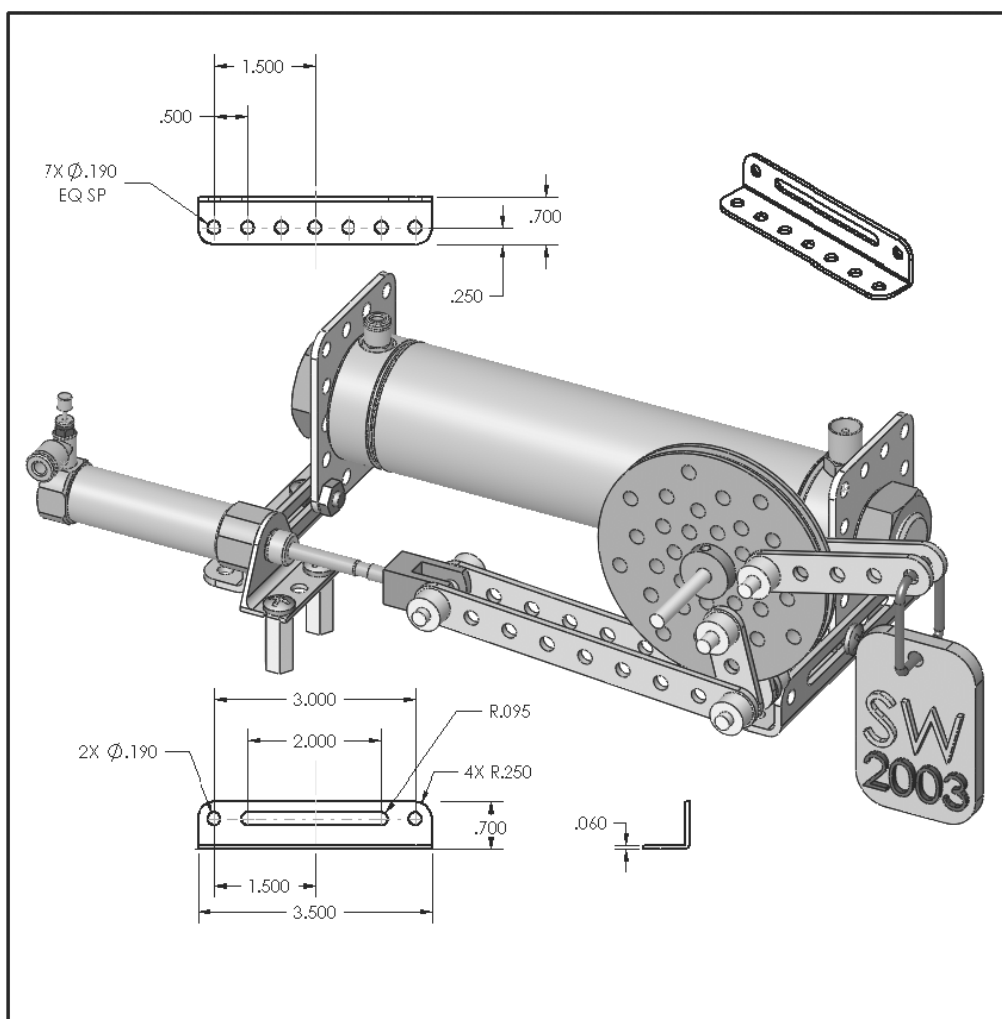




# SolidWorks 2003 Tutorial

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

David C. Planchard & Marie P. Planchard

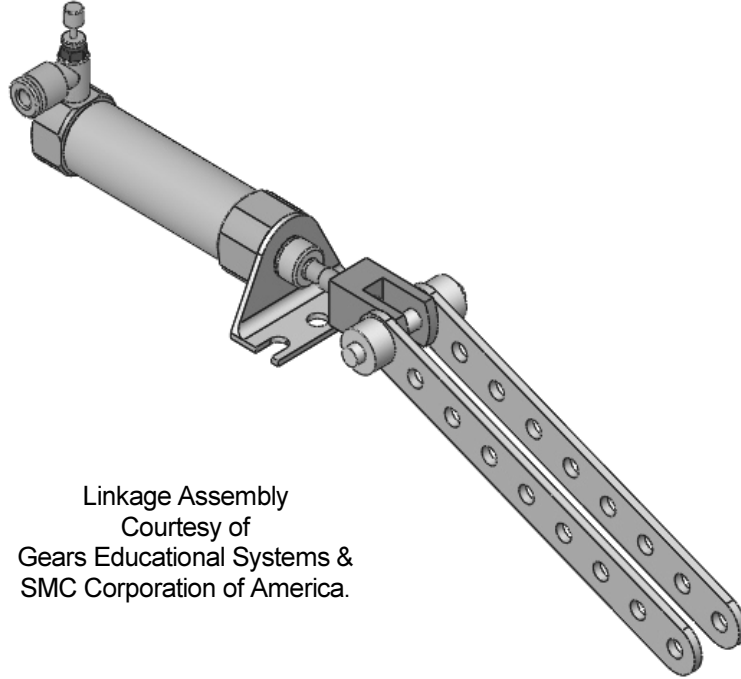


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# Project 1

## Linkage Assembly

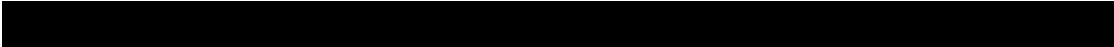


Linkage Assembly  
Courtesy of  
Gears Educational Systems &  
SMC Corporation of America.

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

Desired Outcomes:	Usage Competencies:
Three parts: 1. AXLE. 2. SHAFT-COLLAR. 3. FLATBAR.	Establish a SolidWorks session.  Develop new parts.  Model new features: Extruded-Base, Extruded-Cut, and Linear Pattern.
One assembly: LINKAGE Assembly.	Understand the assembly process including Concentric, Parallel and Coincident Mates.

**Notes:**



## Project 1 – Extrude Features

### Project Objective

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

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

Create three individual parts:

1. AXLE.
2. SHAFT-COLLAR.
3. FLATBAR.

Create a LINKAGE Assembly.

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

- Establish a SolidWorks session.
- Set Units and the Dimensioning Standard.
- Create a part.
- Generate a sketch.
- Add Dimensions.
- Add Geometric Relations.
- Download an assembly.
- Create an assembly.
- Use the following SolidWorks Features:
  - Extruded-Base.
  - Extruded-Cut.
  - Linear Pattern.

**Project Overview**

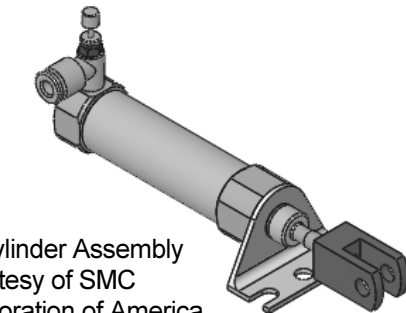
SolidWorks is a design automation software package used to produce and model parts, assemblies and drawings.

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:

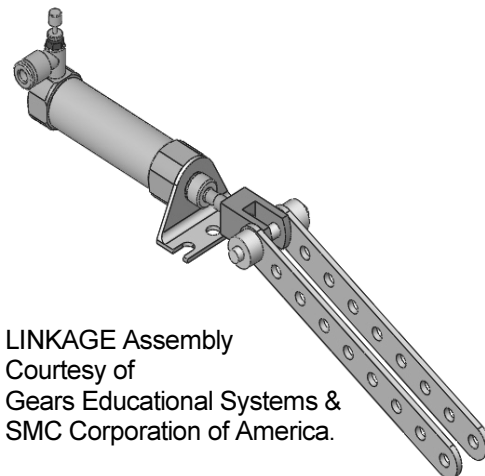
1. AXLE.
2. SHAFT-COLLAR.
3. FLATBAR.

Download the AirCylinder Assembly from the World Wide Web.

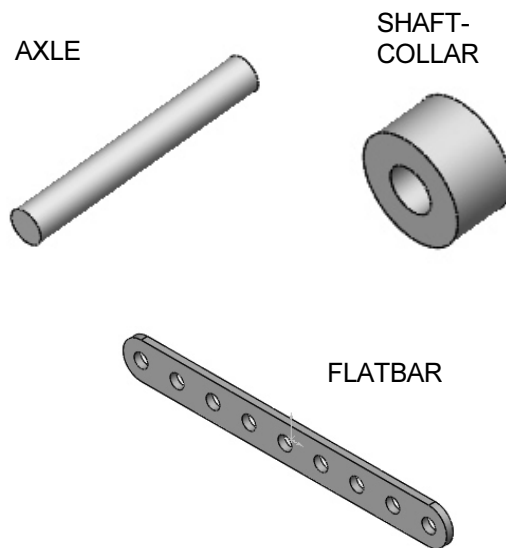


AirCylinder Assembly  
Courtesy of SMC  
Corporation of America

Combine the parts and the AirCylinder Assembly to create the LINKAGE Assembly.

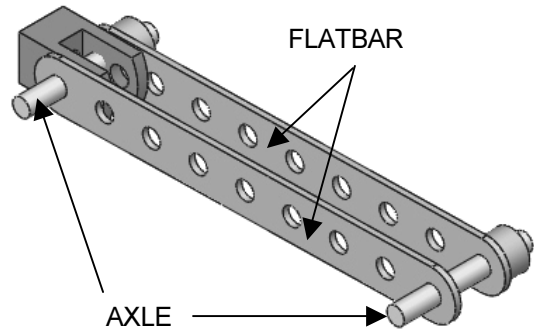


LINKAGE Assembly  
Courtesy of  
Gears Educational Systems &  
SMC Corporation of America.

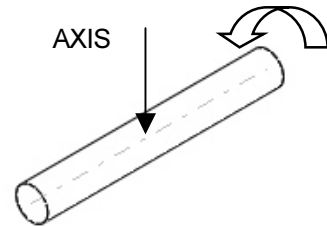


**AXLE Part**

The AXLE is a cylindrical steel rod. Two AXLES support the two FLATBAR parts.



The AXLE rotates about its axis.

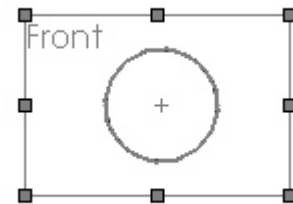


Start a new SolidWorks session. Create the AXLE part.

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

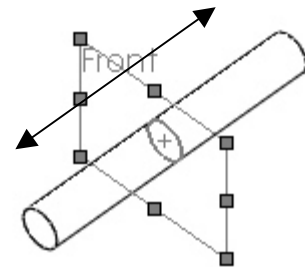
Utilize the Extruded-Base feature. The Extrude Base features add material. The Base feature is the first feature of the part.

Sketch a circular profile on the Front plane, centered at the Origin.

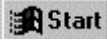

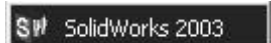


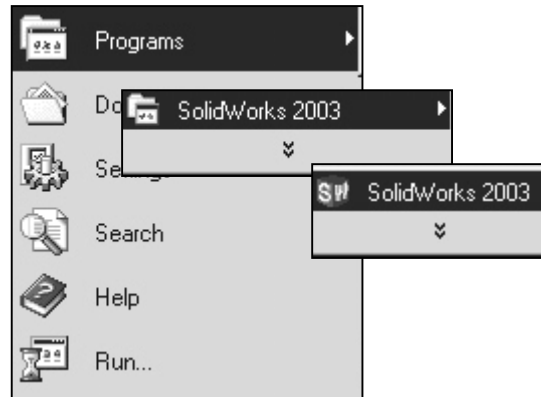
Extend the profile perpendicular ( $\perp$ ) to the Front plane.

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



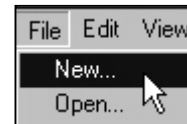
Create a SolidWorks session.

- 1) Click the **Start** button on the Windows Taskbar, .
- 2) Click **Programs**.
- 3) Click the **SolidWorks 2003** folder. .
- 4) Click the **SolidWorks 2003** application. .
- 5) The SolidWorks program window opens. Read the Tip of the Day dialog box. Close the dialog box. Click **Close**.

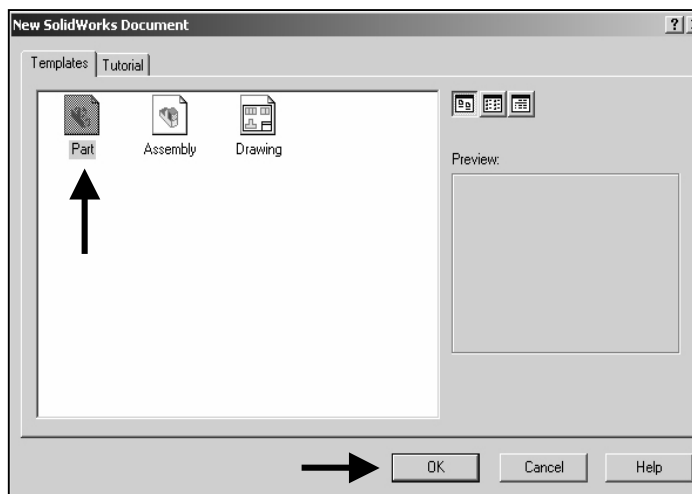


Create a New Part.

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

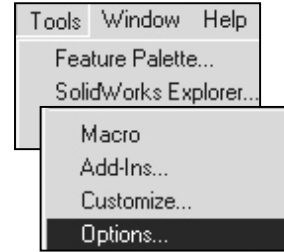


- 8) The Templates tab is the default tab. Part is the default template from the New SolidWorks Document dialog box. Accept the default template. Click **OK**.
- 9) Part1 is displayed. Part1 is the new default part window name.



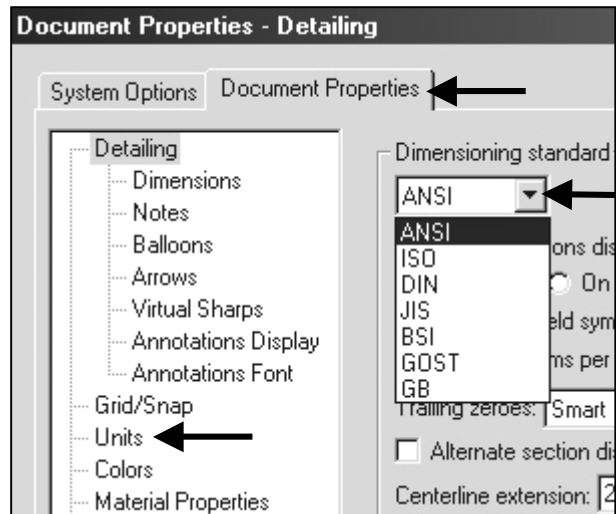
Set the Dimensioning Standard.

10) Click **Tools** from the Main menu. Click **Options**.



11) Click the **Document Properties** Tab.

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

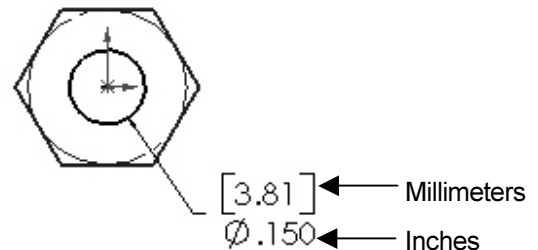


The Dimensioning Standard determines the display of dimension text, arrows, symbols and spacing.

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 inches. The optional secondary units are provided in millimeters and are indicated in brackets [ ].

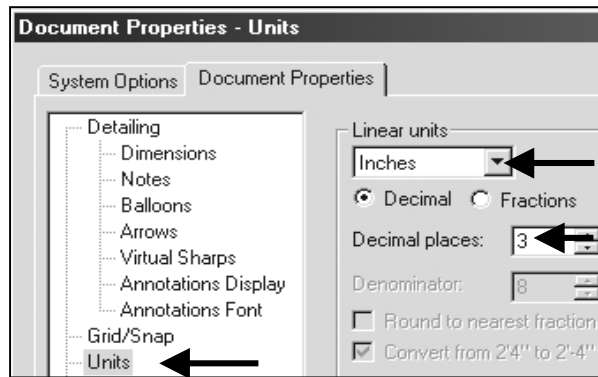
Illustrations are provided in inches and millimeters.








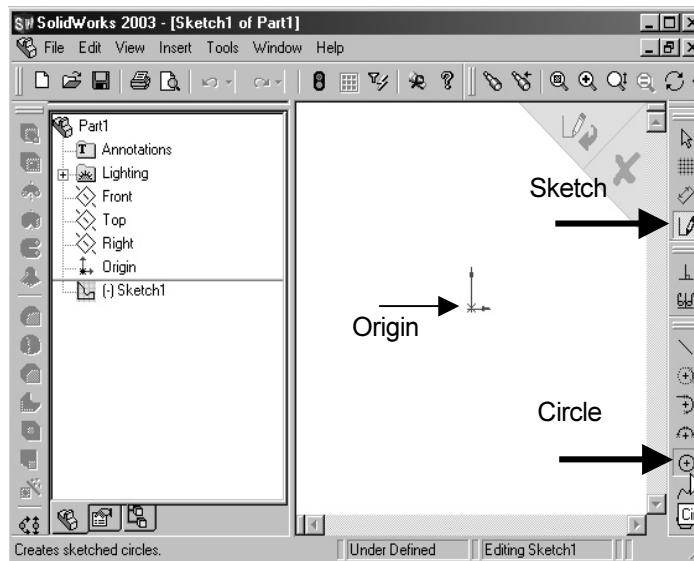
Set the Part Units.



- 13) Click **Units**. Select **Inches** [Millimeters] for Linear units.
- 14) Select **3** [2] for Decimal places.
- 15) Click **OK** to set the document units.

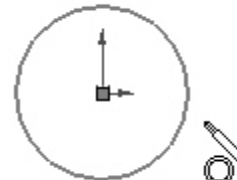


Insert a new Sketch for the Extruded-Base feature.

- 16) Click **Sketch**  from the Sketch toolbar. The Front plane is the default Sketch plane.
- 17) Click **Circle**  from the Sketch Tools toolbar.
- 18) Drag the **mouse pointer** into the Graphics window. The cursor displays the Circle feedback symbol .




- 19) The center point of the circle is positioned at the Origin. 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 display the diameter of the circle.
- 21) Click a **position** in the Graphics window to create a circle.

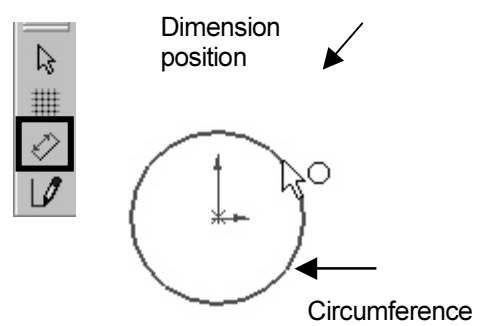
Dimension the circle.

22) Click **Dimension**  from the Sketch toolbar.

23) Click the **circumference** of the circle. The cursor displays the diameter

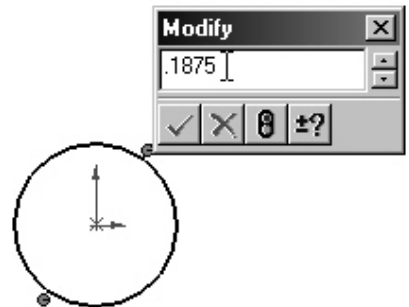
feedback symbol 

24) Click a **position** above the circle in the Graphics window to locate the dimension.



25) Enter **.1875 [4.76]** in the Modify dialog box. The diameter of the circle is .1875.

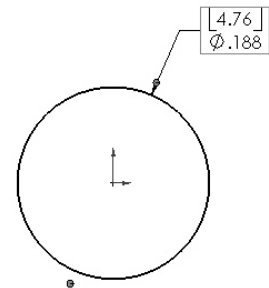
26) Click the **Check Mark** 




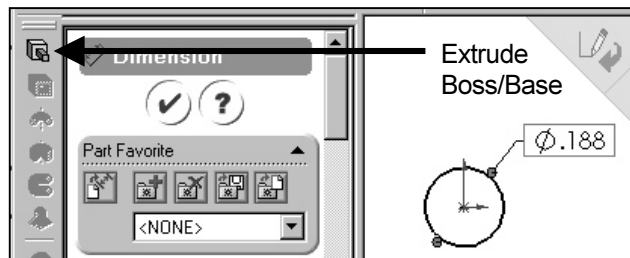
Three decimal places are displayed. The diameter value .1875 rounds to .188.

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


Note: Select **UNDO**  from the Standard toolbar, if your sketch is not correct.

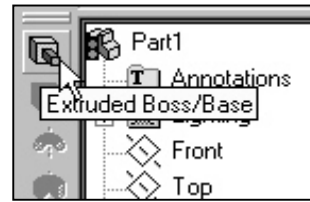


The Extrude Boss/Base  icon is highlighted in the Features toolbar.



Extrude the sketch to create the first feature.

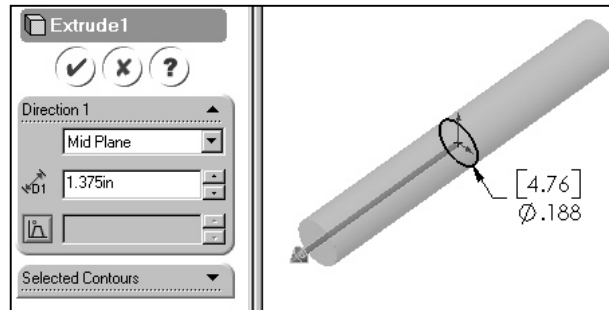
- 27) Click **Extruded Boss/Base**  from the Features toolbar. The Extrude PropertyManager is displayed.



- 28) The Graphics window displays an Isometric view of the sketch. Select **Mid Plane** for Direction 1 End Condition.

- 29) Enter **1.375 [34.93]** for Distance.

- 30) Click **OK**  to create the Extruded-Base feature.



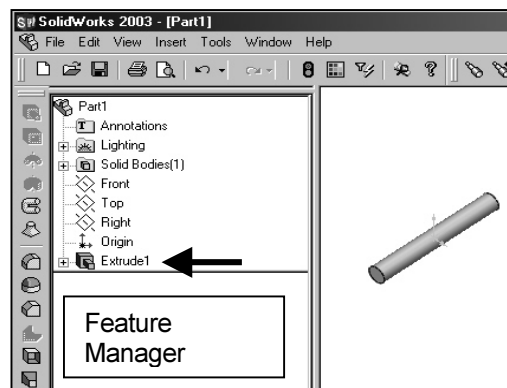
- 31) Fit the model to the Graphics window. Press the **f** key.

The Extrude Property Manager 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.

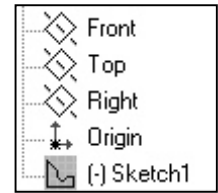
Extrude features add material. Extrude features require the following:

1. Sketch Plane.
2. Sketch.
3. Depth.



The default sketch plane is the Front plane. The Sketch is a circle with the diameter of .1875 [4.76]. The Depth is 1.375 [34.93].

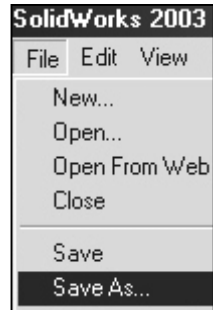
Note: The FeatureManager displays Sketch1 if you exit the sketch before you select a Feature. Select Sketch1 from the FeatureManager. Select the Extrude Boss/Base from the Feature toolbar.



Right-click the Sketch in the FeatureManager. Press the Delete key to delete the sketch.

Save the part.

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

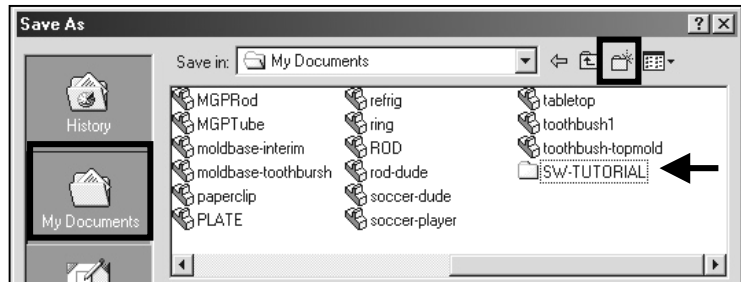


33) Double-click the **MY-DOCUMENTS** file folder.

34) Click **Create New Folder**



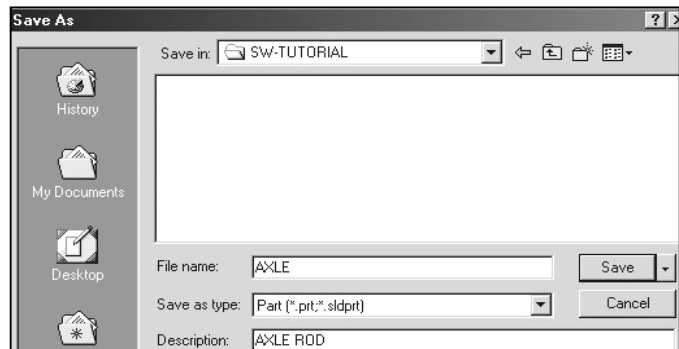
35) Enter **SW-TUTORIAL** for the file folder name.



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

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

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

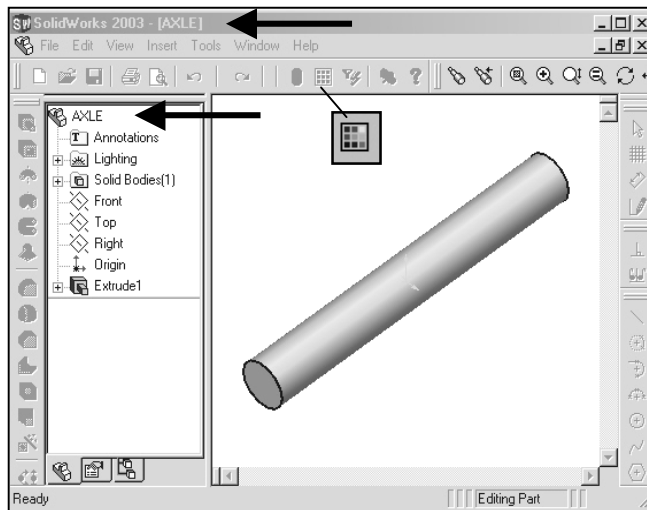


**39)** Click the **Save** button.

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

Modify the color of the part.

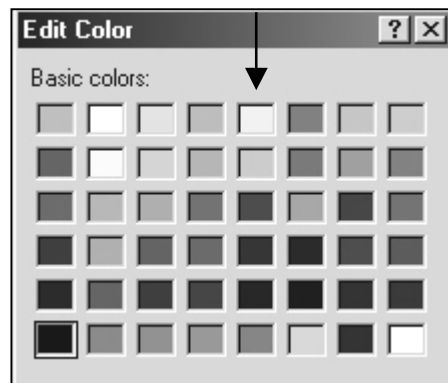
**40)** Click the **AXLE Part**  icon at the top of the FeatureManager.



**41)** Click **Edit Color**  from the Standards toolbar.

**42)** Select a **color swatch** from the Edit Color dialog box.

**43)** Click **OK** to apply the color and exit the Edit Color dialog box.



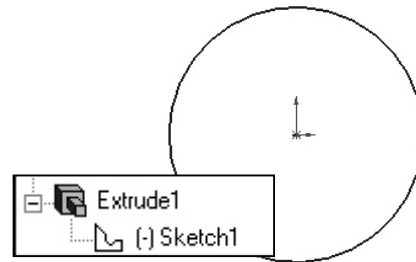
Utilize Edit Color to control part and feature color. SolidWorks utilizes default colors to indicate status of sketches and features.

Example:

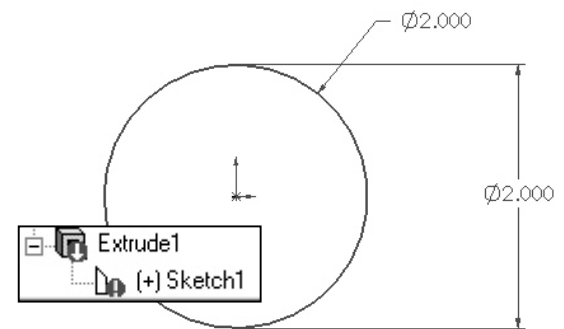
Default Colors indicate the status of a sketch.

Sketches are:

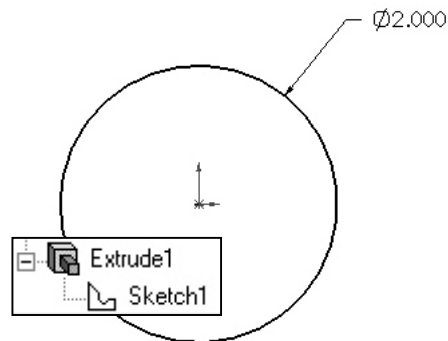
- Under Defined: There is inadequate definition of the sketch, (Blue). The FeatureManager displays a minus (-) symbol before the Sketch name.



- 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 Feature and the Sketch display a red warning tag.

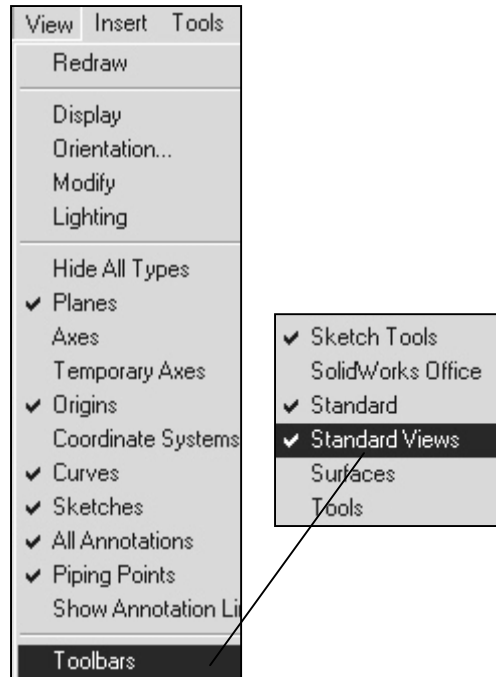


Display the Standard Views of the AXLE.

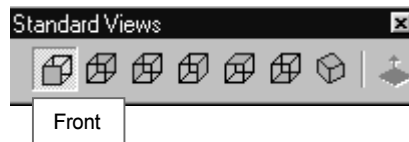
**44)** Display the Standards View toolbars. Click **View** from the Main menu.

**45)** Click **Toolbars**. The system places a Check Mark in the front of the displayed toolbars.

**46)** Click the **Standard Views**. The Standard Views toolbar is displayed below the Main menu.



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

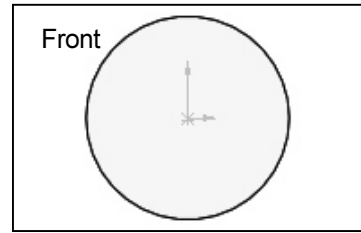


Orthographic projection is the process of projecting views onto parallel planes with  $\perp$  projectors.

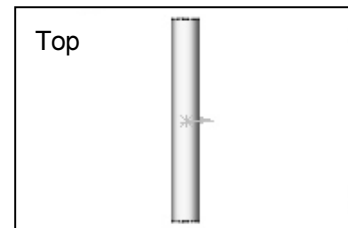
The default reference planes are the Front, Top and Right side viewing planes.

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

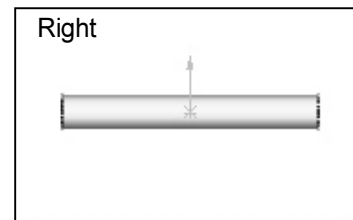
48) Click the **Front**  view.



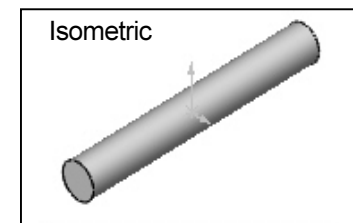
49) Click the **Top**  view.



50) Click the **Right**  view.



51) Click the **Isometric**  view.






The View modes manipulate the model in the Graphics windows.





Display the various View modes.

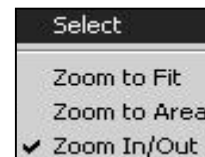
**52)** Click **Zoom to Fit**  to display the full size of the part in the current window.


**53)** Click **Zoom to Area** .


**54)** Select two **opposite corners** of a rectangle to define the boundary of the view. The defined view fits to the current window.


**55)** 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.

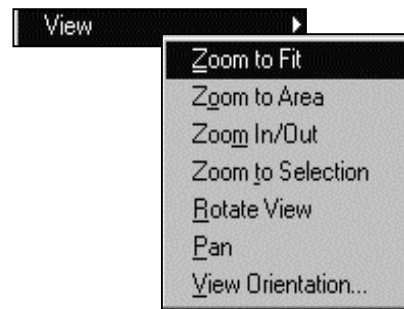
**56)** **Right-click** in the Graphics window. Click **Select**. Click the **front circular edge**. Click **Zoom to Selection** . The selected geometry fills the current window.



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


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

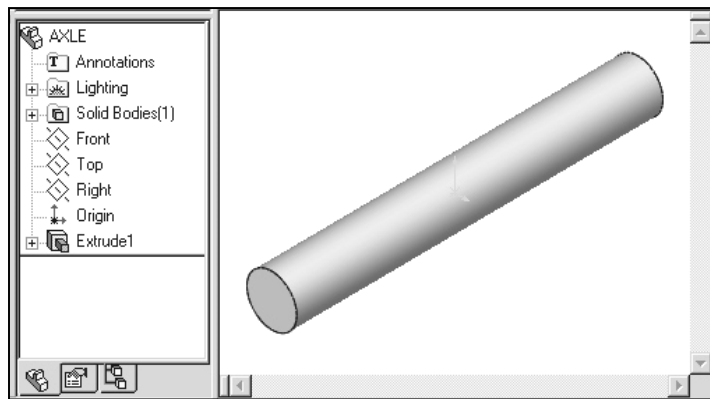
**59)** Right-click in the **Graphics window** area to display the zoom options. Click **Zoom to Fit** .



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

Display the Isometric view.

**60)** Click **Isometric**  from the Standards View toolbar.



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

Review the AXLE Part.

The AXLE part utilized an Extrude feature. The Extrude feature adds material. An Extrude feature requires a Sketch plane, Sketch and Depth.

The Sketch plane is the Front plane (default).

The 2D sketch is a circle centered at the Origin. A dimension defines the overall size of the sketch.

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

Utilize Edit Color to modify the part color. Select the Part icon in the FeatureManager.

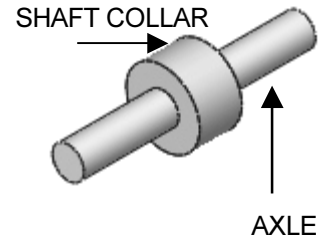
Color defines the sketch status. A blue sketch is under defined. A black sketch is fully defined. A red sketch is over defined.

The default reference planes are the Front, Top and Right side viewing planes. Utilize the Standard Views toolbar to display the principle views of a part.

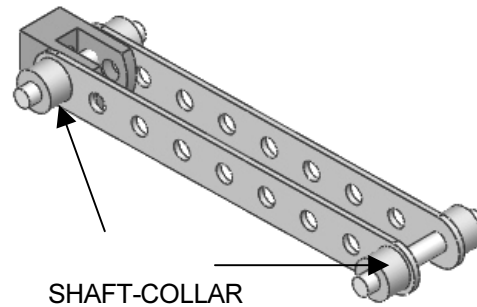
The View modes manipulate the model in the Graphics windows. Utilize Zoom, Pan and Rotate from the View toolbar.

**SHAFT-COLLAR Part**

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



Four SHAFT-COLLAR parts position the two FLATBAR parts and placed on the AXLE parts.

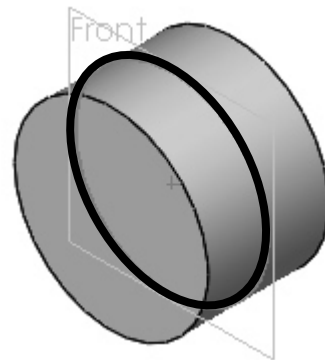


Create the SHAFT-COLLAR part. Utilize an Extruded-Base feature. The Extruded-Base feature requires a 2D circular profile.

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

Extrude the sketch with the Mid Plane End Condition.

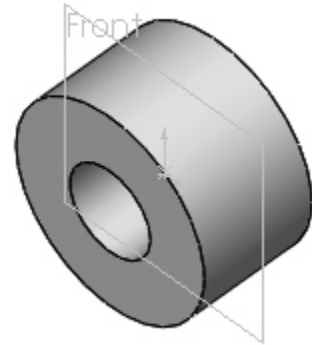
The Extruded-Base feature is centered on both sides of the Front plane.




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.

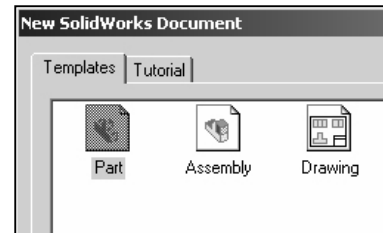
Select the Depth option Through All extends the Extruded Cut feature from the Front plane through all existing geometry.



Create a New Part.

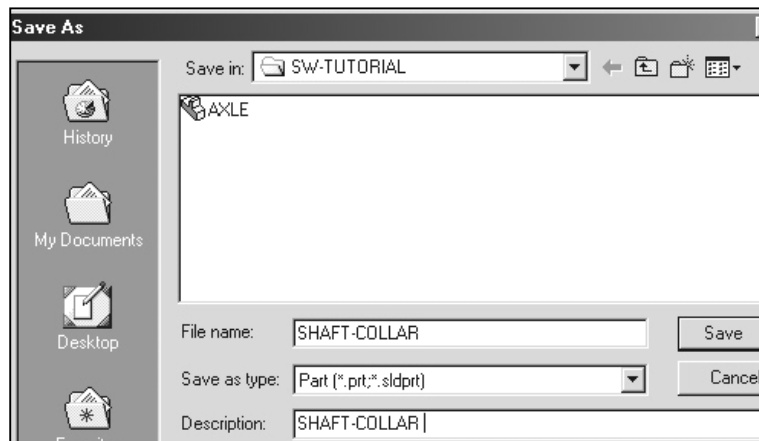
**62)** Click **New**  from the Standard toolbar.

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



**64)** Save the Part. Click **File, Save As**.

**65)** Select **SW-TUTORIAL** for the Save in file folder name.



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

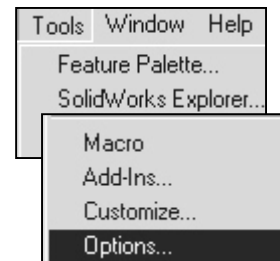
**67)** Enter **SHAFT-COLLAR** for Description.

**68)** Click the **Save** button.

Set the Dimensioning Standard and Part Units.

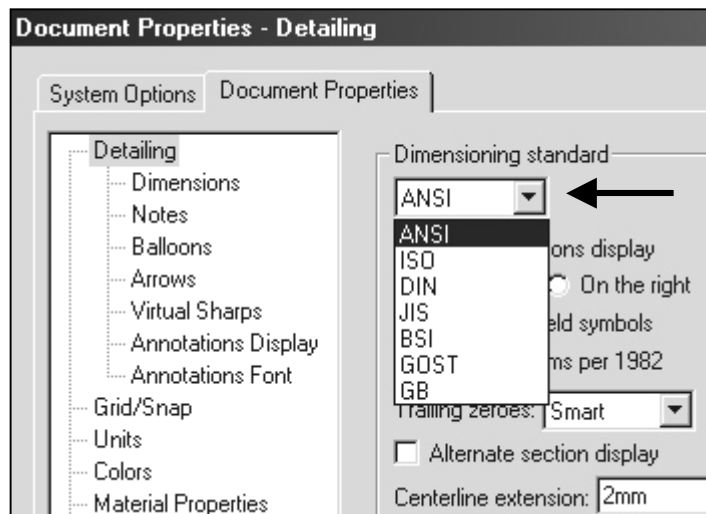
**69)** Click **Tools** from the Main menu.

**70)** Click **Options**.



**71)** Click the **Document Properties** Tab.

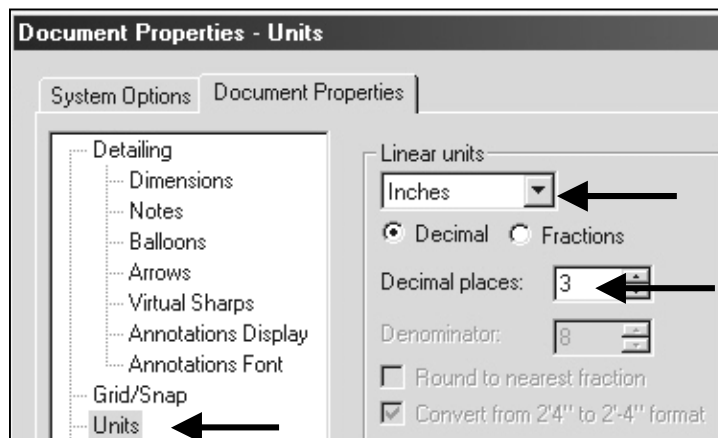
**72)** Select **ANSI** from the Dimensioning Standard list box.



**73)** Click **Units**. Select **Inches [Millimeters]** for Linear units.


**74)** Select **3 [2]** for Decimal places.


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

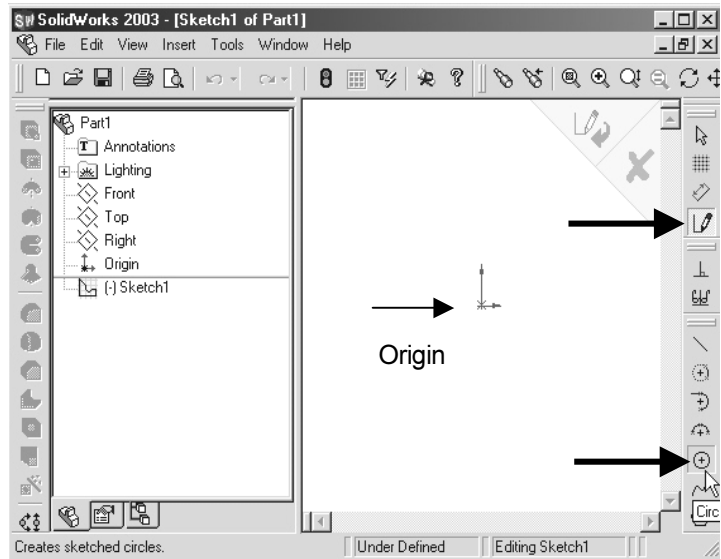




Insert a new Sketch for the Extrude feature.

- 76) Click **Sketch**  from the Sketch toolbar.

- 77) Click **Circle**  from the Sketch Tools toolbar.

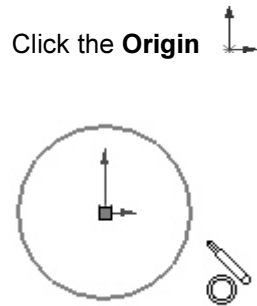
- 78) Drag the **mouse pointer** into the Graphics window. The cursor displays the Circle feedback symbol 



- 79) The center point of the circle is positioned at the Origin. Click the **Origin** . The cursor displays the Coincident to point feedback symbol 


- 80) Drag the **mouse pointer** to the right of the Origin to display the diameter of the circle.

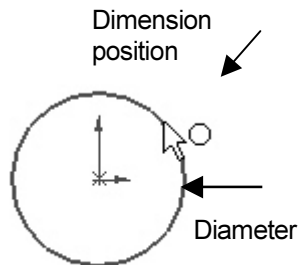
- 81) Click a **position** in the Graphics window to create the circle.



Dimension the circle.

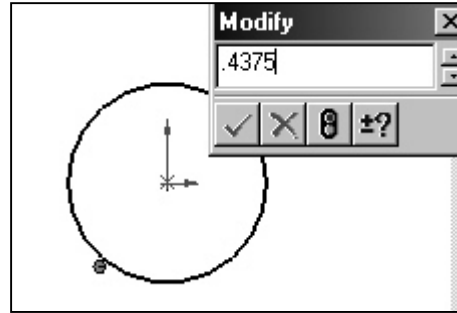
- 82) Click **Dimension**  from the Sketch toolbar.

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



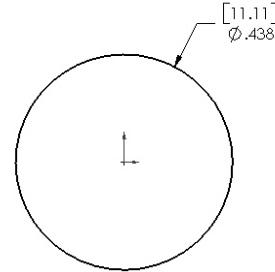
- 84) Click a **position** in the Graphics window above the circle to locate the dimension.

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



- 86) Click the **Check Mark** . The black sketch is fully defined.

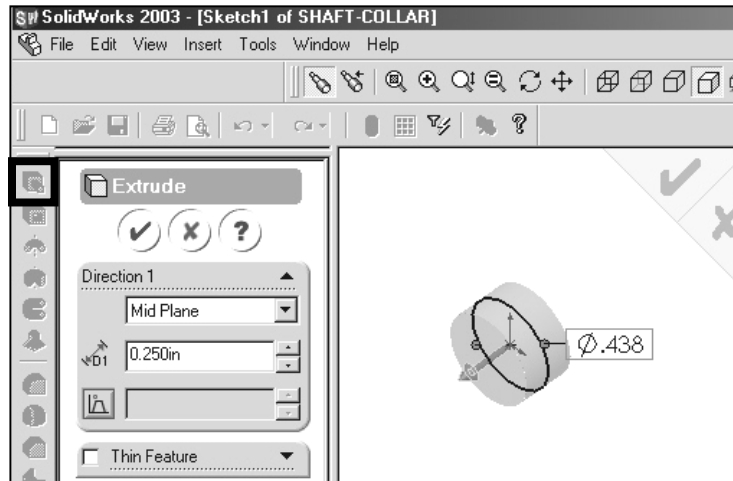
Note: Three decimal places are displayed. The diameter value **.4375** rounds to **.438**.



The Extrude Boss/Base icon is highlighted in the Features toolbar.

Extrude the sketch to create the first feature.

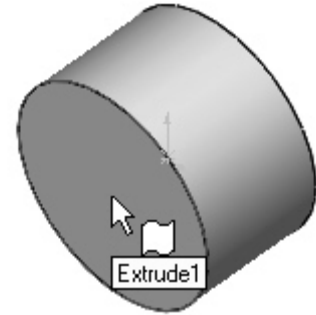
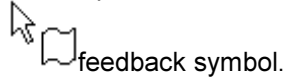
- 87) Click **Extruded Boss/Base** from the Features toolbar.




- 88) Select **Mid Plane** for Direction1 End Condition.
- 89) Enter **.250 [6.35]** for Depth.
- 90) Click **OK** to create the Extruded-Base feature.
- 91) Fit the model to the Graphics window. Press the **f** key.



Insert a new sketch for the Extruded-Cut.

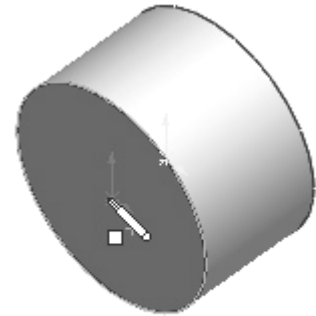
- 92)** Select the Sketch plane. Click the **front circular face** of the Extrude1 feature for the Sketch plane. The cursor displays the Face




- 93)** Click **Sketch**  from the Sketch toolbar.

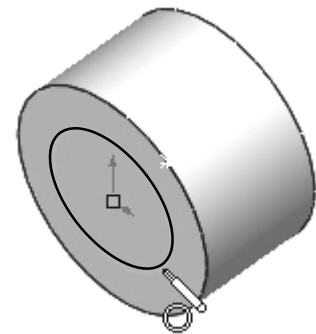
- 94)** Click **Circle**  from the Sketch Tools toolbar.

- 95)** The center point of the circle is positioned at the Origin. Click the **Origin** . The cursor displays the Coincident to point feedback symbol .



- 96)** Drag the **mouse pointer** to the right of the Origin to display the diameter of the circle. The cursor

displays the Circle feedback symbol. 




- 97)** Click a **position** in the Graphics window to create the circle.




Dimension the circle.

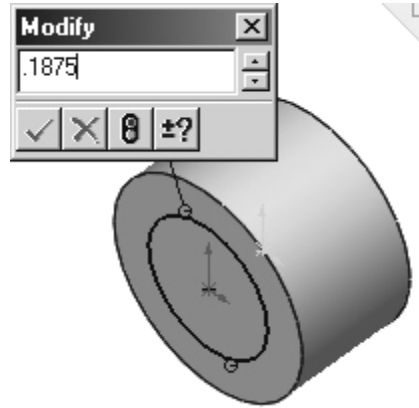
98) Click **Dimension** .

99) Click the **circumference** of the circle.  
The cursor displays the diameter


feedback symbol 

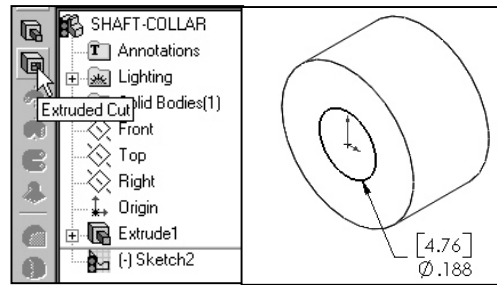
100) Click a **position** in the Graphics window  
above the circle to locate the dimension.

101) Enter **.1875 [4.76]** in the Modify dialog  
box. Click the **Check Mark** .




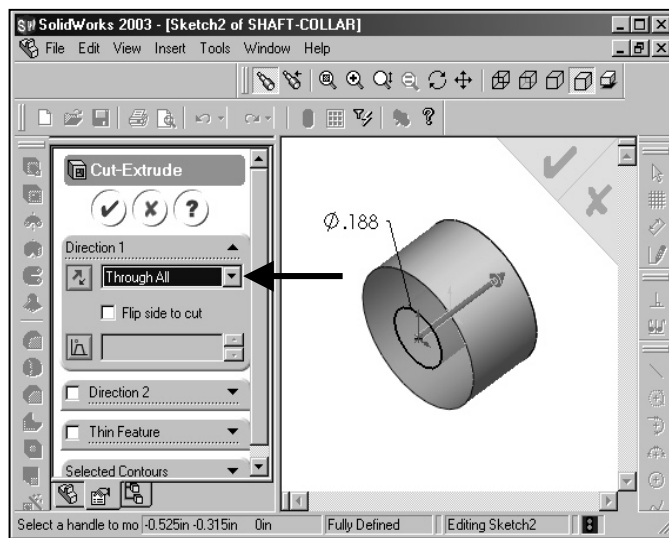
Insert an Extruded-Cut.

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



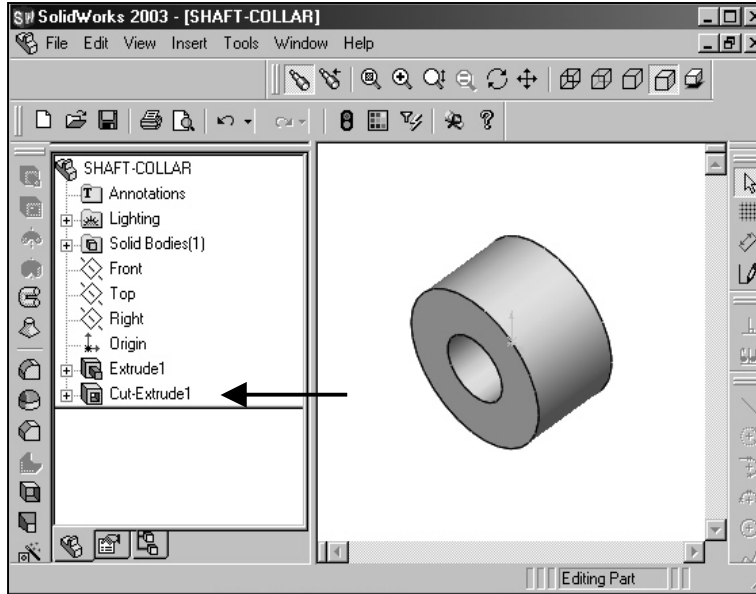
103) Select **Through All** for Direction1 End Condition.

104) Click **OK**  to create the Extruded-Cut feature.



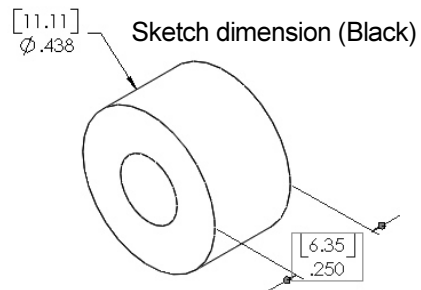
The Extruded-Cut feature is named Cut-Extrude1.

The Through All End Condition removes material from the Front plane through the Extrude1 geometry.



Modify the Dimensions.

**105)** 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.

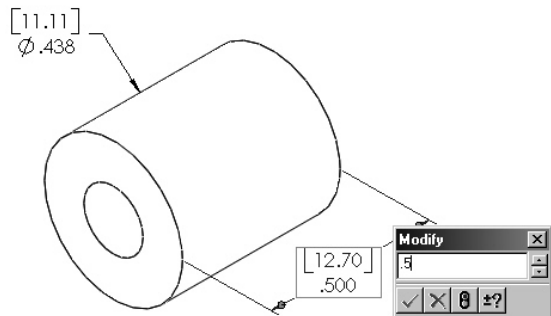


Extrude Depth dimension (Blue)

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

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


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




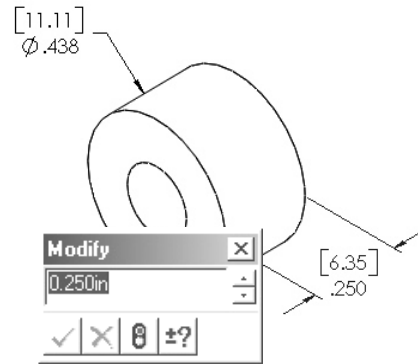
The Extrude1 and Cut-Extrude1 features are modified.

**109)** Return to the original dimensions. Enter **.250 [6.35]** in the Modify dialog box.

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


**111)** Click the green **Check Mark** .

**112)** Click **OK**  from the Dimension PropertyManager to display the FeatureManager.




Modify the part color.

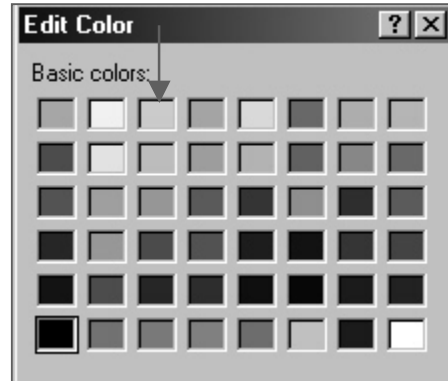
**113)** Click the **SHAFT-COLLAR Part**  **SHAFT-COLLAR** icon at the top of the FeatureManager.

**114)** Click **Edit Color**  from the Standards toolbar.

**115)** Select a **color swatch** from the Edit Color dialog box.

**116)** Click **OK** to apply the color and exit the Edit Color dialog box.

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



Review the SHAFT-COLLAR Part.

The SHAFT-COLLAR utilized an Extrude feature. The Extrude feature adds material. An Extrude feature requires a Sketch plane, Sketch and Depth.

The Sketch plane is the Front plane (default).

The 2D sketch is a circle centered at the Origin. A Dimension defines the overall size of the sketch.

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

The Extruded-Cut feature removes material to create the hole. The Extruded-Cut feature is named Cut-Extrude1.

The Through All End Condition option creates the Cut-Extrude1 feature from the Front plane through all Extrude1 geometry.

Modify feature dimension. Double-click the feature in the Graphics window. Double-click the dimension. Enter the new value. Rebuild the feature.

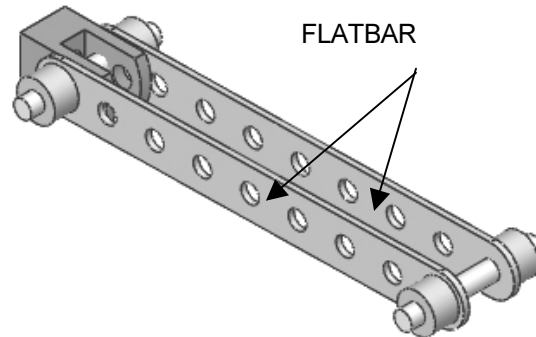
Utilize Edit Color to modify part color. Select the Part icon in the FeatureManager to modify the part color.

**FLATBAR Part**

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

The FLATBAR is manufacture from .060in stainless steel.

Create the FLATBAR part.  
Utilize an Extrude feature.

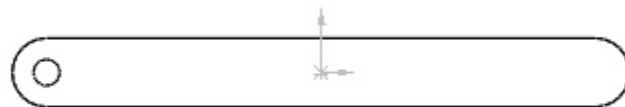


The Extrude feature requires a 2D profile, sketched on the Front plane.

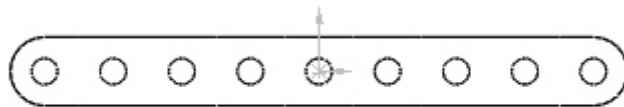
Utilize symmetry. Create the 2D profile centered about the Origin.




Utilize an Extruded-Cut feature to create the first hole.



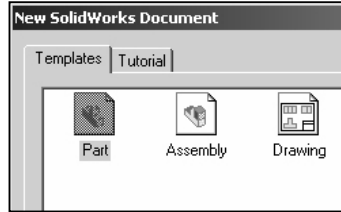
Utilize a Linear Pattern to create the remaining holes. A Linear Pattern creates an array of features in a specified direction.



Create a New Part.

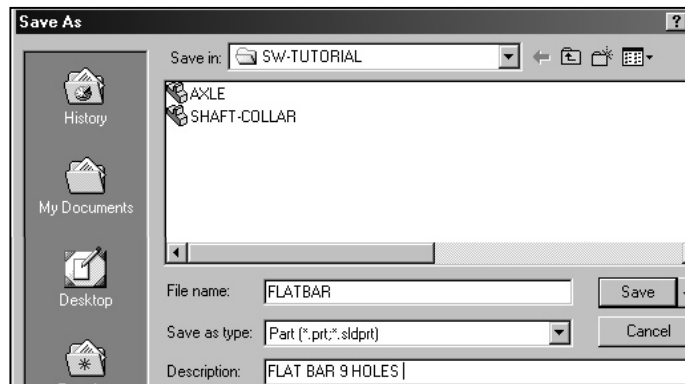
**118)** Click **New**  from the Standard toolbar.

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



**120)** Save the Part. Click **File, Save As**.

**121)** Select **SW-TUTORIAL** for the Save in folder file name.



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

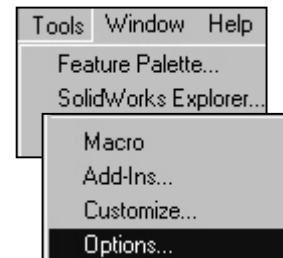
**123)** Enter **FLAT BAR 9 HOLES** for Description.

**124)** Click the **Save** button.

Set the Dimensioning Standard and Part Units.

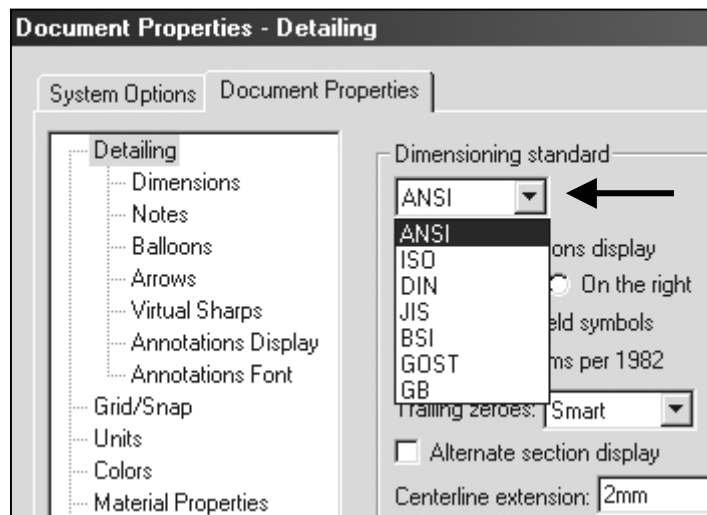
**125)** Click **Tools** from the Main menu.

**126)** Click **Options**.



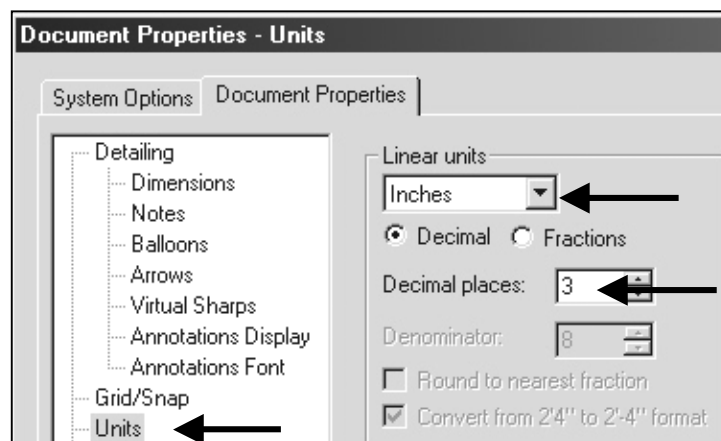
**127)** Click the **Document Properties** Tab.

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



**129)** Click **Units**. Select **Inches [Millimeters]** for Linear units.

**130)** Select **3 [2]** for Decimal places.




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

Insert a new Sketch for the Extruded Base feature.

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

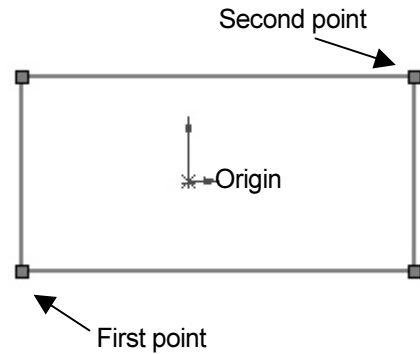
**133)** Click **Rectangle**  from the Sketch Tools toolbar.

**134)** Drag the **mouse pointer** into the Graphics window. The cursor displays the


Rectangle feedback symbol .

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

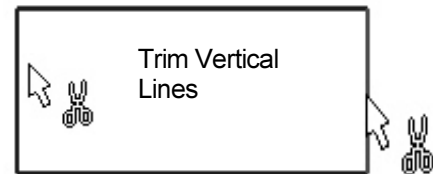
**136)** Click the **second point** of the rectangle above and to the right of the Origin.



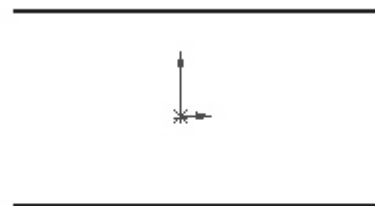
**137)** Trim the vertical lines. Click **Sketch**

**Trim**  from the Sketch Tools toolbar.

**138)** Click the **right vertical line** of the rectangle.



**139)** Click the **left vertical line**. Two equal and parallel horizontal lines are displayed.




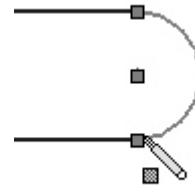


**140)** Sketch the right arc. Click **Tangent Arc**  from the Sketch Tools toolbar.

**141)** Drag the **mouse pointer** along the top horizontal line to the right until a dotted horizontal line is displayed. The referenced top horizontal line is displayed in green.




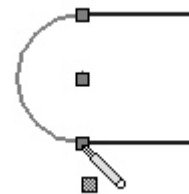
**142)** Click the **top right endpoint**. Drag the mouse pointer to the right and downward. Click the **bottom right endpoint**. The cursor displays the Coincident to point feedback symbol  at each endpoint.



**143)** Sketch the left arc. Drag the **mouse pointer** along the top horizontal line to the left until a dotted horizontal line is displayed. Release the **mouse pointer**. The referenced top horizontal line is displayed in green.

**144)** Click the **top left endpoint**. Drag the **mouse pointer** to the right and downward.

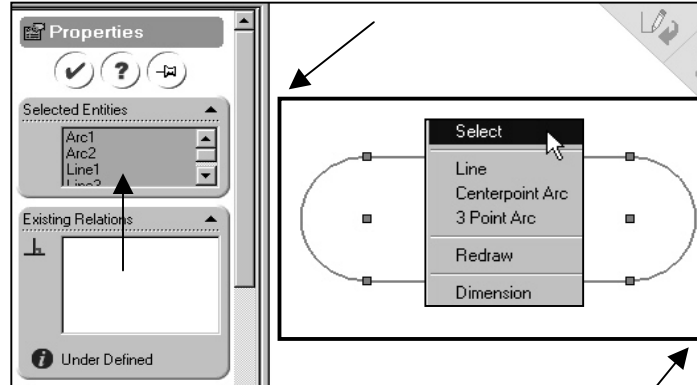
**145)** Click the **bottom left endpoint**. The cursor displays the Coincident to point feedback symbol  at each endpoint.



Select Geometry.

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

**147)** Click a **position** in the upper left corner of the Graphics window. Drag the **mouse pointer** to the lower right corner of the Graphics window. Release the **mouse pointer**. The geometry inside the window is selected. Select geometry is displayed in green.

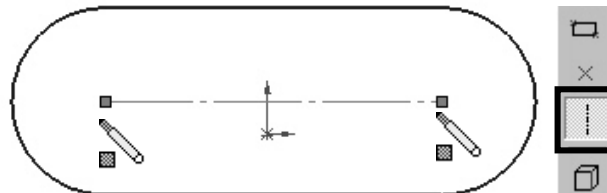


Two arcs and two lines are listed in the Properties Selected Entities box.


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.


Add Relations.

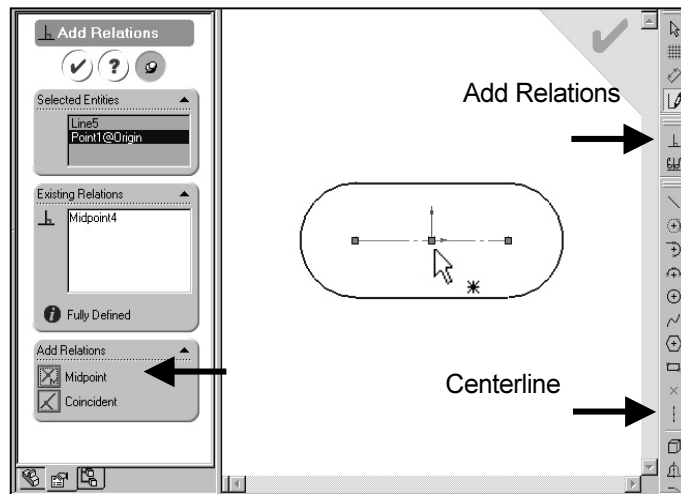
**148)** Sketch a centerline. Click **Centerline**  from the Sketch Tools toolbar.





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


The cursor displays the Coincident to point feedback symbol  at each endpoint.

150) Click Add Relations .




151) Select the **Origin** . The Origin and the Centerline are listed in the Selected Entities box. The cursor displays the Origin  \* feedback symbol.

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

153) Click **OK**  to add a Midpoint relation. The slot sketch is centered about the Origin.

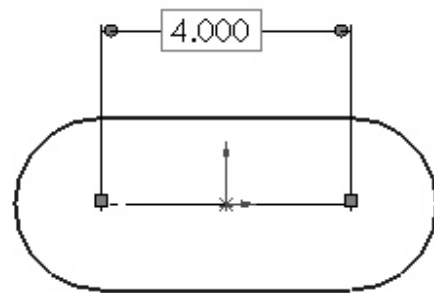
Dimension the slot.

154) Click **Dimension** .

155) Click the **horizontal centerline** of the slot.

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

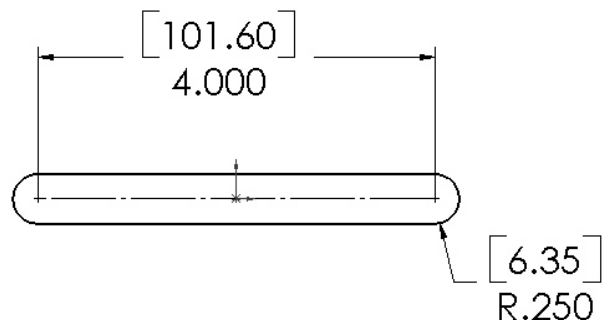
157) Enter **4.0** [101.60] in the Modify dialog box.



158) Click the **Check Mark** .

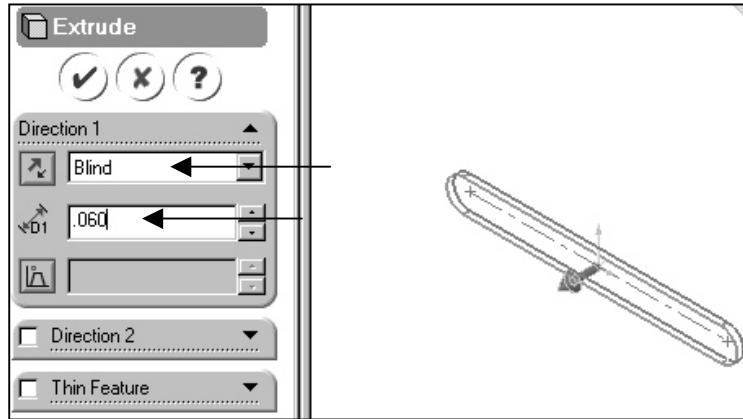
159) Click the **right arc** of the slot.

160) Click a **position** to the right of the slot in the Graphics window to locate the dimension.



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

**162)** Click the **Check Mark** . The black sketch is fully defined.




Extrude the sketch to create the first feature.

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

Blind is the default Direction1 End Condition.

The Direction1 arrow points in the +z direction.

**164)** Enter **.060 [1.5]** for Depth.

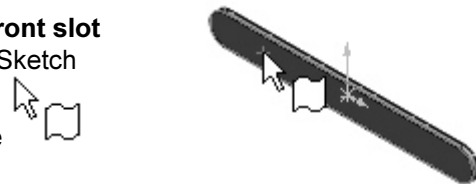
**165)** Click **OK**  to create the Extruded-Base feature.

**166)** Fit the model to the Graphics window. Press the **f** key.


Insert a new sketch for the Extruded-Cut.

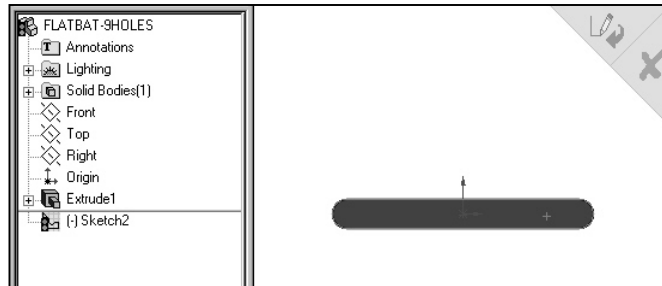
**167)** Select the Sketch plane. Click the **front slot face** of the Extrude1 feature for the Sketch

plane. The cursor displays the Face feedback symbol.




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

**169)** Display the **Front** view. Click the **Front**  view.

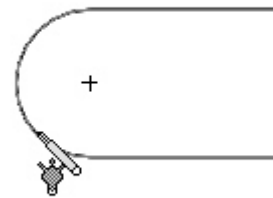


The process of placing the mouse pointer over an existing arc to find its center point is call “wake up”.

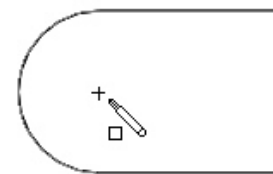
Wake up the center point.

**170)** Click **Circle**  from the Sketch Tools toolbar.

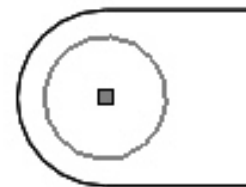
**171)** Place the **mouse pointer** on the left arc. Do not click. The center point of the slot arc is displayed.



**172)** Click the **center point**.



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



Dimension the circle.

**174)** Click **Dimension** .

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

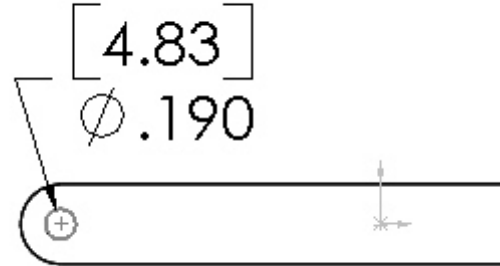


**176)** Click a **position** above the circle in the Graphics window to locate the dimension.

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

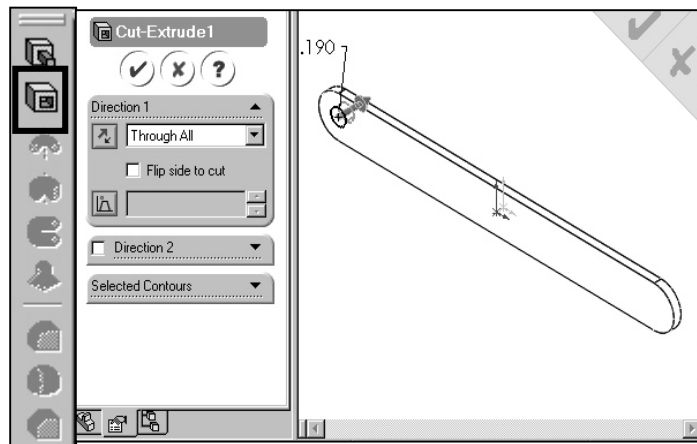
**178)** Click the **Check Mark** .

**179)** Click **Isometric** .




Insert an Extruded-Cut.

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




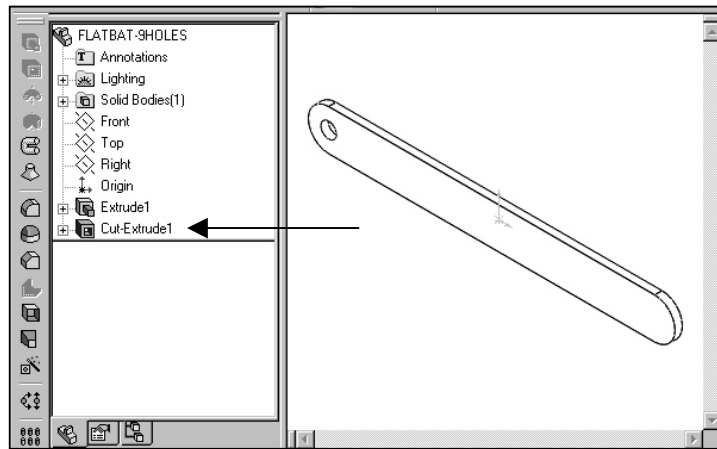
**181)** Select **Through All** for Direction1 End Condition.

**182)** Click **OK**  to create the Extruded-Cut.

**183)** Save the FLATBAR part. Click **Save** .


The Cut-Extrude1 feature is displayed in the FeatureManager.

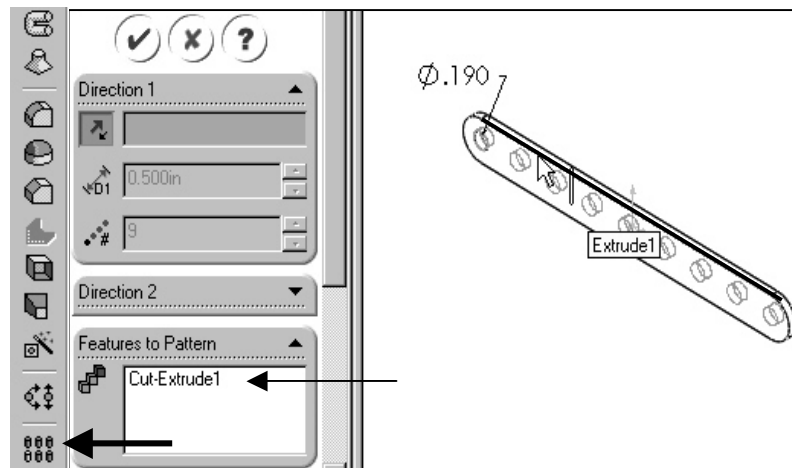
The blue Extruded-Cut  icon indicates the feature is selected.



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

Create a Linear Pattern.

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



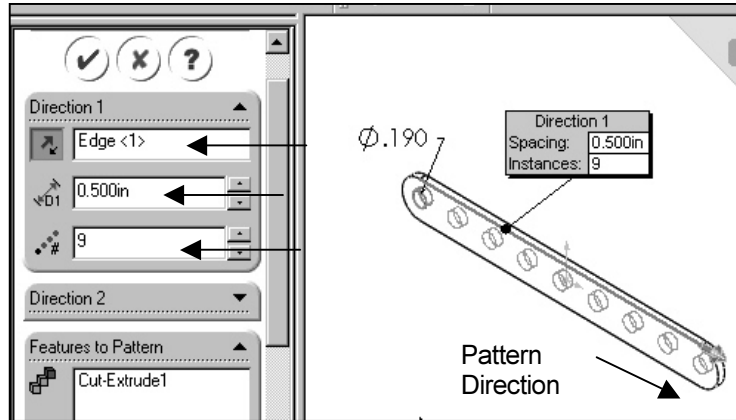
**185)** The Cut-Extrude1 is displayed in the Features to Pattern box.

**186)** Select the **top edge** of the Extrude1 feature for Direction1.

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

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

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



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

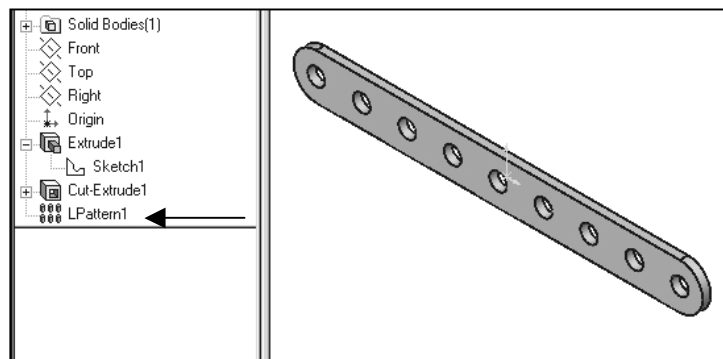


**190)** Click **OK** to create the Linear Pattern.



The LPattern1 feature is listed in the FeatureManager.

**191)** Save the FLATBAR part. Click **Save**. The FLATBAR part is complete.

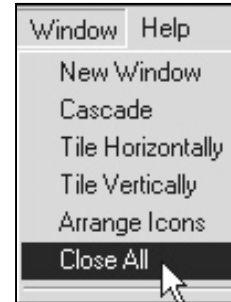




Close all documents.

**192)** Click **Windows** from the Main toolbar.

**193)** Click **Close All**.



Review the FLAT-BAR Part.

The FLAT-BAR utilized an Extrude feature.

The Sketch plane is the Front plane (default).

The 2D sketch utilized the Rectangle and Tangent Arc Sketch tools to create the slot profile.

Utilize a Centerline between the two arc center points. Add a Midpoint relation between the Origin and the centerline to maintain the slot profile symmetric about the Origin.

Add linear and radial dimensions to define the overall size of the sketch.

The name of the feature is Extrude1. Extrude1 utilized the Blind End Condition.

The Extruded-Cut feature removes material to create the hole. The Extruded-Cut feature is named Cut-Extrude1.

The Through All End Condition option creates the Cut-Extrude1 feature from the Front plane through all Extrude1 geometry.

Utilize the Linear Pattern to create an array of 9 holes, equally spaced along the length of the FLATBAR.

## Linkage Assembly

An assembly is a document that contains two or more parts.

An assembly inserted into another assembly is called a sub-assembly.

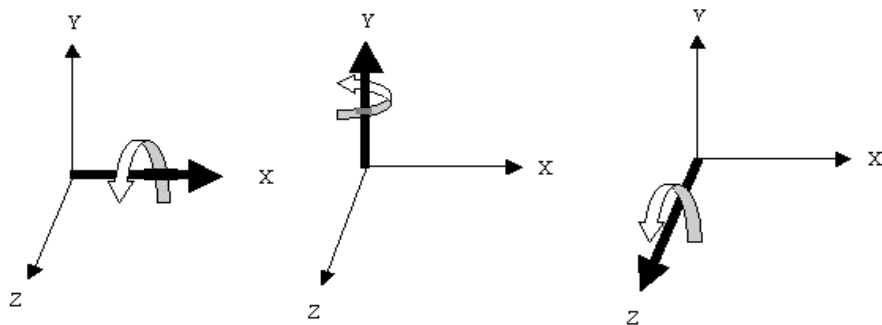
A part or sub-assembly inserted into an assembly is called a component.

The LINKAGE Assembly consists of the following components:

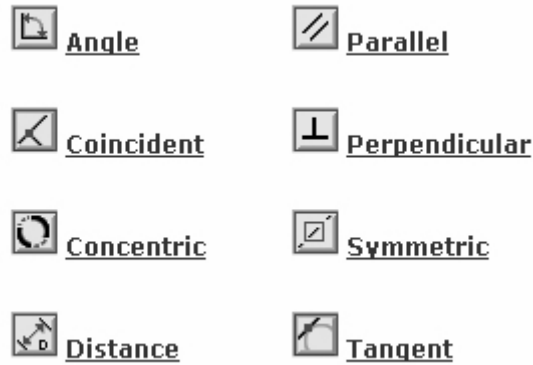
- AXLE part.
- SHAFT-COLLAR part.
- FLATBAR part.
- AirCylinder sub-assembly.

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.

In dynamics, motion of an object is described in linear and rotational terms. Components possess linear motion along the x, y and z-axes and rotational motion around the x, y, and z-axes. In an assembly, each component has 6 degrees of freedom: 3 translational (linear) and 3 rotational. Mates remove degrees of freedom. All components are rigid bodies. The components do not flex or deform.



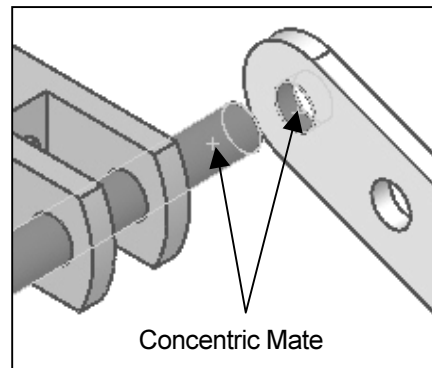
Components are assembled with various Mate types. The Mate types are:



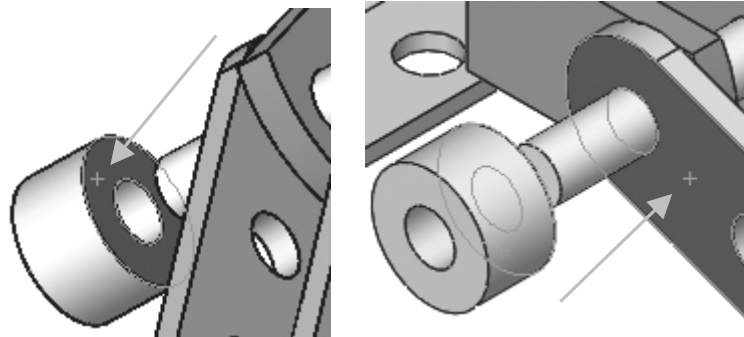
Mates require geometry from two different components. Selected geometry includes Planar Faces, Cylindrical faces, Linear edges, Circular/Arc edges, Vertices, Axes, Temporary axes, Planes, Points and Origins.

Mates reflect the physical behavior of a component in an assembly.

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.



Download the AirCylinder Assembly.

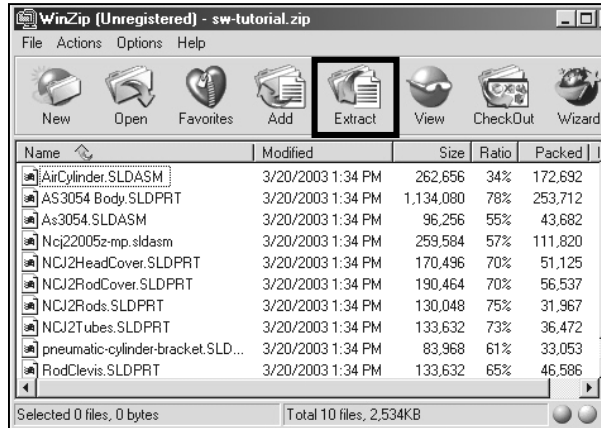
**194)** Invoke a **web browser**.

**195)** Enter the following URL: **www.schroff1.com**.

**196)** Select **SolidWorks Tutorial 2003**.

**197)** **Download** the zip file.


**198)** **Unzip** the downloaded zip file. The zip file contains parts and assemblies.



**199)** Select the **SW-TUTORIAL** as the Save as file folder.

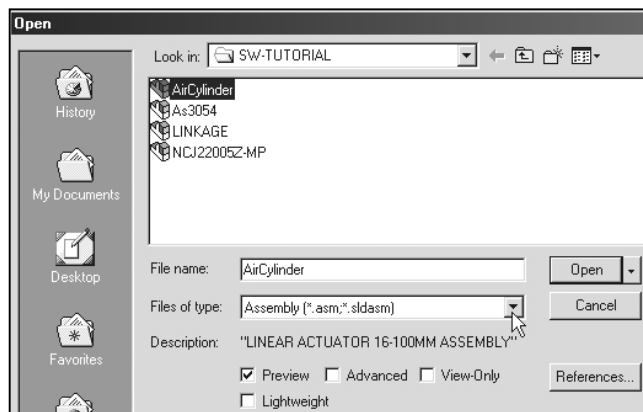
**200)** **Extract** the files to the SW-TUTORIAL file folder.

Open the AirCylinder Assembly.

**201)** Click **Open** .

**202)** Select **Assembly** from the Files of type list.

**203)** Click **AirCylinder** for File name.

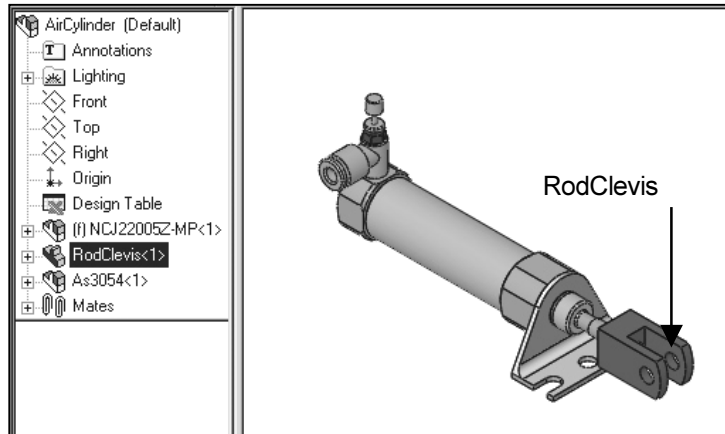


**204)** Click **Open** to view the AirCylinder Assembly in the Graphics window.


The AirCylinder Assembly is displayed in the Graphics window.

- 205)** Click the **RodClevis** from the Feature Manager.

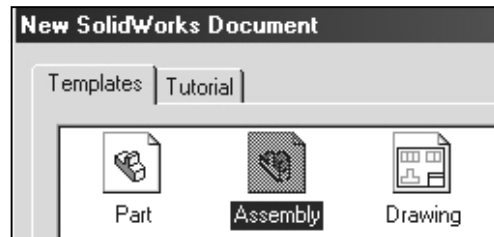
Assemble the AXLE to the holes in the RodClevis in the LINKAGE Assembly. ?



Create a New Assembly.

- 206)** Click **New**  from the Standard toolbar.

- 207)** The Templates tab is the default tab. Click the **Assembly** template from the New SolidWorks Document dialog box.



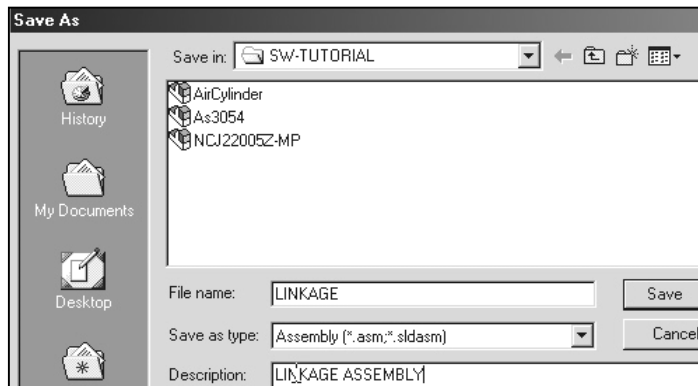
- 208)** Click **OK**.

- 209)** Save the Assembly. Click **File, Save As**.

- 210)** Select **SW-TUTORIAL** for the Save in file folder.

- 211)** Enter **LINKAGE** for File name.

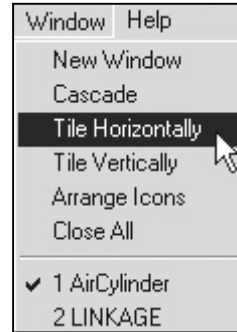
- 212)** Enter **LINKAGE ASSEMBLY** for Description.



**213)** Click the **Save** button.

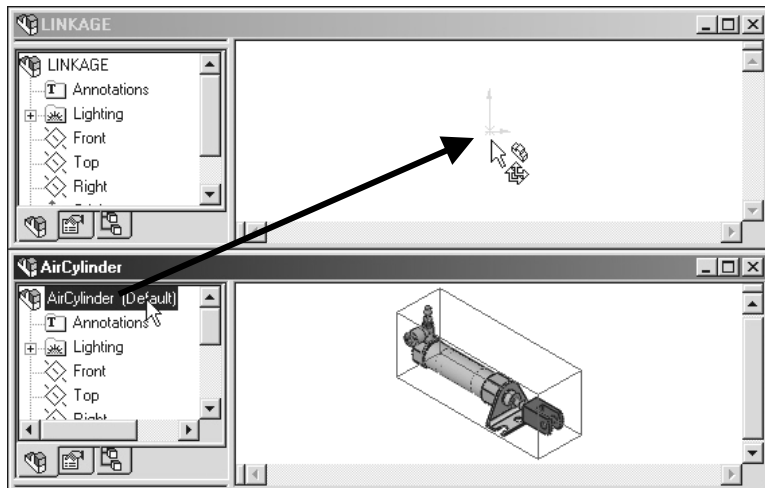
Insert the AirCylinder Assembly.

**214)** Click **Window, Tile Horizontally**.





**215)** Drag the **AirCylinder** assembly icon into the Origin of the LINKAGE Assembly. The cursor displays the Insert Component Fixed at the

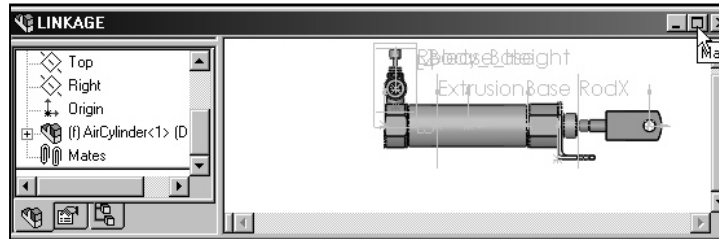
Origin feedback symbol



Display the LINKAGE Assembly.

**216)** Minimize the AirCylinder Assembly. Click **Minimize**  in the AirCylinder Graphics window.

**217)** Maximize the LINKAGE Assembly Graphics window. Click the **Maximize**  button.

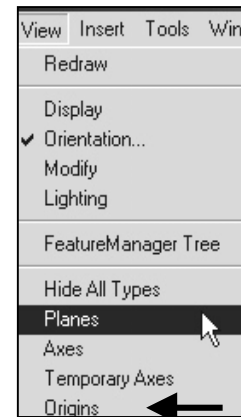


**218)** Hide the Planes and Origins. Click **View** from the Main toolbar.

**219)** Uncheck **Planes**.

**220)** Click **View** from the Main toolbar.

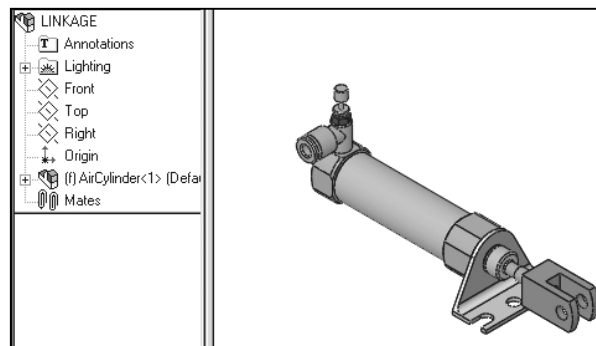
**221)** Uncheck **Origins**.



Display the Isometric view.

**222)** Click **Isometric** .

**223)** Save the LINKAGE Assembly. Click **Save** . Click **Yes** to the question, "Save the documents and the referenced models now."

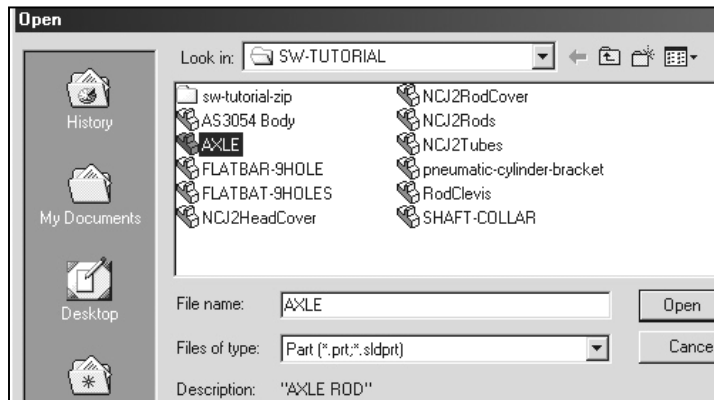


The Air Cylinder is the first component in the LINKAGE Assembly and is fixed to the LINKAGE Assembly Origin.

Open the AXLE part.

**224)** Click **Open** . Select **Part** from the Files of type list.


**225)** Click **AXLE** for File name.

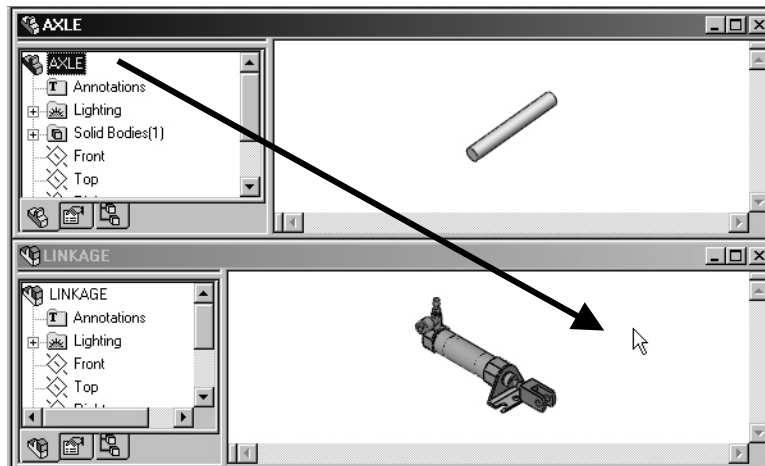


**226)** Click **Open**. The AXLE part is displayed in the Graphics window.

Insert the AXLE.

**227)** Click **Window** from the Main toolbar. Click **Tile Horizontally**.


**228)** Drag the **AXLE**  part icon into the LINKAGE assembly.



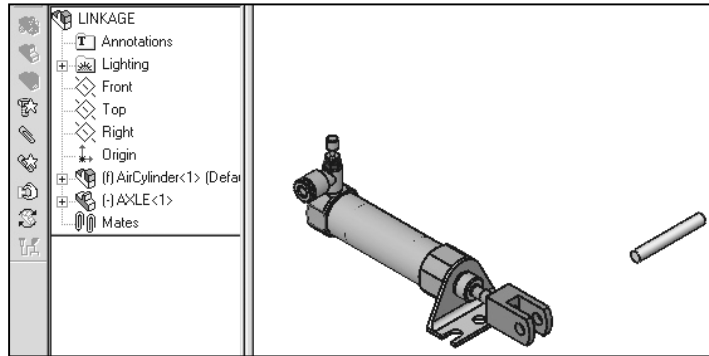
**229)** Click a **position** to the right of the AirCylinder Assembly.

**230)** Minimize the AXLE. Click **Minimize**  in the AXLE Graphics window.



**231)** Maximize the LINKAGE Assembly Graphics window. Click the **Maximize**  button.

**232)** Fit the model to the Graphics window. Press the **f** key.

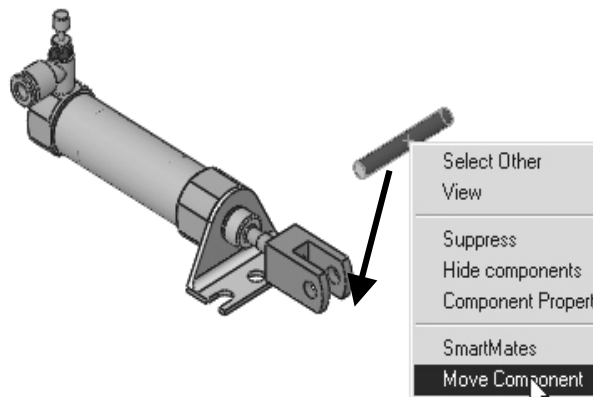


Move Component.

**233)** Right-click on the **AXLE** in the LINKAGE Assembly Graphics window.

**234)** Click **Move Component**.

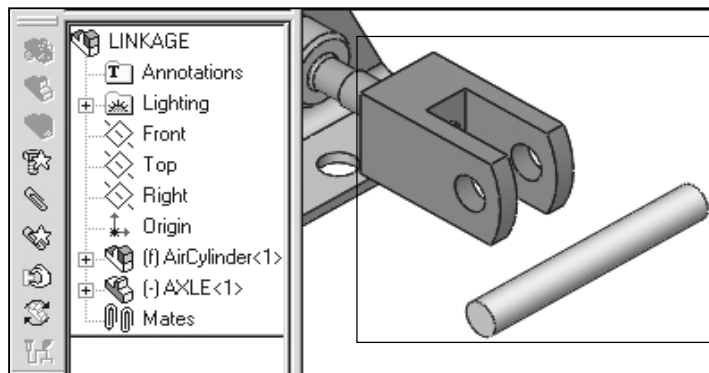
**235)** Drag the **AXLE** in front of the RODCLEVIS.



**236)** Click **OK**  to exit Move Component.

Enlarge the view.

**237)** Click **Zoom to Area** .

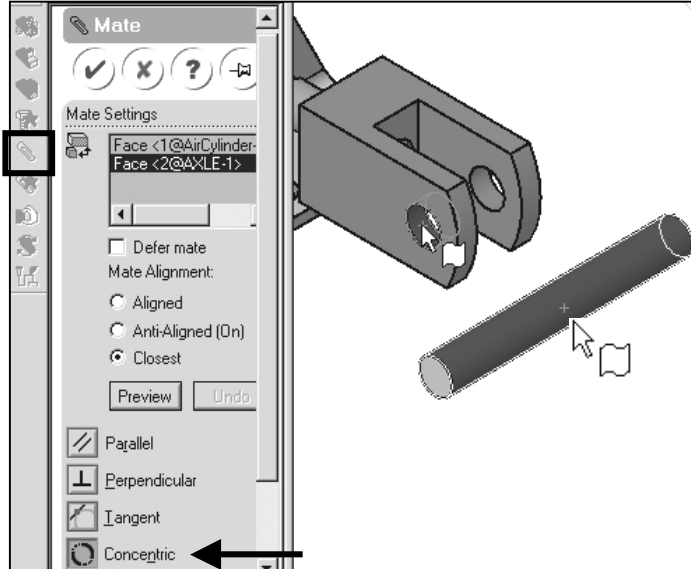


**238)** **Zoom In** on the RodClevis and the AXLE to enlarge the view.

Insert a Concentric Mate.

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

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



**241)** Click the **long cylindrical face** of the AXLE. The cursor displays the Face



feedback symbol. The faces are displayed in the Mate Settings box. The Concentric Mate type is selected by default.

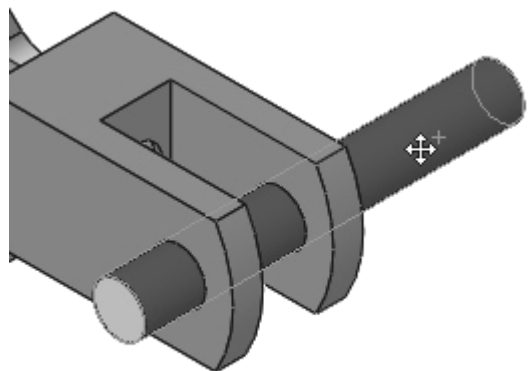
**242)** Click the **Preview** button. The AXLE is positioned concentric to the RodClevis hole

**243)** Utilize the **Undo** button if the Mate is not correct.

**244)** Click **OK**  to create a Concentric mate.


**245)** Right-click **AXLE** in the Graphics window.

**246)** Click **Move Component**.






**247)** Drag the **AXLE** left to right. The AXLE translates in and out of the RodClevis holes.



**248)** Drag the **AXLE** vertically and horizontally. The AXLE rotates about its axis.

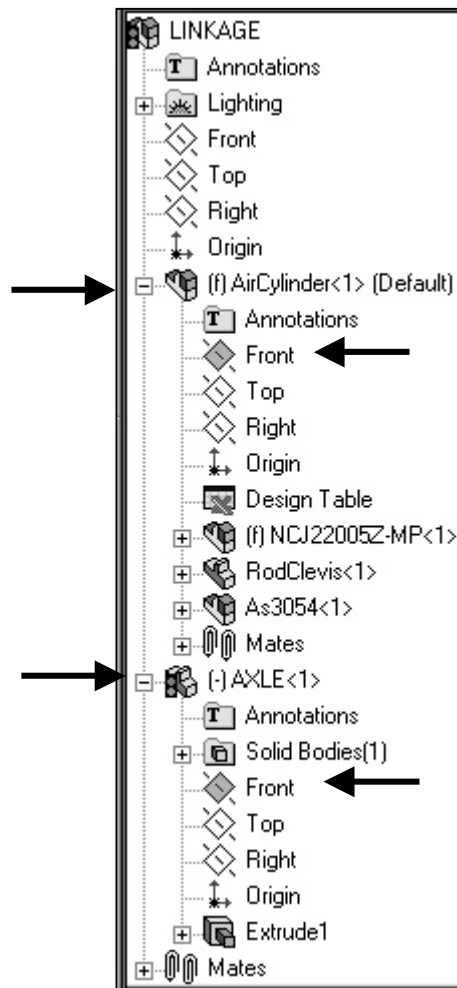
**249)** Click **OK**  to exit Move Component.

Insert a Coincident Mate.

**250)** Display the Top view. Click **Top**  from the Standard Views Toolbar.

**251)** Display the AirCylinder Planes. Click the **Plus**  icon in front of the FeatureManager name  (f) AirCylinder<1> (Default)

**252)** Display the AXLE Planes. Click the **Plus**  icon in front of the FeatureManager name  (-) AXLE<1>



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

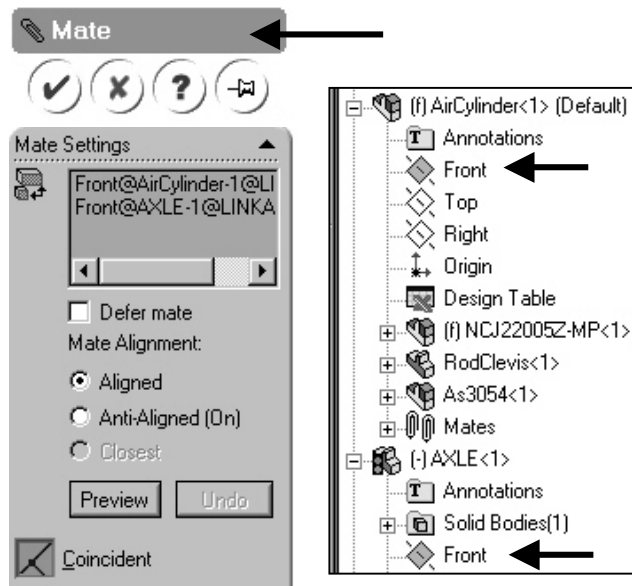
**254)** Click the large **Mate**



button to display the FeatureManager.

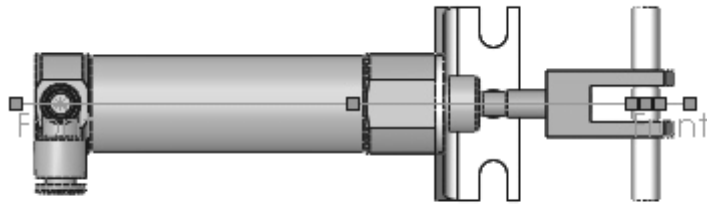
**255)** Click the **Front** plane below the AirCylinder entry in the FeatureManager.

**256)** Click the **Front** plane below the AXLE entry in the FeatureManager.

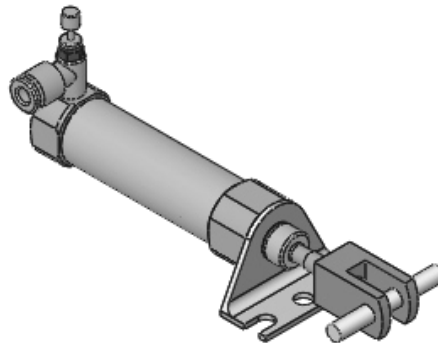


**257)** Click the **Preview** button. The Coincident Mate type is selected by default

**258)** Click **OK**  to create the Coincident Mate.



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



Display the Isometric view.

**259)** Click **Isometric** .

**260)** Save the LINKAGE Assembly. Click **Save** .

**261)** Click **Yes** to the question, "Save the documents and the referenced models now."

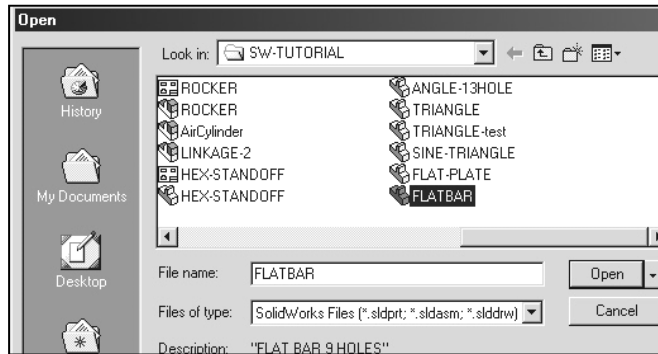
Open the FLATBAR part.

**262)** Click **Open** .

**263)** Select **Part** from the Files of type list.


**264)** Click **FLATBAR** for File name.

**265)** Click **Open**. The FLATBAR part is displayed in the Graphics window.




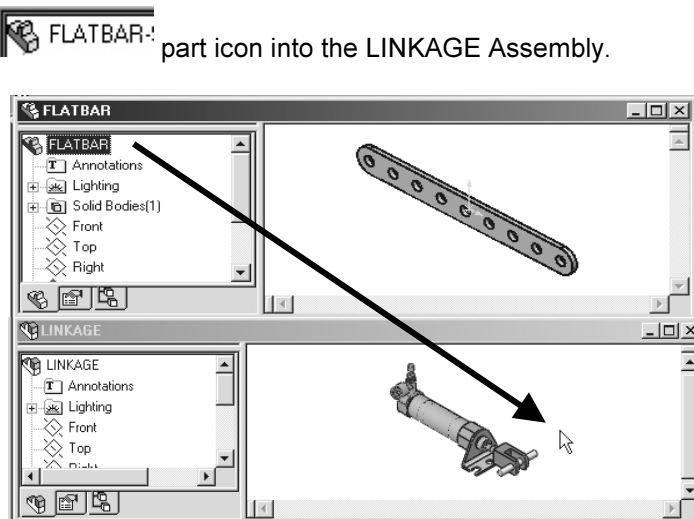
Insert the first FLATBAR.

**266)** Click **Window** from the Main toolbar. Click **Tile Horizontally**.

**267)** Drag the **FLATBAR**  part icon into the LINKAGE Assembly.

**268)** Click a **position** to the right of the AirCylinder Assembly.

**269)** Maximize the LINKAGE Assembly Graphics window. Click the **Maximize**  button.



**270)** Fit the model to the Graphics window. Press the **f** key.

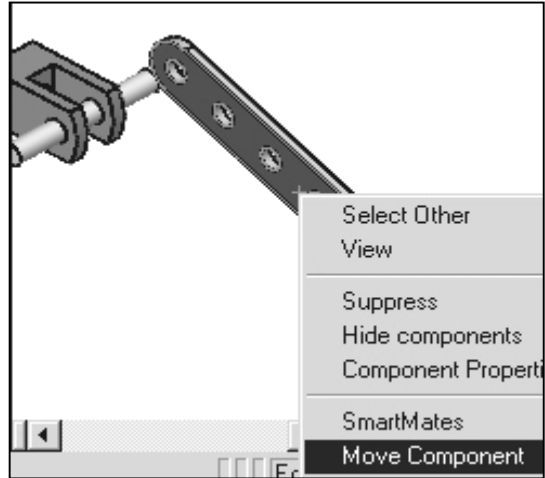
Move Component.

**271)** Right-click on the **FLATBAR** in the LINKAGE Assembly Graphics window.

**272)** Click **Move Component**.

**273)** Drag the **FLATBAR** until the left hole is positioned behind the **AXLE**.

**274)** Click **OK**  to exit Move Component.



Enlarge the view.

**275)** Click **Zoom to Area** .

**276)** **Zoom-in** on the **AXLE** and the left side of the **FLATBAR** to enlarge the view.

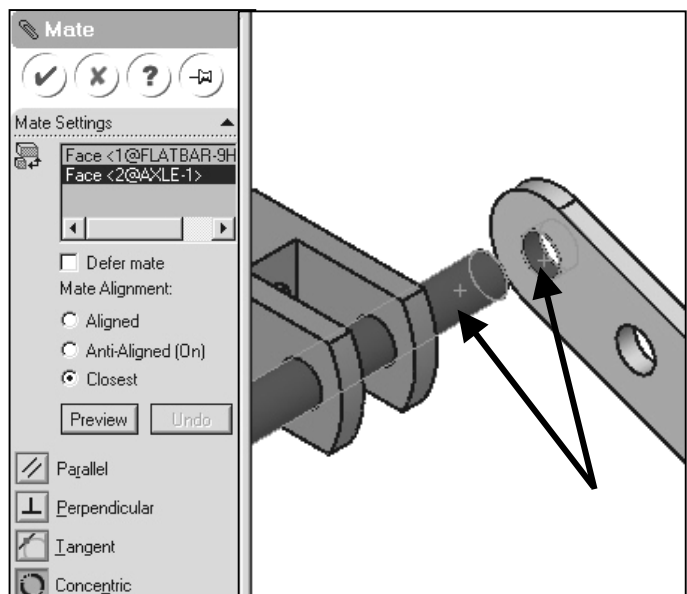
Insert a Concentric Mate.


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

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

**279)** 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.

**280)** Click the **Preview** button.



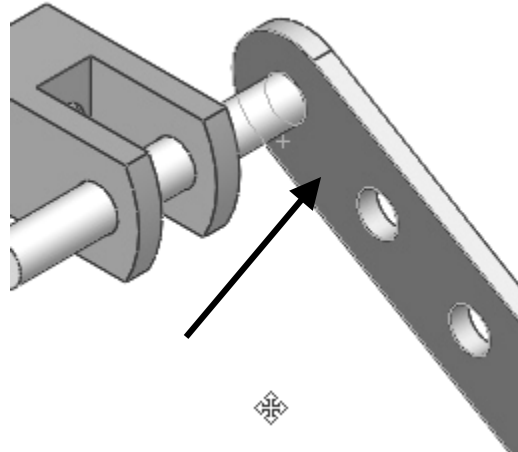
**281)** Click **OK**  to create the Concentric Mate.

Position the FLATBAR.

**282)** Click **Move Component**.

**283)** Drag the **FLATBAR**. The FLATBAR translates and rotates along the AXLE.

**284)** Click **OK**  to exit Move Component.



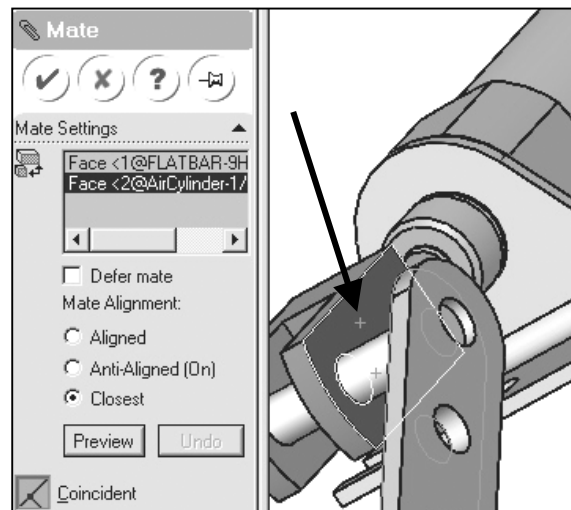
Insert a Coincident Mate.

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

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

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

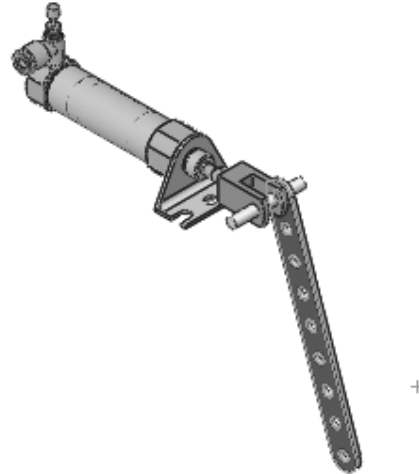
**288)** Click the **back face** of the RodClevis. The faces are displayed in the Mate Settings box. The Coincident Mate type is selected by default.



289) Click the **Preview** button.

290) Click **Isometric** 


291) Click **OK**  to create the Coincident Mate.

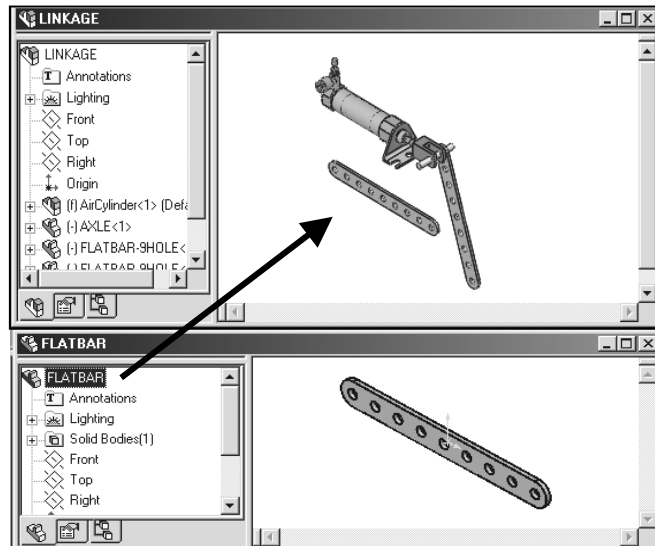


Insert the second FLATBAR.


292) Click **Window** from the Main toolbar.


293) Click **Tile Horizontally**.

294) Drag the **FLATBAR**  part icon into the LINKAGE Assembly.



295) Click a **position** to the left of the AirCylinder.

296) Click **Minimize**  in the FLATBAR Graphics window.

297) Click **Maximize**  in the LINKAGE Graphics window.




Enlarge the view.

**298)** Click **Zoom to Area** 

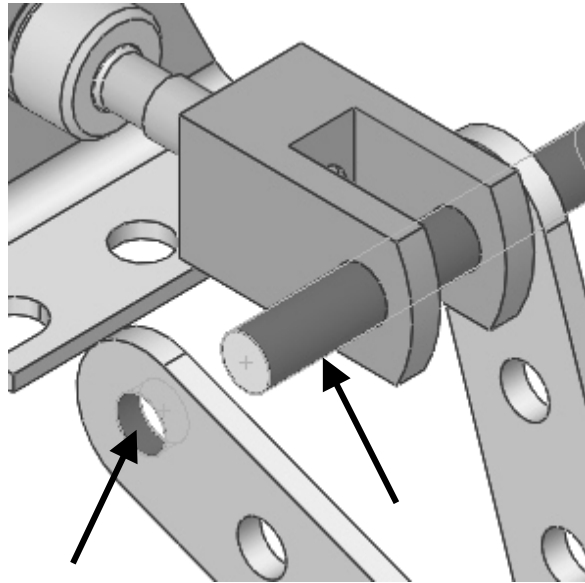
**299)** **Zoom-in** on the second FLATBAR and the AXLE.

Insert a Concentric Mate.


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

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


**302)** 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.



**303)** Click the **Preview** button.

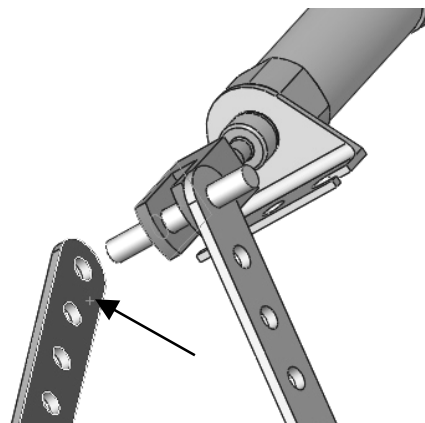
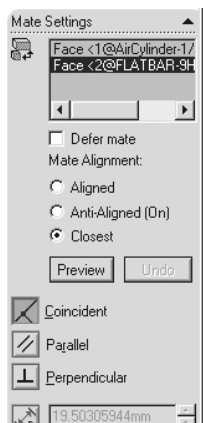
**304)** Click **OK**  to create the Concentric Mate.

Insert a Coincident Mate.

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

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

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




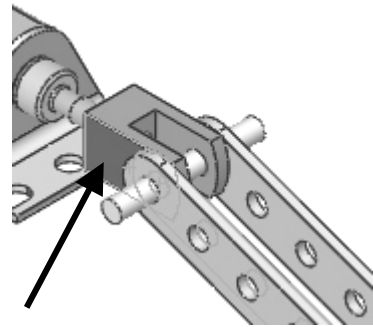
Display the Isometric view.

**308)** Click **Isometric** .

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

**310)** Click the **Preview** button.

**311)** Click **OK**  to create the Coincident Mate.



Insert a Parallel Mate.

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

**313)** Press the **Shift-z** keys 8 times to Zoom in on the model.

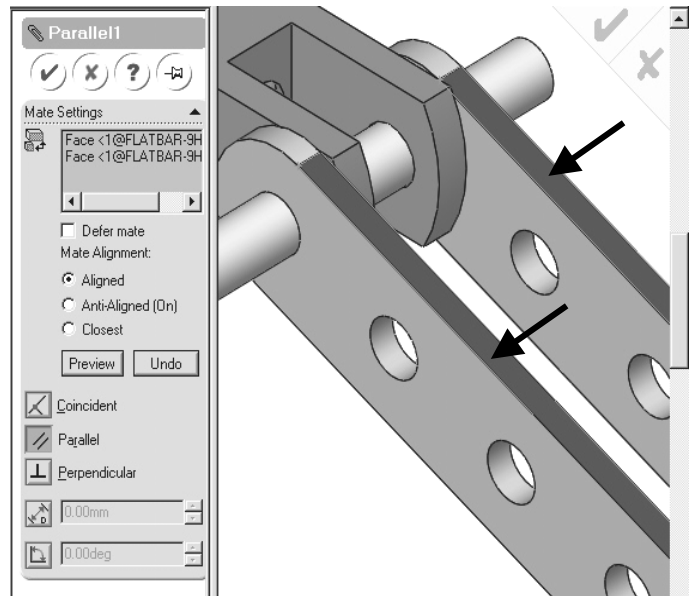
**314)** Click the **narrow top face** of the first FLATBAR.

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

**316)** Click **Parallel**.

**317)** Click the **Preview** button.


**318)** Click **OK**  to create the Parallel Mate.



Move the two FLATBAR parts.

**319)** Right-click the second **FLATBAR** in the Graphics window.

**320)** Click **Move Component**. Both FLATBAR parts move together.

**321)** Click **OK**  to exit Move Component.

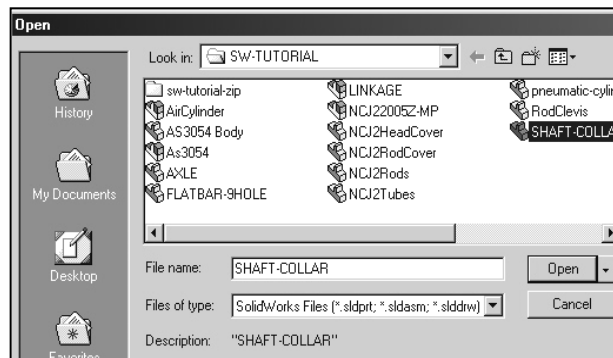
Open the SHAFT-COLLAR part.

**322)** Click **Open** .

**323)** Select **Part** from the Files of type list.


**324)** Click **SHAFT-COLLAR** for File name.

**325)** Click **Open**. The SHAFT-COLLAR part is displayed in the Graphics window.

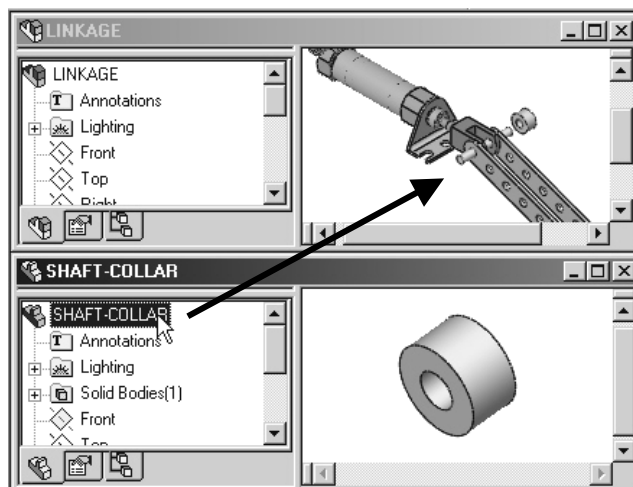



Insert the first SHAFT-COLLAR

**326)** Click **Window** from the Main toolbar. Click **Tile Horizontally**.

**327)** Drag the **SHAFT-COLLAR**  part icon into the LINKAGE Assembly.

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



**329)** Maximize the LINKAGE Assembly Graphics window. Click the **Maximize**  button.

**330)** Fit the model to the Graphics window. Press the **f** key.

Enlarge the view.

**331)** Click **Zoom to Area** 

**332)** **Zoom-in** on the first SHAFT-COLLAR the AXLE to enlarge the view.


Insert a Concentric Mate.

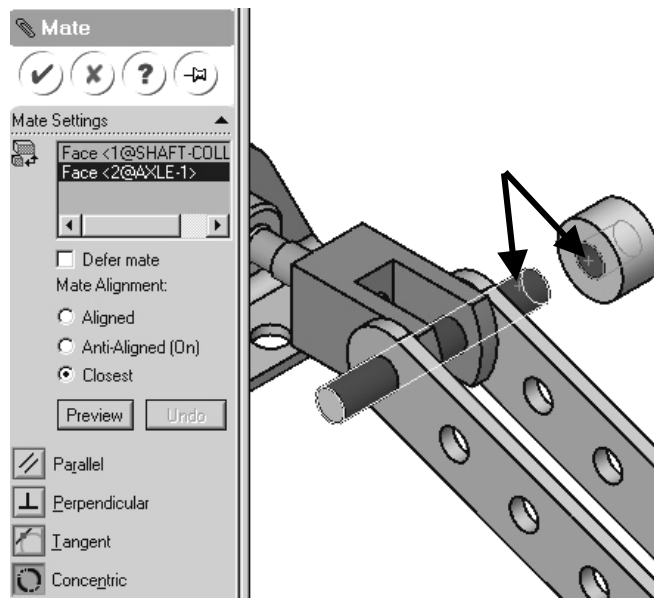
**333)** Click **Mate**  from the Assembly toolbar.

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


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

**336)** Click the **Preview** button.

**337)** Click **OK**  to create the Concentric Mate.

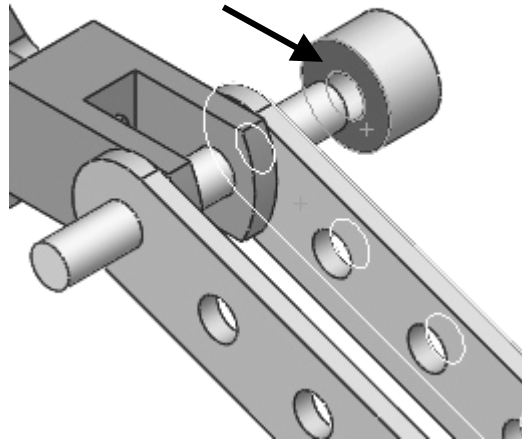


Insert a Coincident Mate.

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

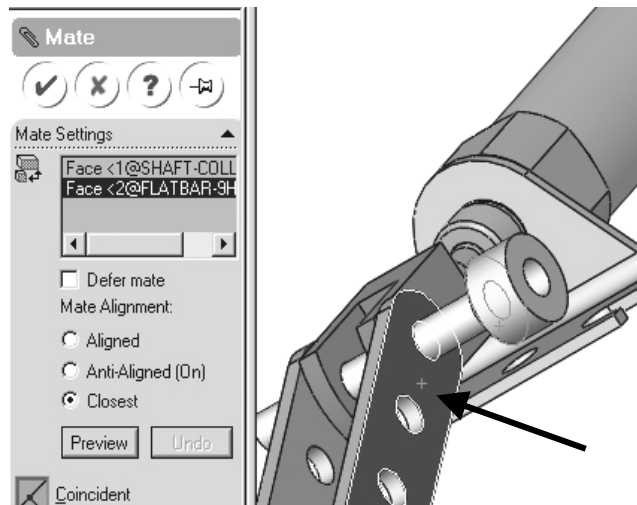
**339)** Press the **Shift-z** keys to Zoom in on the model.

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



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


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



Display the Isometric view.

**343)** Click **Isometric** . The Coincident Mate type is selected by default.

**344)** Click the **Preview** button.

**345)** Click **OK**  to create the Coincident Mate.

Insert the second SHAFT-COLLAR


**346)** Click **Window** from the Main toolbar.

**347)** Click **Tile Horizontally**.

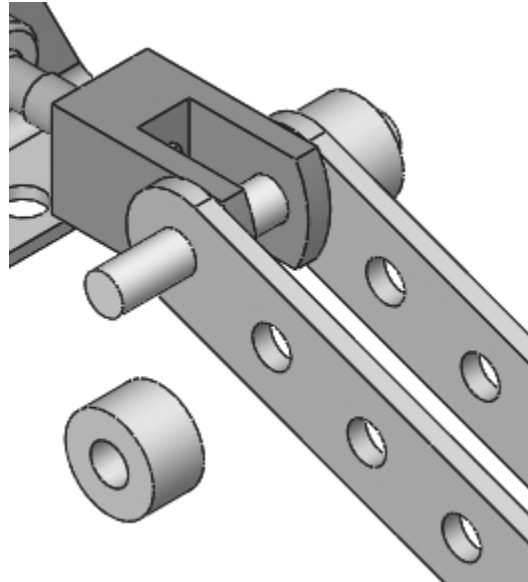
**348)** Drag the **SHAFT-COLLAR**

 **SHAFT-COLLAR** part icon into the LINKAGE Assembly.

**349)** Click a **position** to the left of the AXLE.

**350)** Maximize the LINKAGE Assembly Graphics window. Click the **Maximize**  button.

**351)** Fit the model to the Graphics window. Press the **f** key.



Enlarge the view.

**352)** Click **Zoom to Area** 

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


Insert a Concentric Mate.

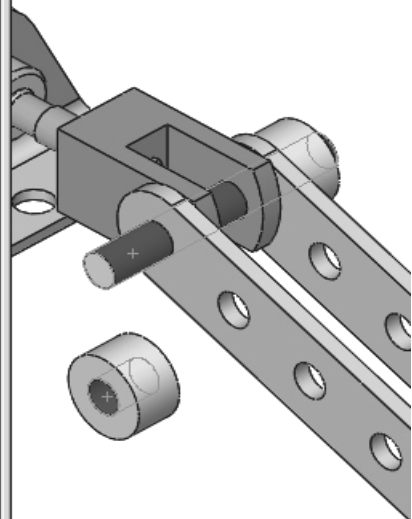
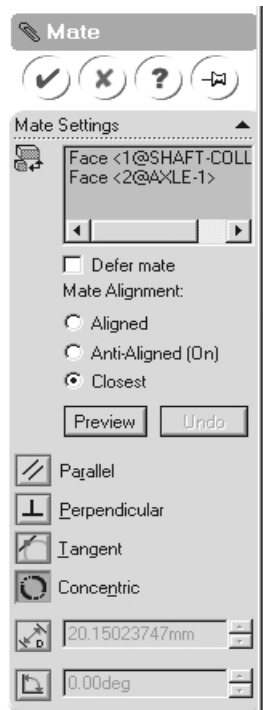
**354)** Click **Mate**  from the Assembly toolbar.

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


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

**357)** Click the **Preview** button.

**358)** Click **OK**  to create the Concentric Mate.

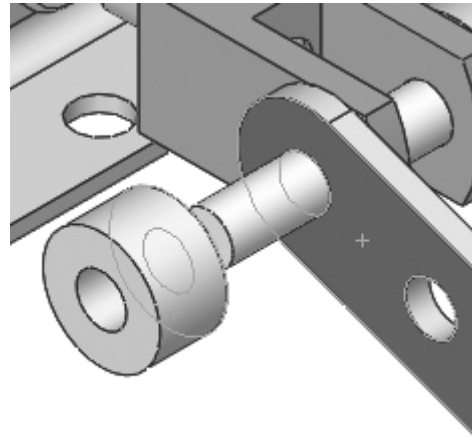


Insert a Coincident Mate.

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

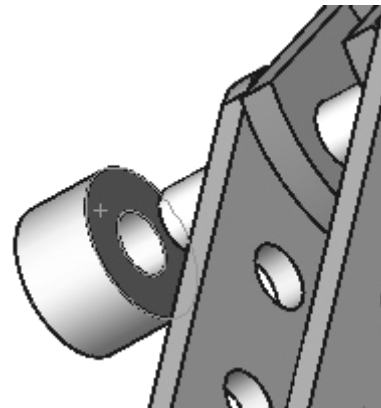
**360)** Press the **Shift-z** keys to Zoom in on the model.

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



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

**363)** Click the **back face** of the first SHAFT-COLLAR.



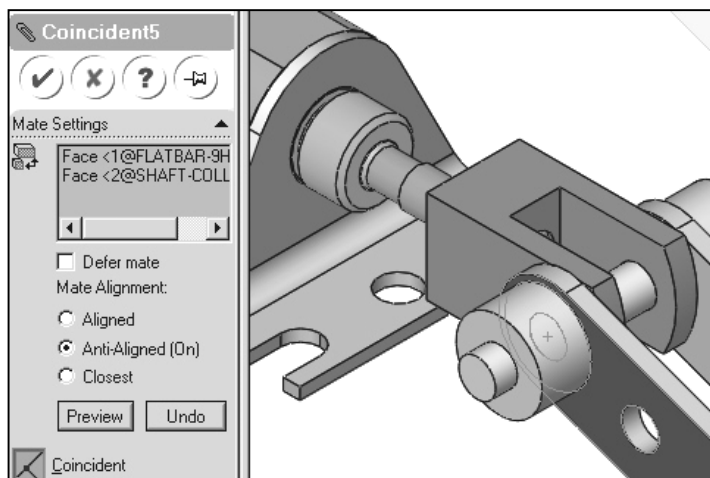
**364)** Click **Isometric**



The Coincident Mate type is selected by default.

**365)** Click the **Preview** button.

**366)** Click **OK**  to create the Coincident Mate.



**367)** Fit the model to the Graphics window. Press the **f** key.

**368)** Save the LINKAGE Assembly. Click **Save** .

**369)** Click **Yes** to the question, "Save the documents and the referenced models now."

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.

Mates are geometric relationships that align and fit components in an assembly. Select geometry from two different components. Select the Mate type. Always Preview the Mate. Utilize Undo if the Mate is not correct.

The AirCylinder sub-assembly is the first component inserted into the LINKAGE Assembly. The AirCylinder Assembly was downloaded and unzipped from the World Wide Web. The AirCylinder Assembly is fixed to the Origin.

The AXLE part is 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 is inserted into the LINKAGE Assembly.

Four SHAFT-COLLAR parts are inserted into the LINKAGE Assembly. Each SHAFT-COLLAR requires a Concentric Mate between 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.

Insert a Rotary Motor Physical Simulation Tool.


**370)** Click **View, Toolbars** from the Main menu.

**371)** Check the **Simulation** toolbar.

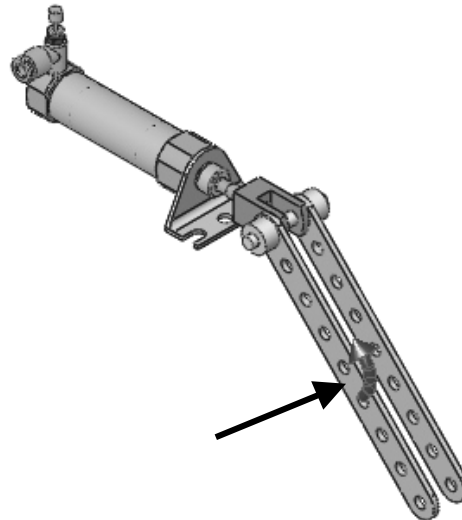


**372)** **Right-click** the front FLATBAR in the Graphics window.

**373)** Click **Move Component** to position the right end of the FLATBAR below the AirCylinder Assembly.

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


**375)** Click the front **FLATBAR front face**. A red Rotary Motor icon is displayed.



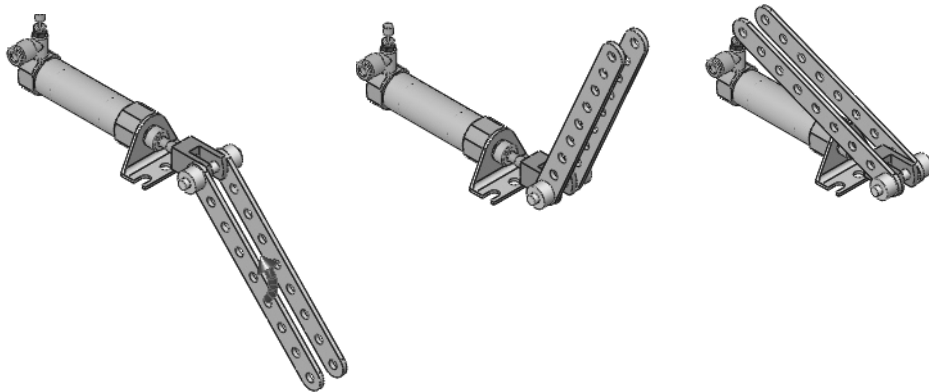
**376)** The red Direction arrow points counterclockwise. Position the **Velocity Slide bar** in the middle of the PropertyManager.

**377)** Click **OK** to apply the Rotary Motor.





**378)** Record the Simulation. Click **Record Simulation** .


**379)** Click **OK** to the warning message, "Model started in colliding position. Simulation will continue and ignore these collisions." The FLATBAR rotates in a counterclockwise direction until collision with the AirCylinder.





#### Linear Assembly Physical Simulation

**380)** Stop the Simulation. Click **Stop Simulation** .

**381)** Return to the starting position. Click **Reset Components** .

**382)** Replay the Simulation. Click **Replay** .

**383)** Stop the Simulation. Click **Stop Simulation** .

**384)** Return to the starting position. Click **Reset Components** .

**385)** Fit the assembly to the Graphics window. Press the **f** key.

**386)** Save the LINKAGE Assembly. Click **Save** .

**387)** Exit **SolidWorks**. Click **File**. Click **Exit**.

The LINKAGE Assembly project is complete.

Review the Physical Simulation.

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

Apply a Rotary Motor to the face of the FLATBAR.

Set the rotation direction and velocity.

Utilize Record to record the simulation. Utilize Play to view the simulation.

**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, Templates, 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 Extrude 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 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

**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.

**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.

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

**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.

**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.

**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 and ellipses.

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

Under Defined: There is inadequate definition of the sketch, (Blue).

Fully Defined: Has complete information, (Black).

Over Defined: Has duplicate dimensions, (Red).

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

**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.

**Dimension:** A value indicating the size of feature geometry.

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

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

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

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

**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.

**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.

### **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.

**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.

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

**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.

a) Determine the radius of the AXLE in mm.

b) Determine the height of the AXLE in mm.

c) Calculate the Volume of the AXLE in  $\text{mm}^3$ .

_____
_____
_____

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

- $\rho$  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.

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

Utilize the table above to determine the following questions:

- a) How many 1-3/8 inch AXLES can be cut from each steel rod?
- b) Twenty AXLE parts are required for a new assembly. What length of Harden Steel Rod should be purchased?

_____
_____
_____
_____
_____
_____
_____
_____
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_____



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.

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

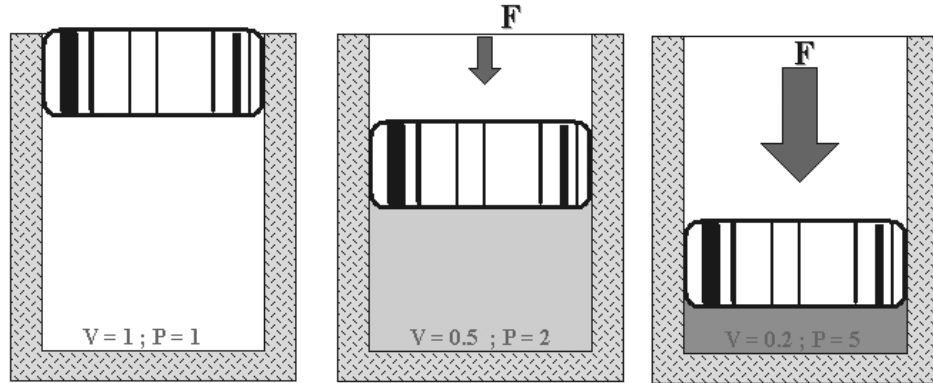


Illustration of Boyle’s Law  
 Courtesy of SMC Corporation of America

a) 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.

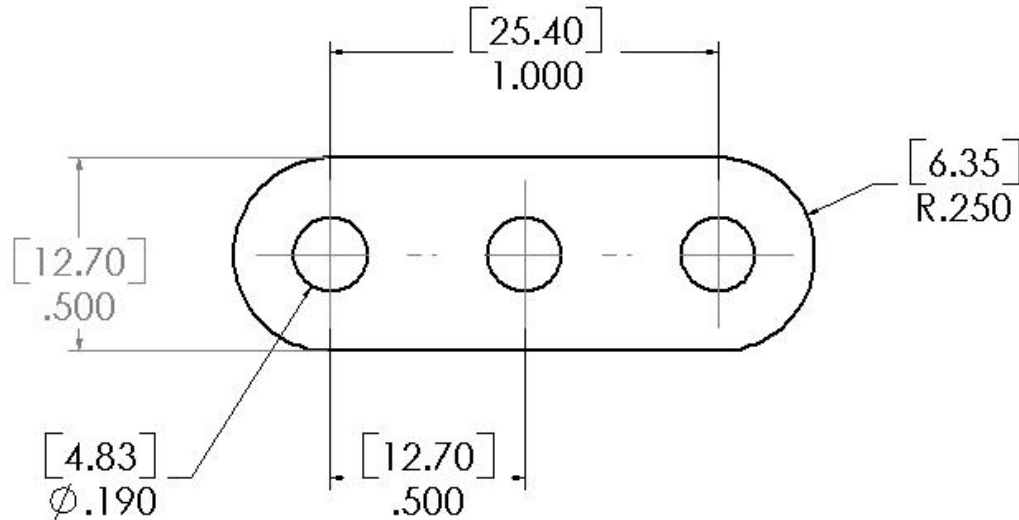
b) Research other contributions made by Robert Boyle and Robert Hooke that are utilized today.

**Questions**

1. Explain the steps in starting a SolidWorks session.
2. Describe a part.
3. Identify the three default reference planes.
4. Identify the default sketch plane.
5. What is the Base feature? Provide an example.
6. Describe the differences between an Extruded-Base feature and an Extruded-Cut feature.
7. The sketch color, black indicates a sketch is \_\_\_\_\_ defined.
8. The sketch color, blue indicates a sketch is \_\_\_\_\_ defined.
9. The sketch color, red indicates a sketch is \_\_\_\_\_ defined.
10. Identify the type of Geometric Relations that can be added to a Sketch.
11. Describe an assembly or sub-assembly.
12. What are Mates and why are they important in assembling components?
13. Describe Dynamic motion.
14. In an assembly, each component has \_\_\_\_\_ # degrees of freedom? Name them.
15. True or False. A fixed component cannot move and is locked to the Origin.

**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.

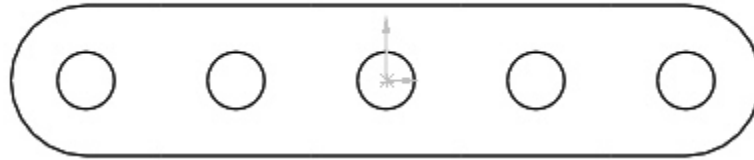


FLATBAR-3HOLE  
Courtesy of GEARS Educational Systems, LLC  
Hanover, MA USA  
[www.gearseds.com](http://www.gearseds.com)

**Exercise 1.2: FLATBAR-5HOLE.**

Design the FLATBAR-5HOLE part.

Manually add dimensions to the 2D profile.



FLATBAR-5HOLE  
Courtesy of GEARS Educational Systems, LLC  
Hanover, MA USA  
[www.gearseds.com](http://www.gearseds.com)

The dimensions for hole spacing, height and end arcs are the same as the FLATBAR-3HOLE part in Exercise 1.1.

Create the part. Utilize the Front Plane for the Sketch Plane. Utilize the Linear Pattern feature to create the hole pattern.

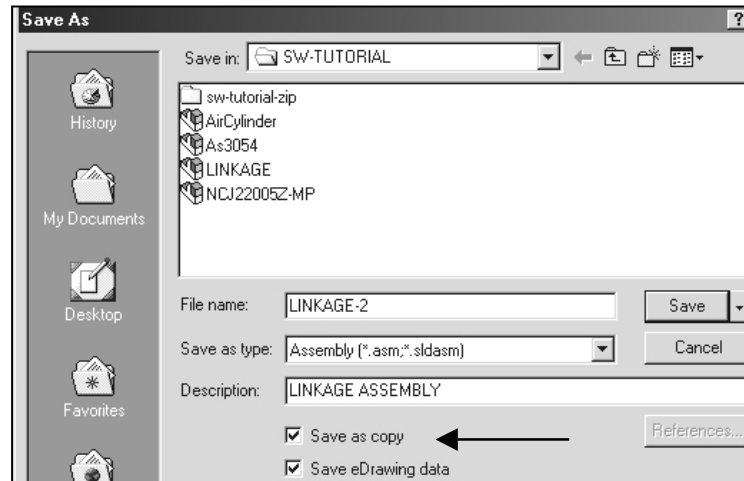
The FLATBAR-5HOLE part is manufactured from 0.060in [1.5mm] Stainless Steel.

**Exercise 1.3a: LINKAGE-2 Assembly.**

Create the LINKAGE-2 Assembly.

Open the LINKAGE Assembly.

Select File, Save As from the Main menu.

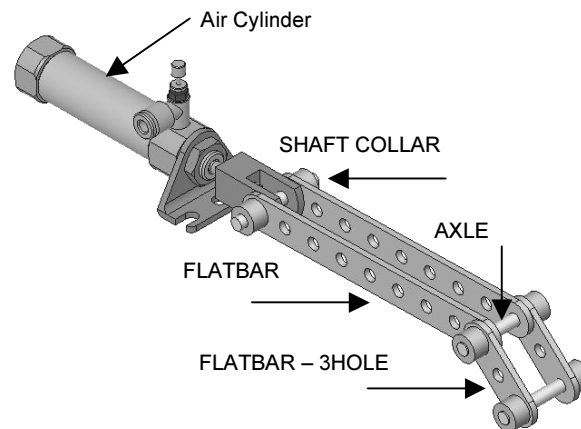


Check the Save a Copy check box.

Enter LINKAGE-2 for file name.

The FLATBAR-3HOLE parts were created in Exercise 1.1.

Utilize 2 AXLE parts, 4 SHAFT COLLAR parts and 2 FLATBAR-3HOLE parts.



Linkage2 Assembly  
Courtesy of  
Gears Educational Systems and  
SMC Corporation of America.

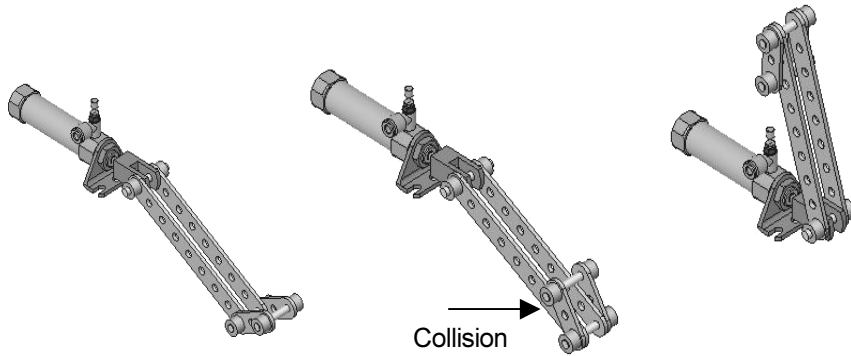
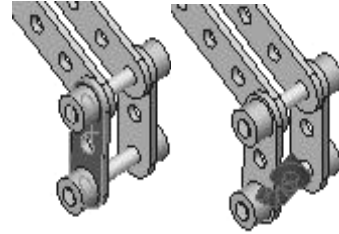
**Exercise 1.3b: LINKAGE-2 Assembly Simulation.**

Create the LINKAGE-2  
Assembly simulation

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

Record the Simulation.

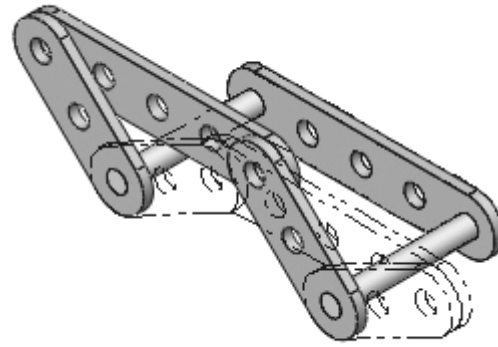
Play the Simulation.



LINKAGE-2 Assembly Simulations

### Exercise 1.4a: ROCKER Assembly.

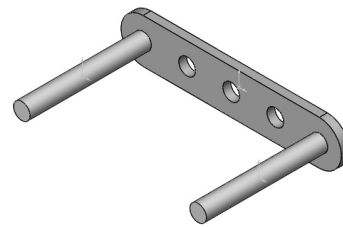
Create the ROCKER Assembly. The ROCKER Assembly consists of 2 AXLE parts; 2 FLATBAR-5HOLE parts, and 2 FLATBAR-3HOLE parts.



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

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

Insert the first FLATBAR-5HOLE part. The FLATBAR-5HOLE is fixed to the Origin of the ROCKER Assembly.



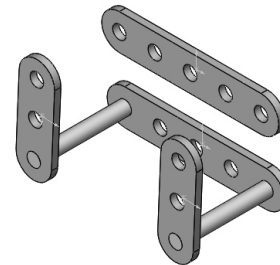
Insert the first AXLE, concentric with the FLATBAR-5HOLE, left hole.

Utilize Concentric and Coincident Mates.

Insert the second AXLE concentric with the FLATBAR-5HOLE, right hole. Utilize Concentric and Coincident Mates.

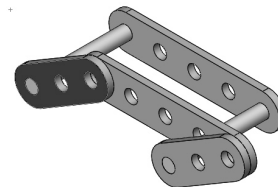
Insert the first FLATBAR-3HOLE concentric with the first AXLE. Utilize Concentric and Coincident Mates.

Insert the second FLATBAR-3HOLE concentric with the second AXLE. Utilize Concentric and Coincident Mates.



Utilize Move Component and position both FLATBAR-3HOLE parts in a vertical position.

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



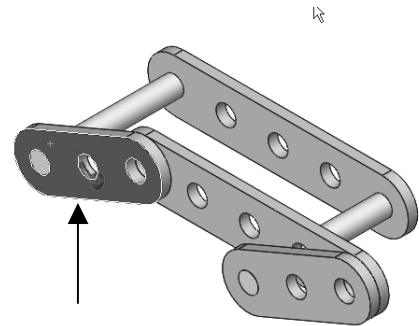
Utilize Move Component to “rock” the FLATBAR-3HOLE part.

**Exercise 1.4b: ROCKER Assembly Physical Simulation.**

Create the ROCKER Physical Simulation.

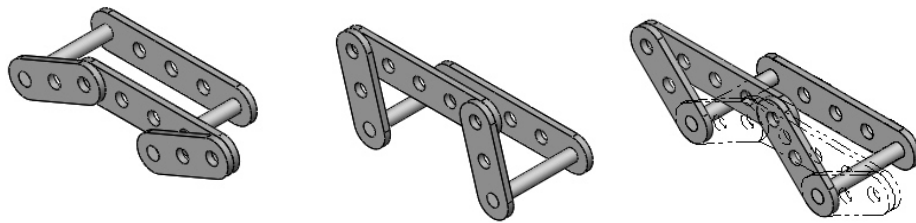
Utilize Move Component and move the FLATBAR-3HOLE part to the right of the AXLE.

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.



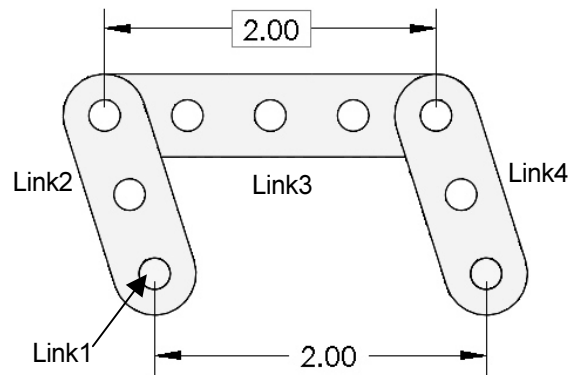
ROCKER Assembly Physical Simulation

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.





**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](http://mikejwilson.com) and [zxys.com](http://zxys.com).

*The models obtained from these web sites for educational purposes only. Learn modeling techniques from others; create your own designs.*



Model Courtesy of  
Mike J. Wilson, CSWP

Scooby Doo® is a registered  
trademark of Hanna-Barbera



Model Courtesy of Paul Salvador, [http:// zxys.com/](http://zxys.com/) "Colgate Navigator"  
Colgate Navigator is a registered trademark of Colgate-Palmolive Company

A common manufacturing procedure for plastic parts is named the Injection Molding Process. Today's automobiles utilize over 50% plastic components.



Model Courtesy of Paul Salvador, <http://zxys.com/>  
"Audi TT by Audi"

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

Review the Injection Molding Process for a small toy car.

### The Injection Molded Process.

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

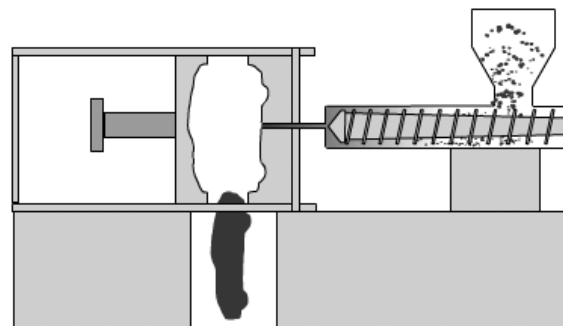
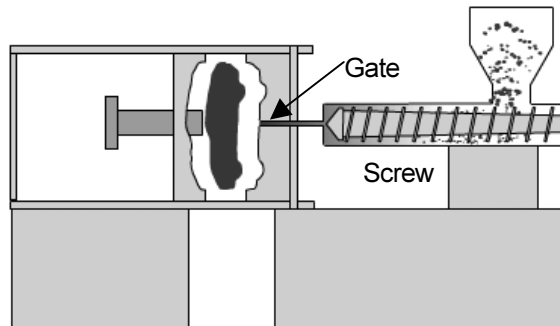
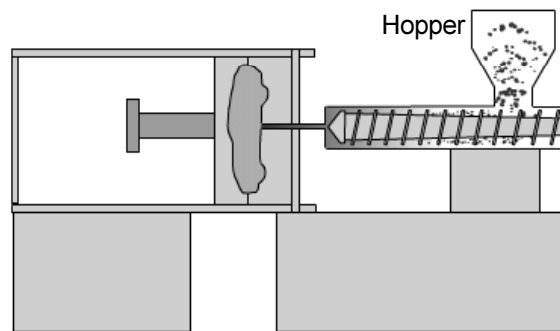
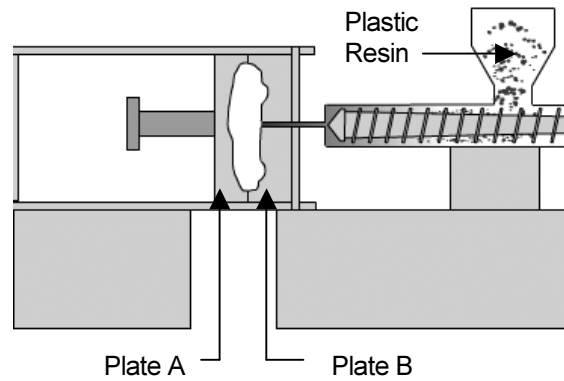
The injection molding process is as follows:

An operator pours the plastic resin in the form of small dry pellets, into a hopper. The hopper feeds a large augur 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.)