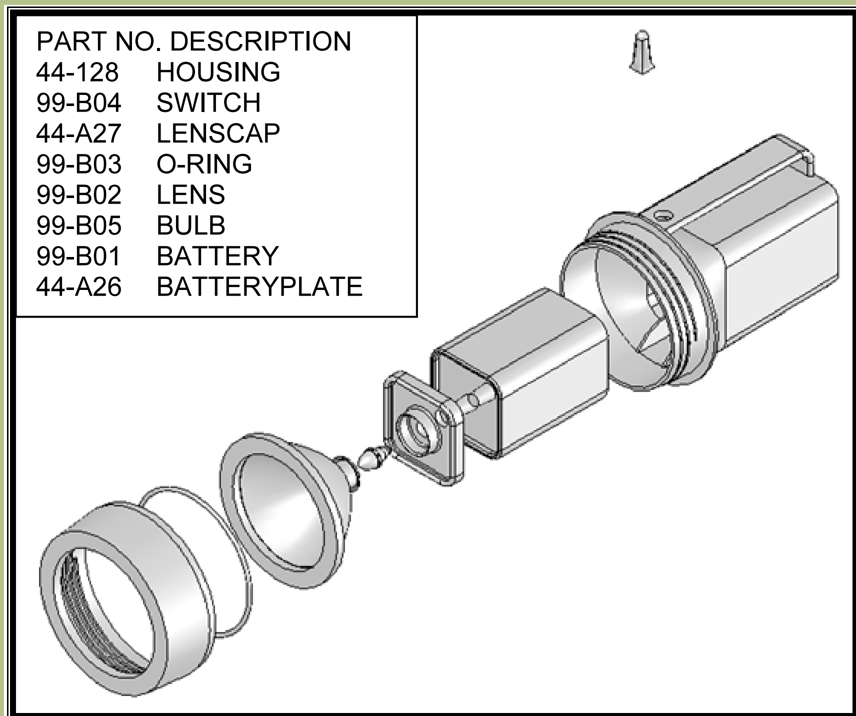
INSIDE:


# SolidWorks 2009: The Basics

## with Multimedia CD

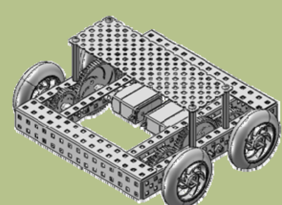
A Step-by-Step Project Based Approach

Introductory Level ♦ Tutorial Style ♦ Video Instruction



| PART NO. | DESCRIPTION |
|---|---|
| 44-128 | HOUSING |
| 99-B04 | SWITCH |
| 44-A27 | LENSCAP |
| 99-B03 | O-RING |
| 99-B02 | LENS |
| 99-B05 | BULB |
| 99-B01 | BATTERY |
| 44-A26 | BATTERYPLATE |

David C. Planchard & Marie P. Planchard CSWP



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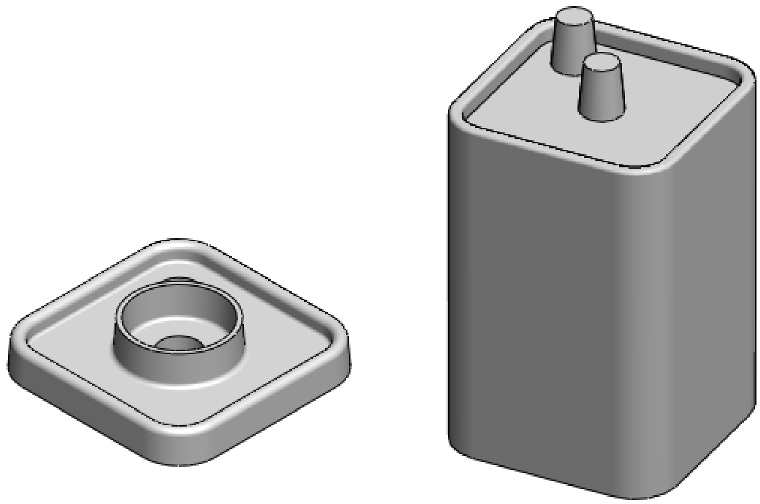
# Project 1

## 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 2009 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:<br>o PART-IN-ANSI.<br>o PART-MM-ISO. | • Skill to address System Options and Document Properties. |
| • Create two FLASHLIGHT Parts:<br>o BATTERY.<br>o BATTERYPLATE. | • Specific knowledge and understanding of 2D sketching and the following 3D features: Extruded Base, Extruded Boss, Extruded Cut, and Fillet. |

## Notes:

# Project 1-Introduction to Part Modeling

## Project Overview

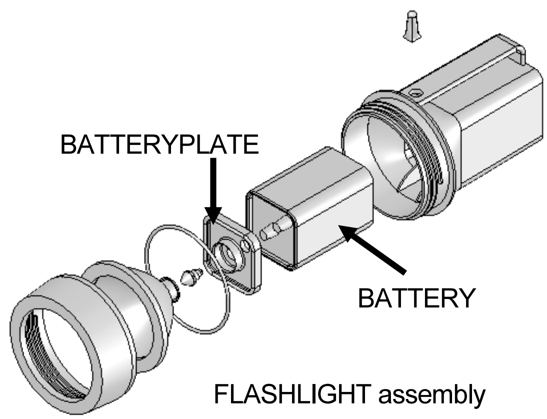
SolidWorks is a design software application used to model and create 2D and 3D sketches, 3D parts, 3D assemblies, and 2D drawings. Project 1 introduces you to the SolidWorks 2009 User Interface and CommandManager: Menu bar toolbar, Menu bar menu, Drop-down menus, Context toolbars, Consolidated drop-down menus, System feedback icons, Confirmation Corner, Heads-up View toolbar, Document Properties and more.

A Template is the foundation for a SolidWorks document. Templates are part, drawing, and assembly documents that include user-defined parameters and are the basis for new documents. Create two part templates:

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

Create two parts for the FLASHLIGHT assembly in this project:

- BATTERY
- BATTERYPLATE



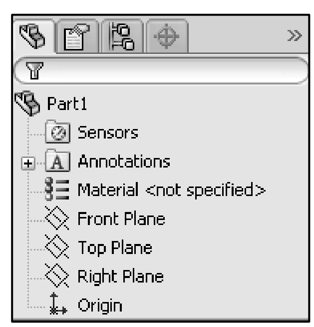

FLASHLIGHT assembly

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

A 2D sketch is required to create an Extruded Base feature. Utilize the sketch geometry and sketch tools to create the following features: *Extruded Base*, *Extruded Boss*, *Extruded Cut*, and *Fillet*.

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

- Establish a SolidWorks session.
- Comprehend the SolidWorks 2009 User Interface.
- Recognize the default Reference Planes.
    - o Front, Top, and Right
- Insert a new 2D sketch and add sketch geometry with the following sketch tools: *Line*, *Center Rectangle*, *Circle*, *Convert Entities*, *Offset Entities*, and *Mirror Entities*.
- Establish Geometric relations, dimensions, and determine the status of the sketch.
    - o Under defined, Fully defined, Over defined



- Utilize the Instant3D tool to create an Extruded Base feature.
- Utilize the Save As, Delete, Edit Feature, and Modify tools.
- 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 first folder named SOLIDWORKS-MODELS 2009. Create two sub-folders named MY-TEMPLATES and PROJECTS.

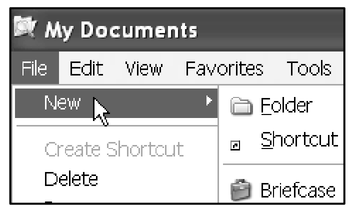
### Activity: File Management

Create a new folder in Windows.

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

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

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




Enter the new folder name.

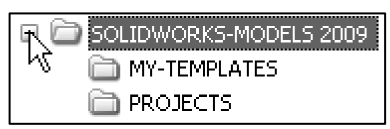
**4)** Enter **SOLIDWORKS-MODELS 2009**.

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 2009** 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 2009** folder.

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

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

Return to the SOLIDWORKS-MODELS 2009 folder.

**10)** Click the **SOLIDWORKS-MODELS 2009** 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 2009 session.

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

**12)** Click **All Programs**.

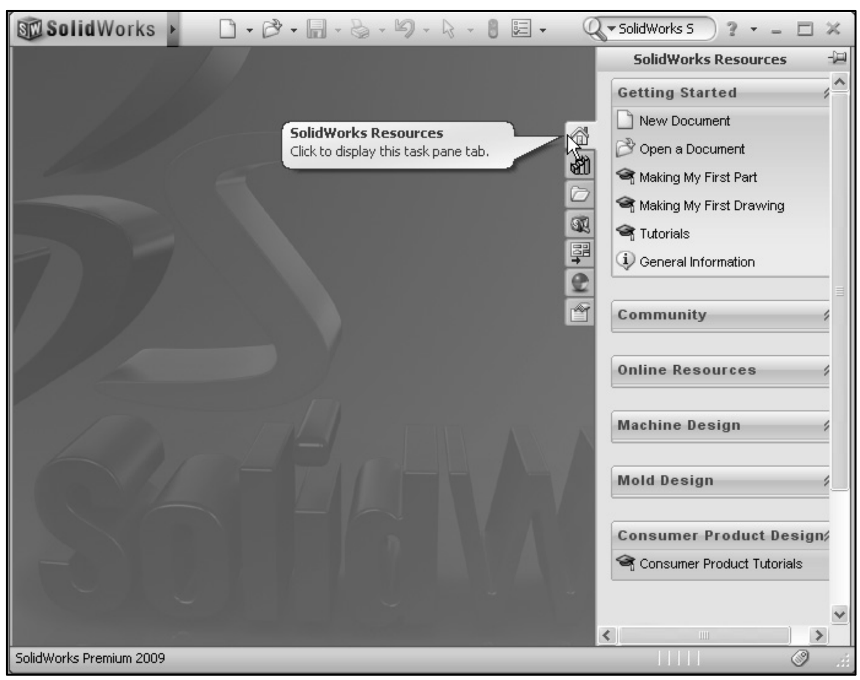
**13)** Click the **SolidWorks 2009** folder.

**14)** Click **SolidWorks 2009** application. The SolidWorks program window opens. Note: Do not open a document at this time.

If available, double-click the SolidWorks 2009 icon on the Windows Desktop to start a SolidWorks session.


Read the Tip of the Day dialog box.

**15)** 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.



## Activity: Understanding the SolidWorks UI and CommandManager

### Menu bar toolbar


SolidWorks 2009 (UI) is design 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:

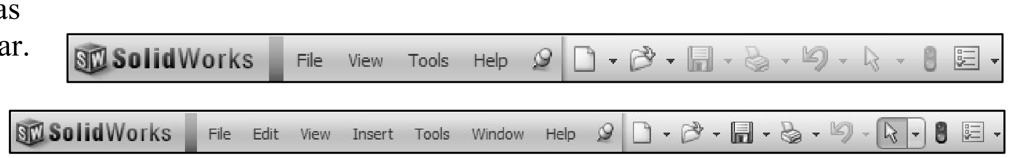
- **New** – Creates a new document.
- **Open** – Opens an existing document.
- **Save** – Saves an active document.
- **Print** – Prints an active document.
- **Undo** – Reverses the last action.
- **Select** – Selects Sketch entities, components, and more.
- **Rebuild** – Rebuilds the active part, assembly,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

### Menu bar menu \ Menu bar toolbar

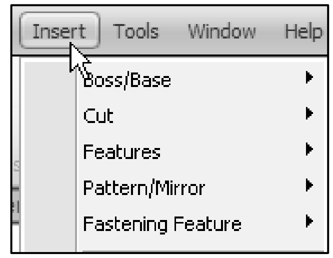
Click SolidWorks in the Menu bar toolbar to display the Menu bar menu. SolidWorks provides a Context-sensitive menu structure. The menu titles remain the same for all types of documents; (part, assembly, and drawing) but the menu items change depending on which type of document is active.


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.



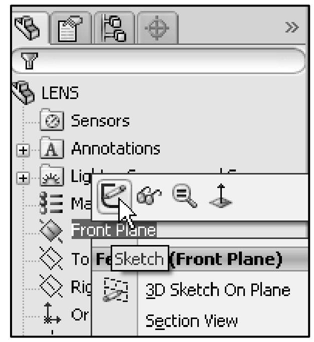
### Drop-down menu




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

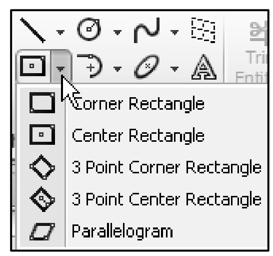
To close a SolidWorks drop-down menu, press the Esc key. You can also click any other section in the SolidWorks Graphics window, or click another drop-down menu.

### Right-click Context toolbar



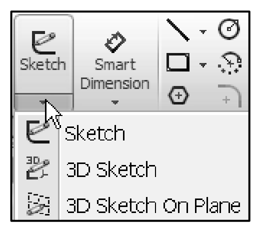

Right-click in the Graphics window either on a model, or in the FeatureManager on a feature or sketch to display a Context-sensitive toolbar. If you are in the middle of a command, the toolbar displays a list of options specifically related to that command.

### Fly-out tool buttons / Consolidated menu



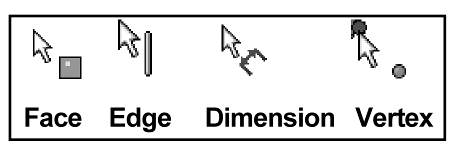

Similar commands are grouped into fly-out buttons on toolbars and the CommandManager. Example: Variations of the rectangle tool are consolidated together in a button with a fly-out control as illustrated. Select the drop-down arrow and view the available tools.

If you select the fly-out button without expanding:



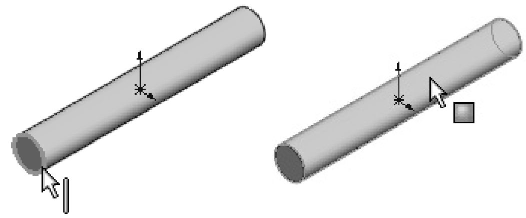

- For some commands such as Sketch, the most commonly used command is performed. This command is the first listed and the command shown on the button.
- For commands such as rectangle, where you may want to repeatedly create the same variant of the rectangle, the last used command is performed. This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

### System feedback icons

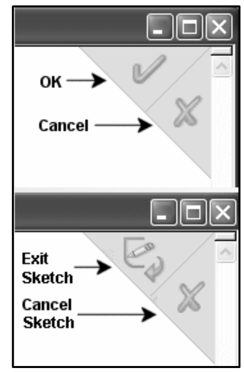



SolidWorks provides system feedback by attaching a symbol to the mouse pointer cursor arrow. The system feedback symbol indicates what you are selecting or what the system is expecting you to select.

As you move the mouse pointer across your model, system feedback is provided in the form of symbols, riding next to the cursor arrow as illustrated.



**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.

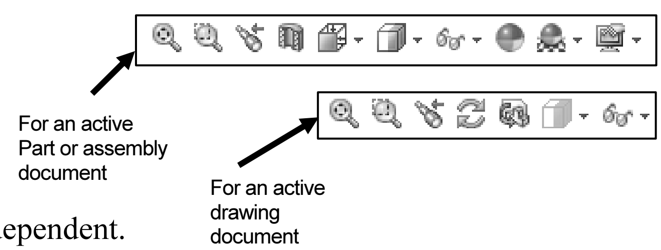
When a sketch is active, the confirmation corner box displays two symbols. The first symbol is the sketch tool icon. The second symbol is a large red X. These three symbols supply a visual reminder that you are in an active sketch. Click the sketch symbol icon to exit the sketch and to saves any changes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

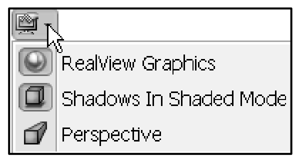



The Heads-up View toolbar is a transparent toolbar that is displayed in the Graphics window when a document is active. You can't hide nor move the Heads-up View toolbar. The following views are available: Note: Views are document dependent.

- *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.
- *Edit Appearance* : Provides the ability to apply appearances from the Appearances PropertyManager.
- *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.*

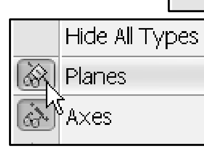


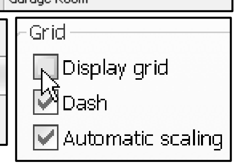


- *Rotate* : Provides the ability to rotate a drawing view.
- *3D Drawing View* : Provides the ability to dynamically manipulate the drawing view to make a selection.

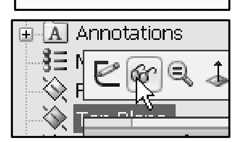
To deactivate the reference planes for an active document, click **View**, uncheck **Planes** from the Menu bar. To deactivate the grid, click **Options** , **Document Properties** tab. Click **Grid/Snaps**, uncheck the **Display grid** box.

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



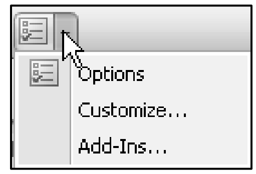




Modify the Heads-up View toolbar with an open document. Press the **space** key. The Orientation dialog box is display. Click the **New View** tool. The Name View dialog box is displayed. Enter a new **named** view. Click **OK**. The new view is displayed in the Heads-up View toolbar



The CommandManager is document dependent. Drop-down tabs are located on the bottom left side of the CommandManager and display the available toolbars and features for each corresponding tab.

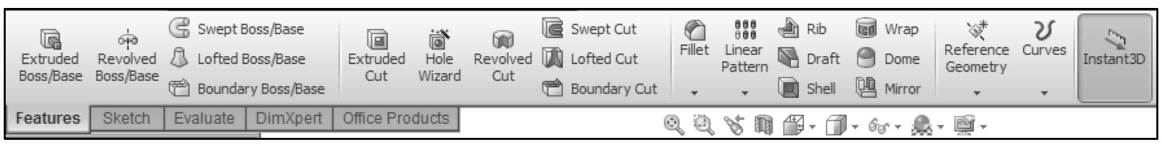
The default Part tabs are: *Features*, *Sketch*, *Evaluate*, *DimXpert*, and *Office Products*. Below is an illustrated CommandManager for a default Part document.

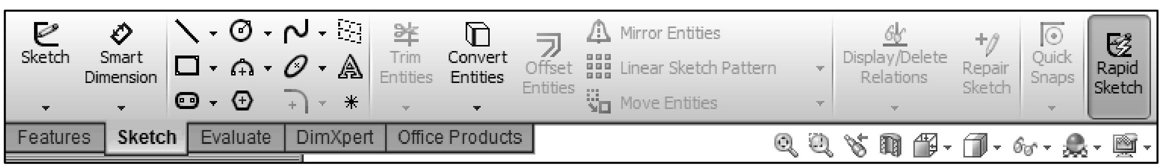


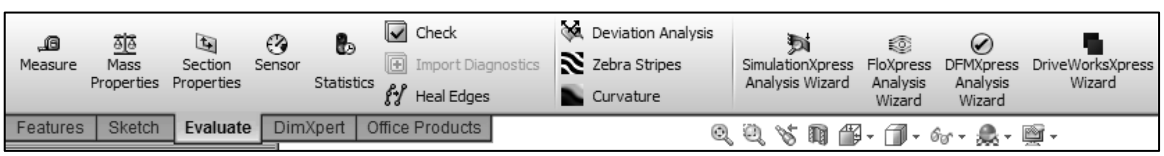

The Office Products toolbar display is dependent on the activated Add-Ins during a SolidWorks session.

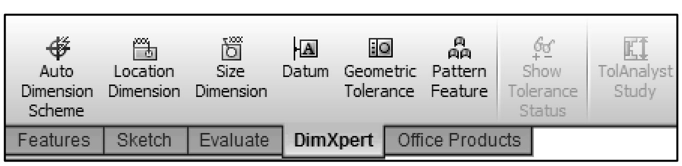
If you have SolidWorks, SolidWorks Professional, or SolidWorks Premium, the Office Products tab is displayed in the CommandManager.

Instant3D and Rapid Sketch are activated by default.



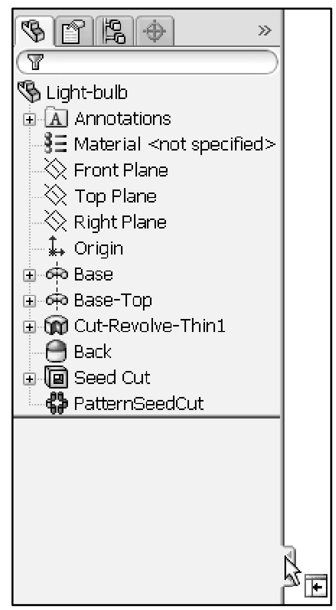




The CommandManager tabs that are displayed are document dependent and can be work flow customize. To customize the CommandManager tabs, **right-click** on a tab, and select the required **custom** option or select **Customize CommandManager** to access the Customize dialog box.


DimXpert provides the ability to graphically check if the model is fully dimensioned and toleranced. DimXpert 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. See SolidWorks Help for additional information.

### FeatureManager Design Tree

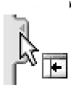
The FeatureManager design tree is located on the left side of the SolidWorks Graphics window. The design tree provides a summarize view of the active part, assembly, or drawing document. The tree displays the details on how the part, assembly, or drawing document is created.




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*

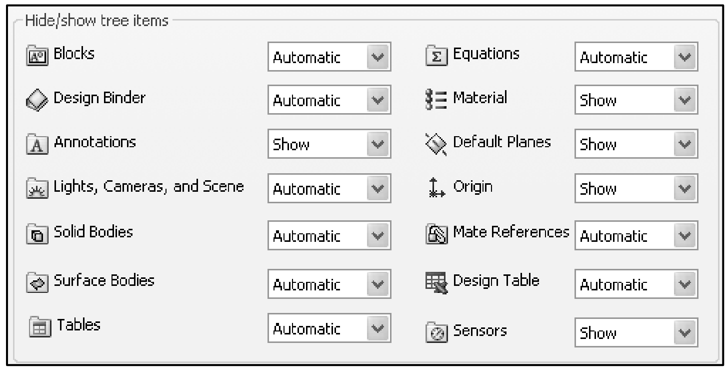
Select the Hide FeatureManager Tree Area arrows tab  from the FeatureManager to enlarge the Graphics window for modeling.

The Sensors tool  located in the FeatureManager monitors selected properties in a part or assembly and alerts you when values deviate from the specified limits. There are four sensor types: *Mass properties*, *Measurement*, *Interference Detection*, and *Simulation data*.

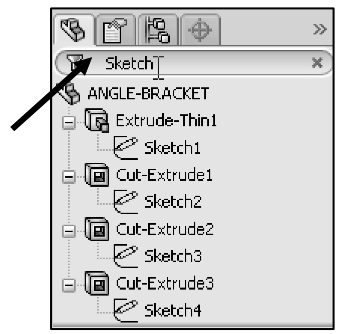
Various commands provide the ability to control what is displayed in the FeatureManager design tree. They are:

1. Show or Hide FeatureManager items.

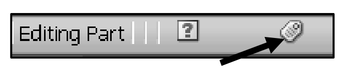
Click **Options** from the Menu bar. Click **FeatureManager** from the System Options tab. **Customize** your FeatureManager from the Hide/Show Tree Items dialog box.




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*.



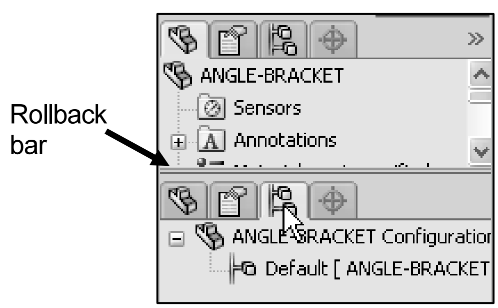

Tags are keywords you can add to a SolidWorks document to make them easier to filter and to search. The Tags icon is located in the bottom right corner of the Graphics window.




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

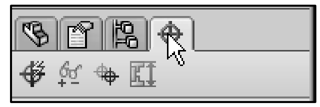



Move between the FeatureManager design tree, PropertyManager, ConfigurationManager, and DimXpertManager by selecting the tabs at the top of the menu.

Press the **s** key to view/access previous command tools in the Graphics window.

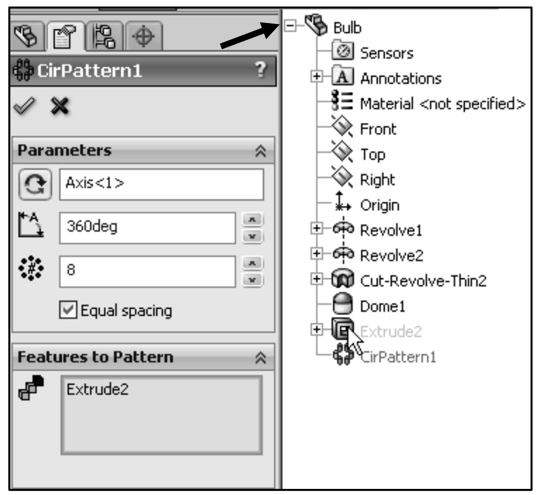
Split the ConfigurationManager and either display two ConfigurationManager instances, or combine the ConfigurationManager with the FeatureManager design tree, PropertyManager, or a third party application that uses the panel.

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



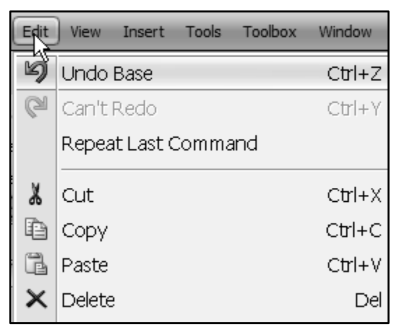
The DimXpertManager tab provides the ability to insert dimensions and tolerances manually or automatically. The DimXpertManager provides the following selections: *Auto Dimension Scheme*, *Show Tolerance Status*, *Copy Scheme*, and *TolAnalyst Study*.

**Fly-out FeatureManager**



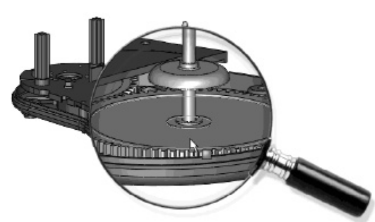

The fly-out FeatureManager design tree provides the ability to view and select items in the PropertyManager and the FeatureManager design tree at the same time.

Throughout the book, you will select commands and command options from the drop-down menu, fly-out FeatureManager, Context toolbar, or from a SolidWorks toolbar.




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.

Press the **g** key to activate the Magnifying glass tool. Use the Magnifying glass tool to inspect a model and make selections without changing the overall view.

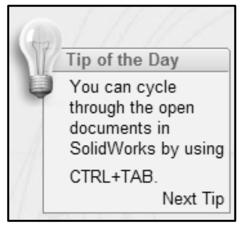


Illustrations may vary slightly depending on your SolidWorks version.

## 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*, *Appearances/Scenes*, and *Custom Properties*.

The Document Recovery tab is 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.





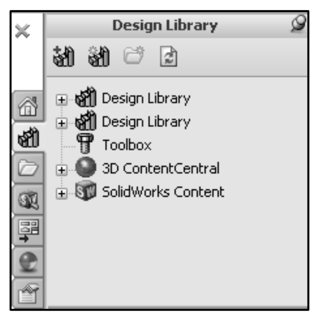
## SolidWorks Resources

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*.

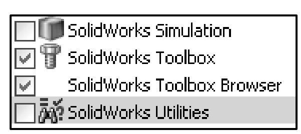
## Design Library

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



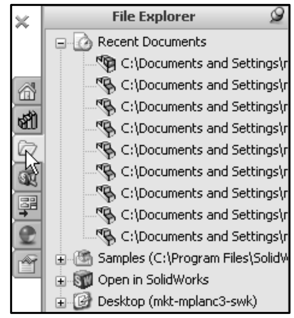

The Design Library tab contains four default selections. Each default selection contains additional sub categories. The default selections are*: Design Library*, *Toolbox*, *3D ContentCentral*, and *SolidWorks Content*.

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 an new installation, click **Add File Location**, enter: **C:\Documents and Settings\All Users\Application Data\SolidWorks\SolidWorks 2009\design library**. Click **OK**. In a network environment, contact your IT department for system details.

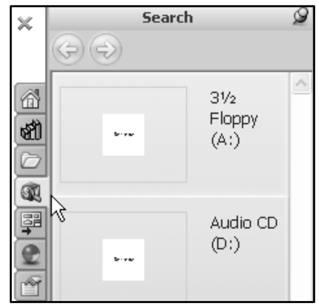
### File Explorer




File Explorer duplicates Windows Explorer from your local computer and displays *Resent Documents*, *directories*, and the *Open in SolidWorks* and *Desktop* folders

### 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 display 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.

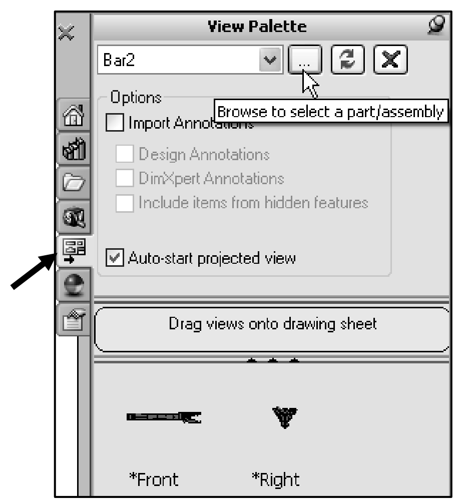
The Search tool in the Task Pane searches the following default locations: *All Locations*, *Local Files*, *Design Library*, *SolidWorks Toolbox*, and *3D ContentCentral*.

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

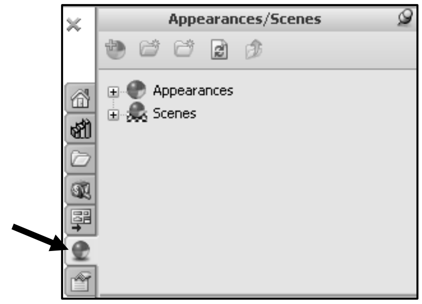
### View Palette

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.



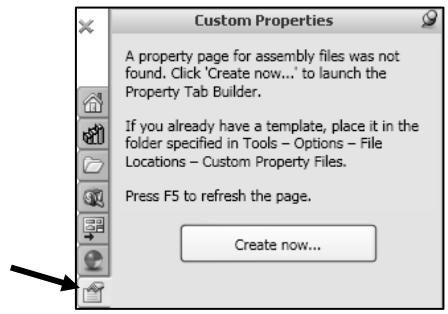
## Appearances/Scenes




Appearances/Scenes provide a simplified way to display models in a photo-realistic setting using a library of Appearances and Scenes. Note: Appearances/Scenes require graphics card support.

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

## Custom Properties




New in 2009 is the Custom Properties tab located in the Task Pane. The Custom Properties tool provides the ability to enter custom and configuration specific properties directly into SolidWorks files.

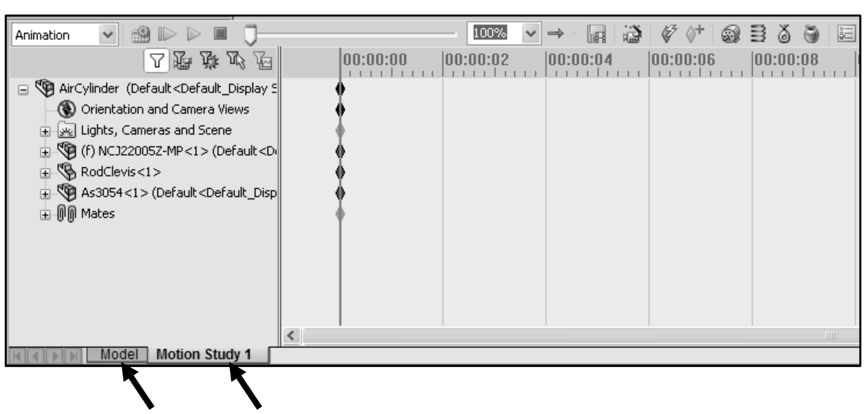
## Document Recovery

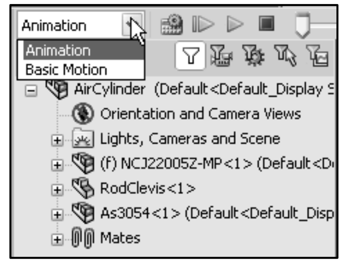
If auto recovery is initiated in the System Options section and the system terminates unexpectedly with an active document, the saved information files are available on the Task Pane Document Recovery tab the next time you start a SolidWorks session.

## Motion Study tab

Motion Studies are graphical simulations of motion for an assembly. Access MotionManager from the Motion Study tab. The Motion Study tab is located in the bottom left corner of the Graphics window.

Incorporate visual properties such as lighting and camera perspective. Click the Motion Study tab to view the MotionManager. Click the Model tab to return to the FeatureManager design tree.






The MotionManager display a timeline-based interface, and provide the following selections from the drop-down menu as illustrated:

*Animation:* Apply Animation to animate the motion of an assembly. Add a motor and insert positions of assembly components at various times using set key points. Use the Animation option to create animations for motion that does not require accounting for mass or gravity.

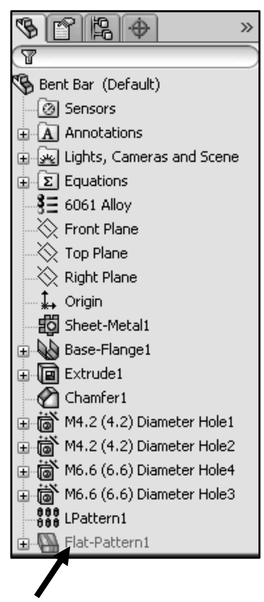
*Basic Motion*. Apply Basic Motion for approximating the effects of motors, springs, collisions, and gravity on assemblies. Basic Motion takes mass into account in calculating motion. Basic Motion computation is relatively fast, so you can use this for creating presentation animations using physics-based simulations. Use the Basic Motion option to create simulations of motion that account for mass, collisions, or gravity.


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

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.

## Activity: Create a new 3D Part

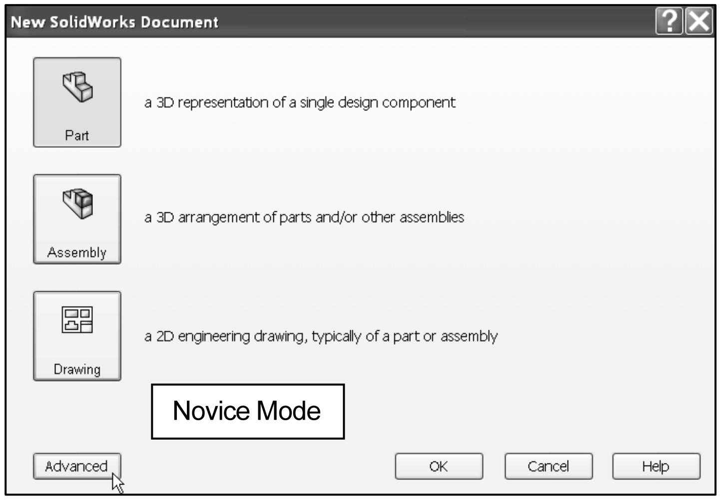
A part is a 3D model which consists of features. What are features?




- 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.
- Features are an individual shape that combined with other features, makes up a part or assembly. Some features, such as bosses and cuts, originate as sketches. Other features, such as shells and fillets, modify a feature's geometry.
- Features are displayed in the FeatureManager as illustrated.

You can suppress a feature, as is *Flat-Pattern1* in the Bent Bar FeatureManager. A suppress feature is display in light gray.

The first sketch of a part is the Base sketch. The Base sketch is the foundation for the 3D model.



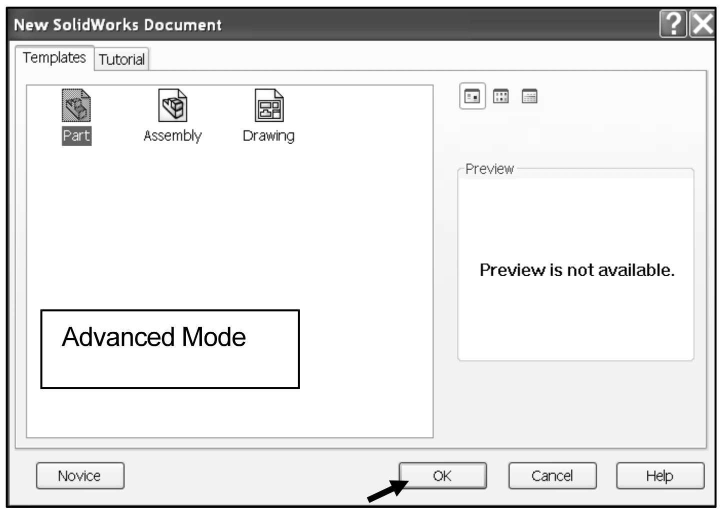

Create a new part.

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

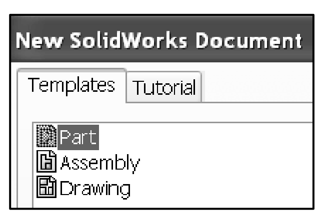
Select Advanced Mode.

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

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




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



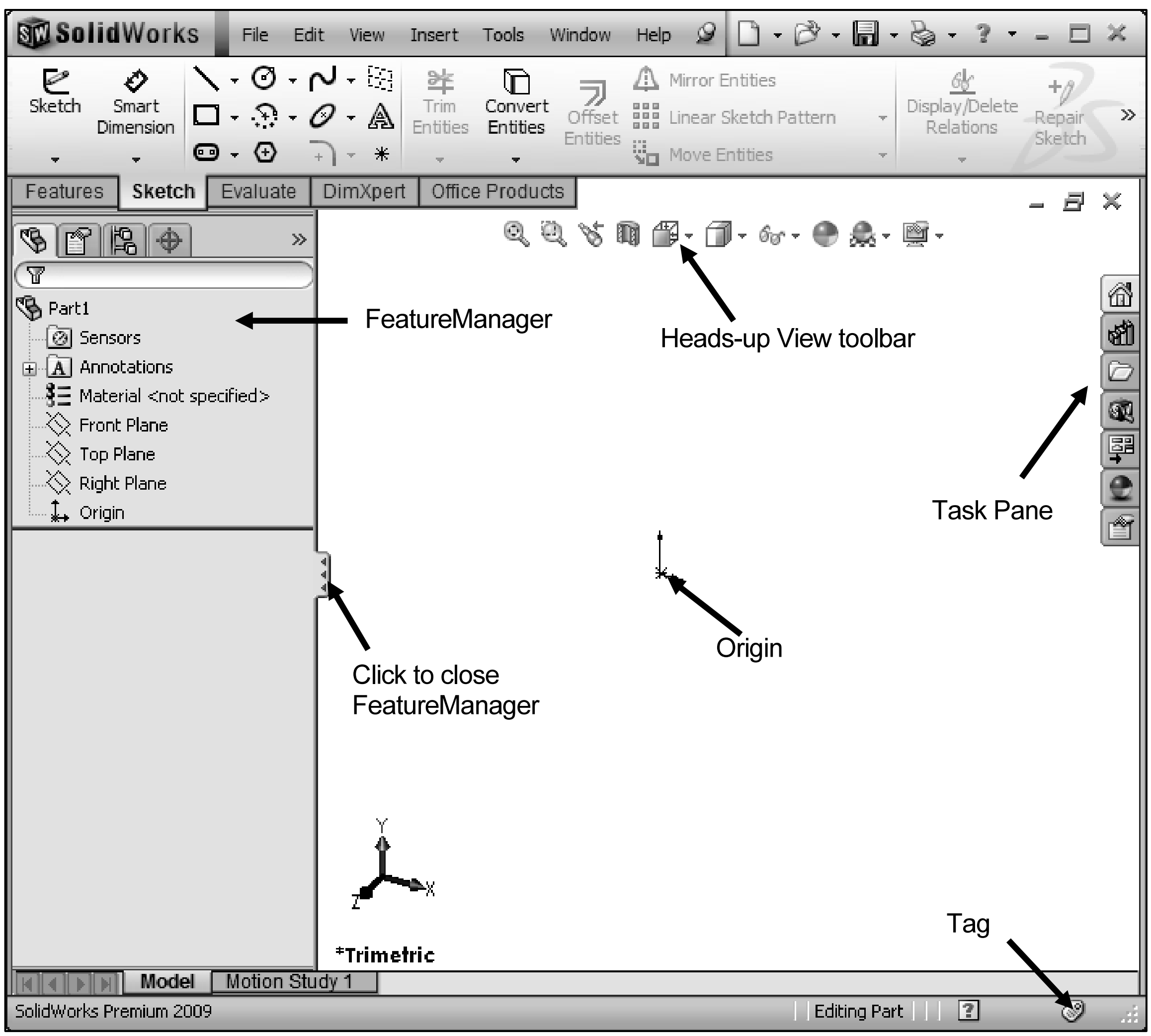

The default SolidWorks installation contains two tabs in the New SolidWorks Document dialog box: Templates and Tutorial.

The Templates tab corresponds to the default SolidWorks templates and the templates utilized in the SolidWorks 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 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 positive Z-axis points out to you 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.

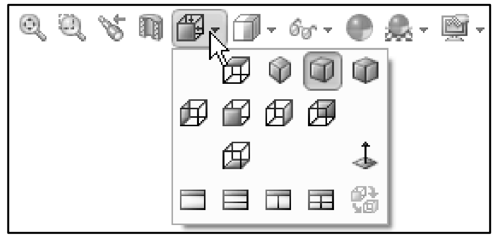
**Activity: Menu Bar toolbar, Menu Bar menu, Heads-up View toolbar**

Display tools and tool tips.

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

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

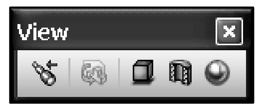
**21)** Select the **drop-down arrow** to view the available view tools.



Display the View toolbar and the Menu bar.

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

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




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

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

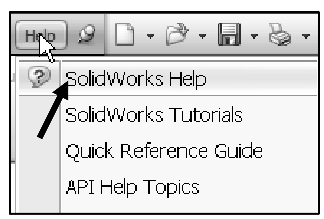

**26)** **Pin** the Menu bar as illustrated. Use both the Menu bar menu and the Menu bar toolbar in this book.

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.

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

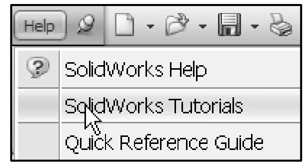
**28)** Click **SolidWorks Help**. The SolidWorks Help dialog box is displayed.




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.

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

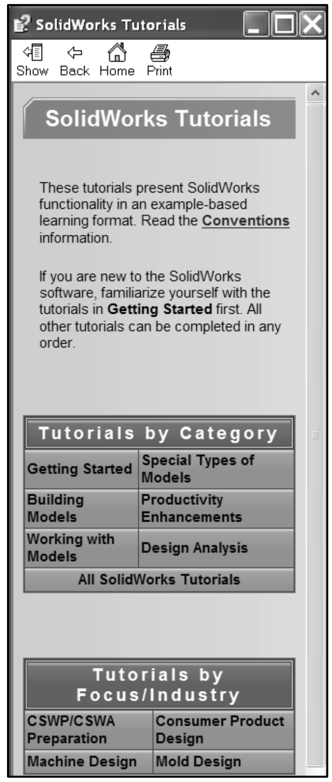


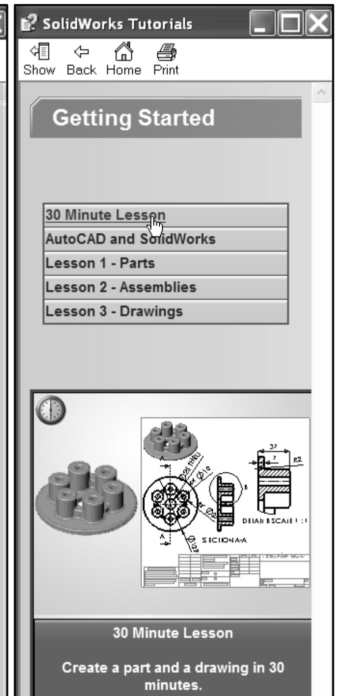

Display and explore the SolidWorks Tutorials.

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

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

**32)** 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.

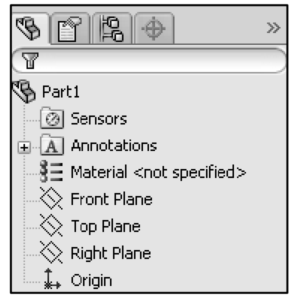
The CSWA certification indicates a foundation in and apprentice knowledge of 3D CAD design and engineering practices and principles.

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.

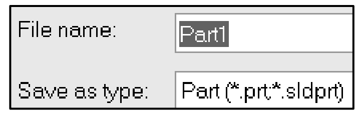
**33)** **Close** the Online Tutorial dialog box.

## Part Template

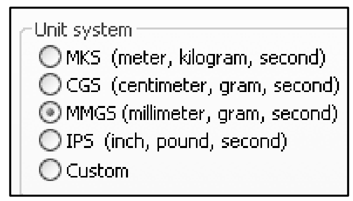



The Part Template is the foundation for a SolidWorks part. Part1 displayed in the FeatureManager utilizes the Part.sldprt default template located in the New SolidWorks dialog box.

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 (drafting) 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.

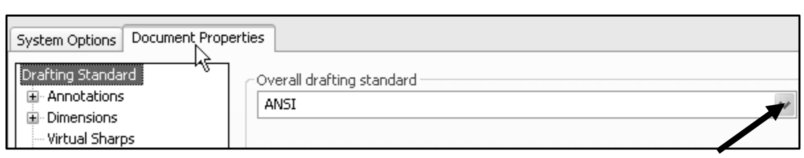
Utilize the File Locations option to reference your Part Templates in the MY-TEMPLATES folder. Add the SOLIDWORKS-MODELS 2009\MY-TEMPLATES folder path name to the Document Templates File Locations list.

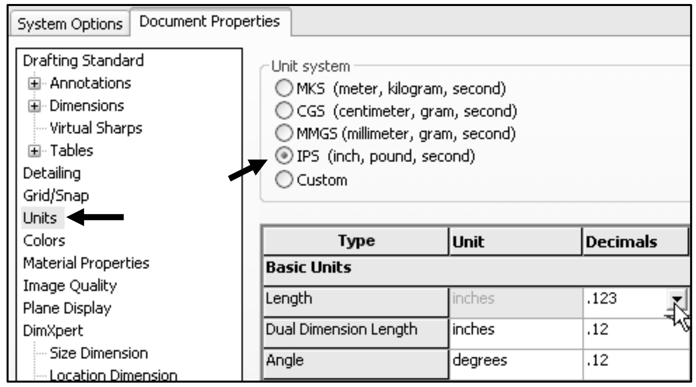
**Activity: Create the PART-IN-ANSI and PART-MM-ISO Part Template**

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

**34)** Click **Options**, **Document Properties** tab from the Menu bar. The Document Properties – Drafting Standard dialog box is displayed.

**35)** Select **ANSI** from the Overall drafting standard drop-down box.






Set the part units.

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

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

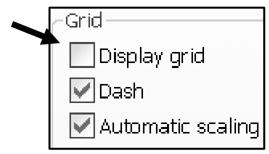
**38)** Select **.123** (three decimal places) for Length basic units.

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

Set the Grid/Snap option.

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

**41)** Un-check the **Display grid** box.

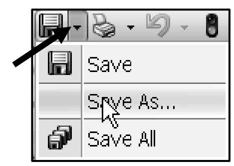



Return to the SolidWorks Graphics window.

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

Save the Part Template.

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

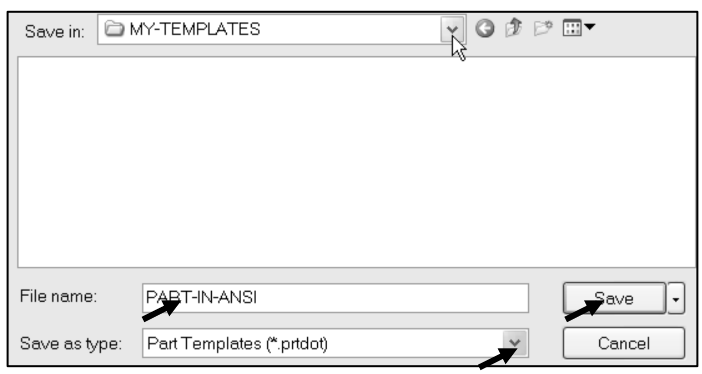



**44)** Select **Part Templates (*.prtdot)** from the Save as type box.

**45)** Select the **SOLIDWORKS-MODELS 2009/MY-TEMPLATES** folder.

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

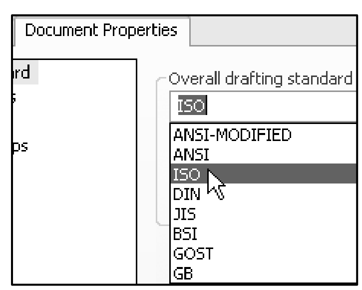
**47)** Click **Save** from the Save As dialog box.




Create the PART-MM-ISO Part Template.

**48)** Click **Options**, **Document Properties** tab from the Menu bar. The Document Properties - Drafting Standard dialog box is displayed.

**49)** Select **ISO** from the Overall drafting standard drop-down box.



Set the part units.

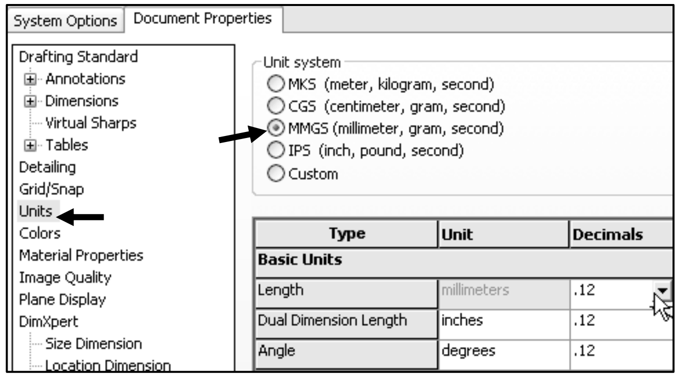
**50)** Click **Units**. The Document Properties - Unit dialog box is displayed.

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

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

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

**54)** Click **OK**.




Save the Part Template.

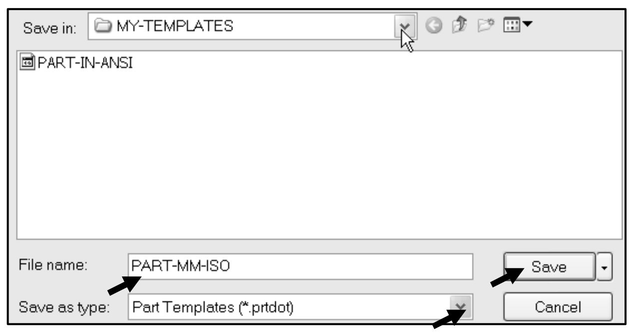
**55)** Click **Save As** from the Menu bar. The Save As dialog box is displayed.


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

**57)** Select the **SOLIDWORKS-MODELS 2009/MY-TEMPLATES** folder.

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

**59)** Click **Save**.



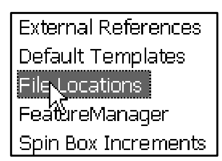

Set System Options to add the two Part Templates.

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

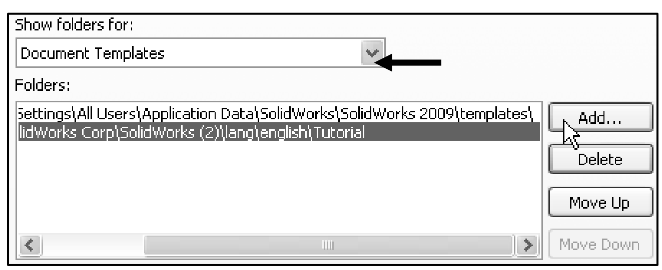
**61)** Click **File Locations** from the System Options tab.

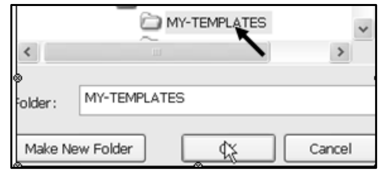
**62)** Select **Document Templates** from Show folders for.

**63)** Click the **Add** button.




**64)** Select the **SOLIDWORKS-MODELS 2009/MY-TEMPLATES** folder.





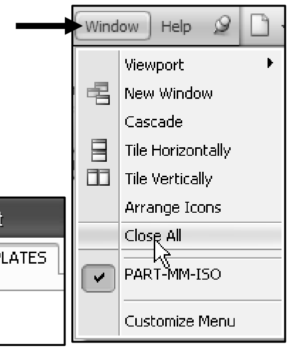

**65)** Click **OK** from the Browse for Folder.

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

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

Close All documents.

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

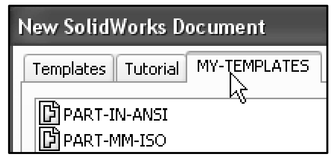



Display the MY-TEMPLATES folder and templates.

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

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

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

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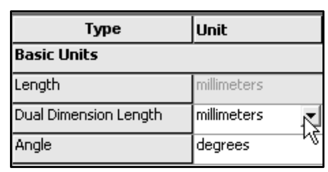
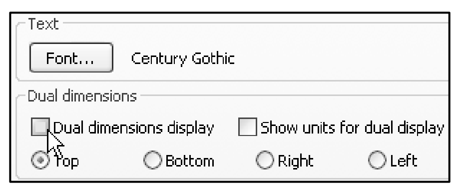
Each folder listed in the System Options, File Locations, Document Templates, Show Folders For option produces a corresponding tab in the New SolidWorks Document dialog box.

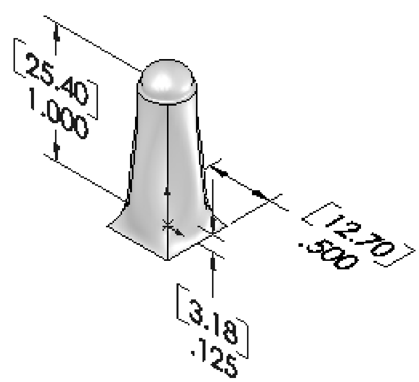
The MY-TEMPLATES tab is only visible when the folder contains a SolidWorks Template document. Create the PART-MM-ANSI template as an exercise.

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.



Additional information on System Options, Document Properties, File Locations, and Templates is located 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 2009 User Interface and CommandManager consist of the following options: Menu bar toolbar, Menu bar menu, Drop-down menus, Context toolbars, Consolidated fly-out 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.

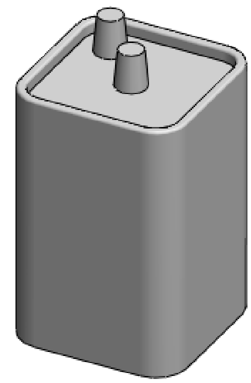
The FeatureManager design tree consist of four default tabs: *FeatureManager design tree,PropertyManager,ConfigurationManager, and DimXertManager.*

The Task Pane contains the following default tabs: *SolidWorks Resources*, *Design Library*, *File Explorer*, *SolidWorks Search*, *View Palette*, *Appearances/Scenes*, and *Custom Properties*.

You created two Part Templates: **PART-MM-ISO** and **PART-IN-ANSI**. The document properties Overall drafting 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.

## BATTERY Part



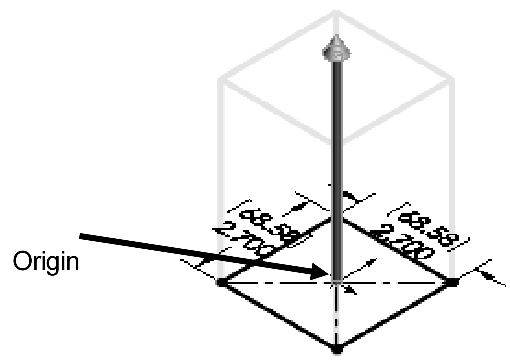
The BATTERY is a simplified representation of a purchased OEM part. Represent the battery terminals as cylindrical extrusions. The battery dimensions are obtained from the ANSI standard 908D.

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

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

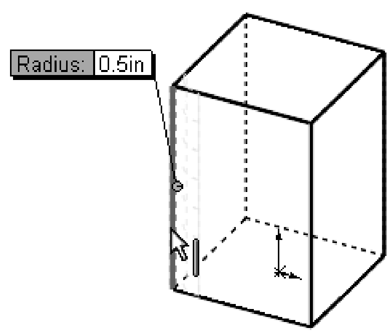
Utilize the Instant3D tool to create the Extruded Base feature. The Extrude Base features add material. The Base feature is the first feature of the part.

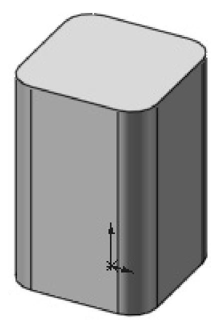
Apply symmetry. Use the Center Rectangle Sketch tool on the Top Plane. The 2D Sketch profile is centered at the Origin.



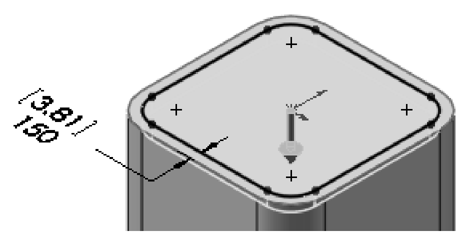

Extend the profile perpendicular (⊥) to the Top Plane.

Utilize the Fillet feature to round the four vertical edges.

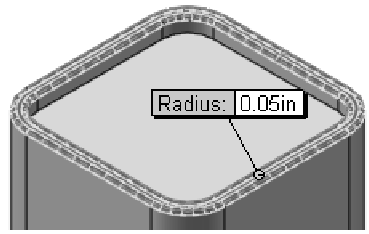


The Extruded Cut feature removes material from the top face. Utilize the top face for the Sketch plane. Utilize the Offset Entity Sketch tool to create the profile.



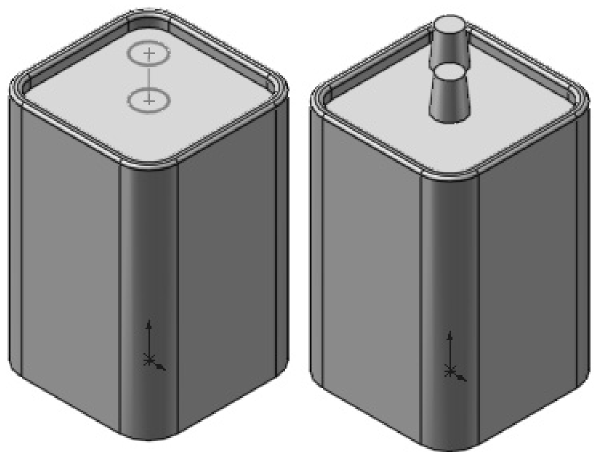
Utilize the Fillet feature to round the top narrow face.




Fillet/Round features creates a rounded internal or external face on the part. You can fillet all edges of a face, selected sets of faces, selected edges, or edge loops.

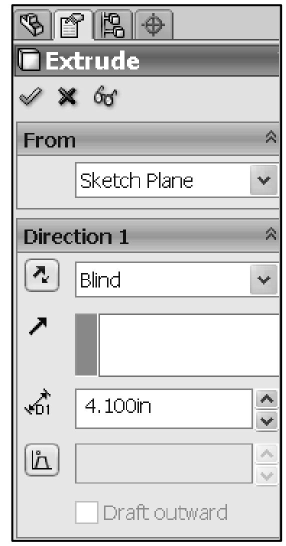
Add larger fillets before smaller ones. When several fillets converge at a vertex, create the larger fillets first

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



Extrude Features creates a feature by extruding a 3D object from a 2D sketch, essentially adding the third dimension. An extrusion can be a base, a boss (which adds material, often on another extrusion), or a cut (which removes material).

**BATTERY Part-Extruded Base Feature**




The Extruded Base feature requires:

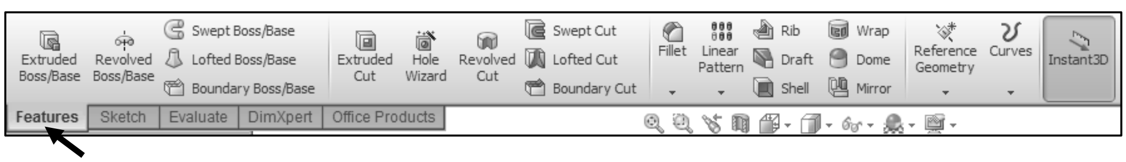
- Sketch plane (Top)
- Sketch profile (Rectangle)
  - Geometric relations and dimensions
- End Condition Depth (Blind) in Direction 1

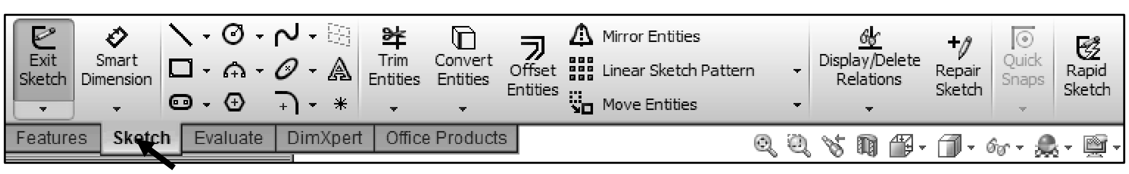
Create a new part named, BATTERY. Insert an Extruded Base feature. Extruded features require a Sketch plane. The Sketch plane determines the orientation of the Extruded Base feature. The Sketch plane locates the Sketch profile on any plane or face.

The Top Plane is the Sketch plane. The Sketch profile is a rectangle. Utilize the Center Rectangle Sketch tool. The Center Rectangle Sketch tool sketches a rectangle that includes a centerpoint. Select the Origin as the centerpoint.

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

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



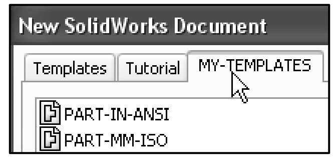
## Activity: BATTERY Part-Create the Extruded Base Feature

Create a new part.

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

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

**74)** Double-click **PART-IN-ANSI**, [**PART-MM-ISO**].



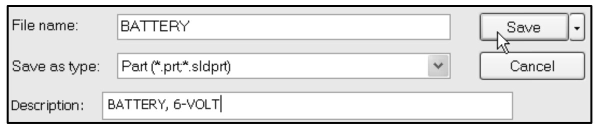

Save the empty part.

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

**76)** Select **PROJECTS** for Save in folder.

**77)** Enter **BATTERY** for File name.

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




**79)** Click **Save**. The Battery FeatureManager is displayed.

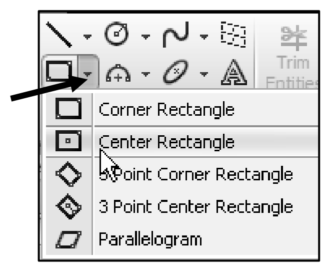
Select the Sketch plane.

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


Sketch the 2D Sketch profile centered at the Origin.

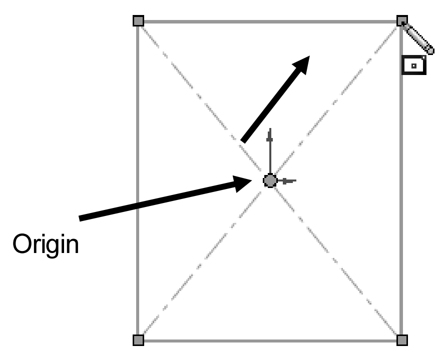
**81)** Click **Sketch** from the Context toolbar. The Sketch toolbar is displayed.

**82)** Click the **Center Rectangle** Sketch tool. The Center Rectangle icon is displayed.




**83)** Click the **Origin**. This is your first point.

**84)** Drag and click the **second point** in the upper right quadrant as illustrated. The Origin is located in the center of the sketch profile. The Center Rectangle Sketch tool automatically applies equal relations to the two horizontal and two vertical lines. A midpoint relation is automatically applied to the Origin.




The book is design to expose the user to different tools and procedures.

Click **View**, **Sketch Relations** from the Menu bar to display the relations in the Graphics window.

Add dimensions.

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

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

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

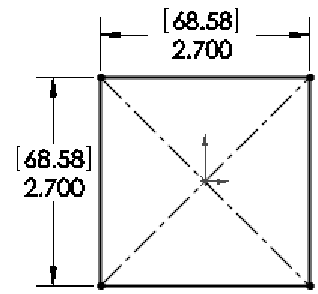
**88)** Enter **2.700**in, [**68.58**] for width.

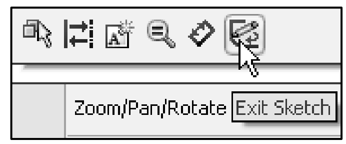
**89)** Click the **Green Check mark** in the Modify dialog box.

**90)** Enter **2.700**in, [68.58] for height as illustrated.

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

**92)** Click **OK** from the Dimension PropertyManager.

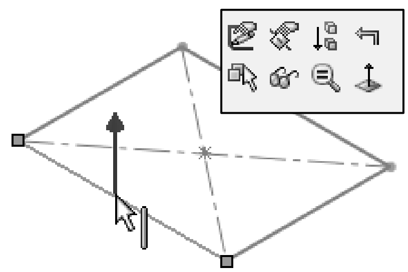



Exit the Sketch.

**93)** Right-click **Exit Sketch**.

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



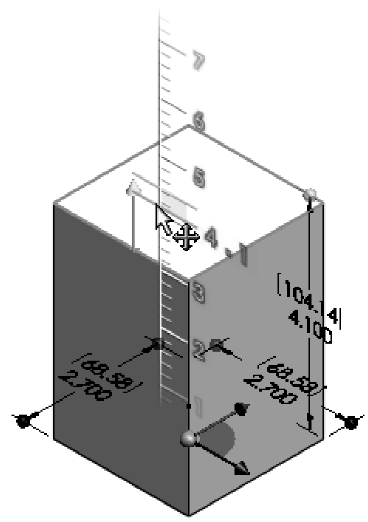
Use the on-screen ruler.

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

**95)** Click the **front horizontal line** as illustrated. A green arrow is displayed.

**96)** Click and drag the **green arrow** upward.

**97)** Click the on-screen ruler at **4.1**in, [104.14] as illustrated. This is the depth in direction 1. The extrude direction is upwards. Extrude1 is displayed in the FeatureManager.

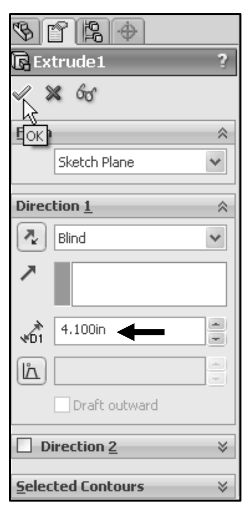



Check the Extrude1 feature depth dimension.

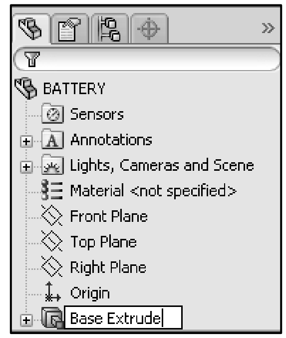
**98)** Right-click **Extrude1** from the FeatureManager.

**99)** Click **Edit Feature** from the Context toolbar. 4.100in is displayed for depth. Blind is the default End Condition. Note: If you did not select the correct depth, input the depth in the Extrude1 PropertyManager.

**100)** Click **OK** from the Extrude1 PropertyManager.




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




Fit the part to the Graphics window.

**101)** Press the **f** key.

Rename the Extruded Base feature.

**102)** Double-click **Extrude1**. View the Sketch.

**103)** Click **Extrude1**.

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

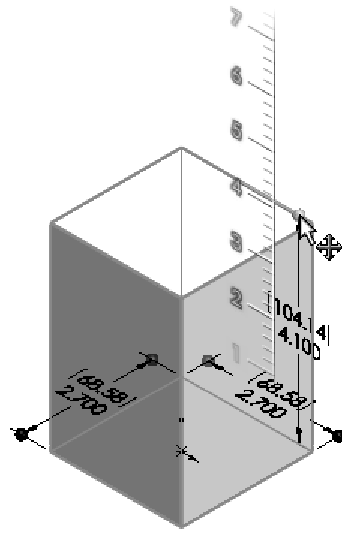
Save the BATTERY.

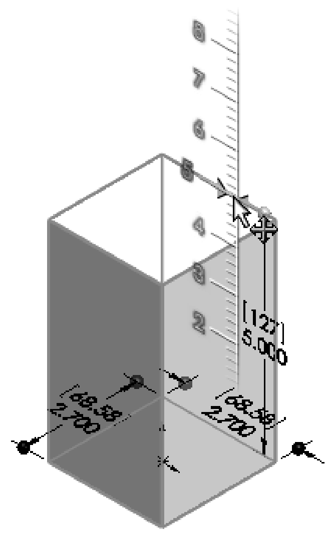
**105)** Click **Save**.

Modify the BATTERY.

**106)** Click **Base Extrude** from the FeatureManager.

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



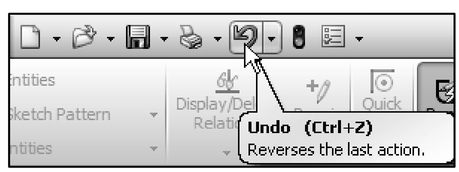




Return to the 4.100 depth.

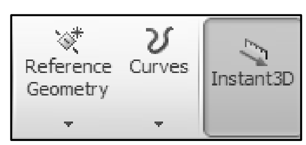
**108)** Click the **Undo** button from the Menu bar. The depth of the model is 4.100in, [104.14]. Blind is the default End Condition. Practice may be needed to select the correct on-screen ruler dimension.

The color of the sketch indicates the sketch status.

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

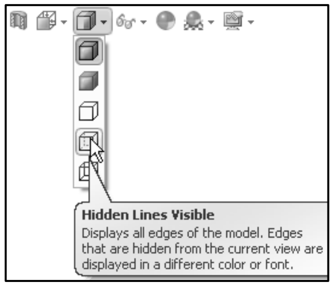


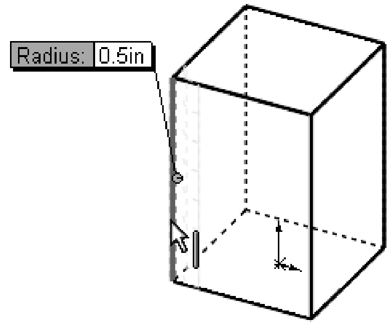

The Instant3D tool is activated by default from the Features toolbar in the CommandManager.



### BATTERY Part-Fillet Feature Edge

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

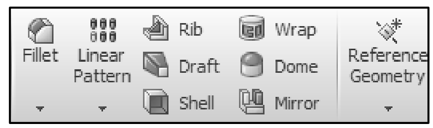


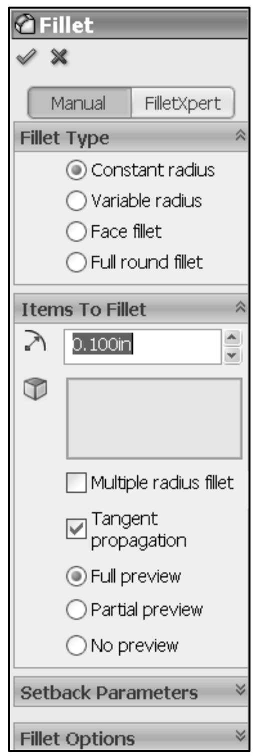

An edge Fillet feature requires:

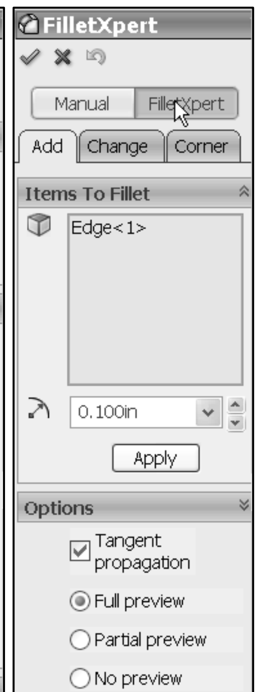
- A selected edge
- Fillet radius

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




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




The Fillet feature uses the Fillet PropertyManager. The Fillet PropertyManager provides the ability to select either the *Manual* or *FilletXpert* tab.

Each tab has a separate menu and PropertyManager. The Fillet PropertyManager and FilletXpert PropertyManager displays the appropriate selections based on the type of fillet you create.

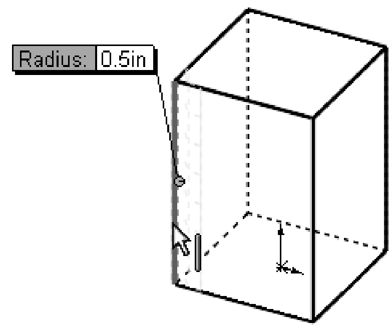
The FilletXpert automatically manages, organizes and reorders your fillets in the FeatureManager design tree. The FilletXpert PropertyManager provides the ability to add, change or corner fillets in your model. The PropertyManager remembers its last used state. View the SolidWorks tutorials for additional information on fillets.

The FilletXpert can ONLY create and edit Constant radius fillets.

## Activity: BATTERY Part-Fillet Feature Edge

Display Hidden Edges.

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



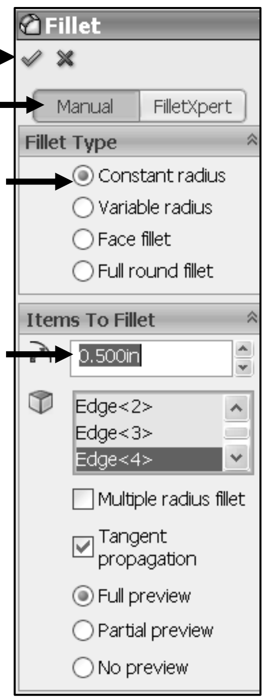

Insert a Fillet feature.

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

**111)** Click the **Features** tab from the CommandManager.

**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.



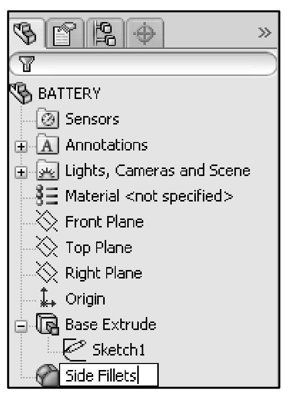

**114)** Constant radius is the default Fillet Type. Click the remaining **3 vertical edges**. The selected entities are displayed in the Items To Fillet box

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

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

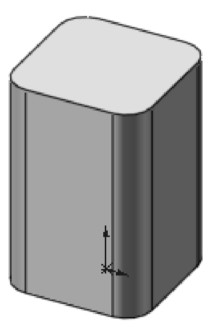



Rename the feature.

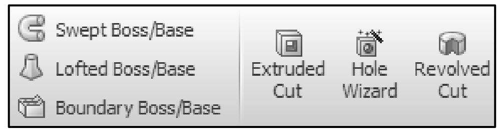
**119)** Rename **Fillet1** to **Side Fillets** in the FeatureManager.

Save the BATTERY.

**120)** Click **Save**.



## BATTERY Part-Extruded Cut Feature




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

- Sketch plane (Top face)
- Sketch profile (Offset Entities)
- End Condition depth (Blind) in Direction 1

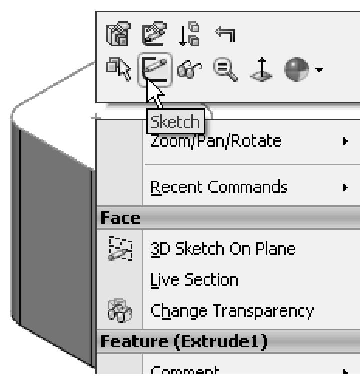
The Offset Entity Sketch tool uses existing geometry, extracts an edge or face and locates the geometry on the current Sketch plane.

Offset the existing Top face for the 2D sketch. Utilize the default Blind End Condition in Direction 1.

### Activity: BATTERY Part-Extruded Cut Feature

Select the Sketch plane.

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




Create a sketch.

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

Display the face.

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

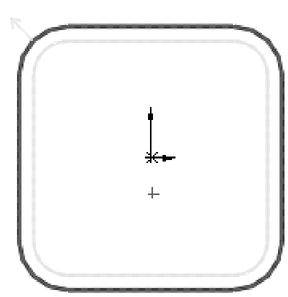
Offset the existing geometry from the boundary of the Sketch plane.

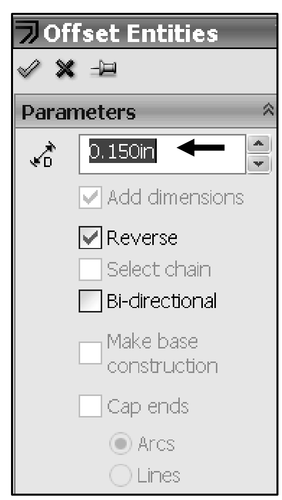
**124)** Click the **Offset Entities** Sketch tool. The Offset Entities PropertyManager is displayed.

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

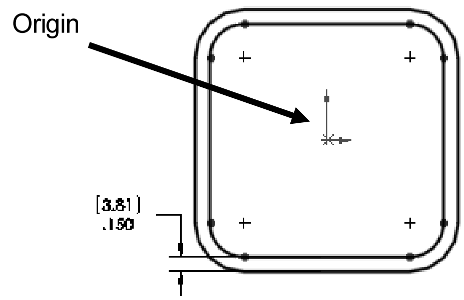
**126)** Click the **Reverse** box. The new Offset yellow profile displays inside the original profile.

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





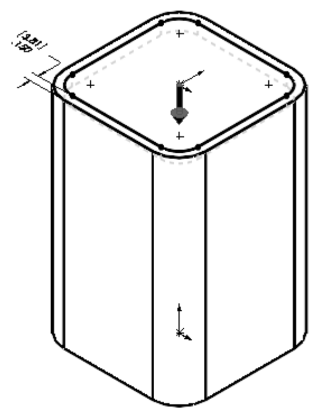
A leading zero is displayed in the spin box. For inch dimensions less than 1, the leading zero is not displayed in the part dimension in the ANSI standard.




Display the profile.

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

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



Insert an Extruded Cut feature. As an exercise, use the Instant3D tool to create the Extruded Cut feature. In this section, the PropertyManager is used.

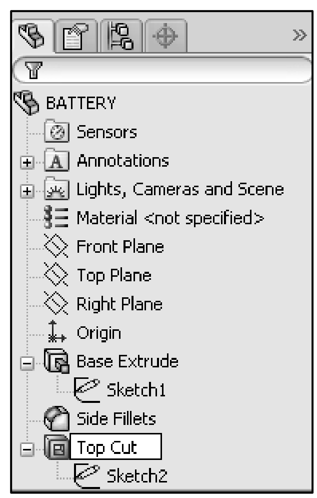
**130)** Click the **Extruded Cut** feature tool. The Extrude PropertyManager is displayed.

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

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

Rename the feature.

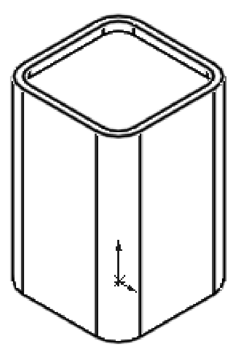
**133)** Rename **Extrude2** to **Top Cut** in the FeatureManager.



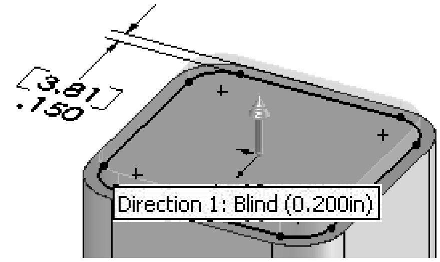

Save the BATTERY

**134)** Click **Save**.

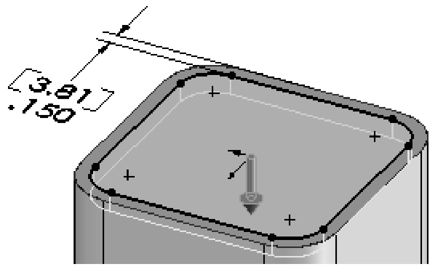
The Extrude PropertyManager contains numerous options. The Reverse Direction option determines the direction of the Extrude. The Extruded Cut feature is valid only when the direction arrow points into material to be removed.

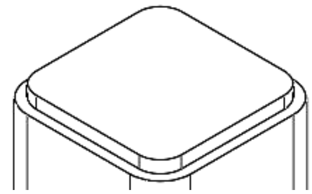


Cut direction not valid, no material to remove



The Flip side to cut option determines if the cut is to the inside or outside of the Sketch profile. The Flip side to cut arrow points outward. The Extruded Cut feature occurs on the outside of the BATTERY.



Extruded Cut with Flip side to cut option checked

## BATTERY Part-Fillet Feature

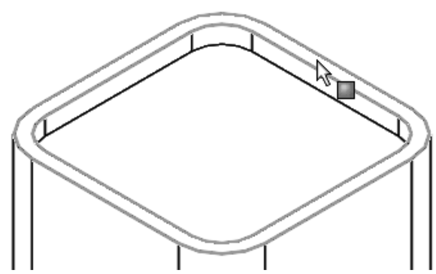
The Fillet feature tool rounds sharp edges by selecting a face. A Fillet requires a:

- A selected face
- Fillet radius

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

Insert a Fillet feature on the top face.

**135)** Click the **top thin face** as illustrated. Note: The face icon feedback symbol.



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

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

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

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

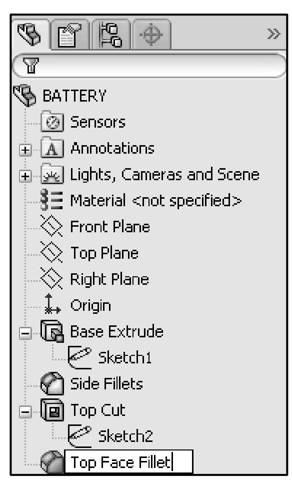
Rename the feature.

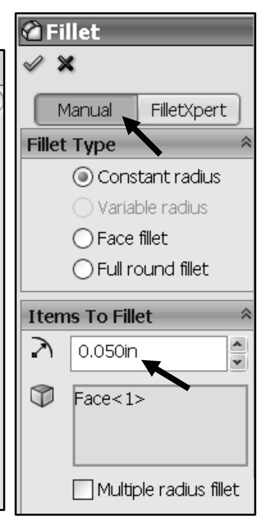
**140)** Rename **Fillet2** to **Top Face Fillet**.

**141)** Press the **f** key.

Save the BATTERY.

**142)** Click **Save**.



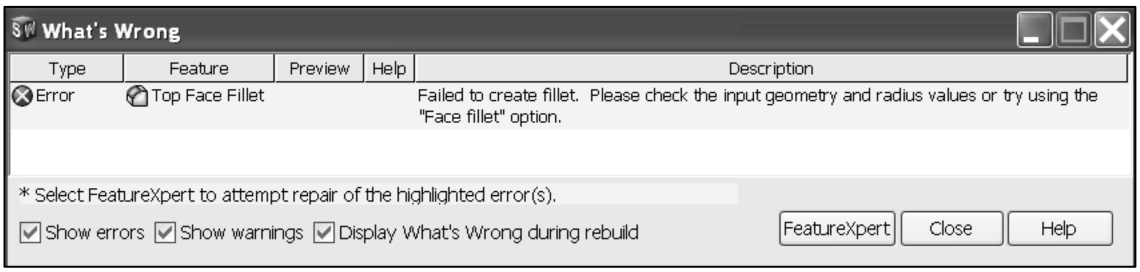



View the mouse pointer for feedback to select Edges or Faces for the fillet.

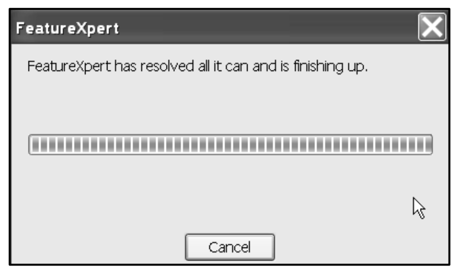
Do not select a fillet radius which is larger then the surrounding geometry.

Example: The top edge face width is .150in, [3.81]. The fillet is created on both sides of the face. A common error is to enter a Fillet too large for the existing geometry. A minimum face width of .200in, [5.08] is required for a fillet radius of .100in, [2.54].

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




Avoid the fillet rebuild error. Use the FeatureXpert to address a constant radius fillet build error or manually enter a smaller fillet radius size. As an exercise, insert a large Fillet radius and use the FeatureXpert option.

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#### BATTERY Part-Extruded Boss Feature

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

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

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

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

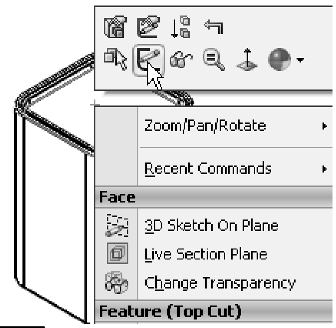
Calculate the depth of the extrusion: For inches: 4.500in – (4.100in Base-Extrude height – .200in Offset cut depth) = .600in  The depth of the extrusion is .600in.

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

## Activity: BATTERY Part-Extruded Boss Feature

Select the Sketch plane.

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



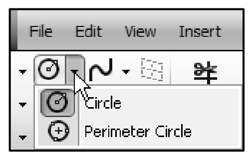

Create the sketch.

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

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

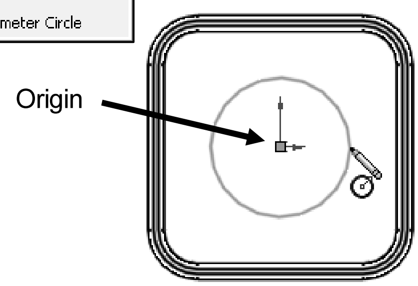
Sketch the profile.

**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 as illustrated.




Add a dimension.

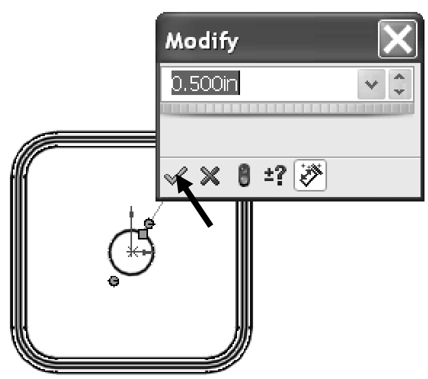
**149)** Click the **Smart Dimension** Sketch tool.

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

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

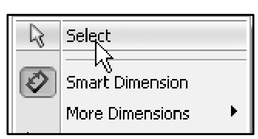
**152)** Enter **.500**in, [**12.7**].

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




Deselect Smart Dimension.

**154)** Right-click **Select**.



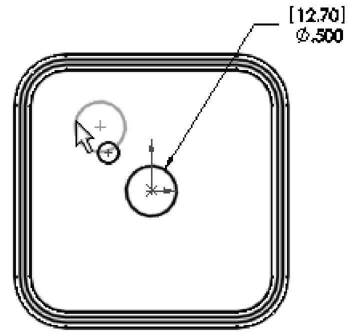

Copy the Sketch circle.

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

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

**157)** Release the **mouse button**.

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




Add an Equal relation.

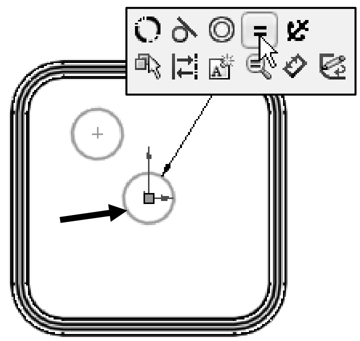
**159)** Hold the **Ctrl** key down.

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

**161)** Release the **Ctrl** key.

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

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



Show the Right Plane for the dimension reference.

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

Add an aligned dimension.

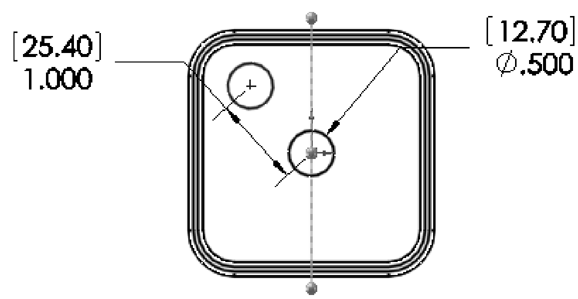
**165)** Click the **Smart Dimension** Sketch tool.

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

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

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

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

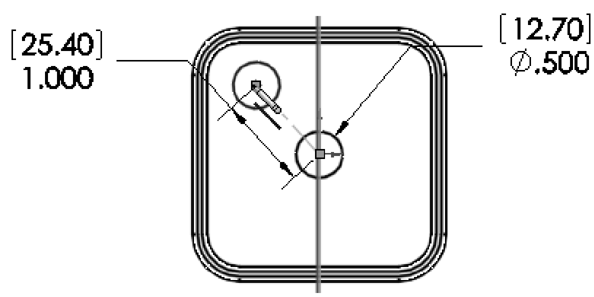



Insert a centerline.

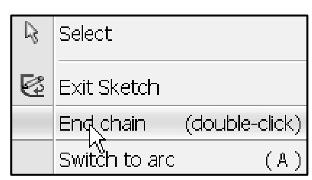
**170)** Click the **Centerline** Sketch tool. The Insert Line PropertyManager is displayed.

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

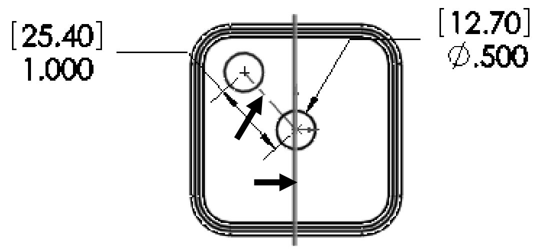
**172)** Right-click **End Chain** to end the line.




Double-click to end the centerline.



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




Add an angular dimension.

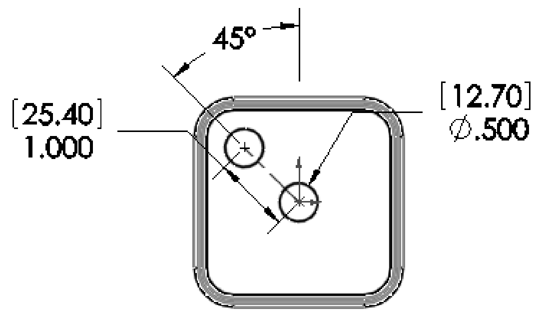
**173)** Click the **Smart Dimension** Sketch tool.

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

**175)** Click the **Right Plane** (vertical line) in the Graphics window. Note: You can also click Right Plane in the FeatureManager.

**176)** Click a **position** between the centerline and the Right Plane, off the profile.

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




Fit the model to the Graphics window.

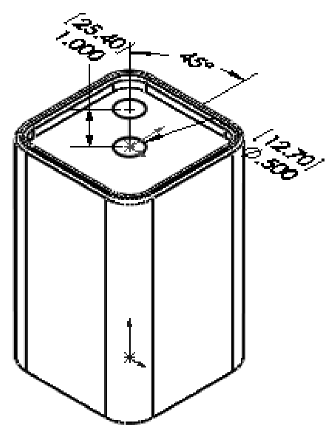
**178)** Press the **f** key.

Hide the Right Plane.

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

**180)** Click **Hide** from the Context toolbar.

**181)** Click **Save**.



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

Insert an Extruded Boss feature.

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

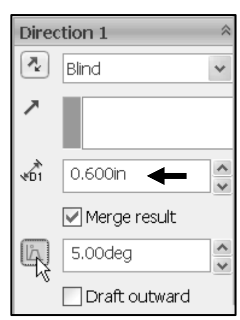
**183)** Click the **Extruded Boss/Base** feature tool. The Extrude PropertyManager is displayed. Blind is the default End Condition Type.

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

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

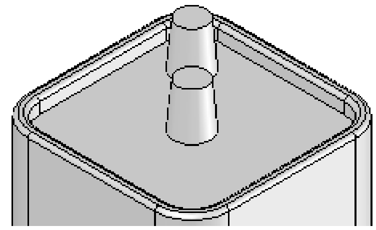
**186)** Enter **5**deg in the Draft Angle box.

**187)** Click **OK** from the Extrude PropertyManager. The Extrude feature is displayed in the FeatureManager.



Rename the feature and sketch.

**188)** Rename **Extrude3** to **Terminals**.



**189)** **Expand** Terminals.

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

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

**192)** Click **Save**.

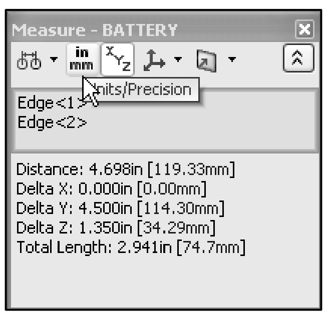
Each time you create a feature of the same feature type, the feature name is incremented by one. Example: Extrude1 is the first Extrude feature. Extrude2 is the second Extrude feature. If you delete a feature, rename a feature or exit a SolidWorks session, the feature numbers will vary from those illustrated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

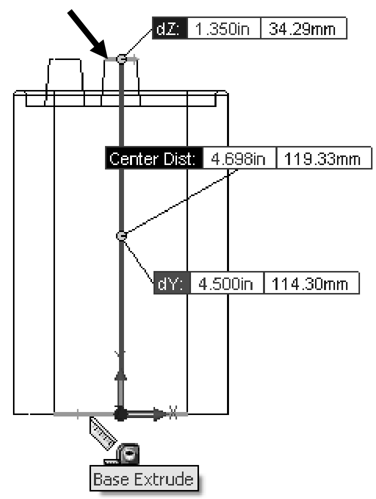
Rename your features with descriptive names. Standardize on feature names that are utilized in mating parts. Example: Mounting Holes.

Measure the overall BATTERY height.

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

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



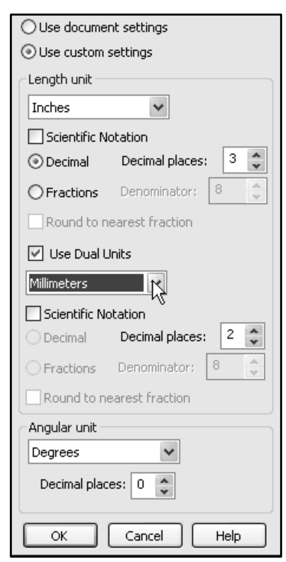




**195)** Click the **top edge** of the battery terminal as illustrated.

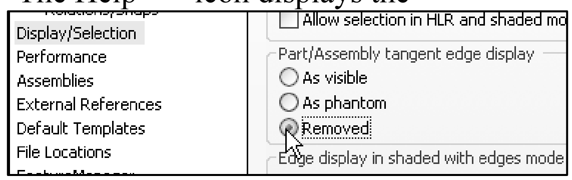
**196)** Click the **bottom edge** of the battery. The overall height, Delta Y is 4.500, [114.3]. Apply the Measure tool to insure a proper design.

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

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

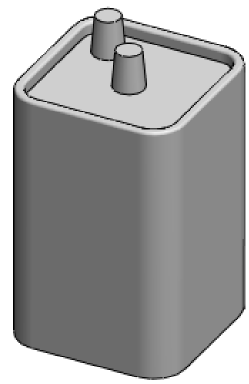


The Selection Filter option toggles the Selection Filter toolbar. When Selection Filters are activated, the mouse pointer displays the Filter icon . The Clear All Filters tool removes the current Selection Filters. The Help icon displays the SolidWorks Online Users Guide.




Display the Trimetric view.

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



Remove Tangent edges.

**199)** Check the **Removed** box from the System options tab: Display/Selection.

Display Tangent edges.

**200)** Check the **As visible** box from the System options tab.

Save the BATTERY.

**201)** Click **Save** .

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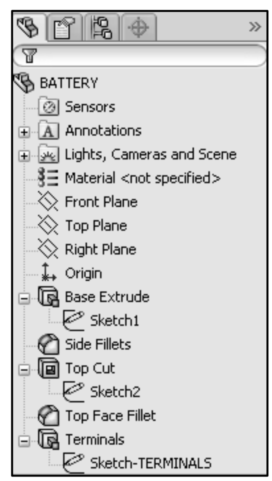
## Review of the BATTERY Part




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

The Extruded Base feature was created using the Instant3D tool. Blind was the default End Condition. The Fillet feature rounded sharp edges. All four edges were selected to combine common geometry into the same Fillet feature. The Fillet feature also rounded the top face. The Sketch Offset Entity created the profile for the Extruded Cut feature.

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

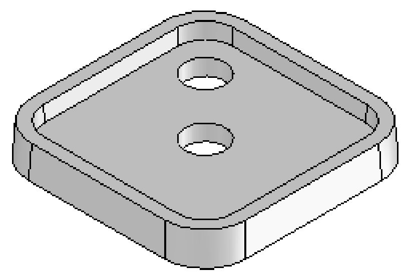
A centerline was required to locate the two holes with an angular dimension. The Draft Angle option tapered the Extruded Boss feature. All feature names were renamed.

### BATTERYPLATE Part

The BATTERYPLATE is a critical plastic part. The BATTERYPLATE:

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

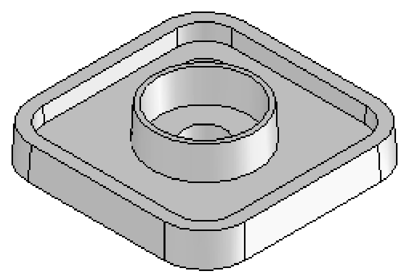
Design the BATTERYPLATE. Utilize features from the BATTERY to develop the BATTERYPLATE. The BATTERYPLATE is manufactured as an injection molded plastic part. Build Draft into the Extruded Base/Boss features.



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

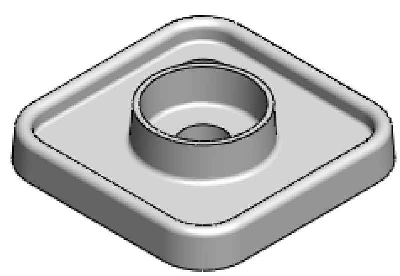
Modify the dimensions of the Base feature. Add a 3 degree draft angle.

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



Insert an Extruded Boss 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 features together into a folder.

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

### Save As, Delete, Edit Feature, and Modify

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

Reuse existing geometry. Create two holes. Delete the Terminals feature and reuse the circle sketch. Select the sketch in the FeatureManager. Create an Extruded Cut feature from the Sketch–TERMINALS using Instant3D. Blind is the default End Condition.

Edit the Base Extrude feature. Modify the overall depth. Rebuild the model.

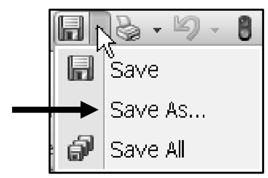
Sketch dimensions are displayed in black.

Feature dimensions are displayed in blue. .

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

Create a new part.

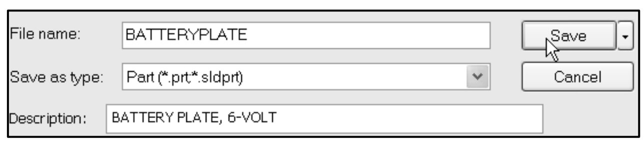
**202)** Click **Save As** from the Menu bar.




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

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

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



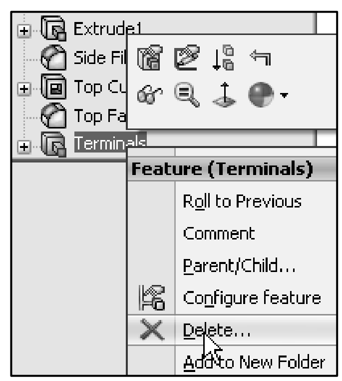

**206)** Click **Save**. The BATTERYPLATE FeatureManager is displayed. The BATTERY part is closed.

Delete the Terminals feature.

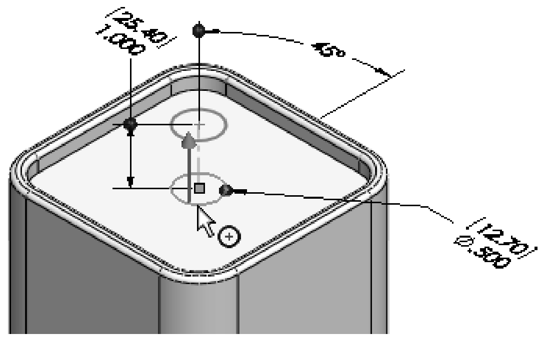
**207)** Right-click **Terminals** from the FeatureManager.

**208)** Click **Delete**.

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



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

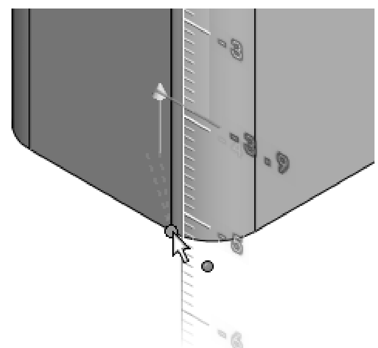



**210)** Click **Sketch-TERMINALS** from the FeatureManager.

**211)** Click the **circumference** of the center circle as illustrated. A green arrow is display.

**212)** Hold the **Alt** key down. Drag the **green arrow** downward below the model to create a hole in Direction 1.

**213)** Release the mouse button on the **vertex** as illustrated. This ensures a Through All End Condition with model dimension changes.



**214)** Release the **Alt** key. Extrude2 is displayed in the FeatureManager.

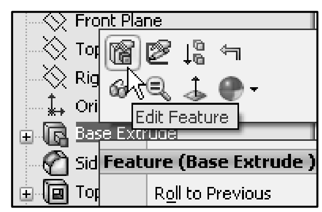
**215)** Rename the **Extrude2** feature to **Holes** in the FeatureManager.

Edit the Base Extrude feature.

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

**217)** Click **Edit Feature** from the Context toolbar. The Base Extrude PropertyManager is displayed.

Modify the overall depth.




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

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

**220)** Enter **3.00**deg in the Angle box.

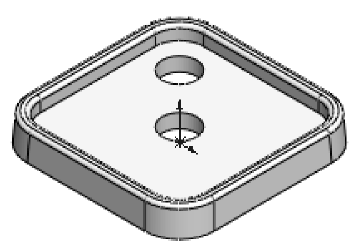
**221)** Click **OK** from the Base Extrude PropertyManager.

Fit the model to the Graphics window.

**222)** Press the **f** key.

Save the BATTERYPLATE.

**223)** Click **Save**.



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

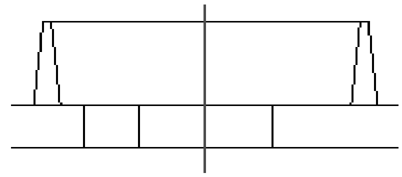
---

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

## BATTERYPLATE Part-Extruded Boss Feature

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

Draft Angle displayed at 5°



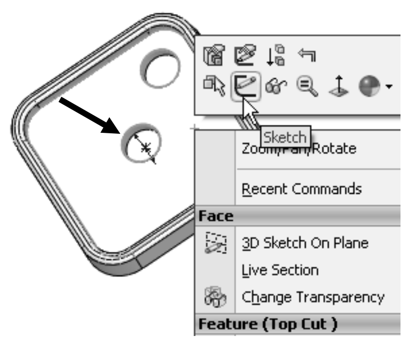
When applying the draft angle to the two concentric circles, the outside face tapers inwards and the inside face tapers outwards.

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

### Activity BATTERYPLATE Part-Extruded Boss Feature

Select the Sketch plane.

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




Create an offset sketch.

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

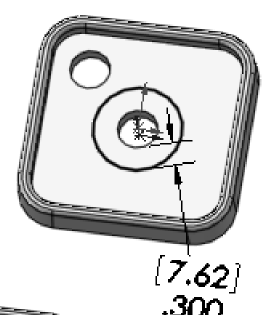
**226)** Click the **top circular edge** of the center hole. Note: Use the keyboard arrow keys or the middle mouse button to rotate the sketch if needed.

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

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

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

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

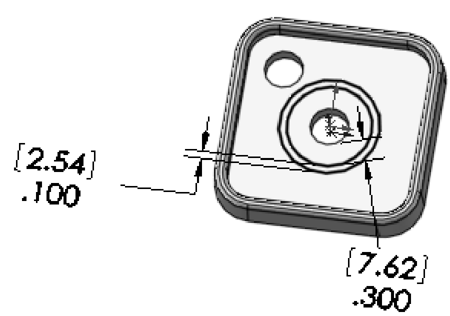


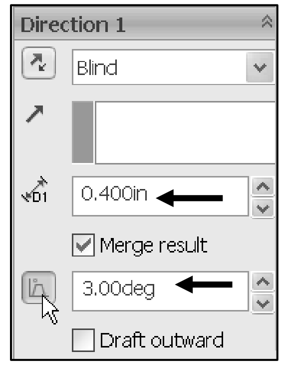

Create the second offset sketch.

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

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

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






**234)** Click **OK** from the Offset Entities PropertyManager. Drag the dimension off the model.

Two offset concentric circles define the sketch.

Insert an Extruded Boss feature.

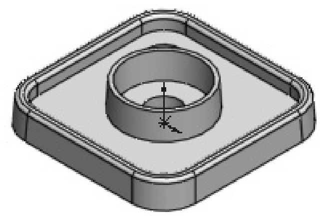
**235)** Click the **Extruded Boss/Base** feature tool. The Extrude PropertyManager is displayed.

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

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

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

**239)** Click **OK** from the Extrude PropertyManager. The Extrude feature is displayed in the FeatureManager.



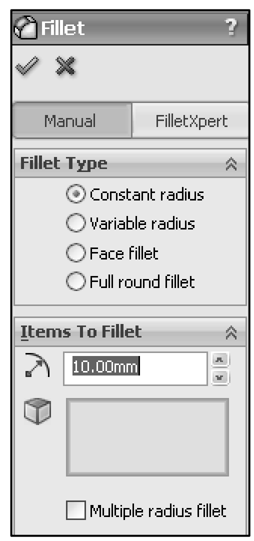
Rename the feature.

**240)** Rename the **Extrude3** feature to **Holder** in the FeatureManager.

Save the model.

**241)** Click **Save** .

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




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

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

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

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

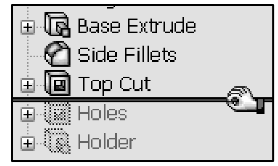
Delete the Top Edge Fillet.

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

**243)** Click **Delete**.

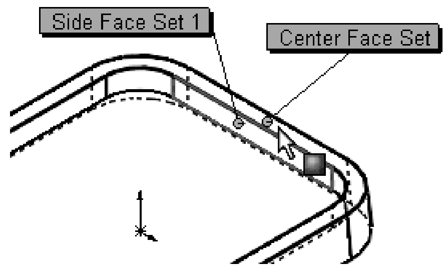
**244)** Click **Yes** to confirm delete.

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




Create a Full round fillet feature.

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




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

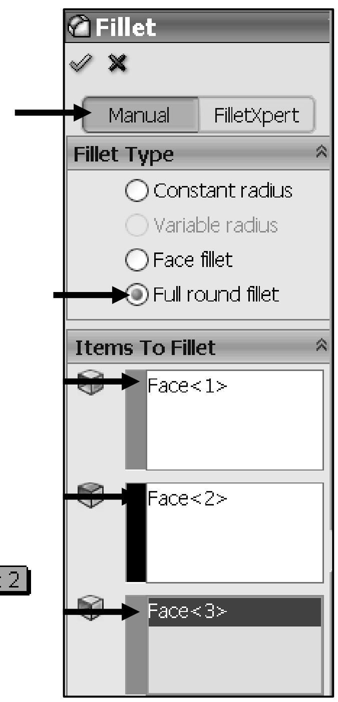
**248)** Click the **Manual** tab.

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

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

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

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




Rotate the part.

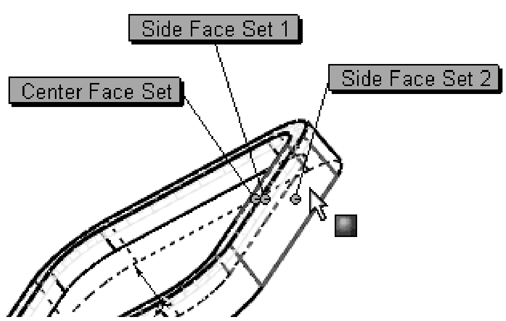
**253)** Press the **Left Arrow** key until you can select the outside Base Extrude face.

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

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

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

**257)** Rename **Fillet1** to **TopFillet**.



Save the BATTERYPLATE.

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

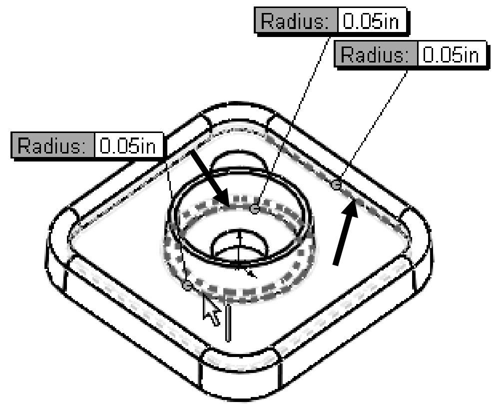
**259)** Click **Hidden Lines Removed** from the Heads-up View toolbar.

**260)** Drag the **Rollback bar** to the bottom of the FeatureManager.

**261)** Click **Save**.

Create a Multiple radius fillet feature.

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




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

**264)** Click the **Constant radius** box.

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

**266)** Click the **bottom inside circular edge** of the Top Cut as illustrated.

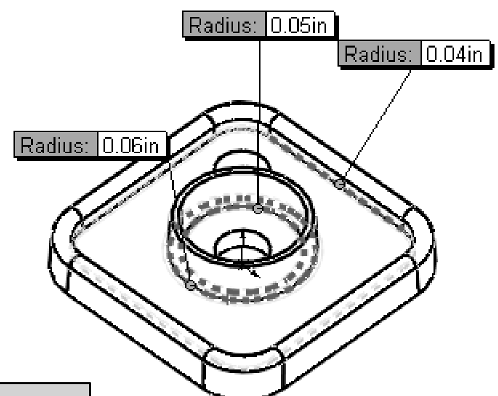
**267)** Click the **inside edge** of the Top Cut.

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

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

Modify the Fillet values.

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




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

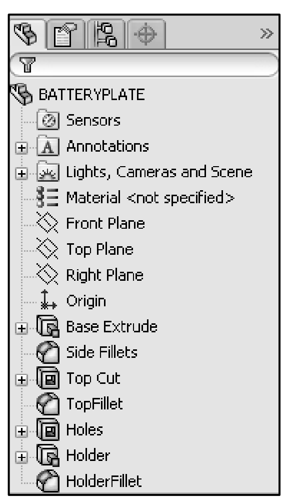
**272)** Click the **Radius** box for the Top Cut inside edge.

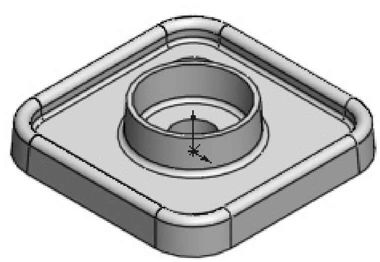
**273)** Enter **0.040**in, [**1.02**].

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

**275)** Rename **Fillet2** to **HolderFillet**.

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



Group the Fillet features into a new folder.

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

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

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

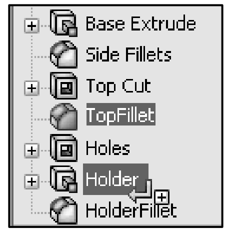
**280)** Hold the **Ctrl** key down.

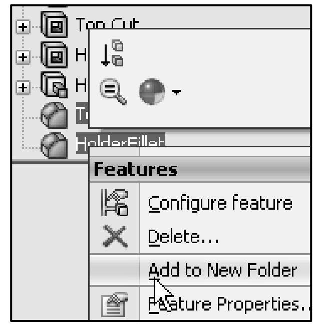
**281)** Click **TopFillet** in the FeatureManager.

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

**283)** Release the **Ctrl** key.

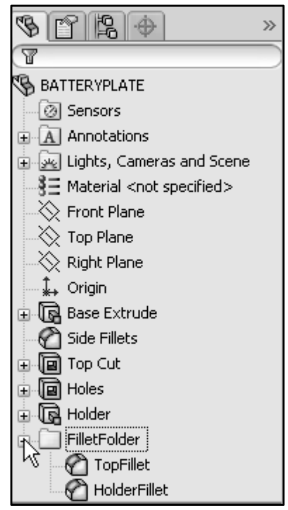
**284)** Rename **Folder1** to **FilletFolder**.



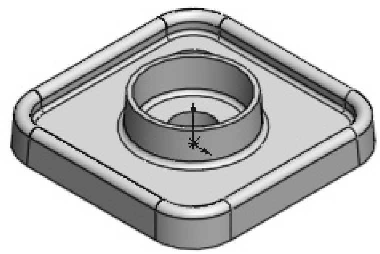

Save the BATTERYPLATE.

**285)** Click **Save**.




Exit SolidWorks.

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

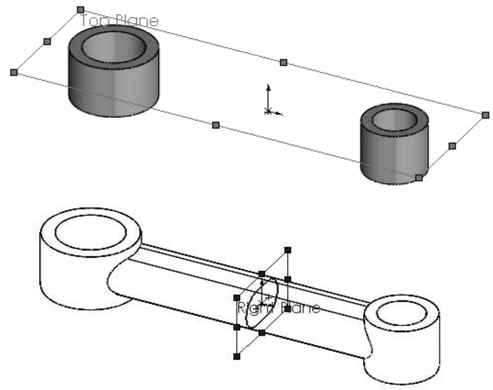


## Multi-body Parts and Extruded Boss Feature

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.

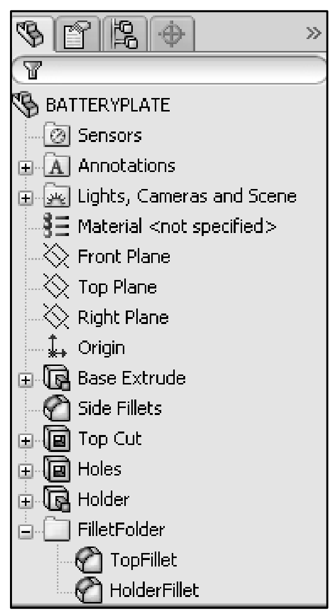



Multi-body part Wrench

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 created a hole in the BATTERYPLATE using Instant3D and modified features using the PropertyManager.




The BATTERYPLATE is a plastic part. The Draft Angle option was added in the Extruded Base feature.

The Holder Extruded Boss utilized a circular sketch and the Draft Angle option. The Sketch Offset tool created the circular ring profile. Multi radius Edge Fillets and Face Fillets removed sharp edges. Similar Fillet features were grouped together into a folder. Features were renamed in the FeatureManager. The BATTERY and BATTERYPLATE utilized an Extruded Base feature.

## Project Summary

SolidWorks is a design software application used to model and create 2D and 3D sketches, 3D parts, 3D assemblies, and 2D drawings. In Project 1, you were introduced to the SolidWorks 2009 User Interface and CommandManager: Menu bar toolbar, Menu bar menu, Drop-down menus, Context toolbars, Consolidated drop-down menus, System feedback icons, Confirmation Corner, Heads-up View toolbar, and Document Properties.

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 three major features in this project: Extruded Boss/Base, Extruded Cut, and Fillet. You addressed the following Sketch tools in this project: Smart Dimension, Sketch Entities, Convert Entities, Offset Entities, Line, Center Rectangle, Circle, and Centerline.

You addressed additional tools that utilized existing geometry: Add Relations, copy, Save As, Edit feature, and more.

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

## Project Terminology

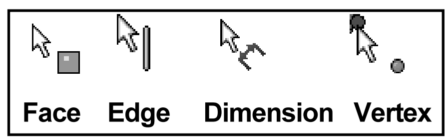
**Assembly:** An assembly is a document in which parts, features, and other assemblies (sub-assemblies) are put together. A part in an assembly is called a component. Adding a component to an assembly creates a link between the assembly and the component. When SolidWorks opens the assembly, it finds 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).

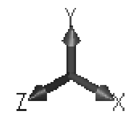
**Template:** A template is the foundation of a SolidWorks document. A Part Template contains the Document Properties such as: Dimensioning Standard, Units, Grid/Snap, Precision, Line Style and Note Font.

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

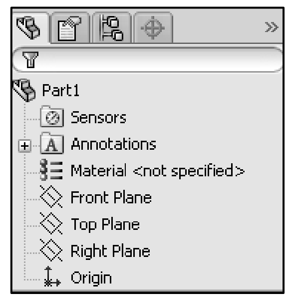
**Units:** Used in the measurement of physical quantities. Decimal inch dimensioning and Millimeter dimensioning are the two types of common units specified for engineering parts and drawings.

## 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.

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 | | |

## Exercises

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

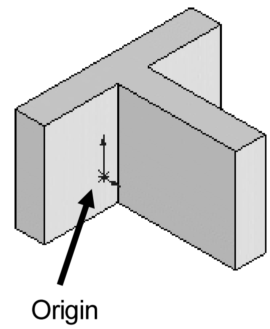
A: Top Plane

B: Front Plane

C: Right Plane

D: Left Plane

Correct answer _______.




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

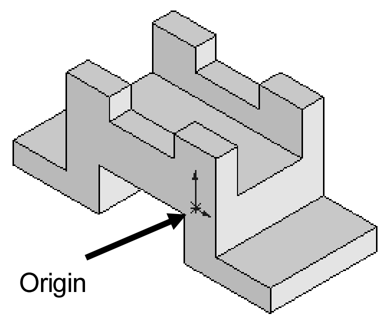
A: Top Plane

B: Front Plane

C: Right Plane

D: Left Plane

Correct answer ________.




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

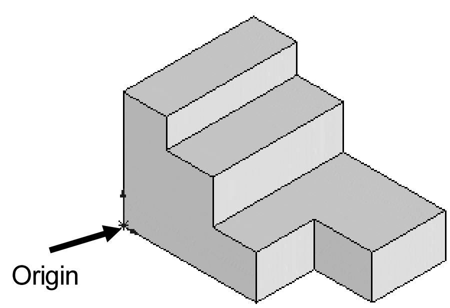
A: Top Plane

B: Front Plane

C: Right Plane

D: Left Plane

Correct answer ________.



**Exercise 1.4: Part Document Templates**

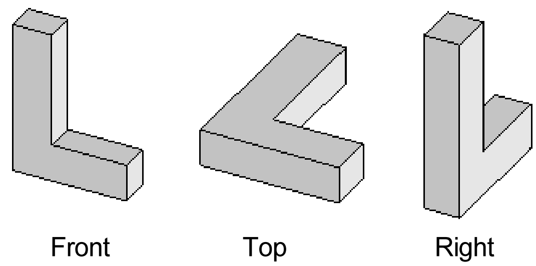
Create a Metric part document template using an ANSI dimension standard.

**Exercise 1.5: L-SHAPE Part**

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

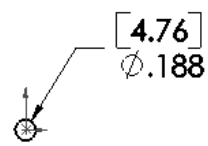
Utilize your own dimensions.

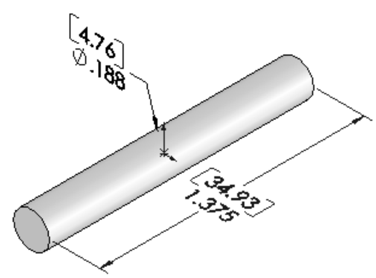
Locate each profile on a different Sketch Plane as illustrated.




**Exercise 1.6: AXLE**

Create an AXLE part as illustrated with dual units.

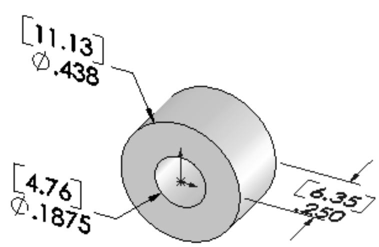



- Utilize the Front Plane for the Sketch plane.
- Utilize the Mid Plane End Condition. The AXLE is symmetric about the Front Plane. Note the location of the Origin.
- Apply 6061 Alloy as a material.
- Display dual dimensions.

**Exercise 1.7: SHAFT-COLLAR**

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

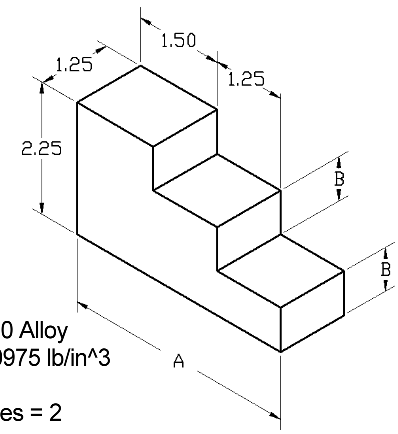



- Utilize the Front Plane for the Sketch plane. Note the location of the Origin.
- Use the provided dimenions.
- Note the location of the Origin.
- Apply 2014 Alloy as a material.
- Apply a color to the part.
- Display dual dimensions.

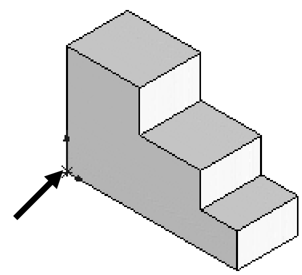
## Exercise 1.8

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

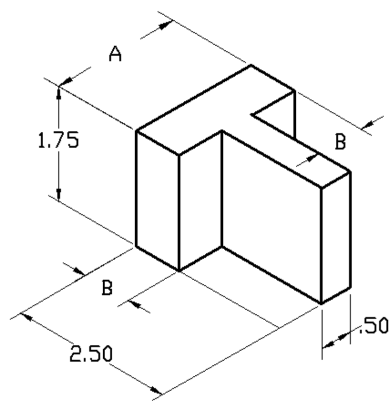


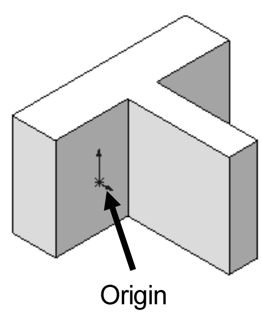
## Exercise 1.9

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



## 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