INSIDE:

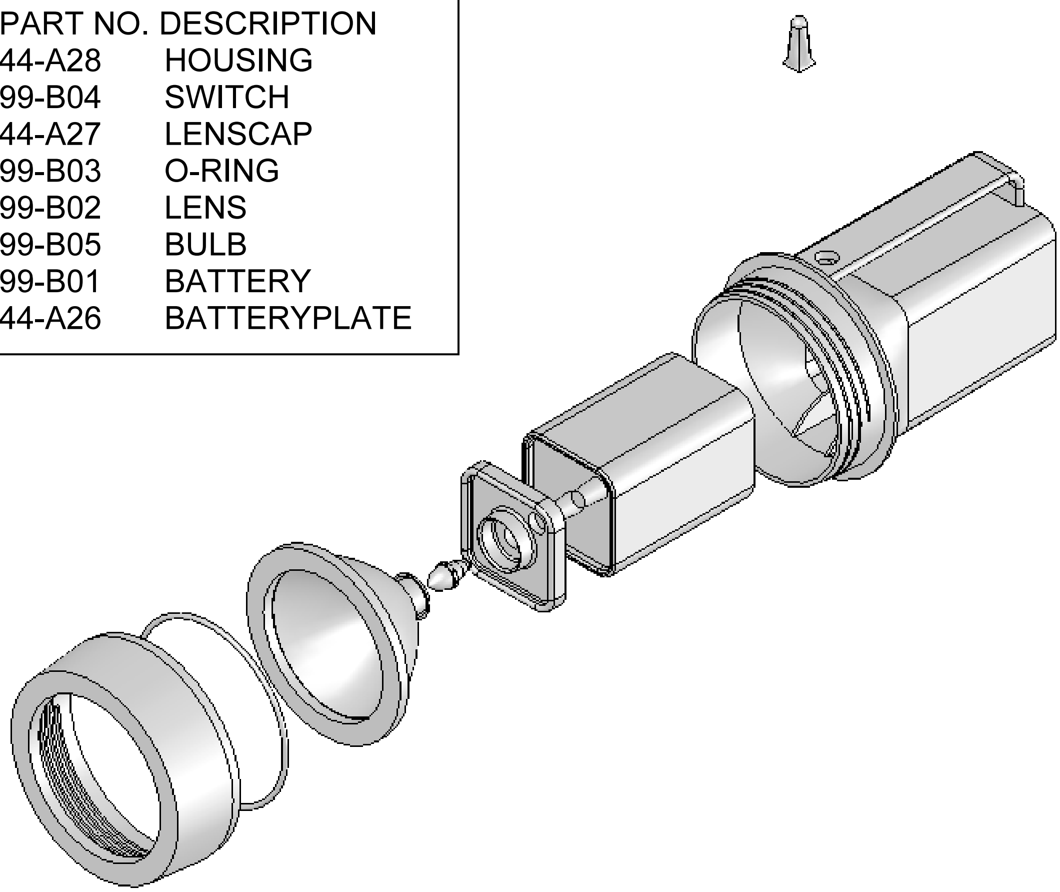
SolidWorks 2008: The Basics

with Multimedia CD

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

Planchard & Planchard

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	BATTERYPLATE



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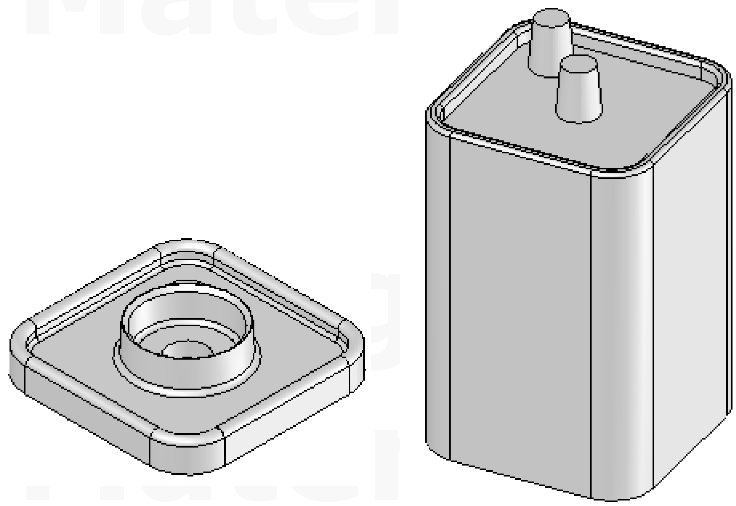
Project 1

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 2008 User Interface.	• Ability to establish a SolidWorks session. Use the SolidWorks User Interface: CommandManager, Toolbars, Task Pane, Search, Confirmation Corner, and more.
• Address File Management with file folders.	• Aptitude to create file folders for various Projects and Templates
• Create two Part Templates: ○ PART-IN-ANSI. ○ PART-MM-ISO.	• Skill to address System Options and Document Properties.
• Create two FLASHLIGH Parts: ○ BATTERY. ○ BATTERYPLATE.	• Specific knowledge and understanding of 2D sketching and the following SolidWorks 3D features: Extruded Base, Extruded Boss, Extruded Cut, Fillet, and Chamfer.

Notes:

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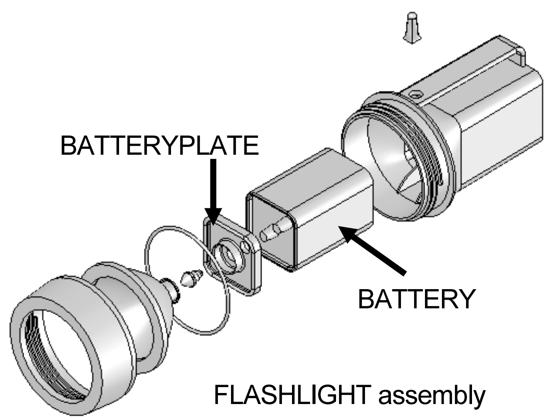
Project 1-Introduction to Part Modeling

Project Overview

SolidWorks is a design software application used to model and create 2D and 3D sketches, 3D parts, assemblies, and 2D drawings. Project 1 introduces you to the SolidWorks 2008 User Interface and CommandManager: Menu bar toolbar, Menu bar menu, Drop-down menus, Short-cut toolbars, Consolidated drop-down menus, System feedback icons, Confirmation Corner, Heads-up View toolbar, and Document Properties.

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.



FLASHLIGHT assembly

Create two parts for the FLASHLIGHT assembly in this project:

- BATTERY.
- BATTERYPLATE.

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

A 2D sketch is required to create an Extruded Base 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 2008 User Interface.
- Recognize the default Reference Planes.

- Insert a new 2D sketch and add sketch geometry with the following sketch tools: Line, Circle, Corner Rectangle, Tangent Arc, and Centerline.
- Establish Geometric relations, dimensions, and determine the status of the sketch.
- Manipulate existing geometry with the following Sketch tools: Line, Corner Rectangle, Circle, Convert Entities, Offset Entities, and Mirror Entities.
- Apply the following 3D 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 2008. Create two sub-folders named MY-TEMPLATES and PROJECTS.

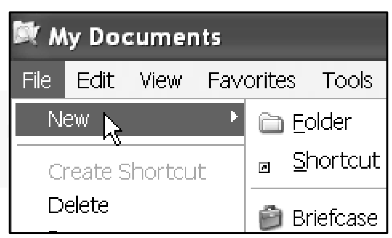
Activity: File Management

Create a new folder in Windows.

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

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

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



Enter the new folder name.

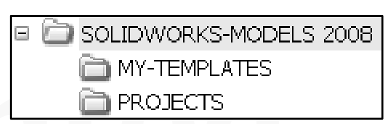
4) Enter **SOLIDWORKS-MODELS 2008**.

The book was written using SolidWorks Office 2003 on Windows XP Professional SP2 with a Windows Classic desktop theme.

Create the first sub-folder.

5) Double-click the **SOLIDWORKS-MODELS 2008** folder.

6) Click **File**, **New**, **Folder** from the Main menu. A New Folder icon is displayed. Enter **MY-TEMPLATES** for the folder name.



Create the second sub-folder.

7) Click the **SOLIDWORKS-MODELS 2008** folder.

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

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

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Return to the SOLIDWORKS-MODELS 2008 folder.

10) Click the **SOLIDWORKS-MODELS 2008** folder.

Note: 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. SolidWorks displays the Tip of the Day box. Read the Tip of the Day every day to obtain additional information on SolidWorks.

Create a new part. Click File, New from the Menu bar menu or click New from the Menu bar toolbar. There are two options for new documents: *Novice* and *Advanced*. Select the Advanced option. Select the Part document.

Activity: Start a SolidWorks Session

Start a SolidWorks 2008 session.

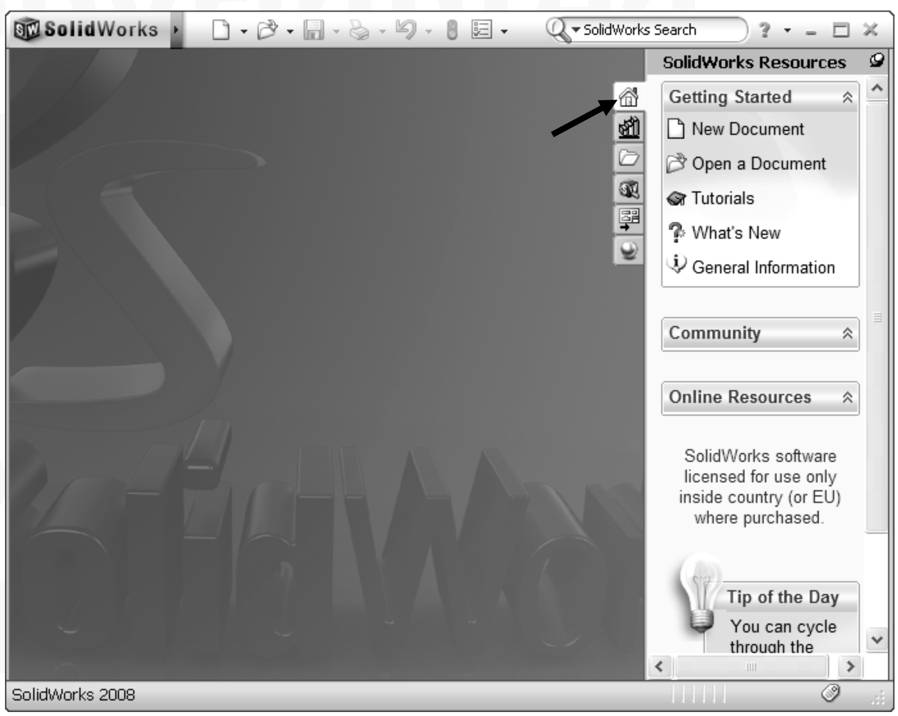
11) Click **Start** on the Windows Taskbar. Click **All Programs**. Click the **SolidWorks 2008** folder.

12) Click **SolidWorks 2008** application. The SolidWorks program window opens. Note: Do not open a document at this time.

If available, double-click the SolidWorks 2008 icon on the Windows Desktop to start a SolidWorks session.

Read the Tip of the Day dialog box.

13) If you do not see this screen, click the SolidWorks **Resources** icon on the right side of the Graphics window located in the Task Pane.



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Activity: Understanding the SolidWorks UI and CommandManager

Menu bar toolbar

SolidWorks 2008 (UI) is redesign to make maximum use of the Graphics window area. The default Menu bar toolbar contains a set of the most frequently used tool buttons from the Standard toolbar. The available tools are:

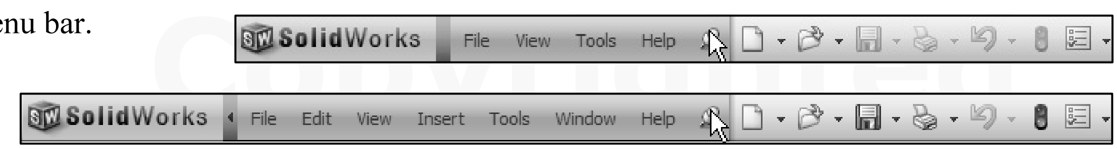
- **New** – Creates a new document.
- **Open** – Opens an existing document.
- **Save** – Saves an active document.
- **Print** – Prints an active document.
- **Undo** – Reverses the last action.
- **Rebuild** – Rebuilds the active part, assembly, or drawing.
- **Options** – Changes system options and Add-Ins for SolidWorks.

Menu bar menu

Click SolidWorks in the Menu bar toolbar to display the Menu bar menu. SolidWorks provides a Context-sensitive menu structure. The menu titles remain the same for all types of documents; (part, assembly, and drawing) but the menu items change depending on which type of document is active.

Example: The Insert menu includes features in part documents, mates in assembly documents, and drawing views in drawing documents. The display of the menu is also dependent on the work flow customization that you have selected. The default menu items for an active document are: *File*, *Edit*, *View*, *Insert*, *Tools*, *Window*, *Help*, and *Pin*.

The Pin tool displays the Menu bar toolbar and the Menu bar menu as illustrated. Throughout the book, the Menu bar menu and the Menu bar toolbar is referred to as the Menu bar.



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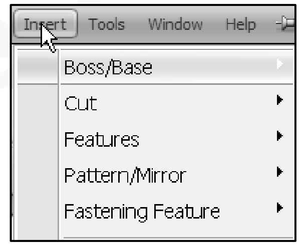
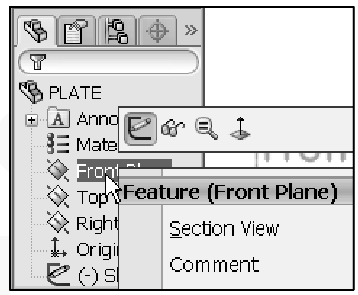
Drop-down menu



SolidWorks takes advantage of the familiar Microsoft® Windows® user interface. Communicate with SolidWorks either through the drop-down menu, pop-up menu, shortcut toolbar, flyout toolbar or the CommandManager. A command is an instruction that informs SolidWorks to perform a task.

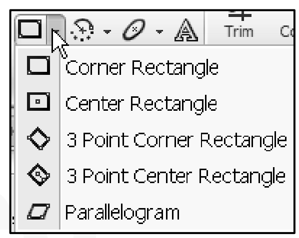
To close a SolidWorks drop-down menu, press the Esc key. You can also click any other section in the SolidWorks Graphics window, or click another drop-down menu.

Right-click Pop-up menus



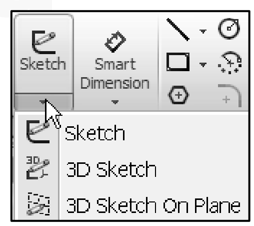
Right-click in the Graphics window either on a model, or in the FeatureManager on a feature or sketch to display a Context-sensitive shortcut toolbar. If you are in the middle of a command, the toolbar displays a list of options specifically related to that command.

Flyout tool buttons / Consolidated menu



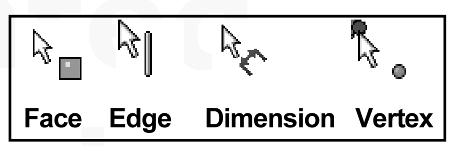
The Flyout tool buttons are new for 2008. Similar commands are grouped into flyout buttons on toolbars and the CommandManager. Example: Variations of the rectangle tool are consolidated together in a button with a flyout control as illustrated. Select the drop-down arrow and view the available tools.

If you select the flyout button without expanding:



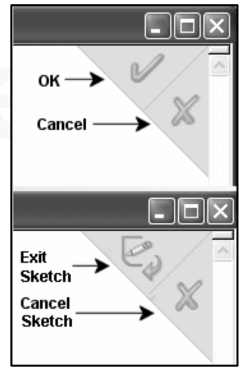
- For some commands such as Sketch, the most commonly used command is performed. This command is the first listed and the command shown on the button.
- For commands such as rectangle, where you may want to repeatedly create the same variant of the rectangle, the last used command is performed. This is the highlighted command when the flyout tool is expanded.

System feedback icons



SolidWorks provides system feedback by attaching a symbol to the mouse pointer cursor arrow. The system feedback symbol indicates what you are selecting or what the system is expecting you to select. As you move the mouse pointer across your model, system feedback is provided.

Confirmation Corner

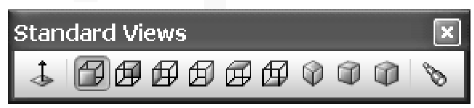


When numerous SolidWorks commands are active, a symbol or a set of symbols are displayed in the upper right corner of the Graphics window. This area is called the Confirmation Corner.

When a sketch is active, the confirmation corner box displays two symbols. The first symbol is the sketch tool icon. The second symbol is a large red X. These three symbols supply a visual reminder that you are in an active sketch. Click the sketch symbol icon to exit the sketch and to saves any changes that you made.

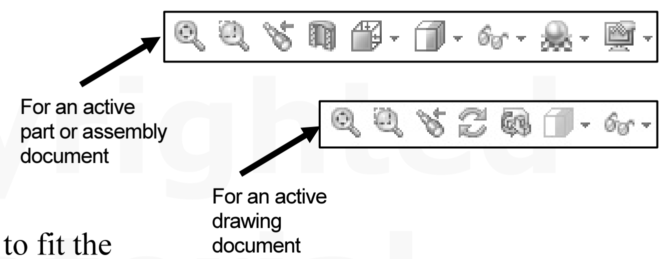
When other commands are active, the confirmation corner box provides a green check mark and a large red X. Use the green check mark to execute the current command. Use the large red X to cancel the command.

Heads-up View toolbar

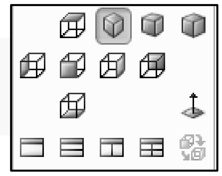


SolidWorks provides the user with numerous view options from the Standard Views, View, and Heads-up View toolbar which are new for 2008.

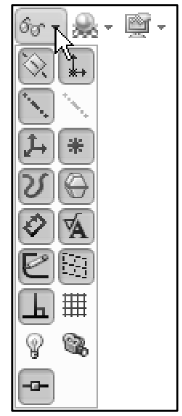
The Heads-up View toolbar is a transparent toolbar that is displayed in the Graphics window when a document is active. You can't hide nor move the Heads-up View toolbar. The following views are available: Note: Views are document dependent.



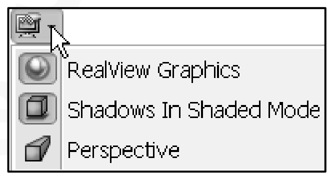
- *Zoom to Fit* : Zooms the model to fit the Graphics window.
- *Zoom to Area* : Zooms to the areas you select with a bounding box.
- *Previous View* : Displays the previous view.
- *Section View* : Displays a cutaway of a part or assembly, using one or more cross section planes.



- *View Orientation*: Provides the ability to select a view orientation or the number of viewports. The available options are: *Top*, *Isometric*, *Trimetric*, *Dimetric*, *Left*, *Front*, *Right*, *Back*, *Bottom*, *Single view*, *Two view - Horizontal*, *Two view - Vertical*, *Four view*.

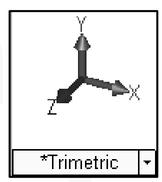


- *Display Style*: Provides the ability to display the style for the active view. The available options are: *Wireframe*, *Hidden Lines Visible*, *Hidden Lines Removed*, *Shaded*, *Shaded With Edges*.
- *Hide/Show Items*: Provides the ability to select items to hide or show in the Graphics window. Note: The available items are document dependent.
- *Apply Scene*: Provides the ability to apply a scene to an active part or assembly document. View the available options.
- *View Setting*: Provides the ability to select the following: *RealView Graphics*, *Shadows in Shaded Mode*, and *Perspective.*

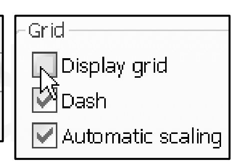


- *Rotate*: Provides the ability to rotate a drawing view.
- *3D Drawing View*: Provides the ability to dynamically manipulate the drawing view to make a selection.

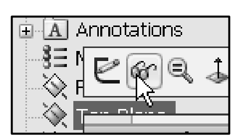
For 2008 the Heads-up View toolbar replaces the Reference triad in the lower left corner of the Graphics window.



The default document setting displays reference planes and the grid in the Graphics window. To deactivate the reference planes for an active document, click **View**, uncheck **Planes** from the Menu bar. To deactivate the grid, click **Options**, **Document Properties** tab. Click **Grid/Snaps**, uncheck the **Display grid** box.



To deactivate a single reference plane in an active document, right-click the **selected plane**, click **Hide**.



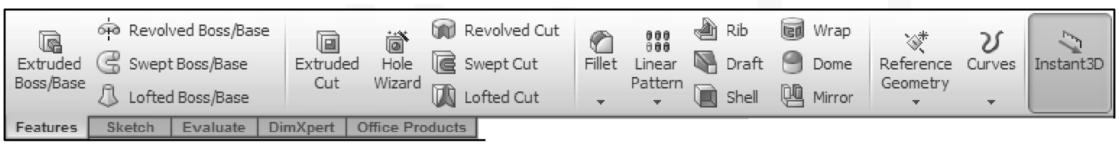
CommandManager

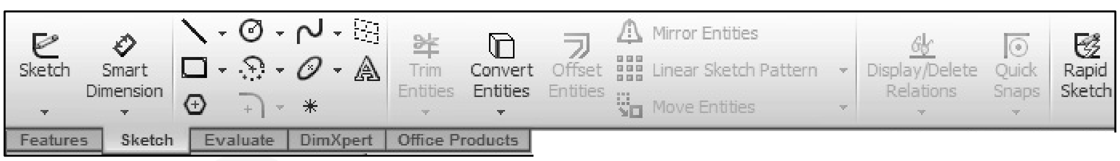
The CommandManager is document dependent. The tabs are located on the bottom left side of the CommandManager and display the available toolbars and features for each corresponding tab. The default tabs are: *Features*, *Sketch*, *Evaluate*, *DimXpert*, and *Office Products*. The tabs are new for 2008.

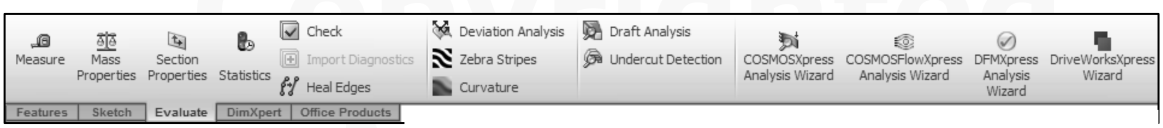
Below is an illustrated CommandManager for a default part document.

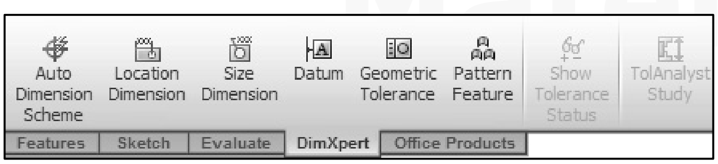
The Office Products toolbar display is dependent on the activated Add-Ins.. during a SolidWorks session.

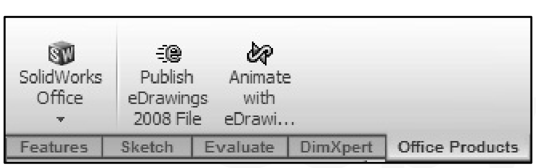
If you have SolidWorks Office, SolidWorks Office Professional, or SolidWorks Office Premium, the Office Products tab is displayed in the CommandManager. The book was written with SolidWorks Office Premium using version SPO.











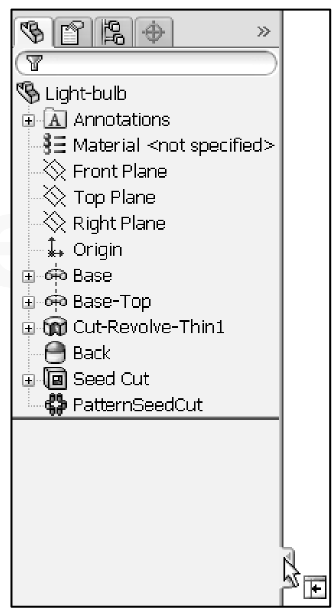
The tabs replace the Control areas buttons from pervious SolidWorks versions. The tabs that are displayed by default depend on the type of document open and the work flow customization that you have selected.

To customize the CommandManager tabs, right-click on a tab, and select the required custom option or select Customize CommandManager to access the Customize dialog box.

DimXpert for parts provides the ability to graphically check if the model is fully dimensioned and toleranced.

Both DimXpert for parts and drawings automatically recognize manufacturing features. Manufacturing features are *not SolidWorks features*. Manufacturing features are defined in 1.1.12 of the ASME Y14.5M-1994 Dimensioning and Tolerancing standard as: “The general term applied to a physical portion of a part, such as a surface, hole or slot.

FeatureManager Design Tree



The FeatureManager design tree is located on the left side of the SolidWorks Graphics window. The design tree provides a summarize view of the active part, assembly, or drawing document. The tree displays the details on how the part, assembly, or drawing document is created.

Understand the FeatureManager design tree to troubleshoot your model. The FeatureManager is use extensively throughout this book.

The FeatureManager consist of four default tabs:

- *FeatureManager design tree.*
- *PropertyManager.*
- *ConfigurationManager.*
- *DimXertManager.*

Select the Hide FeatureManager Tree Area arrows  tab from the FeatureManager to enlarge the Graphics window for modeling.

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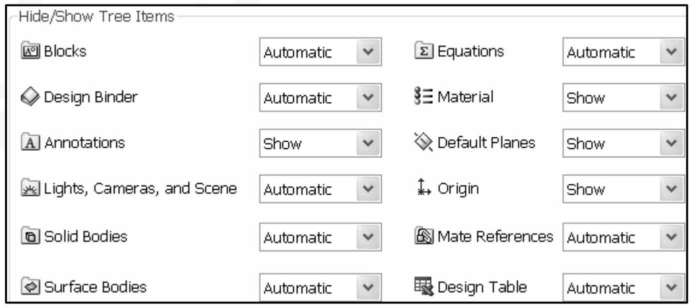
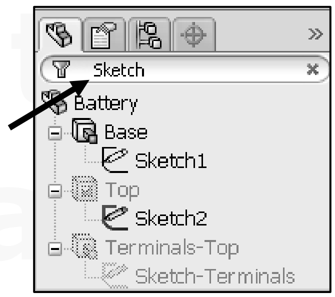
New commands in 2008 provide the ability to control what is displayed in the FeatureManager design tree. They are:

1. Show or Hide FeatureManager items.

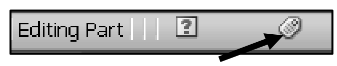


Click **Options** from the Menu Bar toolbar. Click **FeatureManager** from the System Options tab. Customize your FeatureManager from the Hide/Show Tree Items dialog box.

2. Filter the FeatureManager design tree. Enter information in the filter field. You can filter by: *Type of features, Feature names, Sketches, Folders*, *Mates*, *User-defined tags*, and *Custom properties*.



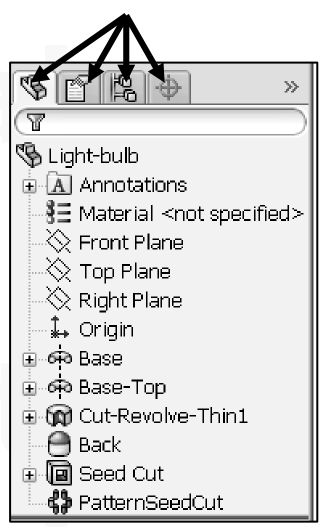
Tags are keywords you can add to a SolidWorks document to make them easier to filter and to search. The Tags icon is located in the bottom right corner of the Graphics window.



To collapse all items in the FeatureManager, **right-click** and select **Collapse items**, or press the **Shift +C** keys.

The FeatureManager design tree and the Graphics window are dynamically linked. Select sketches, features, drawing views, and construction geometry in either pane.

Split the FeatureManager and either display two FeatureManager instances, or combine the FeatureManager design tree with the ConfigurationManager or PropertyManager.



Move between the FeatureManager, PropertyManager, ConfigurationManager, and DimXertManager by selecting the tabs at the top of the menu.

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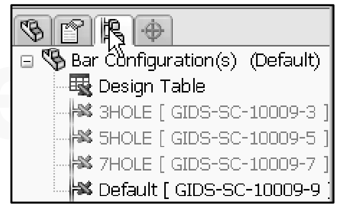
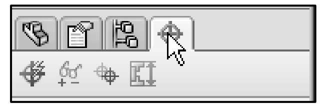


The ConfigurationManager is located to the right of the FeatureManager. Use the ConfigurationManager to create, select, and view multiple configurations of parts and assemblies.

Note: The icons in the ConfigurationManager denote whether the configuration was created manually or with a design table.



DimXpertManager is new for 2008. The DimXpertManager tab provides the ability to insert dimensions and tolerances manually or automatically. The DimXpertManager provides the following selections: *Auto Dimension Scheme*, *Show Tolerance Status*, *Copy Scheme*, and *TolAnalyst Study*.

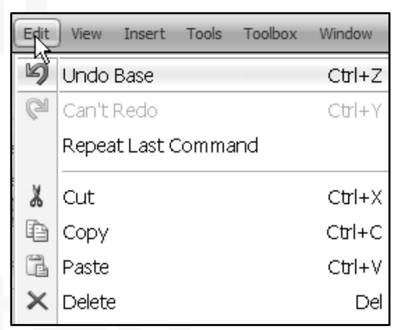
Fly-out FeatureManager

The fly-out FeatureManager design tree provides the ability to view and select items in the PropertyManager and the FeatureManager design tree at the same time.

The fly-out FeatureManager provides the ability to select items which may be difficult to view or select from the Graphics window.

Throughout the book, you will select commands and command options from the drop-down menus, fly-out FeatureManager, shortcut toolbars, or from the SolidWorks toolbars.

Another method for accessing a command is to use the accelerator key. Accelerator keys are special keystrokes which activates the drop-down menu options. Some commands in the menu bar and items in the drop-down menus have an underlined character. Press the Alt key followed by the corresponding key to the underlined character activates that command or option.



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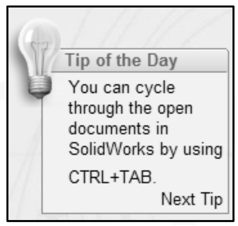
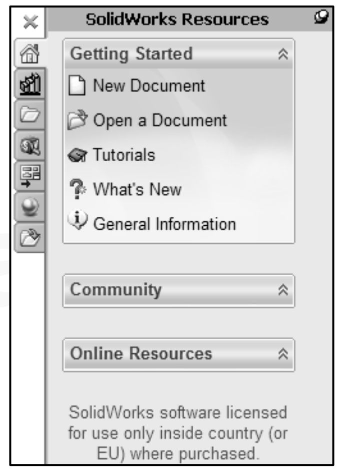
Task Pane

The Task Pane is displayed when a SolidWorks session starts. The Task Pane contains the following default tabs: *SolidWorks Resources*, *Design Library*, *File Explorer*, *SolidWorks Search*, *View Palette*, *RealView*, and *Document Recovery*.

The Document Recovery tab is only displayed in the Task Pane if your system terminates unexpectedly with an active document and if auto-recovery is enabled in the System Options section.

SolidWorks Resources

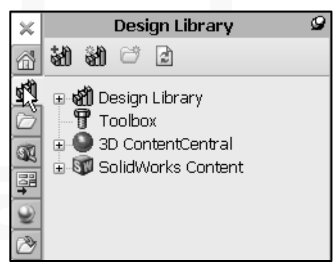




The basic SolidWorks Resources menu displays the following default selections: *Getting Started*, *Community*, *Online Resources*, and *Tip of the Day*.

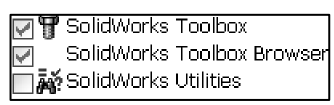
Other user interfaces are available during the initial software installation selection: *Machine Design*, *Mold Design*, or *Consumer Products Design*.

Design Library



The Design Library contains reusable parts, assemblies, and other elements, including library features.

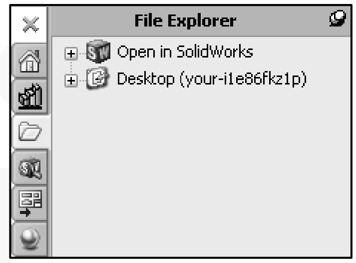
The Design Library tab contains four default selections. Each default selection contains additional sub categories. The default selections are: *Design Library*, *Toolbox*, *3D ContentCentral*, and *SolidWorks Content*.



Click **Tools**, **Add-Ins…**, **SolidWorks Toolbox** and **SolidWorks Toolbox Browser** to active the SolidWorks Toolbox.

To access the Design Library folders in a non network environment for a new installation, click **Add File Location**, enter: **C:\Documents and Settings\All Users\Application Data\SolidWorks\SolidWorks 2008\design library**. Click **OK**. In a network environment, contact your IT department for system details.

File Explorer



File Explorer duplicates Windows Explorer from your local computer and displays the following directories: *Recent Documents*, and *Open in SolidWorks*.

Search

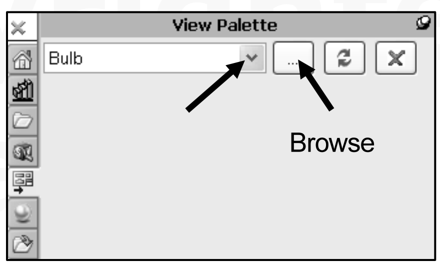
SolidWorks Search is installed with Microsoft Windows Search and indexes the resources once before searching begins, either after installation, or when you initiate the first search.

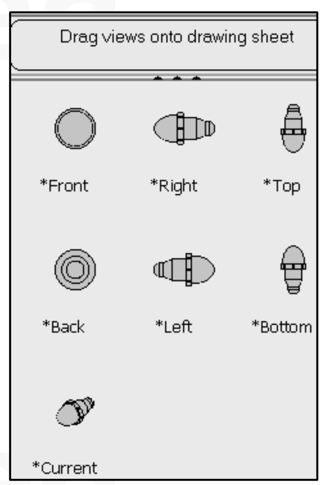
The SolidWorks Search box is displayed in the upper right corner of the SolidWorks Graphics window. Enter the text or key words to search. Click the drop-down arrow to view the last 10 recent searches.

The Search tool in the Task Pane searches the following default locations: *All Locations*, *Local Files*, *Design Library*, *SolidWorks Toolbox*, and *3D ContentCentral*.

Select any or all of the above locations. If you do not select a file location, all locations are searched.

View Palette

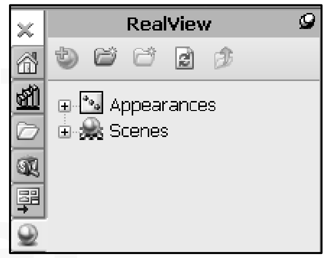




The View Palette tool located in the Task Pane provides the ability to insert drawing views of an active document, or click the Browse button to locate the desired document.

Click and drag the view from the View Palette into an active drawing sheet to create a drawing view.

RealView



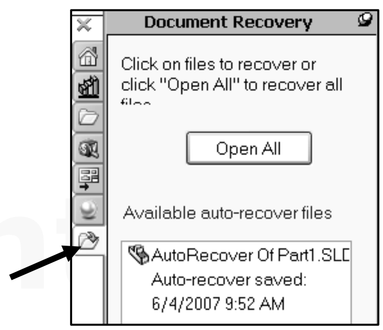
RealView provides a simplified way to display models in a photo-realistic setting using a library of appearances and scenes. Note: RealView requires graphics card support and is memory intensive!

On RealView compatible systems, you can select Appearances and Scenes to display your model in the Graphics window. Drag and drop a selected appearance onto the model or FeatureManager. View the results in the Graphics window.

PhotoWorks needs to be active to apply the scenes tool.

RealView graphics is only available with supported graphics cards. For the latest information on graphics cards that support RealView Graphics display, visit: *www.solidworks.com/pages/services/videocardtesting.html*.

Document Recovery



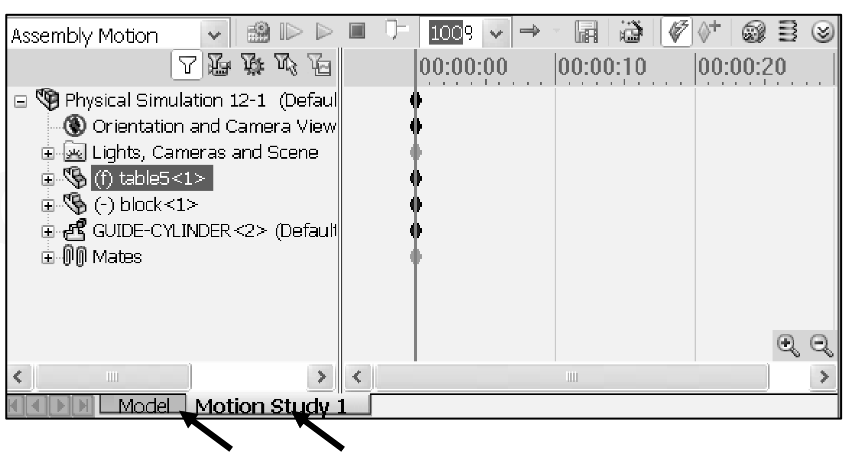
Document Recovery provides the ability to save information files if the system terminates unexpectedly with an active document. The saved files are available on the Task Pane Document Recovery tab the next time you start a SolidWorks session. Note: Auto recovery is activated by default in the System Options section.

Motion Study tab

The Motion Study tab is located in the bottom left corner of the Graphics window. Motion Study uses a key frame-based interface, and provides a graphical simulation of motion for the selected model.

Click the Motion Study tab to view the MotionManager.

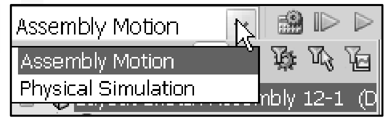
Click the Model tab to return to the FeatureManager design tree.



The MotionManager displays a timeline-based interface, and provides the following selections:

1. *All levels*. Provides the ability to change viewpoints, display properties, and create animations displaying the assembly in motion.

2. *Assembly Motion*. (Available in core SolidWorks.) Provides the ability to animate the assembly and to control the display at various time intervals. The Assembly Motion option computes the sequences required to go from one position to the next.



3. *Physical Simulation*. (Available in core SolidWorks.) Provides the ability to simulating the effects of motors, springs, dampers, and gravity on assemblies. This options combines simulation elements with SolidWorks tools such as mates and Physical Dynamics to move components around the assembly.

4. *COSMOSMotion*. (Available in SolidWorks Office Premium.) Provides the ability to simulate, and analyze the effects of forces, contacts, friction, and motion on an assembly.

If the Motion Study tab is not visible, click **View**, **MotionManager** from the Menu bar. Note: On a model that was created before SolidWorks 2008, the Annotation tab may be displayed in the Motion Study location.

To create a new Motion Study, click **Insert, New Motion Study** from the Menu bar.

View the Assembly Project for additional information on Motion Study.

View the Assembly Project for additional information on Motion Study.

Activity: Create a new 3D Part

A part is a 3D model which consists of features. What are features?

- Features are geometry building blocks.
- Features add or remove material.
- Features are created from 2D or 3D sketched profiles or from edges and faces of existing geometry.

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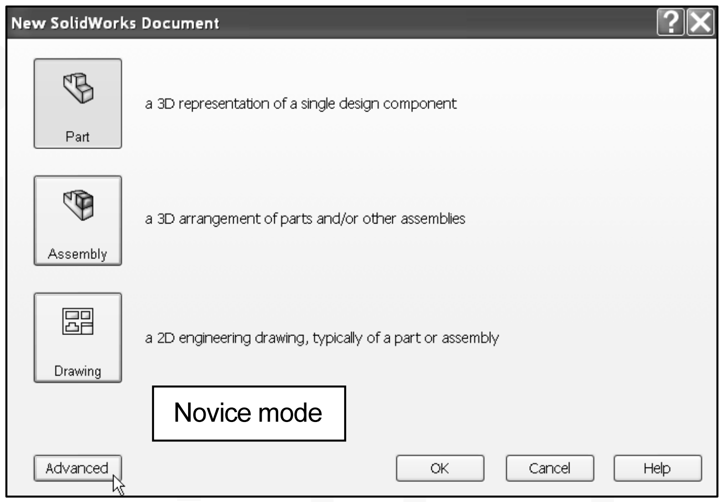
Your default system document templates may be different if you are a new user of SolidWorks 2008 vs. an existing user who has upgraded from a previous version.

In this book, Reference planes and Grid/Snaps are deactivated in the Graphics window for improved model clarity.

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 more templates.

Create a new part.

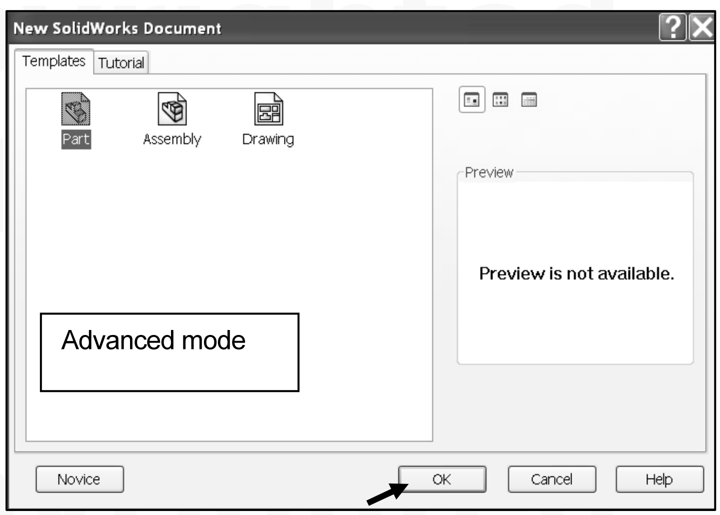
14) Click **New** from the Menu bar. The New SolidWorks Document dialog box is displayed.



Select the Advanced mode.

15) Click the **Advanced** button. The advanced mode is set.

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



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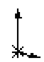
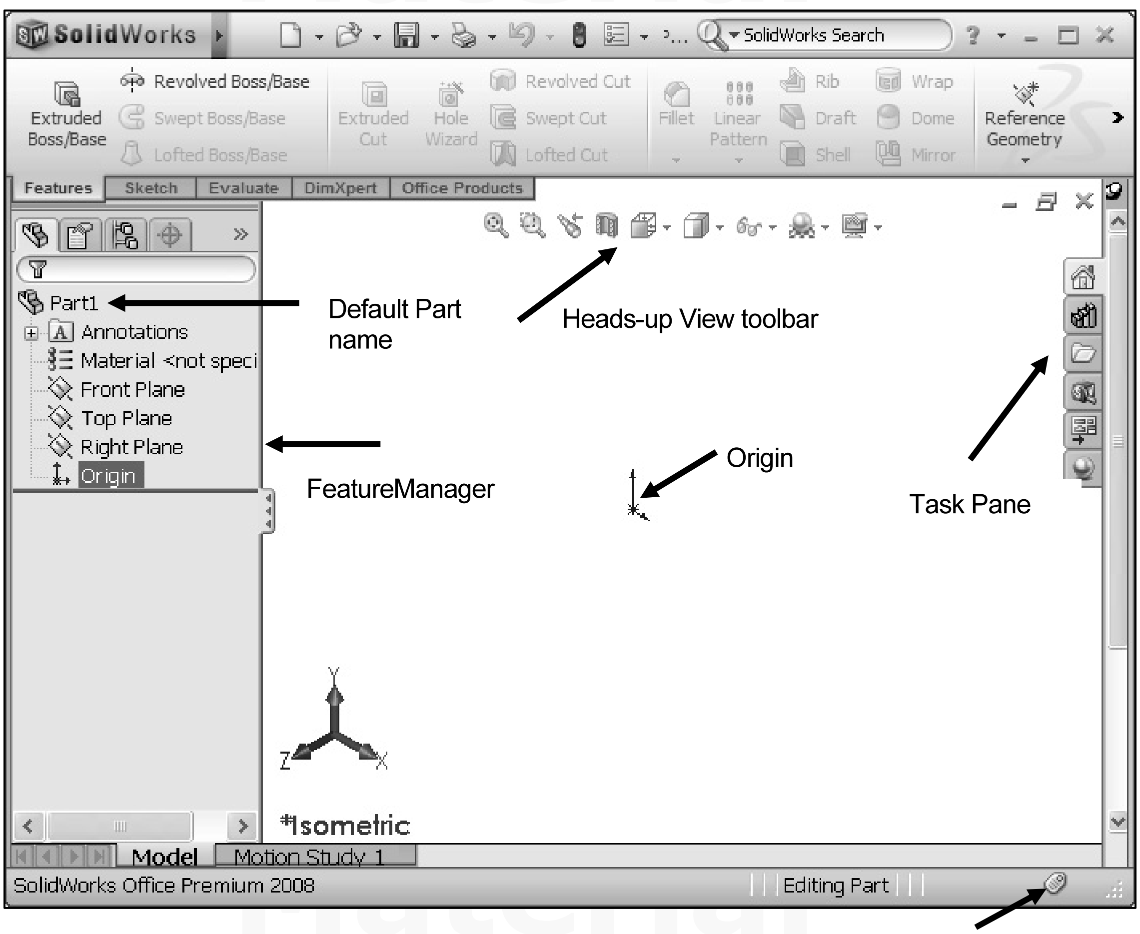
The Advanced mode remains selected for all new documents in the current SolidWorks session. When you exit SolidWorks, the Advanced mode setting is saved.

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.

Part1 is displayed in the FeatureManager and is the name of the document. Part1 is the default part window name. The Menu bar, CommandManager, FeatureManager, Heads-up View toolbar, SolidWorks Resources, SolidWorks Search, Task Pane, and the Origin are displayed in 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.



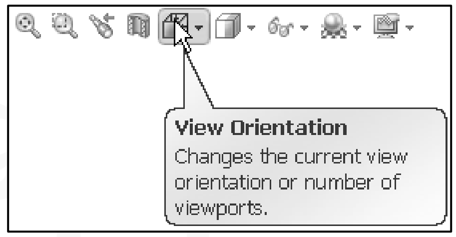
The Tags icon is displayed in the bottom right corner of the Graphics window. Tags are keywords you add to SolidWorks documents and features to make them easier to filter and search for.

In this book, Reference planes and Grid/Snaps are deactivated in the Graphics window for improved model clarity.

Activity: Menu Bar toolbar, Menu Bar menu, Heads-up View toolbar

Display tools and tool tips.

17) Position the **mouse pointer** over the Heads-up View toolbar and view the tool tips.



18) **Read** the large tool tip.

19) Select the **drop-down arrow** to view the available view tools.

Display the View toolbar and the Menu bar.

20) Right-click in the **gray area** of the Menu bar.

21) Click **View**. The View toolbar is displayed.

22) Click and drag the **View toolbar** off the Graphics window.

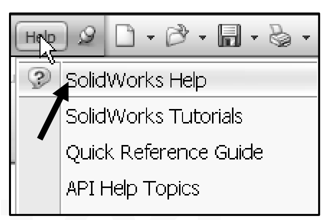
23) Click **SolidWorks** as illustrated to expand the Menu bar menu.

24) **Pin** the Menu bar as illustrated. Use both the Menu bar menu and the Menu bar toolbar in this book.

The SolidWorks Help Topics contains step-by-step instructions for various commands. The Help icon is displayed in the dialog box or in the PropertyManager for each feature.

Display SolidWorks Help. Use SolidWorks Help to locate information on sketches, features, and tools.

25) Click **Help** from the Menu bar. The Help options are displayed.



26) Click **SolidWorks Help**. The SolidWorks Help dialog box is displayed.

The SolidWorks Help dialog box contains the following tabs:

- **Contents** tab: Contains the SolidWorks Online User's Guide documents.
- **Index** tab: Contains additional information on key words.
- **Search** tab. Locates needed information.

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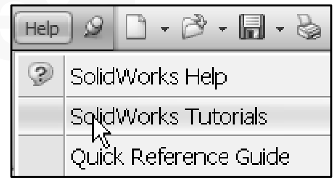
27) **Close** the SolidWorks Help dialog box.

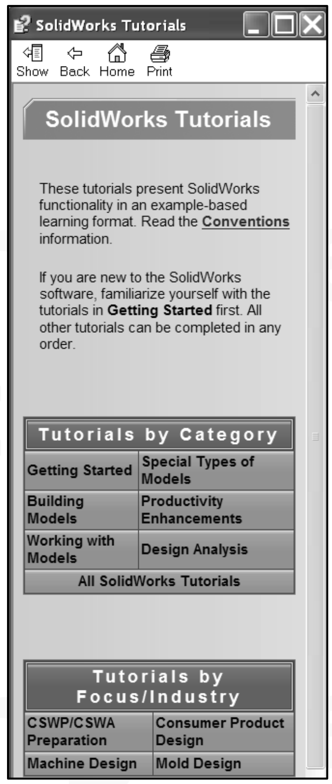
Display and explore the SolidWorks Tutorials.

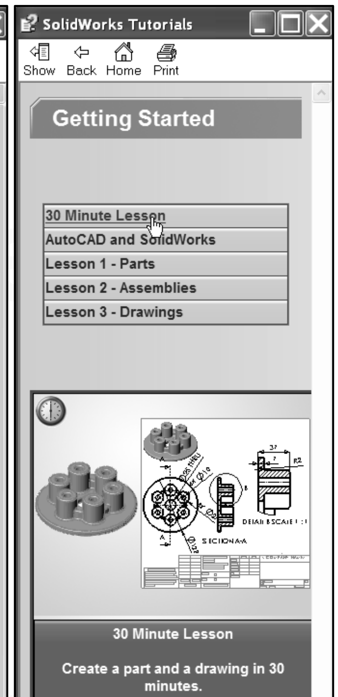
28) Click **Help** from the Menu bar.



29) Click **SolidWorks Tutorials**. The SolidWorks Tutorials are displayed. The SolidWorks Tutorials are presented by category.

30) Click the **Getting Started** category. The Getting Started category provides three 30 minute lessons on parts, assemblies, and drawings. This section also provides information for users who are switching from AutoCAD to SolidWorks. Note: The tutorials provide links to the CSWP and CSWA Certification programs.





SolidWorks Corporation offers two levels of certification representing increasing levels of expertise in 3D CAD design as it applies to engineering: Certified SolidWorks Associate CSWA, and the Certified SolidWorks Professional CSWP.

The CSWA certification indicates a foundation in and apprentice knowledge of 3D CAD design and engineering practices and principles.

The main requirement for obtaining the CSWA certification is to take and pass the three hour, seven question on-line exam at a Certified SolidWorks CSWA Provider, "university, college, technical, vocational, or secondary educational institution" and to sign the SolidWorks Confidentiality Agreement.

Passing this exam provides students the chance to prove their knowledge and expertise and to be part of a world wide industry certification standard.

Return to the SolidWorks Graphics window.

31) **Close** the Online Tutorial dialog box.

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Design Intent

The SolidWorks definition of design intent is the process in which the model is developed to accept future changes. Models behave differently when design changes occur. Design for change. Utilize geometry for symmetry, reuse common features and reuse common parts.

Build change into the following areas:

1. Sketch.

2. Feature.

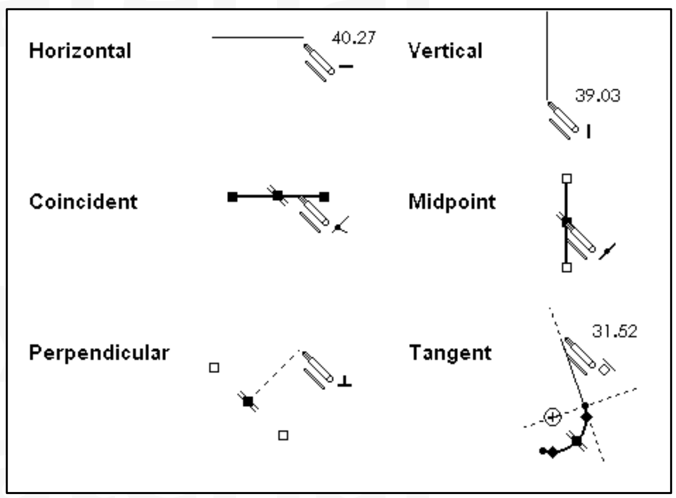
3. Part.

4. Assembly.

5. Drawing.

1. Design Intent in the Sketch

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



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

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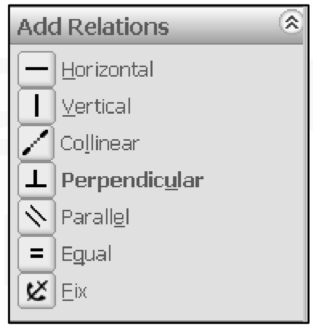
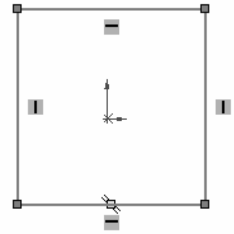
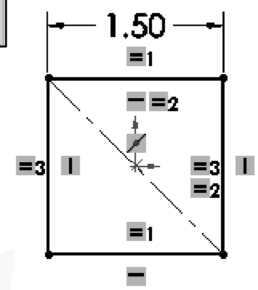
A rectangle contains Horizontal, Vertical, and Perpendicular automatic Geometric relations. Apply design intent using added Geometric relations. Example: Horizontal, Vertical, Collinear, Perpendicular, Parallel, etc.

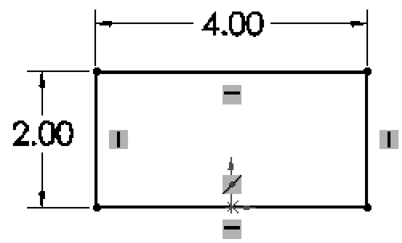




Example A: Apply design intent to create a square profile. Sketch a corner rectangle with the Origin approximately in the center. Insert a construction reference centerline. Add a Midpoint relation. Add an Equal relation between the two vertical and horizontal lines. Insert a dimension to define the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

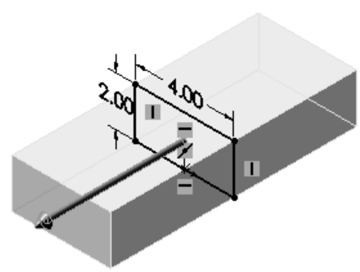


Example B: Develop a corner rectangular profile. The bottom horizontal midpoint of the rectangular profile is located at the Origin. Sketch a rectangle. Add a Midpoint relation between the horizontal edge of the rectangle and the Origin. Insert two dimensions to define the width and height of the rectangle.



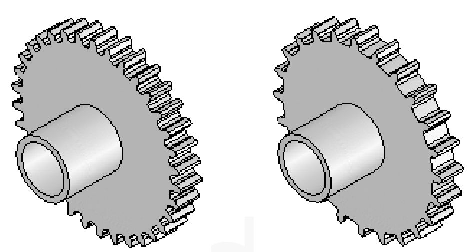
2. Design Intent in the Feature

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



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

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



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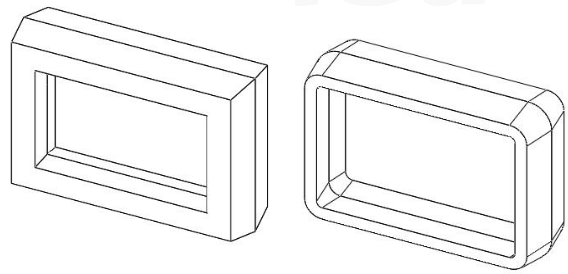
3. Design Intent in the Part

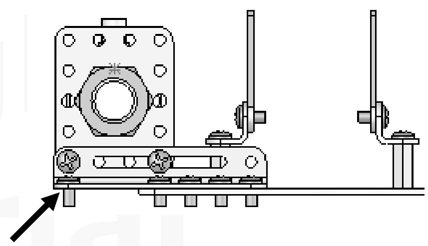
Utilize symmetry, feature order and reusing common features to build design intent into the part.



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

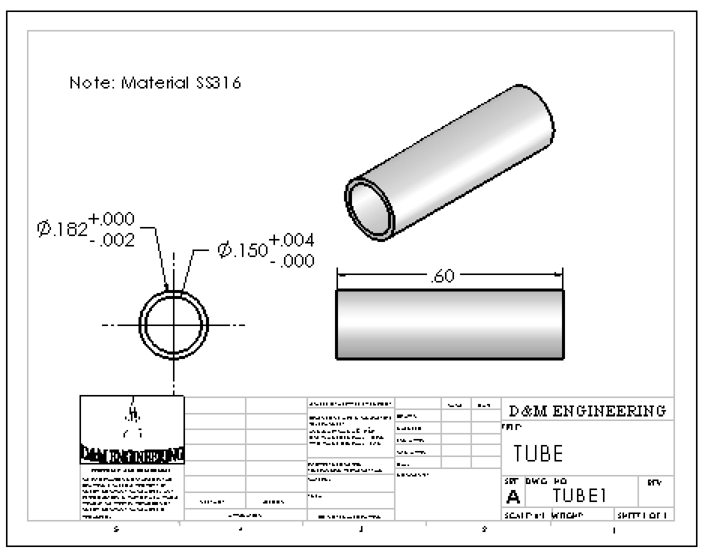
4. Design Intent in the Assembly

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



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

5. Design Intent in the Drawing



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

Example A: Tolerance and material in the drawing.

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

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

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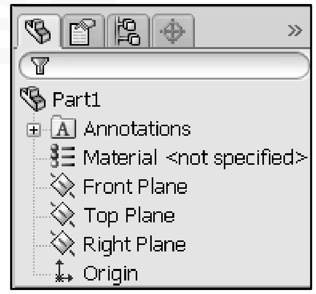
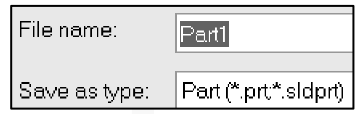
Part Template

The Part template is the foundation for a SolidWorks part. Part1 displayed in the FeatureManager utilizes the Part.sldprt default template located in the New SolidWorks 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. You will modify the following document properties in this Project: 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. MMGS, (millimeter, gram, second) and IPS, (inch, pound, second) are the two most common unit systems 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.

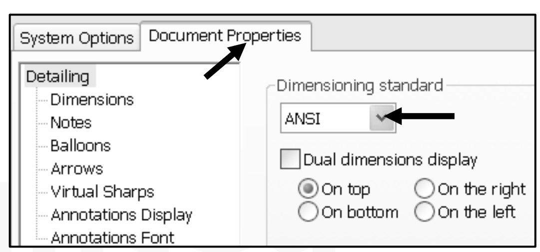
System Options are stored in the registry of your computer. The File Locations option controls the file folder location of SolidWorks documents. Utilize the File Locations option to reference your Part Templates in the MY-TEMPLATES folder. Add the SOLIDWORKS-MODELS 2008\MY-TEMPLATES folder path name to the Document Templates File Locations list.

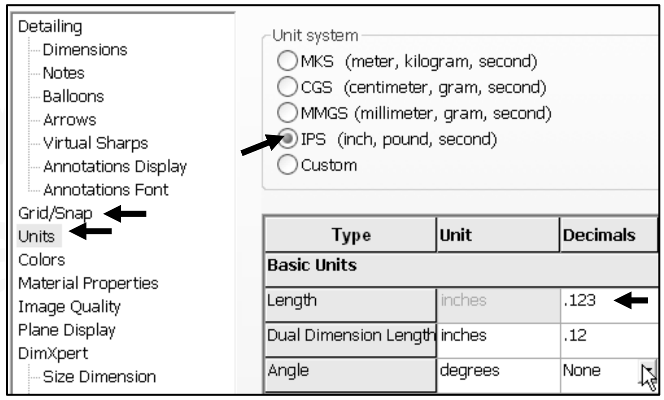
Activity: Create the PART-IN-ANSI and PART-MM-ISO Part Template

Create the **PART-IN-ANSI** Part template.

32) Click **Options**, **Document Properties** tab from the Menu bar. The Document Properties - Detailing dialog box is displayed.

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





Set the part units.

34) Click **Units**. The Document Properties - Unit dialog box is displayed.

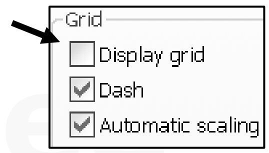
35) Select **IPS**, **(inch, pound, second)** for Unit system.

36) Select **.123** (three decimal places) for Length basic units.

37) Select **None** for Angular units Decimal places.

Set the Grid/Snap option.

38) Click **Grid/Snap**. The Document Properties Grid/Snap dialog box is displayed.



39) Uncheck the **Display grid** box.

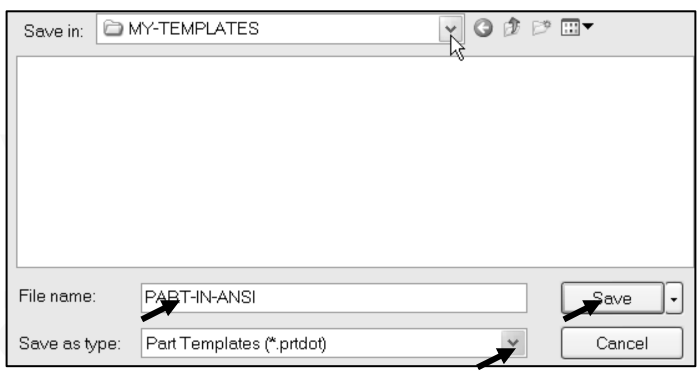
Return to the SolidWorks Graphics window.

40) Click **OK** from the Document Properties Grid/Snap dialog box.

Save the Part Template.

41) Click **Save As** from the Menu bar. The Save As dialog box is displayed.

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



43) Select the **SOLIDWORKS-MODELS 2008/MY-TEMPLATES** folder.

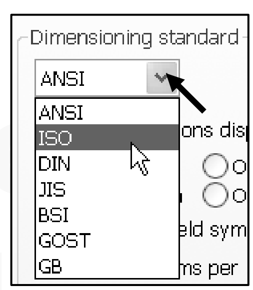
44) Enter **PART-IN-ANSI** in the File name box.

45) Click **Save** from the Save As dialog box.

Create the PART-MM-ISO Part Template.

46) Click **Options**, **Document Properties** tab from the Menu bar. The Document Properties - Detailing dialog box is displayed.

47) Select **ISO** from the Dimensioning standard drop-down menu.



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Set the part units.

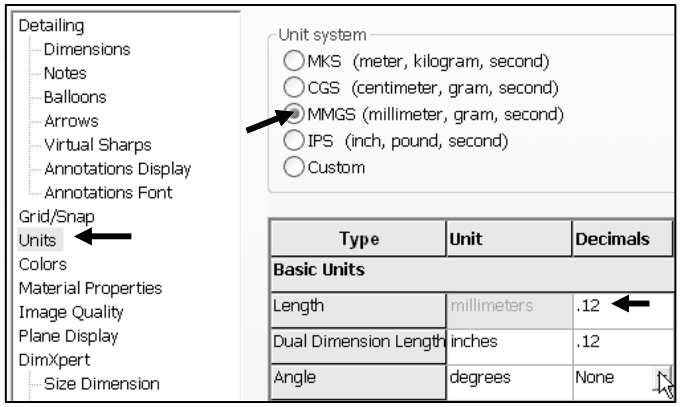
48) Click **Units**. The Document Properties - Unit dialog box is displayed.

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

50) Select .**12** (two decimal places) for Length basic units.

51) Select **None** for Angular units Decimal places.

52) Click **OK**.



Save the Part Template.

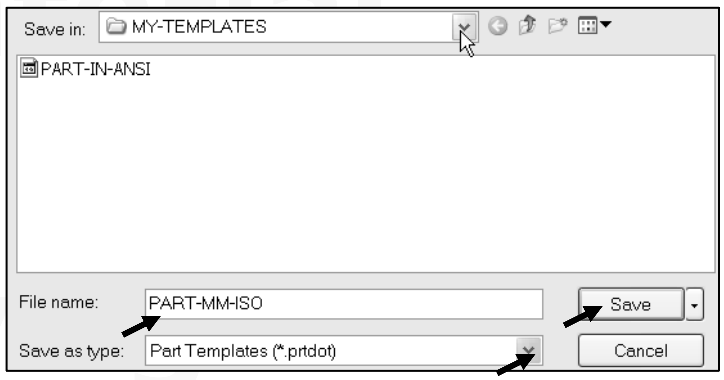
53) Click **Save As** from the Menu bar. The Save As dialog box is displayed.

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

55) Select the **SOLIDWORKS-MODELS 2008/MY-TEMPLATES** folder.

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

57) Click **Save**.

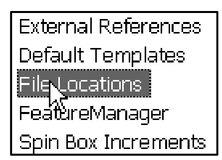


Set the System Options.

58) Click **Options** from the Menu bar. The System Options - General dialog box is displayed.

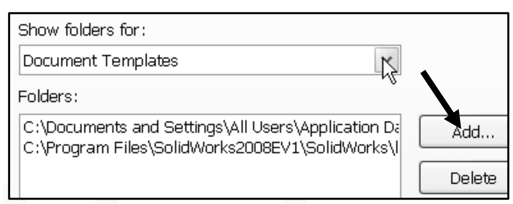
59) Click **File Locations** from the System Options tab.

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



61) Click the **Add** button.

62) Select the **SOLIDWORKS-MODELS 2008/MY-TEMPLATES** folder.



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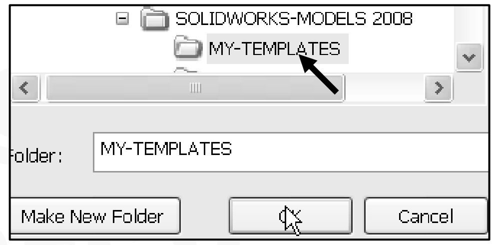
63) Click **OK** from the Browse for Folder.

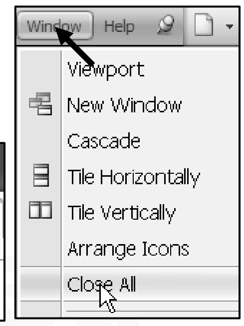
64) Click **OK** from the System Options - File Location dialog box.

65) Click **Yes** to add the new file location.



Close All documents.

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

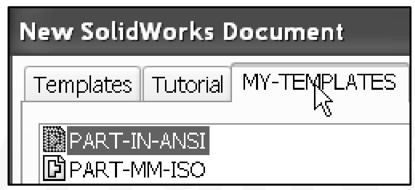


Display the MY-TEMPLATES folder and templates.

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

68) Click the **MY-TEMPLATES** tab. View the two new Part Templates.

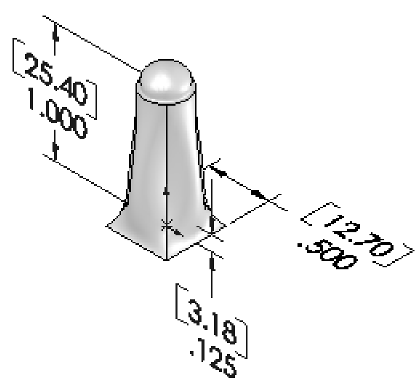
69) Click **Cancel** from the New SolidWorks Document dialog box.



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 only visible when the folder contains a SolidWorks Template document. 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.

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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 SolidWorks User Interface and Part Templates

The SolidWorks 2008 User Interface and CommandManager consist of the following options: Menu bar toolbar, Menu bar menu, Drop-down menus, Short-cut toolbars, Consolidated flyout menus, System feedback icons, Confirmation Corner, and Heads-up View toolbar.

The default CommandManager tabs control the display of the *Features*, *Sketch*, *Evaluate*, *DimXpert*, and *Office Products* toolbars. The tabs are new for 2008.

The FeatureManager design tree consist of four default tabs: *FeatureManager design tree,PropertyManager,ConfigurationManager, and DimXertManager.*

The Task Pane contains the following default tabs: *SolidWorks Resources* , *Design Library* , *File Explorer* , *SolidWorks Search* , *View Palette* , *RealView* , and *Document Recovery* .

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 Menu bar.

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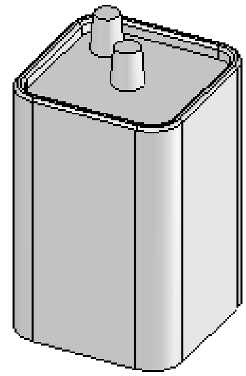
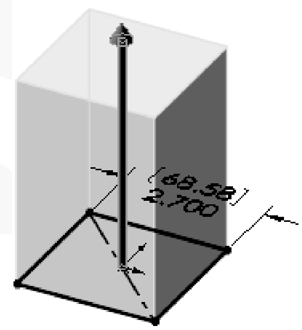
BATTERY Part



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.

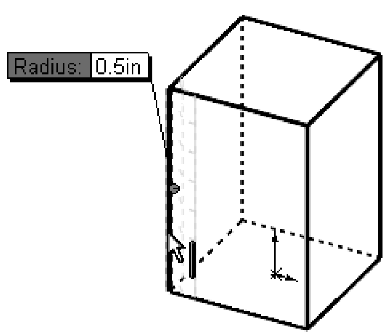
Use features to create parts. Features are building blocks that add or remove material.

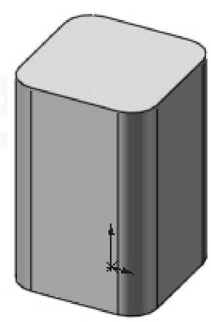


Utilize the Extruded Base feature tool. The Extrude Base feature adds material. The Base feature is the first feature of the part.

Apply symmetry. Sketch a rectangle profile on the Top Plane, centered at the Origin. Extend the profile perpendicular (⊥) to the Top Plane.

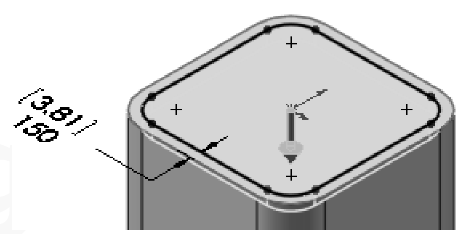
Utilize the Fillet feature tool to round four vertical edges.





The Extruded Cut feature tool removes material from the top face. Utilize the top face for the Sketch plane.

Utilize the Offset Entity Sketch tool to create the profile.



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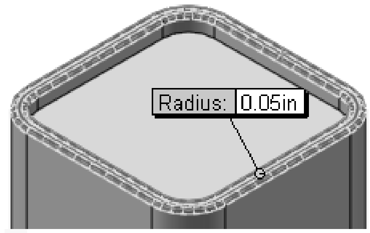
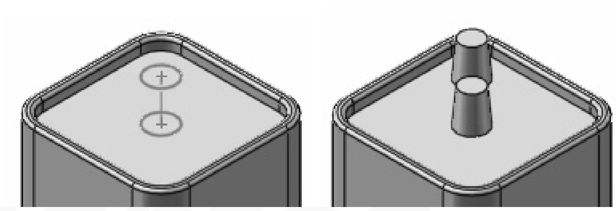
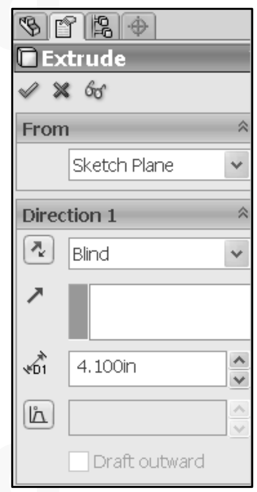
Utilize the Fillet feature tool to round the top narrow face.



Utilize the Extruded Boss feature to add material. Conserve design time. Represent each of the terminals as a cylindrical Extruded Boss feature.



BATTERY Part-Extruded Base Feature



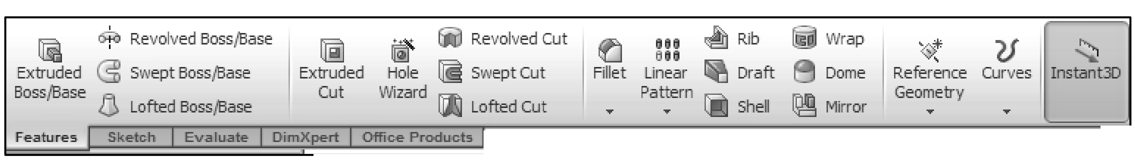
The Extruded Base feature requires:

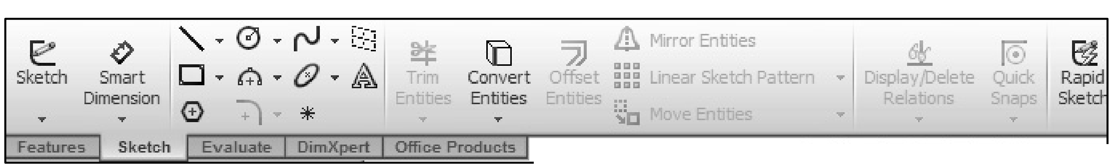
- Sketch plane (Top).
- Sketch Profile (Rectangle).
    - o Geometric relations and dimensions.
- End Condition (Blind is the default End Condition).

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 default Blind End Condition requires a depth value to extrude the 2D sketch profile and to complete the 3D feature.

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





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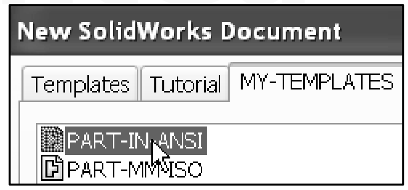
Activity: BATTERY Part

Create a new part.

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

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

72) Double-click **PART-IN-ANSI**, [**PART-MM-ISO**] from the New SolidWorks Document dialog box.



Save the part.

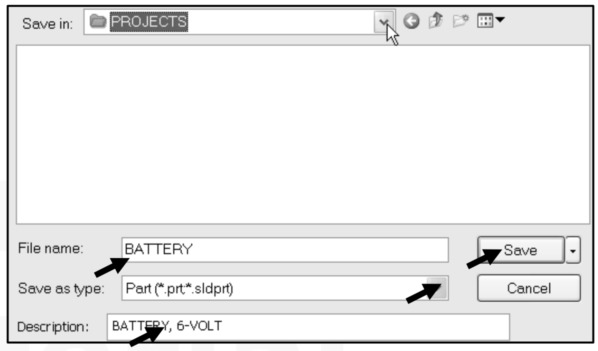
73) Click **Save** from the Menu bar.

74) Select the **SOLIDWORKS-MODELS 2008\PROJECTS** folder.

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

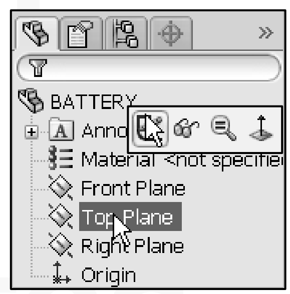
76) Enter **BATTERY**, **6-VOLT** for Description.

77) Click **Save**. The BATTERY FeatureManager is displayed.



Select the Sketch plane.

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



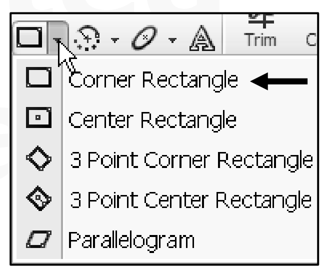
Sketch the profile.

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

80) Click the **Corner Rectangle** Sketch tool from the Consolidated drop-down menu. The Corner Rectangle icon is displayed.

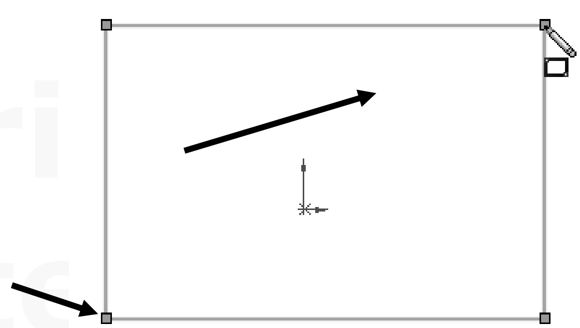
81) Click the **first point** in the lower left quadrant.

82) Drag and click the **second point** in the upper right quadrant as illustrated. The Origin is approximately in the middle of the rectangle.



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

First point



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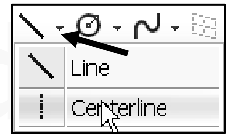
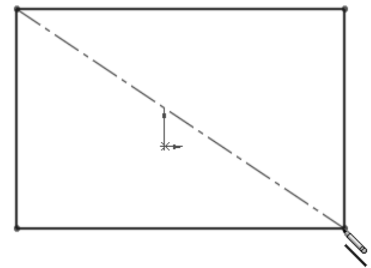


Sketch the centerline.

83) Click the **Centerline** Sketch tool from the Consolidated drop-down menu. The Insert Line PropertyManager is displayed.

84) Sketch a diagonal centerline from the **upper left corner** to the **lower right corner** as illustrated. The endpoints of the centerline are coincident with the corner points of the rectangle.



Deselect the Centerline Sketch tool.

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

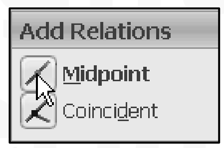
Add a Midpoint relation between the centerline and the Origin.

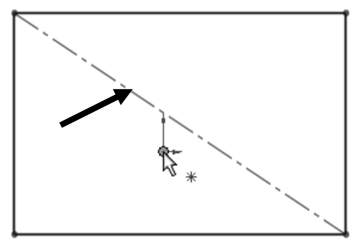
86) Click the **centerline** in the Graphics window.

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

88) Click the **Origin**. The Properties PropertyManager is displayed.

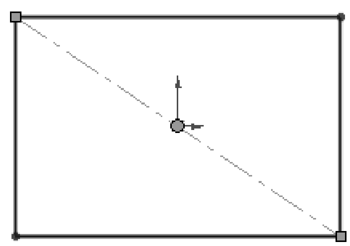
89) Release the **Ctrl** key. The Properties PropertyManager is displayed. The selected entities are displayed.





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

91) Click **OK** from the Properities PropertyManager.



Note: The Line# displayed in the Properties PropertyManager 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.

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

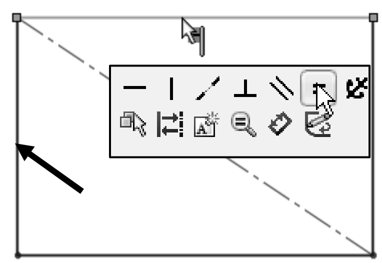
93) Hold the **Ctrl** key down.

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

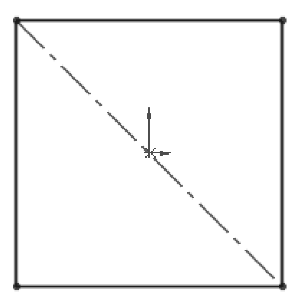
95) Release the **Ctrl** key.

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

97) Click **OK** from the Properties PropertyManager. View the results.



Add Geometric relations directly from the Properties PropertyManager or from the Pop-up shortcut toolbar. The Pop-up shortcut toolbar is new in 2008.



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Add a dimension.

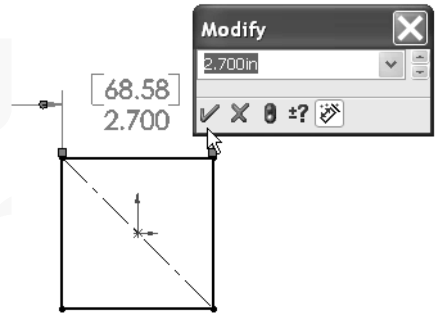
98) Click the **Smart Dimension** Sketch tool. The Smart Dimension icon is displayed.

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

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

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

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



To view the Geometric Sketch relations in the Graphics window, click View, Sketch Relations in the Menu bar. The Geometric relations are displayed.

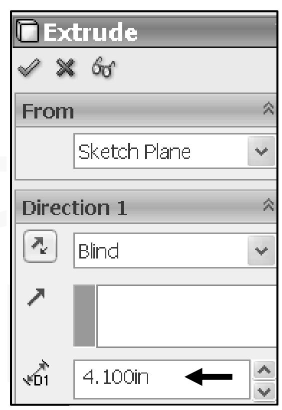
Activity: BATTERY Part-Extruded Base Feature

Insert an Extruded Base feature.

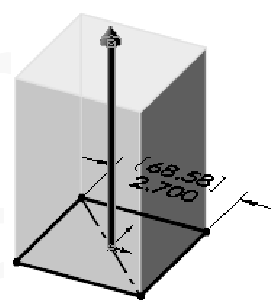
103) Click **Features** tab from the CommandManager. The Features toolbar is displayed.

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

105) Enter **4.100**in, [**104.14**] for Depth.



106) Click **OK** from the Extrude PropertyManager. Extrude1 is displayed in the FeatureManager.



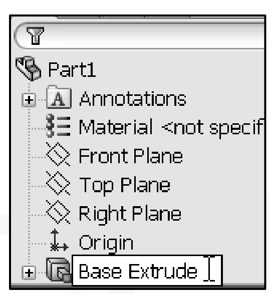
Fit the part to the Graphics window.

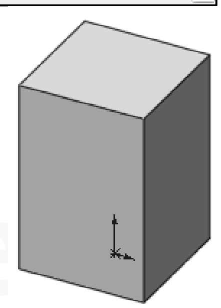
107) Press the **f** key.

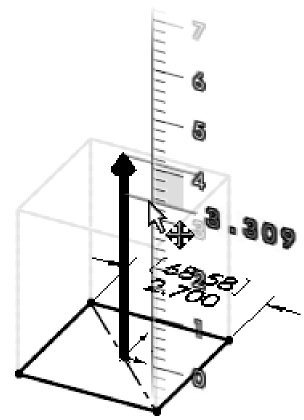
Rename the Extrude1 feature.

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

109) Enter **Base Extrude**.



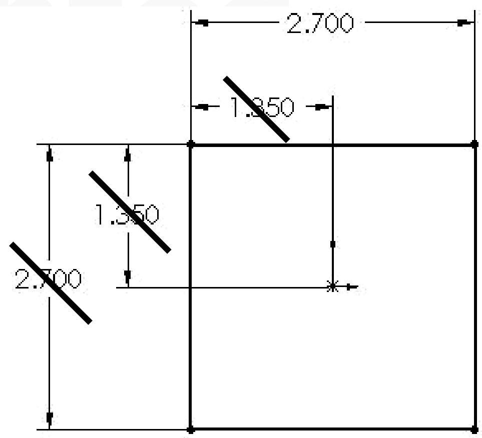




New in 2008 is Instant3D. Instant3D provides the ability to drag geometry and dimension manipulator points to resize features in the Graphics window, and to use on-screen rulers to measure modifications. In this book, you will use the PropertyManager.

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

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.

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

Fillet Feature

The Fillet feature tool removes sharp edges. Use the Hidden Lines Visible view option to display hidden edges of a model.

An edge Fillet requires:

- Edge.
- Fillet radius.

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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.

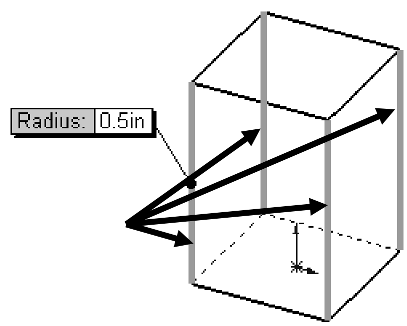
The Fillet feature uses the Fillet PropertyManager. The Fillet PropertyManager provides the ability to select either the Manual or FilletXpert tab. Each tab has a separate menu and option selections. The Fillet PropertyManager displays the appropriate selections based on the type of fillet you create.

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 tabs. Use the Add tab to create new constant radius fillets. The PropertyManager remembers its last used state.

Activity: BATTERY Part-First Fillet Feature

Display the hidden edges.

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



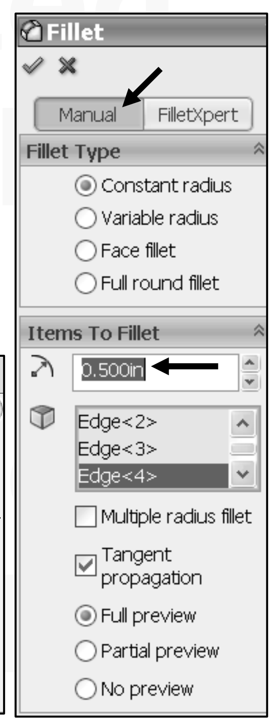
Insert the Fillet feature.

111) Click the **left front vertical edge** of the Base Extrude feature. Note the mouse pointer edge icon.

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

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

114) Click the remaining **3 vertical edges**. The four selected edges are displayed in the Items To Fillet box.



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

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

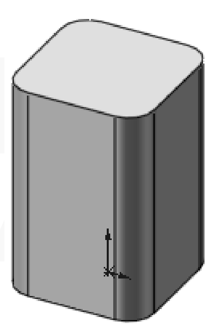
117) Rename **Fillet1** to **Side Fillet**.

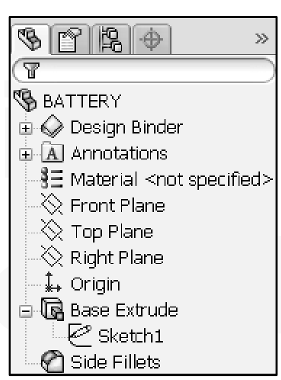
118) Click **Isometric view**.

119) Click **Shaded With Edges**.

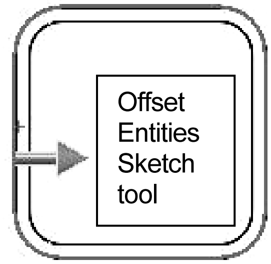
120) Click **Save**.

Note: Select edges to produce the correct result.





Extruded Cut Feature



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

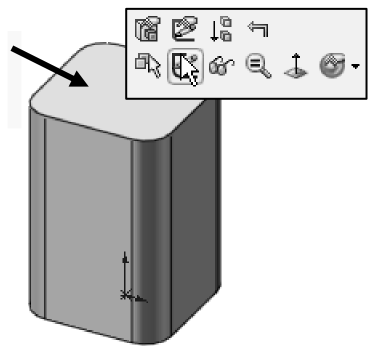
- Sketch plane, (top face).
- Sketch Profile, (Offset Entities).
- End Condition, (Blind is the default End Condition).

The Offset Entities Sketch tool uses existing geometry, extracts an edge or face and locates the geometry on the selected 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-Edge

Select the Sketch plane.

121) Right-click the **Top face** of the BATTERY as illustrated.



Create the sketch.

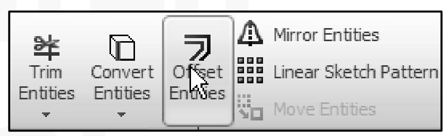
122) Click **Sketch** from the shortcut toolbar. The Sketch toolbar is displayed.

Display the face.

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

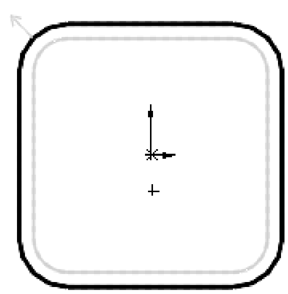
Offset the existing geometry from the boundary of the Sketch plane.

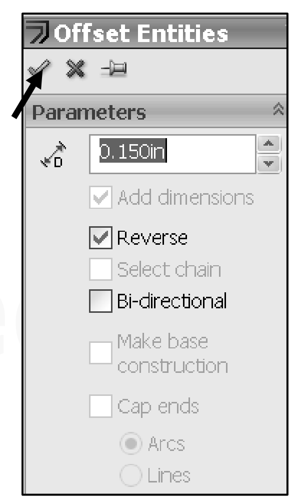
124) Click the **Offset Entities** Sketch tool. The Offset Entities PropertyManager is displayed.



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

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





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

128) Click **Features** tab from the CommandManager. The Features toolbar is displayed.

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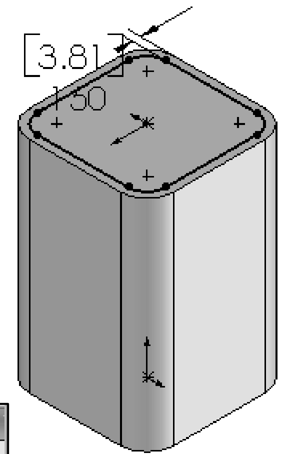


A leading zero is displayed in the spin box. For inch dimensions less than 1, the leading zero is not displayed in the part dimension under the ANSI standard.

Display the profile.

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

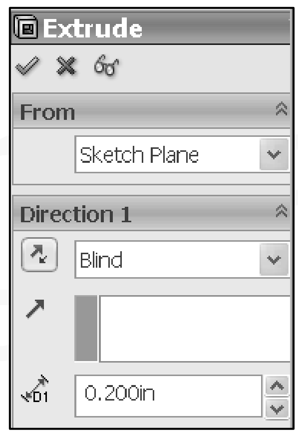
Insert an Extruded Cut feature.

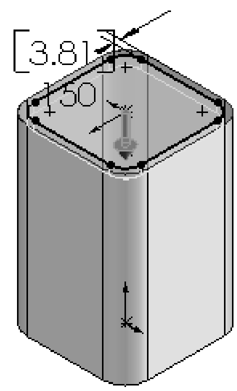
130) Click the **Extruded Cut** feature tool. The Extrude PropertyManager is displayed. Blind is the default End Condition in Direction 1.

131) Enter **.200**in, [**5.08**] for Depth.

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

133) Rename **Extrude2** to **Top Cut** as illustrated.





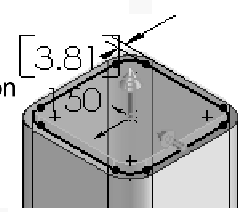
Save the BATTERY part.

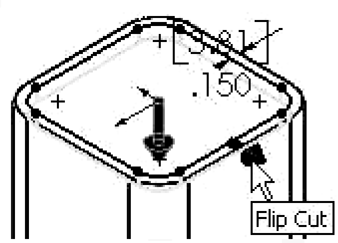
134) Click **Save**.

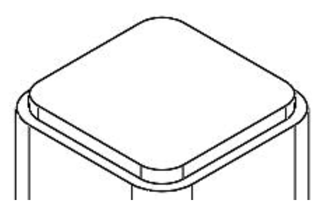
The Extrude PropertyManager contains numerous options. The Reverse Direction option determines the direction of the Extruded Cut. The Extruded Cut is valid when the direction arrow points into material to be removed.

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.

Cut direction not valid.







Extruded Cut with Flip side to cut option checked

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Fillet Feature

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

- Face.
- Fillet Radius.

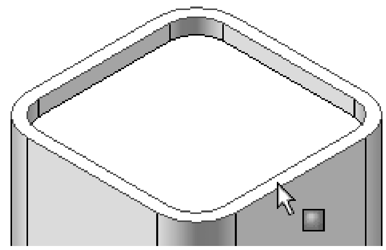
Activity: BATTERY Part-Second Fillet Feature

Insert the Fillet feature on the top face.

135) Use the middle mouse button and **Zoom in** on the Top face of the BATTERY.

136) Click the **top thin face** of the BATTERY as illustrated.

Note: The face icon feedback symbol.



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

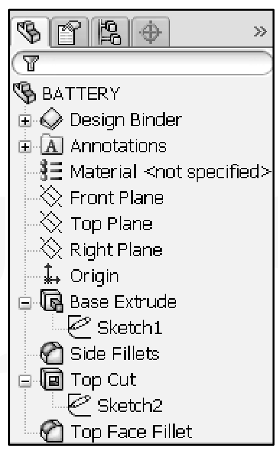
138) Click the **Manual** tab. Create a Constant Radius for Fillet Type. Face<1> is displayed in the Items To Fillet box.

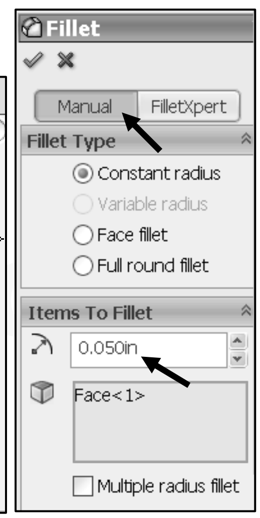
139) Enter **.050**in, [**1.27**] for Radius. Accept the default settings.

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

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

142) Click **Save**.

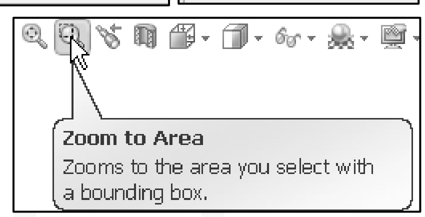




View the mouse pointer for feedback to select Edges or Faces for the Fillet.

Press the z key to Zoom out.

Use the Zoom to Area tool to zoom in on a section of your model.



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

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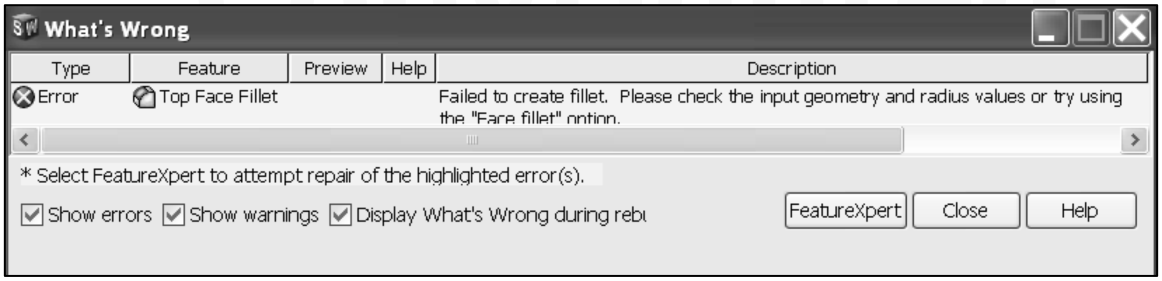
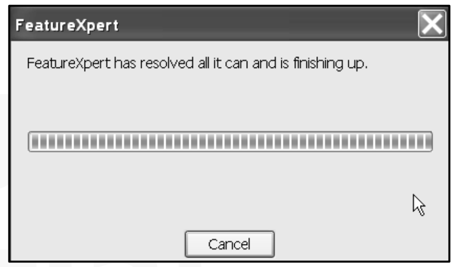
The following error occurs when the Fillet radius is too large for the existing geometry:



Avoid the Fillet Rebuild error. Reduce the fillet size or increase the face width. The FeatureXpert can change the feature order in the FeatureManager design tree or adjust the tangent properties so you can successfully rebuild the part. The FeatureXpert can also, to a lesser extent, repair reference planes that have lost their references. See SolidWorks Help for additional information.



Extruded Boss Feature

The Extruded Boss feature tool 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:

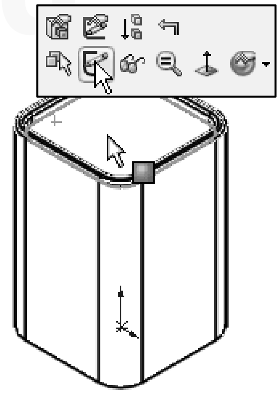
For inches: 4.500in – (4.100in height – .200in Offset cut depth) = .600in. Note: The depth of the extrusion is .600in.

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

Activity: BATTERY Part-Extruded Boss Feature

Select the Sketch plane.

143) Right-click the **top face** of the Top Cut feature. The top face is your Sketch plane.



Create the sketch.

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

Display the Sketch plane.

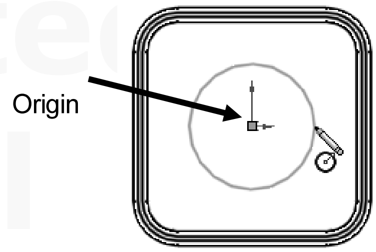
145) Click **Top view** from the Heads-up View toolbar.

Sketch the profile.

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

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

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



Add a dimension.

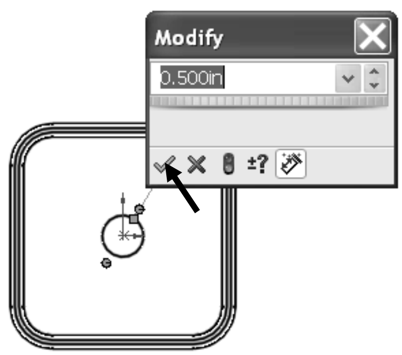
149) Click the **Smart Dimension** Sketch tool. The Smart Dimension icon is displayed.

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

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

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

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



Deselect the Smart Dimensions Sketch tool.

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

Copy the sketched circle.

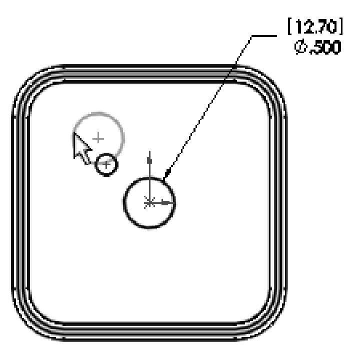
155) Hold the **Ctrl** key down.

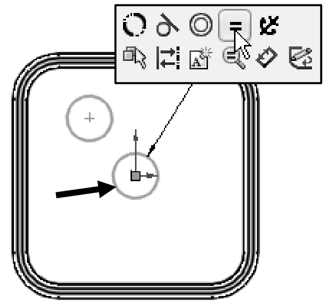
156) Click the **circumference** of the circle.

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

158) Release the **mouse button**.

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





Add an Equal relation.

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

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

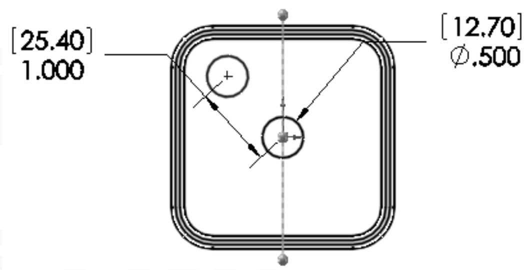
162) Right-click **Make Equal** from the shortcut toolbar.

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

Show the Right Plane for the dimension reference.

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

165) Click **Show**. The Right Plane is displayed.



Add a dimension.

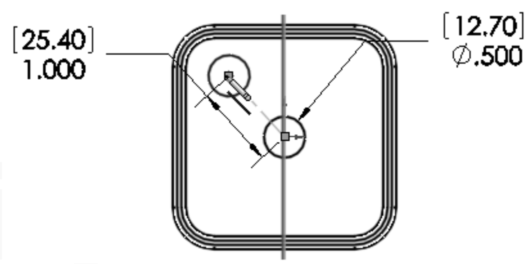
166) Click the **Smart Dimension** Sketch tool.

The Smart Dimension icon is displayed.

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

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

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

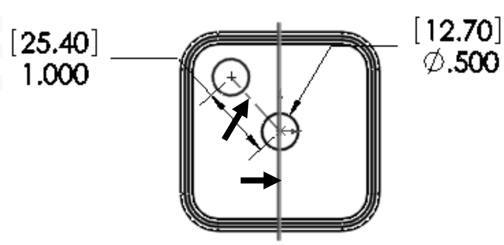


170) Click the **Green Check mark** from the Modify dialog box.

Add a centerline.

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

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



Deselect the Centerline Sketch tool.

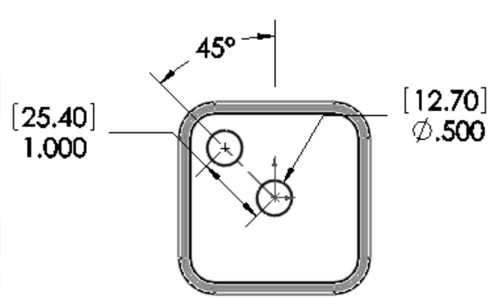
173) Right-click **End Chain**.

Add a dimension.

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

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

176) Click **Right Plane** from the fly-out FeatureManager or from the Graphics window.



177) Click a **position** between the centerline and the Right Plane, off the profile as illustrated.

178) Enter **45**deg.

179) Click the **Green Check mark** from the Modify dialog box.

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.

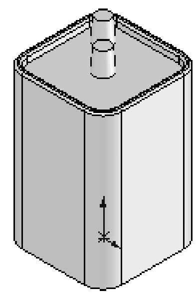
180) Click the **Extruded Boss/Base** feature tool. The Extrude PropertyManager is displayed. Blind is the default End Condition.

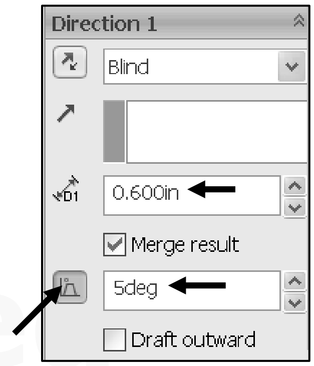
181) Enter **.600**in, [**15.24**] for Depth. Click the **Draft ON/OFF** button. Enter **5**deg in the Draft Angle box.

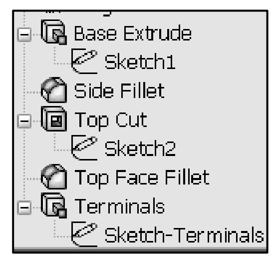
182) Click **OK** from the Extrude PropertyManager. Extrude3 is displayed.

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

184) Click **Hide** .







Rename the Feature and Sketch.

185) Rename **Extrude3** to **Terminals**.

186) **Expand** Terminals from the FeatureManager. 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 this text.

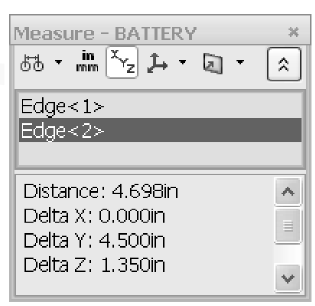
Rename features with descriptive names. Standardize on feature names that are utilized in mating parts. Example: Mounting Holes.

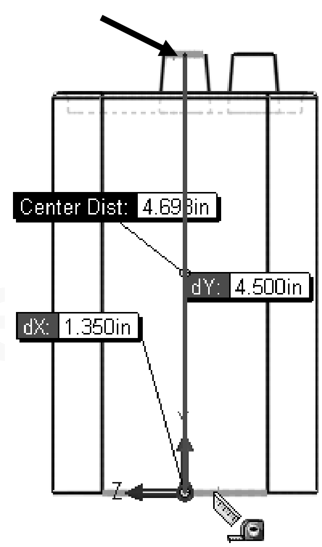
Measure the overall BATTERY height.

187) Click **Right view** .

188) Click **Measure** from the Evaluate tab in the CommandManager. Click the **top edge** of the BATTERY terminal.

189) Click the **bottom edge** of the BATTERY. View the results.





190) Click **Close** from the Measure – BATTERY dialog box.

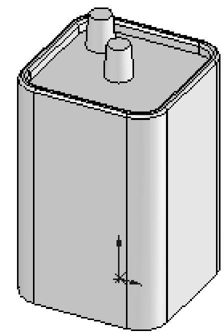
Right-click Clear Selections in the Selected items block to measure the distance between various edges or faces.

Hide all planes and display a Trimetric view.

191) Click **View**, uncheck **Planes** from the Menu bar.

192) Click **Trimetric view** from the Heads-up Views toolbar.

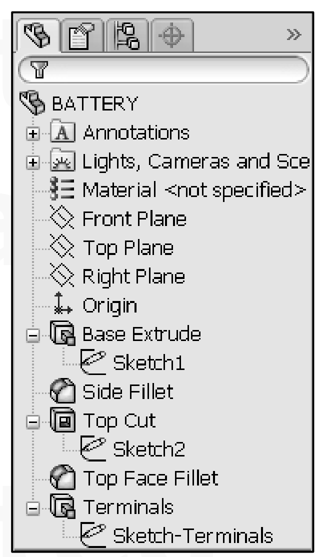
193) Click **Save** .



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.

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 at 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 copied 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.

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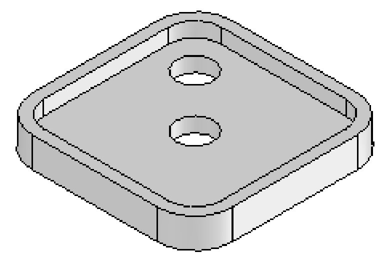
BATTERYPLATE Part

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 Boss\Base features.

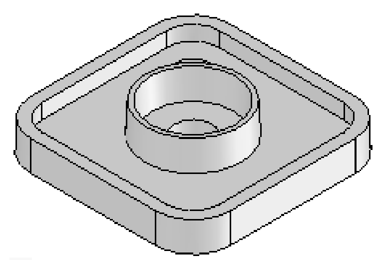
Edit the BATTERY features. Create two holes from the original sketched circles.



Use the Extruded Cut feature tool. Modify the dimensions of the Base feature. Add a 3° draft angle.

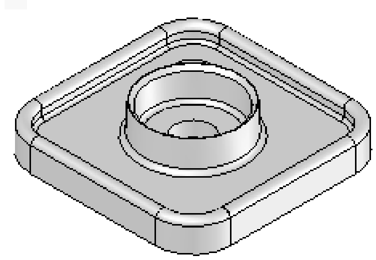
Note: A sand pail contains a draft angle. The draft angle assists the sand to leave the pail when the pail is flipped upside down.

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 a 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.

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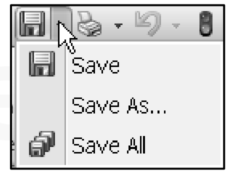
Save As, Delete, Modify, and Edit Feature



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

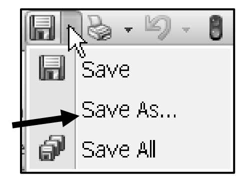
Reuse existing geometry. Create two holes. Delete the Terminals feature and reuse the circle sketch. Select the sketch in the FeatureManager. Insert an Extruded Cut feature. The Through All End Condition option creates two holes that cut through the entire Extruded Base feature.

Right-click the Extruded Cut feature from the FeatureManager. Select the Edit Feature option. The Edit Feature option returns to the Extrude PropertyManager. Modify the End Condition from Blind to Through All. Modify the depth dimension. 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.

194) Click **Save As** from the Consolidated Menu bar.



195) Select the **PROJECTS** folder.

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

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

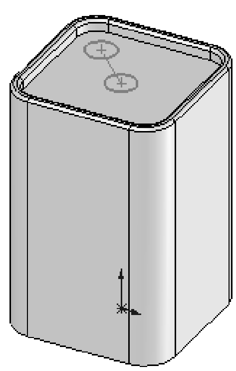
198) Click **Save**. The BATTERYPLATE FeatureManager is displayed. The BATTERY part is closed.

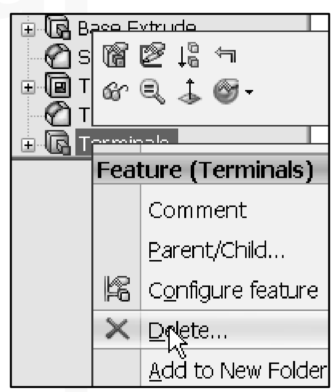
Delete the BATTERY Terminals.

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

200) Click **Delete** Delete....

201) Click **Yes** from the Confirm Delete box. Do not delete Sketch-Terminals.





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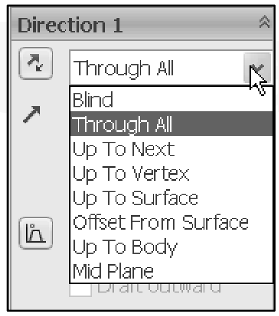
Activity: BATTERYPLATE Part-Extruded Cut Feature

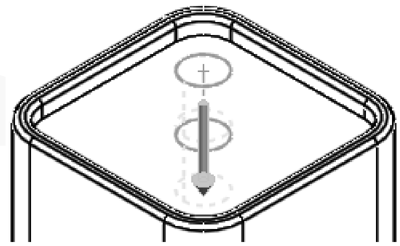
Create an Extruded Cut feature from the Sketch–Terminals.

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

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

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





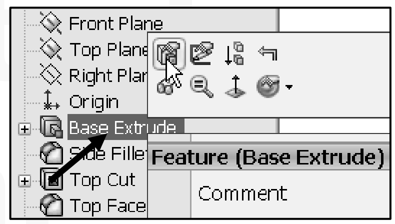
205) Click **OK** from the Extrude PropertyManager. The Exturde feature is displayed.

206) Rename the **Extrude** feature to **Holes**.

207) Click **Save**.

Edit the Base Extrude feature.

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



209) Click **Edit Feature**. The Base Extrude PropertyManager is displayed.

Modify the overall depth and draft.

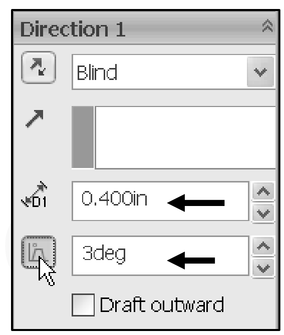
210) Click the **4.100**in, [**104.14**] dimension.

211) Enter **.400**in, [**10.16**] for new Depth.

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

213) Enter **3**deg in the Draft Angle box.

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

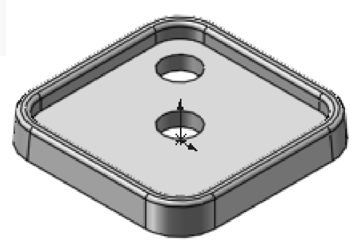


Fit the model to the Graphics window.

215) Press the **f** key.

Save the BATTERYPLATE part.

216) Click **Save**.



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

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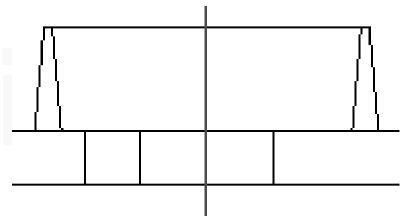
Extruded Boss Feature

The Holder is created with a circular Extruded Boss feature. Utilize the Offset Sketch Entity to create the second circle. Utilize a draft angle of 3° in the Extrude PropertyManager. Note: 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°

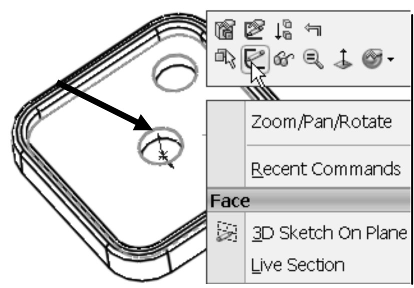


Plastic parts require a draft angle. A rule of thumb; 1deg to 5deg is the draft angle. The draft angle is created in the direction of pull from the mold. This is defined by geometry, material selection, mold production and cosmetics. Always verify the draft with the mold designer and manufacturer.

Activity: BATTERYPLATE Part-Offset Entities

Select the Sketch plane.

217) Right-click the **top face** of the BATTERYPLATE part.



Create the sketch.

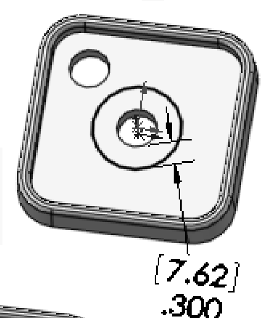
218) Click **Sketch** from the shortcut toolbar. The Sketch toolbar is displayed.

219) Click the **top circular edge** of the center hole as illustrated.

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

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

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



Create the second offset circle.

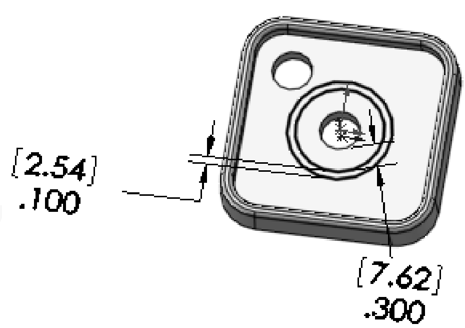
223) Click the **offset circle**.

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

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

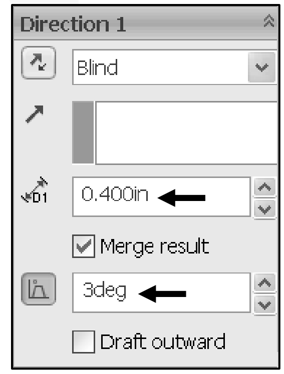
226) Enter **.100**in, [**2.54**] for the Offset Distance. Accept the default settings.

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



Activity: BATTERYPLATE Part-Extruded Boss Feature

Insert an Extruded Boss feature.



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

229) Enter **.400**in, [**10.16**] for Depth.

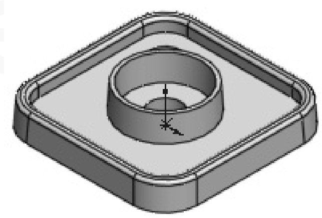
230) Click the **Draft ON/OFF** button.

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

232) Click **OK** from the Extrude PropertyManager. Extrude5 is displayed in the FeatureManager

233) Rename **Extrude5** to **Holder**.

Display an Isometric view.



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

Save the BATTERYPLATE part.

235) Click **Save** .

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

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

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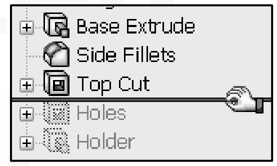
Activity: BATTERYPLATE Part-Fillet Features: Full Round, Multiple Radius Options

Delete the Top Edge Fillet.

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



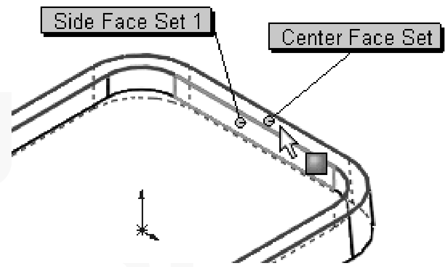
237) Click **Delete** Delete... .

238) Click **Yes**.

239) Drag the **Rollback** bar below Top Cut in the FeatureManager as illustrated.

Insert a Full Round Fillet feature type.

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



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

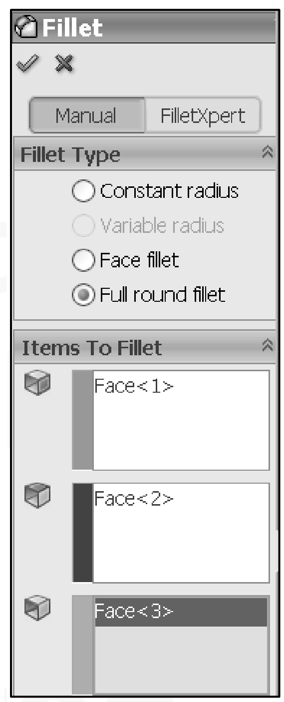
242) Click the **Manual** tab.

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

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

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

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



Rotate the part.

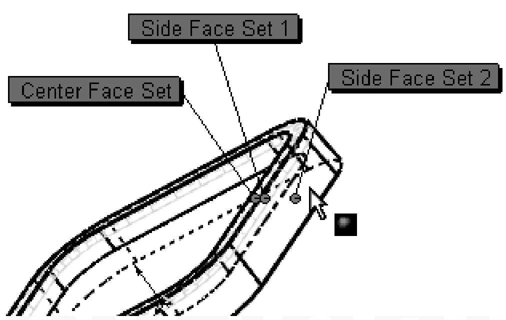
247) Press the **left arrow** key until you can select the outside Base Extrude face. Note: Use the middle mouse button to rotate a model. The rotate icon is displayed.

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

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

250) Click **OK** from the Fillet PropertyManager. The Fillet feature is displayed in the FeatureManager.

251) Rename **Fillet3** to **TopFillet**.



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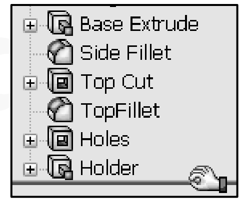


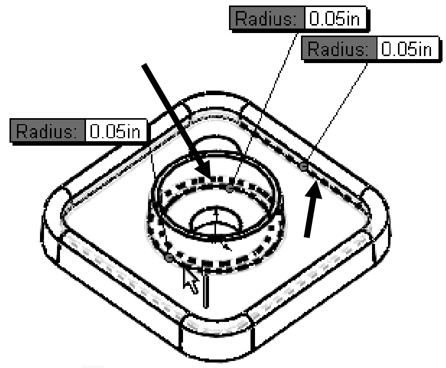
Display an Isometric view.

252) Click **Isometric view**.

253) Click **Hidden Lines Removed**.

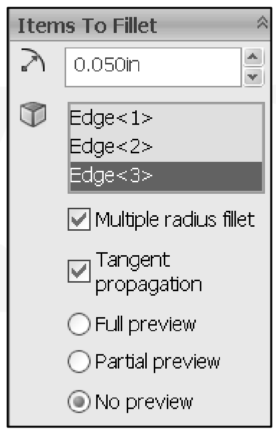
254) Drag the **Rollback** bar back below Holder in the FeatureManager.

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



Insert a Multiple Radius Fillet feature.

255) Click the **bottom outside circular edge** of the Holder as illustrated.

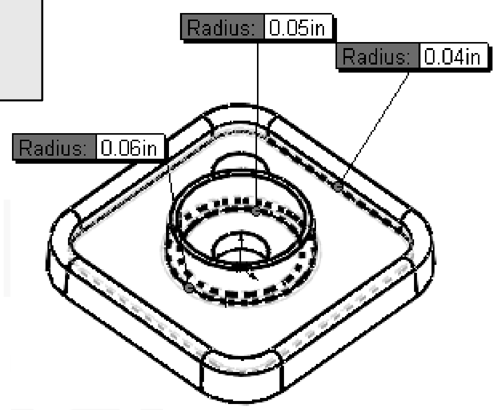


256) Click the **Fillet** feature tool. The Fillet PropertyManager is displayed.

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

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

259) Click the **inside edge** of the Top Cut. The selected edges are displayed in the Items To Fillet box.



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

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

Modify the Fillet values.

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

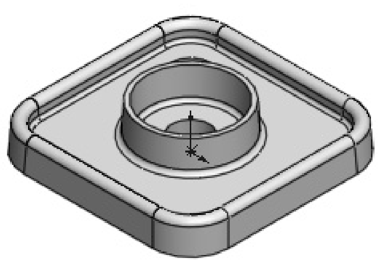
263) Enter **0.060**in, [**1.52**].

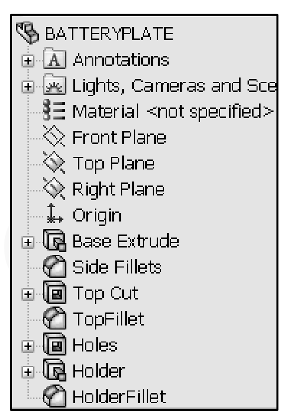
264) Click the **Radius** box Radius: 0.05in for the Top Cut inside edge.

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

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

267) Rename **Fillet4** to **HolderFillet**.





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Group the Fillets into a folder.

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

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

270) Click **HolderFillet** from the FeatureManager.

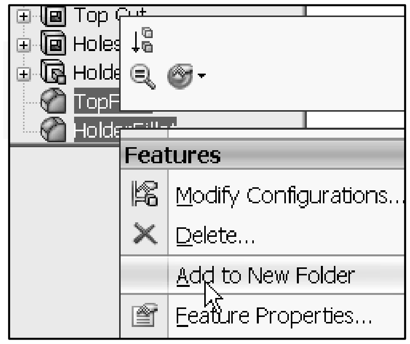
271) Hold the **Ctrl** key down.

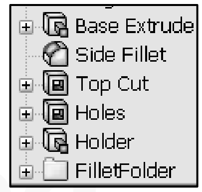
272) Click **Top Fillet** from the FeatureManager.

273) Release the **Ctrl** key.

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

275) Rename **Folder1** to **FilletFolder**.





Save the BATTERYPLATE.

276) Click **Save**.

Chamfer Feature

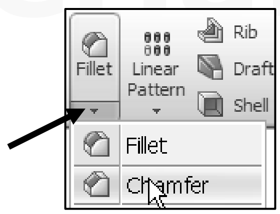
The Chamfer feature tool is located in the Consolidated drop-down menu as illustrated.

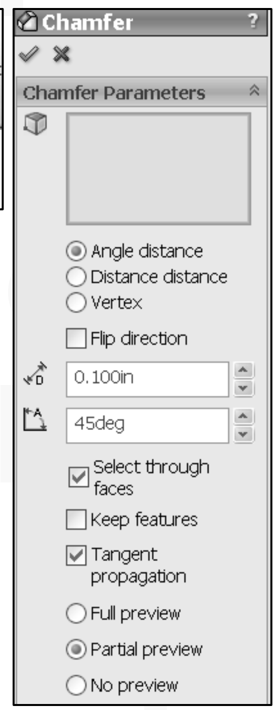
A Chamfer feature tool bevels an edge or a face. There are three options for the Chamfer feature:

- Angle distance, (default setting).
- Distance distance.
- Vertex.

The Chamfer feature for the Holder requires an:

- Edge or face.
- Angle and distance.





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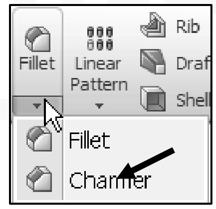
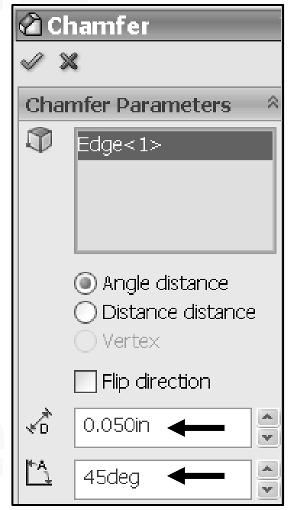
Activity: BATTERYPLATE Part-Chamfer Feature

Insert a Chamfer feature.

277) Click the **inside circular edge** of the Holder as illustrated.

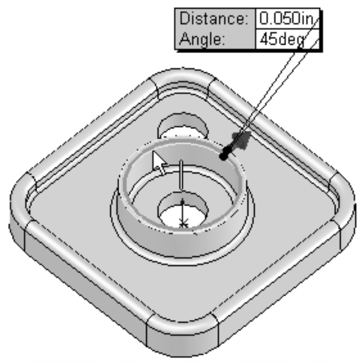
278) Click the **Chamfer** feature tool from the Consolidated drop-down menu. The Chamfer PropertyManager is displayed.





279) Enter **.050**in, [**1.27**] for Distance.

280) Enter **45**deg for Angle. Accept the default settings.

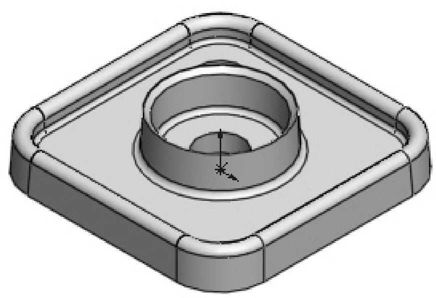


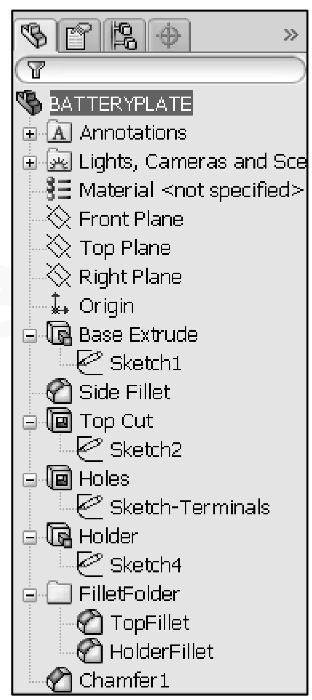
281) Click **OK** from the Chamfer PropertyManager. Chamfer1 is displayed in the FeatureManager.

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

Save the BATTERYPLATE.

283) Click **Save**. View the model and the Part FeatureManager.





Exit SolidWorks.

284) Click **File**, **Exit** from the Menu bar.

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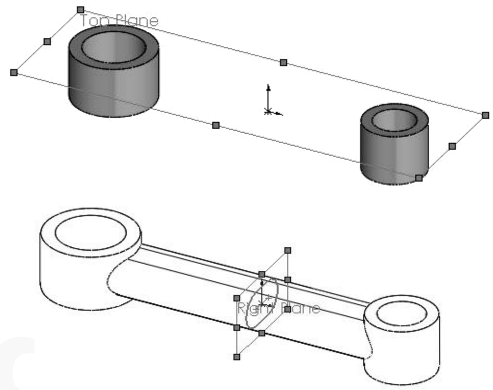
Multi-body Parts and Extruded Boss Feature

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

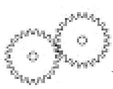
A WRENCH consists of two cylindrical bodies. Each extrusion is a separate body. The oval profile is sketched on the right plane and extruded with the Up to Body option.

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 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 Offset Entities Sketch tool created the circular ring profile.

Multi radius Edge Fillets and Face Fillets removed sharp edges. Similar Fillets were grouped together into a folder. Features were renamed in the FeatureManager. The BATTERY and BATTERYPLATE utilized an Extruded Base feature.

Project Summary

SolidWorks is a design software application used to model and create 2D and 3D sketches, 3D parts, assemblies, and 2D drawings. In Project 1 you were introduced to the SolidWorks 2008 User Interface and CommandManager: Menu bar toolbar, Menu bar menu, Drop-down menus, Short-cut toolbars, Consolidated drop-down menus, System feedback icons, Confirmation Corner, Heads-up View toolbar, and Document Properties.

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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.

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 Part Template.

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: Smart Dimension, Sketch Entities, Line, Corner Rectangle, Circle, 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.

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Project Terminology

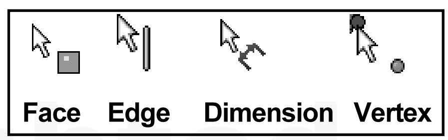
Assembly: An assembly is a document in which parts, features, and other assemblies (sub-assemblies) are put together. A part in an assembly is called a component. Adding a component to an assembly creates a link between the assembly and the component. When SolidWorks opens the assembly, it finds the component file to show it in the assembly. Changes in the component are automatically reflected in the assembly. The filename extension for a SolidWorks assembly file name is .SLDASM. The FLASHLIGHT is an assembly. The BATTERY is a part/component in the FLASHLIGHT assembly.

Chamfer: The chamfer tool creates a beveled feature on selected edges, faces, or a vertex.

CommandManager: The CommandManager is a context-sensitive toolbar that dynamically updates based on the toolbar you want to access. By default, it has toolbars embedded in it based on the document type. When you click a tab below the Command Manager, it updates to show that toolbar. For example, if you click the **Sketches** tab, the Sketch toolbar is displayed.

Convert Entities: A sketch tool that extracts sketch geometry to the current Sketch plane. You can create one or more curves in a sketch by projecting an edge, loop, face, curve, or external sketch contour to the selected Sketch plane.

Cursor Feedback: The system feedback symbol indicates what you are selecting or what the system is expecting you to select. As you move the mouse pointer across your model, system feedback is provided.



Dimension: A value indicating the size of the 2D sketch entity or 3D feature.

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 Sketch 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: In SolidWorks, Geometric relations between sketch entities and model geometry, in either 2D or 3D sketches, are an important means of building in design intent. Example: Concentric, Tangent, Vertical, etc.

Menus: Menus, (drop-down, pop-out) provides 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. When you create mirrored entities, the SolidWorks software applies a Symmetric relation between each corresponding pair of sketch points (the ends of mirrored lines, the centers of arcs, and so on). If you change a mirrored entity, its mirror image also changes.

Mouse Buttons: The left, middle, and right mouse buttons have distinct functions in SolidWorks. The left mouse button is utilized to select geometry. The right-mouse button is utilized to invoke commands. The middle button is used to rotate and Zoom in and Zoom out.

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.

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, corner rectangles, circles, polygons, and ellipses.

Status of a Sketch: Three states are utilized in this Project: *Fully Defined*: Has complete information, (Black), *Over Defined*: Has duplicate dimensions, (Red), or *Under Defined*: There is inadequate definition of the sketch, (Blue).

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.

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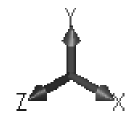
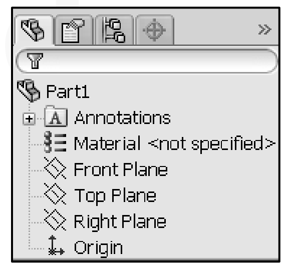
Questions

1. Identify and describe the function of the following features:
    - Extruded Boss/Base.
    - Fillet.
    - Chamfer.
    - Extruded Cut.
2. Explain the differences between a Template and a Part.



3. Explain the steps in starting a SolidWorks session.
4. Describe the procedure to develop a new 2D sketch.



5. Explain the procedure required to change part unit dimensions from inches to millimeters.
6. Identify the three default Reference planes.
7. What is a Base feature? Provide an example.
8. Describe the differences between an Extruded Base feature, an Extruded Cut feature and a Fillet 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. Describe is a draft angle. Provide an example.
16. An arc requires _______ points?
17. Identify the properties of a Multi-body part.

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18. Identify the name of the following Feature tool icons.

A B C D E F

A	B	C	D
E	F		

19. Identify the name of the following Sketch tool icons.

A B C D E F G H I J

A	B	C	D
E	F	G	H
I	J		

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Exercises

Exercise 1.1: Identify the Sketch plane for the Extrude1 feature as illustrated.

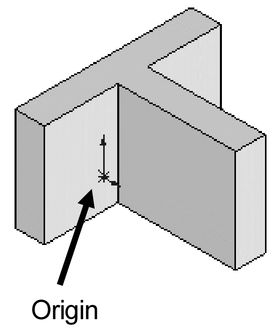
A: Top Plane

B: Front Plane

C: Right Plane

D: Left Plane

Correct answer _______.



Exercise 1.2: Identify the Sketch plane for the Extrude1 feature as illustrated.

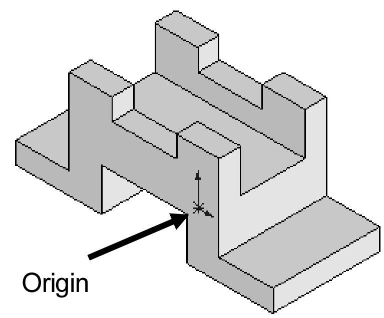
A: Top Plane

B: Front Plane

C: Right Plane

D: Left Plane

Correct answer ________.



Exercise 1.3: Identify the Sketch plane for the Extrude1 feature as illustrated.

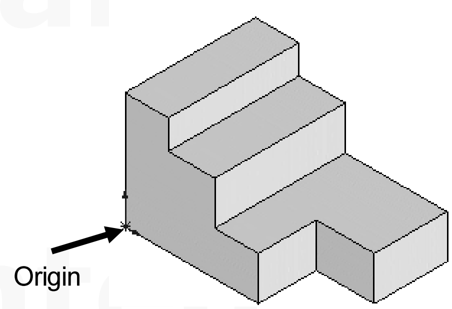
A: Top Plane

B: Front Plane

C: Right Plane

D: Left Plane

Correct answer ________.



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Exercise 1.4: Part Document Templates

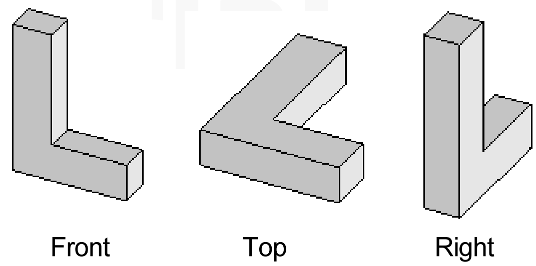
Create a Metric part document template using an ANSI dimension standard.

Exercise 1.5: L-SHAPE Part

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

Utilize your own dimensions.

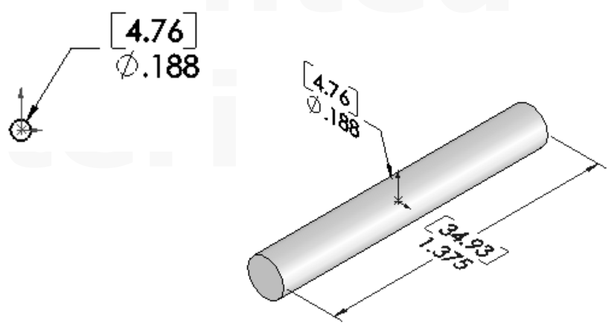
Locate each profile on a different Sketch Plane as illustrated.



Exercise 1.6: AXLE

Create an AXLE part as illustrated with dual units.

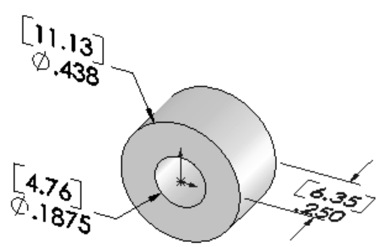
- Utilize the Front Plane for the Sketch plane.
- Utilize the Mid Plane End Condition. The AXLE is symmetric about the Front Plane. Note the location of the Origin.
- Apply 6061 Alloy as a material.
- Display dual dimensions.



Exercise 1.7: SHAFT-COLLAR

Create a SHAFT-COLLAR part as illustrated with dual units.

- Utilize the Front Plane for the Sketch plane. Note the location of the Origin.
- Use the provided dimenions.
- Note the location of the Origin.
- Apply 2014 Alloy as a material.
- Apply a color to the part.
- Display dual dimensions.

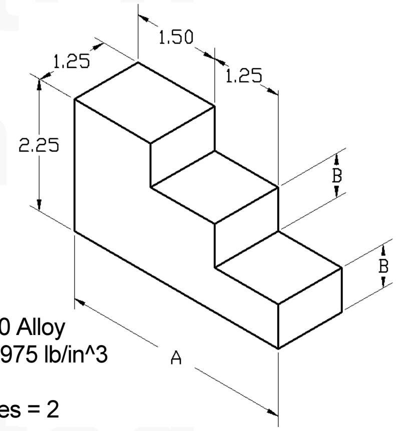


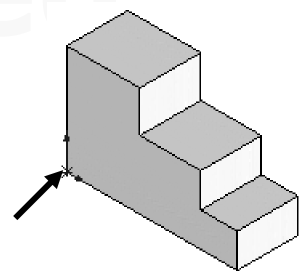
Exercise 1.8

Create the illustrated part. Note the location of the Origin.

- Calculate the overall mass of the illustrated model.
- Apply the Mass Properties tool.
- Think about the steps that you would take to build the model.
- Review the provided information carefully.
- Units are represented in the IPS, (inch, pound, second) system.
- A = 3.50in, B = .70in

Given:
A = 3.50
B = .70
Material: 1060 Alloy
Density = 0.0975 lb/in^3
Units: IPS
Decimal places = 2



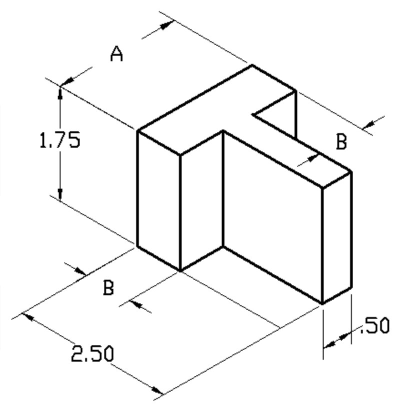


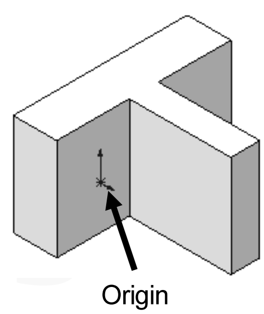
Exercise 1.9

Create the illustrated part. Note the location of the Origin.

- Calculate the overall mass of the illustrated model.
- Apply the Mass Properties tool.
- Think about the steps that you would take to build the model.
- Review the provided information carefully. Units are represented in the IPS, (inch, pound, second) system.
- A = 3.00in, B = .75in

Given:
A = 3.00
B = .75
Material: Copper
Density = 0.321 lb/in^3
Units: IPS
Decimal places = 2





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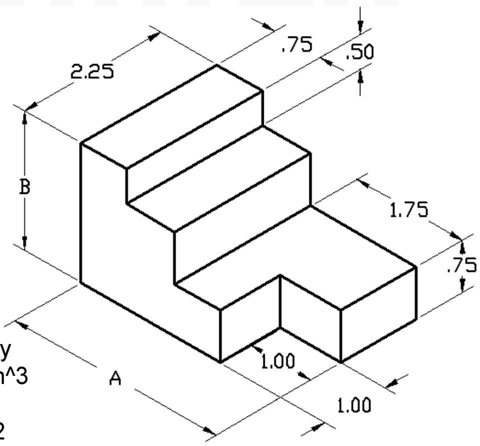
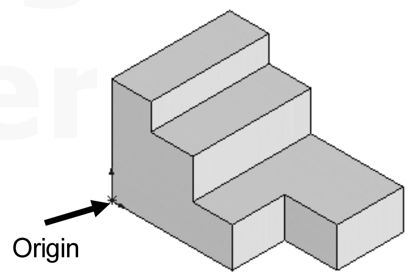
Exercise 1.10

Create the illustrated part. Note the location of the Origin.

- Calculate the volume of the part and locate the Center of mass with the provided information.
- Apply the Mass Properties tool.
- Think about the steps that you would take to build the model.
- Review the provided information carefully.



Given:
A = 3.30
B = 2.00
Material: 2014 Alloy
Density = .101 lb/in^3
Units: IPS
Decimal places = 2



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