

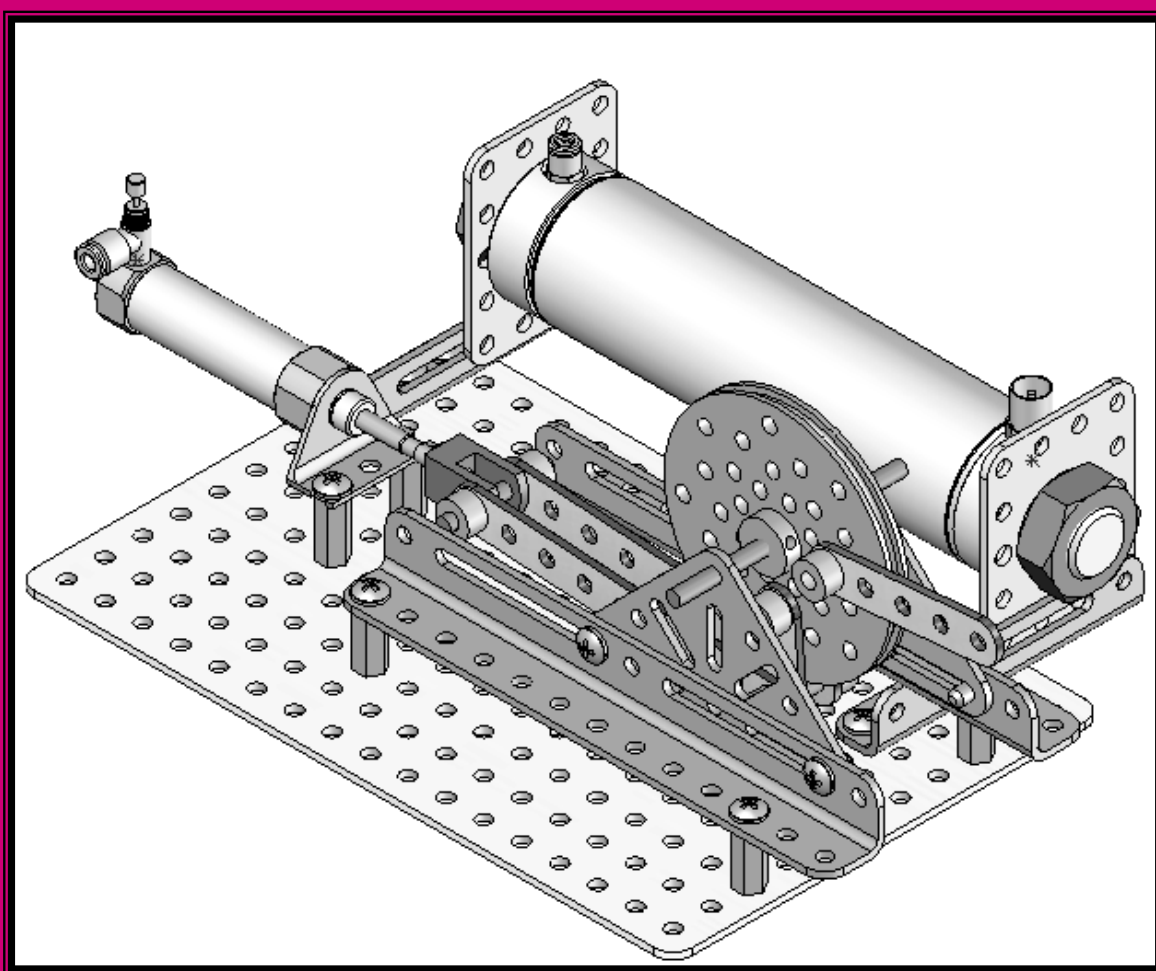
INSIDE:



SolidWorks 2007 Tutorial with MultiMedia CD

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

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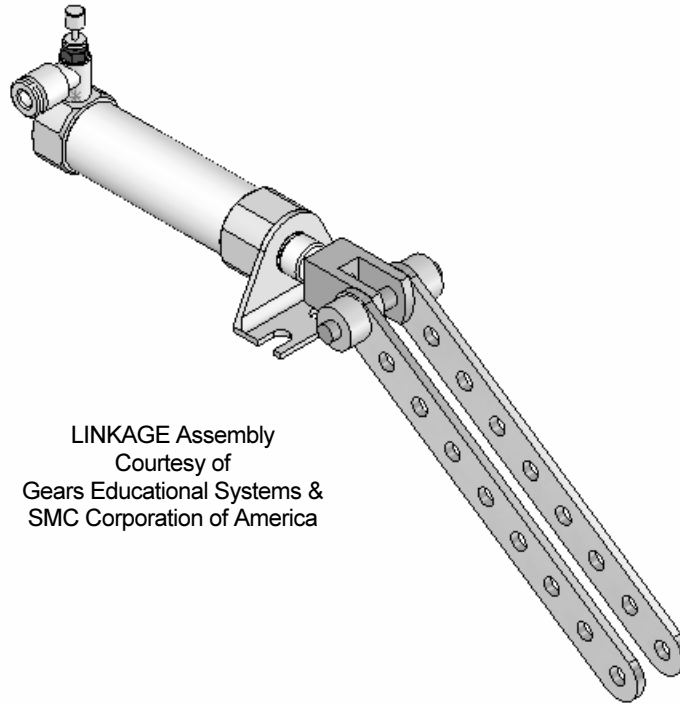
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3S
SolidWorks

Solution
Partner

Project 1

LINKAGE Assembly



LINKAGE Assembly
Courtesy of
Gears Educational Systems &
SMC Corporation of America

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

Desired Outcomes:	Usage Competencies:
<ul style="list-style-type: none">• Create three parts:<ul style="list-style-type: none">○ AXLE.○ SHAFT-COLLAR.○ FLATBAR.	<ul style="list-style-type: none">• Establish a SolidWorks session.• Understand the SolidWorks Interface.• Develop new parts.• Apply the following features: Extruded Base, Extruded Cut, and Linear Pattern.
<ul style="list-style-type: none">• Create an assembly:<ul style="list-style-type: none">○ LINKAGE assembly.	<ul style="list-style-type: none">• Insert Components into an Assembly.• Insert the following Standard Mates: Concentric, Coincident, and Parallel.

Notes:



Project 1 – LINKAGE Assembly

Project Objective

Provide a basic understanding of the SolidWorks User Interface: Menus, Toolbars, System feedback, System options, Keyboard shortcuts, and Document Properties.

Obtain the knowledge of the following SolidWorks features: Extruded Base, Extruded Cut and Linear Pattern.

Create three individual parts:

- AXLE.
- SHAFT-COLLAR.
- FLATBAR.

Create an assembly using the three created parts and a sub-assembly downloaded from the CD in the book:

- LINKAGE assembly.

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

- Establish a SolidWorks session.
- Set units and dimensioning standards for a SolidWorks document.
- Generate a 2D sketch.
- Add and modify dimensions.
- Create a 3D model.
- Insert Geometric Relations.
- Download an assembly into SolidWorks.
- Create an assembly.
- Insert Coincident, Concentric, and Parallel Mates into an assembly.
- Use and understand the following SolidWorks Features:
 - Extruded Base, Extruded Cut, and Linear Pattern.

Project Overview

SolidWorks is a design automation software package used to produce and model parts, assemblies, and drawings.

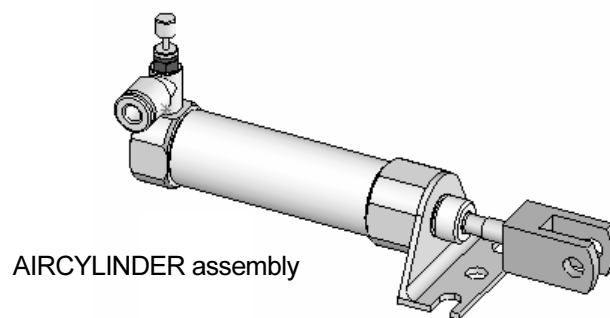
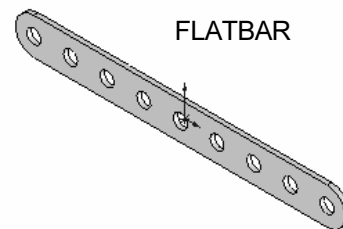
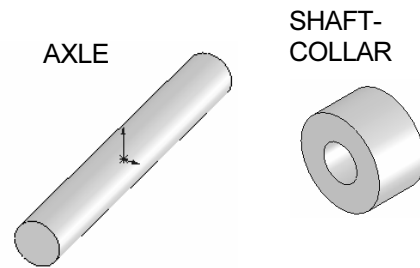
SolidWorks is a 3D solid modeling CAD program. SolidWorks provides design software to create 3D models and 2D drawings.

Create three parts in this project:

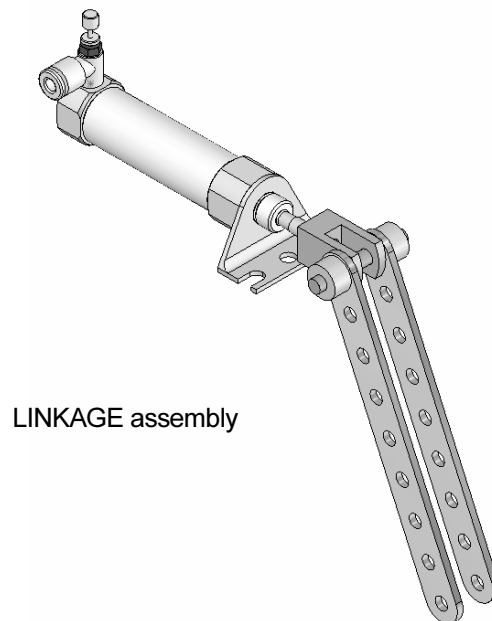
- AXLE.
- SHAFT-COLLAR.
- FLATBAR.

Download the AIRCYLINDER assembly from the enclosed CD.

The AIRCYLINDER assembly is also available to download from the World Wide Web.

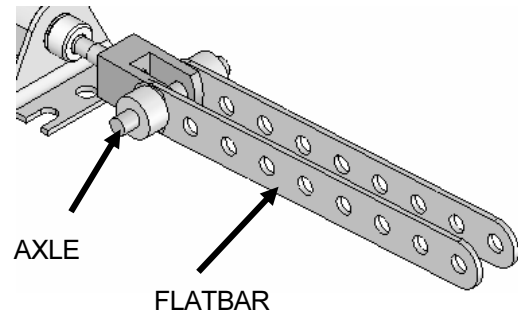


Combine the created parts and the downloaded AIRCYLINDER assembly to create the LINKAGE assembly.



AXLE Part

The AXLE is a cylindrical steel rod. The AXLE supports the two FLATBAR parts.



The AXLE rotates about its axis. The dimensions for the AXLE are determined from the other components in the LINKAGE assembly.

Start a new SolidWorks session. Create the AXLE part.

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

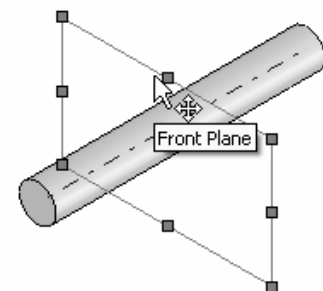
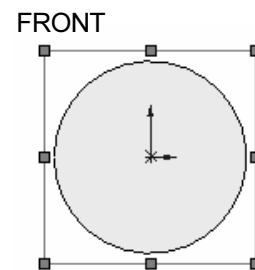
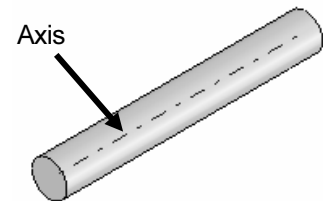
Utilize the Extruded Base feature. The Extruded Base feature adds material. The Base feature is the first feature of the part. The Base feature is the foundation of the part. Keep the Base feature simple!

The Base feature geometry for the AXLE is an extrusion. How do you create a solid Extruded Base feature for the AXLE?

- Sketch a circular profile on the Front plane, centered at the Origin.
- Extend the profile perpendicular (\perp) to the Front plane.

Utilize symmetry. Extrude the sketch with the Mid Plane End Condition in Direction 1. The Extruded Base feature is centered on both sides of the Front plane.

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 to obtain additional knowledge on SolidWorks.

Open a new part. Select File, New from the Main menu. 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 2007 session.

- 1) Click **Start** from the Windows Taskbar. Click **All Programs**  .
- 2) Click the **SolidWorks 2007** folder.
- 3) Click the **SolidWorks 2007**  SolidWorks 2007 application. The SolidWorks program window opens. Note: Do not open a document at this time.



If available, double-click the SolidWorks 2007 icon on the Windows desktop to start a SolidWorks session.

Read the Tip of the Day dialog box.

- 4) Click the **Collapse arrow**  in the Task Pane to close the Tip of the Day. 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.




The SolidWorks 2007 Task Pane is displayed when a SolidWorks session starts. The Task Pane can be displayed in the following states: Visible or hidden, expanded or collapsed, pinned or unpinned, docked or floating. The Task Pane contains the following standard tabs: SolidWorks Resources, Design Library, File Explorer, Search, View Palette, and PhotoWorks Items.



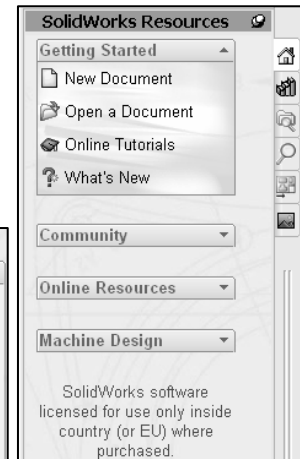
If you do not see the PhotoWorks Items tab, click Tools, Add-Ins from the Main menu. Check the PhotoWorks box. Click OK.



SolidWorks Resources

Utilize the left/right arrows  to expand or collapse the Task Pane options. The basic SolidWorks Resources menu displays the following default selections:

- Getting Started.
- Community.
- Online Resources.
- Tip of the Day.

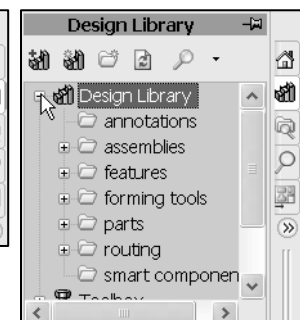
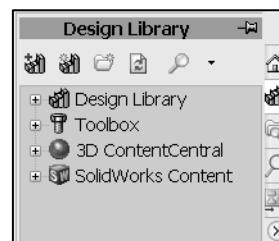


Other user interfaces are available to be displayed during the initial software installation selection: Machine Design, Mold Design, or Consumer Products Design. The illustration displays the Machine Design user interface.

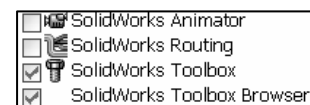
Design Library


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

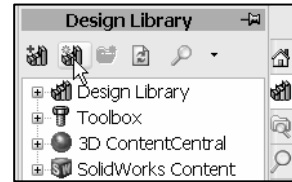
The Design Library tab contains four default selections. Each default selection contains additional sub categories. The selections are: Design Library, Toolbox, 3D ContentCentral, and SolidWorks Content.



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

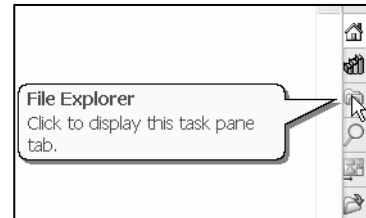


To access the Design Library folders in a non network environment, click Add File Location , enter: C:\Program Files\SolidWorks\data\design library. Click OK. In a network environment, contact your IT department for system details.



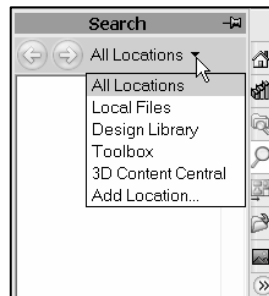
SolidWorks File Explorer

File Explorer in the Task Pane duplicates the function of Windows Explorer from your local computer and displays the recent and open documents in the SolidWorks directories.

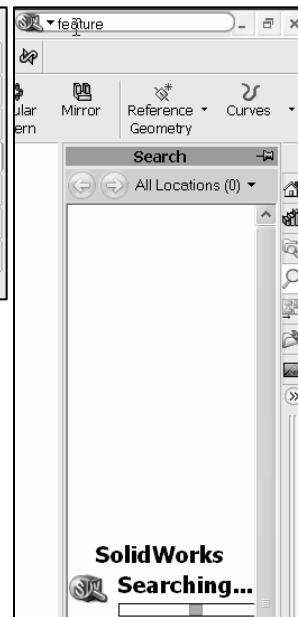



SolidWorks Search

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



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



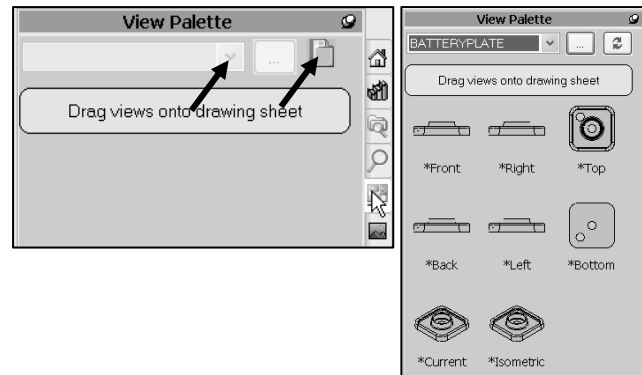
The Search tool  in the Task Pane searches the following locations:

- All locations.
- Local Files.
- Design Library.
- SolidWorks Toolbox.
- 3D ContentCentral.
- Added location.

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

View Palette

Use the View Palette, located in the Task Pane, to insert drawing views of an active part, or click the Browse button to locate the desired model. Click and drag the desired view from the View Palette into an active drawing sheet to create a drawing view.



Auto Recovery

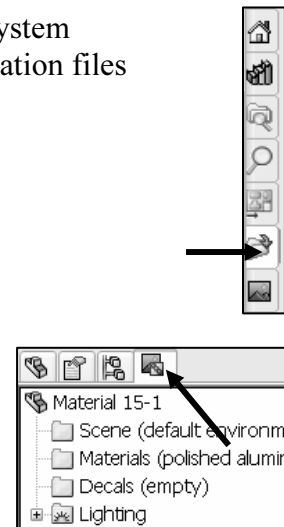
If auto recovery is initiated in the System Options section and the system terminates unexpectedly with an active document, the saved information files are available on the Task Pane Document Recovery tab.

PhotoWorks

PhotoWorks Items create photo-realistic images of SolidWorks models. PhotoWorks provides many professional rendering effects. PhotoWorks contains the following default folders: Scene, Materials, Decals, and Lighting.

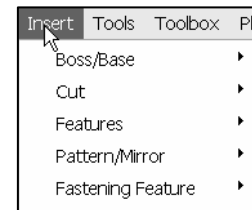


Click Tools, Add-Ins..., PhotoWorks from the Main menu to activate the PhotoWorks feature.



Drop-Down Menu

SolidWorks takes advantage of the familiar Microsoft Windows[®] graphical user interface. Communicate with SolidWorks either through the drop-down menus or the application toolbars. A command is an instruction that informs SolidWorks to perform a task.

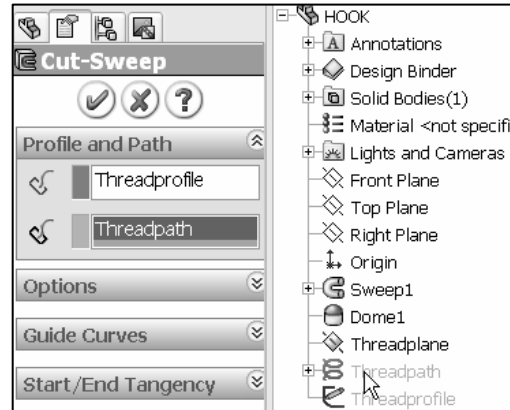


To close a SolidWorks drop-down menu, press the Esc key. You can also click inside the SolidWorks Graphics window, or click another drop-down menu. The drop-down menu options perform three basic functions. They are:

- Displays a SolidWorks dialog box.
- Submits a command to create or modify a model or drawing.
- Offers an expanded set of tools located in the SolidWorks toolbars.

Fly-out FeatureManager

The fly-out FeatureManager design tree allows you to view and select items in the PropertyManager and the FeatureManager design trees at the same time. The fly-out FeatureManager provides the ability to select items which may be difficult to view or select from the Graphics window. You can hide, modify the transparency of, go to, or zoom to selected items.



You cannot suppress items or roll back the build.

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

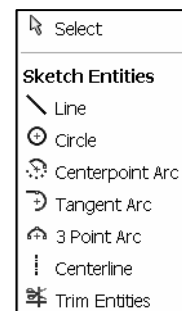


Another method for accessing a command is to use an accelerator key. Accelerator keys are special keystrokes which activate the drop-down menu options. Some commands in the menu bar and items in the drop-down menus have an underlined character. Pressing the Alt key followed by the key corresponding to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

Cut	Ctrl+X
Copy	Ctrl+C
Paste	Ctrl+V
Delete	Del

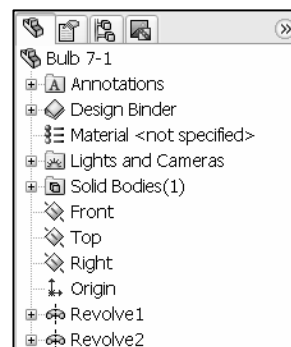
Right Click Pop-up menus

Right-click in the Graphics window to display a context-sensitive shortcut menu. If you are in the middle of a command, this menu displays a list of options related to that command. Example: if you right-click your mouse before picking the first point of the Rectangle tool, a menu is displayed in the Graphics window. The menu displays Sketch Entities, Selected Entity, and other Relations and menu options.



FeatureManager design tree

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



Understand the FeatureManager design tree to troubleshoot your model. The FeatureManager is used extensively throughout this book. Expand, collapse, and scroll the FeatureManager design tree.

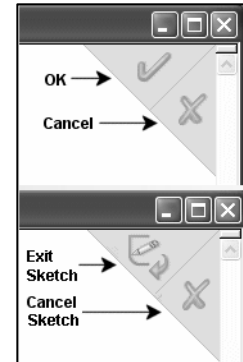


To collapse all items in the FeatureManager, right-click and select Collapse items, or press the Shift+C keys.

Confirmation Corner

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

When you activate or open a sketch, the confirmation corner box displays two symbols. The first symbol is the sketch tool icon. The second symbol is a large red X. These two symbols supply a visual reminder that you are in an active sketch.



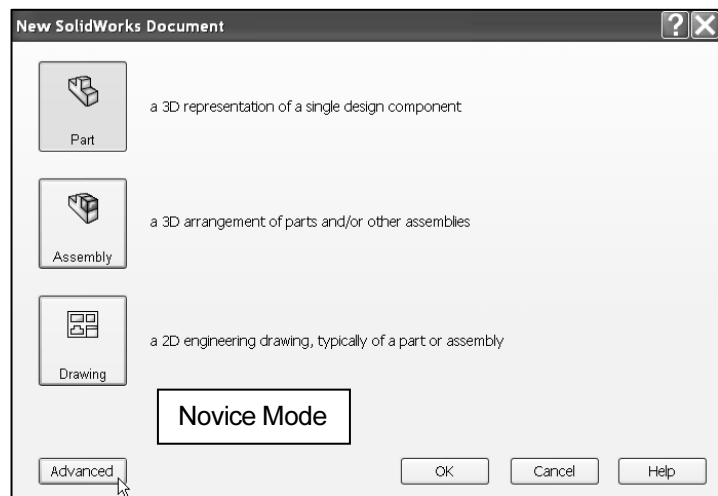
There are two modes in the New SolidWorks Document dialog box: Novice and Advanced. The Novice option is the default option with three templates. The Advanced option contains access to more templates.

Create a new part.

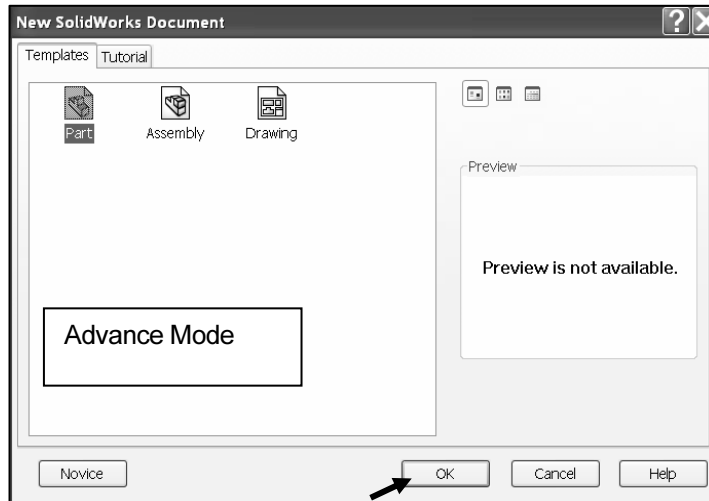
- 5) Click **File, New** from the Main menu.

Select Advanced Mode.

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

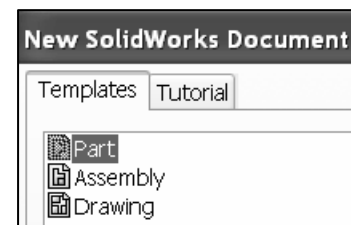


- 7) The Templates tab is the default tab. Part is the default template from the New SolidWorks Document dialog box. Click **OK**.



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


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



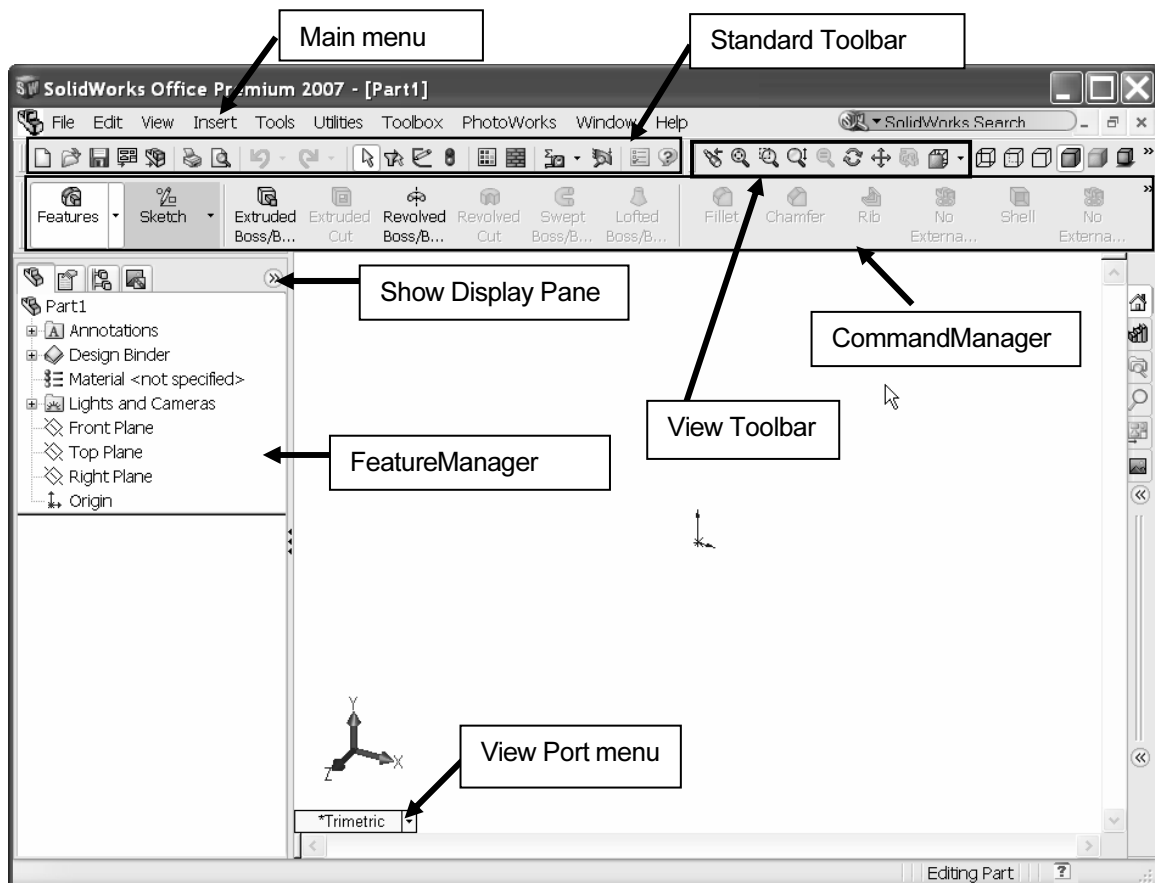
User Interface and CommandManager

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

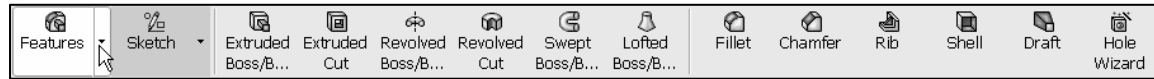
Part1 is displayed. Part1 is the new default part window name. The Main menu, Standard Toolbar, View Toolbar, and CommandManager are displayed above the Graphics window.

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

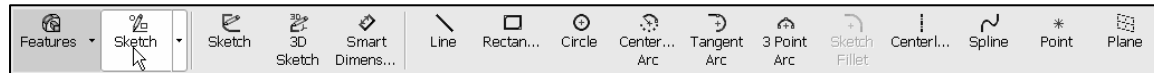
The FeatureManager contains a list of features, reference geometry, and settings utilized in the part.



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



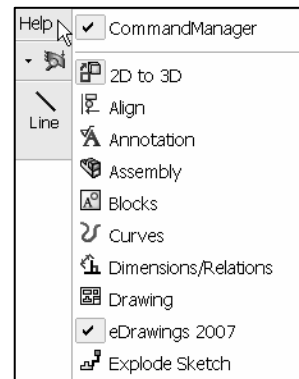
Features Toolbar



Sketch Toolbar

The CommandManager is utilized in this text. Control the CommandManager display. Right-click in the gray area to the right of the Help entry, in the Main menu. A complete list of toolbars is displayed. Check CommandManager if required.

Select individual toolbars from the toolbar list to display in the Graphics window. Reposition toolbars by clicking and dragging their drag handle.

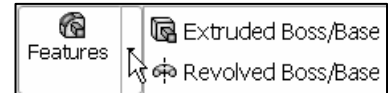


Drag Handle

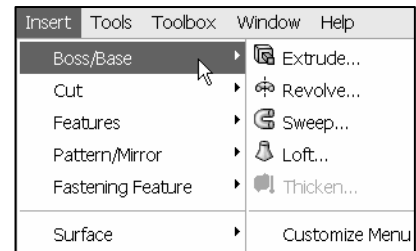
Enter commands from the CommandManager, individual toolbars, and Main menus.

Example:

- Click Extruded Boss/Base in the CommandManager.
- Click Extruded Boss/Base from the Features toolbar.
- Click Insert, Boss/Base, Extrude from the Main menu.



Enter additional commands through Shortcut keys and the Right Mouse button Pop-up menus.



Activity: AXLE Part

Set the Document Properties.

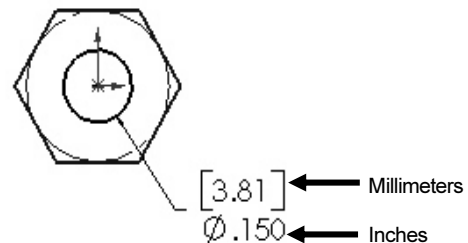
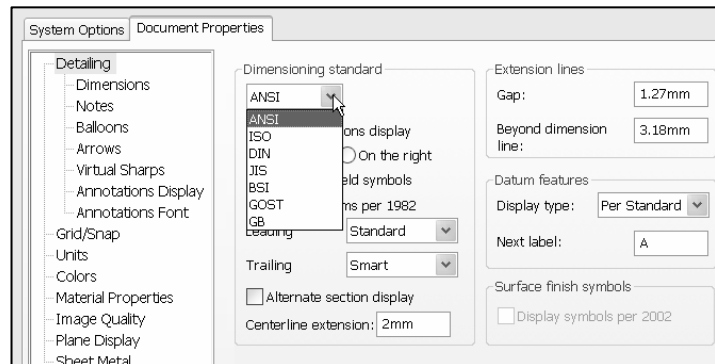
- 8) Click **Tools, Options, Document Properties** tab from the Main menu.
- 9) Select **ANSI** from the Dimensioning standard box.



Various detailing options are available depending on the selected standard.

The Dimensioning standard determines the display of dimension text, arrows, symbols, and spacing. Units are the measurement of physical quantities. Millimeter dimensioning and decimal inch dimensioning are the two most common unit types specified for engineering parts and drawings.

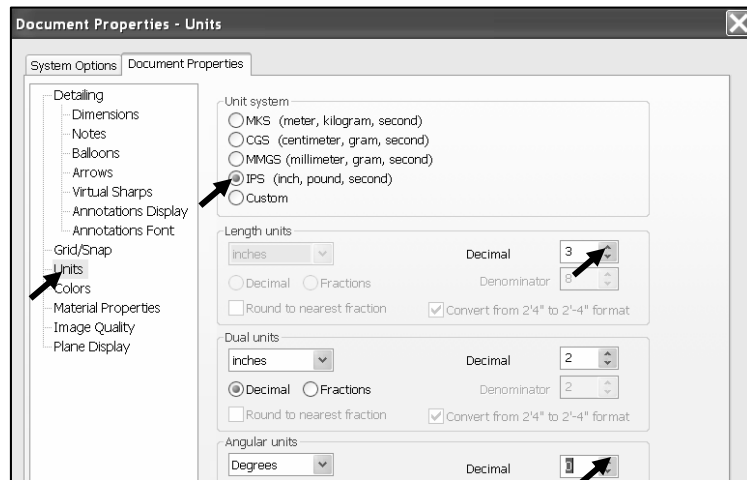
The primary units in this book are provided in IPS, (inch, pound, seconds). The optional secondary units are provided in MMGS, (millimeters, grams, second) and are indicated in brackets [].



Illustrations are provided in both inches and millimeters.

Set the part units.

- 10) Click **Units**. Click **IPS**, (inch, pound, second) [MMGS] for Unit system.
- 11) Select **3**, [2] for Length units Decimal places.
- 12) Select **0**, [0] for Angular units Decimal places.
- 13) Click **OK** from the Document Properties - Units dialog box. The Part FeatureManager is displayed.




Activity: AXLE Part-Extruded Base Feature


Insert a new Sketch for the Extruded Base feature.


- 14) Click **Front Plane** from the FeatureManager. This is your Sketch plane.

- 15) Click **Sketch**  from the Control Area.

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

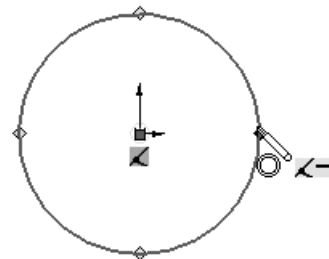
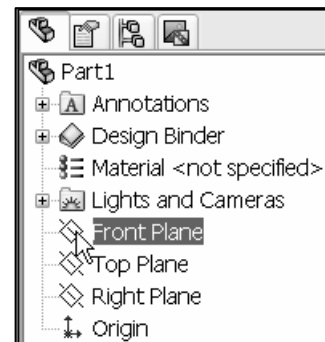
- 17) Drag the **mouse pointer** into the Graphics window. The

cursor displays the Circle feedback symbol . The center point of the circle is positioned at the Origin.

- 18) Click the **Origin**  of the circle. The cursor displays the Coincident to point feedback symbol.

- 19) Drag the **mouse pointer** to the right of the Origin to create the circle as illustrated.

- 20) Click a **position** to create the circle.



Add a dimension.

- 21) Click **Smart Dimension**  from the Sketch toolbar.

- 22) Click the **circumference** of the circle. The cursor displays the diameter feedback symbol.


- 23) Click a **position** diagonally above the circle in the Graphics window.

- 24) Enter **.188in, [4.78]** in the Modify dialog box.


- 25) Click the **Green Check mark**  in the Modify dialog box. The diameter of the circle is .188 inches.


The circular sketch is centered at the Origin. The dimension indicates the diameter of the circle.



If your sketch is not correct, select the **UNDO**  tool. To fit your sketch to the Graphics window, press the **f** key.

Extrude the sketch to create the Extruded Base Feature.

- 26) Click **Features**  from the CommandManager.

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

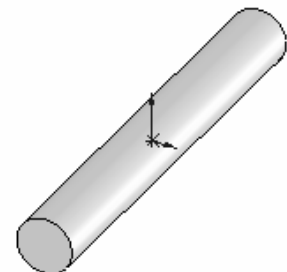
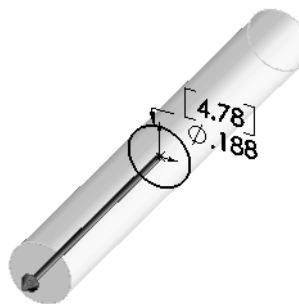
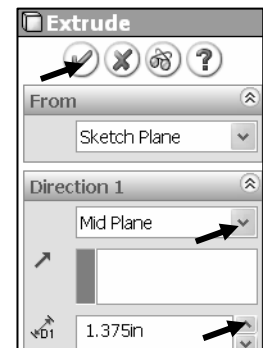
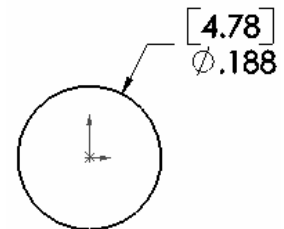
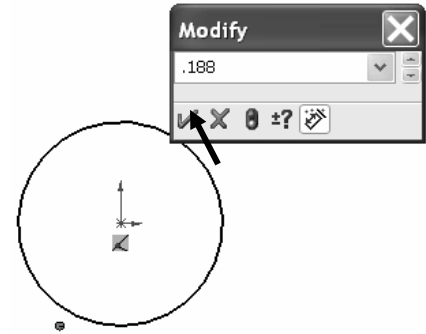
- 28) Select **Mid Plane** for End Condition in Direction 1.

- 29) Enter **1.375in, [34.93]** for Depth in Direction 1. Accept the default conditions.

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

Fit the model to the Graphics window.

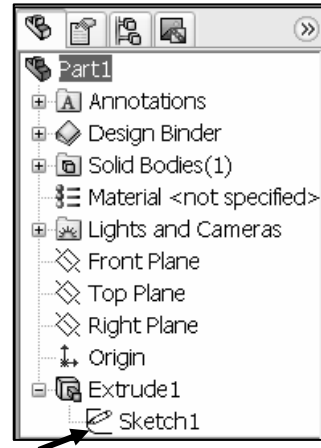
- 31) Press the **f** key.



The Extrude PropertyManager displays the parameters utilized to define the feature. The Mid Plane End Condition in the Direction 1 box extrudes the sketch equally on both sides of the Sketch plane. The Depth defines the extrude distance.

The Extrude1 feature name is displayed in the FeatureManager. The FeatureManager lists the features, planes and other geometry that construct the part. Extrude features add material. Extrude features require the following:

- Sketch plane.
- Sketch.
- Depth.



The Sketch plane is the Front plane. The Sketch is a circle with the diameter of .188in, [4.76]. The Depth is 1.375in, [34.93].

Activity: AXLE Part-Save

Save the part.

32) Click **File, Save As** from the Main Menu.

33) Double-click the **MY-DOCUMENTS** file folder.

34) Click **Create New Folder** .

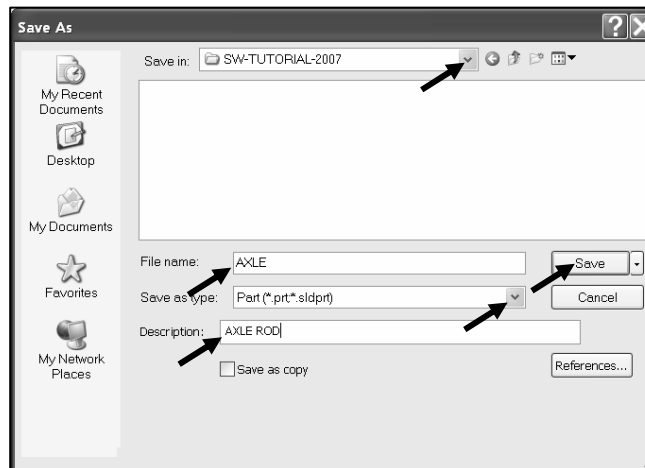
35) Enter **SW-TUTORIAL-2007** for the file folder name.

36) Double-click the **SW-TUTORIAL-2007** file folder. SW-TUTORIAL-2007 is the Save in file folder name.

37) Enter **AXLE** for the File name.

38) Enter **AXLE ROD** for the Description.


39) Click **Save**. The AXLE FeatureManager is displayed.



Organize parts into file folders. The file folder for this project is named: SW-TUTORIAL-2007. Save all documents in the SW-TUTORIAL-2007 file folder. Copy all files from the CD in the book to the SW-TUTORIAL-2007 folder.

Activity: AXLE Part-Edit Color

Modify the color of the part.

- 40) Click the **AXLE**  icon at the top of the FeatureManager.

- 41) Click **Edit Color**  from the Standard toolbar. The Color And Optics PropertyManager is displayed. AXLE is displayed in the Selection box.

- 42) Select a **light yellow** color from the Edit Color box.


- 43) Click **OK**  from the Color And Optics PropertyManager.

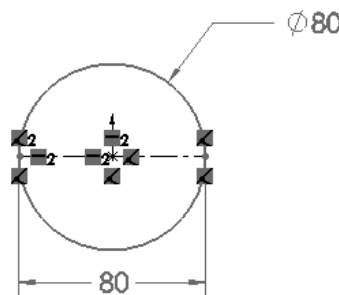
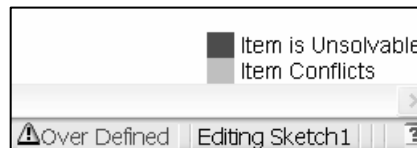


The SolidWorks FeatureManager design tree provides an indicator informing you on the status of your sketch. The sketch can either be:

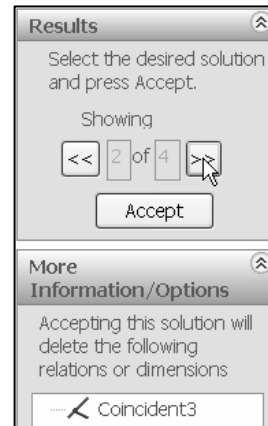
- 1.) (+) Over defined. The sketch is displayed in red.
- 2.) (-) Under defined. The sketch is displayed in blue.
- 3.) (?) Cannot be solved.
- 4.) No prefix: the sketch is fully defined. This is the ideal state. A fully defined sketch has complete information and is displayed in black.



 A new feature for 2007 is the SketchXpert PropertyManager. The SketchXpert PropertyManager provides the ability to diagnose an over defined sketch to create a fully defined sketch. Click Over Defined at the bottom of the Graphics window toolbar. The SketchXpert PropertyManager is displayed. Click the Diagnose button.



Select the desired solution and click the Accept button from the Results box.

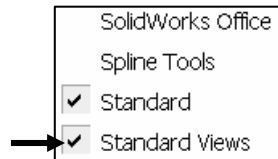


Activity: AXLE Part-Standard Views and View Modes

Display the Standard Views toolbar.

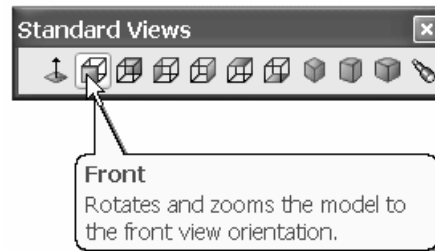
- 44)** If required, click **View, Toolbars, Standard Views** from the Main menu. The Standard Views toolbar is displayed below the Main menu.

- 45)** Position the **mouse pointer** on an individual toolbar icon to receive a ToolTip.



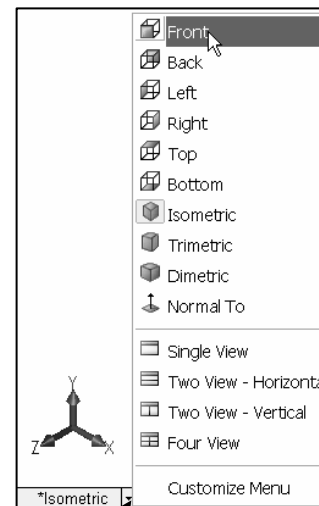
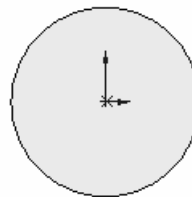
Orthographic projection is the process of projecting views onto parallel planes with \perp projectors. The default reference planes are the Front, Top and Right side viewing planes.

The Isometric view displays the part in 3D with two equal projection angles.



Display the Standard Views for the AXLE.

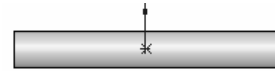
- 46)** Click **Front view**  **Front** from the Standard Views menu or from the View Port Menu in the Graphics window.



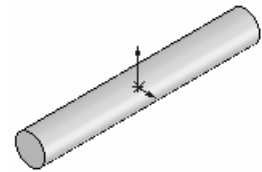
- 47) Click **Top view**  **Top**.



- 48) Click **Right view**  **Right**.




- 49) Click **Isometric view**  **Isometric**.




View modes manipulate the model in the Graphics window.

Display the various View modes.

- 50) Click **Zoom to Fit**  to display the full size of the part in the current window.




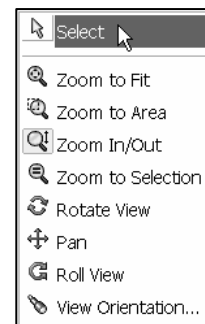
The defined view fits to the current window.


- 51) Click **Zoom In/Out** . Drag upward to **zoom in**.

- 52) Drag downward to **zoom out**. Press the lower case **z** key to zoom out. Press the upper case **Z** key to zoom in.


- 53) **Right-click** in the Graphics window. Click **Select**. Click the **front circular edge** of the AXLE in the Graphics window.

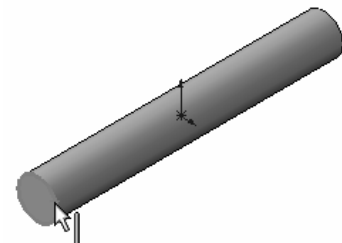
- 54) Click **Zoom to Selection** . The selected geometry fills the current window.



- 55) Click **Rotate** . Drag the **mouse pointer** to rotate about the screen center.

- 56) Use the computer keyboard **arrow keys** to rotate in 15-degree increments.

- 57) Click **Pan** . Drag the **mouse pointer** up, down, left, or right. The model scrolls in the direction of the mouse.



- 58) Right-click in the **Graphics window** area to display the zoom options.


- 59)** Click **Zoom to Fit** .

Note: View modes remain active until deactivated from the View toolbar or unchecked from the pop-up menu.




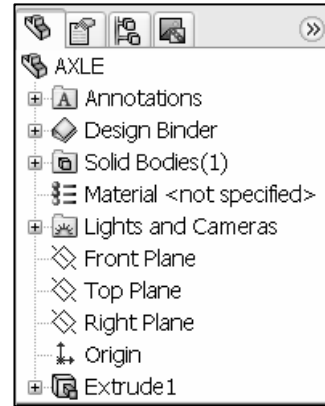
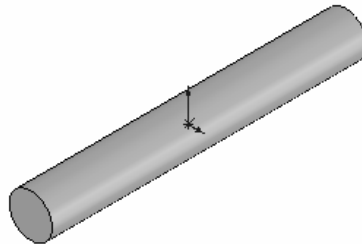
Utilize the center wheel of the mouse to Zoom In/Zoom Out and Rotate the model in the Graphics window.

Display the Isometric view.

- 60)** Click **Isometric view**  from the Standard Views toolbar.

Save the AXLE part.

- 61)** Click **Save** . The AXLE part is complete.



Review the AXLE Part.

The AXLE part utilized an Extruded Base feature. The Extruded Base feature adds material. The Extruded feature required a Sketch plane, Sketch, and Depth. The AXLE Sketch plane was the Front plane. The 2D circle was sketched centered at the Origin. A dimension defined the overall size of the sketch based on the dimensions of mating parts in the LINKAGE assembly.

The name of the feature is Extrude1. Extrude1 utilized the Mid Plane End Condition. The Extrude1 feature is symmetrical about the Front plane.

The Edit Color option modified the part color. Select the Part icon in the FeatureManager to modify the color of the part. Color and a prefix defines the sketch status. A blue sketch is under defined. A black sketch is fully defined. A red sketch is over defined.

The default Reference planes are the Front, Top, and Right side viewing planes. Utilize the Standard Views toolbar or the View Port menu in the lower left side of the Graphics window to display the principle views of a part. The View modes manipulate the model in the Graphics windows. Utilize Zoom, Pan, and Rotate from the View toolbar.

SHAFT-COLLAR Part

The SHAFT-COLLAR part is a hardened steel ring fastened to the AXLE part.

Two SHAFT-COLLAR parts are used to position the two FLATBAR parts on the AXLE.

Create the SHAFT-COLLAR part.

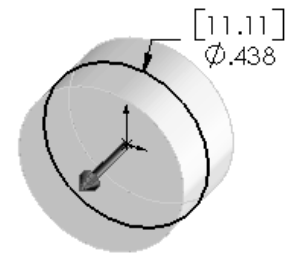
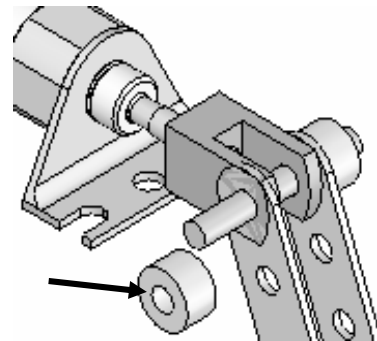
Utilize the Extruded Base feature. The Extruded Base feature requires a 2D circular profile.

Utilize symmetry. Sketch a circle on the Front plane centered at the Origin.

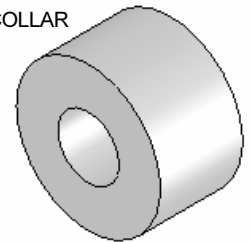
Extrude the sketch with the Mid Plane End Condition. The Extruded Base feature is centered on both sides of the Front plane.

The Extruded Cut feature removes material. Utilize an Extruded Cut feature to create a hole. The Extruded Cut feature requires a 2D circular profile. Sketch a circle on the front face centered at the Origin.

The Through All End Condition extends the Extruded Cut feature from the front face through all existing geometry.




SHAFT-COLLAR

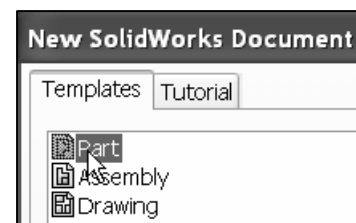


Activity: SHAFT-COLLAR Part-Extruded Base Feature

Create a new part.

62) Click **File, New**  from the Main Menu. The Templates tab is the default tab. Part is the default template from the New SolidWorks Document dialog box.

63) Double-click **Part**. The Part FeatureManager is displayed.



Save the Part.

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

65) Enter **SHAFT-COLLAR** for File name in the SW-TUTORIAL-2007 folder.

66) Enter **SHAFT-COLLAR** for Description.

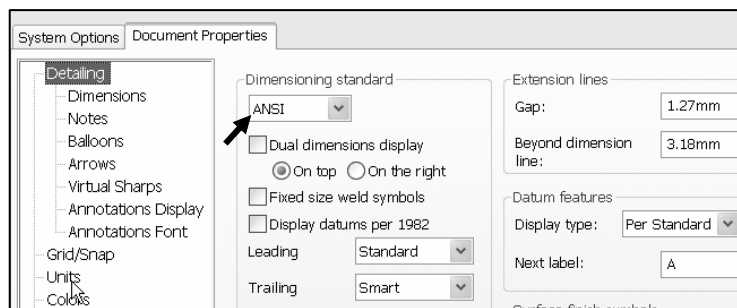
67) Click **Save**. The SHAFT-COLLAR FeatureManager is displayed.



Set the Dimension standard and part units.

68) Click **Tools, Options, Document Properties** tab from the Main menu.

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



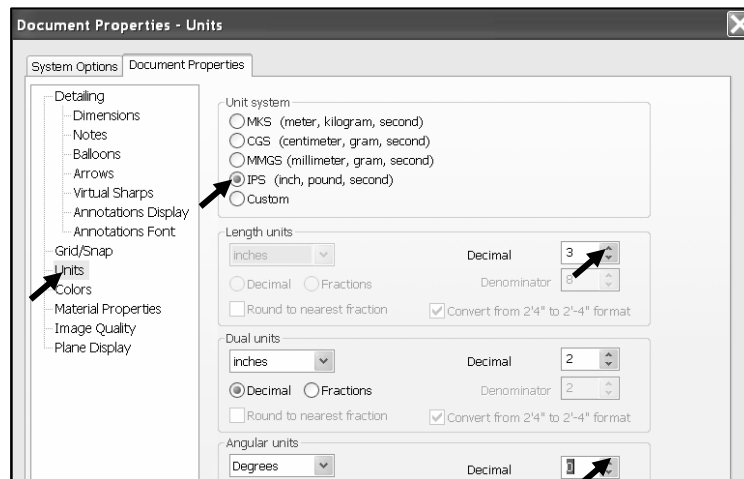
70) Click **Units**.

71) Click **IPS** (inch, pound, second), **[MMGS]** for Unit system.

72) Select **3, [2]** for Length units Decimal places.

73) Select **0, [0]** for Angular units Decimal places.


74) Click **OK** from the Document Properties - Units dialog box.





Insert a new Sketch for the Extruded Base feature.

- 75)** Click **Front Plane** from the FeatureManager. This is the Sketch plane.




- 76)** Click **Sketch**  from the Control Area.

- 77)** Click **Circle**  from the Sketch toolbar. Click the **Origin** . The cursor displays the Coincident to point feedback symbol.


- 78)** Drag the **mouse pointer** to the right of the Origin as illustrated. Click a **position** to create the circle.

Add a dimension.

- 79)** Click **Smart Dimension**  from the Sketch toolbar. Click the **circumference** of the circle. The cursor displays the diameter feedback symbol.

- 80)** Click a **position** diagonally above the circle in the Graphics window.

- 81)** Enter **.4375in**, **[11.11]** in the Modify dialog box.

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

Note: Three decimal places are displayed. The diameter value .4375 rounds to .438.

Extrude the sketch to create the Base feature.

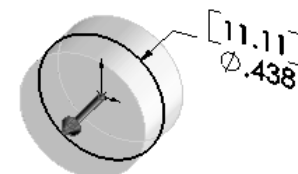
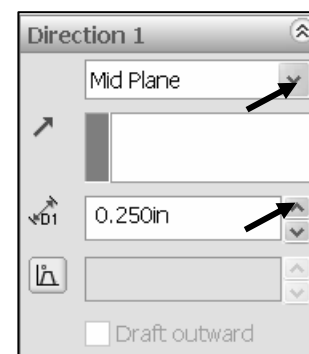
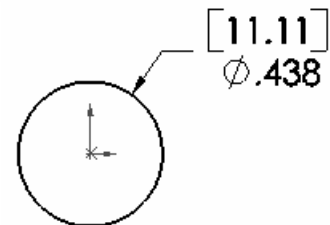
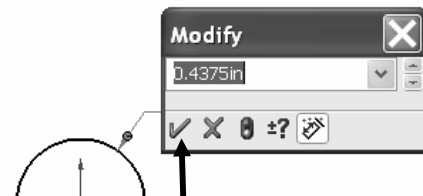
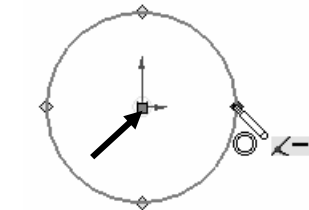
- 83)** Click **Features**  from the CommandManager.

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

- 85)** Select **Mid Plane** for End Condition in Direction 1.

- 86)** Enter **.250in**, **[6.35]** for Depth. Accept the default conditions.

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



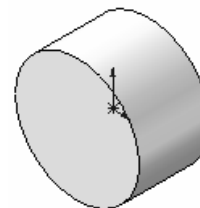
Fit the model to the Graphics window.

88) Press the **f** key.

89) Click **Isometric view** .

Save the model.


90) Click **Save** .

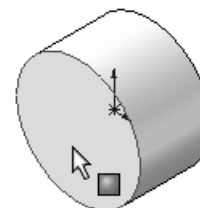


Activity: SHAFT-COLLAR Part-Extruded Cut Feature


Insert a new sketch for the Extruded Cut feature.


91) Click the **front circular face** of the Extrude1 feature for the Sketch plane. The mouse pointer displays the face feedback icon.

 View the mouse pointer for the correct geometry: line, face, point, or vertex.



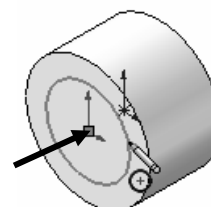
92) Click **Sketch**  from the Control Area.

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

94) Click the **Origin** . The cursor displays the Coincident to point feedback symbol.

95) Drag the **mouse pointer** to the right of the Origin.

96) Click a **position** to create the circle as illustrated.



Add a dimension.

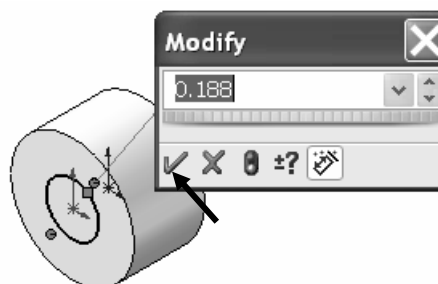
97) Click **Smart Dimension**  from the Sketch toolbar.

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


99) Click a **position** diagonally above the circle in the Graphics window.


100) Enter **.188in, [4.78]** in the Modify dialog box.

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

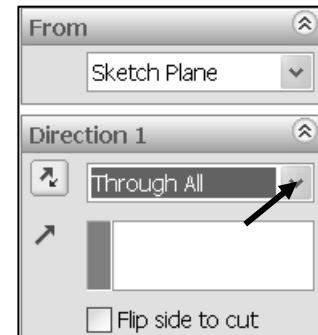
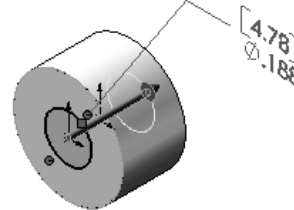



Insert an Extruded Cut feature.


102) Click **Features**  from the CommandManager.

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

104) Select **Through All** for End Condition in Direction 1. The direction arrow points to the right. Accept the default conditions.



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

 The Extruded Cut feature is named Cut-Extrude1. The Through All End Condition removes material from the Front plane through the Extrude1 geometry.

Activity: SHAFT-COLLAR-Modify Dimensions and Edit Color

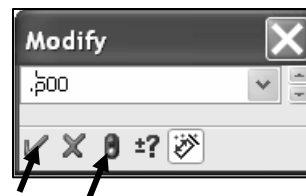
Modify the dimensions.

106) Click **Trimetric view**. Double-click the **outside face** of the SHAFT-COLLAR. The Extrude1 dimensions are displayed. Sketch dimensions are displayed in black. The Extrude depth dimensions are displayed in blue.

107) Double-click the .250in, [6.35] depth dimension.

108) Enter .500in, [12.70].

109) Click **Rebuild**  from the Modify pop-up box.



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

The Extrude1 and Cut-Extrude1 features are modified.

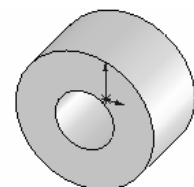
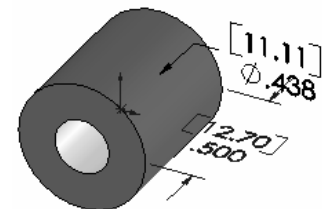
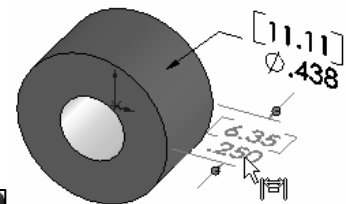
Return to the original dimensions.

111) Double-click the .500in, [6.35] depth dimension.

112) Enter .250in, [6.35] in the Modify dialog box.

113) Click **Rebuild**  from the Modify dialog box.

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



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

Modify the part color.

116) Click the **SHAFT-COLLAR Part**  icon at the top of the FeatureManager.

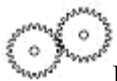
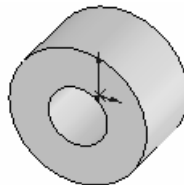
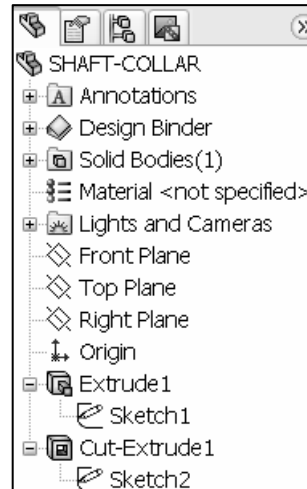
117) Click **Edit Color**  from the Standard toolbar.

118) Select a **light blue** color from the Color and Optics PropertyManager.

119) Click **OK**  from the Color And Optics PropertyManager.

Save the SHAFT-COLLAR part.

120) Click **Save** . The SHAFT-COLLAR part is complete.



Review the SHAFT-COLLAR Part.

The SHAFT-COLLAR utilized an Extruded Base feature. The Extruded Base feature adds material. An Extruded feature required a Sketch plane, Sketch, and Depth.

The Sketch plane was the Front plane. The 2D circle was sketched centered at the Origin. A dimension defined the overall size of the sketch.


The name of the feature was Extrude1. Extrude1 utilized the Mid Plane End Condition. The Extrude1 feature was symmetric about the Front plane.

The Extruded Cut feature removed material to create the hole. The Extruded Cut feature was named Cut-Extrude1. The Through All End Condition option created the Cut-Extrude1 feature. Feature dimensions were modified. The Edit Color option was utilized to modify the part color.



Additional details on Circle, Modify, Smart Dimensions, Sketch Color, Extruded Base and Extruded Cut are available in SolidWorks Help. Keywords: Circle, Modify, Sketch (color), Dimension, and Extrude.



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 Help for a rectangle.

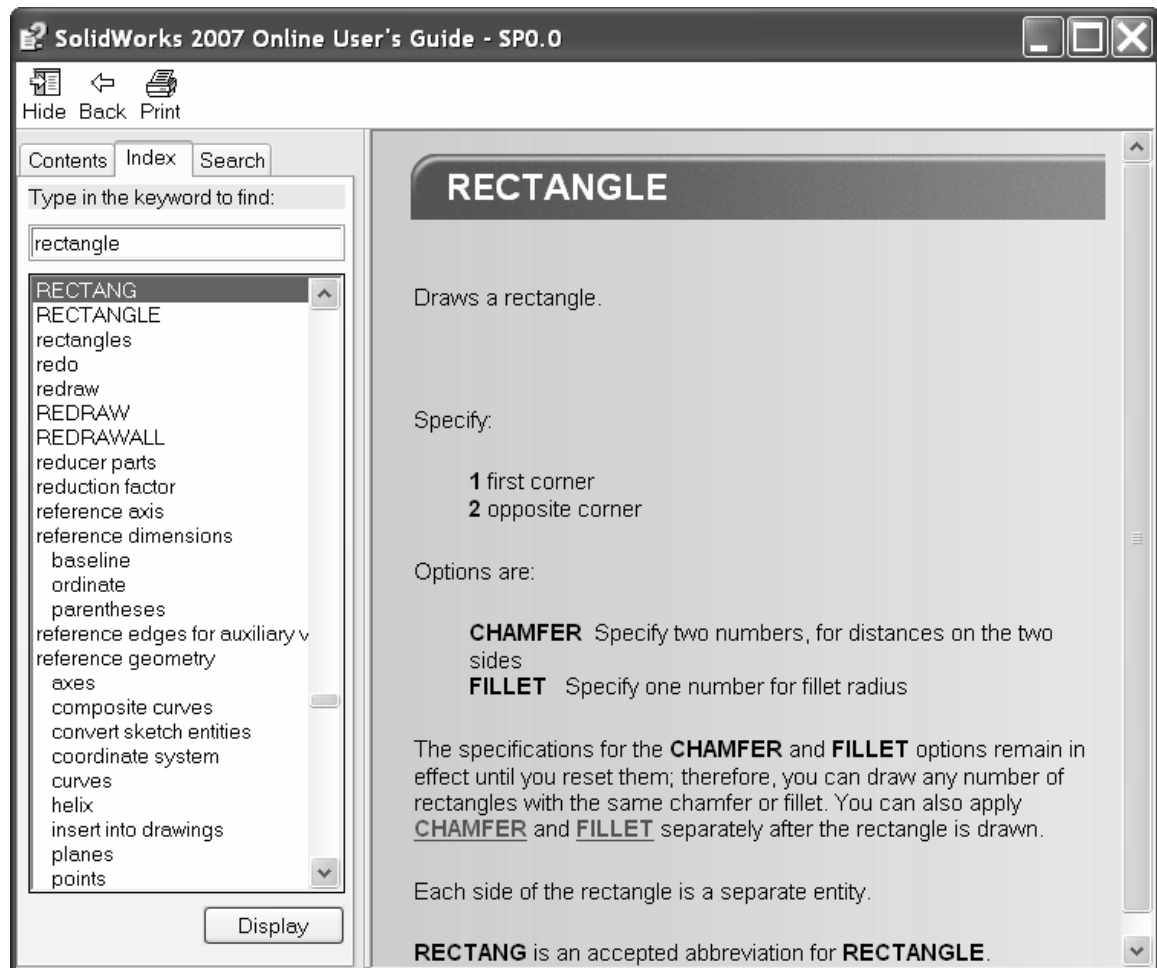
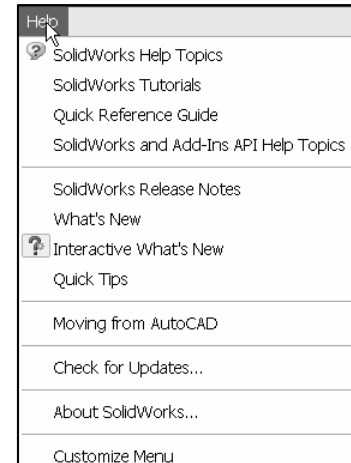
121) Click **Help** from the Main menu.

122) Click **SolidWorks Help Topics**  SolidWorks Help Topics.

123) Click the **Index** tab.

124) Enter **rectangle**. The description is displayed in the right window.

125) Click **Close**  to close the Help window.



The Help option contains tools to assist the user. The SolidWorks Help Topics contain:

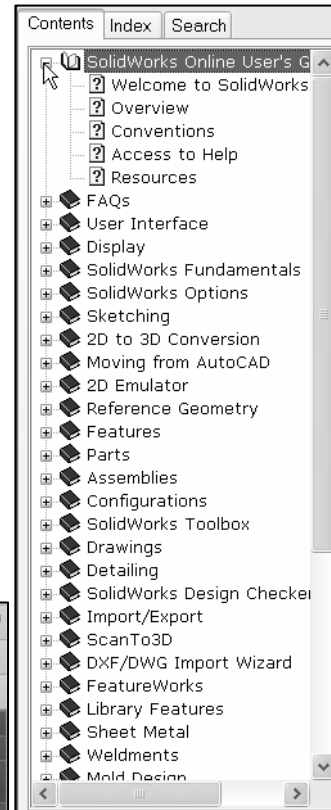
- **Contents** tab: Contains the SolidWorks Online User's Guide documents.
- **Index** tab: Contains additional information on key words.
- **Search** tab: To locate information.

Display the SolidWorks Tutorials.

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

127) Click **SolidWorks Tutorials**. The SolidWorks Tutorials are displayed. Review Lesson 1. This is a great location for additional information.

128) Click **Close**  from the Online Tutorial dialog box. Return to the Graphics window.



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

FLATBAR Part

The FLATBAR part fastens to the AXLE. The FLATBAR contains nine, $\varnothing.190$ in holes spaced 0.5in apart.

The FLATBAR part is manufactured from .060in stainless steel.

Create the FLATBAR part. Utilize an Extruded feature.

The Extruded feature requires a 2D profile sketched on the Front plane.

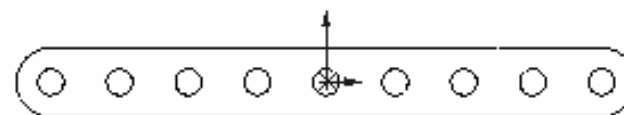
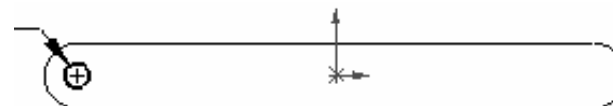
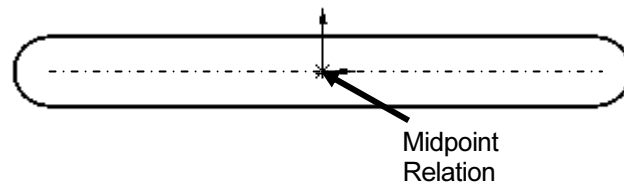
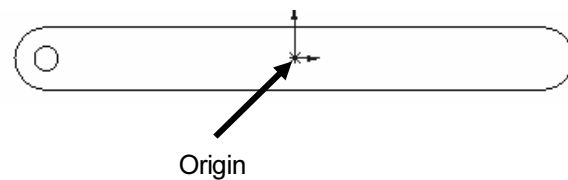
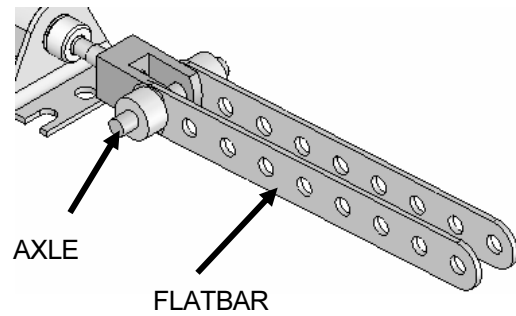
Utilize symmetry. Create the 2D profile centered about the Origin.

Relations control the size and position of entities with constraints.

Utilize the Add Relations sketch tool to define a Midpoint geometric relation in the sketch.

Utilize an Extruded Cut feature to create the first hole.

Utilize a Linear Pattern to create the remaining holes. A Linear Pattern creates an array of features in a specified direction.



Activity: FLATBAR Part-Extruded Base Feature

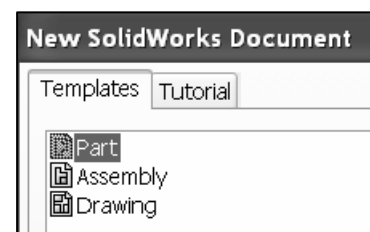
Create a new Part.

129) Click **File, New** from the Main menu. The Templates tab is the default tab. Part is the default template from the New SolidWorks Document dialog box.

130) Double-click **Part**. The Part FeatureManager is displayed.

Save the part.

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



132) Enter **FLATBAR** for File name in the SW-TUTORIAL-2007 folder

133) Enter **FLAT BAR 9 HOLES** for Description.

134) Click **Save**. The FLATBAR FeatureManager is displayed.

Set the Dimension standard and part units.

135) Click **Tools, Options, Document Properties** tab from the Main menu.



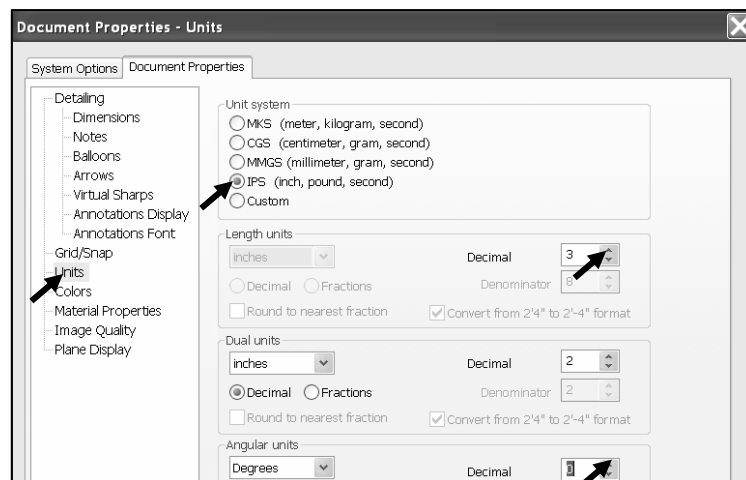
136) Select **ANSI** from the Dimensioning standard box.

137) Click **Units**.

138) Click **IPS**, **[MMGS]** for Unit system.

139) Select **3**, **[2]** for Length units Decimal places.

140) Select **0**, **[0]** for Angular units Decimal places.



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

Insert a new Sketch for the Extruded Base feature.

142) Click **Front Plane** from the FeatureManager. This is the Sketch plane.

143) Click **Sketch**  from the Control Area.

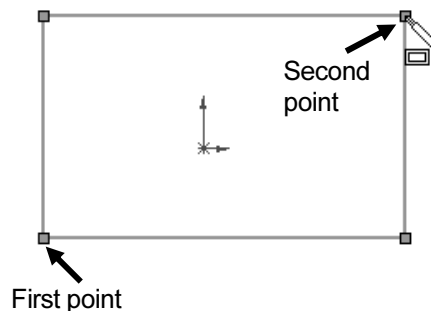
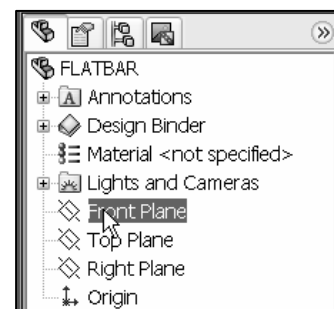
144) Click **Rectangle**  from the Sketch toolbar.

145) Click the **first point** of the rectangle below and to the left of the Origin in the Graphics window.

146) Drag the **mouse pointer** up and to the right of the Origin. Release the **mouse button** to create the second point of the rectangle as illustrated.

Trim the vertical lines.

147) Click **Trim Entities**  from the Sketch toolbar. The Trim PropertyManager is displayed.




148) Click **Trim to closest** from the Options box.

149) Click the **left vertical** line. The left vertical line is removed.

150) Click the **right vertical** line. The right vertical line is removed.

151) Click **OK**  from the Trim PropertyManager.

Sketch the right 180 degree Tangent Arc.

152) Click **Tangent Arc**  from the Sketch toolbar.

153) Click the **top right** endpoint of the top horizontal line.

154) Drag the **mouse pointer** to the right and downward.

155) Click the **bottom right endpoint** to complete the arc.

Sketch the left 180 degree Tangent Arc.

156) Click the **top left** endpoint of the top horizontal line.

157) Drag the **mouse pointer** to the left and downward.

158) Click the **bottom left endpoint** to complete the arc.

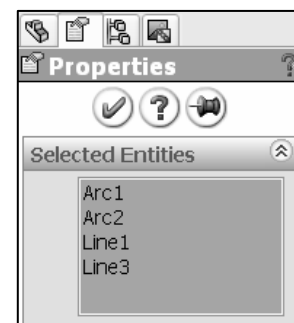
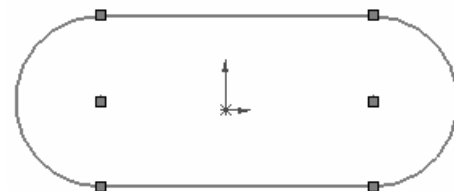
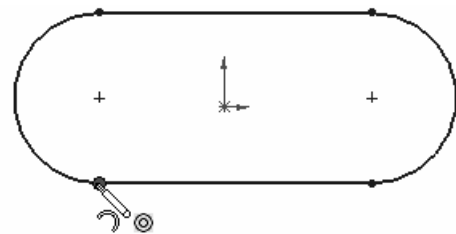
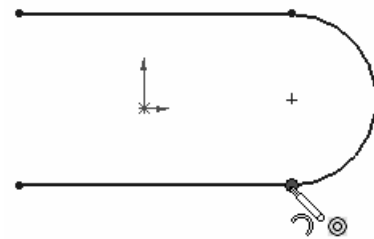
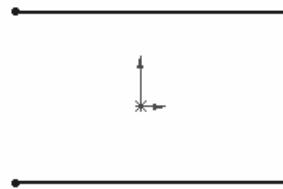
Window-select geometry in the Graphics window.

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

160) Click a **position** in the upper left corner of the Graphics window.


161) Drag the **mouse pointer** to the lower right corner of the Graphics window.

162) Release the **mouse pointer**. The selected geometry is displayed in the Selected Entities box. The selected geometry is displayed in green in the Graphics window.



Maintain the slot sketch symmetric about the Origin. Utilize relations. A relation is a geometric constraint between sketch geometry. Position the Origin at the Midpoint of the centerline.

Sketch a centerline.

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

164) Sketch a **horizontal centerline** from the left arc center point to the right arc center point.

165) Right-click **Select**  **Select**.

Add a Midpoint relation.

166) Click the **Origin** . Hold the **Ctrl** key down.

167) Click the **centerline**. The Properties PropertyManager is displayed. Release the **Ctrl** key. The Origin and the centerline is displayed in the Selected Entities box.


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

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

Add an Equal relation.

170) Click the **top horizontal line**. Hold the **Ctrl** key down.

171) Click the **bottom horizontal line**. The Properties PropertyManager is displayed. Release the **Ctrl** key.

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

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

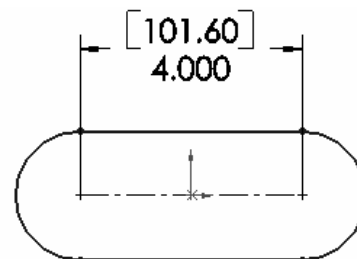
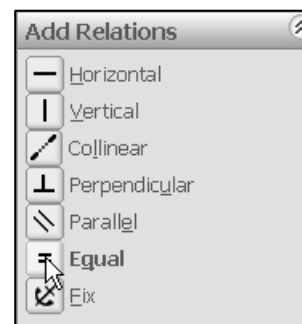
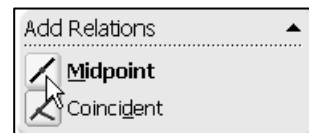
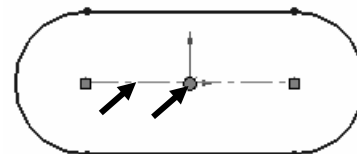
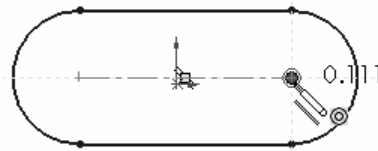
Add a dimension.

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

175) Click the **horizontal centerline**. Click a **position** above the top horizontal line in the Graphics window.

176) Enter **4.000in**, **[101.60]** in the Modify dialog box.

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




178) Click the **right arc** of the FLATBAR.

179) Click a **position** diagonally to the right in the Graphics window.


180) Enter **.250in**, **[6.35]** in the Modify dialog box.

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

Extrude the sketch to create the Base feature.

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

183) Enter **.060in**, **[1.5]** for Depth. Accept the default conditions.

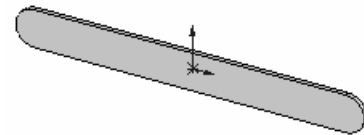
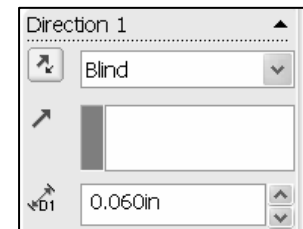
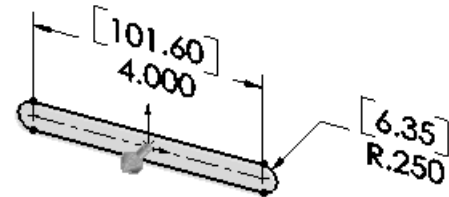
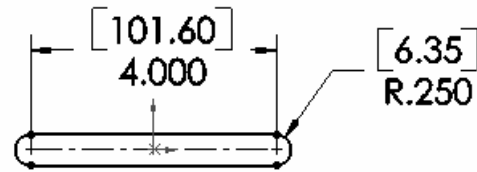
184) Click **OK**  from the Extrude PropertyManager. Extrude1 is displayed in the FeatureManager.

Fit the model to the Graphics window.

185) Press the **f** key.

Save the FLATBAR part.

186) Click **Save** .



Activity: FLATBAR Part-Extruded Cut Feature

Insert a new sketch for the Extruded Cut Feature.

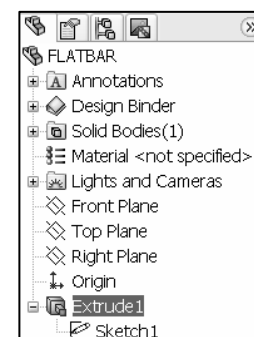
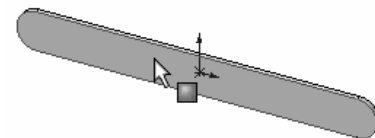
187) Click the **front face** of the Extrude1 feature in the Graphics window. This is the Sketch plane. Extrude1 is highlighted in the FeatureManager.

188) Click **Sketch**  from the Control Area.


Display the Front view.

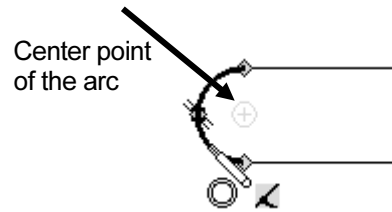
189) Click **Front view** .

The process of placing the mouse pointer over an existing arc to locate its center point is called “wake up”.



Wake up the center point.

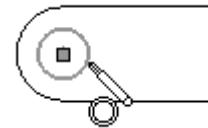
- 190)** Click **Circle**  from the Sketch toolbar. The Circle PropertyManager is displayed.



- 191)** Place the **mouse pointer** on the left arc. Do not click. The center point of the slot arc is displayed.

- 192)** Click the **center point** of the arc.

- 193)** Click a **position** to the right of the center point to create the circle as illustrated.

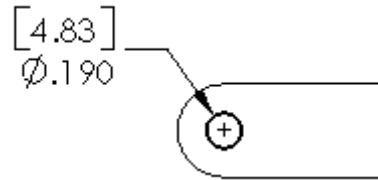


Add a dimension.

- 194)** Click **Smart Dimension**  from the Sketch toolbar.

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

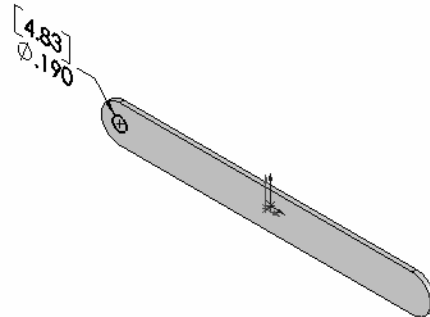
- 196)** Click a **position** diagonally above and to the left of the circle in the Graphics window.



- 197)** Enter .190in, [4.83] in the Modify box.

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


- 199)** Click **Isometric view** .

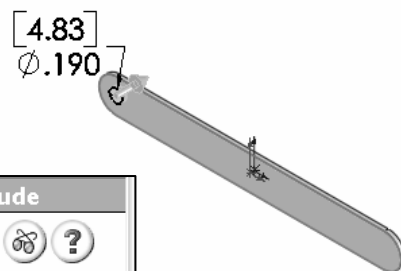
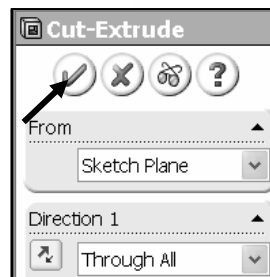


Insert an Extruded Cut feature.

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


- 201)** Select **Through All** for End Condition in Direction 1. The direction arrow points to the right. Accept the default conditions.

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

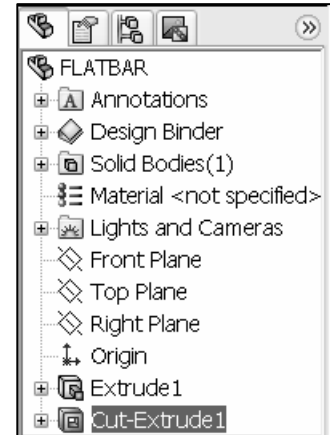


Save the FLATBAR part.

- 203)** Click **Save** .

The blue Cut-Extrude1  Cut-Extrude1 icon indicates that the feature is selected.

Select features by clicking their icons in the FeatureManager or by selecting their geometry in the Graphics window.



Activity: FLATBAR Part-Linear Pattern Feature


Create a Linear Pattern.


204) Click **Linear Pattern**  from the Features toolbar. The Linear Pattern PropertyManager is displayed.

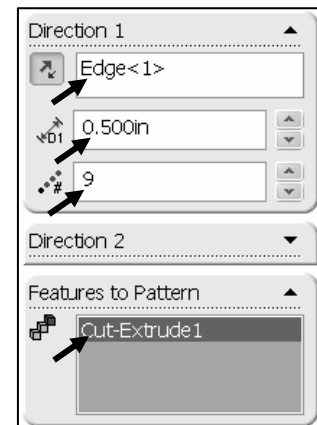
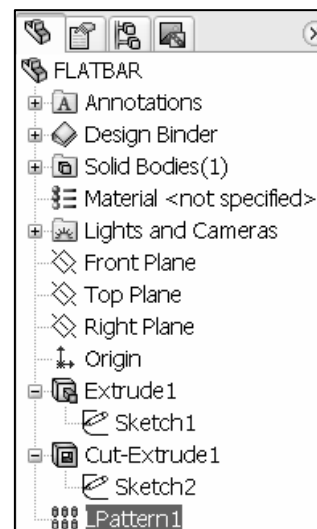
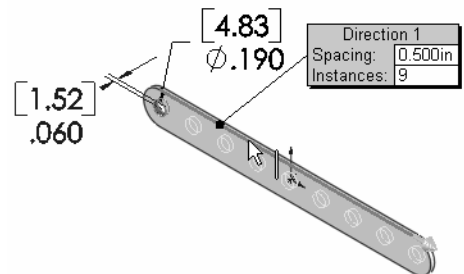
205) Click the **top edge** of the Extrude1 feature for Direction1. Edge<1> is displayed in the Pattern Direction box.

206) Enter **0.5in**, [12.70] for Spacing.


207) Enter **9** for Number of Instances. Instances are the number of occurrences of a feature.

208) The Direction arrow points to the right. Click the **Reverse Direction**  button if required.

209) Cut-Extrude1 is displayed in the Features to Pattern box. Click **OK**  from the Linear Pattern PropertyManager. The LPattern1 feature is displayed in the FeatureManager.



Save the FLATBAR part.

210) Click **Save** . The FLATBAR part is complete.

Close all documents.

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



Review the FLATBAR Part.

The FLATBAR utilized an Extruded Base feature. The Sketch plane was the Front plane. The 2D sketch utilized the Rectangle and Tangent Arc Sketch tools to create the slot profile. You created a centerline between the two arc center points.

The Midpoint relation maintained the slot profile symmetric about the Origin. Linear and radial dimensions were added to define the overall size of the sketch. The name of the feature was Extrude1. Extrude1 utilized the Blind End Condition.

The Extruded Cut feature removed material to create the hole. The Extruded Cut feature was named Cut-Extrude1. The Through All End Condition option created the Cut-Extrude1 feature from the Front plane. The Linear Pattern created an array of 9 holes, equally spaced along the length of the FLATBAR Part.



Additional details on Rectangle, Trim Entities, Extruded Base, Extruded Boss, Extruded Cut and Linear Pattern are available in SolidWorks Help. Keywords: Rectangle, Trim, Extruded Features, and Linear Pattern.

LINKAGE Assembly

An assembly is a document that contains two or more parts. An assembly inserted into another assembly is called a sub-assembly. A part or sub-assembly inserted into an assembly is called a component. The LINKAGE assembly consists of the following components: AXLE part, SHAFT-COLLAR part, FLATBAR part, and AIRCYLINDER sub-assembly.

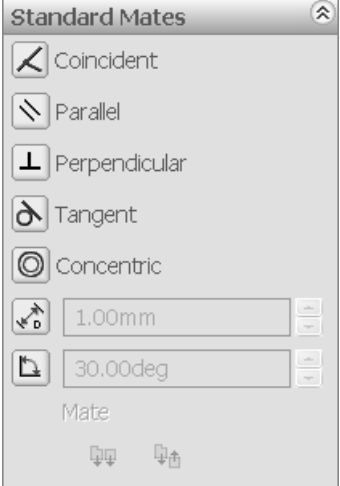
Establishing the correct component relationship in an assembly requires forethought on component interaction. Mates are geometric relationships that align and fit components in an assembly. Mates remove degrees of freedom from a component.

Mate Types

Mates reflect the physical behavior of a component in an assembly. The components in the LINKAGE assembly utilize Standard Mate types. Review the Standard and Advanced Mates types.

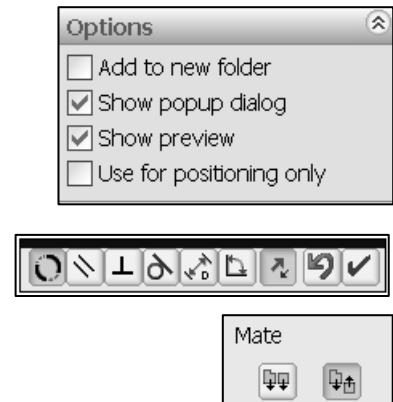
Standard and Advanced Mates:

The Mate Property Manager displays Standard Mate types and Advanced Mate types. Components are assembled with various Mate types. The Standard Mate types are:

Standard Mate Types:	
	Coincident: Positions selected faces, edges, and planes so they share the same infinite line. Positions two vertices so they touch.
	Parallel: Positions the selected geometry so they lie in the same direction and remain a constant distance apart.
	Perpendicular: Positions the selected geometry at 90°.
	Tangent: Positions the selected items in a tangent mate (at least one selection must be a cylindrical, conical, or spherical face).
	Concentric: Positions the selected geometry so that they share the same center point.
	Distance: Positions the selected geometry at a specified distance between them.
	Angle: Positions the selected geometry at a specified angle between them.

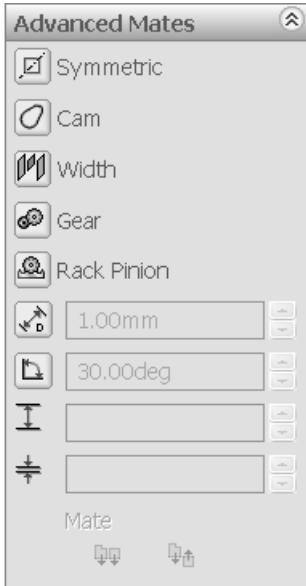
The Mate dialog box displays the Pop-up toolbar during the Mate procedure. The Standard Mate types, Aligned/Anti-Aligned, Undo, and OK are displayed in the Pop-up toolbar.

There are two Mate Alignment options. The Aligned option positions the components so that the normal vectors from the selected faces point in the same direction. The Anti-Aligned option positions the components so that the normal vectors from the selected faces point in opposite directions.



Advanced Mates:

The Advanced Mate types are:

Advanced Mate Types:	
	<p>Symmetric: Positions two selected entities to be symmetric about a plane or planar face. A Symmetric Mate does not create a Mirrored Component.</p>
	<p>Cam: A cam-follower mate is a type of tangent or coincident mate. It positions a cylinder, plane, or point to a series of tangent extruded Cam faces. The Cam profile is comprised of tangent lines, arcs, and/or splines in a closed loop.</p>
	<p>Width: Centers a tab within the width of a groove.</p>
	<p>Gear: Positions two components to rotate relative to one another about selected axes. The axis of rotation includes cylindrical and conical faces, axes, and linear edges. Gear components are not required for a Gear Mate; example, two rolling cylinders.</p>
	<p>Rack Pinion: Linear translation of one part (the rack) causes circular rotation in another part (the pinion), and vice versa.</p>
	<p>Limit: Defines a range of motion for a Distance Mate or Angle Mate. Specify a starting value, minimum value and maximum value.</p>

SolidWorks Help Topics list the rules governing Mate Type valid geometry. The valid geometry selection between components in a Coincident Mate is displayed in the Coincident Mate Combinations Table.

SolidWorks Help Topics also display Standard Mates by Entity. Specific combinations of geometry create valid Mates.

Coincident Mate		
You can add a coincident mate between the combinations shown below:		
Cone - Circular/Arc Edge - Cone - Point	Plane - Line - Plane - Point - Circular/Arc Edge	Circular/Arc Edge - Cone - Plane - Cylinder - Circular/Arc Edge
Cylinder - Line - Point - Circular/Arc Edge	Point - Cam - Cone - Cylinder - Extrusion - Line - Plane - Point - Sphere - Surface	Surface - Point
Extrusion - Point		
Line - Cylinder - Line - Plane - Point	Sphere - Point	

Example: Utilize a Concentric Mate between the AXLE cylindrical face and the FLATBAR Extruded Cut (hole). Utilize a Coincident Mate between the SHAFT-COLLAR back face and the FLATBAR front flat face.

The LINKAGE assembly requires the AIRCYLINDER assembly. The AIRCYLINDER assembly is located on the SolidWorks Tutorial Multimedia CD in the pneumatic components folder.

Activity: AIRCYLINDER Assembly-Open and Save As option

Open the AIRCYLINDER assembly.

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

213) Double-click **Assembly** from the New SolidWorks Document dialog box. The Insert Component PropertyManager is displayed.

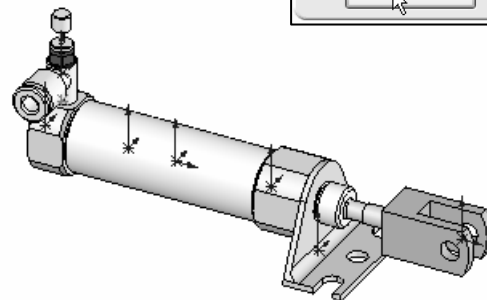
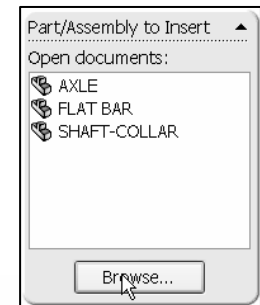
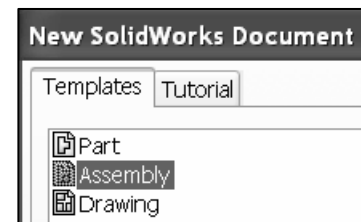
214) Click **View**; check **Origins** from the Main menu.

215) Click **Browse** from the Insert Component PropertyManager.

216) Select the **pneumatic components** folder from the CD in the book.

217) Double-click the **AIRCYLINDER** assembly. The AIRCYLINDER assembly is displayed in the Graphics window.

218) Click the **Origin** to fix the AIRCYLINDER assembly in the Graphics window. The (f) symbol is placed in front of the AIRCYLINDER name in the FeatureManager.



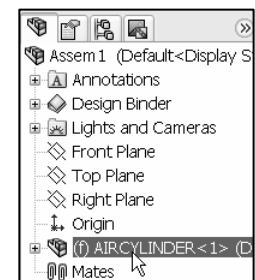
The AIRCYLINDER assembly is displayed in the Graphics window. Copy the AIRCYLINDER assembly with the Save As command. Save all sub-assemblies and part references to the SW-TUTORIAL-2007 folder.

Save the AIRCYLINDER assembly to the SW-TUTORIAL-2007 folder.

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

220) Select **SW-TUTORIAL-2007** for Save in folder.

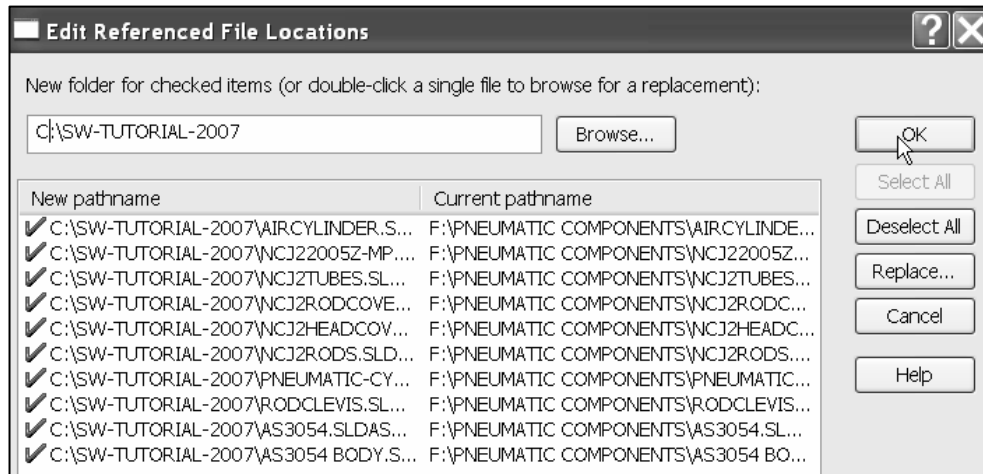
221) Click the **References** button.



222) Click the **Select All** button to check all components contained in the AIRCYLINDER assembly.

223) Click the **Browse** button from the Edit Referenced File Locations.

224) Select the **SW-TUTORIAL-2007** folder. Click **OK** from the Browse for Folder dialog box.



225) Click **OK** from the Edit Referenced File Locations dialog box.

226) Enter **LINKAGE** for File name.

227) Click **Save**.

228) Click **YES**.

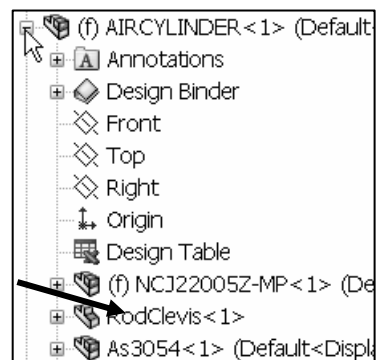
The AIRCYLINDER assembly and its references are copied to the SW-TUTORIAL-2007 folder. Assemble the AXLE to the holes in the RodCleviis.



Display the RodCleviis Component.

229) **Expand** the AIRCYLINDER assembly in the FeatureManager.

230) Click **RodCleviis<1>** from the FeatureManager. Note: The RodCleviis is displayed in green in the Graphics window.



Hide the Origins.

231) Click **View**; uncheck **Origins** from the Main menu.

The AIRCYLINDER is the first component in the LINKAGE assembly and is fixed (f) to the LINKAGE assembly Origin.

Display the Isometric view.

232) Click **Isometric view** .

Insert the AXLE part.

233) Click **Insert Component**  from the Assembly toolbar. The Insert Component PropertyManager is displayed.

234) Click **Browse**. Select **All Files** from the Files of type box.

235) Double-click **AXLE** from the SW-TUTORIAL-2007 folder.

236) Click a **position** to the left of the AIRCYLINDER assembly as illustrated.


Move the AXLE component.

237) Click and drag a **position** in front of the RODCLEVIS.

Enlarge the view.

238) Click **Zoom to Area**  on the RodClevis and the AXLE.

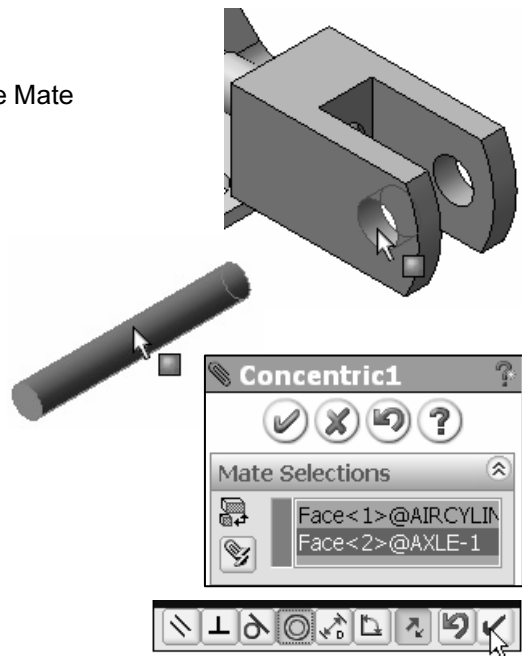
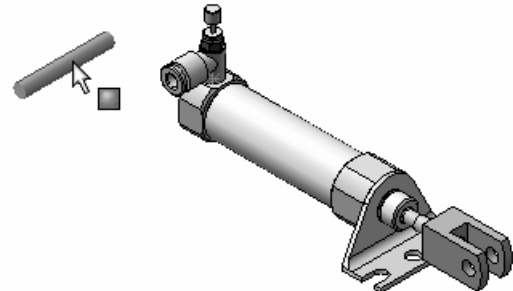
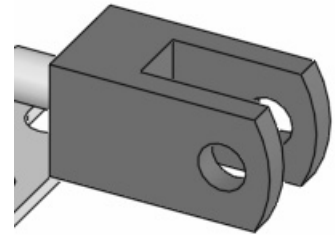
Insert a Concentric Mate.

239) Click **Mate**  from the Assembly toolbar. The Mate PropertyManager is displayed.

240) Click the inside **left hole face** of the RodClevis.

241) Click the **long cylindrical face** of the AXLE. The cursor displays the Face feedback symbol. The selected faces are displayed in the Mate Selections box. Concentric Mate is selected by default. The AXLE is positioned concentric to the RodClevis hole.

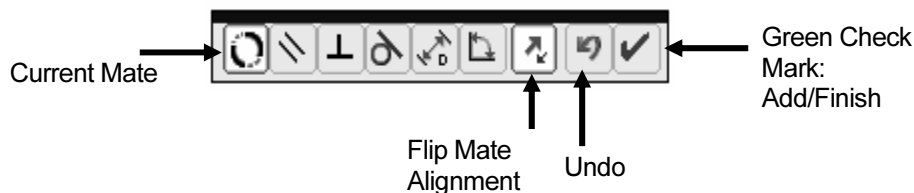
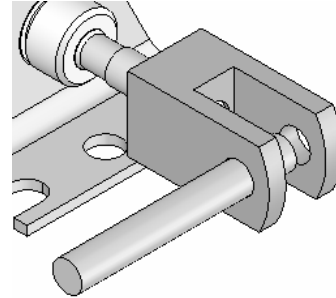
242) Click the **Green Check mark** .



Move the AXLE.

243) Click and drag the **AXLE** left to right. The AXLE translates in and out of the RodClevish holes.

The Mate Pop-up toolbar is displayed after selecting the two cylindrical faces. The Mate Pop-up toolbar minimizes the time required to create a Standard Mate.



Position the mouse pointer in the middle of the face to select the entire face. Do not position the mouse pointer near the edge of the face. If the wrong face or edge is selected, perform one of the following actions:

- Click the face or edge again to remove it from the Items Selected text box.
- Right-click in the Graphics window. Click Clear Selections to remove all geometry from the Items Selected text box.
- Utilize the UNDO button to begin the Mate command again.

Display the Top view.

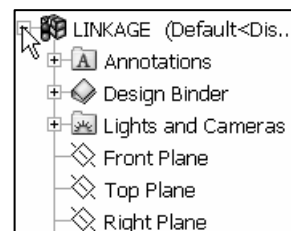
244) Click **Top view**

Expand the LINKAGE assembly and components in the fly-out FeatureManager.

245) **Expand** the LINKAGE assembly from the fly-out FeatureManager.

246) **Expand** the AIRCYLINDER assembly from the fly-out FeatureManager.

247) **Expand** the AXLE part from the fly-out FeatureManager.



Clear all sections from the Mate Selections box.

248) Right-click **Clear Selections** inside the Mate Selections box.



Insert a Coincident Mate.

249) Click the **Front Plane** of the AIRCYLINDER assembly from the fly-out FeatureManager.

250) Click the **Front Plane** of the AXLE part from the fly-out FeatureManager. The selected planes are displayed in the Mate Selections box. Coincident Mate is selected by default.

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

252) Click **OK** from the Mate PropertyManager.

The AIRCYLINDER Front plane and the AXLE Front plane are Coincident. The AXLE is centered in the RodClevis.



Display the Mates in the FeatureManager to check that the components and the Mate Types correspond to the design intent. Note: If you delete a Mate and then recreate it, the Mate numbers will be in a different order.

Display an Isometric view.

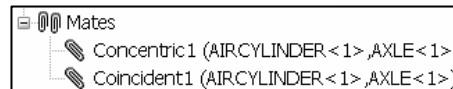
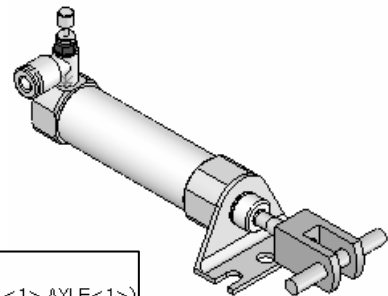
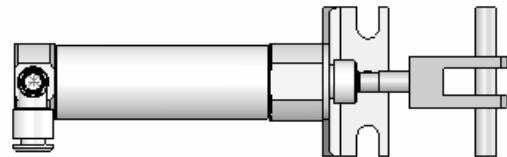
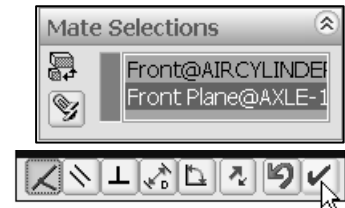
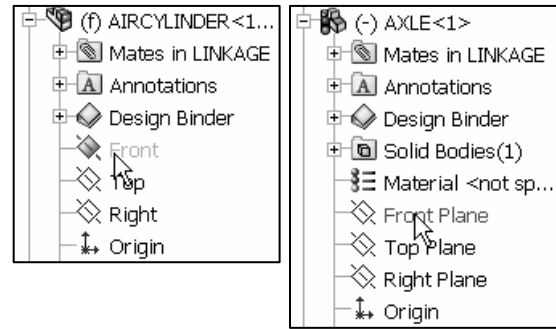
253) Click **Isometric view** .

Display the Mates.

254) **Expand** the Mates folder in the FeatureManager.

Save the LINKAGE assembly.

255) Click **Save** .



Activity: LINKAGE Assembly-Insert FLATBAR Part

Insert the FLATBAR part.



256) Click **Insert Component** from the Assembly toolbar. The Insert Component PropertyManager is displayed.

257) Click **Browse**.


258) Select **Part** for Files of type from the SW-TUTORIAL-2007 folder.

259) Double-click **FLATBAR**.


Place the Component.

260) Click a **position** in the Graphics window as illustrated.

Enlarge the view.

261) Click **Zoom to Area**  on the AXLE and the left side of the FLATBAR to enlarge the view.

Insert a Concentric Mate.

262) Click **Mate**  from the Assembly toolbar. The Mate PropertyManager is displayed. If required, right-click **Clear Selections** inside the Mate Selections box.

263) Click the inside **left hole face** of the FLATBAR.

264) Click the **long cylindrical face** of the AXLE. The selected faces are displayed in the Mate Selections box. Concentric is selected by default.

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

Fit the model to the Graphics window.

266) Press the **f** key.

Move the FLATBAR.

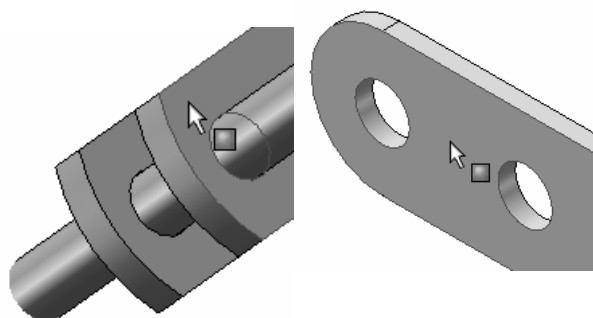
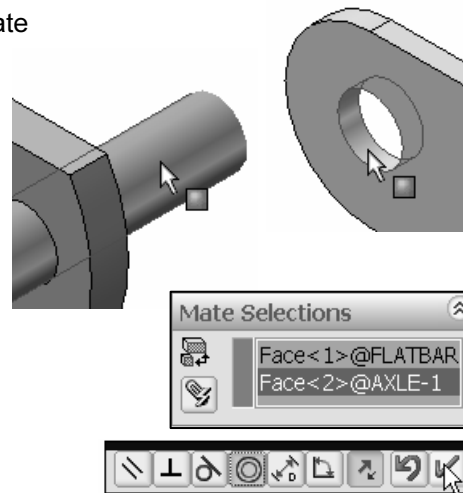
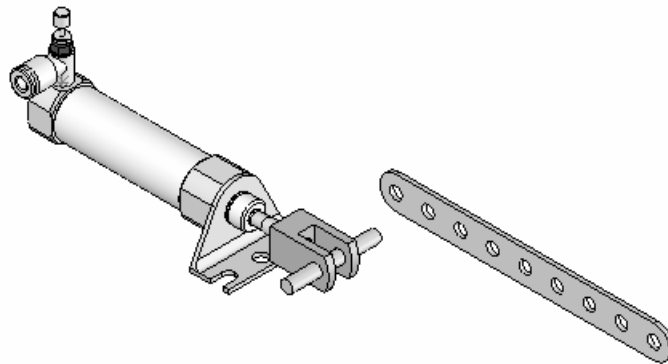
267) Click and drag the **FLATBAR**. The FLATBAR translates and rotates along the AXLE.


Insert a Coincident Mate.

268) Click the **front face** of the FLATBAR.


269) Press the **left arrow key** approximately 5 times to rotate the model and to view the back face of the RodClevis. Click the **back face** of the RodClevis as illustrated. The selected faces are displayed in the Mate Selections box. Coincident is selected by default.

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




271) Click **OK**  from the Mate PropertyManager.

Display the Isometric view.

272) Click **Isometric view** .

Insert the second FLATBAR Component.

273) Click **Insert Component**  from the Assembly toolbar. The Insert Component PropertyManager is displayed.


274) Click **Browse**.

275) Select **Part** for Files of type from the SW-TUTORIAL-2007 folder.


276) Double-click **FLATBAR**.

277) Click a **position** to the left of the AIRCYLINDER in the Graphics window as illustrated.

Enlarge the view.

278) Click **Zoom to Area**  on the second FLATBAR and the AXLE.

Insert a Concentric Mate.

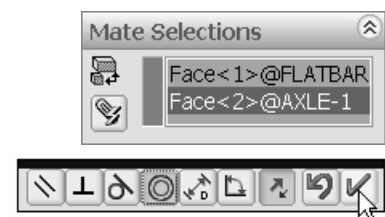
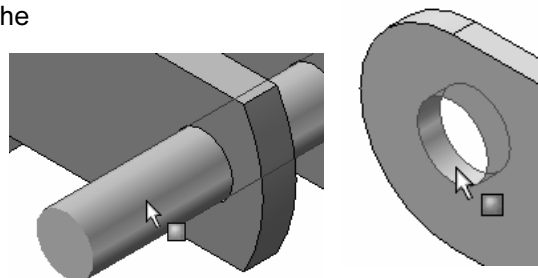
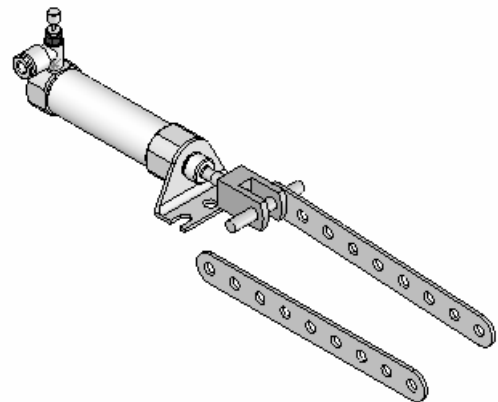
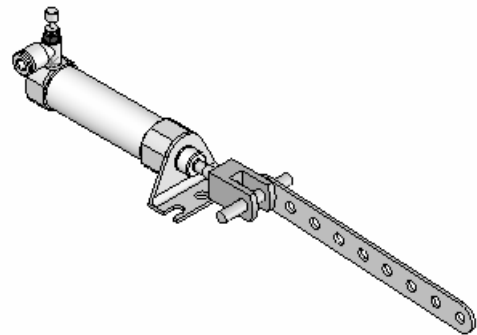
279) Click **Mate**  from the Assembly toolbar. The Mate PropertyManager is displayed.

280) Click the **left inside hole face** of the second FLATBAR.

281) Click the **long cylindrical face** of the AXLE. The selected faces are displayed in the Mate Selections box. Concentric is selected by default.

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

283) Press the **f** key to fit the model to the Graphics window.



Insert a Coincident Mate.

284) Press the **left arrow key** approximately 5 times to rotate the model to view the back face of the second FLATBAR.

285) Click the **back face** of the second FLATBAR.

286) Press the **right arrow key** approximately 5 times to rotate the model to view the front face of the RodClevis.

287) Click the **front face** of the RodClevis. The selected faces are displayed in the Mate Selections box. Coincident is selected by default.

288) Click the **Green Check mark** ✓.

Create a Parallel Mate.

289) Press the **Shift-z** keys to Zoom in on the model.

290) Click the **top narrow face** of the first FLATBAR.

291) Click the **top narrow face** of the second FLATBAR. The selected faces are displayed in the Mate Selections box.

292) Click **Parallel** ∥.

293) Click the **Green Check mark** ✓.

294) Click **OK** (✓) from the Mate PropertyManager.

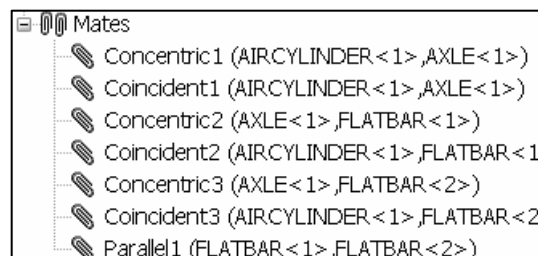
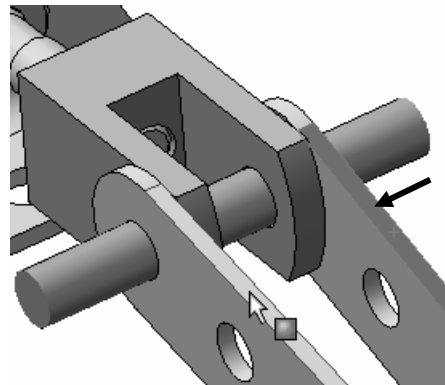
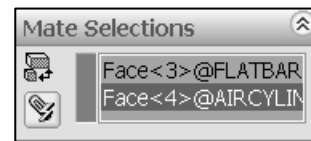
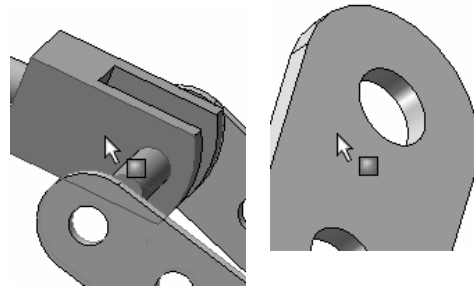
295) Click **Isometric view** .

Move the two FLATBAR parts.

296) Click and drag the **second FLATBAR**. Both FLATBAR parts move together.

View the Mates.

297) **Expand** the Mates folder from the FeatureManager.



Activity: LINKAGE Assembly-Insert SHAFT-COLLAR Part

Insert the first SHAFT-COLLAR.



298) Click **Insert Component** from the Assembly toolbar. The Insert Component PropertyManager is displayed.

299) Click **Browse**.

300) Select **Part** for Files of type from the SW-TUTORIAL-2007 folder.

301) Double-click **SHAFT-COLLAR**.

302) Click a **position** to the right of the AXLE as illustrated.

Enlarge the view.

303) Click **Zoom to Area** . **Zoom-in** on the SHAFT-COLLAR and the AXLE to enlarge the view.

Insert a Concentric Mate.

304) Click **Mate** from the Assembly toolbar. The Mate PropertyManager is displayed.

305) Click the inside **hole face** of the SHAFT-COLLAR.

306) Click the **long cylindrical face** of the AXLE. The selected faces are displayed in the Mate Selections box. Concentric is selected by default.

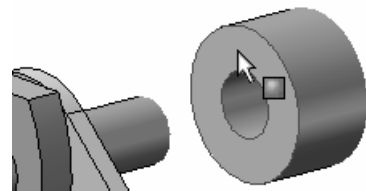
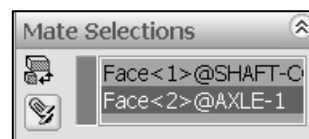
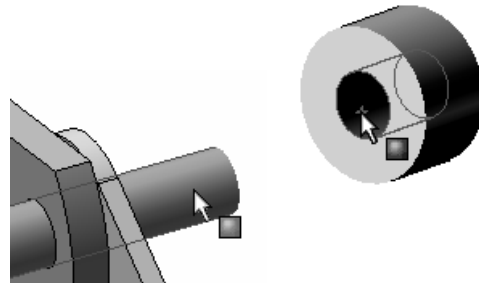
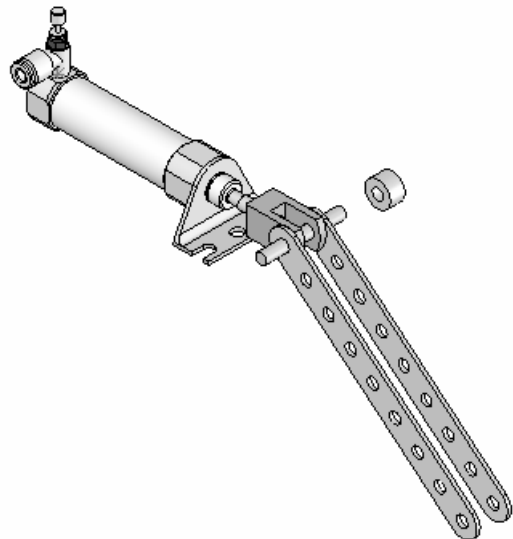
307) Click the **Green Check mark** .

Insert a Coincident Mate.

308) Press the **Shift-z** keys to Zoom in on the model.


309) Click the **front face** of the SHAFT-COLLAR as illustrated.

310) Press the **left arrow key** approximately 5 times to rotate the model to view the back face of the first FLATBAR.



311) Click the **back face** of the first FLATBAR. The selected faces are displayed in the Mate Selections box. Coincident is selected by default.

312) Click the **Green Check mark** ✓.

313) Click **OK**  from the Mate PropertyManager.

Display the Isometric view.

314) Click **Isometric view** .

Insert the second SHAFT-COLLAR.



315) Click **Insert Component**  from the Assembly toolbar. The Insert Component PropertyManager is displayed.


316) Click **Browse**.

317) Select **Part** for Files of type from the SW-TUTORIAL-2007 folder.

318) Double-click **SHAFT-COLLAR**.


319) Click a **position** near the AXLE as illustrated.

Enlarge the view.

320) Click **Zoom to Area** . Zoom-in on the second SHAFT-COLLAR and the AXLE to enlarge the view.

Insert a Concentric Mate.

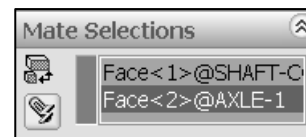
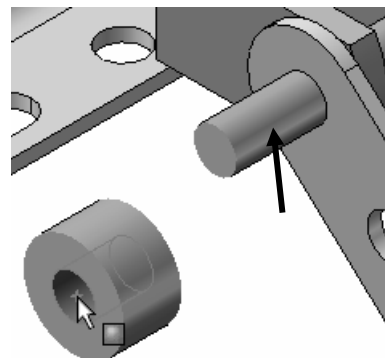
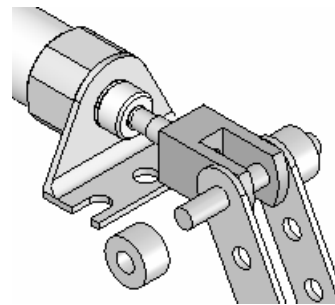
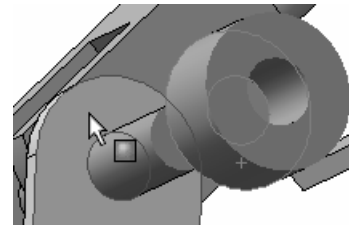


321) Click **Mate**  from the Assembly toolbar. The Mate PropertyManager is displayed.

322) Click the **inside hole face** of the second SHAFT-COLLAR.

323) Click the **long cylindrical face** of the AXLE. Concentric is selected by default. The selected faces are displayed in the Mate Selections box.

324) Click the **Green Check mark** ✓.

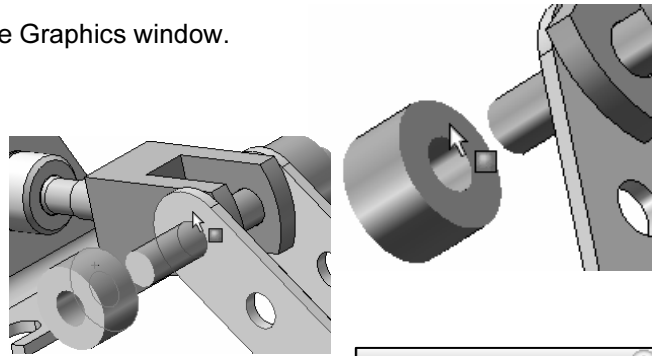


Insert a Coincident Mate.


325) Press the **f** key to fit the model to the Graphics window.

326) Click the **back face** of the second SHAFT-COLLAR.

327) Click the **front face** of the second FLATBAR. The selected faces are displayed in the Mate Selections box. Coincident is selected by default.



328) Click the **Green Check mark** ✓.

329) Click **OK**  from the Mate PropertyManager.


Display the Isometric view.

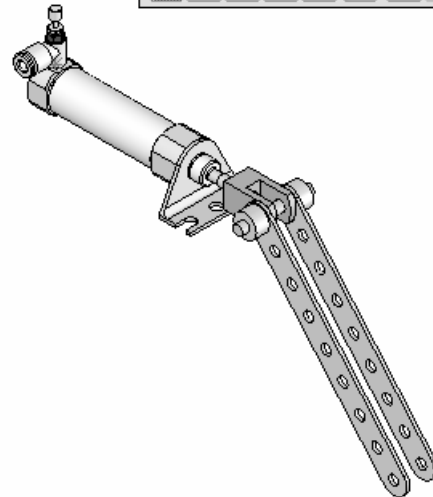
330) Click **Isometric view** .

Fit the model to the Graphics window.

331) Press the **f** key.

Save the LINKAGE assembly.

332) Click **Save** . The LINKAGE assembly is complete.



Review the LINKAGE Assembly.

An assembly is a document that contains two or more parts. A part or sub-assembly inserted into an assembly is called a component. You created the LINKAGE assembly.

The AIRCYLINDER sub-assembly was the first component inserted into the LINKAGE assembly. The AIRCYLINDER assembly was obtained from the CD in the book and copied to the SW-TUTORIAL-2007 folder.

The AIRCYLINDER assembly was fixed to the Origin. The Concentric and Coincident Mates added geometric relationships between components in the LINKAGE assembly.

The AXLE part was the second component inserted into the LINKAGE assembly. The AXLE required a Concentric Mate between two cylindrical faces and a Coincident Mate between two Front planes.

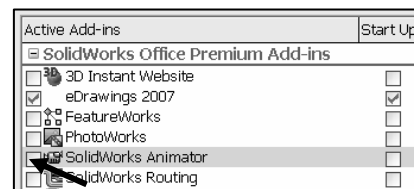
The FLATBAR part was the third component inserted into the LINKAGE assembly. The FLATBAR required a Concentric Mate between two cylindrical faces and a Coincident Mate between two flat faces. A second FLATBAR was inserted into the LINKAGE assembly. A Parallel Mate was added between the two FLATBARs.

Two SHAFT-COLLAR parts were inserted into the LINKAGE assembly. Each SHAFT-COLLAR required a Concentric Mate between the two cylindrical faces and a Coincident Mate between two flat faces.

Physical Simulation Tools

The Physical Simulation tools represent the effects of motors, springs, and gravity on an assembly. The Physical Simulation tools are combined with Mates and Physical Dynamics to translate and rotate components in an assembly.

The Simulation Toolbar contains four simulation tools: Linear Motor, Rotary Motor, Spring, and Gravity. Activate SolidWorks Animator. Select Tools, Add-Ins, SolidWorks Animator from the Main menu.



Activity: LINKAGE Assembly-Physical Simulation

Insert a Rotary Motor Physical Simulation Tool.

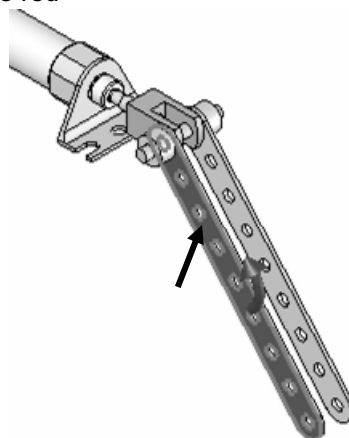
333) Click **View, Toolbars, Simulation** from the Main menu. The Simulation toolbar is displayed



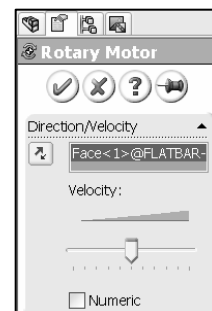
334) Click **Rotary Motor** from the Simulation toolbar. The Rotary Motor PropertyManager is displayed.

Rotary Motor
Moves a component as if acted upon by a rotary motor.

335) Click the **FLATBAR front face** as illustrated. A red Rotary Motor icon is displayed. The red direction arrow points counterclockwise.





336) Position the **Velocity Slide bar** in the middle of the Rotary Motor PropertyManager.




337) Click **OK** from the Rotary Motor PropertyManager. Note: Recording the simulation to an avi file requires the SolidWorks Animator application.

Record the Simulation.

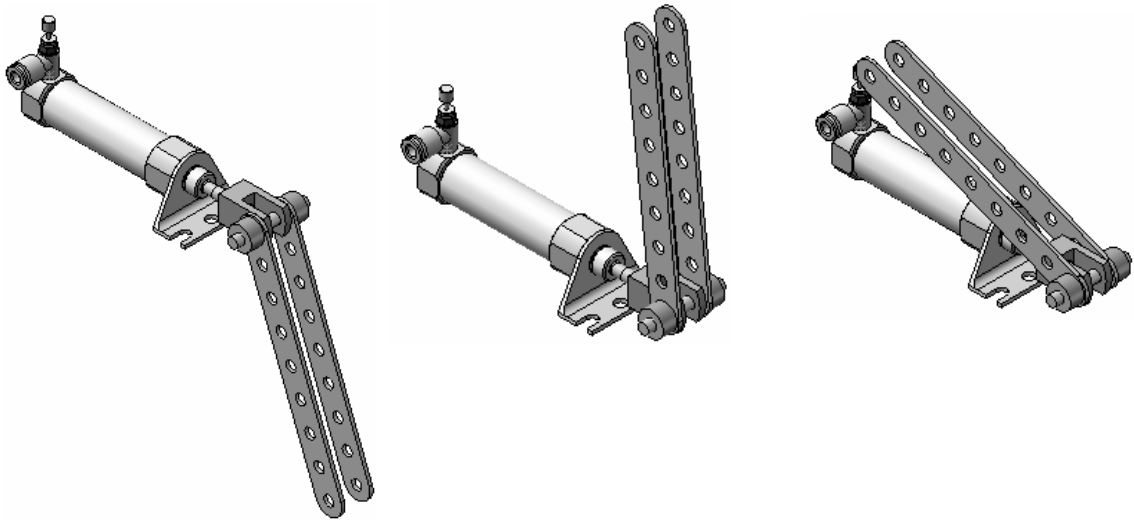
338) Click **Calculate Simulation** . The FLATBAR rotates in a counterclockwise direction.

339) Click **Replay Simulation**  from the Simulation toolbar.

340) Click **Play**  from the Animation Controller.




Calculate Simulation
Calculates the effects of simulation elements.



Linear Assembly Physical Simulation



Stop the Simulation.

341) Click **Stop**  from the Animation Controller.

342) Save the simulation in an AVI file to the SW-

TUTORIAL-2007 folder. Click **Save** .

Note: SW Animator is required to save the simulation in an .AVI file. Accept the default file name.

343) Click **Save**.

344) Click **OK** from the Video Compression box. Note: This should take approximately 2 minutes.

Close the Simulation.

345) Click **Close** from the Animation Controller.

Fit the assembly to the Graphics window.

346) Press the **f** key.

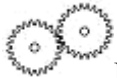
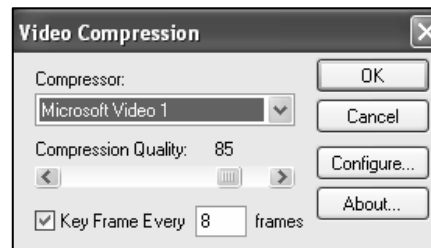
Save the LINKAGE assembly.

347) Click **Save** .

Exit SolidWorks.

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

The LINKAGE assembly project is complete.



Review the Physical Simulation.

The Rotary Motor Physical Simulation tool combined Mates and Physical Dynamics to rotate the FLATBAR components in the LINKAGE assembly. The Rotary Motor was applied to the front face of the FLATBAR. You utilized the Calculate option to play the simulation. You saved the simulation in an .AVI file. Note: SW Animator is required to save the simulation in an .AVI file.



Additional details on Assembly, Mates, and Simulation are available in SolidWorks Help. Keywords: Standard Mates, Mate PropertyManager, Design Methods in Assembly, and Physical Simulation.



Review the Keyboard Short Cuts in the Appendix. Utilize the Keyboard Short Cuts to save modeling time.

Project Summary

In this project you created three parts, copied the AIRCYLINDER assembly from the CD in the book, and created the LINKAGE assembly.

You developed an understanding of the SolidWorks User Interface: Menus, Toolbars, Task Pane, System icon feedback, Keyboard shortcuts, Document Properties, Parts, and Assemblies.

You created 2D sketches and addressed the three key states of a Sketch: Fully Defined, Over Defined, and Under Defined.
Note: Always review your FeatureManager for the proper Sketch state.

You obtained the knowledge of the following SolidWorks features: Extruded Base, Extruded Cut, and Linear Pattern. Features are the building blocks of parts. The Extruded Base feature required a Sketch plane, Sketch, and Depth.

The Extruded Base feature added material to a part. The Extruded Base feature was utilized in the AXLE, SHAFT-COLLAR, and FLATBAR parts.

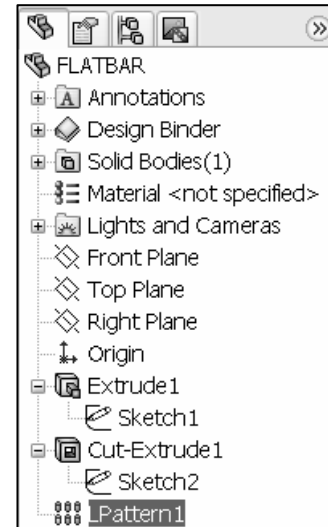
The Extruded Cut feature removed material from the part. The Extruded Cut feature was utilized to create a hole in the SHAFT-COLLAR, and FLATBAR parts.

The Linear Pattern feature was utilized to create an array of holes in the FLATBAR part.

When parts are inserted into an assembly, they are called components. You created the LINKAGE assembly by inserting the AIRCYLINDER assembly, AXLE, SHAFT-COLLAR, and FLATBAR parts.

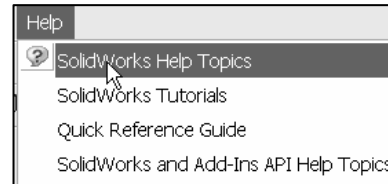
Mates are geometric relationships that align and fit components in an assembly. Concentric, Coincident, and Parallel Mates were utilized to assemble the components.

The Rotary Motor Physical Simulation tool combined Mates and Physical Dynamics to rotate the FLATBAR components in the LINKAGE assembly.



Project Terminology

Utilize SolidWorks Help for additional information on the terms utilized in this project.



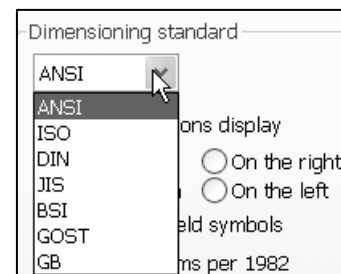
Assembly: An assembly is a document which contains parts, features, and other sub-assemblies. When a part is inserted into an assembly it is called a component. Components are Mated together. The filename extension for a SolidWorks assembly file name is .SLDASM.

Component: A part or sub-assembly within an assembly.

Cursor Feedback: Feedback is provided by a symbol attached to the cursor arrow indicating your selection. As the cursor floats across the model, feedback is provided in the form of symbols, riding next to the cursor.

Dimension: A value indicating the size of feature geometry.

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



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.

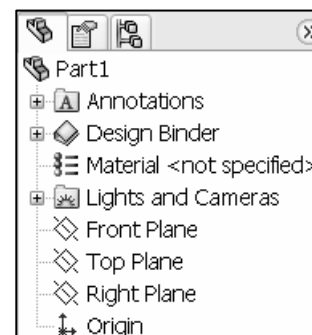
Mates: A Mate is a geometric relationship between components in an assembly.

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

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

Part: A part is a single 3D object made up of features. The filename extension for a SolidWorks part file name is .SLDPRT.

Plane: To create a sketch, select a plane. Planes are flat and infinite. They are represented on the screen with visible edges. The reference plane for this project is the Front plane.



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

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

Status of a Sketch: Three states are utilized in this Project:

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

Toolbars: The toolbar menus provide shortcuts enabling you to quickly access the most frequently used commands.

Trim Entities: Deletes selected sketched geometry. Extends a sketch segment unit it is coincident with another entity.

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

Project Features

Extruded Boss/Base: An Extruded Base feature is the first feature in a part. An Extruded Boss feature occurs after the Extruded Base feature. The Extruded Boss/Base feature adds material by extrusion.

Steps to create an Extruded Boss/Base Feature:

- Select the Sketch plane; Sketch the profile; Add Dimensions and Relations; Select Extruded Boss/Base from the Features toolbar; Select an End Condition and/or options; Enter a depth; Click OK from the Extrude PropertyManager.

Extruded Cut: The Extruded Cut feature removes material from a solid. The Extruded Cut feature performs the opposite function of the Extruded Boss/Base feature. The Extruded Cut feature starts with either a 2D or 3D sketch and removes material by extrusion.

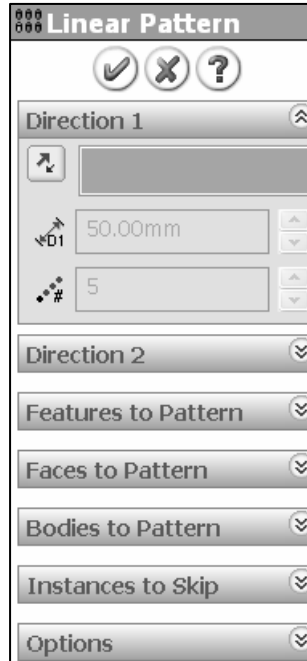
Steps to create an Extruded Cut Feature:

- Select the Sketch plane; Sketch the profile, Add Dimensions and Relations; Select Extruded Cut from the Features toolbar; Select an End Condition and/or options; Enter a depth; Click OK from the Cut-Extrude PropertyManager.

Linear Pattern: A Linear Pattern repeats features or geometry in an array. A Linear Pattern requires the number of instances and the spacing between instances.

Steps to create a Linear Pattern Feature:

- Select the features to repeat; Select Linear Pattern from the Feature toolbar; Enter Direction of the pattern; Enter Number of pattern instances in each direction; Enter Distance between pattern instances; Optional: Pattern instances to skip; Click OK from the Linear Pattern PropertyManager.



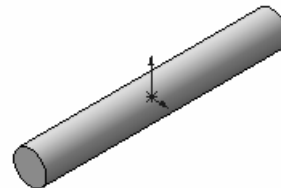
Engineering Journal

Engineers and designers utilize mathematics, science, economics and history to calculate additional information about a project. Answers to questions are written in an engineering journal.

1. Volume of a cylinder is provided by the formula, $V = \pi r^2 h$. Where:

- V is volume.
- r is the radius.
- h is the height.

- Determine the radius of the AXLE in mm.
- Determine the height of the AXLE in mm.
- Calculate the Volume of the AXLE in mm^3 .



2. Density of a material is provided by the formula: $\rho = m/V$. Where:

- ρ is density.
- m is mass.
- V is volume.

a) Determine the mass of the AXLE in grams if the AXLE is manufactured from hardened steel. The density of hardened steel is $.007842 \text{ g/mm}^3$.

3. The material supplier catalog lists Harden Steel Rod in foot lengths.

Harden Steel Rod (Ø 3/16):		
Part Number:	Length:	Cost:
23-123-1	1 ft	\$10.00
23-123-2	2 ft	\$18.00
23-123-3	3ft	\$24.00

Utilize the table above to determine the following questions:

How many 1-3/8 inch AXLES can be cut from each steel rod?

Twenty AXLE parts are required for a new assembly. What length of Harden Steel Rod should be purchased?

4. Air is a gas. Boyle's Law states that with constant temperature, the pressure, P of a given mass of a gas is inversely proportional to its volume, V .

- $P_1 / P_2 = V_2 / V_1$
- $P_1 \times V_1 = P_2 \times V_2$

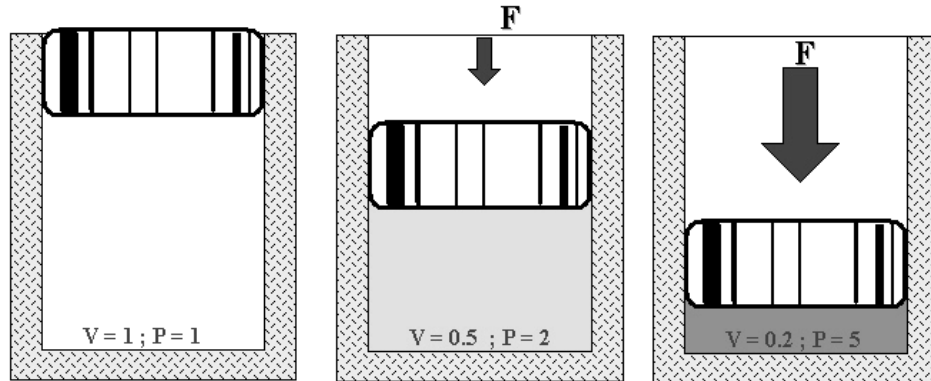


Illustration of Boyle's Law
Courtesy of SMC Corporation of America

The pressure in a closed container is doubled. How will the volume of air inside the container be modified?

Robert Boyle (1627-1691) was an Irish born, English scientist, natural philosopher and a founder of modern chemistry. Boyle utilized experiments and the scientific method to test his theories. Along with his student, Robert Hooke (1635-1703), Boyle developed the air pump.

Research other contributions made by Robert Boyle and Robert Hooke that are utilized today.

Questions

1. Explain the steps in starting a SolidWorks session.
2. Describe the procedure to begin a new sketch.
3. Explain the steps required to change part unit dimensions from inches to millimeters.
4. Describe a part.
5. Identify the three default reference planes.
6. What is the Base feature? Provide an example.
7. Describe the differences between an Extruded Base feature and an Extruded Cut feature.
8. The sketch color black indicates a sketch is _____ defined.
9. The sketch color blue indicates a sketch is _____ defined.
10. The sketch color red indicates a sketch is _____ defined.
11. Describe the procedure to “wake up” a centerpoint.
12. Define a relation. Provide an example.
13. What is a Linear Pattern? Provide an example.
14. Describe an assembly or sub-assembly.
15. What are Mates and why are they important in assembling components?
16. In an assembly, each component has _____ # degrees of freedom? Name them.
17. True or False. A fixed component cannot move in an assembly.
18. Review the Design Intent section in the book. Identify how you incorporated design intent into the parts and assembly.

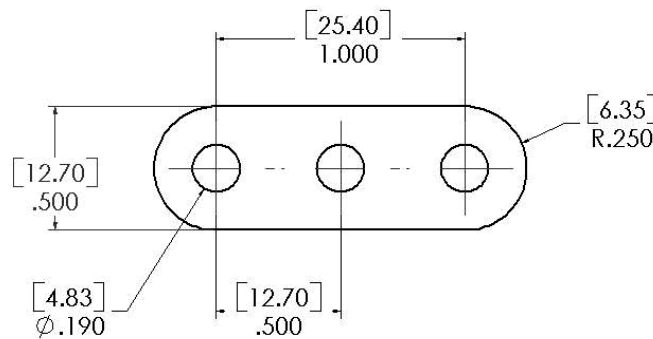
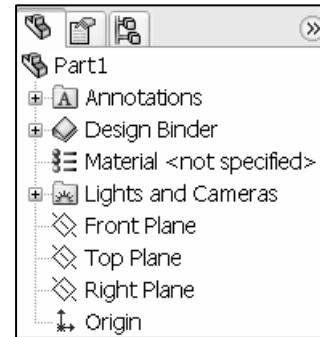


Exercises

Exercise 1.1: FLATBAR-3HOLE.

Create the FLATBAR-3HOLE part.

- Utilize the Front plane for the Sketch plane.
- Utilize a Linear Pattern for the three holes. The FLATBAR-3HOLE part is manufactured from 0.060in, [1.5mm] Stainless Steel.

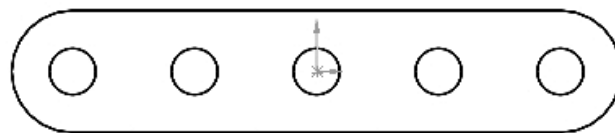


FLATBAR-3HOLE

Exercise 1.2: FLATBAR-5HOLE.

Create the FLATBAR-5HOLE part.

- Utilize the Front plane for the Sketch plane.
- Utilize a Linear Pattern for the five holes. The FLATBAR-5HOLE part is manufactured from 0.060in, [1.5mm] Stainless Steel.
- Calculate the required dimensions for the FLATBAR-5HOLE part. Use the following information:
 - Holes are .500in on center.
 - Radius is .250in.
 - Hole diameter is .190in.

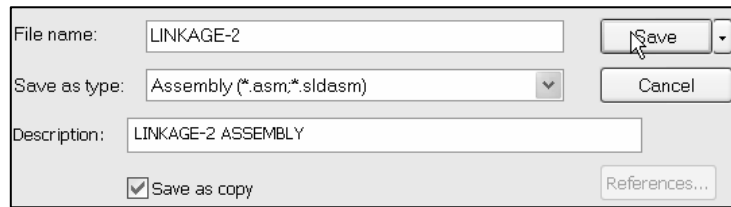


FLATBAR-5HOLE

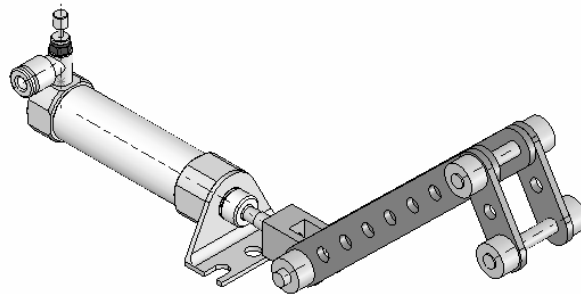
Exercise 1.3a: LINKAGE-2 Assembly.

Create the LINKAGE-2 assembly.

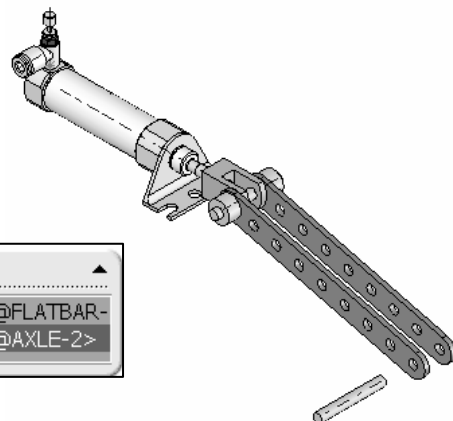
- Open the LINKAGE assembly.
- Select File, Save As from the Main menu.
- Check the Save as Copy check box.
- Enter LINKAGE-2 for file name. LINKAGE-2 ASSEMBLY for Description.



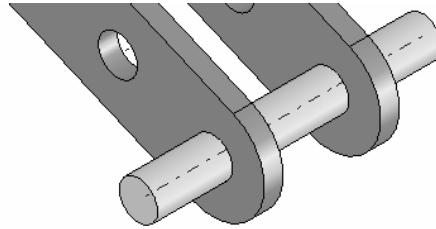
The FLATBAR-3HOLE part was created in Exercise 1.1. Utilize two AXLE parts, four SHAFT COLLAR parts, and two FLATBAR-3HOLE parts to create the LINKAGE-2 assembly as illustrated.



- Insert the first AXLE part.
- Insert a Concentric Mate.



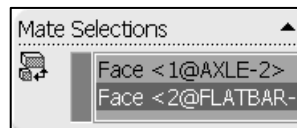
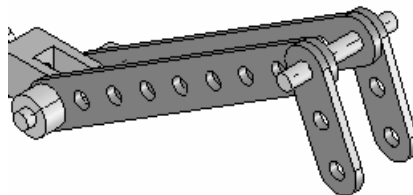
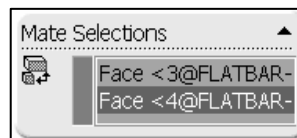
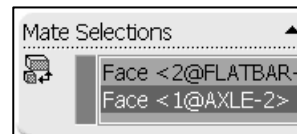
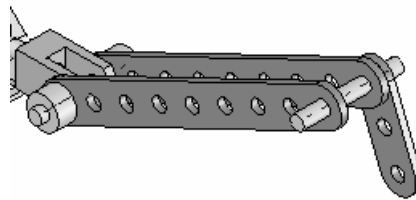
- Insert a Coincident Mate.



- Insert the first FLATBAR-3HOLE part.

- Insert a Concentric Mate.

- Insert a Coincident Mate. Perform the same procedure for the second FLATBAR-3HOLE part.



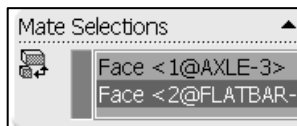
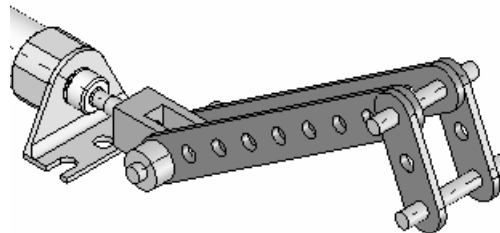
- Insert a Parallel Mate between the 2 FLATBAR-3HOLE parts. Note: The 2 FLATBAR-3HOLE parts move together.



- Insert the second AXLE part.

- Insert a Concentric Mate.

- Insert a Coincident Mate.

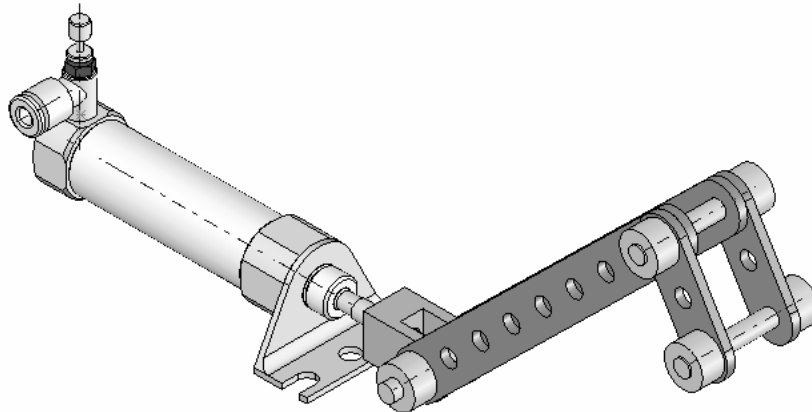
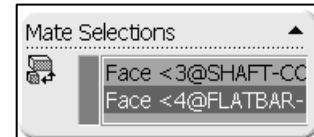


- Insert the first SHAFT-COLLAR part.

- Insert a Concentric Mate.



- Insert a Coincident Mate.
- Perform the same tasks to insert the other three required SHAFT-COLLAR parts.



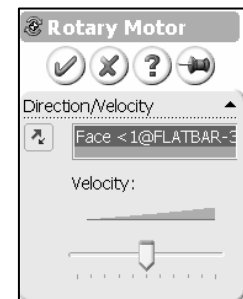
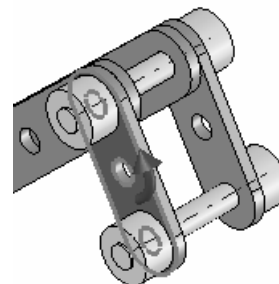
Exercise 1.3b: LINKAGE-2 Assembly Simulation.

Use the LINKAGE-2 assembly created in the previous exercise for the simulation

- Apply a Rotary Motor to the front FLATBAR-3HOLE.
- Record the Simulation.
- Play the Simulation.



Rotary Motor
Moves a component as if acted upon by a rotary motor.



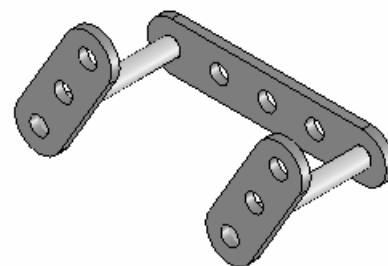
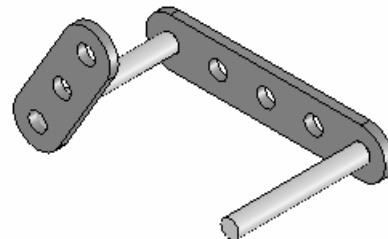
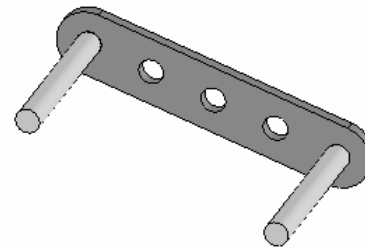
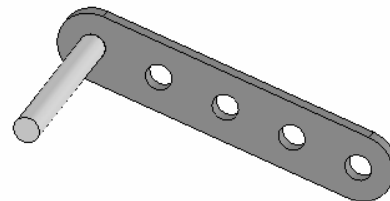
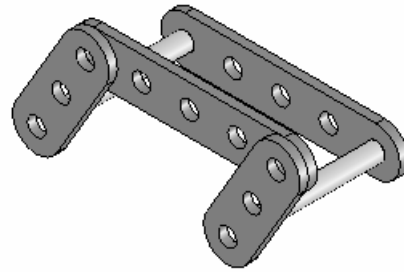
Exercise 1.4a: ROCKER Assembly.

Create a ROCKER assembly. The ROCKER assembly consists of two AXLE parts, two FLATBAR-5HOLE parts, and two FLATBAR-3HOLE parts.

The FLATBAR-3HOLE parts are linked together with the FLATBAR-5HOLE.

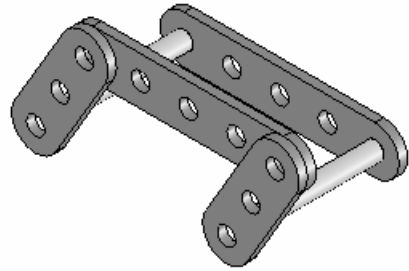
The three parts rotate clockwise and counterclockwise, above the Top plane. Create the ROCKER assembly.

- Insert the first FLATBAR-5HOLE part. The FLATBAR-5HOLE is fixed to the Origin of the ROCKER assembly.
- Insert the first AXLE part.
- Insert a Concentric Mate.
- Insert a Coincident Mate.
- Insert the second AXLE part.
- Insert a Concentric Mate.
- Insert a Coincident Mate.
- Insert the first FLATBAR-3HOLE part.
- Insert a Concentric Mate.
- Insert a Coincident Mate.
- Insert the second FLATBAR-3HOLE part.
- Insert a Concentric Mate.
- Insert a Coincident Mate.




- Insert the second FLATBAR-5HOLE part.
- Insert a Concentric Mate.
- Insert a Coincident Mates.

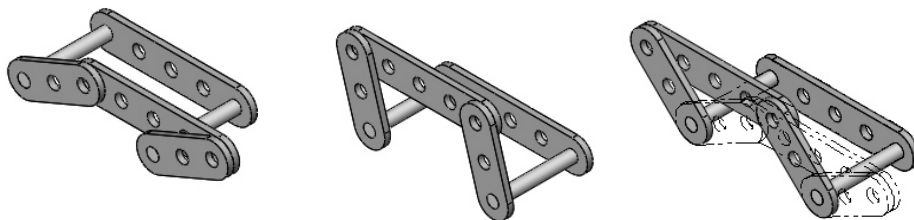
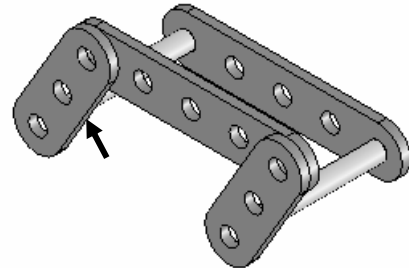
Note: The end holes of the second FLATBAR-5HOLE are concentric with the end holes of the FLATBAR-3HOLE parts.



Exercise 1.4b: ROCKER Assembly Physical Simulation.

Create the ROCKER Physical Simulation.

- Apply a Rotary Motor to the left FLATBAR-3HOLE in a counterclockwise direction.
- Record the simulation.
- Select Reciprocating Replay .
- Select Play. The Reciprocating Replay continuously plays the simulation from the beginning to the end and then from the end to the beginning.



ROCKER Assembly Physical Simulation

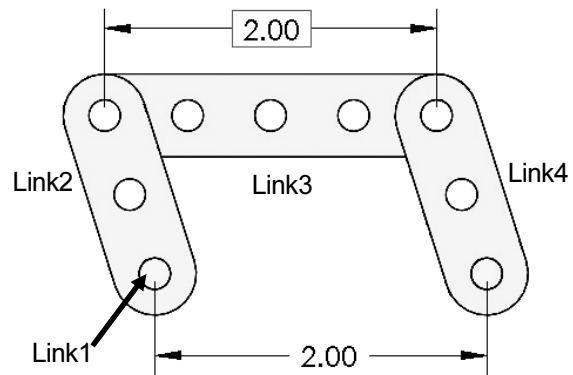
In mechanical design, the ROCKER assembly is classified as a mechanism. A Four-Bar Linkage is a common mechanism comprised of four links.

Link1 is called the Frame.

The AXLE part is Link1.

Link2 and Link4 are called the Cranks.

The FLATBAR-3HOLE parts are Link2 and Link4. Link3 is called the Coupler.
The FLATBAR-5HOLE part is Link3.



The Injection Molded Process

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

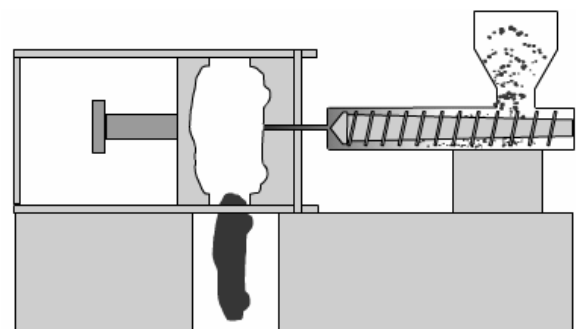
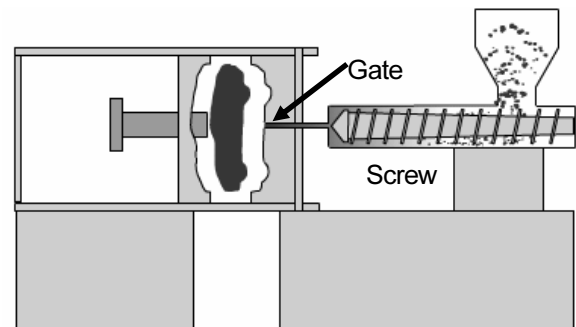
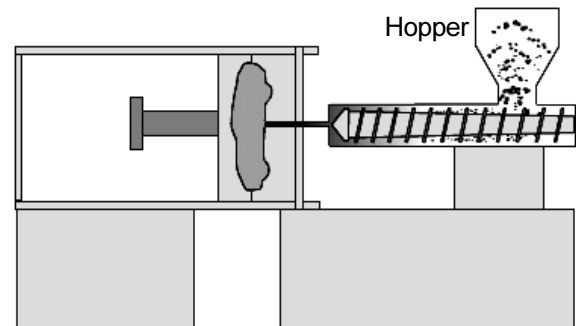
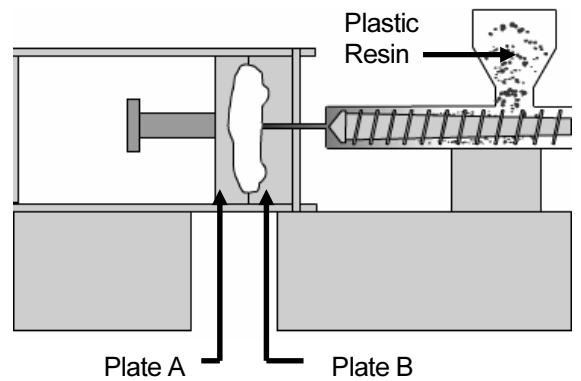
The injection molding process is as follows:

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

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

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

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



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

Exercise 1.5: Industry Application

Engineers and designers develop a variety of products utilizing SolidWorks.

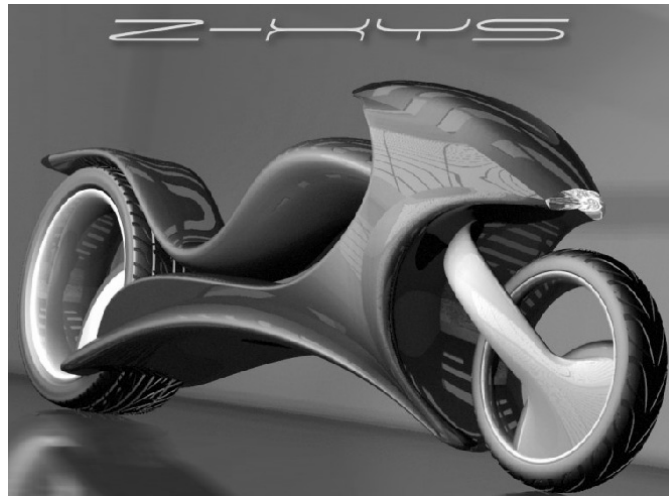
Model information is utilized to create plastic molds for products from toys to toothbrushes.

- Utilize the World Wide Web and review the following web sites: mikejwilson.com and zxys.com.

The models obtained from these web sites are for educational purposes only.

Learn modeling techniques from others; create your own designs. A common manufacturing procedure for plastic parts is named the Injection Molding Process. Today's automobiles utilize over 50% plastic components.

Engineers and designers work with mold makers to produce plastic parts. Cost reduction drives plastic part production.



Model Courtesy of
Mike J. Wilson,
CSWP

