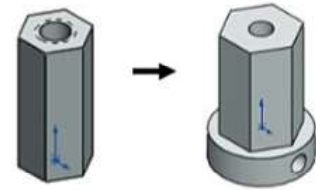


## Modify a Part

Conserve design time and cost. Modify existing parts and assemblies to create new parts and assemblies. Utilize the Save as copy tool to avoid updating the existing assemblies with new file names.

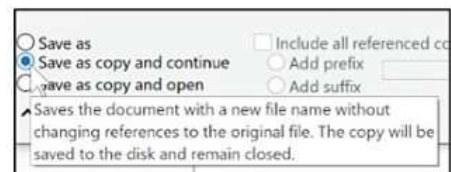


The HEX-STANDOFF part was created in a previous chapter. The HEX-ADAPTER is required to fasten the WHEEL to the AXLE. Start with the HEX-STANDOFF part.

Utilize the Save As command and enter the HEX-ADAPTER for the new file name. Check the Save as copy and continue check box. The HEX-ADAPTER is the new part name. Open the HEX-ADAPTER. Modify the dimensions of the Extruded Base feature.



Utilize Edit Definition to modify the Hole Wizard Tap Hole to a Standard Hole. Insert an Extruded Boss/Base feature to create the head of the HEX-ADAPTER.



Insert an Extruded Cut feature. Sketch a circle on the Right Plane. Extrude the circle in Direction1 and Direction2 with the Through All End Condition option. Note: You can use the Hole Wizard feature with a 3D Sketch. Feature order determines the internal geometry of the Hole. If the Hole feature is created before the Extrude2-Head feature, the Through All End Condition will extend through the Boss-Extrude1 feature.

If the Hole feature is created after the Extrude2-Head feature, the Through All End Condition will extend through the Boss-Extrude1 feature and the Extrude2-Head feature.

Modify feature order by dragging feature names in the FeatureManager. Utilize the Save As command to create the AXLE3000 part from the AXLE part.

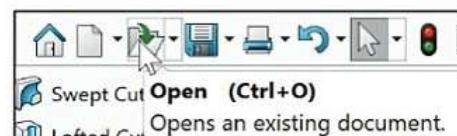
Utilize the Save As command to create the SHAFTCOLLAR-500 part from the SHAFT-COLLAR part. Save the HEX-STANDOFF as the HEX-ADAPTER part.

### Activity: HEX-ADAPTER Part

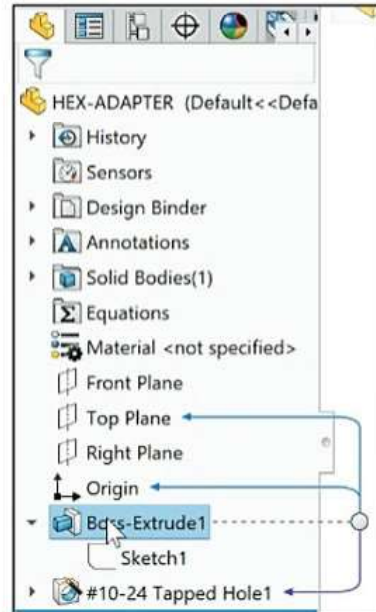
Create the HEX-ADAPTER.

**379)** Click **Open**  from the Menu bar.

**380)** Double-click **HEX-STANDOFF**. The HEX-STANDOFF FeatureManager is displayed. Note: HEX-STANDOFF was created in a previous chapter. Click **Save As** .



- 381) Select the **SW-TUTORIAL-2020** folder.
- 382) Enter **HEX-ADAPTER** for File name.
- 383) Enter **HEX-ADAPTER 10-24** for Description.
- 384) Check the **Save as copy and continue** box.
- 385) Click **Save**.
- 386) Click **File, Close** from the Menu bar.

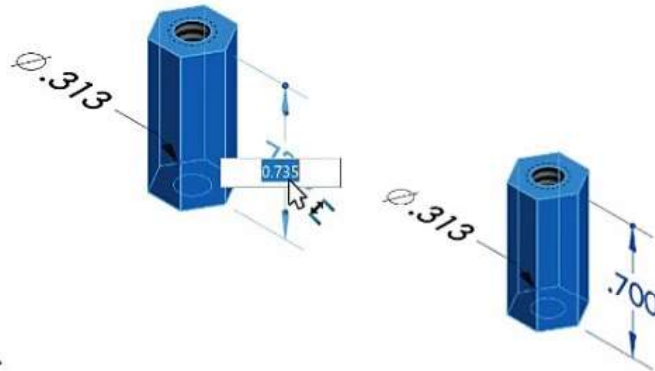


Open the HEX-ADAPTER.

- 387) Click **Open** from the Menu bar.
- 388) Double-click **HEX-ADAPTER**. The HEX-ADAPTER FeatureManager is displayed.

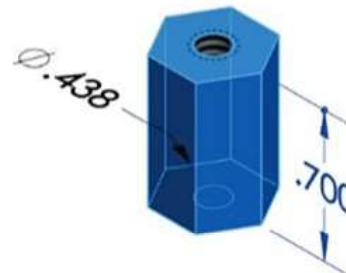
Modify the Boss-Extrude1 dimensions.

- 389) Double-click **Boss-Extrude1** from the FeatureManager.
- 390) Move the **dimensions** off the model.
- 391) Double-click **.735in, [18.67]**.
- 392) Enter **.700in, [17.78]** for depth.
- 393) Double-click **.313in, [7.95]**.
- 394) Enter **.438in, [11.13]** for diameter.



Modify the #10-24 Tapped Hole1 feature.

- 395) Right-click **#10-24 Tapped Hole1** from the FeatureManager.
- 396) Click **Edit Feature** from the Context toolbar. The Hole Specification PropertyManager is displayed.



The Hole Specification PropertyManager is part of the Hole Wizard tool located in the Features toolbar.

The Type tab is selected by default.


**397)** Select the **Hole** tab from the Hole Specification box.

**398)** Select **ANSI Inch** for Standard.

**399)** Select **Tap Drills** for Type.

**400)** Select **#10-24** for Size.

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

**402)** Click **OK**  from the Hole Specification PropertyManager. The Tap Hole is modified.

Insert a sketch for the Extruded Boss feature.

**403)** **Rotate** the model to view the bottom face.

**404)** Right-click the **bottom hexagonal face** of the Boss-Extrude1 feature as illustrated.

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

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

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

**408)** Click the **Origin**  as illustrated.

**409)** Click a **position** in the Graphics window to the right of the Origin.

Add a dimension.

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

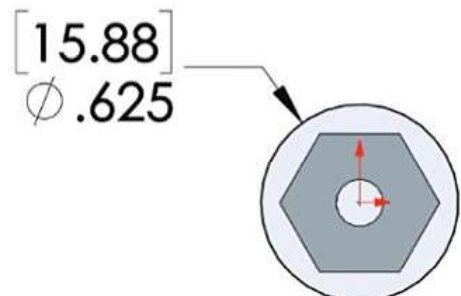
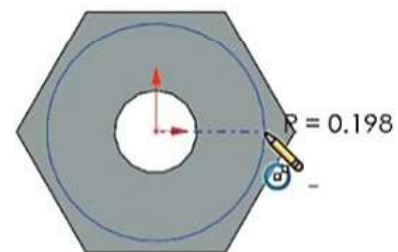
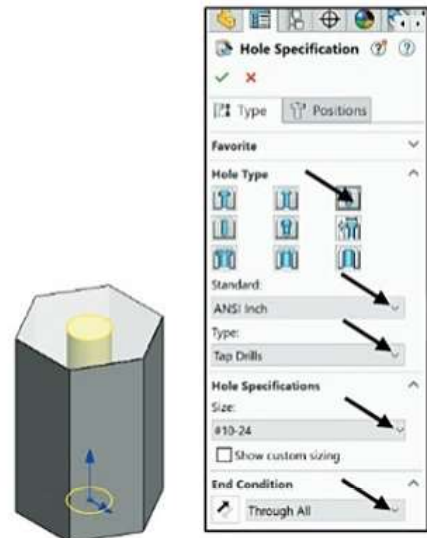
**411)** Click the **circumference** of the circle.

**412)** Click a **position** above the circle to locate the dimension.

**413)** Enter **.625in**, [**15.88**] in the Modify dialog box.

Fit the model to the Graphics window.

**414)** Press the **f** key.



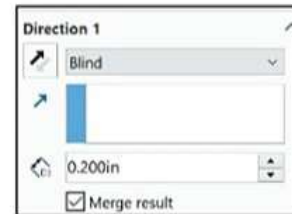


### Activity: HEX-ADAPTER Part - Extruded Boss/Base Feature

Extrude the sketch to create the Extruded Boss/Base feature.

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

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

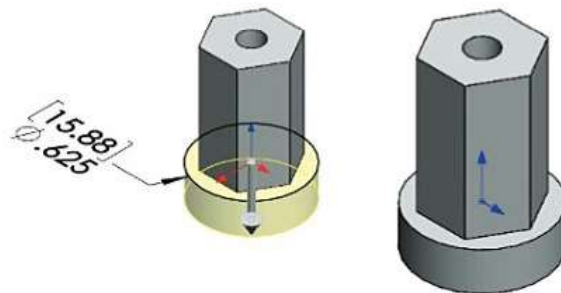



417) Enter .200in, [6.35] for Depth. The Direction arrow points downward. Flip the **Direction arrow** if required.

418) Click **OK**  from the Boss-Extrude PropertyManager. Boss-Extrude2 is displayed in the FeatureManager.

Rename Boss-Extrude2. Save the model.

419) Rename **Boss-Extrude2** to **Extrude2-Head**.



420) Click **Save** .




Rename a feature or sketch for clarity. Slowly click the feature or sketch name twice and enter the new name when the old one is highlighted.

### Activity: HEX-ADAPTER Part - Extruded Cut Feature


Insert a new sketch for the Extruded Cut on the Right Plane.

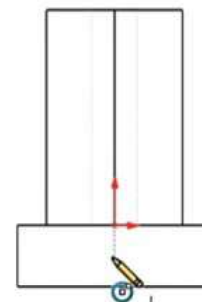
421) Right-click **Right Plane** from the FeatureManager.

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

423) Click **Right view**  from the Heads-up View toolbar. Note the location of the Origin.

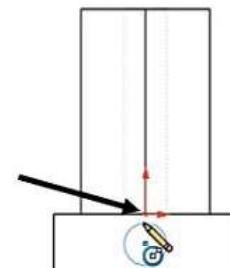
424) Click **Hidden Lines Visible**.

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



426) Sketch a circle below the Origin . The center point is vertically aligned to the Origin (vertical relation).

427) If required, add a **Vertical relation** between the center point of the circle and the Origin.



Add dimensions.

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

429) Click the **middle horizontal edge**.

430) Click the **center point** of the circle.

431) Click a **position** to the right of the profile.

432) Enter **.100in, [2.54]**.

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

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

435) Click a **position** below the profile.

436) Enter **.120in, [3.95]**.

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


Insert an Extruded Cut feature.

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

439) Select **Through All - Both** for End Condition in Direction 1.

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

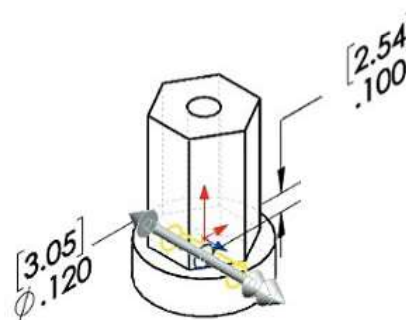
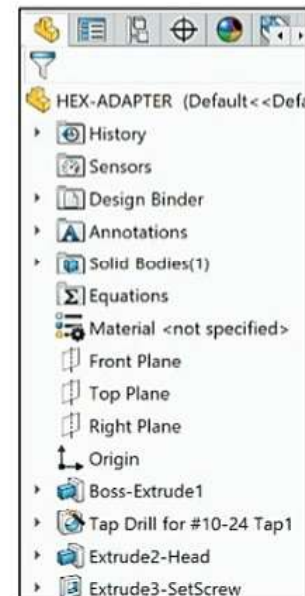
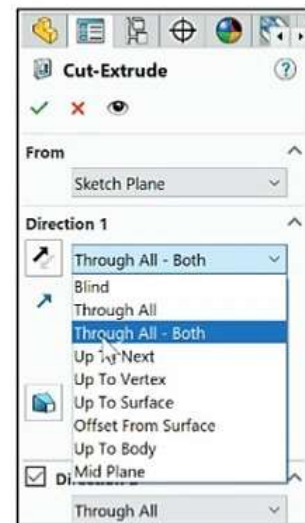
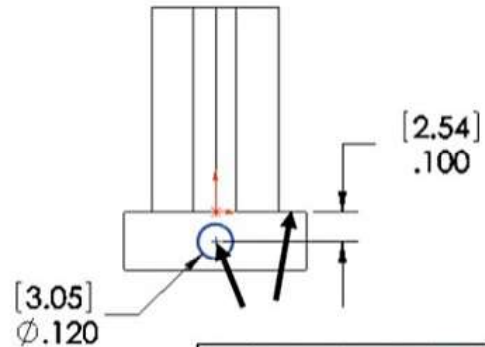
Display an Isometric view. Rename the feature.

441) Click **Isometric view** .

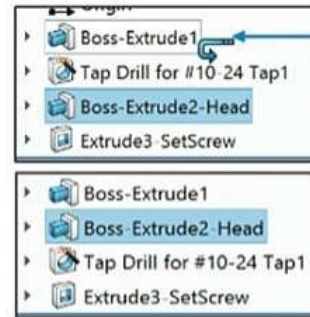
442) Rename the Cut-Extrude# feature to **Extrude3-SetScrew**.

Save the HEX-ADAPTER part.

443) Click **Save** .



The Through All End Condition is required to penetrate both the Boss-Extrude1 and Extrude2 features. Reorder features in the FeatureManager. Position the Extrude2 feature before the Tap Drill for # 10-24 Tap 1 feature in the FeatureManager.




Reorder the Features.

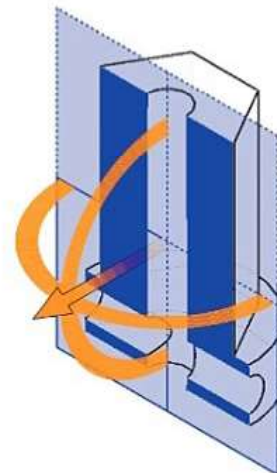
**444)** Click and drag **Extrude2-Head** from the FeatureManager upward as illustrated.


**445)** Click a **position** below Boss-Extrude1. The Through All End Condition option for the Tap Drill for # 10-24 Tap 1 feature creates a hole through both Boss-Extrude1 and Boss-Extrude2.

Display a Section view.


**446)** Click **Front Plane** from the FeatureManager.

**447)** Click **Section view**  from the Heads-up View toolbar in the Graphics window. The Section View PropertyManager is displayed. View the results.




**448)** Click **OK**  from the Section View PropertyManager.

Display the full view.

**449)** Click **Section view**  from the Heads-up View toolbar in the Graphics window.

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

Save the HEX-ADAPTER.

**451)** Click **Save** . Note the location of the Origin in the model.

Close all documents.

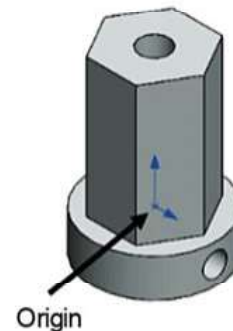
**452)** Click **Windows, Close All** from the Menu bar.



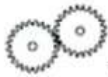
Utilize the Save As command and work on the copied version of the document before making any changes to the original. Keep the original document intact.



Tangent edges and Origins are displayed for educational purposes in this book.





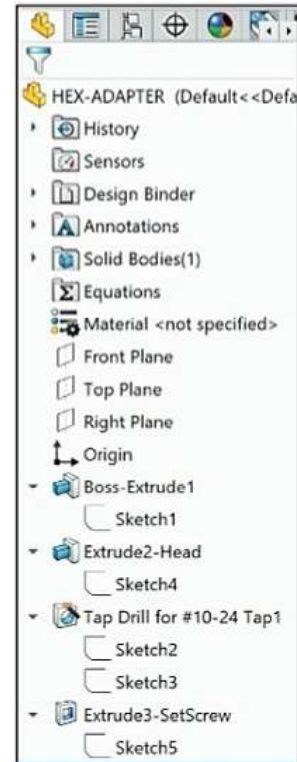



Review the HEX-ADAPTER Part


The HEX-ADAPTER part was created by utilizing the Save As and the Save as copy command with the HEX-STANDOFF part. The Boss-Extrude1 feature dimensions were modified. Edit Definition was utilized to modify the Hole type from the Hole Wizard feature.

An Extruded Boss feature added material. An Extruded Cut feature, sketched on the Right Plane with the Through All End Condition for both Direction1 and Direction2, created a hole through the Extruded Boss feature. Reordering features in the FeatureManager modified the Hole. Utilizing existing geometry saved time with the Save as copy command. The original part and its references to other assemblies are not affected with the Save as copy command.

You require additional work before completing the PNEUMATIC-TEST-MODULE assembly. The AXLE and SHAFT-COLLAR were created in Chapter 2. Utilize the Save as copy command to save the parts.



 Additional details on Save (Save As copy), Reorder (features), Section View PropertyManager are available in SOLIDWORKS Help.

 Utilize Design Table configurations for the AXLE part and SHAFT-COLLAR part developed in the previous chapter.



The AXLE-3000 part and SHAFT-COLLAR-500 part utilize the Save As option in the next section.

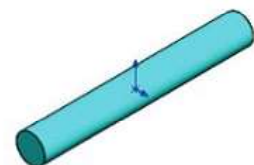
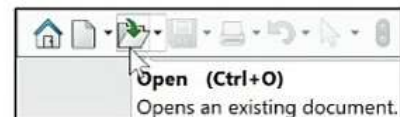
Utilize the Save As components or the configurations developed with Design Tables to create the WHEEL-AXLE assembly.


**Activity: AXLE-3000 Part**

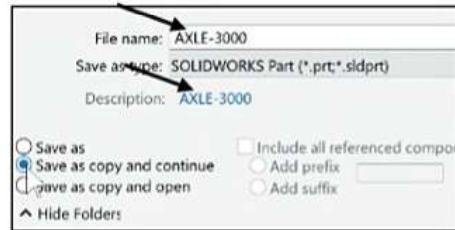
Create the AXLE-3000 part from the AXLE part.

**453)** Click **Open**  from the Menu bar.

**454)** Double-click **AXLE** from the SW-TUTORIAL-2020 folder. The AXLE FeatureManager is displayed.




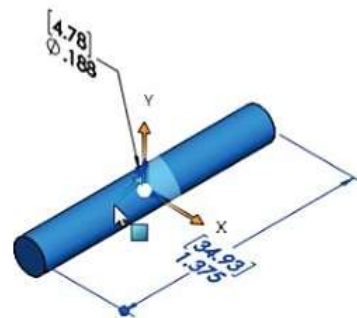
- 455) Click **Save As** .
- 456) Select the **SW-TUTORIAL-2020** folder.
- 457) Enter **AXLE-3000** for File name.
- 458) Enter **AXLE-3000** for Description.
- 459) Check the **Save as copy and continue** check box. Note: You can use the Save as copy and open command. This will open the new copy.
- 460) Click **Save**.



- Close the AXLE part.
- 461) Click **File, Close** from the Menu bar.

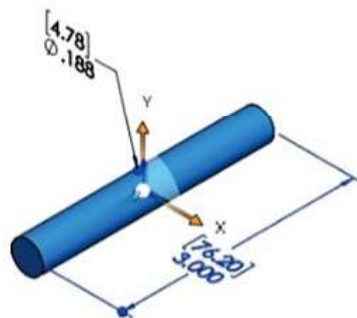
Open AXLE-3000 part.

- 462) Click **Open**  from the Menu bar.
- 463) Double-click **AXLE-3000** from the SW-TUTORIAL-2020 folder. The AXLE-3000 FeatureManager is displayed.



Modify the depth dimension.


- 464) Double-click the **cylindrical face** in the Graphics window.
- 465) **Move** the dimensions off the model.
- 466) Click 1.375in, [34.93]. Enter 3.000in, [76.20].



Fit the model to the Graphics window.

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

Save the AXLE-3000 part.

- 468) Click **Save** .
- 469) Click **inside** the Graphics window.

**Activity: SHAFTCOLLAR-500 Part**

Create the SHAFTCOLLAR-500 part.

- 470) Click **Open**  from the Menu bar.
- 471) Double-click **SHAFT-COLLAR** from the SW-TUTORIAL-2020 folder.



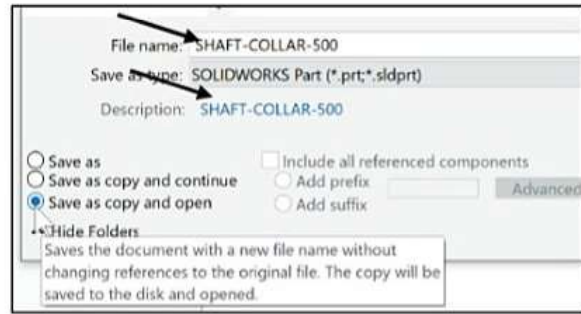


472) Click **Save As** .

473) Enter **SHAFT-COLLAR-500** for File name.

474) Enter **SHAFT-COLLAR-500** for Description.

475) Click the **Save as copy and open** check box.



476) Click **Save**. Both models are open at this time.

Modify the diameter dimensions of the SHAFT-COLLAR-500 Part.

477) Right-click **Annotations** in the FeatureManager.

478) Check the **Show Feature Dimensions** box.

479) Press the **f** key to fit the model to the graphics area.

480) Click **.438in, [11.11]**.

481) Enter **.750in, [19.05]** for outside diameter.

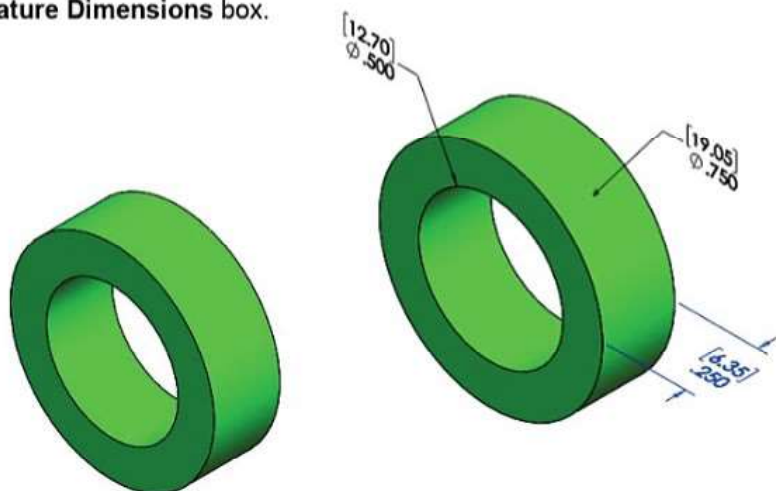
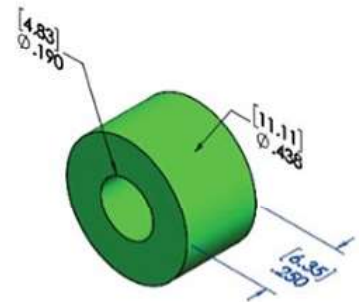
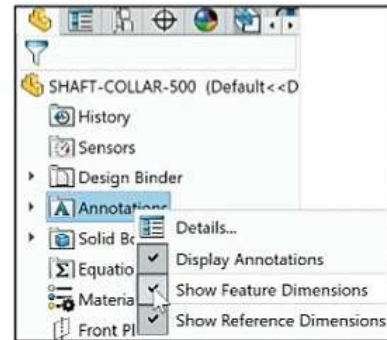
482) Click **.190in, [4.83]**.

483) Enter **.500in, [12.70]** for inside diameter. View the results.

484) Right-click **Annotations** in the FeatureManager.

485) Un-check the **Show Feature Dimensions** box.

486) **Rebuild**  the model.

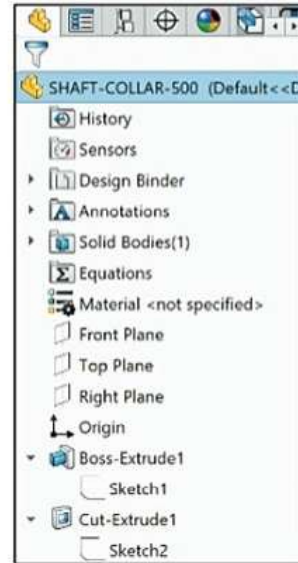


Fit the model to the Graphics window.

**487)** Press the f key.

Save the SHAFT-COLLAR-500 part.

**488)** Click **Save** .



Close all documents.

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



Press the s key in the Graphics window. A Context Pop-up features toolbar is displayed. The features toolbar displays the last few feature tools applied.



Select the types of Annotations that you want to display and set text scale and other Annotations options. In the FeatureManager design tree, right-click the **Annotations** folder, and click **details**. View the options from the Annotation Properties dialog box.



Rename a feature or sketch for clarity. Slowly click the feature or sketch name twice and enter the new name when the old one is highlighted.



Add relations, then dimensions. This will keep the user from having too many unnecessary dimensions. This helps to show the design intent of the model. Dimension what geometry you intend to modify or adjust.



## Chapter Summary

In this chapter, you created six parts. The WEIGHT part utilized the Plane feature, Lofted Base feature and the Extruded Cut (Instant3D tool) feature. The HOOK part utilized the Swept Base feature, Dome feature, and Thread feature. The WHEEL part utilized the Extruded Base feature, Revolved Cut feature, Extruded Cut feature, Circular Pattern feature and Axis feature.

The second three parts utilized existing parts created in an early chapter. The HEX-ADAPTER part, AXLE-3000 part and the SHAFTCOLLAR-500 part utilized existing part geometry along with the Hole Wizard feature.

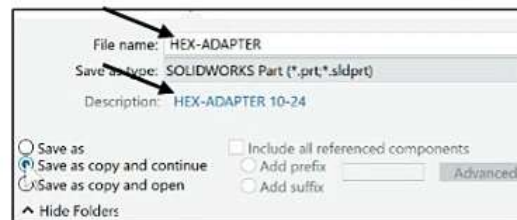
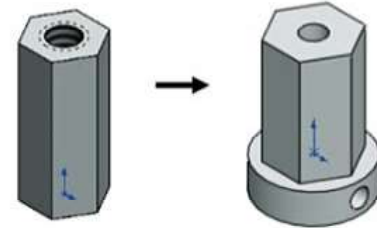
Conserve design time and cost. Modify existing parts and assemblies to create new parts and assemblies. Utilize the Save As/Save as command to save the file in another file format.

Utilize the Save as copy and continue command to save the document to a new file name without replacing the active document. Utilize the Save as copy and open command to save the document to a new file name that becomes the active document. The original document remains open. References to the original document are not automatically assigned to the copy.

You applied the following Sketch tools in this chapter: Circle, Line, Centerline, Tangent Arc, Polygon, Smart Dimension, Center Rectangle, Centerpoint Arc, 3 Point Arc and Convert Entities.



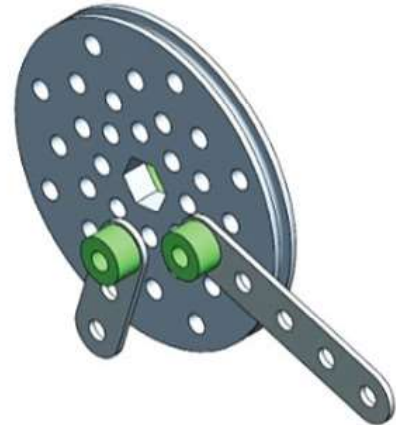
The book is designed to expose the new SOLIDWORKS user to many different tools, techniques and procedures. It may not always use the most direct tool or process.





The WHEEL-FLATBAR assembly consists of the following:

- WHEEL part.
- 3HOLE-SHAFTCOLLAR assembly.
- 5HOLE-SHAFTCOLLAR assembly.



### FLATBAR Sub-assembly


There are two similar sub-assemblies contained in the WHEEL-FLATBAR assembly:

- 3HOLE-SHAFTCOLLAR assembly.
- 5HOLE-SHAFTCOLLAR assembly.

Create the 3HOLE-SHAFTCOLLAR assembly. Utilize parts and mating techniques developed in Chapter 2 & 3.



Utilize the Save As command and create the 5HOLE-SHAFTCOLLAR assembly.

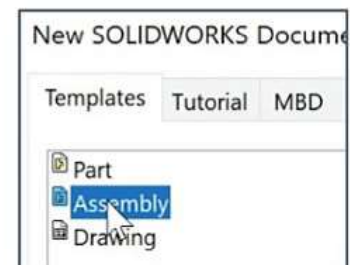
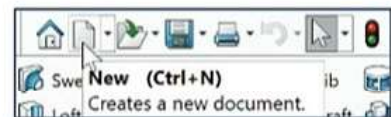
Combine the 3HOLE-SHAFTCOLLAR assembly, 5HOLE-SHAFTCOLLAR assembly and the WHEEL part to create the WHEEL-FLATBAR assembly.


 The FLATBAR-3HOLE and FLATBAR 5HOLE parts were created in the Chapter 3 exercises. If needed, copy the models from the Chapter 6 Models folder.

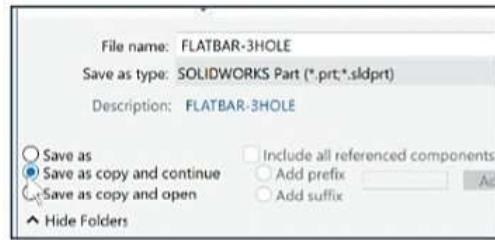
#### Activity: 3HOLE - SHAFTCOLLAR Assembly

Create the 3HOLE-SHAFTCOLLAR assembly.

- 1) Click **New**  from the Menu bar.
- 2) Double-click **Assembly** from the Templates tab.
- 3) Double-click **FLATBAR-3HOLE** from the SW-TUTORIAL-2020\Chapter 6 Models folder. The FLATBAR-3HOLE part is in the Chapter 6 Models folder. If you want to create this part, follow the below procedure; otherwise, skip the next few steps to create the part.
- 4) Click **Open**  from the Menu bar.





- 5) Double-click **FLATBAR** from the SW-TUTORIAL-2020 folder.
- 6) Click **Save As** .
- 7) Select the **SW-TUTORIAL-2020** folder.
- 8) Enter **FLATBAR-3HOLE** for File name.
- 9) Enter **FLATBAR-3HOLE** for Description.
- 10) Check the **Save as copy and continue** box.

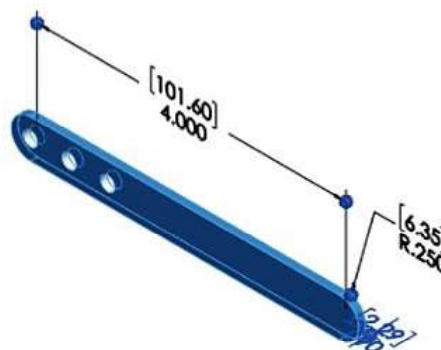
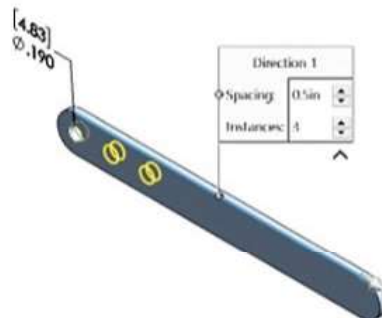
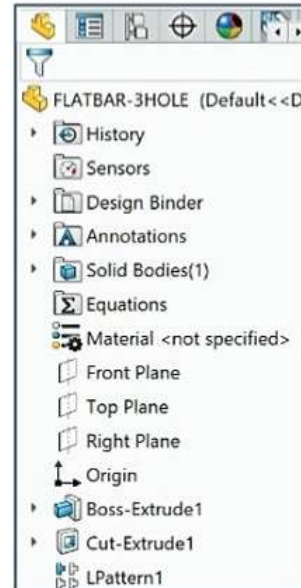


Save and close the FLATBAR model.

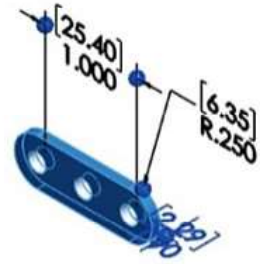
- 11) Click **Save**.
- 12) **Close** the FLATBAR model.

Open the FLATBAR-3HOLE model.


- 13) Click **Open**  from the Menu bar.
- 14) Double-click **FLATBAR-3HOLE** from the SW-TUTORIAL-2020 folder.
- 15) Right-click **LPATTERN1** from the FeatureManager.
- 16) Click **Edit Feature** from the Context toolbar. The LPattern1 PropertyManager is displayed.
- 17) Enter 3 in the Number of Instances.
- 18) Click **OK**  from the LPattern1 PropertyManager. Note: If needed, delete the Design Table in the CommandManager.
- 19) Click **Boss-Extrude1** from the FeatureManager.
- 20) **View** the dimensions in the Graphics window.
- 21) Click the 4.000in, [101.60] dimension.
- 22) Enter 1.000in, [25.4].




- 23) Click **Cut-Extrude1** from the FeatureManager.
- 24) **View** the dimensions in the Graphics window.
- 25) Right-click **9X** dimension text in the Graphics window. The Dimension PropertyManager is displayed.
- 26) Delete the **9X** text in the Dimension Text box.
- 27) Enter **3X** in the Dimension Text box.
- 28) Click **OK** ✓ from the Dimension PropertyManager.

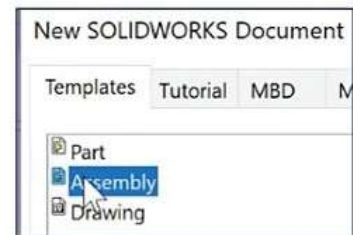
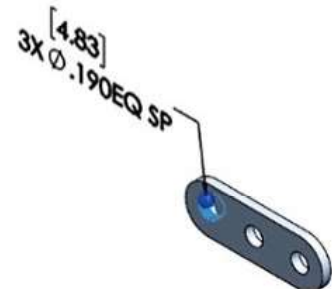


Save the model.

- 29) Click **Save** .


Create a new assembly.


- 30) Click **New**  from the Main menu.
- 31) Double-click **Assembly** from the Templates tab.
- 32) Double-click **FLATBAR-3HOLE** from the SW-TUTORIAL-2020\Chapter 6 Models folder. The FLATBAR-3HOLE part is in the Chapter 6 Models folder.
- 33) Double-click **FLATBAR-3HOLE**.
- 34) Click **OK** ✓ from the Begin Assembly PropertyManager. The FLATBAR-3HOLE is fixed to the Origin.

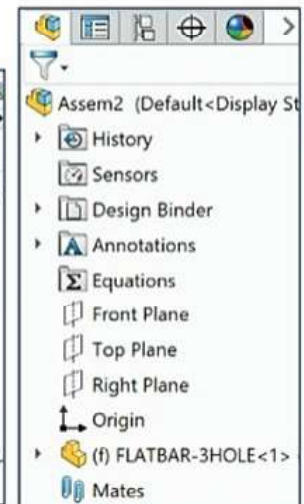
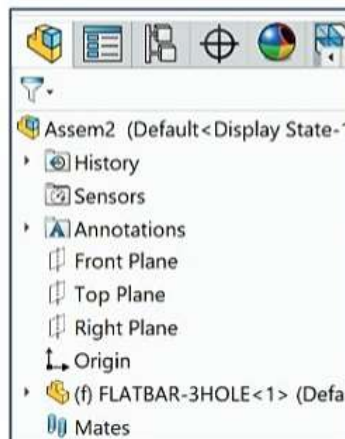


Save the assembly. Enter name. Enter description.

- 35) Click **Save As** from the drop-down Menu bar.

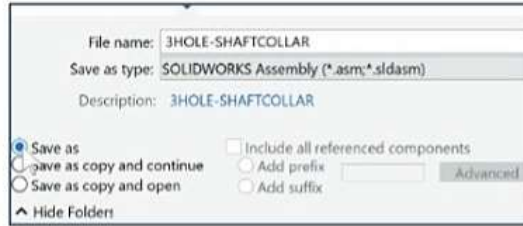
 Tangent edges and Origin are displayed for educational purposes.

 CommandManager and Feature Manager tabs and folder files will vary, depending on system setup and Add-Ins.





- 36) Enter **3HOLE-SHAFTCOLLAR** for File name.
- 37) Enter **3HOLE-SHAFTCOLLAR** for Description.
- 38) Click **Save**.





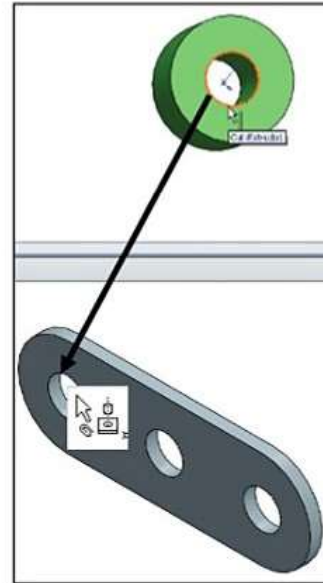
Save the 3HOLE-SHAFTCOLLAR assembly.

- 39) Click **Save** .



Utilize a Concentric/Coincident  SmartMate between the SHAFT-COLLAR and the FLATBAR-3HOLE.

Open the **SHAFT-COLLAR** part.


- 40) Click **Open**  from the Menu bar.
- 41) Double-click **SHAFT-COLLAR** from the SW-TUTORIAL-2020 folder. SHAFT-COLLAR is the current document name.
- 42) **Rotate** the SHAFT-COLLAR to view the back circular edge.
- 43) Click **Window, Tile Horizontally** from the Menu bar.
- 44) Drag the **back circular edge** of the SHAFT-COLLAR to the left circular hole edge of the FLATBAR-3HOLE in the Assembly Graphics window as illustrated. The mouse pointer displays the Concentric/Coincident  icon.
- 45) **Release** the mouse button. Note: Select the back circular edge of the SHAFT-COLLAR, not the face.

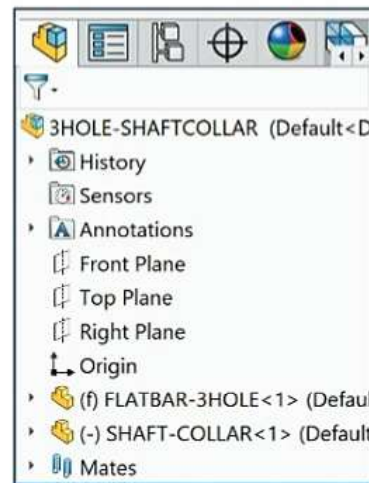


Save the 3HOLE-SHAFTCOLLAR assembly.

- 46) **Close**  the SHAFT-COLLAR window.
- 47) **Maximize**  the 3HOLE-SHAFTCOLLAR assembly.

Fit the model to the Graphics window. Save the assembly.

- 48) Press the **f** key.
- 49) Click **Save** .



Create the 5HOLE-SHAFTCOLLAR assembly.  
Utilize the Save As command with the Save as copy and continue option. Recover from Mate errors.



Save the 3HOLE-SHAFTCOLLAR assembly as the 5HOLE-SHAFTCOLLAR assembly.

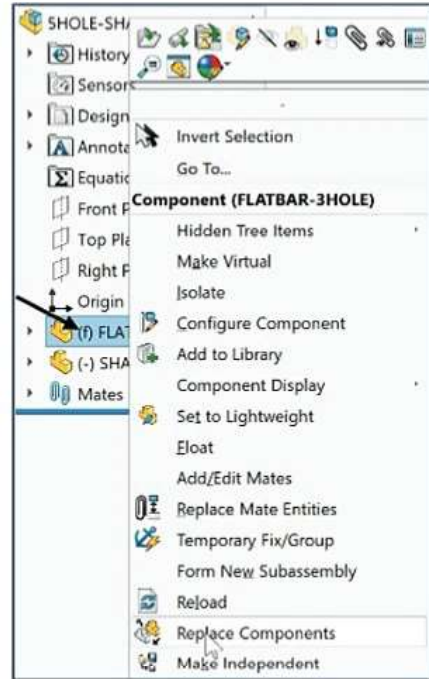
- 50) Click **Save As** from the drop-down Menu bar.
- 51) Check the **Save as copy and continue** box.
- 52) Enter **5HOLE-SHAFTCOLLAR** for File name.
- 53) Enter **5HOLE-SHAFTCOLLAR** for Description.
- 54) Click **Save**.

Close the model.

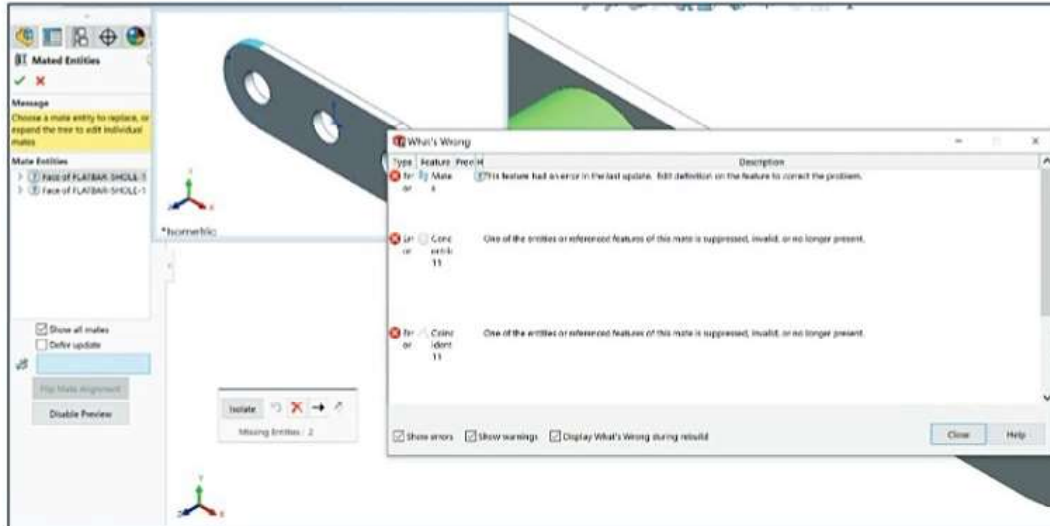
- 55) Click **File, Close** from the Menu bar.
- 56) Click **Save All**.

Open the 5HOLE-SHAFTCOLLAR assembly.

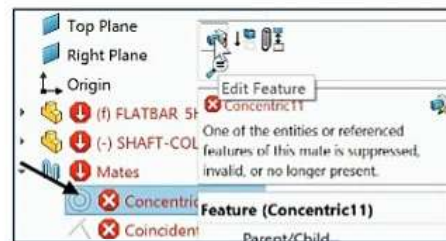
- 57) Click **Open**  from the Menu bar.
- 58) Double-click **5HOLE-SHAFTCOLLAR**. The 5HOLE-SHAFTCOLLAR FeatureManager is displayed.
- 59) Right-click **FLATBAR-3HOLE** from the FeatureManager.
- 60) **Expand** the Pop-up menu if needed.
- 61) Click **Replace Components**. The Replace PropertyManager is displayed.
- 62) Click the **Browse** button.
- 63) Double-click **FLATBAR-5HOLE**. Note: The FLATBAR-5HOLE part is located in the Chapter 6 Models folder.
- 64) Check the **Re-attach mates** box.
- 65) Click **OK**  from the Replace PropertyManager. The Mate Entities PropertyManager and the What's Wrong dialog box is displayed. There are two red Mate error marks displayed in the Mate Entities box.



- 66) The What's Wrong dialog box is displayed. Recover from the Mate errors. Click **Close** from the What's Wrong dialog box.

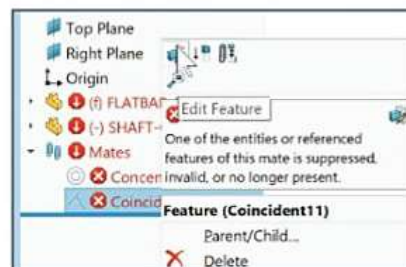
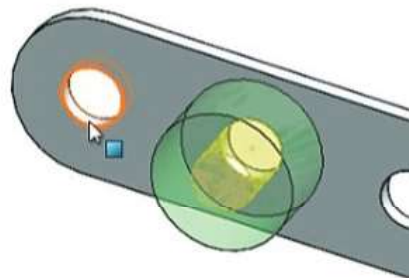


- 67) Click **OK** from the Mated Entities PropertyManager.
- 68) Click **Close** from the What's Wrong dialog box. View the location of the SHAFT-COLLAR in the Graphics window.



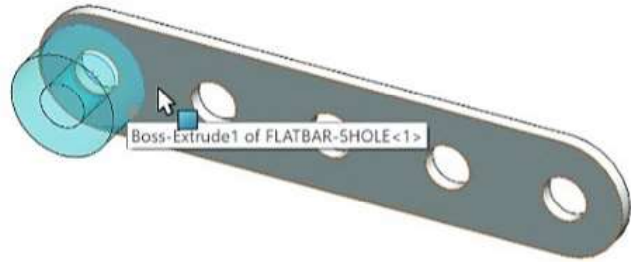
Recover from the Mate errors.

- 69) **Expand** the Mates folder from the FeatureManager. Right-click the first mate, **Concentric #** from the Mates folder.
- 70) Click **Edit Feature** from the Context toolbar. The Mate PropertyManager is displayed.
- 71) Right-click the **Mate Face** error in the Mate Selections box. Click **Delete**.
- 72) Click the **inside face** of the left hole of the FLATBAR as illustrated. Concentric is selected by default.
- 73) Click **OK** from the Mate PropertyManager.
- 74) Right-click the second mate, **Coincident #** from the Mates folder.
- 75) Click **Edit Feature** from the Context toolbar. The Mate PropertyManager is displayed.

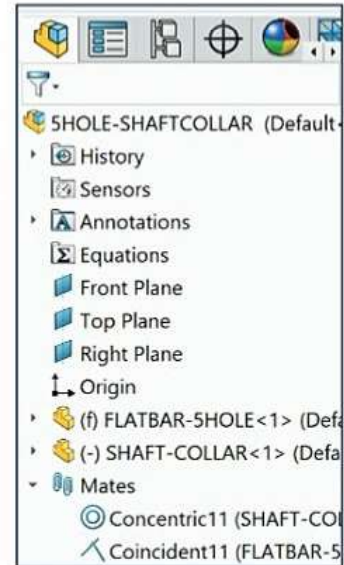




- 76) Right-click the **Mate Face** error in the Mate Selections box.
- 77) Click **Delete** as illustrated.
- 78) Click the **front face** of the **FLATBAR** as illustrated. The selected faces are displayed in the Mate Selections box. Coincident is selected by default.



- 79) Click **OK** ✓ from the Mate PropertyManager.
- 80) **Expand** the Mate folder from the FeatureManager.
- 81) **View** the created mates.



💡 The Mate Entities box will list red X's if the faces, edges or planes are not valid. Expand the Mate Entities and select new references in the Graphics window to redefine the mates.

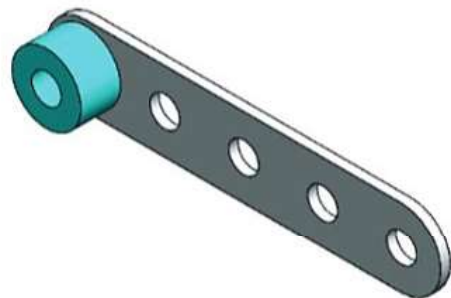
The FLATBAR-3HOLE is replaced with the FLATBAR-5HOLE part. The Mates are updated.

Fit the model to the Graphics window.

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

Display an Isometric view. Save the 5HOLE-SHAFTCOLLAR assembly.

- 83) Click **Isometric view** 📐 .
- 84) Click **Save** 💾 .



💡 Incorporate symmetry into the assembly. Divide large assemblies into smaller sub-assemblies.

💡 CommandManager and Feature Manager tabs and folder files will vary, depending on system setup and Add-Ins.

### WHEEL-FLATBAR Assembly

The WHEEL-FLATBAR assembly consists of the following components:

- 3HOLE-SHAFTCOLLAR assembly.
- 5HOLE-SHAFTCOLLAR assembly.
- WHEEL part.

Create the WHEEL-FLATBAR assembly. Mate the 3HOLE-SHAFTCOLLAR assembly 67.5 degrees counterclockwise from the Top Plane.

The 3HOLE-SHAFTCOLLAR assembly is concentric with holes on the second and fourth bolt circle.

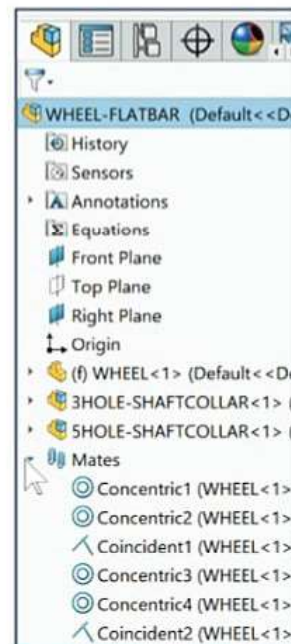
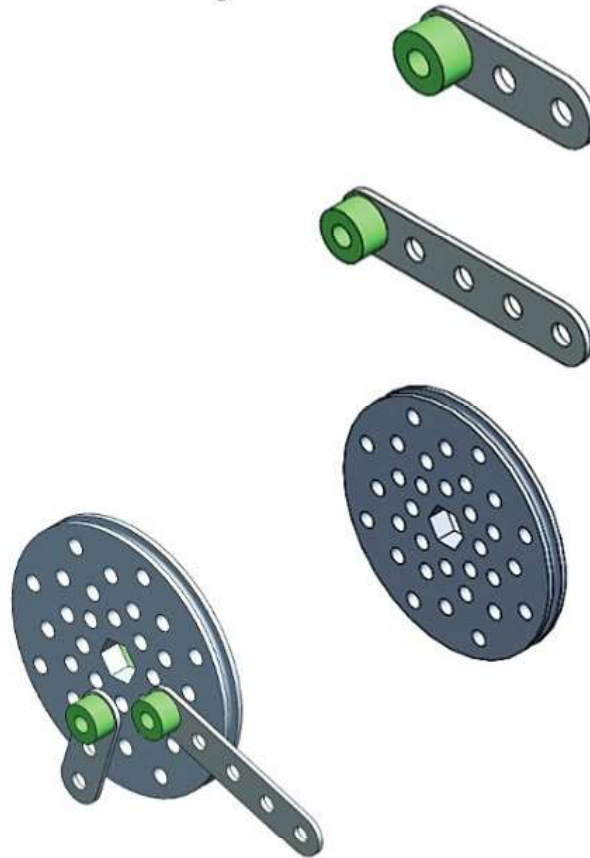
Mate the 5HOLE-SHAFTCOLLAR assembly 22.5 degrees clockwise from the Top Plane.

The 5HOLE-SHAFTCOLLAR assembly is concentric with holes on the second and fourth bolt circle.

Use the Quick mate procedure for reference geometry (such as planes, axes, and points) along with model geometry (such as faces, edges and vertices).



Remove the fixed state. Right-click the component name in the FeatureManager. Click **Float**. The component is free to move.

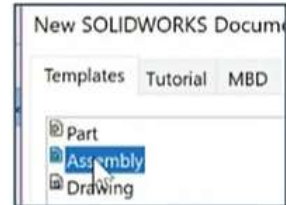


**Activity: WHEEL-FLATBAR Assembly**

Create the WHEEL-FLATBAR assembly.


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

86) Double-click **Assembly** from the Templates tab.



Insert the WHEEL.

87) Double-click **WHEEL** from the SW-TUTORIAL-2020 folder. The WHEEL was created in Chapter 5. The WHEEL is in the Chapter 6 Models folder.

88) Click **OK**  from the Begin Assembly PropertyManager. The WHEEL part is fixed to the assembly Origin.

If needed, deactivate the Origins.

89) Click **View, Hide/Show**, uncheck **Origins** from the Menu bar menu.



Save the assembly. Enter name. Enter description.

90) Click **Save As** from the drop-down Menu bar.

91) Select the **SW-TUTORIAL-2020** folder.


92) Enter **WHEEL-FLATBAR** for File name.

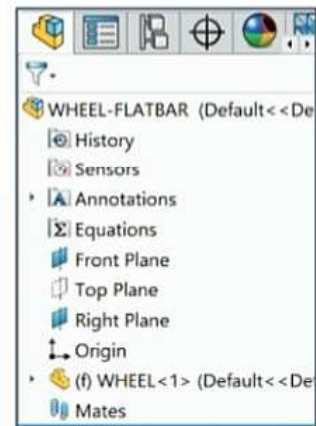
93) Enter **WHEEL-FLATBAR** for Description.

94) Click **Save**.



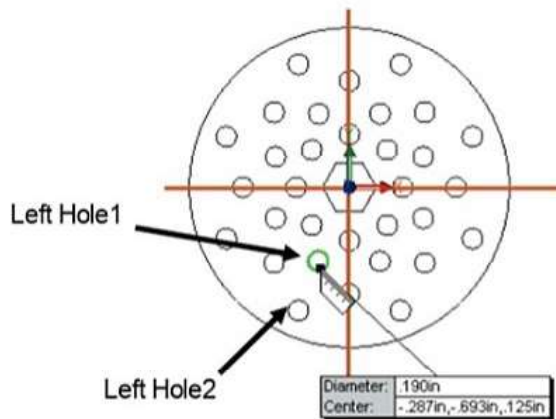
Display the Top Plane in the Front view.

95) Click **Front view**  from the Heads-up View toolbar. View the WHEEL part.





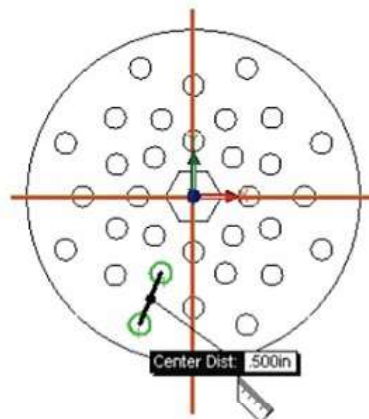
Locate the first set of holes from the Right plane (-Y-axis). Left Hole1 and Left Hole2 are positioned on the second and fourth bolt circle, 22.5° from the Right plane. Select Left Hole1. The x, y, z coordinates, -.287, -.693, .125 are displayed.




$$\tan^{-1} (-.287/.693) = 22.5^\circ$$

As an exercise, utilize the Measure tool to determine the center-to-center distance between the Left Hole1 and Left Hole2. The center-to-center distance is .500in.

Insert two Concentric mates between Left Hole1 and Left Hole2 and the 3HOLE-SHAFTCOLLAR assembly holes. The FLATBAR-3HOLE center-to-center distance is also .500in.

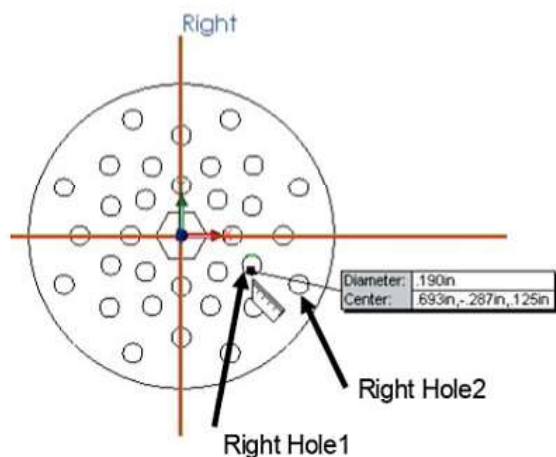


 To determine tolerance issues, utilize two Concentric mates between components with mating cylindrical geometry. If the mating components center-to-center distance is not exact, a Mate error is displayed on the second Concentric mate.

Insert a Coincident mate between the back face of the 3HOLE-SHAFTCOLLAR assembly and the front face of the WHEEL.

Right Hole1 and Right Hole2 are 22.5° from the Top Plane.



Insert two Concentric mates between Right Hole1 and Right Hole2 and the 5HOLE-SHAFTCOLLAR assembly holes.

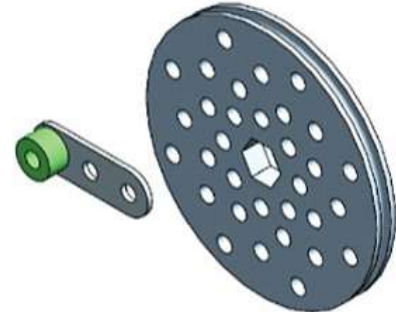


Insert a Coincident mate between the back face of the 5HOLE-SHAFTCOLLAR assembly and the front face of the WHEEL.

**Activity: WHEEL-FLATBAR Assembly - Insert 3HOLE-SHAFTCOLLAR Assembly**

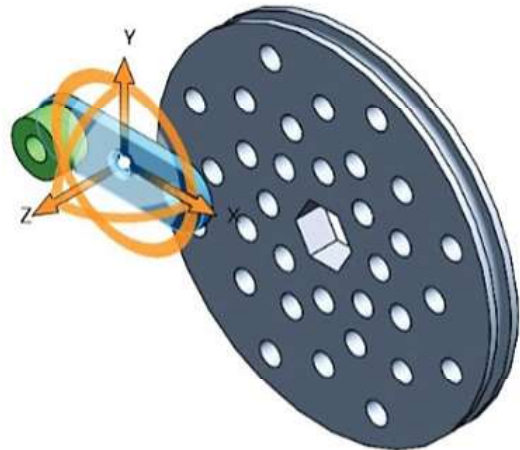
Insert the 3HOLE-SHAFTCOLLAR assembly.

- 96)** Click **Isometric view**  from the Heads-up View toolbar.
- 97)** Click **Insert Components**  from the Assembly toolbar.
- 98)** Double-click the **3HOLE-SHAFTCOLLAR** assembly from the SW-TUTORIAL-2020 folder. The assembly is also located in the Chapter 6 Models folder.
- 99)** Click a **position** to the left of the WHEEL as illustrated.



Move and rotate the 3HOLE-SHAFTCOLLAR component.

- 100)** Click the front face of the **3HOLE-SHAFTCOLLAR**.
- 101)** Right-click **Move with Triad**.
- 102)** Hold the **left mouse button** down on the X-axis (red).
- 103)** Drag the **component** to the left. View the ruler.
- 104)** Hold the **right mouse button** down on the Z-axis (blue).
- 105)** Drag the **component** and rotate it about the Z-axis. View the ruler.
- 106)** **Position** the component until the SHAFT-COLLAR part is approximately in front of the WHEEL Left Hole1.
- 107)** Release the **right mouse button**.
- 108)** Click a **position** in the Graphics window to deselect the face.



Insert a Concentric mate.

**109)** Click the **back top inside cylindrical face** of the SHAFT-COLLAR.

**110)** Rotate the model to view the WHEEL.

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

**112)** Click the **WHEEL Left Hole1** cylindrical face as illustrated.

**113)** Release the **Ctrl** key. The Mate pop-up menu is displayed.

**114)** Click **Concentric** from the Mate pop-up menu.

Insert the second Concentric mate.

**115)** Click the **back middle inside cylindrical face** of the FLATBAR.

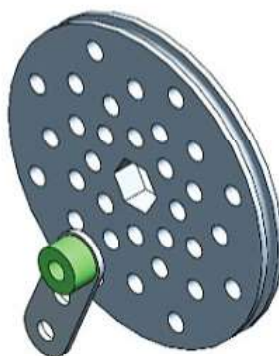
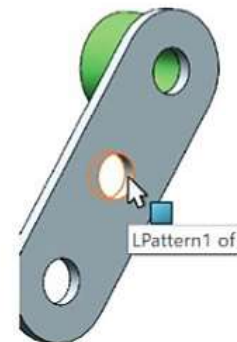
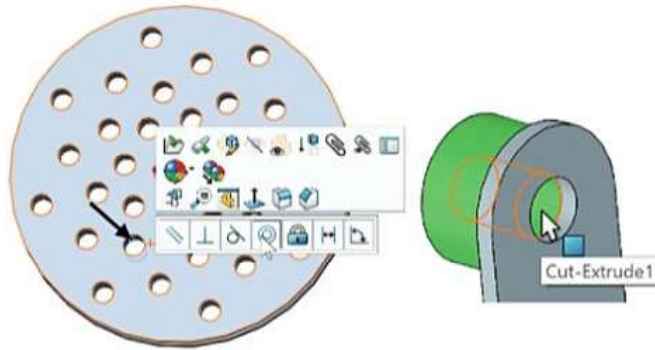
**116)** Rotate the model to view the WHEEL.

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

**118)** Click the **WHEEL Left Hole2** inside cylindrical face as illustrated.

**119)** Release the **Ctrl** key. The Mate pop-up menu is displayed.

**120)** Click **Concentric** from the Mate pop-up menu.





Insert a Coincident mate.

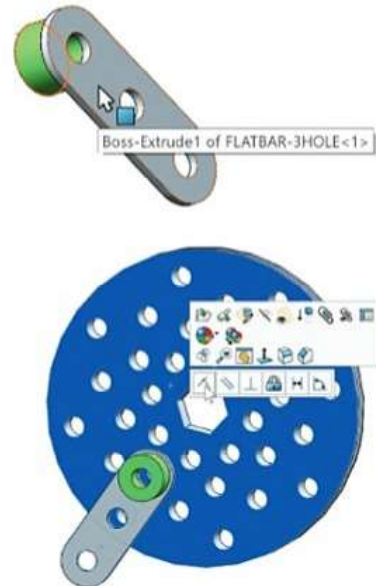
- 121) Click the **FLATBAR-3HOLE** back face.
- 122) **Rotate** the model to view the **WHEEL**.
- 123) Hold the **Ctrl** key down.
- 124) Click the front face of the **WHEEL**.
- 125) Release the **Ctrl** key. The Mate pop-up menu is displayed.
- 126) Click **Coincident** from the Mate pop-up menu.

Display a Front view.

- 127) Click **Front view** .


Save the WHEEL-FLATBAR assembly.

- 128) Click **Save** .




#### Activity: WHEEL-FLATBAR Assembly - Insert 5HOLE-SHAFTCOLLAR Assembly

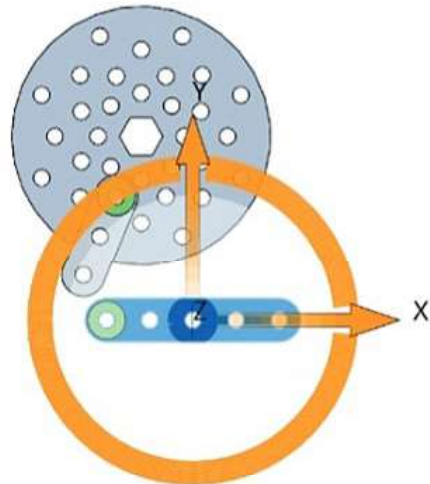
Insert the 5HOLE-SHAFTCOLLAR assembly.

- 129) Click **Insert Components**  from the Assembly toolbar.
- 130) Double-click the **5HOLE-SHAFTCOLLAR** assembly from the SW-TUTORIAL-2020 folder. The assembly is in the Chapter 6 Models folder.

- 131) Click a **position** to the right of the **WHEEL**.

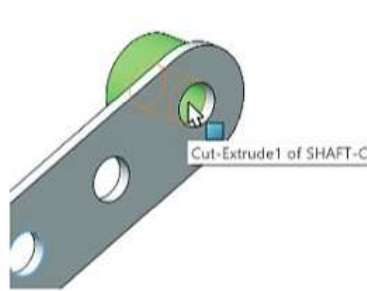
Move the 5HOLE-SHAFTCOLLAR component.

- 132) Click the **5HOLE-SHAFTCOLLAR** front face.
- 133) Right-click **Move with Triad**.
- 134) Hold the **left mouse button** down on the **X-axis** (red).
- 135) Drag the **component** to the right. View the results.
- 136) Click **Isometric view** .
- 137) Click **inside** the Graphics window.



Mate the 5HOLE-SHAFTCOLLAR assembly.  
Insert a Concentric mate.

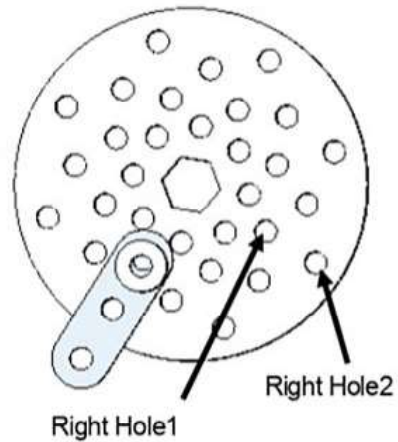
**138)** Click the **back inside cylindrical face** as illustrated on the **FLATBAR-5HOLE** assembly.



**139)** Rotate the model to view the **WHEEL**.

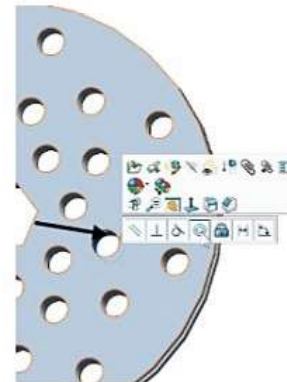
**140)** Hold the **Ctrl** key down.

**141)** Click the **WHEEL Right Hole1** cylindrical face.



**142)** Release the **Ctrl** key. The Mate pop-up menu is displayed.

**143)** Click **Concentric** from the Mate pop-up menu.



Insert a Concentric mate.

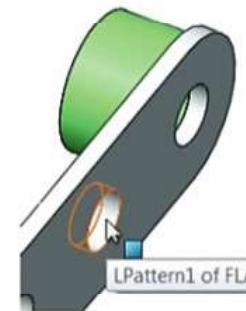
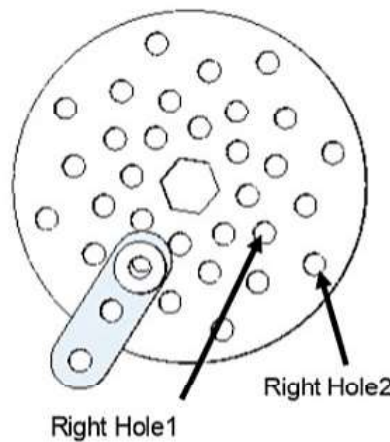
**144)** Move the 5HOLE-SHAFTCOLLAR to view the back side.

**145)** Click the **back inside cylindrical face** of the second hole on the **FLATBAR-5HOLE** assembly.

**146)** Rotate the model to view the **WHEEL**.

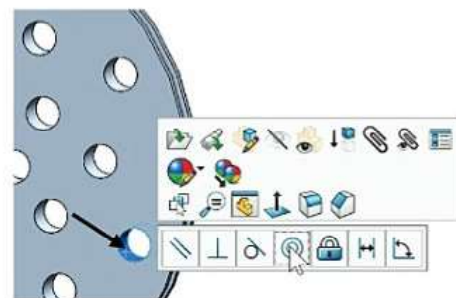
**147)** Hold the **Ctrl** key down.

**148)** Click the **WHEEL Right Hole2** cylindrical face.



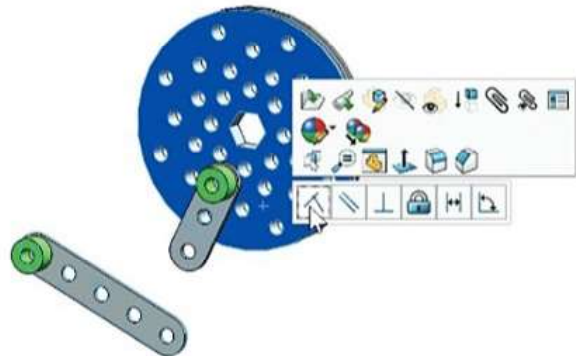
**149)** Release the **Ctrl** key. The Mate pop-up menu is displayed.

**150)** Click **Concentric** from the Mate pop-up menu.



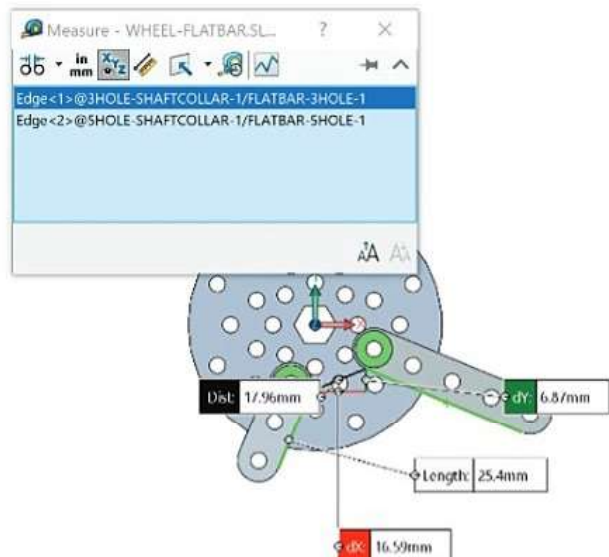
Insert a Coincident mate.

- 151) Click the **back face** of the **FLATBAR-5HOLE**.
- 152) **Rotate** the model to view the **WHEEL**.
- 153) Hold the **Ctrl** key down.
- 154) Click the **front face** of the **WHEEL**.
- 155) Release the **Ctrl** key. The **Mate** pop-up menu is displayed.
- 156) Click **Coincident** from the **Mate** pop-up menu.




Measure the angle between the **3HOLE-SHAFTCOLLAR** assembly and the **5HOLE-SHAFTCOLLAR** assembly.

- 157) Click **Front view**.
- 158) Click the **Measure** tool from the **Evaluate** tab in the **CommandManager**. The **Measure** dialog box is displayed.
- 159) Select the **two inside edges** of the **FLATBAR** assemblies. View the results.
- 160) If required, click the **Show XYZ Measurements** button. The items are perpendicular.




The **Measure** tool provides the ability to display dual units. Click the **Units/Precision** icon from the **Measure** dialog box. Set the desired units.

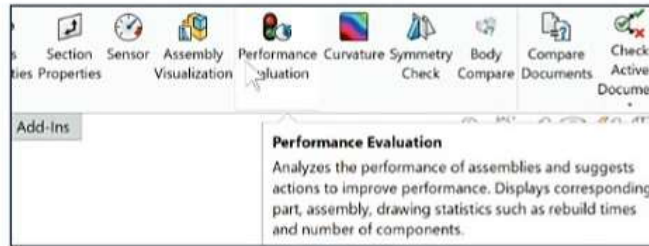
 Utilize the **Measure** tool to measure distances and add reference dimensions between the **COM** point and entities such as vertices, edges, and faces.





**161)** Close  the Measure dialog box.

Apply the Performance Evaluation tool. The Performance Evaluation tool displays statistics and checks the health of the current assembly.



**162)** Click the **Performance Evaluation** tool in the Evaluate toolbar. The Performance Evaluation dialog box is displayed. Review the Status and description for the assembly.

**163)** Click **OK** from the Performance Evaluation dialog box.

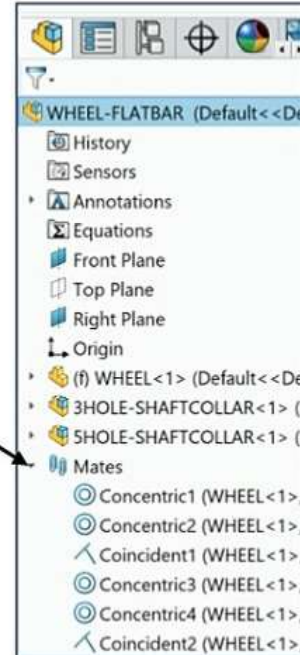
Fit the model to the Graphics window. Display an Isometric view.


**164)** Press the f key.


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

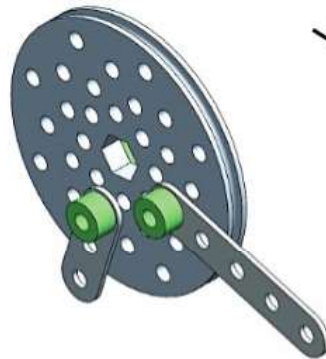
Save the WHEEL-FLATBAR assembly.

**166)** Click **Save** .



 To remove the fixed state, Right-click the component name in the FeatureManager. Click **Float**. The component is free to move.

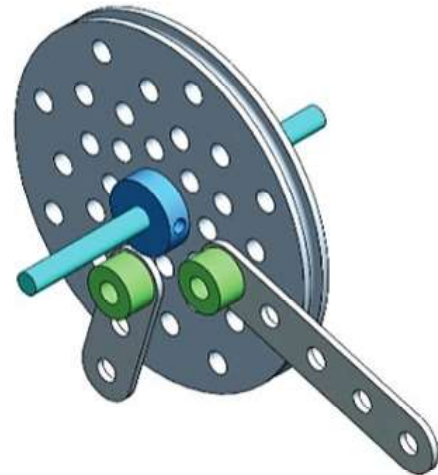
 Mate to the first component added to the assembly. If you mate to the first component or base component of the assembly and decide to change its orientation later, all the components will move with it.



## WHEEL-AND-AXLE Assembly

The WHEEL-AND-AXLE assembly contains the following items:

- WHEEL-FLATBAR assembly.
- AXLE-3000 part.
- SHAFTCOLLAR-500 part.
- HEX-ADAPTER part.



Create the WHEEL-AND-AXLE assembly. The AXLE-3000 part is the first component in the assembly. A part or assembly inserted into a new assembly is called a component. The WHEEL-FLATBAR assembly rotates about the AXLE part.

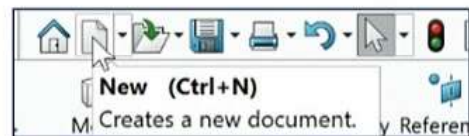
Combine the created new assemblies and parts to develop the PNEUMATIC-TEST-MODULE assembly.

### Activity: WHEEL-AND-AXLE Assembly

Create the WHEEL-AND-AXLE assembly.


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

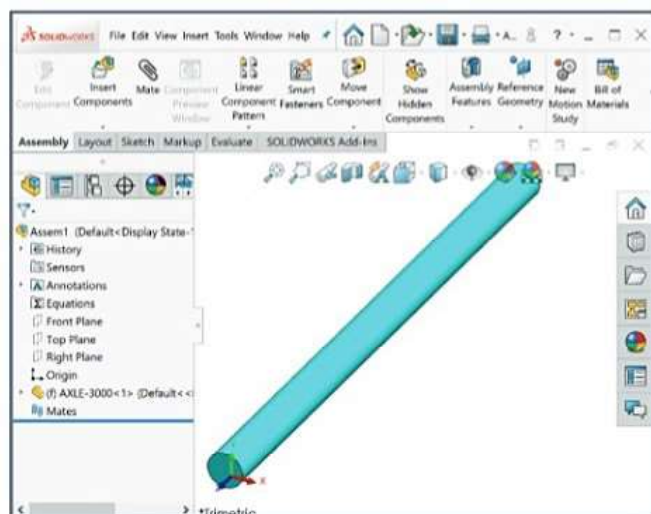
**168)** Double-click **Assembly** from the Templates tab.



Insert the AXLE-3000 part.

**169)** Double-click **AXLE-3000** from the SW-TUTORIAL-2020 folder. AXLE-3000 is in the Chapter 6 Models folder if you did not create it in Chapter 5.

**170)** Click **OK**  from the Begin Assembly PropertyManager. The AXLE-3000 part is fixed to the assembly Origin.



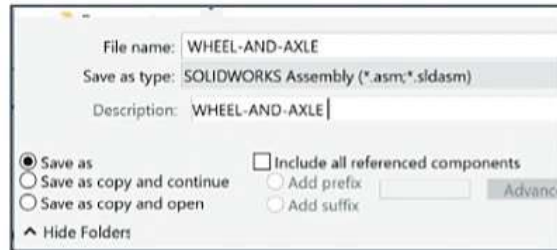
Save the assembly. Enter name. Enter description.

171) Click **Save As** from the drop-down Menu bar.

172) Enter **WHEEL-AND-AXLE** for File name in the SW-TUTORIAL-2020 folder.

173) Enter **WHEEL-AND-AXLE** for Description.

174) Click **Save**.



Insert a Coincident mate between the Axis of the AXLE-3000 and the Axis of the WHEEL. Insert a Coincident mate between the Front Plane of the AXLE-3000 and the Front Plane of the WHEEL. The WHEEL-FLATBAR assembly rotates about the AXLE-3000 axis.

Display the Temporary Axes.

175) Click **View, Hide/Show**, check **Temporary Axes** from the Menu bar.

Insert the WHEEL-FLATBAR assembly.

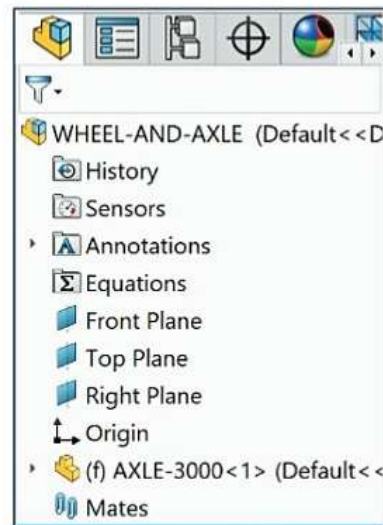
176) Click **Insert Components**  from the Assembly toolbar.

177) Double-click the **WHEEL-FLATBAR** assembly from the SW-TUTORIAL-2020 folder. The assembly is in the Chapter 6 Models folder.

178) Click a **position** to the right of AXLE-3000.

View the Reference WHEEL Axis.

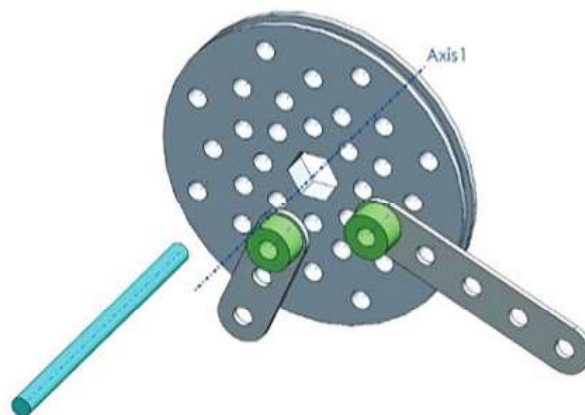
179) Click **View, Hide/Show**, check **Axes** from the Menu bar.



Reference geometry defines the shape or form of a surface or a solid. Reference geometry includes planes, axes, coordinate systems, and points.



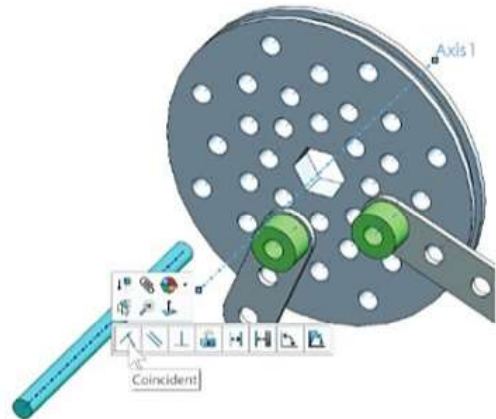
The book is designed to expose the new SOLIDWORKS user to many different tools, techniques and procedures. It may not always use the most direct tool or process.





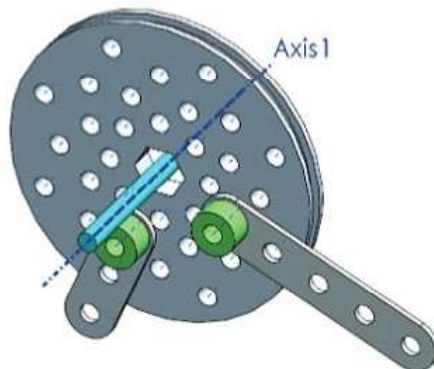
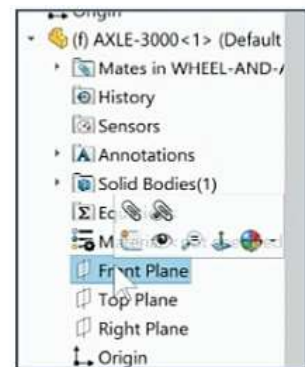
Insert a Coincident mate.

- 180) Click **Axis1** in the Graphics window.
- 181) Hold the **Ctrl** key down.
- 182) Click the **AXLE-3000 Temporary Axis**.
- 183) Release the **Ctrl** key. The Pop-up Mate menu is displayed.
- 184) Click **Coincident** from the Mate Pop-up menu.
- 185) Click **OK** ✓ from the Coincident PropertyManager.



Insert a Coincident mate.

- 186) **Expand** WHEEL-AND-AXLE from the fly-out FeatureManager.
- 187) **Expand** AXLE-3000 from the fly-out FeatureManager.
- 188) Click **Front Plane** of AXLE-3000<1>.
- 189) **Expand** WHEEL from the fly-out FeatureManager.
- 190) Hold the **Ctrl** key down.
- 191) Click **Front Plane** of the WHEEL.
- 192) Release the **Ctrl** key. The Mate Pop-up menu is displayed.
- 193) Click **Coincident** from the Mate Pop-up menu.

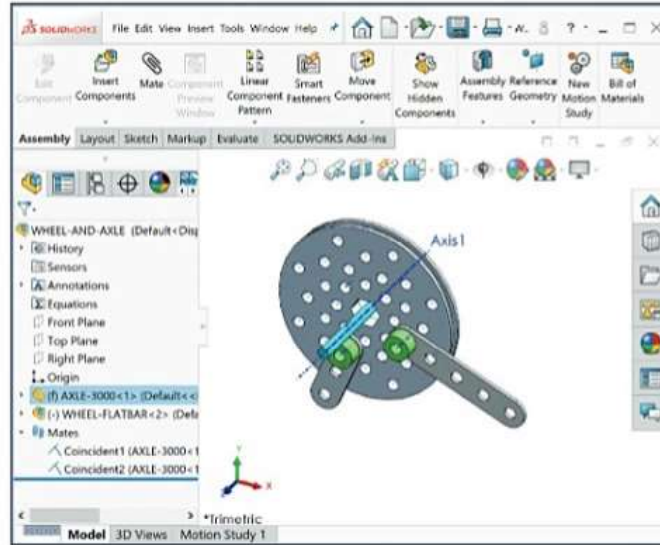


Rotate the WHEEL-FLATBAR assembly about AXLE-3000.

- 194)** Click and drag the **WHEEL** around AXLE-3000.


Save the WHEEL-AND-AXLE assembly.

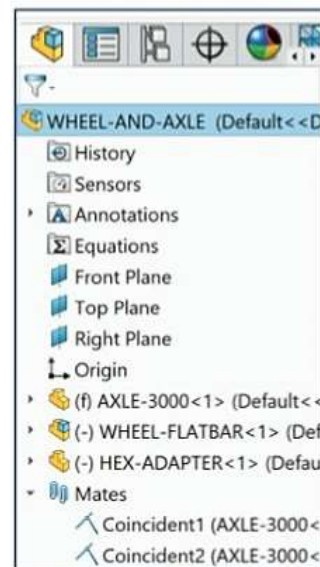
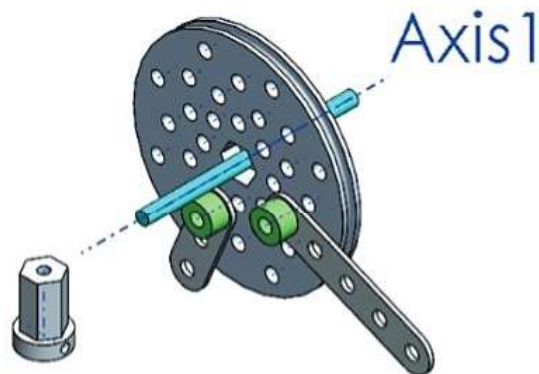
- 195)** Click **Save** .






**Activity: WHEEL-AND-AXLE Assembly - Insert the HEX-ADAPTER Part**

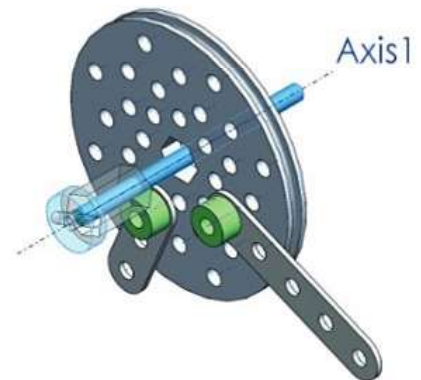
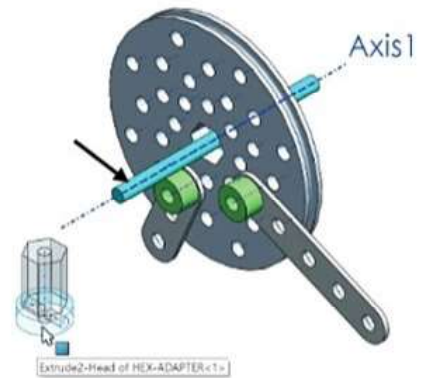
Insert the HEX-ADAPTER part.

- 196)** Click the **Insert Components**  tool from the Assembly toolbar.
- 197)** Double-click **HEX-ADAPTER** from the SW-TUTORIAL-2020 folder.
- 198)** Click a **position** near the assembly as illustrated. The HEX-ADAPTER is displayed in the FeatureManager.
- 199)** **Expand** the Mates folder.
- 200)** **View** the created mates for the assembly. View the inserted components: AXLE-3000, WHEEL-FLATBAR and HEX-ADAPTER.



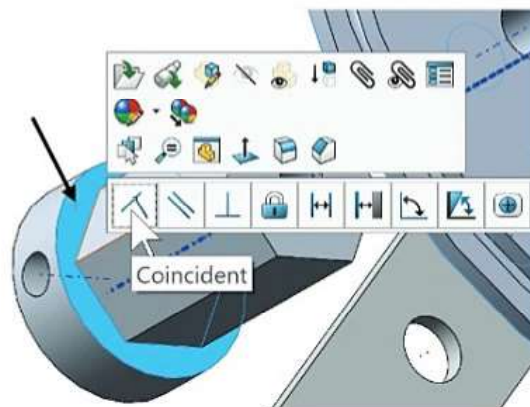
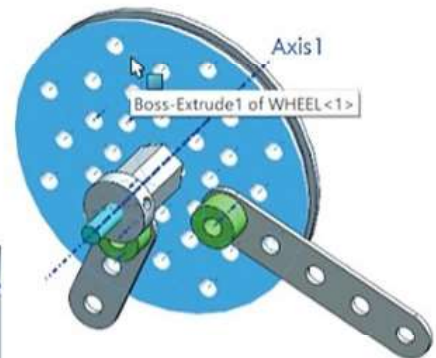
Insert a Concentric mate. Mate the HEX-ADAPTER.

- 201) Click the **Mate**  tool from the Assembly toolbar. The Mate PropertyManager is displayed.
- 202) Click the **HEX-ADAPTER** cylindrical face as illustrated.
- 203) Click the **AXLE-3000** cylindrical face as illustrated. Concentric is selected by default. The selected faces are displayed in the Mate Selections box.
- 204) Click **Aligned**  from the Concentric1 PropertyManager to flip the HEX-ADAPTER, if required.
- 205) Click **OK**  from the Concentric PropertyManager.



Insert a Coincident mate.

- 206) Click the **front face** of the WHEEL.
- 207) **Rotate** the assembly to view the flat back circular face of the HEX-ADAPTER as illustrated.
- 208) Hold the **Ctrl** key down.
- 209) Click the **flat back circular face** of the HEX-ADAPTER.
- 210) Release the **Ctrl** key. The Mate pop-up menu is displayed.
- 211) Click **Coincident** from the Mate pop-up menu.





Insert a Parallel mate.

**212)** Rotate the WHEEL-AND-AXLE assembly to view the back-bottom edge of the HEX-ADAPTER.

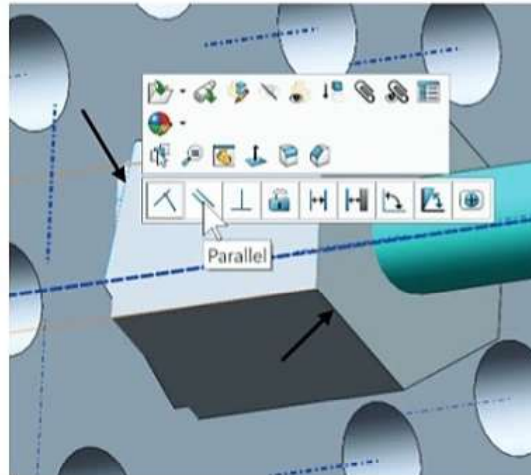
**213)** Zoom in on the back-bottom edge of the HEX-ADAPTER.


**214)** Click the back-bottom edge of the WHEEL. Do not select the midpoint.

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

**216)** Click the top edge of the HEX-ADAPTER. Do not select the midpoint.

**217)** Release the Ctrl key. The Mate pop-up menu is displayed.




**218)** Click **Parallel**  from the Mate pop-up menu.

Fit the model to the Graphics window.

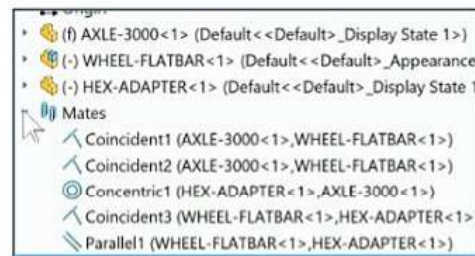
**219)** Press the f key.

Display an Isometric view.

**220)** Click **Isometric view** .

View the created mates.

**221)** Expand the Mates folder.




Save the WHEEL-AND-AXLE assembly.

**222)** Click **Save** .

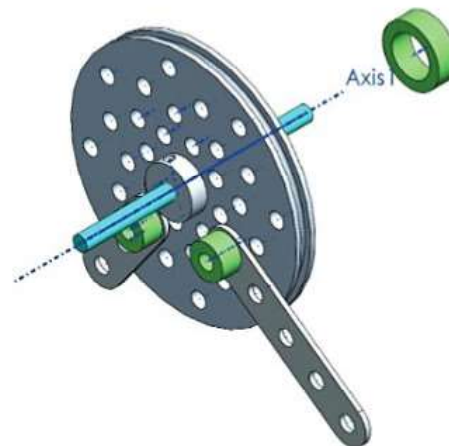
**Activity: WHEEL-AND-AXLE Assembly - Insert SHAFTCOLLAR-500 Part**

Insert the SHAFTCOLLAR-500 part.

**223)** Click the **Insert Components**  tool from the Assembly toolbar.

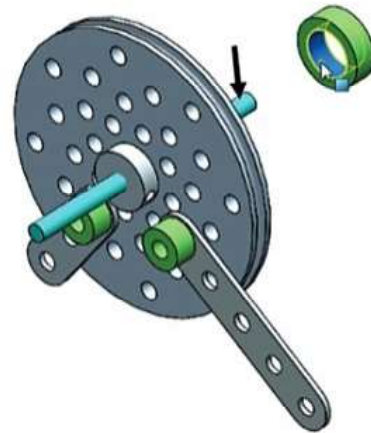
**224)** Double-click **SHAFTCOLLAR-500** from the SW-TUTORIAL-2020 folder.

**225)** Click a **position** behind the WHEEL-AND-AXLE assembly.



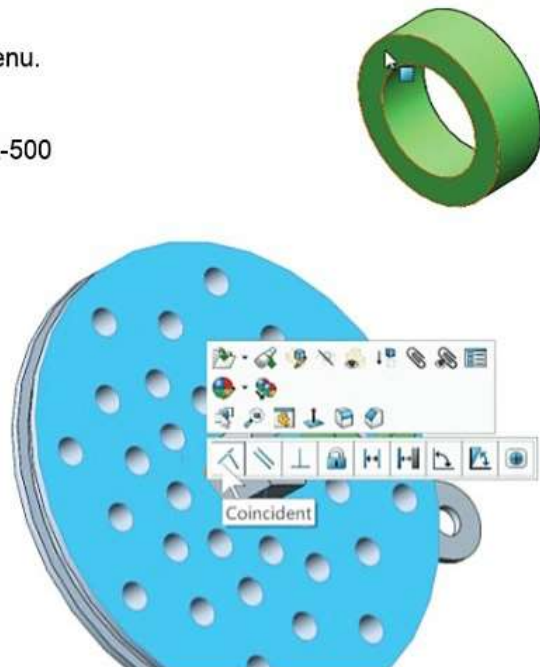
Insert a Concentric mate.

- 226) Click **View, Hide/Show**, un-check **Temporary Axes** from the Menu bar.
- 227) Click **View, Hide/Show**, un-check **Axes** from the Menu bar.
- 228) Hold the **Ctrl** key down.
- 229) Click the **inside cylindrical face** of the SHAFTCOLLAR-500 part.
- 230) Click the **cylindrical face** of the AXLE-3000 part.
- 231) Release the **Ctrl** key. The Mate pop-up menu is displayed.
- 232) Click **Concentric** from the Mate pop-up menu.



Insert a Coincident mate.

- 233) Click the **front face** of the SHAFTCOLLAR-500 part.
- 234) **Rotate** the WHEEL to view the back face.
- 235) Hold the **Ctrl** key down.
- 236) Click the **back face** of the WHEEL.
- 237) Release the **Ctrl** key. The Mate pop-up menu is displayed.
- 238) Click **Coincident** from the Mate pop-up menu.




Display an Isometric view.

- 239) Click **Isometric view** .

View the created Mates.

- 240) **Expand** the Mates folder. View the created mates.

Save the WHEEL-AND-AXLE assembly.

- 241) Click **Save** .

Close all files.

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

