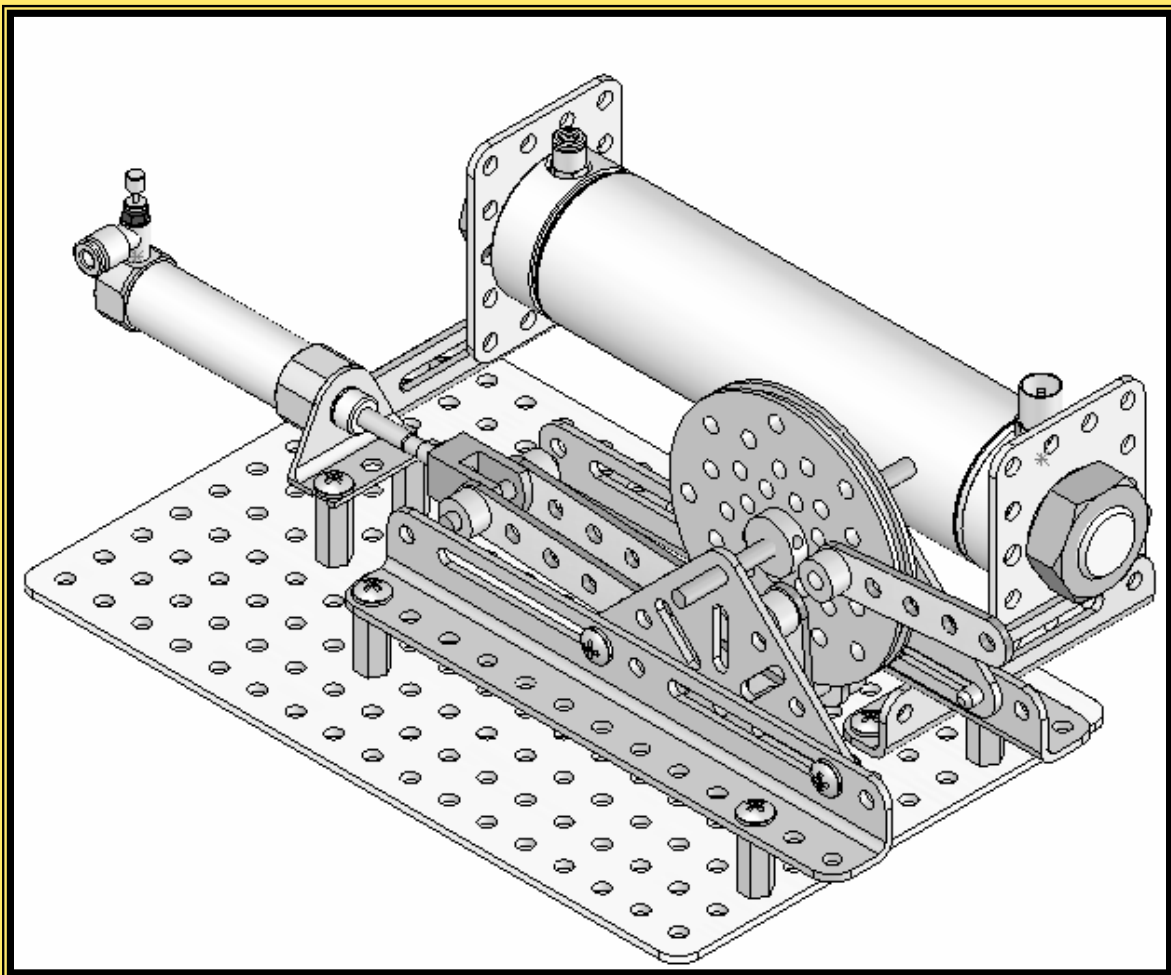


SolidWorks 2006 Tutorial

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

David C. Planchard & Marie P. Planchard



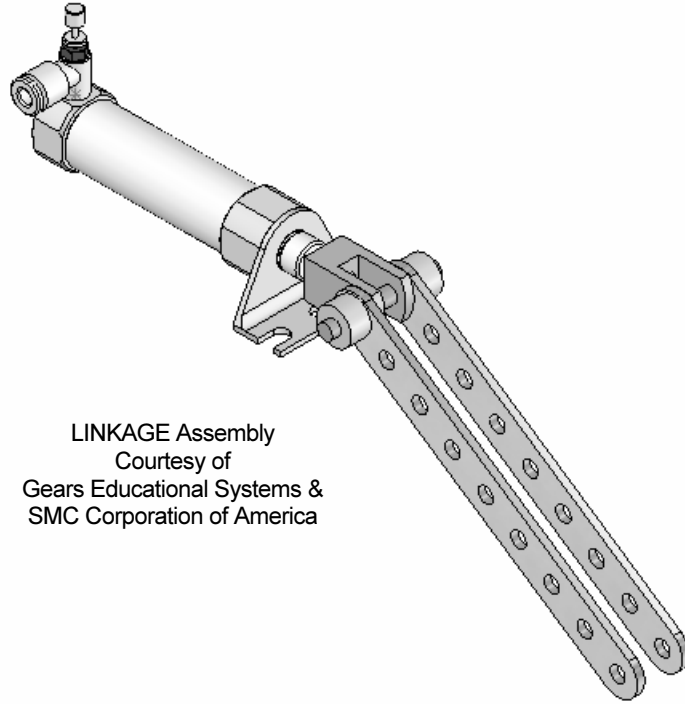
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Project 1

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.• Develop new parts.• Model new features: Extruded Base, Extruded Cut and Linear Pattern.
<ul style="list-style-type: none">• Develop an assembly:<ul style="list-style-type: none">○ LINKAGE assembly.	<ul style="list-style-type: none">• Use the assembly process with the following 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 downloaded sub-assembly:

- LINKAGE assembly.

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

- Establish a SolidWorks session.
- Set units and dimensioning standards.
- Create a part.
- Generate a sketch.
- Add and modify dimensions.
- Add Geometric Relations.
- Download an assembly into SolidWorks.
- Create an assembly using existing parts.
- Insert Coincident, Concentric and Parallel Mates.
- Use 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.

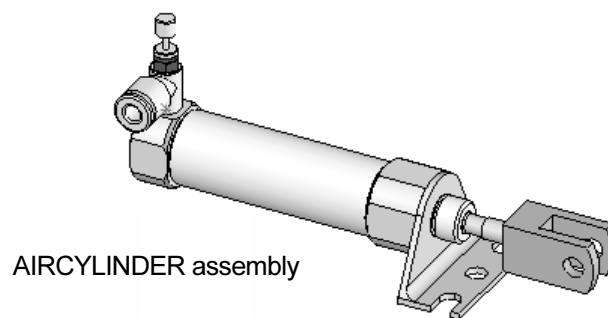
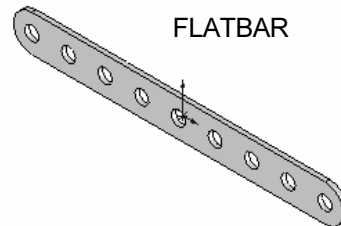
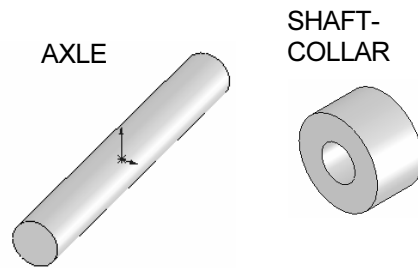
SolidWorks is a 3D solid modeling CAD program. SolidWorks provides design software to create 3D models and 2D drawings.

Create three parts in this project:

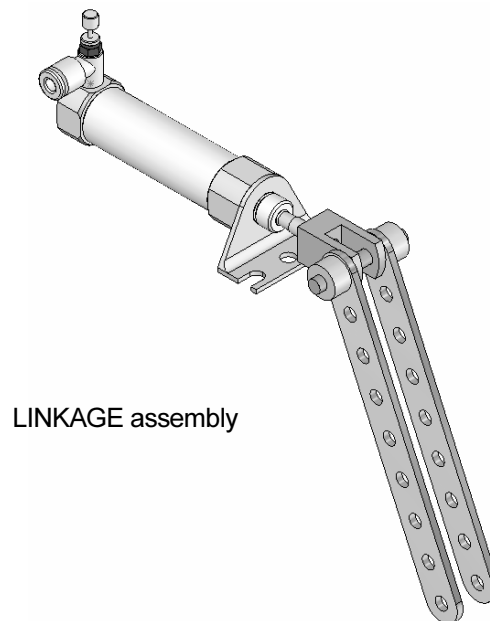
- AXLE.
- SHAFT-COLLAR.
- FLATBAR.

Download the AIRCYLINDER assembly from the enclosed CD.

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

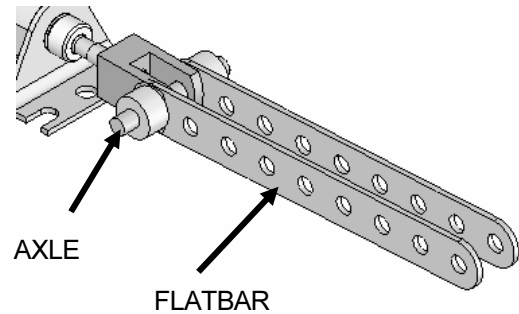


Combine the created parts and the downloaded AIRCYLINDER assembly to create the LINKAGE assembly.



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.

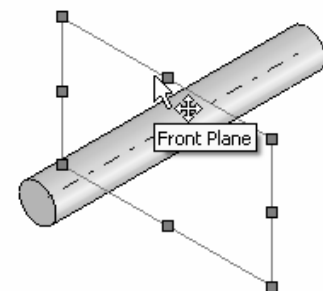
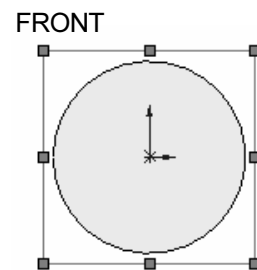
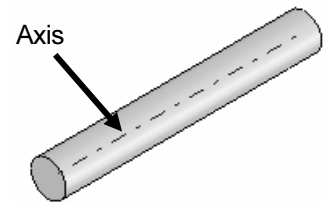
Utilize the Extruded Base feature. The Extruded Base feature adds material. The Base feature is the first feature of the part. The Base feature is the foundation of the part. Keep the Base feature simple!

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. 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.

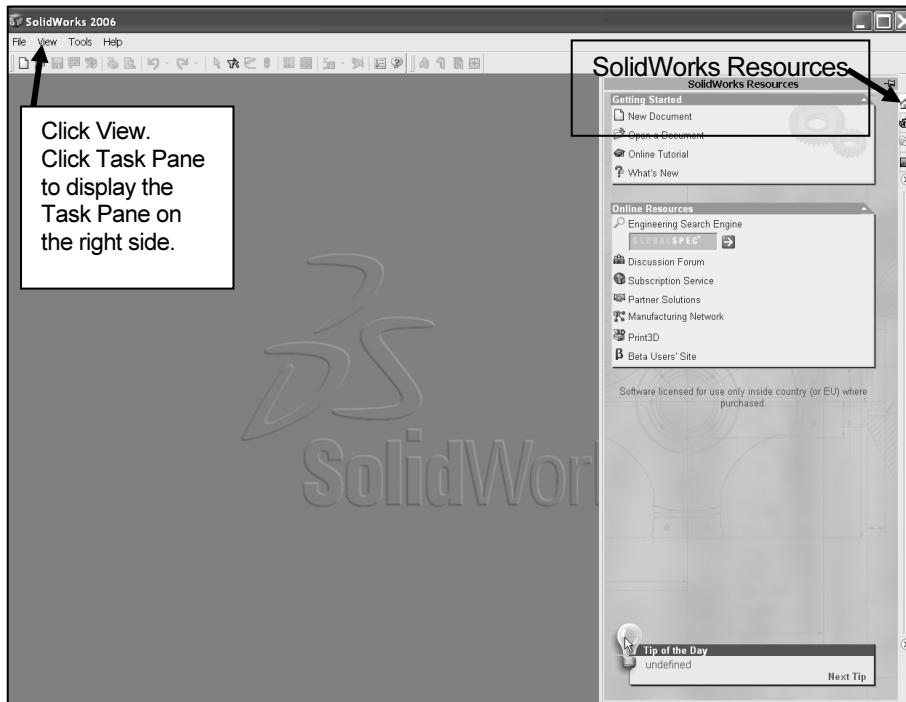
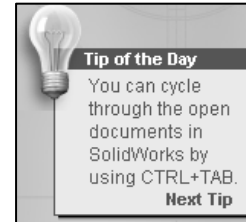
Activity: Start a SolidWorks Session

Start a SolidWorks 2006 session.

- 1) Click **Start**  on the Windows Taskbar.
- 2) Click **All Programs** .
- 3) Click the **SolidWorks 2006**  SolidWorks 2006 folder.
- 4) Click **SolidWorks 2006**  SolidWorks 2006 application. The SolidWorks program window opens. Note: Do not open a document.

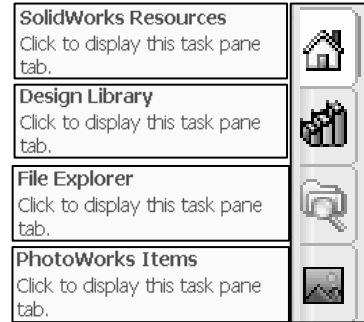
Read the Tip of the Day dialog box.


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



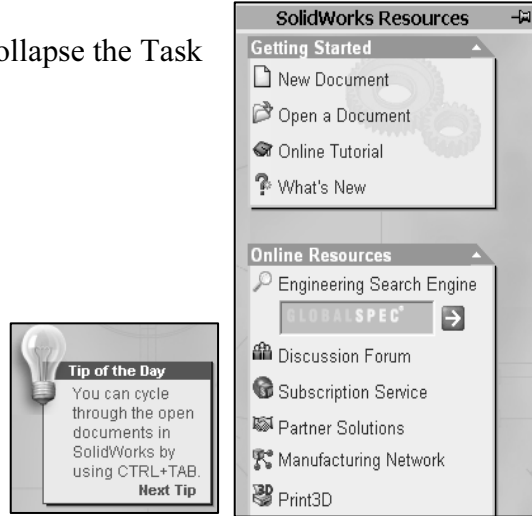
The SolidWorks 2006 Task Pane contains four options:

- SolidWorks Resources.
- Design Library.
- File Explorer.
- PhotoWorks Items. Note: If PhotoWorks Items is not displayed, click Tools, Add-Ins from the Main menu. Check PhotoWorks. Click OK.




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

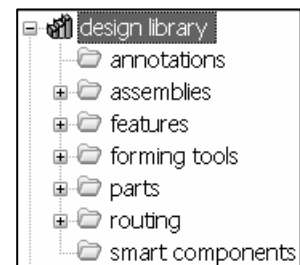
SolidWorks Resources contains the Getting Started menu, the Online Resources menu and the Tip of the Day.



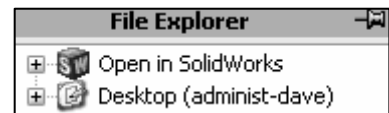
The Design Library includes entries for the Design Library, Toolbox and 3D ContentCentral. The Design Library contains the following folders: annotations, assemblies, features, forming tools parts routing and smart components.



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



File Explorer duplicates Windows Explorer in functionality.



PhotoWorks Items create photo-realistic images of SolidWorks models. PhotoWorks provides many professional rendering effects. PhotoWorks is based on the mental ray® rendering engine.



You will explore the options in SolidWorks Resources, Design Library, File Explorer and PhotoWorks as you use this book.

There are two modes in the New SolidWorks Document dialog box: Novice and Advanced.

The Novice option is the default option with three templates.

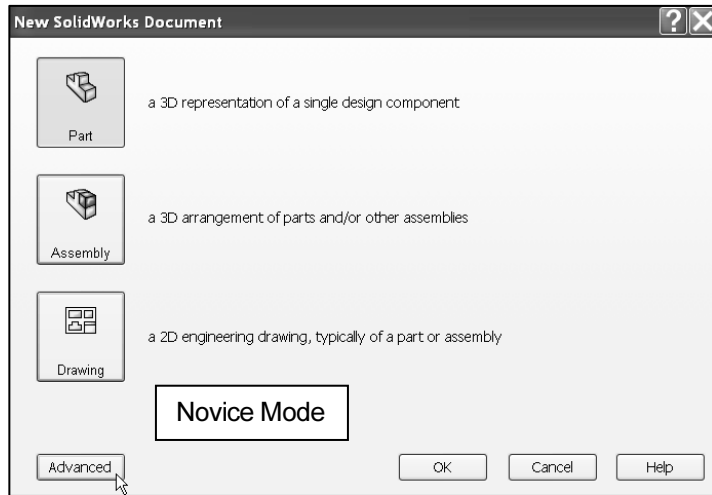
The Advanced option contains access to more templates.

Create a new part.

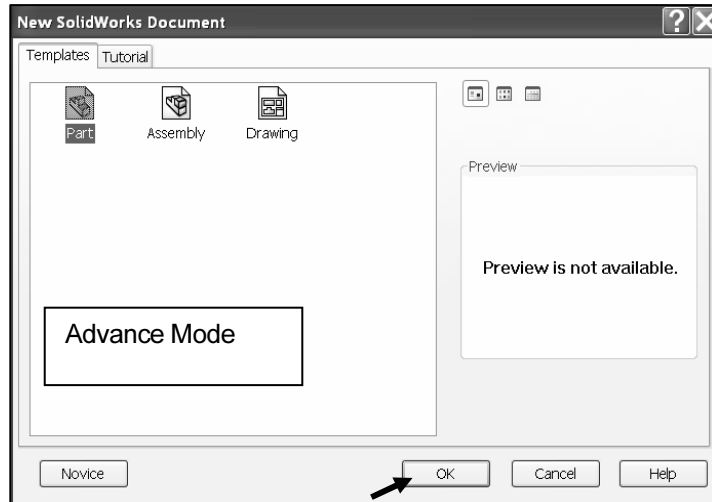
- 6) Click **File, New** from the Main menu.

Select Advanced Mode.

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




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



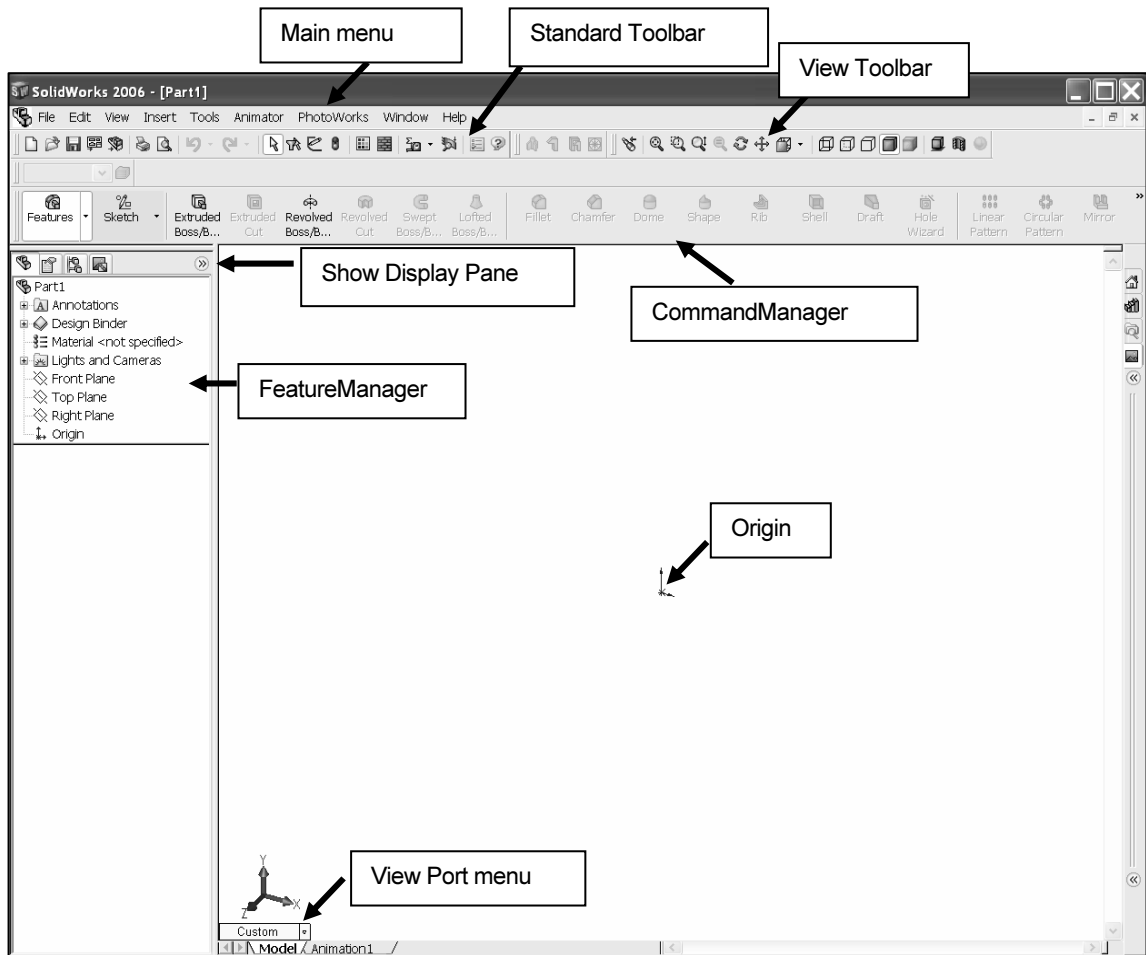
User Interface and CommandManager

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

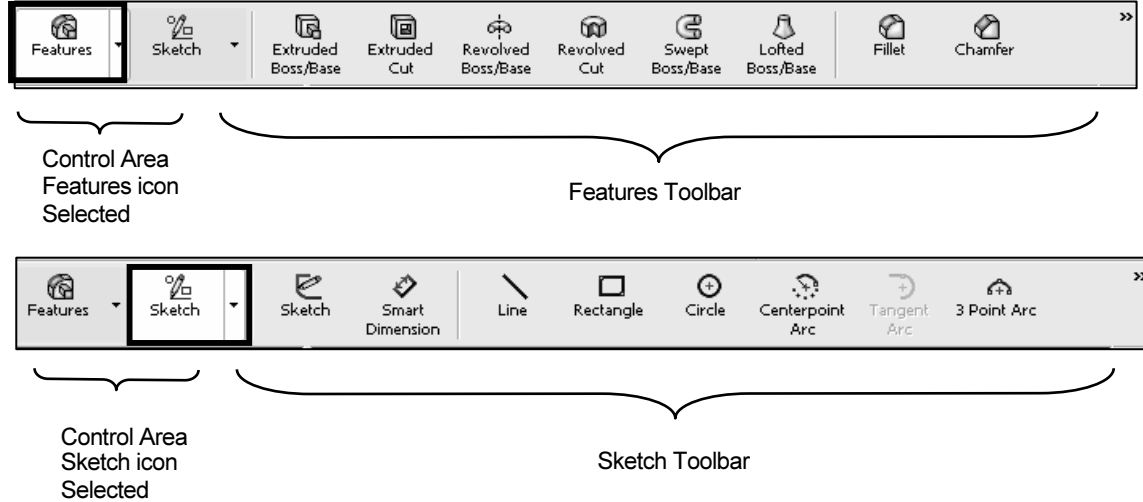
Part1 is displayed. Part1 is the new default part window name. The Main menu, Standard Toolbar, View Toolbar and CommandManager are displayed above the Graphics window.

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

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

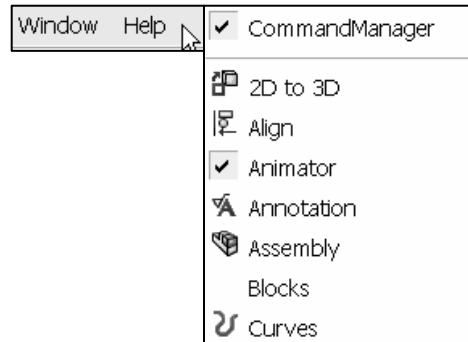


The CommandManager is divided into the Control Area and an expanded Toolbar. Select a Control Area icon to display the corresponding toolbar. The Features icon and Features Toolbar are selected by default in Part mode.

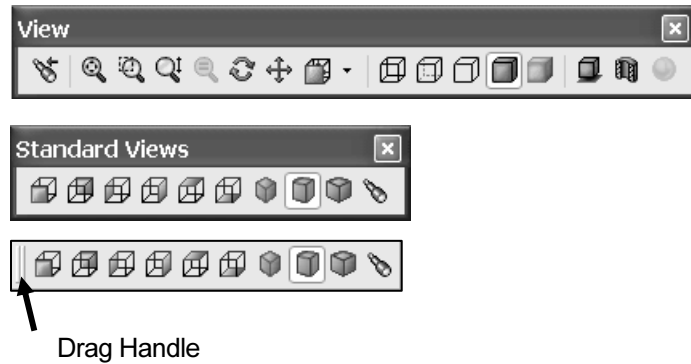


The CommandManager is utilized in this text. Control the CommandManager display.

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.



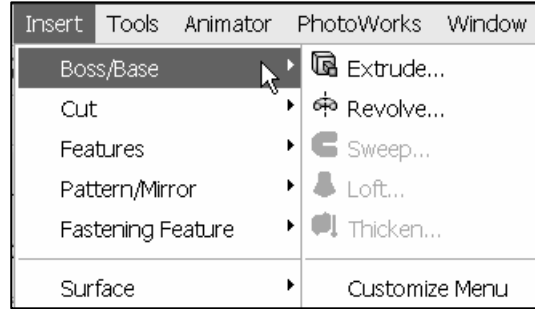
Select individual toolbars from the toolbar list to display in the Graphics window. Reposition toolbars by moving their drag handle.



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

Example:

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



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

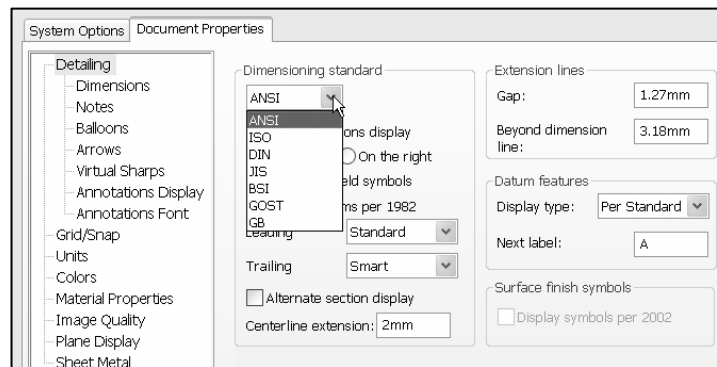
Activity: AXLE Part

Set the Document Properties.

- 9) Click **Tools, Options, Document Properties** tab from the Main menu.
- 10) Select **ANSI** from the Dimensioning standard drop down list. Various Detailing options are available depending on the selected standard.

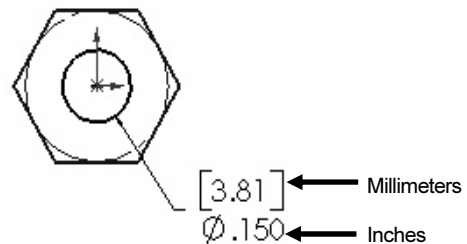
The Dimensioning Standard determines the display of dimension text, arrows, symbols and spacing.

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).

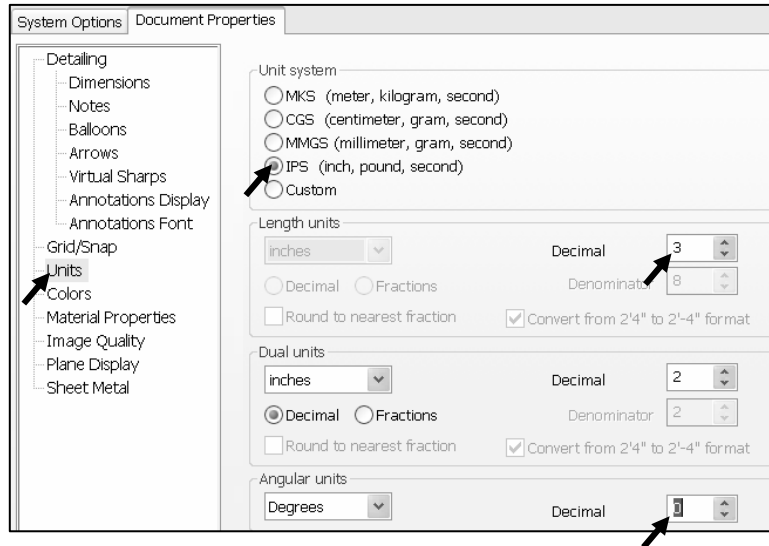
The optional secondary units are provided in MMGS (millimeters, grams, second) and are indicated in brackets [].



Illustrations are provided in both inches and millimeters.






Set the part units.

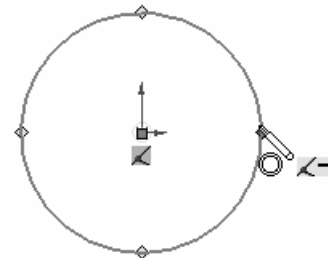
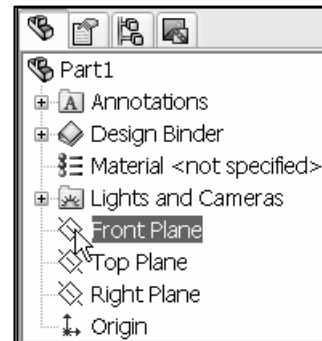
- 11) Click **Units**. Click **IPS** [MMGS] for Unit System.
- 12) Select **3** [2] for Length units Decimal places.
- 13) Select **0** [0] for Angular units Decimal places.
- 14) Click **OK** to set the document units.





Activity: AXLE Part-Extruded Base Feature

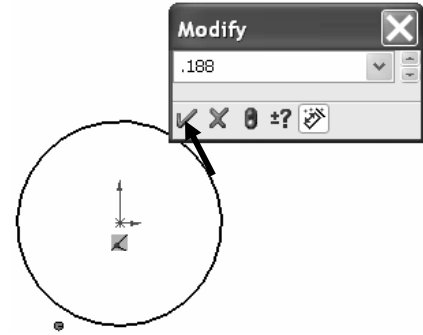
Insert a new Sketch for the Extruded Base feature.

- 15) Click the **Front Plane**  Front Plane from the FeatureManager for the Sketch plane.
- 16) Click **Sketch**  Sketch from the Control Area.
- 17) Click **Circle**  Circle from the Sketch toolbar.
- 18) Move 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.
- 19) Click the **Origin**  of the circle. The cursor displays the Coincident to point feedback symbol.
- 20) Drag the **mouse pointer** to the right of the Origin to create the circle.
- 21) Click a **position** to create the circle.





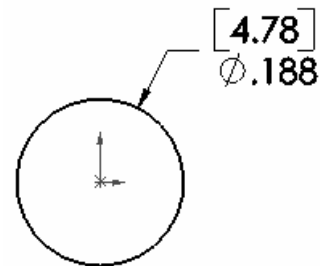
Add a dimension.

- 22) Click **Smart Dimension**  from the Sketch toolbar.
- 23) Click the **circumference** of the circle. The cursor displays the diameter feedback symbol.
- 24) Click a **position** diagonally above the circle in the Graphics window.
- 25) Enter **.188 [4.78]** in the Modify dialog box.
- 26) Click the **Green Check mark**  in the Modify pop-up box. The diameter of the circle is .188 inches.






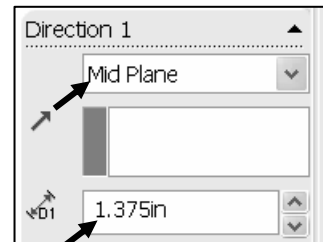
The circular sketch is centered at the Origin. The dimension indicates the diameter of the circle.

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



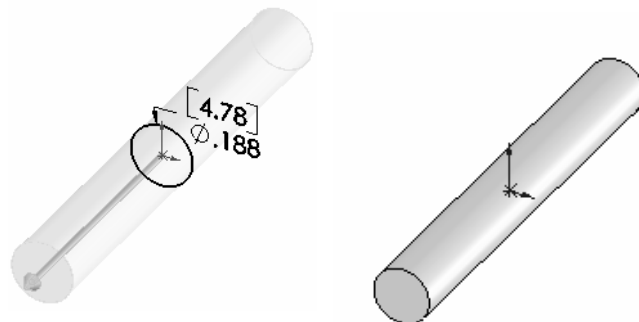
Extrude the sketch to create the first feature.

- 27) Click **Features**  from the Control Area.
- 28) Click **Extruded Boss/Base**  from the Features toolbar. The Extrude PropertyManager is displayed.
- 29) Select **Mid Plane** for Direction 1 End Condition.
- 30) Enter **1.375 [34.93]** for Depth.
- 31) Click **OK**  from the Extrude PropertyManager.



Fit the model to the Graphics window.

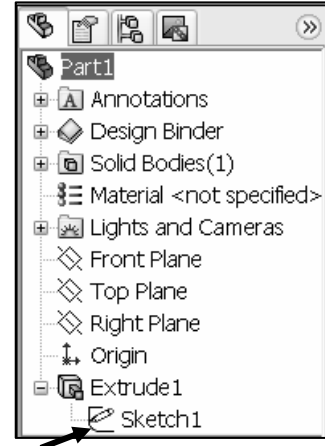
- 32) Press the **f** key.



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 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 .188 [4.76]. The Depth is 1.375 [34.93].

Activity: AXLE Part-Save

Save the part.

33) Click **File, Save As** from the Main toolbar.

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

35) Click **Create New Folder** .

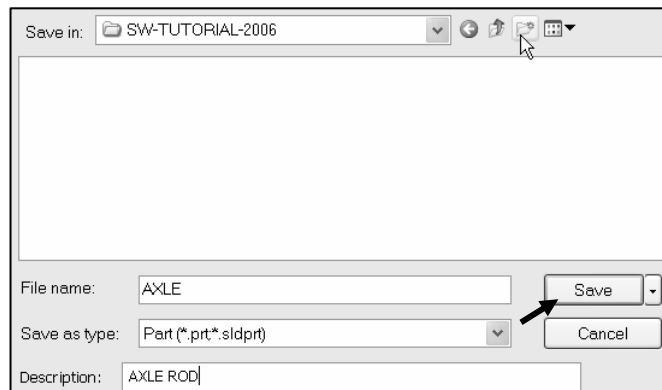
36) Enter **SW-TUTORIAL-2006** for the file folder name.

37) Double-click the **SW-TUTORIAL-2006** file folder. SW-TUTORIAL-2006 is the Save in file folder name.

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

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


40) Click the **Save** button.




Organize parts into file folders. The file folder for this project is named: SW-TUTORIAL-2006. Save all documents in the SW-TUTORIAL-2006 file folder.

Activity: AXLE Part-Edit Color

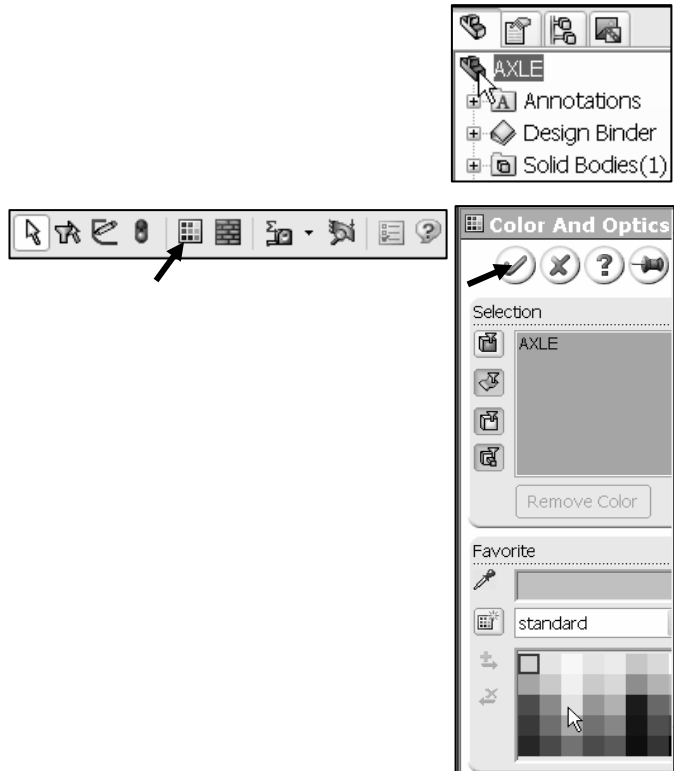
Modify the color of the part.

41) Click the **AXLE Part**  icon at the top of the FeatureManager.

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

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

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

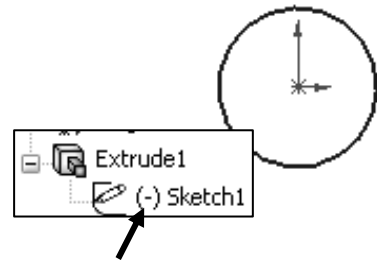


Utilize Edit Color to control part and feature color. SolidWorks utilizes default colors to indicate status of sketches and features.

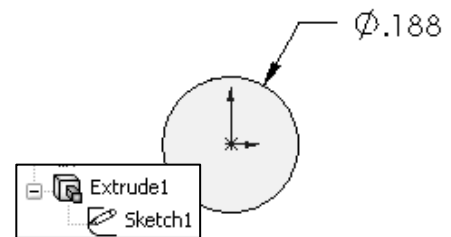
Example: Default Colors indicate the status of a sketch.

Sketches are:

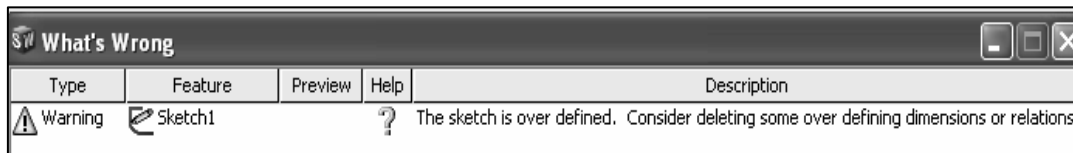
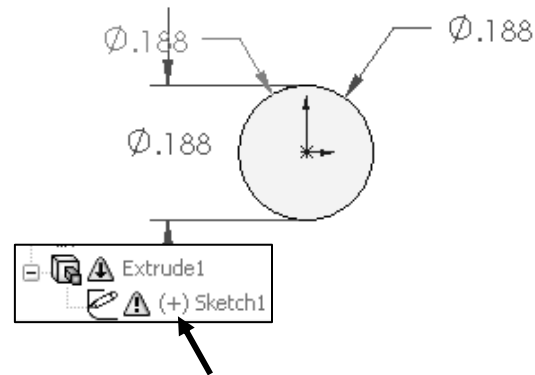
Under Defined: There is inadequate definition of the sketch, (Blue). The FeatureManager displays a minus (-) symbol before the Sketch name.



Fully Defined: Has complete information, (Black). The FeatureManager displays no symbol before the Sketch name.



Over Defined: Has duplicate dimensions, (Red). The FeatureManager displays a (+) symbol before the Sketch name. The What's Wrong dialog box appears.

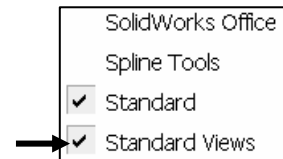


Activity: AXLE Part-Standard Views and View Modes

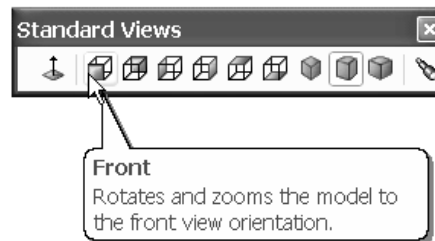
Display the Standard Views toolbar.

45) Click **View, Toolbars, Standard Views** from the Main menu. The Standard Views toolbar is displayed below the Main menu.

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



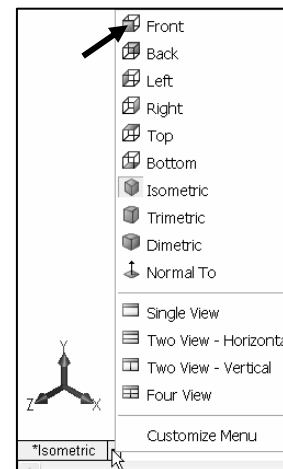
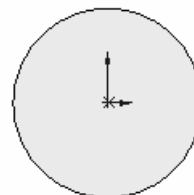
Orthographic projection is the process of projecting views onto parallel planes with \perp projectors. The default reference planes are the Front, Top and Right side viewing planes.



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

Display the Standard Views for the AXLE.

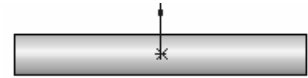
47) Click **Front view**  from the Standard Views menu or from the View Port Menu.



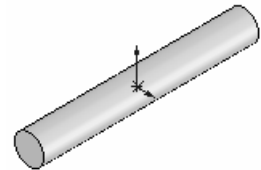
- 48) Click **Top view**.  Top .



- 49) Click **Right view**.  Right .




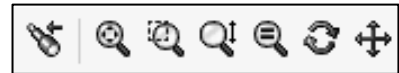
- 50) Click **Isometric view**.  Isometric .




View modes manipulate the model in the Graphics window.


Display the various View modes.


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




- 52) Click **Zoom to Area**  . Select two **opposite corners** of a rectangle to define the boundary of the view.

The defined view fits to the current window.

- 53) Click **Zoom In/Out**  . Drag upward to **zoom in**. Drag downward to **zoom out**. Press the lower case **z** key to zoom out. Press the upper case **Z** key to zoom in.

- 54) **Right-click** in the Graphics window. Click **Select**. Click the **front circular edge**. Click **Zoom to Selection**  . The selected geometry fills the current window.

- 55) Click **Rotate**  . Drag the **mouse pointer** to rotate about the screen center. Use the computer keyboard **arrow keys** to rotate in 15-degree increments.

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

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


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

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




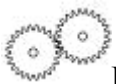
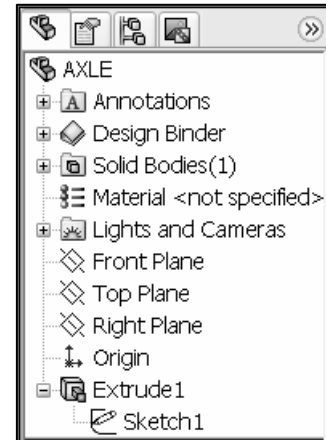
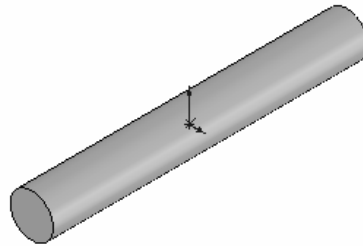
Utilize the center wheel of the mouse to **Zoom In/Zoom Out** and **Rotate** the model in the Graphics window.

Display the Isometric view.

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

Save the AXLE part.

- 60)** 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 change the color of the part. Color 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 hand 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.

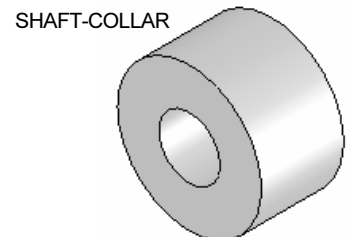
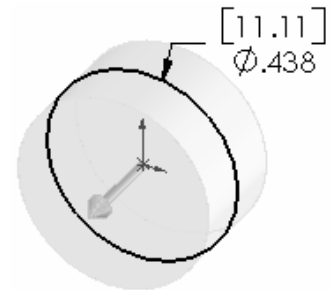
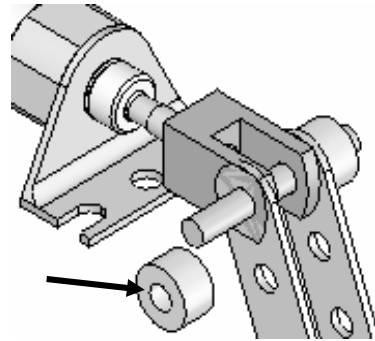
Utilize the Extruded Base feature. The Extruded Base feature requires a 2D circular profile.

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.

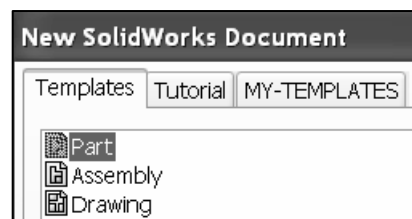
The Depth Through All option extends the Extruded Cut feature from the front face through all existing geometry.



Activity: SHAFT-COLLAR Part-Extruded Base Feature

Create a new part.

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



Save the Part.

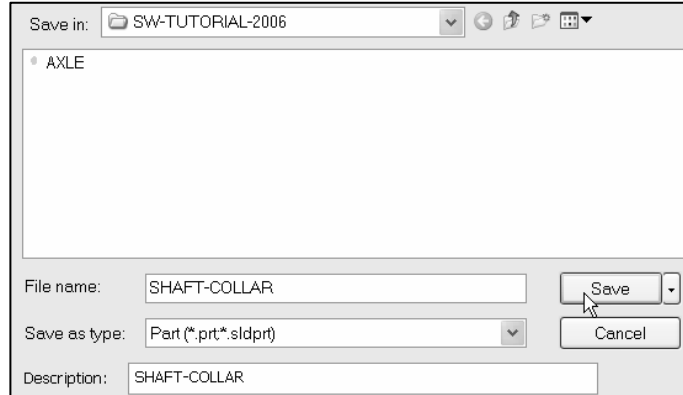
63) Click **File, Save As** from the Main menu.

64) Select **SW-TUTORIAL-2006** for file folder.

65) Enter **SHAFT-COLLAR** for File name.

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

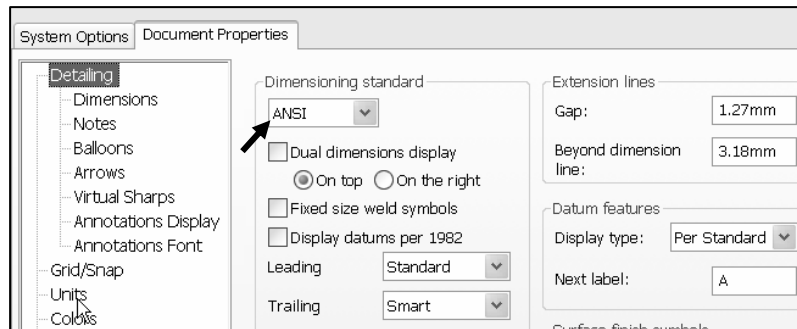
67) Click **Save**.



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 list box.



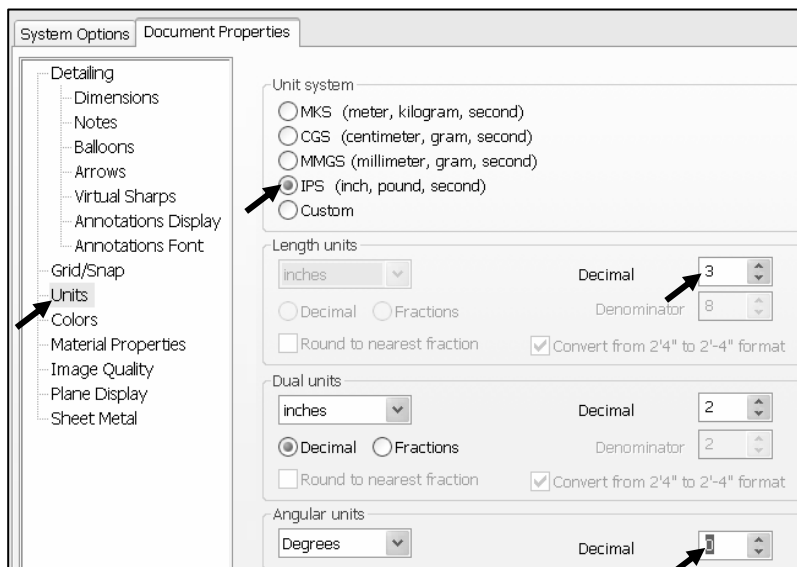
70) Click **Units**.

71) Click **IPS [MMGS]** for Unit System.

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

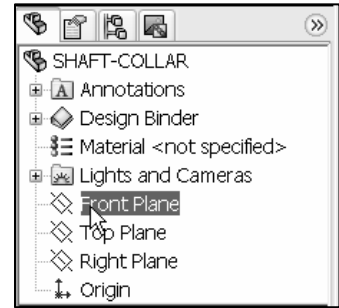
73) Select **0 [0]** for Angular units Decimal places.



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




Insert a new Sketch for the Extruded Base feature.

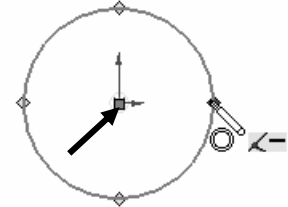
75) Click the **Front Plane**  Front Plane from the FeatureManager for the Sketch plane.




76) Click **Sketch**  Sketch from the Control Area. Click **Circle**  Circle from the Sketch toolbar.

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

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



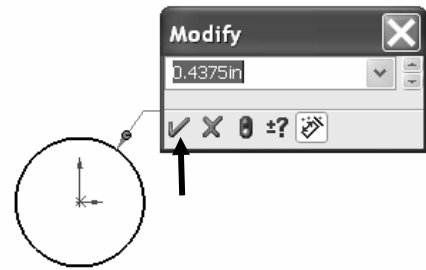
Add a dimension.

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

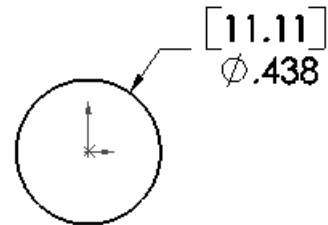
80) Click a **position** diagonally above the circle in the Graphics window.

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

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




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




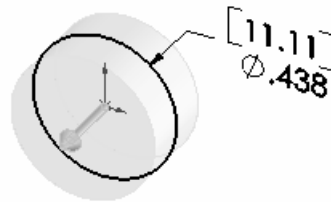
Extrude the sketch to create the first feature.

83) Click **Features**  Features from the Control Area.

84) Click **Extruded Boss/Base**  Extruded Boss/B... from the Features toolbar.

85) Select **Mid Plane** for Direction1 End Condition.

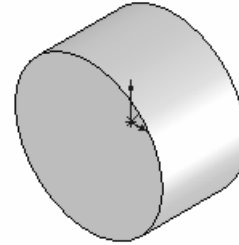
86) Enter **.250 [6.35]** for Depth. Click **OK**  from the Extrude PropertyManager.



Fit the model to the Graphics window.

87) Press the **f** key.

88) Click **Isometric view** .







Save the model.

89) Click **Save** .

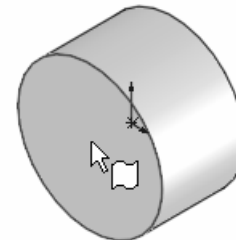
Activity: SHAFT-COLLAR Part-Extruded Cut Feature

Insert a new sketch for the Extruded Cut feature.


90) Click the **front circular face** of the Extrude1 feature for the Sketch plane. The cursor displays the Face feedback symbol.

 | Select Line
 Select Face
 * Select Point
 □ Select Vertex

Feedback Symbols



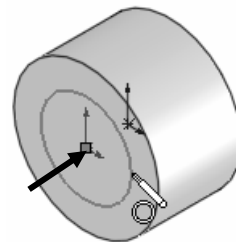
91) Click **Sketch**  from the Control Area.

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

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

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

95) Click a **position** to create the circle.



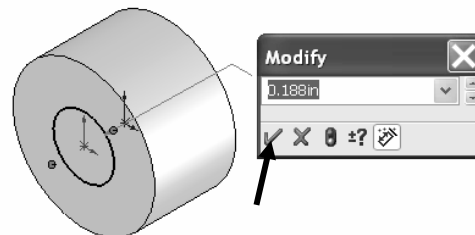
Add a dimension.

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

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

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


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




100) Click the **Green Check mark**  in the Modify pop-up box.

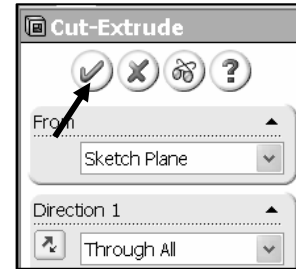
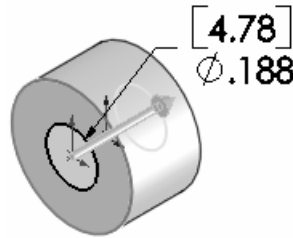
Insert an Extruded Cut feature.

- 101)** Click **Features**  from the Control Area.

- 102)** Click **Extruded Cut**  from the Features toolbar.

- 103)** Select **Through All** for Direction1 End Condition.

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



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.

- 105)** 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.

- 106)** Double-click the **.250 [6.35]** Depth dimension.

- 107)** Enter **.500 [12.70]**.

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



- 109)** Click the **Green Check mark**  from the Modify pop-up box.

The Extrude1 and Cut-Extrude1 features are modified.

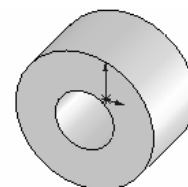
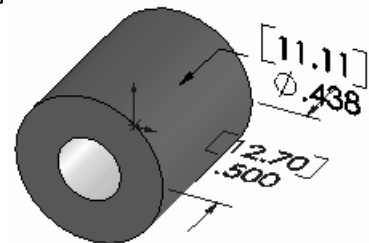
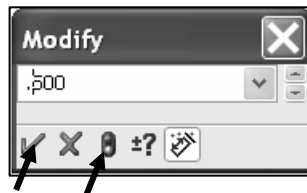
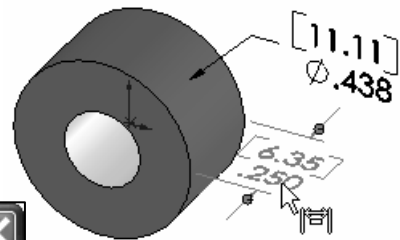
Return to the original dimensions.

- 110)** Double-click the **.500 [6.35]** Depth dimension.

- 111)** Enter **.250 [6.35]** in the Modify dialog box.

- 112)** Click **Rebuild**  from the Modify pop-up box. Click the **Green Check mark**  from the Modify pop-up box.

- 113)** Click **OK**  from the Dimension PropertyManager.




Modify the part color.


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

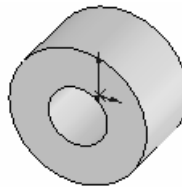
115) Click **Edit Color**  from the Standard toolbar.

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

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

Save the SHAFT-COLLAR part.

118) 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 Online Help. Select Help, SolidWorks Help topics. Keywords: Circle, Modify, Sketch (color), Dimension, Extrude.

 The SolidWorks Help Topics  SolidWorks Help Topics contains step-by-step instructions for various commands.

The Help  icon appears in the dialog box or PropertyManager for each tool.

Display Help for a rectangle.


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

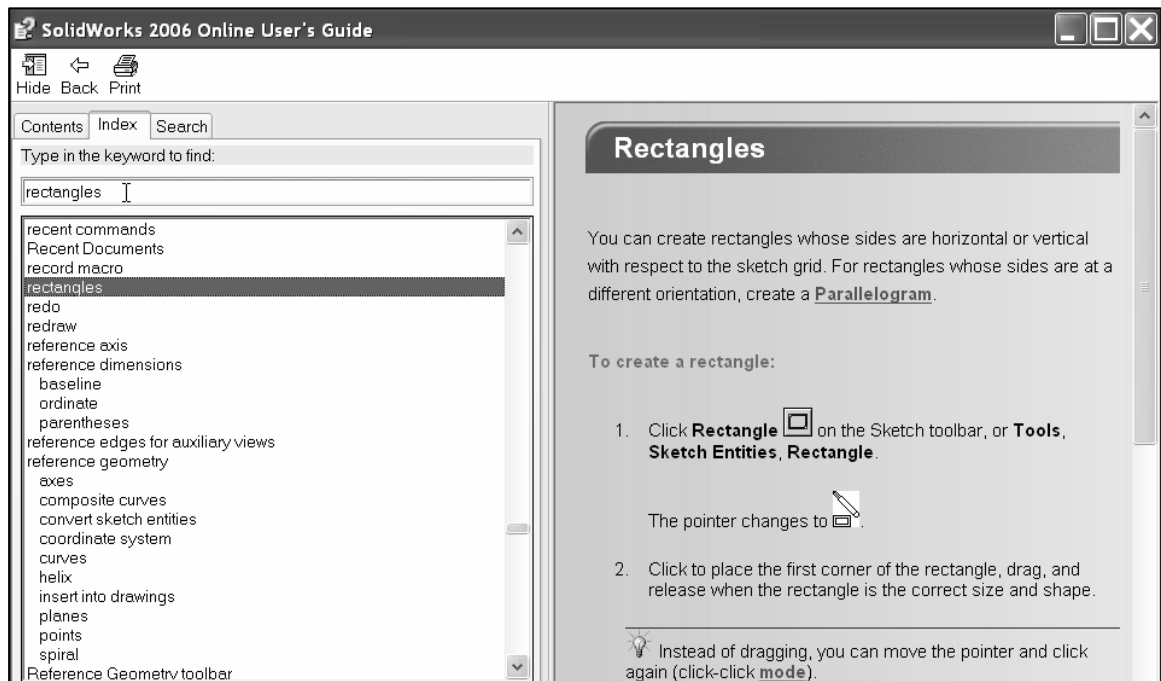
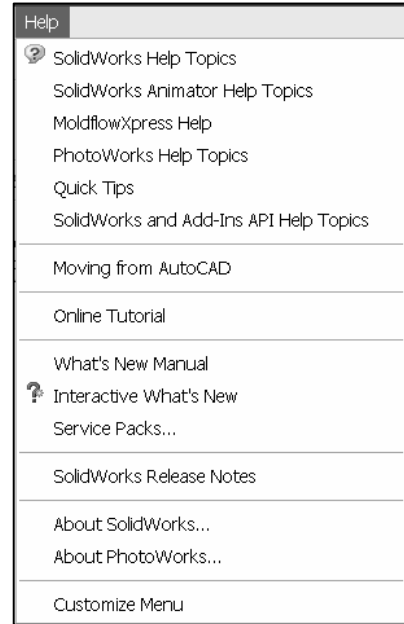
120) Select **SolidWorks Help Topics**

 SolidWorks Help Topics

121) Click the **Index** tab.

122) Enter **rectangle**. The description appears in the right window.

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



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.

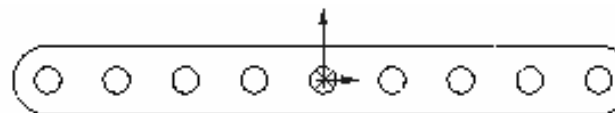
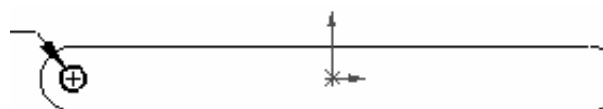
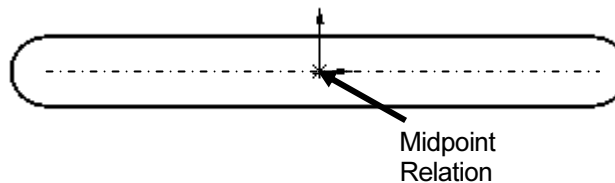
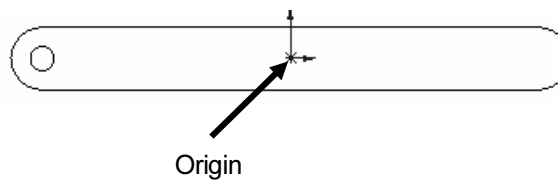
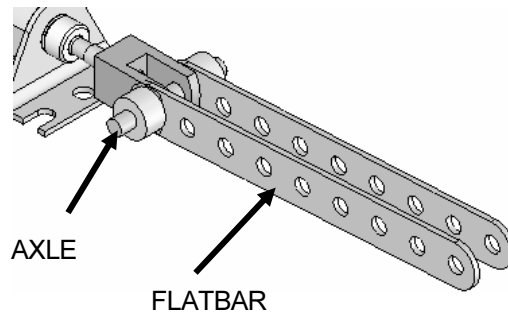
Utilize symmetry. Create the 2D profile centered about the Origin.

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.

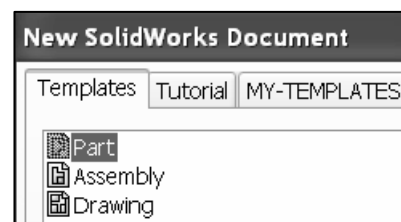
124) Click **File, New**  from the Main menu.

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

Save the part.

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

127) Select the **SW-TUTORIAL-2006** file folder.



128) Enter **FLATBAR** for File name.

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

130) Click **Save**.

Set the dimension standard and part units.

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

132) Select **ANSI** from the Dimensioning Standard list box.

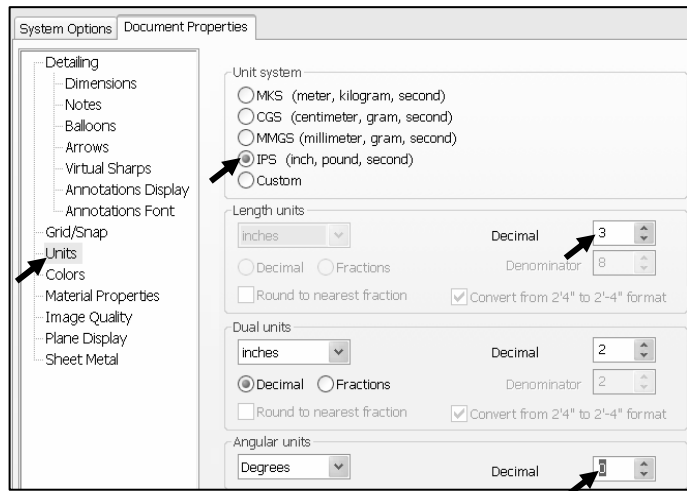
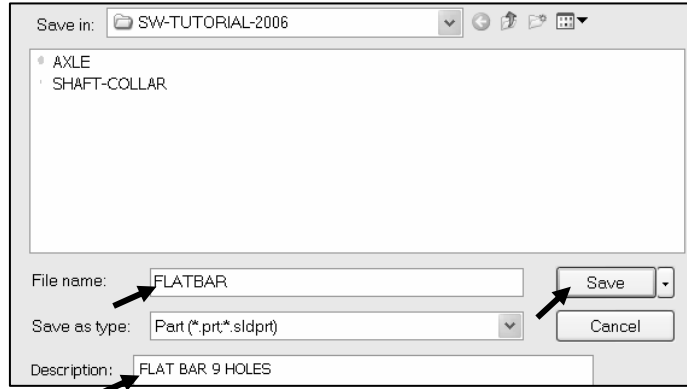
133) Click **Units**.

134) Click **IPS [MMGS]** for Unit System.

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

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

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



Insert a new Sketch for the Extruded Base feature.

138) Click **Front Plane** from the FeatureManager for the Sketch plane.

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

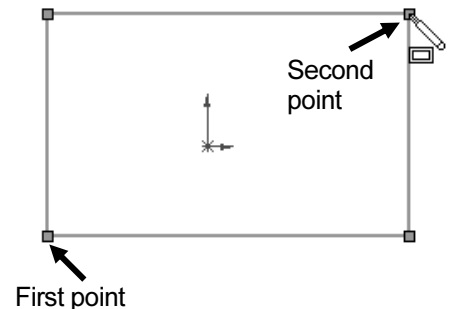
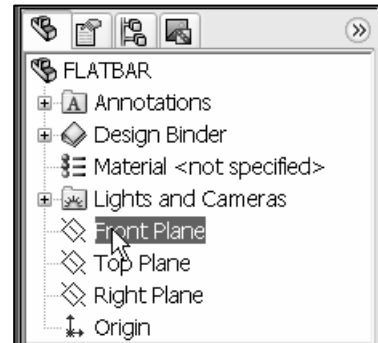
140) Click **Rectangle** from the Sketch toolbar.


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

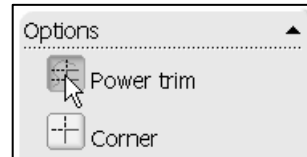
142) Drag the **mouse pointer** up and to the left of the Origin. Release the **mouse button** to create the second point of the rectangle.

Trim the vertical lines.

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



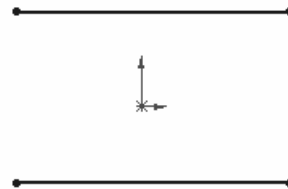
144) Click **Power trim**  Power trim from the Trim PropertyManager.




145) Click a point to the **right** of the right vertical line of the rectangle.


146) Drag the mouse to **intersect** the right vertical line. The line is removed.

147) Click a position to the **left** of the left vertical line of the rectangle. Drag the mouse to **intersect** the left vertical line. The line is removed.



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

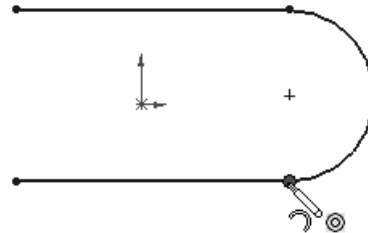
Sketch the right 180 degree Tangent Arc.

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

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

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

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

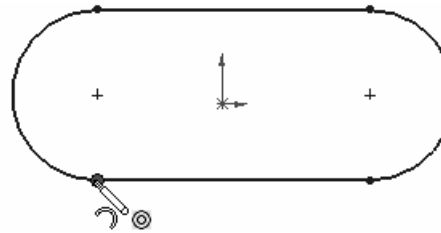


Sketch the left 180 degree Tangent Arc.

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

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

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

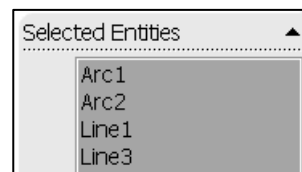
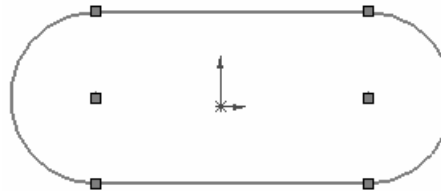


Select Geometry.

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

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


158) Drag the **mouse pointer** to the lower right corner of the Graphics window. Release the **mouse pointer**. The geometry inside the window is selected. The selected geometry is displayed in green.



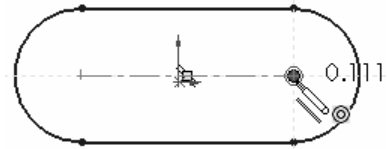
Two arcs and two lines are listed in the Properties Selected Entities box.

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

Sketch a centerline.

159) Click **Centerline**  from the Sketch toolbar.

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

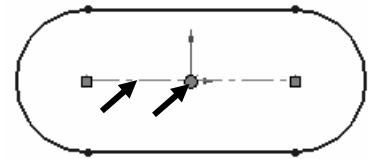


161) Right-click **Select**  Select.

Add a Midpoint relation.

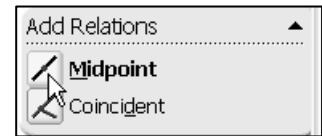
162) Click the **Origin** . Hold the **Ctrl** key down.

163) Click the **Centerline**. Release the **Ctrl** key. The Origin and the Centerline are listed in the Selected Entities box.



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

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



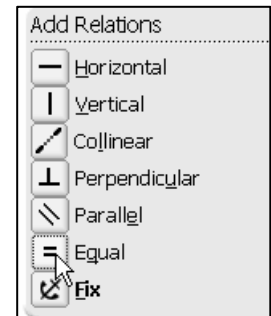
Add an Equal relation.

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

167) Click the **bottom horizontal line**. Release the **Ctrl** key.

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

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



Add a dimension.

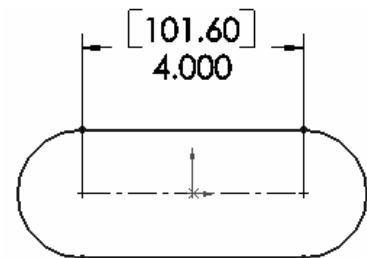
170) Click **Smart Dimension**  **Smart Dimension** .

171) Click the **horizontal centerline**.

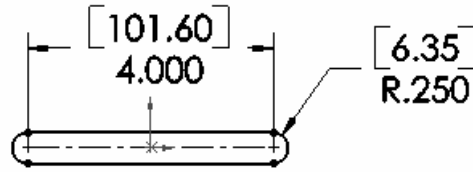
172) Click a **position** above the top horizontal line in the Graphics window.

173) Enter **4.000 [101.60]** in the Modify box.

174) Click the **Green Check mark**  in the Modify pop-up box.

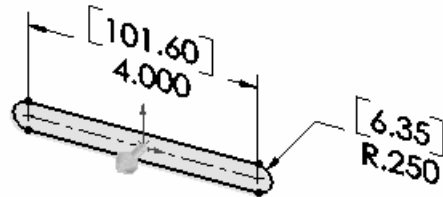



- 175) Click the **right arc** of the FLATBAR.
- 176) Click a **position** diagonally to the right in the Graphics window.
- 177) Enter **.250 [6.35]** in the Modify dialog box.
- 178) Click the **Green Check mark** ✓. The black sketch is fully defined.



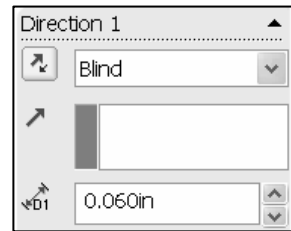
Extrude the sketch to create the first feature.

- 179) Click **Features**  from the Control Area.



- 180) Click **Extruded Boss/Base**  from the Features toolbar.

- 181) Enter **.060 [1.5]** for Depth.



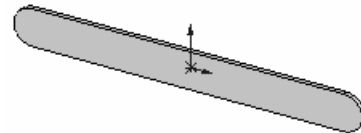
- 182) Click **OK**  from the Extrude PropertyManager.

Fit the model to the Graphics window.

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

Save the FLATBAR part.

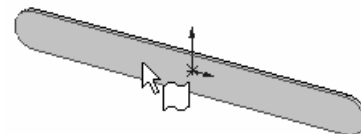
- 184) Click **Save** .



Activity: FLATBAR Part-Extruded Cut Feature

Insert a new sketch for the Extruded-Cut.

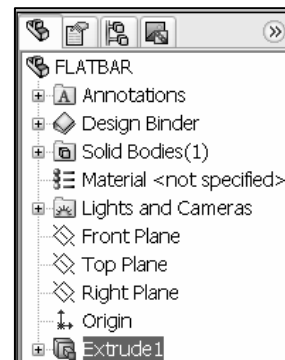
- 185) Click the **front face** of the Extrude1 feature for the Sketch plane.



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

Display the Front view.

- 187) Click **Front view** .



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

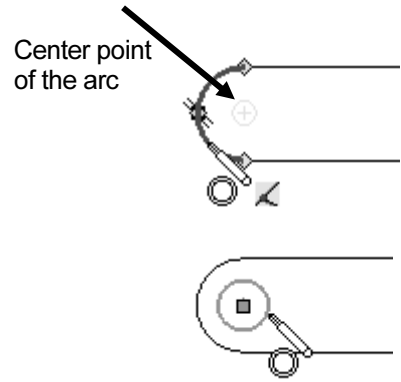
Wake up the center point.

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

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

190) Click the **center point** of the arc.

191) Click a **position** to the right of the center point to create the circle.



Add a dimension.

192) Click **Smart Dimension**  **Smart Dimens....**

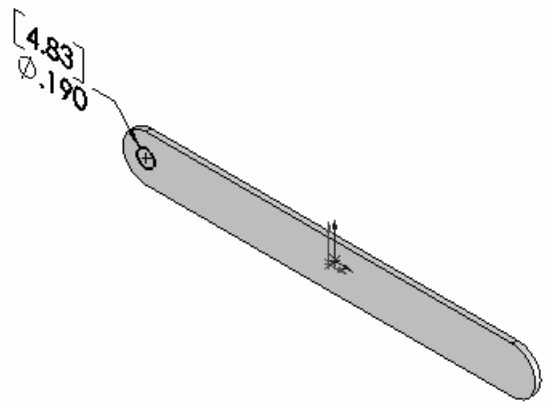
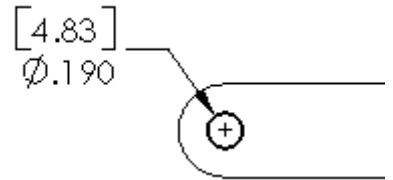
193) Click the **circumference** of the circle.

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

195) Enter **.190 [4.83]** in the Modify box.


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

197) Click **Isometric view** .



Insert an Extruded Cut feature.

198) Click **Features**  **Features** from the Control Area.

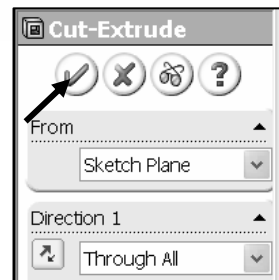
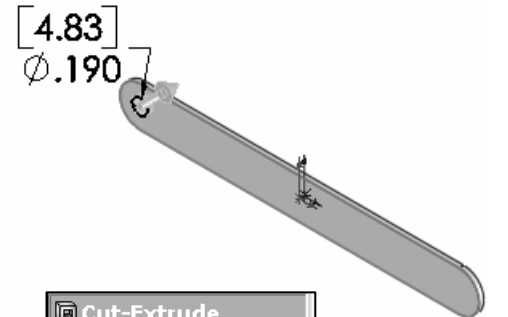
199) Click **Extruded Cut**  **Extruded Cut** from the Features toolbar.

200) Select **Through All** for Direction1 End Condition.


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

Save the FLATBAR part.

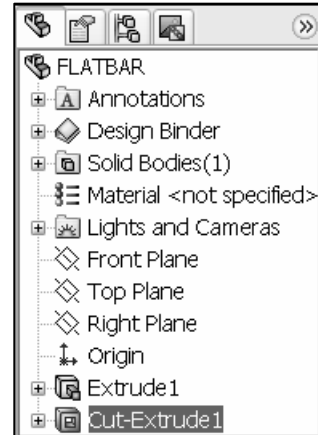
202) Click **Save** .



The Cut-Extrude1 feature is displayed in the FeatureManager.


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

Select Features by clicking their icon in the FeatureManager or selecting geometry in the Graphics window.



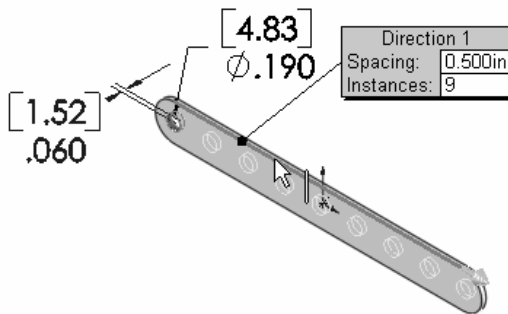
Activity: FLATBAR Part-Linear Pattern Feature

Create a Linear Pattern.

203) Click **Linear Pattern**  from the Features toolbar.


204) Click the **top edge** of the Extrude1 feature for Direction1.

Edge<1> is displayed in the Pattern Direction box for Direction1.



205) Enter **0.5 [12.70]** for Spacing.

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

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

208) Click the **Features to Pattern** box.

209) Expand FLATBAR.

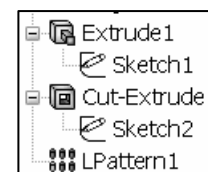
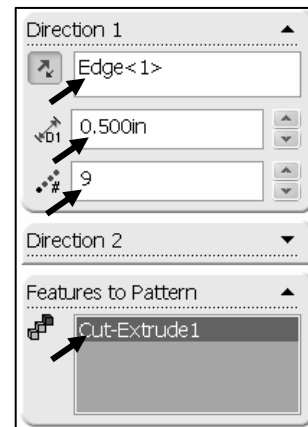
210) Click **Cut-Extrude1** from the FeatureManager.

211) Click **OK**  from the Linear Pattern PropertyManager.

The LPattern1 feature is listed in the FeatureManager.

Save the FLATBAR part.

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



Close all documents.

213) 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 Online Help. Select Help, SolidWorks Help topics. Keywords: Rectangle, Trim, Extruded, Features, 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.
- AIRCYLINDER sub-assembly.

Establishing the correct component relationship in an assembly requires forethought on component interaction.

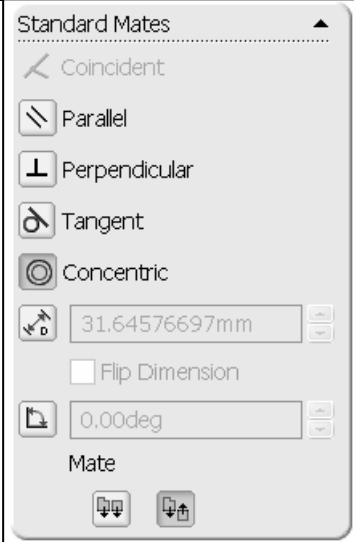
Mates are geometric relationships that align and fit components in an assembly. Mates remove degrees of freedom from a component.

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.

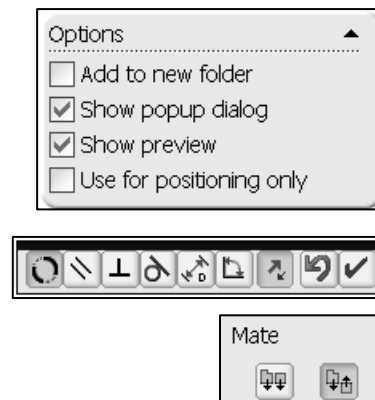
Standard and Advanced Mates:

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

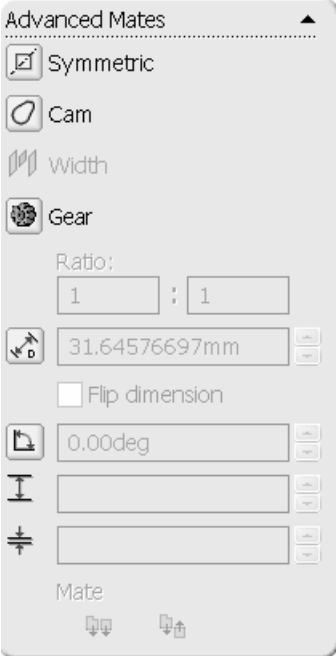
The Mate, Show popup dialog box, displays the Pop-up toolbar during the Mate options. The Standard Mate Types, Aligned/Anti-Aligned, Undo and OK are displayed in the Pop-up toolbar.

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>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>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:		
<p>Cone</p> <ul style="list-style-type: none"> - Circular/Arc Edge - Cone - Point 	<p>Plane</p> <ul style="list-style-type: none"> - Line - Plane - Point - Circular/Arc Edge 	<p>Circular/Arc Edge</p> <ul style="list-style-type: none"> - Cone - Plane - Cylinder - Circular/Arc Edge
<p>Cylinder</p> <ul style="list-style-type: none"> - Line - Point - Circular/Arc Edge 	<p>Point</p> <ul style="list-style-type: none"> - Cam - Cone - Cylinder - Extrusion - Line - Plane - Point - Sphere - Surface 	<p>Surface</p> <ul style="list-style-type: none"> - Point
<p>Extrusion</p> <ul style="list-style-type: none"> - Point 		
<p>Line</p> <ul style="list-style-type: none"> - Cylinder - Line - Plane - Point 	<p>Sphere</p> <ul style="list-style-type: none"> - 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.

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

215) Double-click **Assembly**. The Insert Component PropertyManager is displayed.

216) Click **View**, check **Origins** from the Main menu.

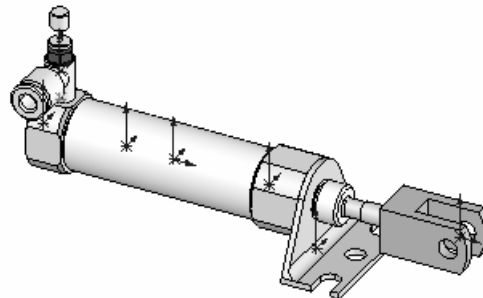
217) Click **Browse** from the Insert Component PropertyManager.

218) Select the **CD pneumatic components** folder.



219) Double-click the **AIRCYLINDER** assembly. The AIRCYLINDER assembly appears in the Graphics window.

220) Click the **Origin** to fix the AIRCYLINDER assembly in the Graphics window.



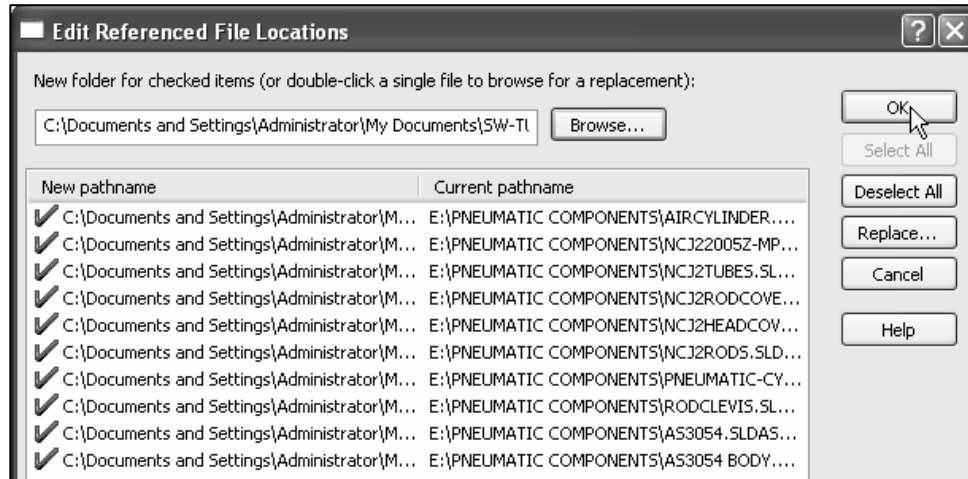
The AIRCYLINDER assembly is displayed in the Graphics window. Copy the AIRCYLINDER assembly with the Save As command. Save all sub-assemblies and part references to the SW-TUTORIAL-2006 folder.

Save the AIRCYLINDER assembly to the SW-TUTORIAL-2006 folder.

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

222) Select **SW-TUTORIAL-2006** for Save in folder. Click the **References** button. Click the **Select All** button to check all components contained in the AIRCYLINDER assembly. Click the **Browse** button from the Edit Referenced File Locations.

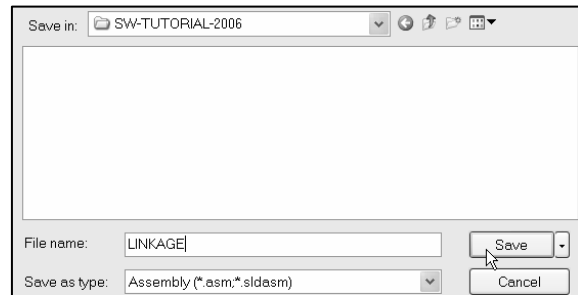
223) Select the **SW-TUTORIAL-2006** folder. Click **OK** from the Browse for Folder dialog box.



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

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

226) Click **Save**. Click **YES**.

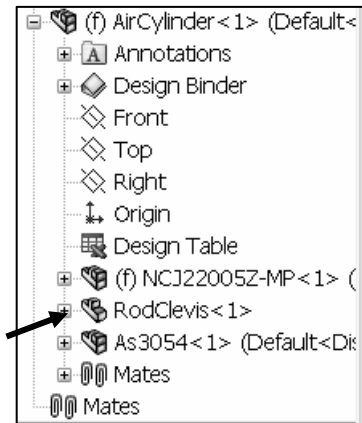


The AIRCYLINDER assembly and its references are copied to the SW-TUTORIAL-2006 folder. Assemble the AXLE to the holes in the RodClevvis.

Display the RodClevvis.

227) Click the **Plus** icon to expand the AIRCYLINDER assembly.

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



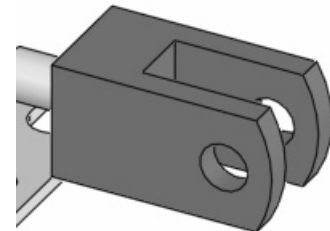
Hide the Origins.

229) 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. The (f) symbol is placed in front of the AIRCYLINDER name in the FeatureManager.

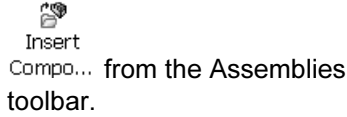
Display the Isometric view.

230) Click **Isometric view** icon.



Insert the AXLE part.

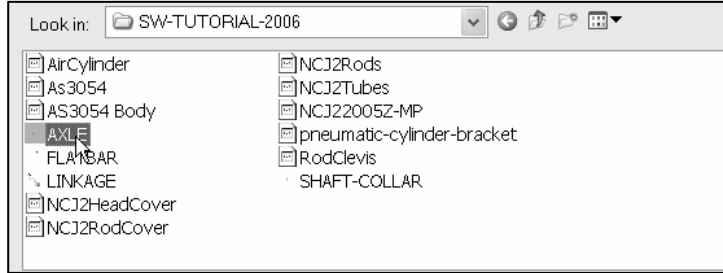
231) Click **Insert Component**



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

233) Select **All Files** from the Files of type list.

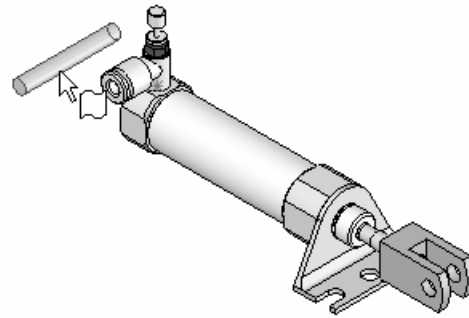
234) Double-click **AXLE** from the SW-TUTORIAL-2006 folder.



235) Click a **position** to the left of the AIRCYLINDER assembly.

Move the AXLE component.

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



Enlarge the view.


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

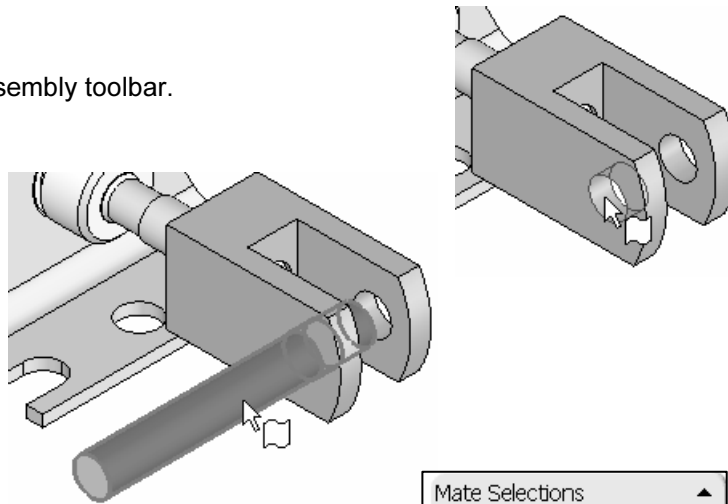
Insert a Concentric Mate.


238) Click **Mate**  from the Assembly toolbar.

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

240) Click the **long cylindrical face** of the AXLE. The cursor displays the Face feedback symbol. The faces are displayed in the Mate Selections box. The

Concentric Mate  type is selected by default. The AXLE is positioned concentric to the RodClevis hole.



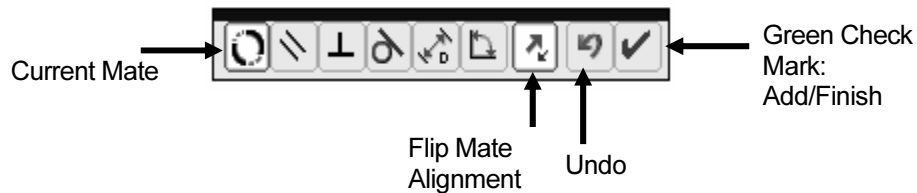
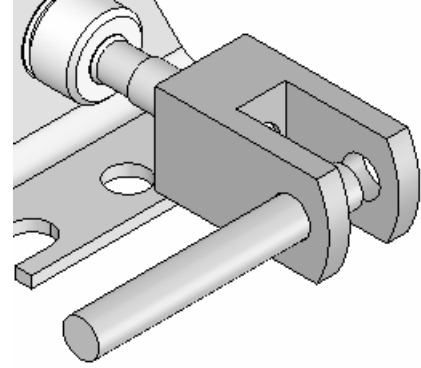
241) Click the **Green Check mark** .




Move the AXLE.

242) Click and drag the **AXLE** left to right. The AXLE translates in and out of the RodClevis 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.



243) Click **Top view** .

Expand the LINKAGE assembly in the Graphics window.

244) Click the **Plus**  icon in front of the LINKAGE assembly
 LINKAGE (Default<Dis... Click the **Plus**  icon in front of the AIRCYLINDER assembly  (f) AirCylinder<1> (...)



Expand the AXLE.

245) Click the **Plus**  icon in front of the AXLE part
 (-) AXLE<1>.

Clear all sections from the Mate Selections box.

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

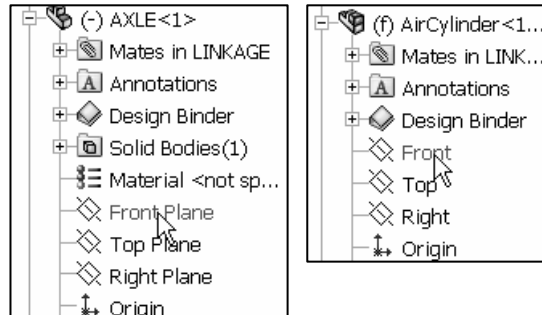


Insert a Coincident Mate.

247) Click the **Front Plane** of the AIRCYLINDER assembly from the FeatureManager.

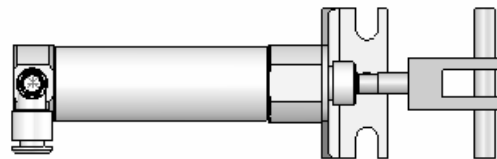
248) Click the **Front Plane** of the AXLE part from the FeatureManager.

249) Click the **Green Check mark** ✓ in the Mate pop-up box.




250) Click **OK** ✓ from the Mate PropertyManager.

The Coincident Mate type is selected by default. The AIRCYLINDER Front Plane and the AXLE Front Plane are Coincident. The AXLE is centered in the RodClevis.



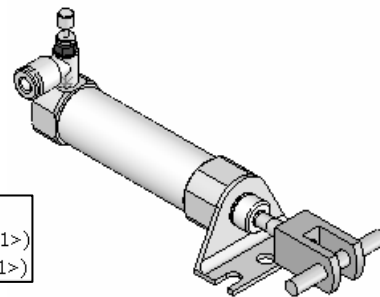
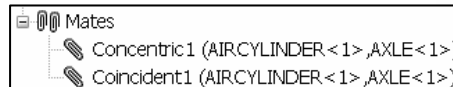
Display the Mates in the FeatureManager to check that the components and the Mate Types correspond to the design intent. Note: If you delete a Mate and then recreate it, the Mate numbers will be in a different order.

Display the Isometric view.

251) Click **Isometric view** .

Display the Mates.

252) Click the **Plus sign** + of the Mates in the FeatureManager.



Save the LINKAGE assembly.

253) Click **Save** .

Activity: LINKAGE Assembly-Insert FLATBAR Part

Insert the FLATBAR part.



254) Click **Insert Component** Compo... from the Assemblies toolbar.

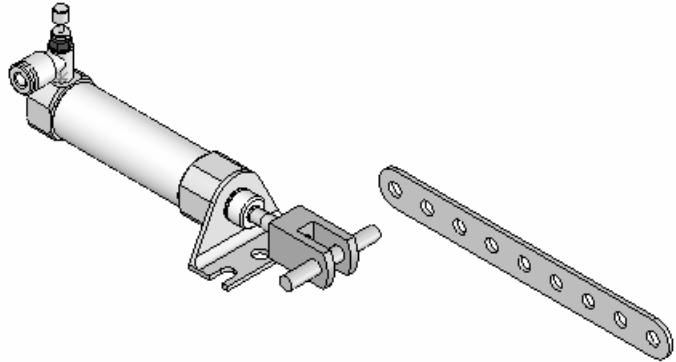
255) Click **Browse** from the Insert Component PropertyManager.

256) Select **Part** for Files of Type from the SW-TUTORIAL-2006 folder.


257) Double-click **FLATBAR**.

Place the Component.


258) Click a **position** behind the left hole of the AXLE.



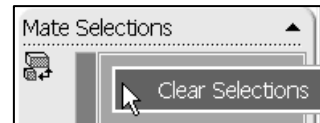
Enlarge the view.

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

Insert a Concentric Mate.

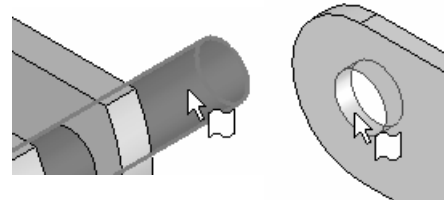
260) Click **Mate**  from the Assembly toolbar.

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

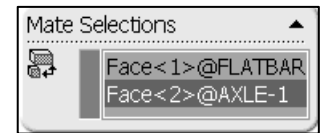


262) Click the inside **left hole face** of the FLATBAR.

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



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



Fit the model to the Graphics window.

265) Press the **f** key.



Move the FLATBAR.

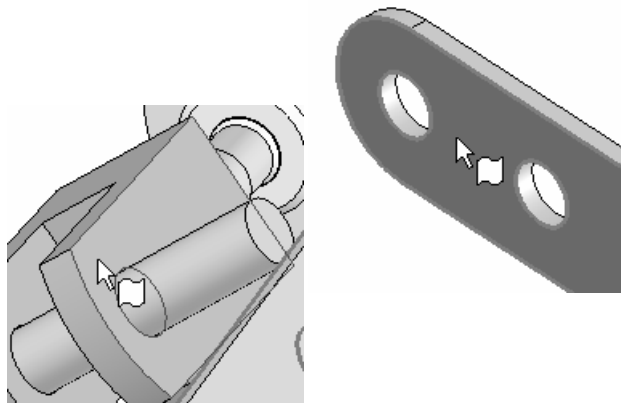
266) Click and drag the **FLATBAR**. The FLATBAR translates and rotates along the AXLE.

Insert a Coincident Mate.

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


268) Press the **left arrow key** 5 times to rotate the model.

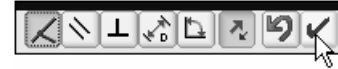
269) Click the **back face** of the RodClevis.



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

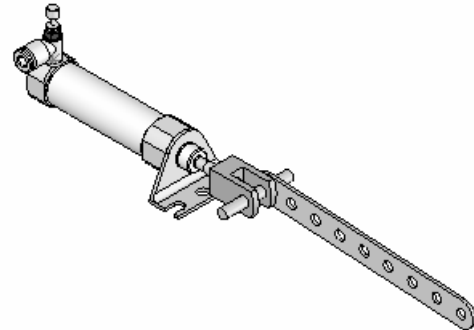


271) Click **OK**  from the Mate PropertyManager.




Display the Isometric view.

272) Click **Isometric view** .



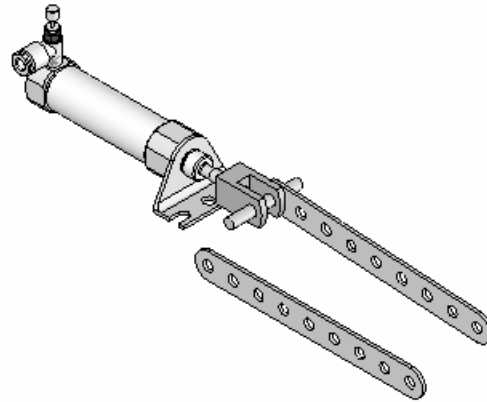
Insert the second FLATBAR.

273) Click **Insert Component**  from the Assemblies toolbar.

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


275) Select **Part** for Files of Type from the SW-TUTORIAL-2006 folder.

276) Double-click **FLATBAR**.




277) Click a **position** to the left of the AIRCYLINDER.

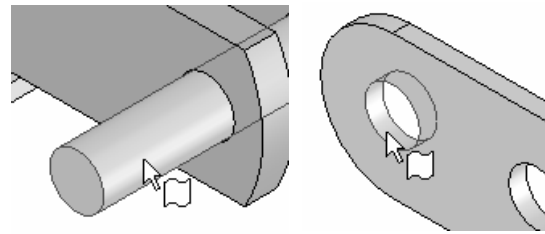
Enlarge the view.

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

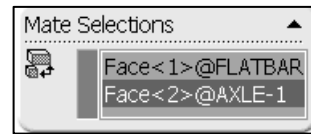
Insert a Concentric Mate.

279) Click **Mate**  from the Assembly toolbar.

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



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



282) Click the **Green Check mark** ✓ from the Mate pop-up box to create the Concentric Mate. Press the **f** key to fit the model to the Graphics window.



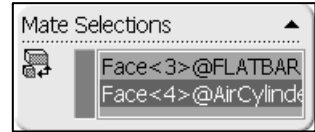
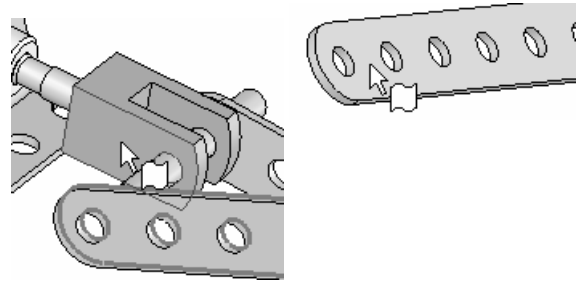
Insert a Coincident Mate.

283) Press the **left arrow key** approximately 5 times to rotate the model.

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

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

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



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

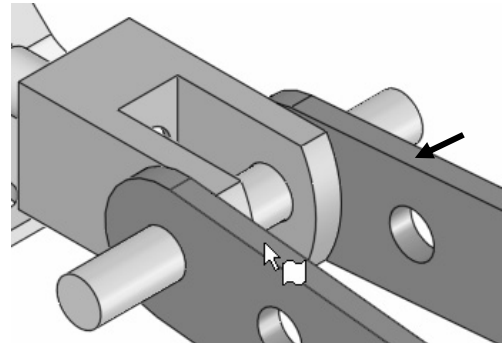
Create a Parallel Mate.

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

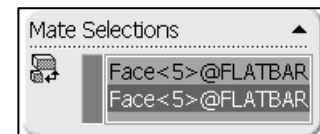
289) Click the **top narrow face** of the first FLATBAR.

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

291) Click **Parallel** .



292) Click the **Green Check mark** ✓ to create the Parallel Mate.



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

294) Click **Isometric view** .


Move the two FLATBAR parts.

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

Activity: LINKAGE Assembly-Insert SHAFT-COLLAR Part

Insert the first SHAFT-COLLAR.



296) Click **Insert Component** , from the Assemblies toolbar.


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

298) Select **Part** for Files of Type from the SW-TUTORIAL-2006 folder.

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

300) Click a **position** to the right of the AXLE.


Enlarge the view.

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

Save the LINKAGE assembly.


302) Click **Save** .

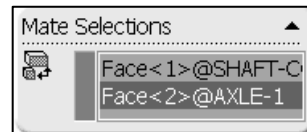
Insert a Concentric Mate.

303) Click **Mate**  from the Assembly toolbar.

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

305) Click the **long cylindrical face** of the AXLE. The Concentric Mate type is selected by default.

306) Click the **Green Check mark**  to create the Concentric Mate.



Insert a Coincident Mate.


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

308) Click the **front face** of the SHAFT-COLLAR.

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

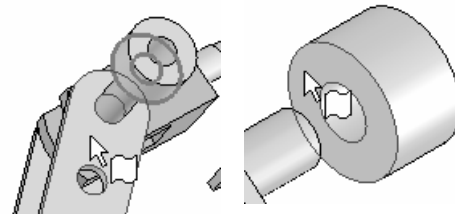
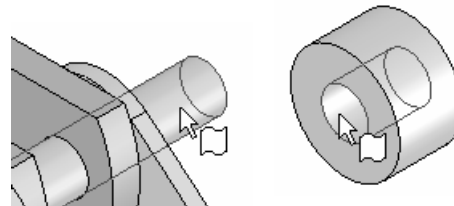
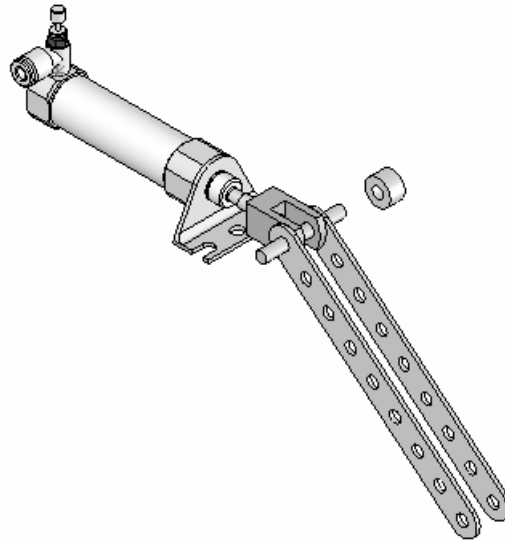
310) Click the **back face** of the first FLATBAR.

311) Click the **Green Check mark**  to create the Coincident Mate.

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

Display the Isometric view.

313) Click **Isometric view** .



Insert the second SHAFT-COLLAR



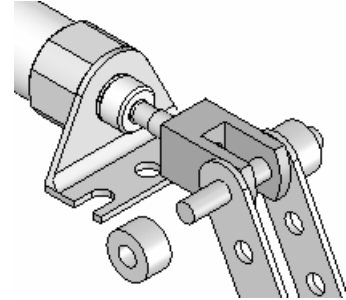
314) Click **Insert Component** from the Assemblies toolbar.

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

316) Select **Part** for Files of Type from the SW-TUTORIAL-2006 folder.

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

318) Click a **position** near the AXLE.



Enlarge the view.

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

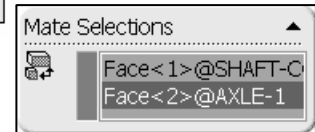
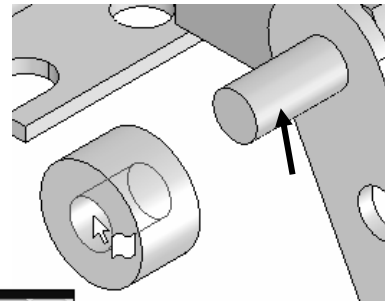
Insert a Concentric Mate.

320) Click **Mate** from the Assembly toolbar.

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

322) Click the **long cylindrical face** of the AXLE. The Concentric Mate type is selected by default.

323) Click the **Green Check mark** to create the Concentric Mate.



Insert a Coincident Mate.

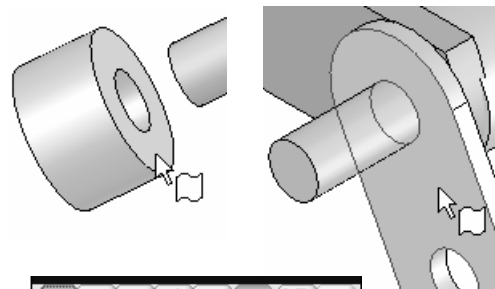
324) Press the **f** key to fit the model to the Graphics window.

325) Use the **Shift-z** keys to Zoom in on the front face of the second FLATBAR.

326) Click the **front face** of the second FLATBAR.

327) Press the **left arrow key** approximately 5 times to rotate the model to view the back face of the second SHAFT-COLLAR.

328) Click the **back face** of the second SHAFT-COLLAR. Click the **Green Check mark** to create the Coincident Mate.



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




Display the Isometric view.

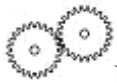
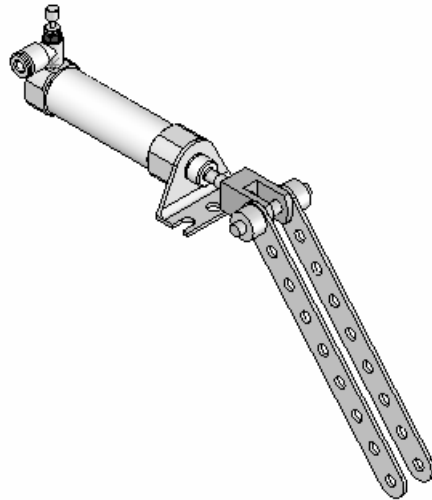
330) Click **Isometric view** .

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-2006 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.

Activity: LINKAGE Assembly-Physical Simulation

Insert a Rotary Motor Physical Simulation Tool.

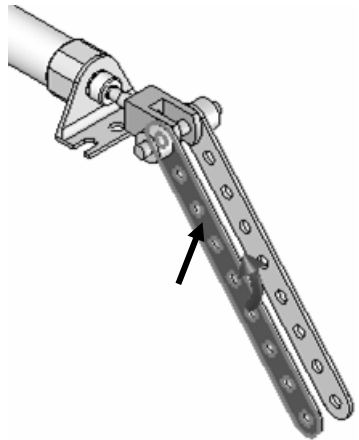
333) Click **View, Toolbars, Simulation** from the Main menu.

334) Click **Rotary Motor**  from the Simulation toolbar.

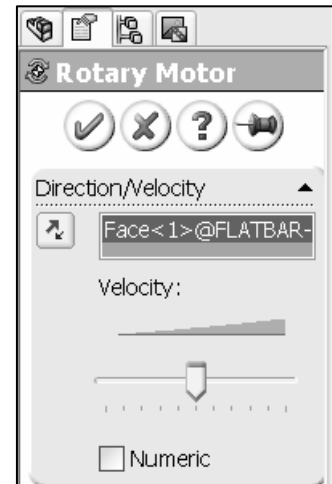



Rotary Motor
Moves a component as if acted upon by a rotary motor.

335) Click the **FLATBAR front face**. 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.

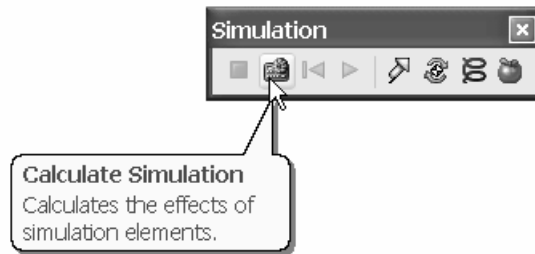



337) Click **OK**  from the Rotary Motor PropertyManager. Note: Recording the simulation to an avi. file requires the SW Animator application. Select Tools, Add-Ins, SolidWorks Animator from the Main toolbar if required.


Record the Simulation.

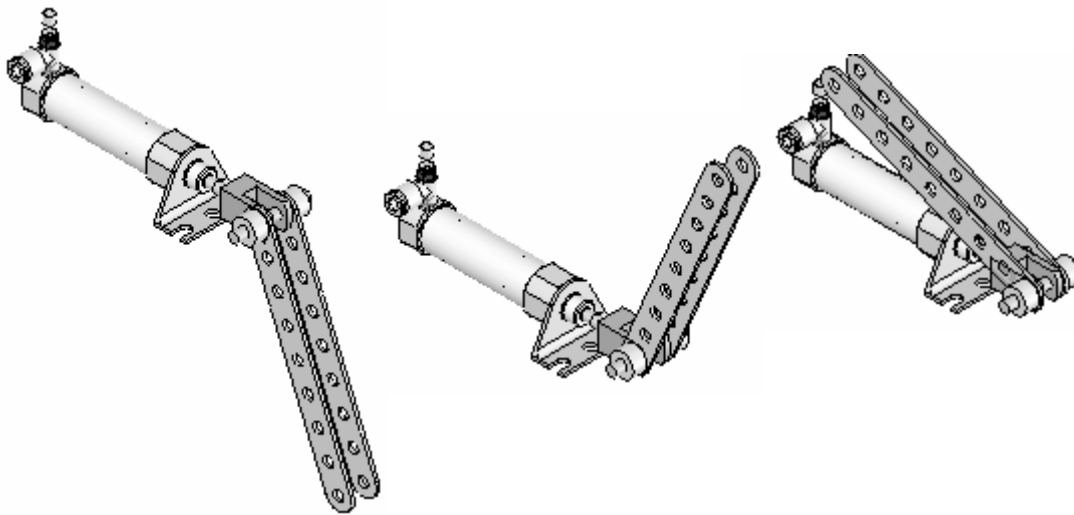
338) Click **Calculate Simulation**  .

339) Click **OK** to the warning message, "Model started in colliding position". The simulation continues and ignores the collisions. The FLATBAR rotates in a counterclockwise direction.



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


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




Linear Assembly Physical Simulation



Stop the Simulation.

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

343) Save the simulation in an AVI file to the SW-TUTORIAL-2006 folder. Click **Save** . Note: SW Animator is required to save the simulation in an .AVI file.

344) Accept the default file name. Click **Save**.

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

Close the Simulation.

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

Fit the assembly to the Graphics window.

347) Press the **f** key.

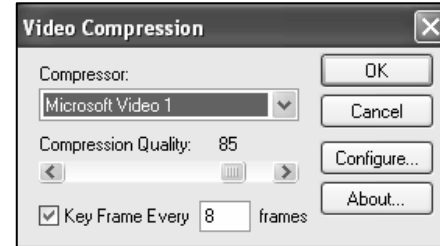
Save the LINKAGE assembly.

348) Click **Save** .

Exit SolidWorks.

349) 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 Online Help. Select Help, SolidWorks Help topics. Keywords: Standard Mates, Mate PropertyManager, Design Methods in Assembly, 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, downloaded the AIRCYLINDER assembly and created the LINKAGE assembly.

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

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 Online Help for additional information about the terms utilized in this project.

Assembly: An assembly is a document in which parts, features, and other assemblies (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, choose 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 until 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 Base/Boss: Use to add material by extrusions. The Extruded is the first feature in a part. An Extruded Boss occurs after the first feature.

Steps to create an Extruded Base/Boss:

- Select the Sketch plane.
- Sketch the profile.
- Add Dimensions and Relations.
- Select the Extruded Boss/Base from the Features toolbar.
- Enter Depth, select end conditions and or options.

Extruded Cut: Use to remove material from a solid. This is the opposite of the boss. Cuts begin as a 2D sketch and remove materials by extrusions.

- Steps to create an Extruded Cut:
- Select the Sketch plane.
- Sketch the profile.
- Add Dimensions and Relations.

- Select Extruded Cut from the Features toolbar.
- Enter Depth, select end conditions and or options.

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

Steps to create a Linear Pattern:

- 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.

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.

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

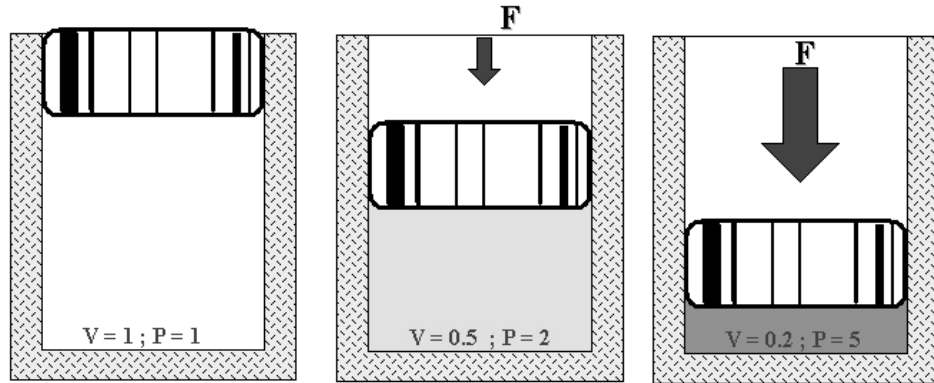


Illustration of Boyle’s Law
 Courtesy of SMC Corporation of America

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

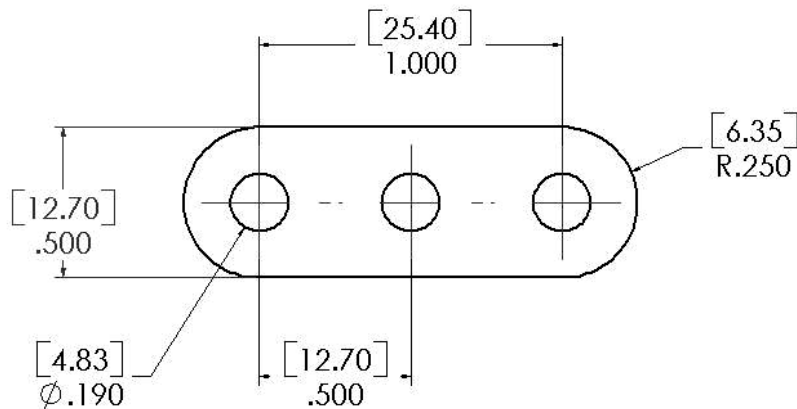
1. Explain the steps in starting a SolidWorks session.
2. Describe the procedure to begin a new sketch.
3. Explain the steps required to change part unit dimensions from inches to millimeters.
4. Describe a part.
5. Identify the three default reference planes.
6. What is the Base feature? Provide an 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 Introduction. 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.

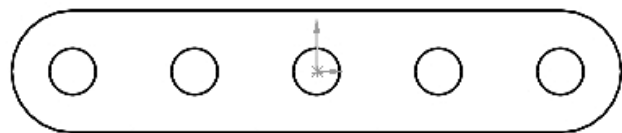


FLATBAR-3HOLE

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:



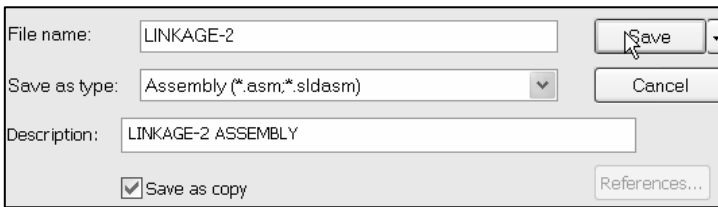
FLATBAR-5HOLE

- Holes are .500in on center.
- Radius is .250in.
- Hole diameter is .190in.

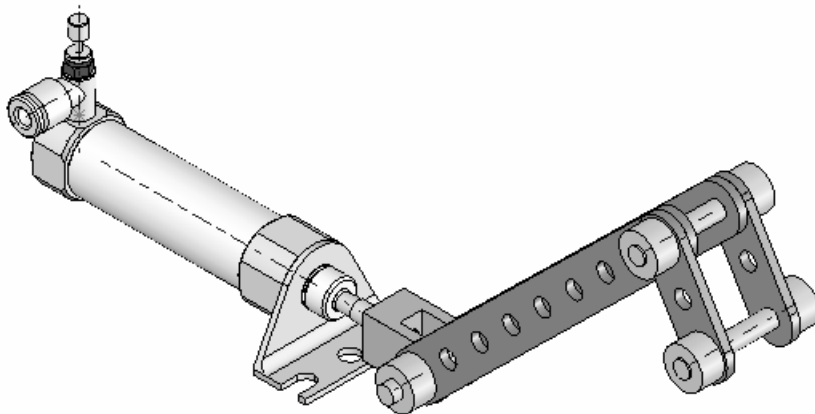
Exercise 1.3a: LINKAGE-2 Assembly.

Create the LINKAGE-2 assembly.

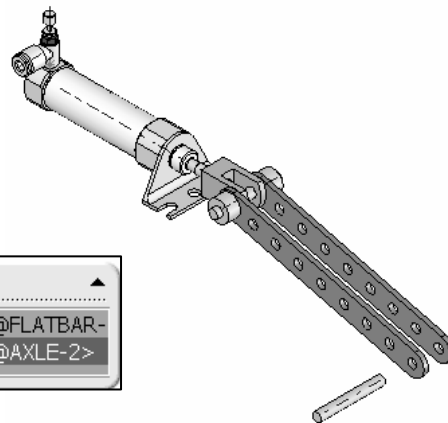
- Open the LINKAGE assembly.
- Select File, Save As from the Main menu.
- Check the Save as Copy check box.
- Enter LINKAGE-2 for file name. LINKAGE-2 ASSEMBLY for Description.



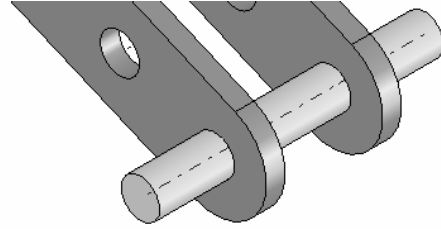
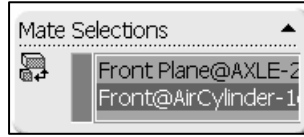
The FLATBAR-3HOLE part was created in Exercise 1.1. Utilize 2, AXLE parts, 4, SHAFT COLLAR parts and 2, FLATBAR-3HOLE parts to create the LINKAGE-2 assembly.



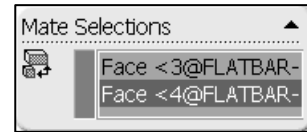
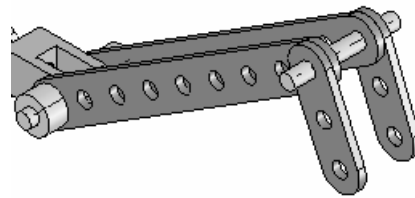
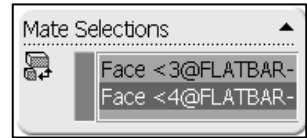
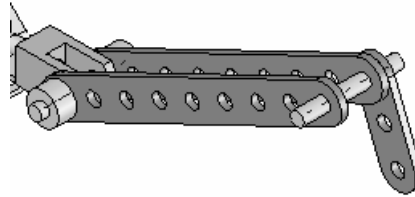
- Insert the first AXLE part.
- Insert a Concentric Mate.



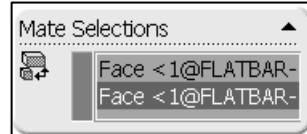
Insert a Coincident Mate.



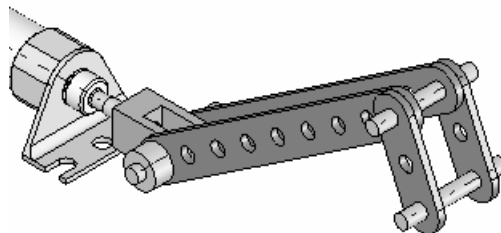
- Insert the first FLATBAR-3HOLE part.
- Insert a Concentric Mate.
- Insert a Coincident Mate. Perform the same procedure for the second FLATBAR-3HOLE part.



- 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.



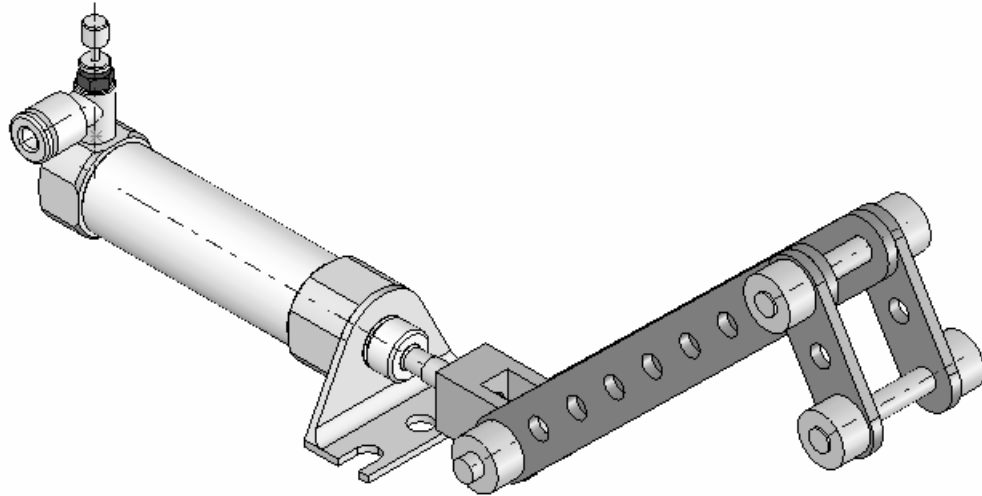
- Insert a Coincident Mate.



- 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.



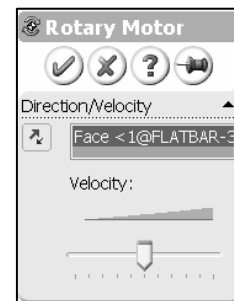
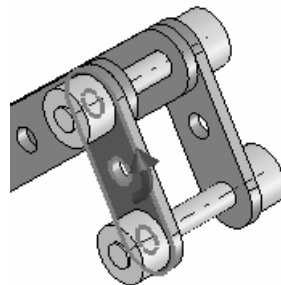
Exercise 1.3b: LINKAGE-2 Assembly Simulation.

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.



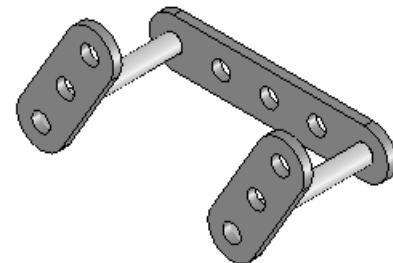
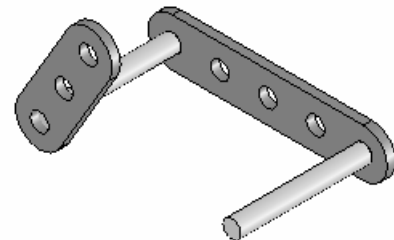
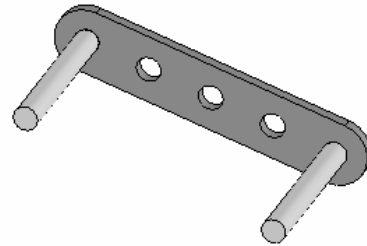
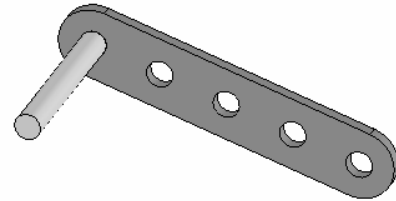
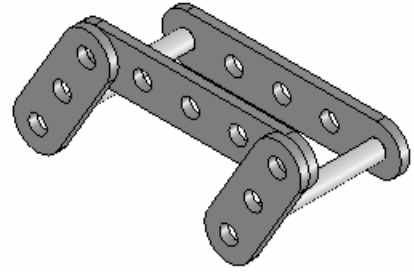
Exercise 1.4a: ROCKER Assembly.

Create a ROCKER assembly. The ROCKER assembly consists of 2 AXLE parts, 2 FLATBAR-5HOLE parts, and 2 FLATBAR-3HOLE parts.

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

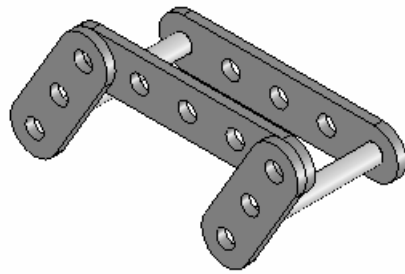
The three parts rotate clockwise and counterclockwise, above the top plane. Create the ROCKER assembly.

- 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 Mate.

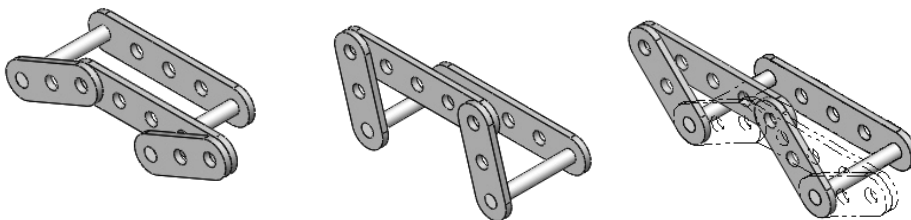
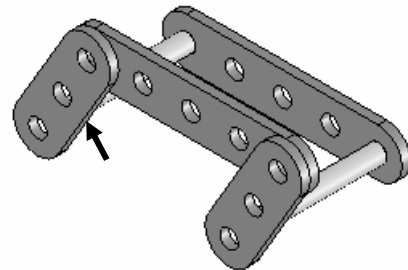
Note: The end holes of the second FLATBAR-5HOLE are concentric with the end holes of the FLATBAR-3HOLE parts.



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

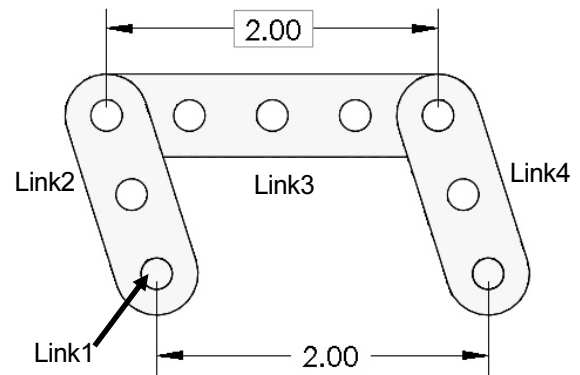
In mechanical design, the ROCKER assembly is classified as a mechanism. A Four-Bar Linkage is a common mechanism comprised of four links.

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.

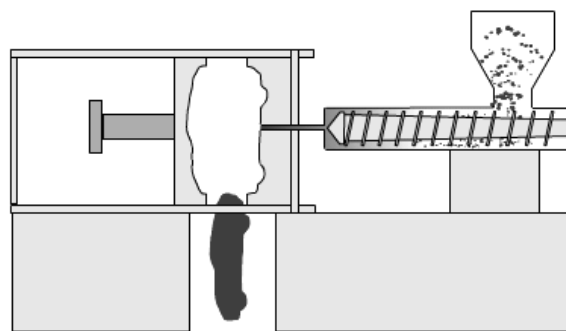
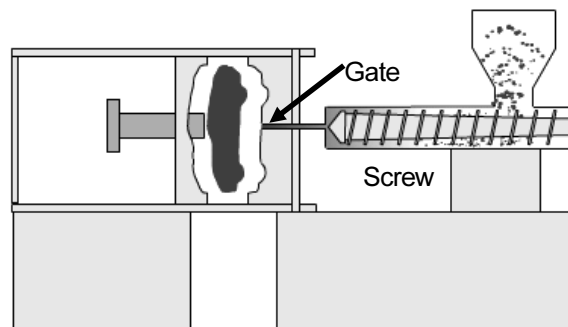
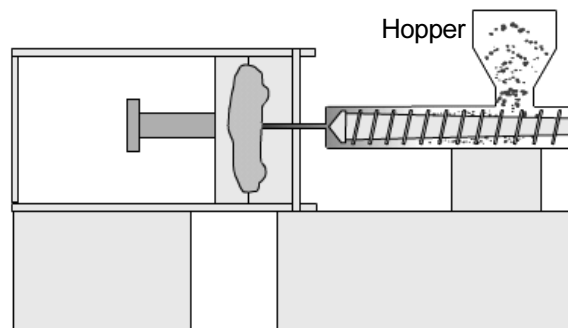
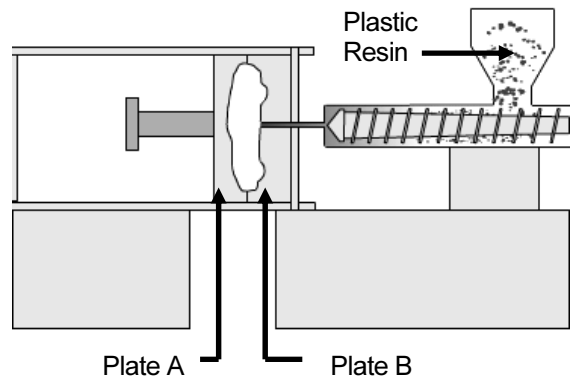
The injection molding process is as follows:

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 web sites mikejwilson.com and zxys.com.

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



Model Courtesy of
Mike J. Wilson, CSWP

Scooby Doo® is a registered
trademark of Hanna-Barbera

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.



Notes: