Properties PropertyManager.

Show the Right Plane for the dimension reference.

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

161) Click **Show**.



Add a dimension.

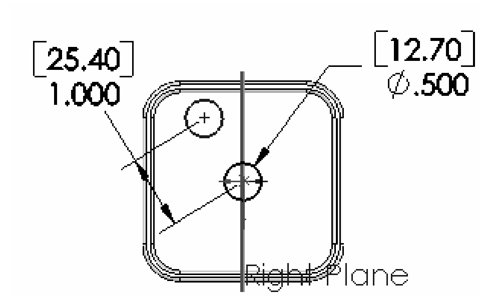
162) Click **Smart Dimension** Smart Dimens....

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

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

165) Enter **1.000** [**25.4**] for the aligned dimension.

166) Click the **Green Check mark**.

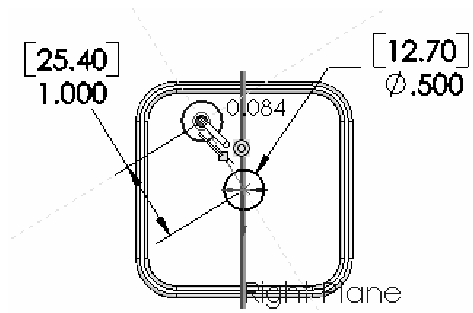


Add a Centerline.

167) Click **Centerline** Centerl... from the Sketch toolbar.

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

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



Add a dimension.

170) Click Smart **Dimension** Smart Dimens....

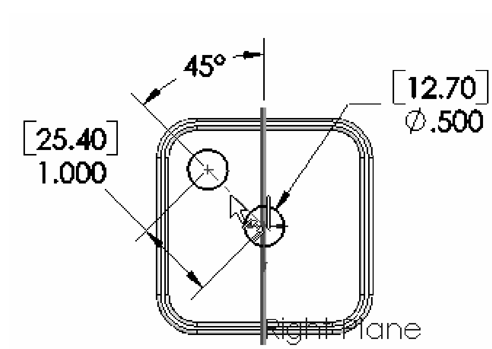
171) Click the **centerline** between the two circles.

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

173) Click a **position** between the centerline and the Right plane, off the profile.

174) Enter **45**.

175) Click the **Green Check mark**.

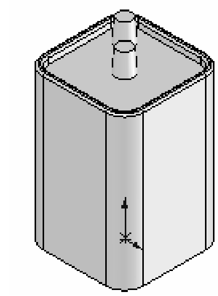


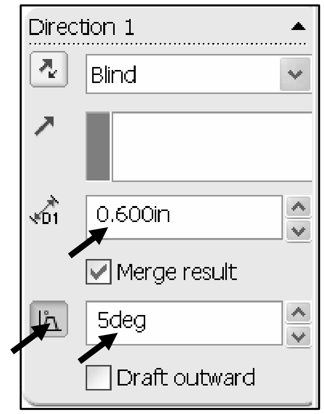
Create an angular dimension between three points or two lines. Sketch a centerline/construction line when an additional point or line is required.

Insert an Extruded Boss feature.

176) Click **Features**, **Extruded Boss/Base** from the Features toolbar. Blind is the default Type option.

177) Enter **.600** [**15.24**] for Depth. Click the **Draft ON/OFF** button. Enter **5** in the Draft Angle text box. Click **OK** from the Extrude PropertyManager.

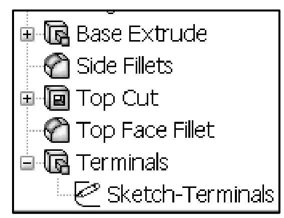




178) Click **Isometric view**. Right-click **Right Plane** from the FeatureManager. Click **Hide**.

Rename the Feature and Sketch.

179) Rename **Extrude2** to **Terminals**. Expand **Terminals**. Rename **Sketch3** to **Sketch-Terminals.**



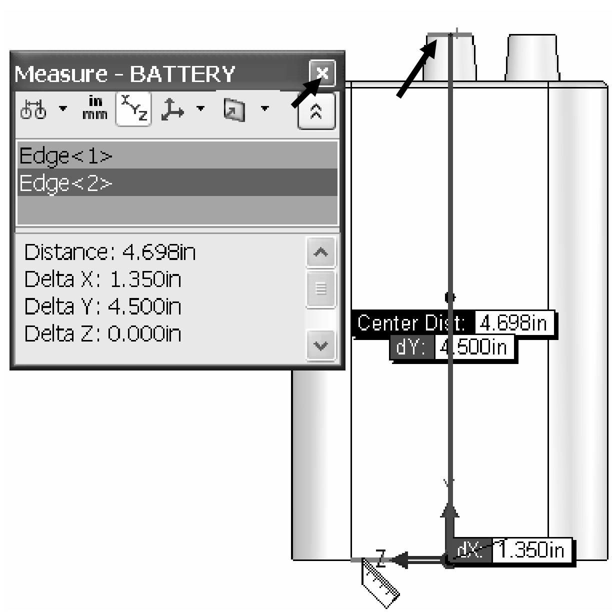
Each time you create a feature of the same feature type, the feature name is incremented by one. Example: Extrude1 is the first Extrude feature. Extrude2 is the second Extrude feature. If you delete a feature, rename a feature or exit a SolidWorks session, the feature numbers will vary from those illustrated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

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

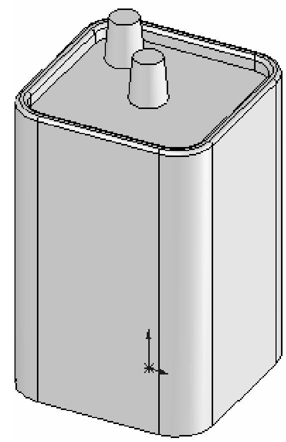
Measure the overall BATTERY height.

180) Click **Right view** from the Standard Views toolbar. Click **Tools**, **Measure** from the Main menu. Click the **top edge** of the BATTERY terminal. Click the **bottom edge** of the BATTERY. The overall height, Delta Y is 4.500 [114.3].

181) Click **Close** from the Measure – BATTERY box.



Right-click Clear Selections in the Selected items block to measure the distance between various edges or faces.



Hide all planes and display the Trimetric view.

182) Click **View**, uncheck **Planes** from the Main menu.

183) Click **Trimetric view** .

184) 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).

Review of the BATTERY Part

The BATTERY utilized an Extrude 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.

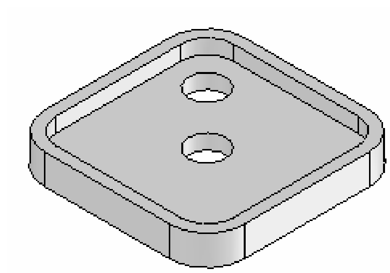
The Terminals were created with an Extruded Boss feature. You sketched a circular profile and utilized the Ctrl key to copy the sketched geometry. A centerline was required to locate the two holes with an angular dimension. The Draft Angle option tapered the Extruded Boss feature. All features were renamed.

BATTERYPLATE Part

The BATTERYPLATE is a critical FLASHLIGHT part. The BATTERYPLATE:

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

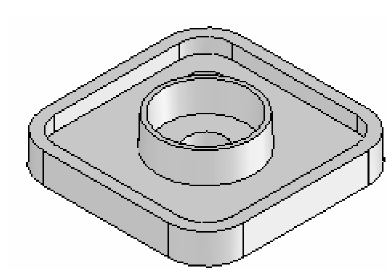
Create the BATTERYPLATE. Utilize features from the BATTERY to develop the BATTERYPLATE. The BATTERYPLATE is manufactured as an injection molded plastic part. Build Draft into the Extruded Base\Boss features.



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

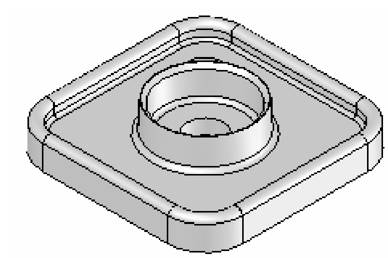
Modify the dimensions of the Base feature. Add a 3° draft angle.

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 feature together into a Folder. Perform a Draft Analysis on this part.

Group fillets together into a folder to locate 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 part. 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.

Right-click the Extruded Cut feature from the FeatureManager. Select the Edit Feature option. The Edit Feature option returns to the Extruded Cut PropertyManager. Modify the End Condition from Blind to Through All. Modify the depth dimension or the Extruded Base feature. Sketch dimensions are displayed in black. Feature dimensions are displayed in blue. Select Rebuild to update the part.

Activity: Save As option and Delete, Modify and Edit Feature

Create a new part.

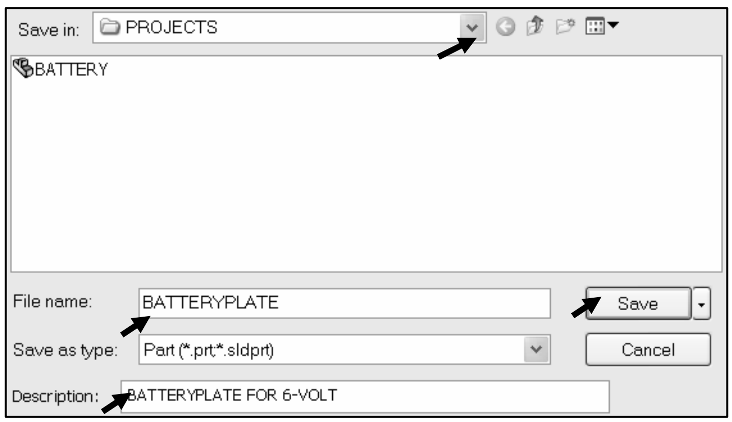
185) Click **File**, **Save As** from the Main menu.

186) Select **PROJECTS** for Save In Folder.

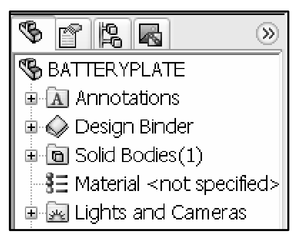
187) Enter **BATTERYPLATE** for File name.

188) Enter **BATTERYPLATE FOR 6-VOLT** for Description.

189) Click **Save**.



The BATTERYPLATE part icon is displayed at the top of the FeatureManager. The BATTERY part is closed.

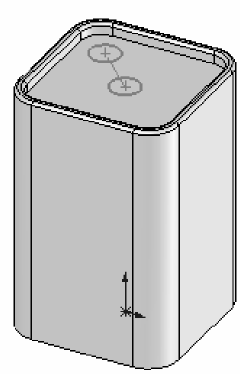


Delete the BATTERY Terminals.

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

191) Click **Delete** Delete... .

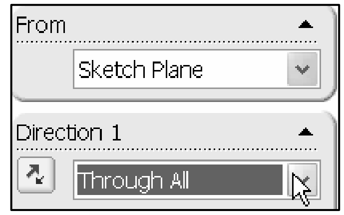
192) Click **Yes** from the Confirm Delete box. Do not delete the two-circle sketch, Sketch-TERMINALS.



Activity: BATTERYPLATE Part-Extruded Cut Feature

Create an Extruded Cut feature from the Sketch–Terminals.

193) Click **Sketch-Terminals** from the FeatureManager.



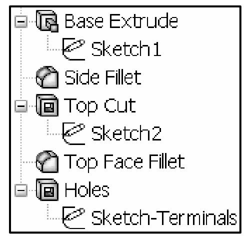
194) Click **Features** Features, **Extruded Cut** Extruded Cut from the Features Toolbar.

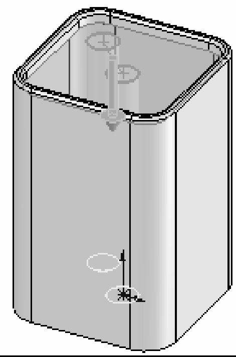
195) Select **Through All** for End Condition.

196) Click **OK** from the Cut-Extrude PropertyManager.

197) Rename **Cut-Extrude1** to **Holes**.

198) Click **Save**.





Edit the Base Extrude feature.

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

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

Modify the overall Depth and Draft.

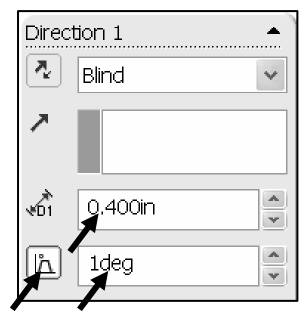
201) Click **4.100** [**104.14**].

202) Enter **.400** [**10.16**] for new Depth.

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

204) Enter **1** in the Draft Angle box.

205) Click **OK** from the Base Extrude PropertyManager.

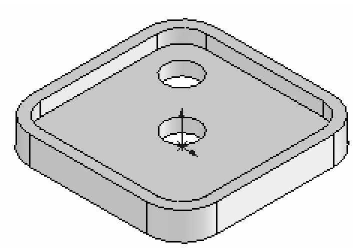


Fit the model to the Graphics window.

206) Press the **f** key.

Save the BATTERYPLATE part.

207) Click **Save**.



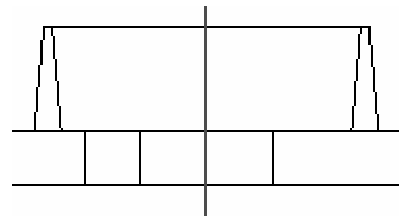
Select the Also Delete Absorbed Feature check box to delete both the feature and the sketch at the same time.

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 Extrude Boss options.

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

Draft Angle displayed at 5°

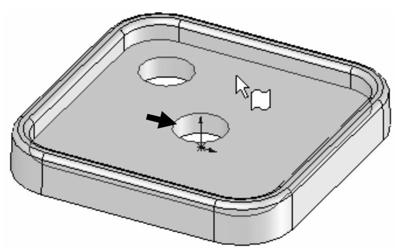


Plastic parts require a draft angle. A rule of thumb; 1° to 5° is the draft angle. The draft angle is created in the direction of pull from the mold. This is defined by geometry, material selection, mold production and cosmetics. Always verify the draft with the mold designer and manufacturer.

Activity: BATTERYPLATE Part-Offset Entities

Select the Sketch plane.

208) Click the **top face** of the BATTERYPLATE part.



Create the Sketch.

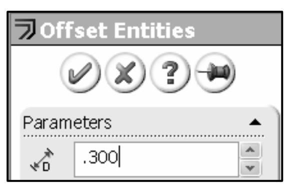
209) Click **Sketch** Sketch .

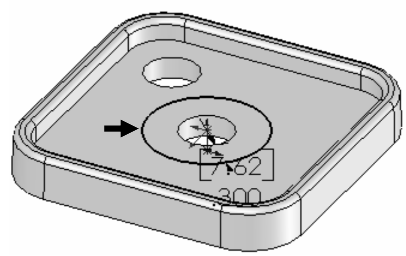
210) Click the **top circular edge** of the center Hole.

211) Click **Offset Entities** Offset .

212) Enter **.300** [**7.62**] for Offset Distance.

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





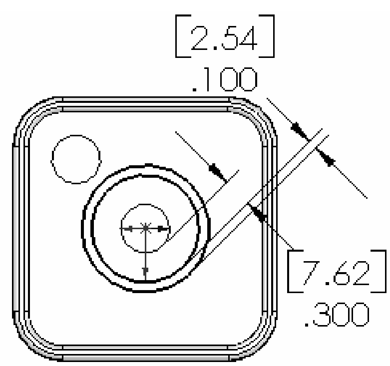
Create the second offset circle.

214) Click the **offset circle**.

215) Click **Top view** .

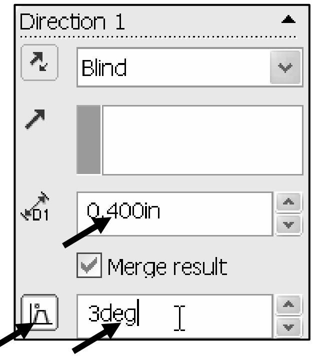
216) Click **Offset Entities** Offset .

217) Enter **.100** [**2.54**] for the Offset Distance. Click **OK** from the Offset Entities PropertyManager.



Activity: BATTERYPLATE Part-Extruded Boss Feature

Insert the Extruded Boss feature.



218) Click **Features** Features, **Extruded Boss/Base** Extruded Boss/B... .

219) Enter **.400** [**10.16**] for Depth.

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

221) Enter **3** in the Angle text box.

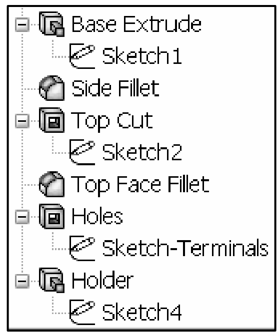
222) Click **OK** from the Extrude PropertyManager.

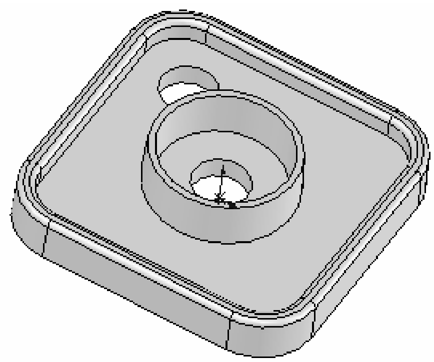
223) Rename **Extrude3** to **Holder**.

224) Click **Isometric view** .

Save the BATTERYPLATE part.

225) 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. Use two different techniques to create the Fillets.

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 were 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.

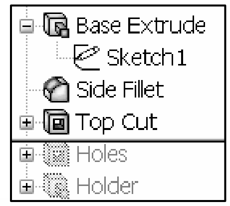
Activity: BATTERYPLATE Part-Fillet Features: Full Round, Multiple Radius Options

Delete the Top Edge Fillet.

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

227) Click **Delete** Delete... . Click **Yes**.

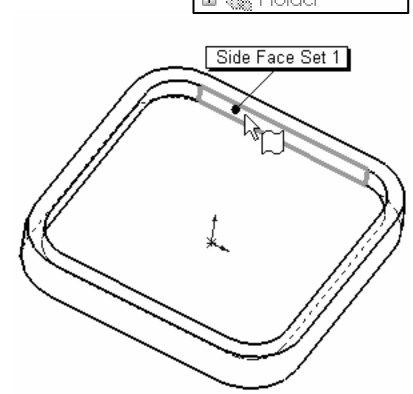
228) Drag the **Rollback** bar below Top Cut in the FeatureManager.



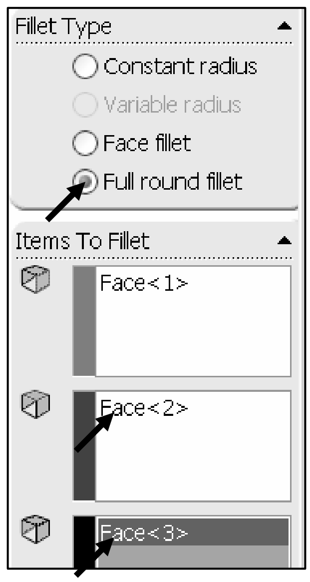
Insert the Full Round Fillet feature.

229) Click **Hidden Lines Visible** .

230) Click **Features** Features , **Fillet** Fillet from the Features toolbar.



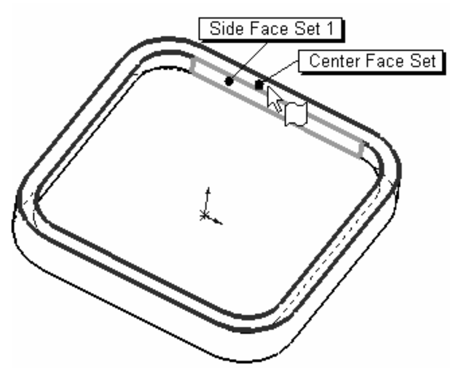
231) Click **Full round fillet** in the Fillet Type box.



232) Click the **inside Top Cut face** for Side Face Set 1.

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

234) Click the **top face** for Center Face Set.

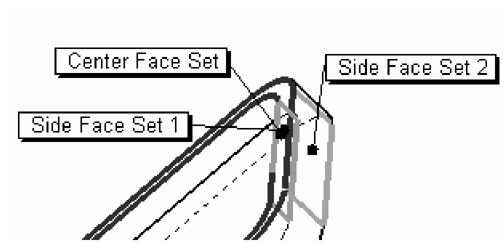


Rotate the part.

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

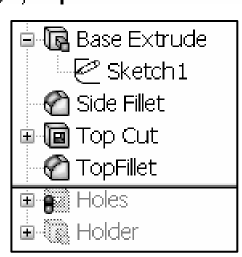
236) Click **inside** the Side Face Set 2 box.

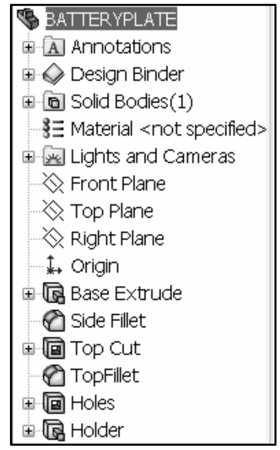
237) Click the **outside Base Extrude face** for Side Face Set 2.



238) Click **OK** from the Fillet PropertyManager.

239) Rename **Fillet3** to **TopFillet**.





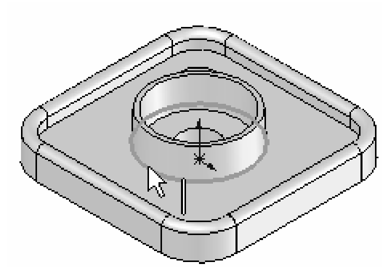
Save the BATTERYPLATE.

240) Click **Isometric view** .

241) Click **Shaded With Edges** . Drag the **Rollback** bar back below Holder in the FeatureManager.

242) Click **Save** .

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



Insert a Multiple Radius Fillet feature.

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

244) Click **Features** Features , **Fillet** Fillet from the Features toolbar.

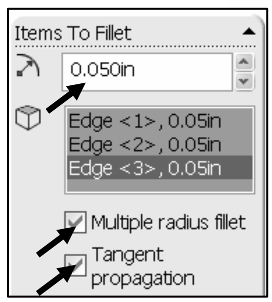
245) Enter **.050** [**1.27**] for Radius.

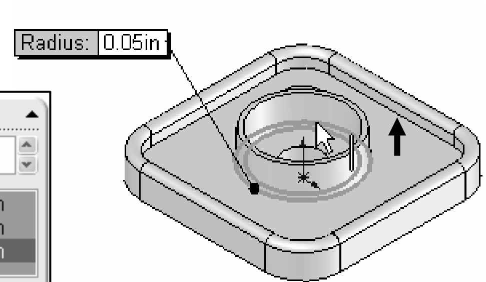
246) Click the **bottom inside circular edge** of the Holder.

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

248) Check **Tangent Propagation**.

249) Check **Multiple radius fillet**.





Modify the Fillet values.

250) Click the **Radius** box Radius: 0.05in for the Holder outside edge.

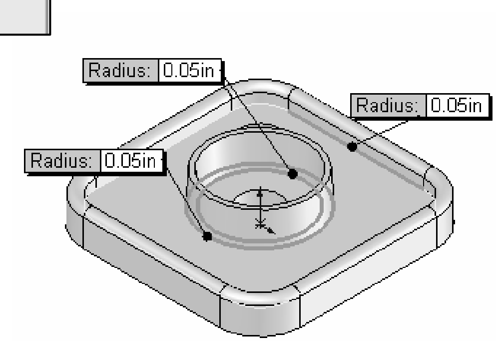
251) Enter **0.060** [**1.52**].

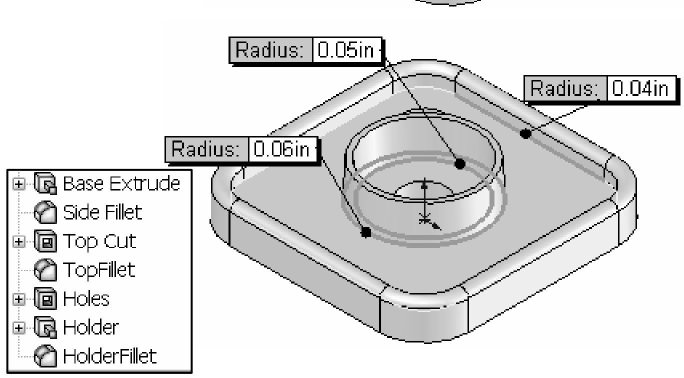
252) Click the **Radius** box for the Top Cut inside edge.

253) Enter **0.040** [**1.02**].

254) Click **OK** from the Fillet PropertyManager.

255) Rename **Fillet4** to **HolderFillet**.

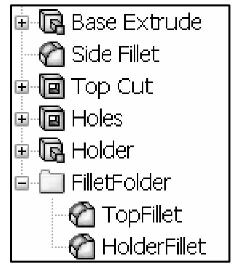




Group the Fillets into a Folder.

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

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



258) Click **HolderFillet** from the FeatureManager. Hold the **Ctrl** key down.

259) Click **Top Fillet** from the FeatureManager.

260) Right-click **Add to New Folder**. Release the **Ctrl** key.

261) Rename **Folder1** to **FilletFolder**.

Save the BATTERYPLATE.

262) Click **Save** .

Chamfer Feature

A Chamfer feature bevels an edge or a face. There are three options for the Chamfer feature:

- Angle – distance.
- Distance – distance.
- Vertex - point.

The Chamfer feature for the Holder requires:

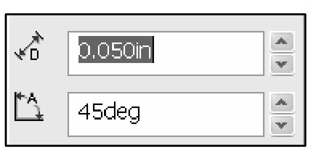
- Edge or face.
- Angle and distance.

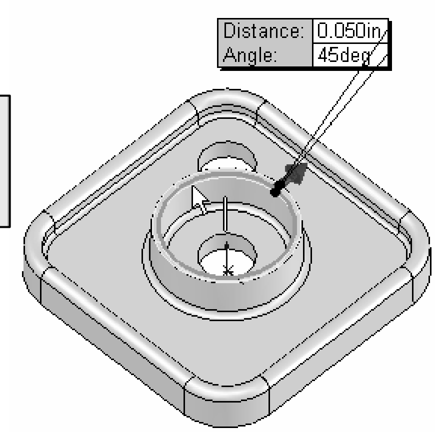
Activity: BATTERYPLATE Part-Chamfer Feature

Insert a Chamfer feature.

263) Click the **inside circular edge** of the Holder.

264) Click **Chamfer** Chamfer from the Features toolbar.





265) Enter **.050** [**1.27**] for Distance.

266) Enter **45** for Angle.

267) Click **OK** from the Chamfer PropertyManager.

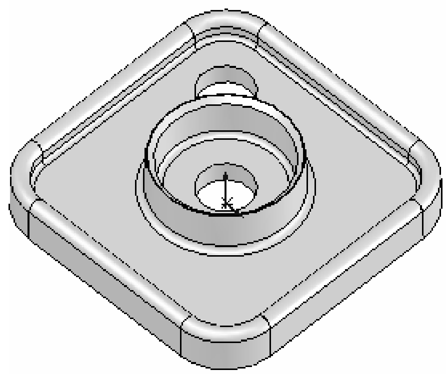
268) Click **Isometric view** .

Save the BATTERYPLATE.

269) Click **Save** .

Exit SolidWorks.

270) Click **File**, **Exit** from the Main menu.

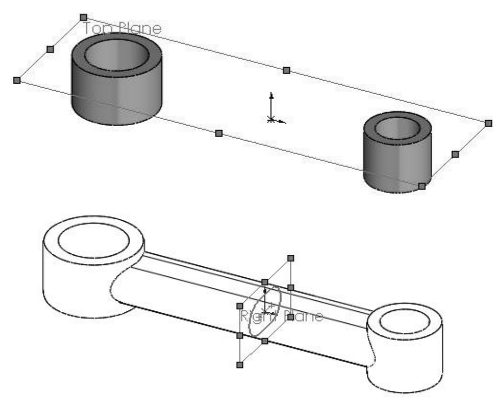


Multi-body Parts and Extruded Boss Feature

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 option.

The BATTERY consisted of a solid body with one sketched profile. The BATTERY is a single body part.



Multi-body part
Wrench

Additional information on Save, Extrude Boss/Base, Extrude 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), Multi-body (extrude, modeling techniques).

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.

Project Summary

SolidWorks is a 3D design software application utilized to create parts, assemblies and drawings. You are designing a FLASHLIGHT assembly that is cost effective, serviceable and flexible for future design revisions. The FLASHLIGHT assembly consists of various parts. The BATTERY and BATTERYPLATE parts were modeled in this project.

The SolidWorks Windows based user interface is divided into: Pull down menus, toolbars, Pop-up menus, the CommandManager, FeatureManager Status bar and the Graphics window.

Folders organized your models and templates. The Part Template is the foundation for all parts in the FLASHLIGHT assembly. You created the PART-IN-ANSI and PART-MM-ISO Templates.

Project 1 concentrated on the Extruded Base feature. The Extruded Base feature required a Sketch Plane, Sketch Profile and End Condition (Depth). The BATTERY and BATTERYPLATE parts incorporated an Extruded Base feature:

You addressed four major features in this project: Extruded Boss/Base, Extruded Cut, Fillet and Chamfer.

You addressed the following Sketch tools in this project: Sketch, Smart Dimension, Line, Rectangle, Circle, Tangent Arc and Centerline.

You addressed additional Sketch tools that utilized existing geometry: Add Relations, Display/Delete Relations, Mirror Entities, Convert Entities and Offset Entities.

Geometric Relations were utilized to build symmetry into the sketches. Practice these concepts with the project exercises.

Project Terminology

Assembly: An assembly is a document in which parts, features and other assemblies (sub-assemblies) are put together. The filename extension for a SolidWorks assembly file name is .SLDASM. The FLASHLIGHT is an assembly. The BATTERY is a part in the FLASHLIGHT assembly.

Chamfer: A feature that bevels sharp edges or faces by a specified distance and angle or by two specified distances.

Convert Entities: A sketch tool that extracts sketch geometry to the current sketch plane.

Cursor Feedback: Feedback is provided by a symbol attached to the cursor arrow indicating your selection.

Dimension: A value indicating the size of feature geometry.

Dimensioning Standard: A set of drawing and detailing options developed by national and international organizations. A few key dimensioning standard options are: ANSI, ISO, DIN, JIS, BSI, GOST and GB.

Draft angle: A draft angle is the degree of taper applied to a face. Draft angles are usually applied to molds or castings.

Drawing: A document containing a 2D representation of a 3D part or assembly. The filename extension for a SolidWorks drawing file name is .SLDDRW.

Edit Feature: A tool utilized to modify existing feature parameters. Right-click the feature in the FeatureManager. Click Edit Feature.

Edit Sketch: A tool utilized to modify existing sketch geometry. Right-click the feature in the FeatureManager. Click Edit Sketch.

Extruded Boss/Base: A feature that adds material utilizing a 2D sketch profile and a depth perpendicular to the sketch plane. The Base feature is the first feature in the part.

Extruded Cut: A feature that removes material utilizing a 2D sketch profile and a depth perpendicular to the sketch plane.

Features: Features are geometry building blocks. Features add or remove material. Features are created from sketched profiles or from edges and faces of existing geometry.

Fillet: A feature that rounds sharp edges or faces by a specified radius.

Geometric relationships: Relations between geometry that are captured as you sketch.

Menus: Menus provide access to the commands that the SolidWorks software offers.

Mirror Entities: A sketch tool that mirrors sketch geometry to the opposite side of a sketched centerline.

Mouse Buttons: The left and right mouse buttons have distinct meanings in SolidWorks. The left mouse button is utilized to select geometry. The right-mouse button is utilized to invoke commands.

Offset Entities: A sketch tool that offsets sketch geometry to the current sketch plane by a specific amount.

Part: A part is a single 3D object that consists of various features. The filename extension for a SolidWorks part is .SLDPRT.

Plane: Planes are flat and infinite. Planes are represented on the screen with visible edges. The reference plane in Project 1 is the Top Plane.

Relation: A relation is a geometric constraint between sketch entities or between a sketch entity and a plane, axis, edge or vertex. Utilize Add Relations to manually connect related geometry.

Sketch: The name to describe a 2D profile is called a sketch. 2D sketches are created on flat faces and planes within the model. Typical geometry types are lines, arcs, rectangles, circles, polygons and ellipses.

States of a Sketch: There are four key states that are utilized in this Project:

- Fully Defined: Has complete information, (Black).
- Over Defined: Has duplicate dimensions, (Red).
- Under Defined: There is inadequate definition of the sketch, (Blue).
- Selected: The current selected entity, (Green).

Template: A template is the foundation of a SolidWorks document. A Part Template contains the Document Properties such as: Dimensioning Standard, Units, Grid/Snap, Precision, Line Style and Note Font.

Toolbars: The toolbars provide shortcuts enabling you to access the most frequently used commands.

Units: Used in the measurement of physical quantities. Decimal inch dimensioning and Millimeter dimensioning are the two types of common units specified for engineering parts and drawings.

Questions

1. Identify and describe the function of the following features:
    - Extruded Base/Boss.
    - Fillet.
    - Chamfer.
    - Extruded Cut.
2. Explain the differences between a Template and a Part.
3. Explain the steps in opening a SolidWorks session.
4. Describe the procedure to develop a new sketch.
5. Explain the steps required to change part unit dimensions from inches to millimeters.
6. Identify the three default reference planes.
7. What is a Base feature? Provide two examples.
8. Describe the differences between an Extruded Base feature and an Extruded Cut feature.
9. The sketch color black indicates a sketch is ___________ defined.
10. The sketch color blue indicates a sketch is ___________ defined.
11. The sketch color red indicates a sketch is ___________ defined.
12. True or False. Folders are utilized to only store part documents.
13. Describe a symmetric relation.
14. Describe an angular dimension.
15. What is a draft angle? Provide an example.
16. An arc requires _______ points?
17. Identify the properties of a Multi-body part.

Exercises

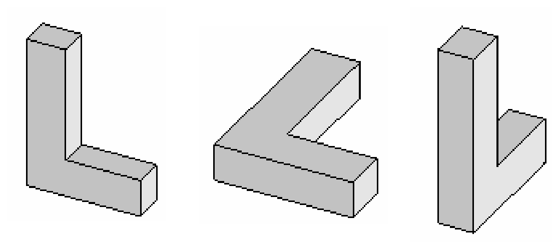
Exercise 1.1: Part Document Templates

Create a Metric part document template using an ANSI dimension standard.

Exercise 1.2: L-SHAPE Part

Create 3 parts: L-SHAPE-FRONT, L-SHAPE-TOP and L-SHAPE-RIGHT.

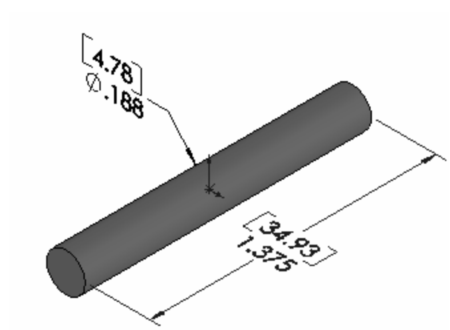
Utilize your own dimensions. Locate each profile on a different Sketch Plane.



Front Top Right

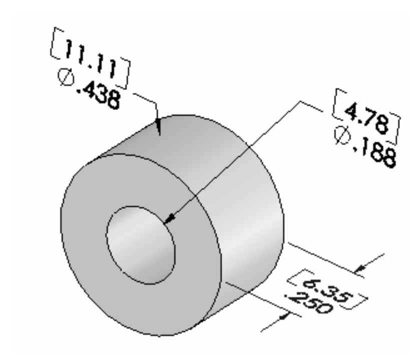
Exercise 1.3: AXLE Part

Create the AXLE part. Utilize the Front Plane for the Sketch plane. Use the provided dimensions.



Exercise 1.4: SHAFT COLLAR Part

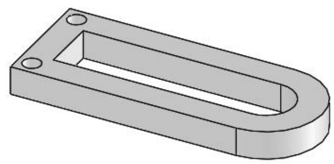
Create the SHAFT COLLAR part. Utilize the Front Plane for the Sketch plane. Use the provided dimensions.

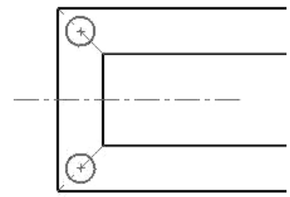


Exercise 1.5a -15.d: Create the following parts utilizing the Extrude Boss/Base, Extruded Cut, Fillet and Chamfer features. Dimensions are not provided. Utilize symmetry.

Exercise 1.5a: RING Part

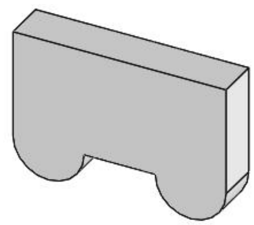
Use the Top Plane as the sketch plane. Use the Offset Entities tool. Utilize a Tangent Arc Sketch tool. The part is symmetrical about the Front Plane. Utilize two diagonal centerlines to locate the centerpoints of the circles at the Midpoint of the centerline.



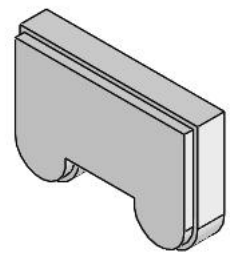


Exercise 1.5b: PLAQUE Part

Utilize the Offset Entities Sketch tool and Extruded Cut (Flip side) feature. The Base feature is symmetric about the Right Plane.



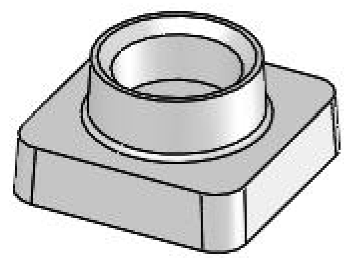
Extruded Base



Extruded Cut

Exercise 1.5c: CASTING Part

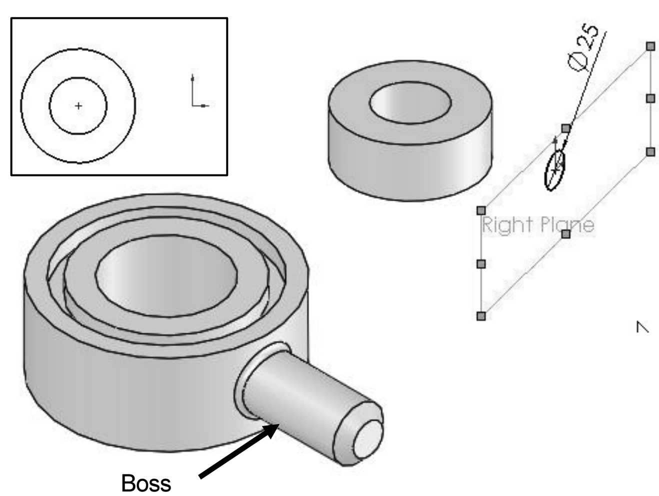
Utilize a 3º Draft Angle for the Extruded Base and Extrude Boss features. Add Fillets and Chamfers. Center the Base feature about the Origin.



Exercise 1.5d: FITTING Part

Sketch the profile for the Extruded Base feature to the left of the Origin. Insert the Extruded Boss feature on the Right Plane. Utilize the Up to Surface option. Add an Extruded Cut utilizing Offset Entities.

Insert Fillets and Chamfers.



Notes: