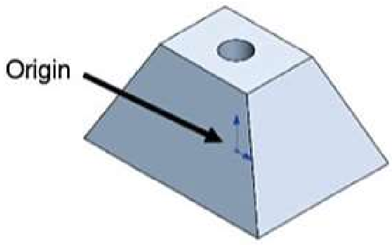
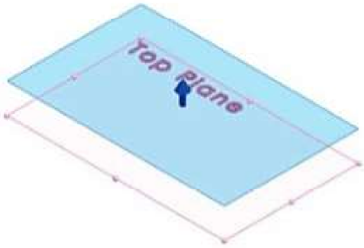
WEIGHT Part

The WEIGHT part is a machined part. Utilize the Lofted feature. Create a Loft by blending two or more profiles. Each profile is sketched on a separate plane.



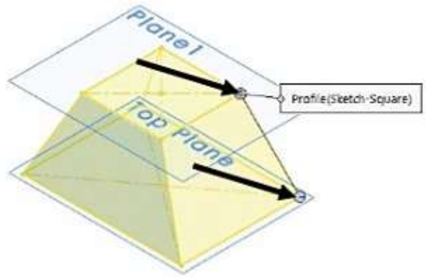
Create Plane1. Offset Plane1 from the Top Plane.



Sketch a rectangle for the first profile on the Top Plane.

Sketch a square for the second profile on Plane1.

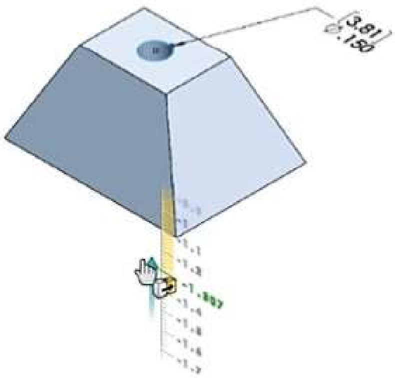
Select the corner of each profile to create the Lofted feature.



Utilize the Instant3D tool to create an Extruded Cut feature to create a Through All hole centered on the top face of the Loft feature.

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

When using the Instant3D tool, you lose the ability to select various End Conditions to maintain design intent.

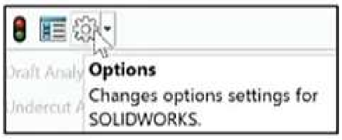


All parts in this chapter utilize a custom part template. Create the custom part template from the default part template. Save the Custom Part template in the SW-TUTORIAL-2020\MY-TEMPLATE folder. If needed, create the SW-TUTORIAL-2020\MY-TEMPLATE folder.

Activity: Create the WEIGHT Part

Create a New part template.

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



2) Double-click **Part** from the Templates tab. The Part FeatureManager is displayed.

Set Document Properties. Set drafting standard.

3) Click **Options** from the Main menu.

4) Click the **Document Properties** tab from the dialog box.

5) Select **ANSI** from the Overall drafting standard drop-down menu.

Set document units and precision.

6) Click **Units**.

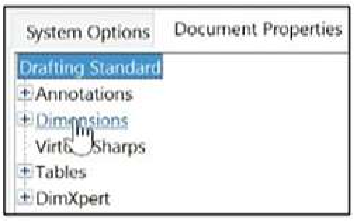
7) Select **IPS**, [**MMGS**] for Unit system.

8) Select **.123**, [**.12**] for linear units Decimal places.

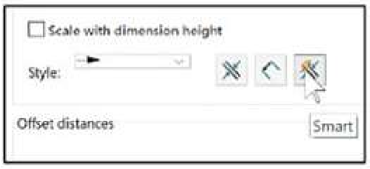
9) Select **None** for Angular units Decimal places.

Set Leader arrow direction.

10) Click **Dimensions**. Check the **Smart** box as illustrated.



11) Click **OK** from the Document Properties - Detailing - Dimensions dialog box.



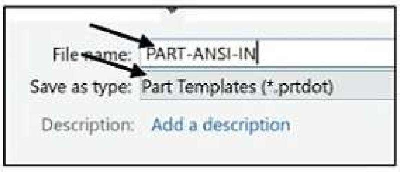
Save the Part template. Enter name.

12) Click **Save As**.

13) Select **Part Templates (*.prtdot)** for Save as type.

14) Select **SW-TUTORIAL-2020\MY-TEMPLATES** for Save in folder.

15) Enter **PART-ANSI-IN**, [PART-ANSI-MM] for File name.



16) Click **Save**.

Close the Part template.

17) Click **File**, **Close** from the Menu bar.

Create a New part.

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

19) Click the **SW-TUTORIAL-2020\MY-TEMPLATES** tab. Note: Additional templates are displayed.

20) Double-click **PART-ANSI-IN**, [PART-ANSI-MM]. The Part FeatureManager is displayed.

Save the part. Enter name and description.

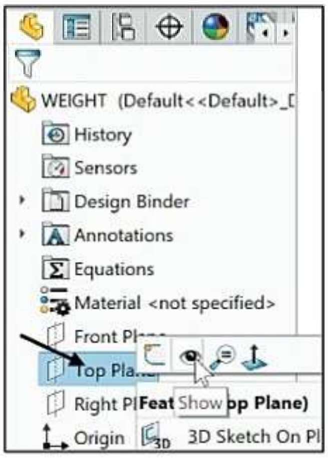
21) Click **Save As**.

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

23) Enter **WEIGHT** for File name.

24) Enter **WEIGHT** for Description.

25) Click **Save**.

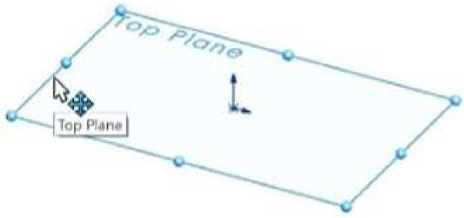


Insert Plane1. Display and Isometric view.

26) Click the **Isometric view** icon.

27) Right-click **Top Plane** from the FeatureManager.

28) Click **Show**. The Top Plane is displayed in the Graphics window.



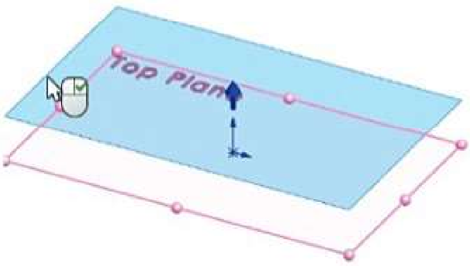
29) Hold the **Ctrl** key down.

30) Click the **boundary** of the Top Plane as illustrated.

31) Drag the **mouse pointer** upward.

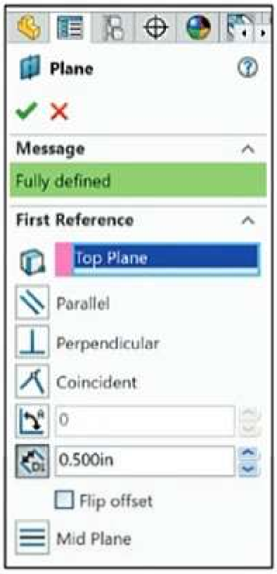
32) Release the **mouse pointer**.

33) Release the **Ctrl** key. The Plane PropertyManager is displayed. Top Plane is displayed in the First Reference box.



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

34) Enter **.500**in, [**12.70**] for Distance.



35) Click **OK** from the Plane PropertyManager.

Plane1 is displayed in the Graphics window and is listed in the FeatureManager. Plane1 is offset from the Top Plane.

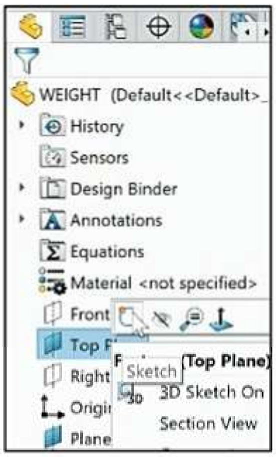
A Lofted feature requires two sketches. The first sketch, Sketch1, is a rectangle sketched on the Top Plane centered about the Origin. The second sketch, Sketch2, is a square sketched on Plane1 centered about the Origin.

Create Sketch1 in the Top Plane.

36) Right-click **Top Plane** from the FeatureManager.

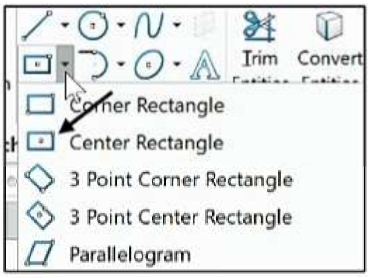
37) Click **Sketch** from the Context toolbar. The Sketch toolbar is displayed.

38) Click **Center Rectangle** from the Consolidated Sketch tool. The Center Rectangle icon is displayed.

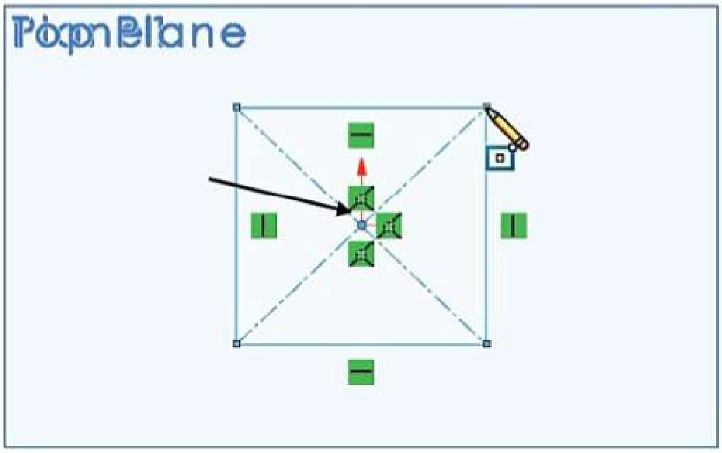


39) Click the **Origin**.

40) Click a **position** to the top right as illustrated.

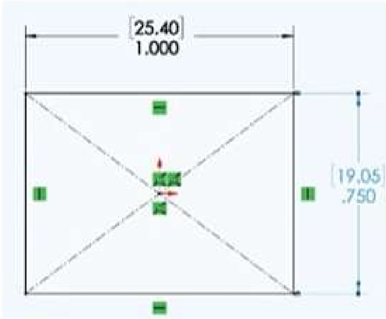


The Center Rectangle tool provides the ability to sketch a rectangle located at a center point, in this case the Origin. This eliminates the need for centerlines to the Origin with a Midpoint geometric relation.



CommandManager and FeatureManager tabs and tree folders will vary depending on system setup and Add-ins.

Add dimensions.



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

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

43) Click a **position** above the line.

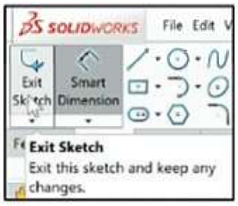
44) Enter **1.000**in, [25.40].

45) Click the **right vertical** line.

46) Click a **position** to the right.

47) Enter .**750**in, [19.05].

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

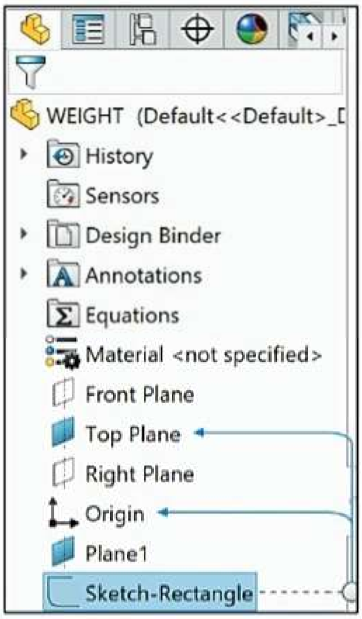


Close Sketch1.

49) Click **Exit Sketch** from the Sketch toolbar. The sketch is fully defined and is displayed in black.

Rename Sketch1.

50) Rename **Sketch1** to **Sketch-Rectangle**.



Save the part.

51) Click **Save** .

Display an Isometric view.

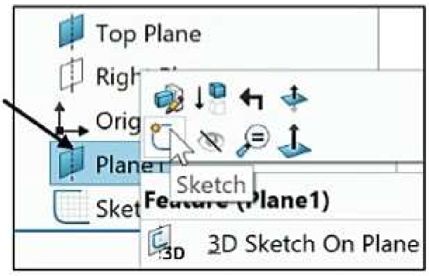
52) Click **Isometric view** from the Heads-up View toolbar.

Create Sketch2 on Plane1. Plane1 is your Sketch plane.

53) Right-click **Plane1** from the FeatureManager. Plane1 is your Sketch plane.

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

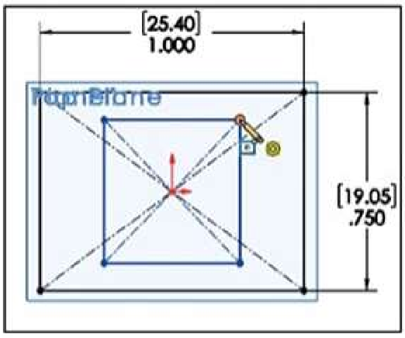
55) Click the **Center Rectangle** Consolidated Sketch tool. The Center Rectangle icon is displayed.



56) Click **Top View**.

57) Click the **Origin** .

58) Click a **position** as illustrated.



59) Right-click **Select** to de-select the Center Rectangle tool.

If needed, add an Equal relation between the left vertical line and the top horizontal line.

60) Click the **left vertical line** of the rectangle.

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

62) Click the **top horizontal line** of the rectangle.

63) Release the **Ctrl** key.

64) Click **Equal** = from the Add Relations box.

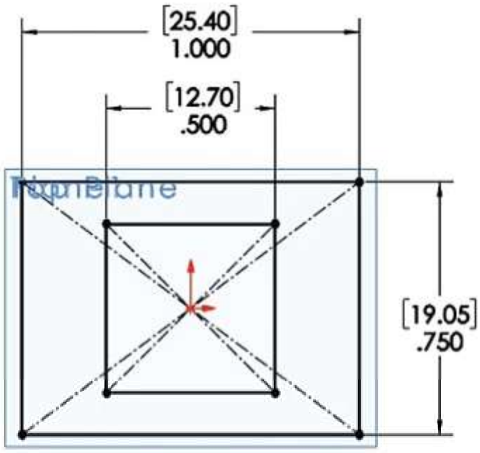
65) Click **OK** from the Properties PropertyManager.

Add a dimension.

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

67) Click the **top horizontal** line. Click a **position** above the line.

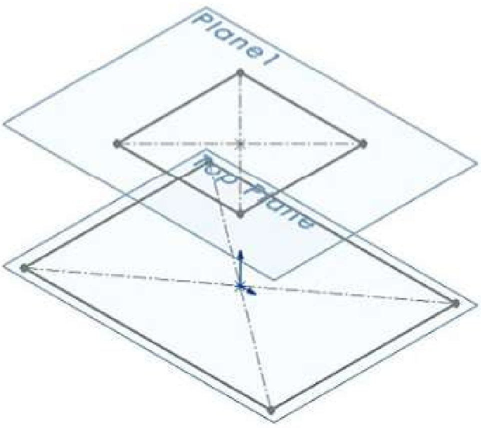
68) Enter **.500**in, **[12.70]**. Click the **Green Check mark**. The sketch is displayed in black.



Close Sketch2. Display an Isometric view.

69) Click **Exit Sketch** from the Sketch toolbar. Sketch2 is fully defined.

70) Click **Isometric view** from the Heads-up View toolbar. View the results in the Graphics window.

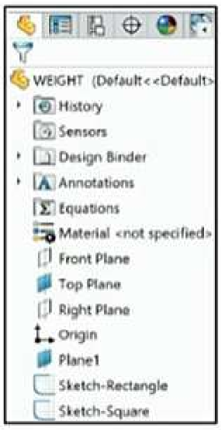


If you did not select the Origin, insert a Coincident relation between the rectangle and the Origin to fully define Sketch2.

Think design intent. When do you use the various End Conditions and Geometric sketch relations? What are you trying to do with the design? How does the component fit into an assembly?

Rename Sketch2.

71) Rename **Sketch2** to **Sketch-Square**.



Save the WEIGHT part.

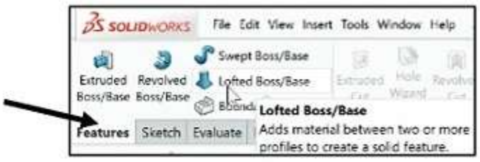
72) Click **Save**.

Lofted features are comprised of multiple sketches. Name sketches for clarity.

Activity: WEIGHT Part - Lofted Feature

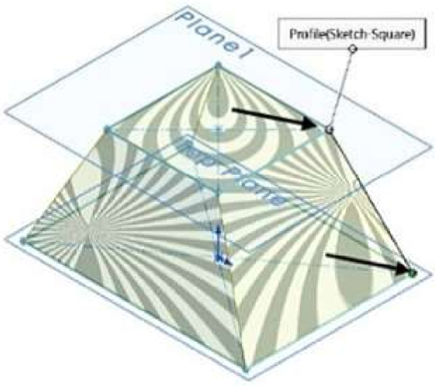
Insert a Lofted feature.

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



74) Click the **Lofted Boss/Base** Feature tool. The Loft PropertyManager is displayed.

75) Clear the **Profiles** box.



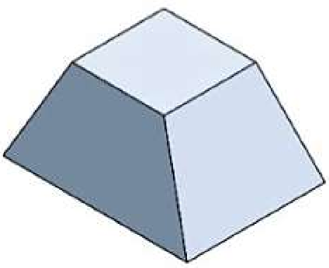
76) Click the **back right corner** of Sketch-Rectangle as illustrated.

77) Click the **back right corner** of Sketch-Square. Sketch-Rectangle and Sketch-Square are displayed in the Profiles box.

78) Click **OK** from the Loft PropertyManager. Loft1 is displayed in the FeatureManager.

79) **Hide** all planes.

To display the Selection Filter toolbar, right-click in the Graphics window, and click the **Selection Filters** drop-down menu icon. The Selection Filter is displayed.



To clear a Filter icon, click **Clear All Filters** from the Selection Filter toolbar.

80) **Expand** Loft1 in the FeatureManager. Sketch-Rectangle and Sketch-Square are the two sketches that contain the Loft feature.

81) **Zoom in** on the Loft1 feature.

Activity: WEIGHT Part - Instant3D - Extruded Cut Feature

Insert a New sketch for the Extruded Cut feature.

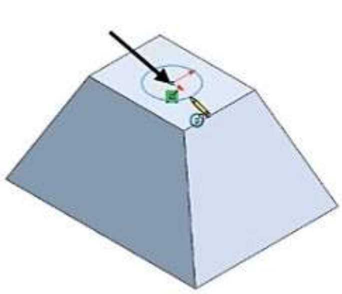
82) Right-click the **top square face** of the Loft1 feature for the Sketch plane.

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

84) Click the **Circle** Sketch tool. The circle icon is displayed.

85) Click the **center** as illustrated.

86) Click a **position** to the right as illustrated.



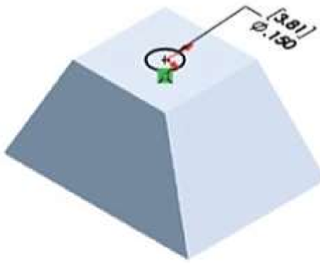
Add a dimension.

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

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

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

90) Enter **.150**in, [**3.81**] in the Modify box.

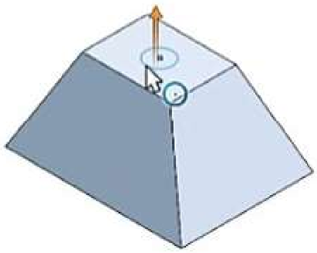


Insert an Extruded feature using the Instant3D tool.

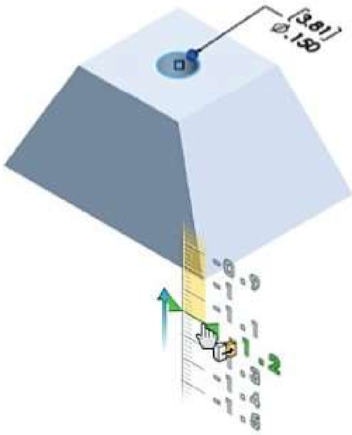
91) **Exit** the Sketch. By default, Instant3D is active.

92) Click the **diameter** of the circle, Sketch3, as illustrated.

93) Click the **Arrowhead** and drag it below the model.



94) Click a **position** on the Instant3D ruler. The Extrude feature is displayed in the FeatureManager.

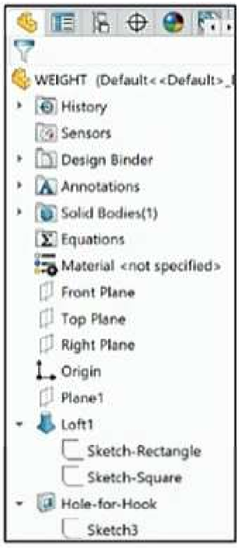


Display Wireframe style.

95) Click **Wireframe** from the Heads-up View toolbar. View the Extrude feature.

Rename the Extrude feature.

96) Rename the **Extrude** feature to **Hole-for-Hook**.

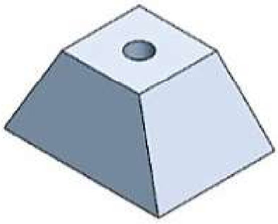


Display an Isometric view. Display Shaded With Edges. Save the WEIGHT part.

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

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

99) Click **Save**. The WEIGHT part is complete. Later, apply material to the part.



Review the WEIGHT Part

The WEIGHT part was created with the Loft feature. The Loft feature required two planes: Top Plane and Plane1. Profiles were sketched on each plane. Profiles were selected to create the Loft feature.

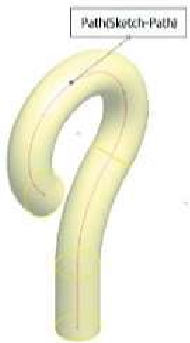
An Extruded Cut feature was created using the Instant3D tool to create a Through All center hole in the WEIGHT.

HOOK Part

The HOOK part fastens to the WEIGHT. The HOOK is created with a Swept Base feature. The Swept Base feature adds material by moving a profile along a path.

The Swept Base feature requires two sketches (path and profile) or a sketch path and a circular profile diameter. If the sketch profile is a circle, enter the circular profile diameter in the Swept PropertyManager.

For non-circular sketch profiles, create the sketch on a perpendicular plane to the path and use the pierce relation to locate the profile on the path.



Create the HOOK part with a Swept Base feature.

The Swept Base feature uses:

- A path sketched on the Right Plane.
- A circular profile diameter.

Utilize the Dome feature tool to create a spherical feature on a circular face.

Utilize the Thread feature tool to create a right-hand #8-36 thread for the HOOK. The Thread tool can add or removes material.

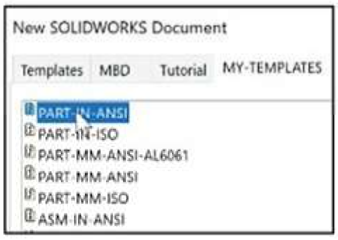
Reference geometry defines the shape or form of a surface or a solid. Reference geometry includes planes, axes, coordinate systems and points.

Activity: Create the HOOK Part

Create the New part.

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

101) Select the **SW-TUTORIAL-2020\MY-TEMPLATES** tab. Additional templates are displayed.



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

Save the part. Enter name. Enter description.

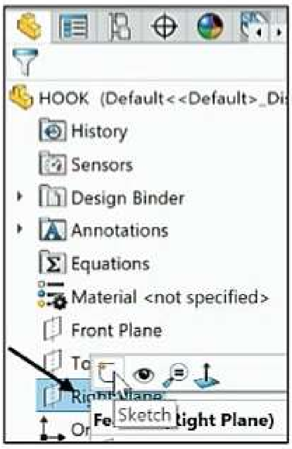
103) Click **Save As**.

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

105) Enter **HOOK** for File name.

106) Enter **HOOK** for Description.

107) Click **Save**. The HOOK FeatureManager is displayed.



If the sketch profile is a circle, enter the diameter in the Swept PropertyManager. For non-circular sketch profiles, create the sketch on a perpendicular plane to the path and use the pierce relation to locate the profile on the path.

Sketch1 is the Sweep Path sketched on the Right Plane.

Sketch the Sweep Path.

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

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

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

111) Sketch a **vertical line** from the Origin as illustrated.

Add a dimension.

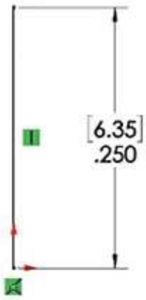
112) Click the **Smart Dimension** Sketch tool.

113) Click the **vertical line**.

114) Click a **position** to the right.

115) Enter **.250**in, **[6.35]**.

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

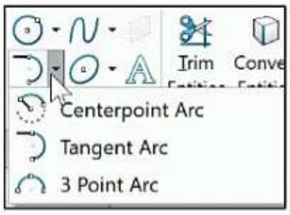


Fit the model to the Graphics window.

117) Press the **f** key.

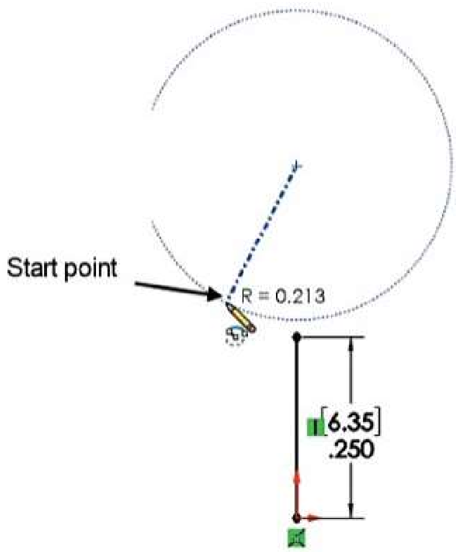
Create the Centerpoint arc.

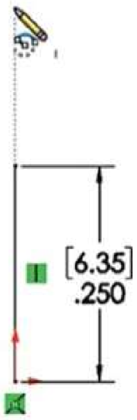
118) Click the **Centerpoint Arc** Sketch tool from the Consolidated Sketch toolbar. The Centerpoint Arc icon is displayed.



119) Click the **arc center point** vertically aligned to the Origin as illustrated.

120) Click the **arc start point** as illustrated.





121) Move (do not drag) the **mouse pointer** clockwise approximately 270°.

122) Click a point **horizontally aligned** to the arc start point. If needed add a horizontal relationship.

123) Click the **3 Point Arc** Sketch tool from the Consolidated Sketch toolbar. The Arc PropertyManager is displayed.

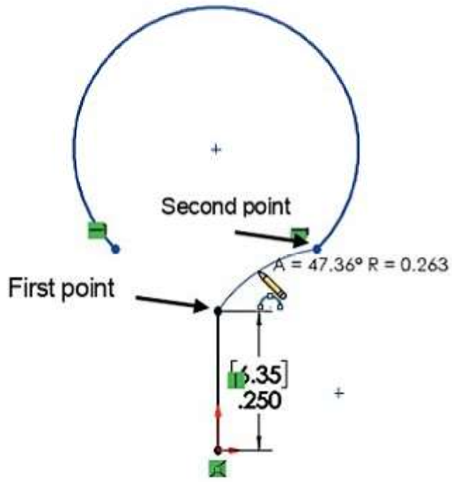
124) Click the **vertical line** endpoint.

125) Click the **center point arc** endpoint.

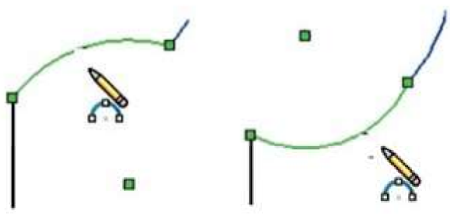
126) Drag and pull the center of the **3 Point Arc downwards**.

127) Click the center of the **center point arc line** as illustrated.

128) Click **OK** from the Arc PropertyManager.



It is important to draw the correct shape with the 3 Point Arc tool as illustrated.



Correct shape **Incorrect shape**

Add a Vertical relation between the Origin and the center point of the arc.

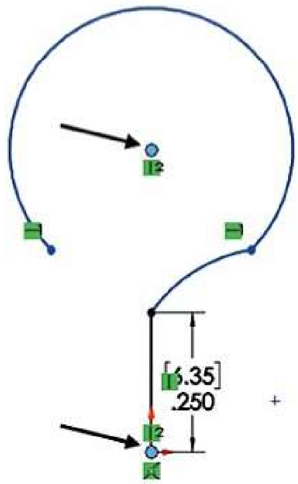
129) Click the **Origin**.

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

131) Click the **center point** of the Center point arc.

132) Release the **Ctrl** key.

133) Click **Vertical** from the Add Relations box.



If needed, add a Horizontal relation.

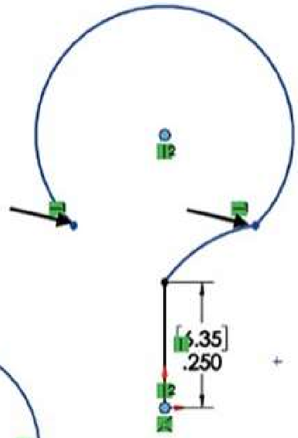
134) Click the **start point** of the Center point arc.

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

136) Click the **end point** of the Center point arc.

137) Release the **Ctrl** key.

138) Click **Horizontal** — from the Add Relations box.



Add a Tangent relation.

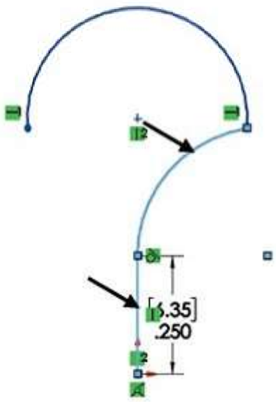
139) Click the **vertical line.**

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

141) Click the **3 Point Arc.**

142) Release the **Ctrl** key.

143) Click **Tangent** from the Add Relations box.



Add a second Tangent relation.

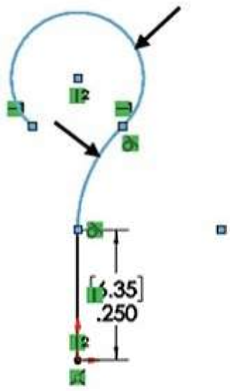
144) Click the **3 Point Arc.**

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

146) Click the **Center point** arc.

147) Release the **Ctrl** key.

148) Click **Tangent** from the Add Relations box.



Add dimensions.

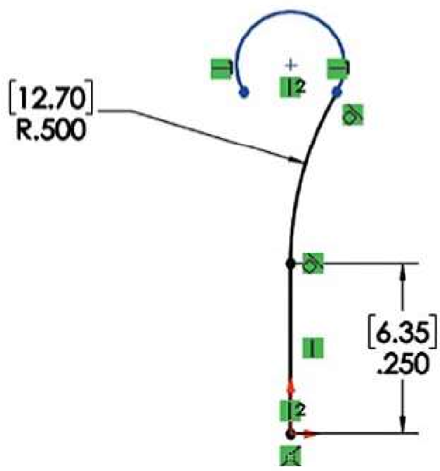
149) Click the **Smart Dimension** Sketch tool.

150) Click the **3 Point Arc.**

151) Click a **position** to the left.

152) Enter **.500**in, **[12.70]**.

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

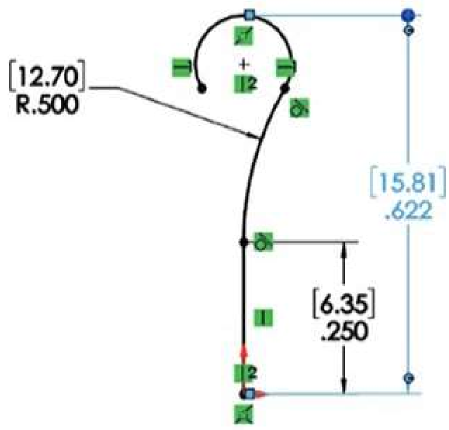


Dimension the overall length of the sketch.

154) Click the **top of the arc**.

155) Click the **Origin**.

156) Click a **position** to the right of the profile. Accept the default dimension.



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

Modify the overall length.

158) Double-click the default **dimension**.

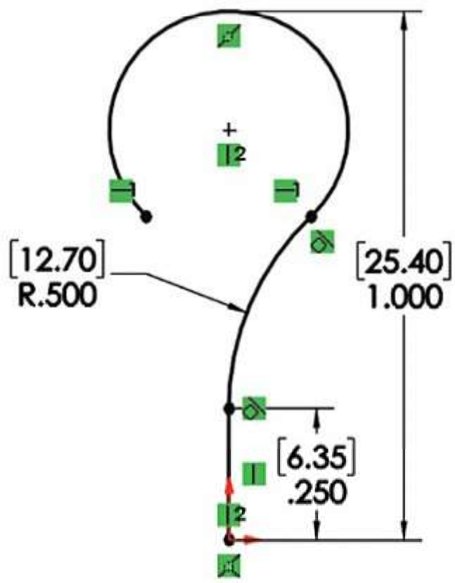
159) Enter **1.000**in, **[25.40]**.

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

Fit the model to the Graphics window.

161) Press the **f** key.

162) Move the **dimensions** as illustrated.



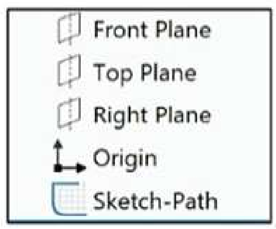
By default, the Dimension tool utilizes the center point of an arc or circle. Select the circle profile during dimensioning. Utilize the Leaders tab in the Dimension PropertyManager to modify the arc condition to Minimum or Maximum.

Close the sketch.

163) Click **Exit Sketch** from the Sketch toolbar.

Rename Sketch1.

164) Rename **Sketch1** to **Sketch-Path**.



Save the HOOK.

165) Click **Save**.

Activity: HOOK Part - Swept Base Feature

Insert the Swept feature.

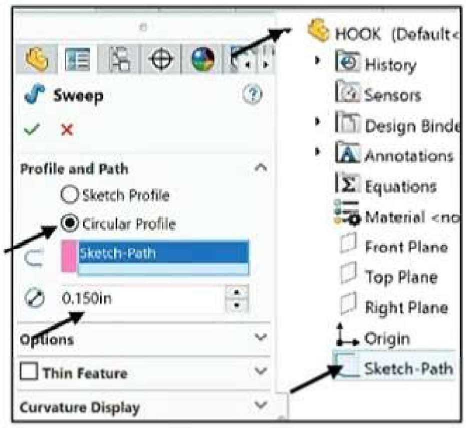
166) Click the **Swept Boss/Base** Features tool. The Sweep PropertyManager is displayed.

167) Select **Circular Profile**.

168) Click **Sketch-Path** from the fly-out FeatureManager. Sketch-Profile is displayed in the Profile box.

169) Enter **0.150**in diameter.

170) Click **OK** from the Sweep PropertyManager. Sweep1 is displayed in the FeatureManager.



Save the HOOK part.

171) Click **Save**.

Activity: HOOK Part - Dome Feature

Insert a Dome feature.

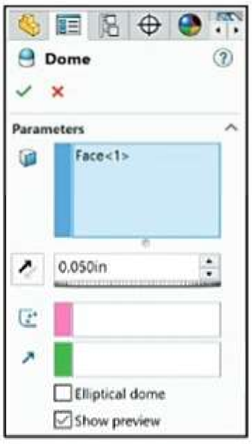
172) **Rotate** the model with the middle mouse button.

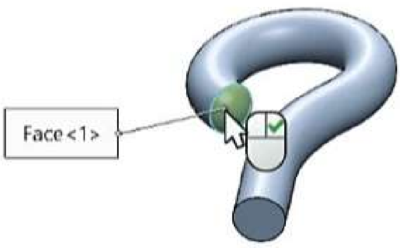
173) Click the **flat face** of the Sweep1 feature in the Graphics window as illustrated.

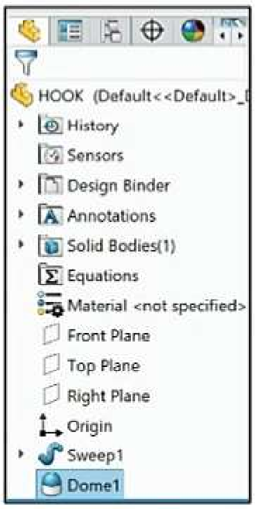
174) Click the **Dome** Features tool (Insert, Features, Dome). The Dome PropertyManager is displayed. Face<1> is displayed in the Parameters box.

175) Enter **.050**in, [**1.27**] for Distance.

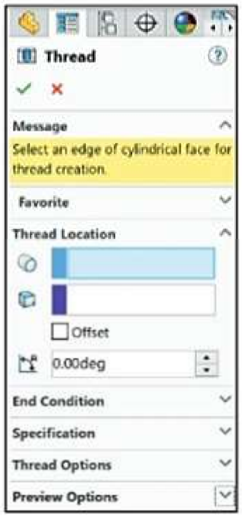
176) Click **OK** from the Dome PropertyManager. Dome1 is displayed in the FeatureManager.







The HOOK requires a simplified right-hand #8-36 thread from the bottom of the Sweep1 feature. Utilize the Thread feature. The Thread feature provides the ability to create helical threads on cylindrical faces using profile sketches. Store custom thread profiles as library features.



The Thread tool Type and Size profiles are nominal thread profiles only. Do not use them for production-quality threads. To create production-quality threads, modify the nominal profiles to meet your design requirements.

Define the start thread location, specify an offset, set end conditions, specify the type, size, diameter, pitch and rotation angle, and choose options such as right-hand or left-hand thread.

Activity: HOOK Part - Thread Feature

Create a right-hand #8-36 thread from the bottom of the Sweep1 feature.

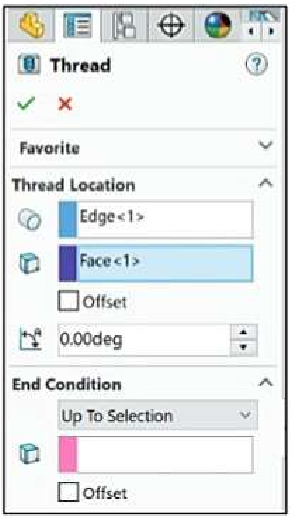
177) Click **Thread** from the Features toolbar.

178) Click **OK**. The Thread PropertyManager is displayed.

179) Click the **bottom circular edge** of Sweep1. Edge <1> is displayed in the Edge of cylinder box.

180) Click inside the **Optional start location** box.

181) Click the **bottom face**. Face<1> is displayed.

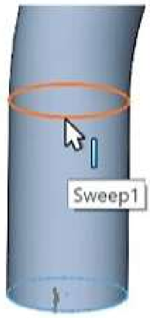


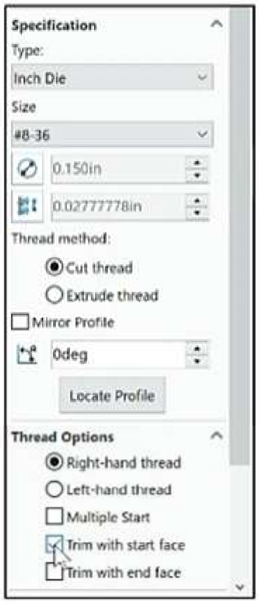
Select End Condition.

182) Select **Up To Selection** for End Condition.

183) Click inside the **End Condition** box.

184) Click the **ending edge** as illustrated. Edge<2> is displayed in the End Condition box.





Set Thread Specification.

185) Click **Inch Die** from the drop-down menu.

Set Thread Size.

186) Click **#8-36** for size from the drop-down menu. View the Override diameter and Override pitch number.

Set the Thread method.

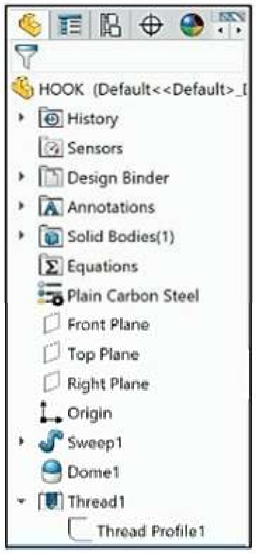
187) Click the **Cut thread** box.

Create a Right-hand thread.

188) Click **Right-hand thread** as illustrated.

189) Click the **Trim with start face** box.

190) Click **OK** from the Thread PropertyManager. Thread1 is displayed in the FeatureManager.



Apply Material.

191) Right-click the **Material** folder in the FeatureManager.

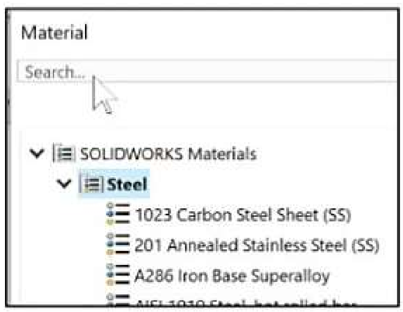
192) Click **Edit Material**. The Material dialog box is displayed.

193) Apply **Plain Carbon Steel**.

Display an Isometric view. Save the model.

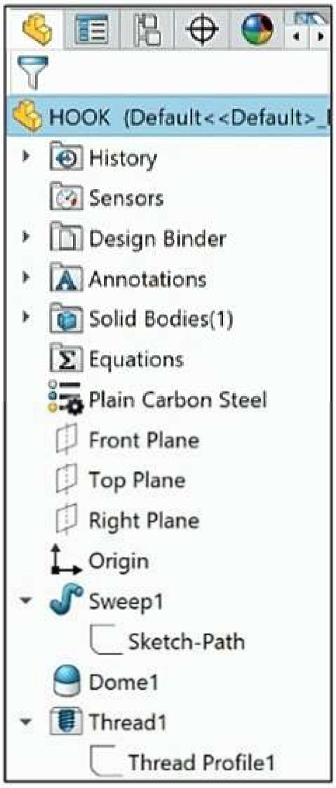
194) Click **Isometric view** from the Heads-up View toolbar.

195) Click **Save**. The HOOK part is finished.



Utilize the new Search feature in the Material dialog box to quickly locate the desired material.

Review the HOOK Part



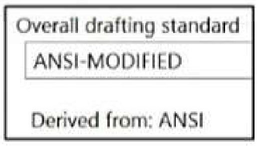
The HOOK part was created with a Swept, Dome, and Thread feature. A Swept Base feature added material by moving a profile along a path. The Swept Base feature requires two sketches (path and profile) or a sketch path and a circular profile diameter.

If the sketch profile is circular, enter the diameter in the Swept PropertyManager. For non-circular sketch profiles, create the sketch on a perpendicular plane to the path and use the pierce relation to locate the profile on the path.

The Dome feature created a spherical face on the end of the Swept Base feature.

The Thread feature provides the ability to create helical threads on cylindrical faces using profile sketches. Store custom thread profiles as library features.

The Thread tool Type and Size profiles are nominal thread profiles only. Do not use them for production quality threads. To create production quality threads,modify the nominal profiles to meet your design requirements.



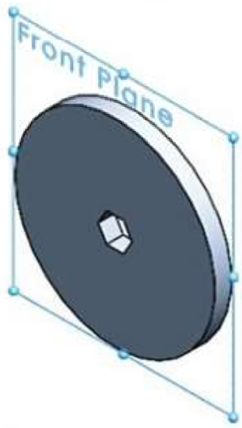
If you modify a document property from an Overall drafting standard, a modify message is displayed as illustrated.

When you create a new part or assembly, the three default Planes (Front, Right and Top) are aligned with specific views. The Plane you select for the Base sketch determines the orientation of the part.

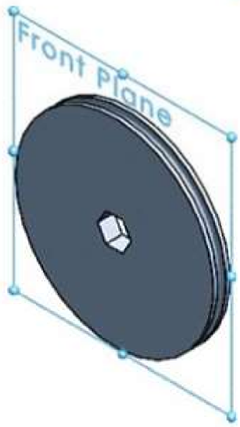
WHEEL Part

The WHEEL part is a machined part.

Create the WHEEL part with the Extruded Boss/Base feature tool. Utilize the Mid Plane option to center the WHEEL on the Front Plane.



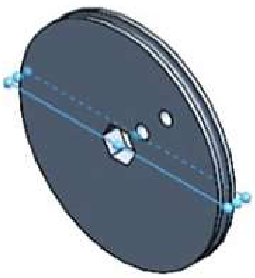
Utilize the Revolved Cut feature tool to remove material from the WHEEL and to create a groove for a belt.



The WHEEL contains a complex pattern of holes. Apply the Extruded Cut feature tool.

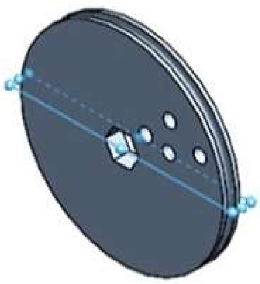
Simplify the geometry by dividing the four holes into two Extruded Cut features.

The first Extruded Cut feature contains two small circles sketched on two bolt circles. The bolt circles utilize Construction geometry.



Utilize the Hole Wizard feature when creating non-Through All complex geometry holes.

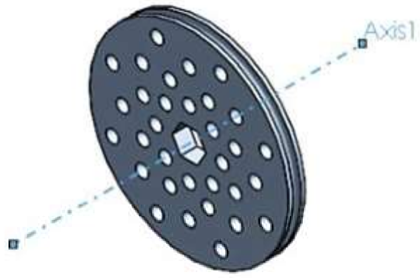
The second Extruded Cut feature utilizes two small circles sketched on two bolt circles. The bolt circles utilize Construction geometry.



Utilize the Circular Pattern Feature tool. The two Extruded Cut features are contained in the Circular Pattern. Revolve the Extruded Cut features about the Temporary Axis located at the center of the Hexagon.

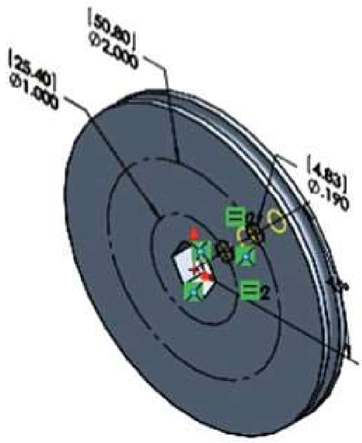
Create a Reference Axis. The Reference Axis is utilized in the WHEEL-AXLE assembly.

Construction geometry is used only to assist in creating the sketch entities and geometry that are ultimately incorporated into the part. Construction geometry is ignored when the sketch is used to create a feature. Construction geometry uses the same line style as centerlines.



You can utilize the Hole Wizard feature tool instead of the Cut-Extrude feature tool, or use the Instant3D tool to create a Through All hole for any part. See SOLIDWORKS Help for additional information.

Slots are available in the Hole Wizard. Create regular slots as well and counterbore and countersink slots. You also have options for position and orientation of the slot. If you have hardware already mated in place, the mates will not be broken if you switch from a hole to a slot.



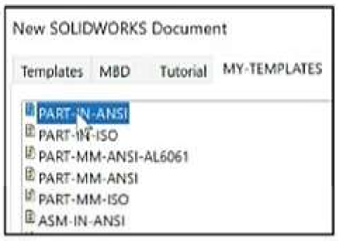
Activity: WHEEL Part

Create the New part.

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

197) Click the **SW-TUTORIAL-2020\MY-TEMPLATES** tab. Additional templates are displayed.

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



Save the part. Enter name. Enter description.

199) Click **Save As**.

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

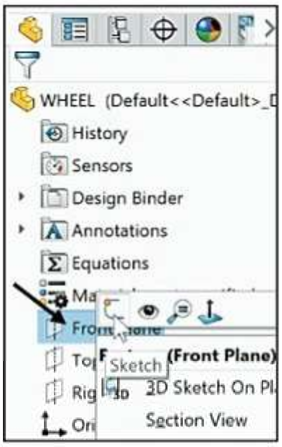
201) Enter **WHEEL** for File name.

202) Enter **WHEEL** for Description.

203) Click **Save**. The WHEEL FeatureManager is displayed.

Insert the sketch for the Extruded Base feature.

204) Right-click **Front Plane** from the FeatureManager.

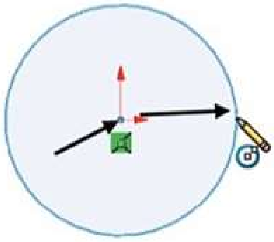


205) Click **Sketch** from the Context toolbar. The Sketch toolbar is displayed.

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

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

208) Click a **position** to the right of the Origin.



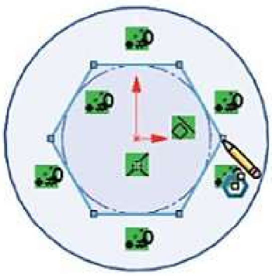
Insert a polygon.

209) Click the **Polygon** Sketch tool. The Polygon PropertyManager is displayed.

210) Click the **Origin**.

211) Drag and click the **mouse pointer** horizontally to the right of the Origin to create the hexagon as illustrated.

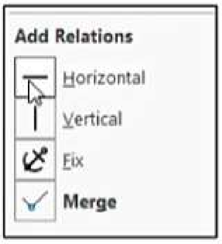
212) Click **OK** from the Polygon PropertyManager.

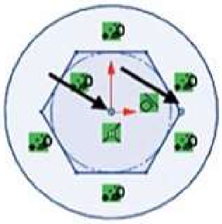


De-select the Polygon Sketch tool.

213) Right-click **Select**.

Add a Horizontal relation.





214) Click the **Origin**.

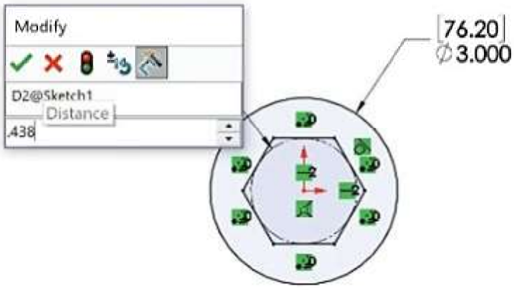
215) Hold the **Ctrl** key down.

216) Click the **right point** of the hexagon.

217) Release the **Ctrl** key.

218) Click **Horizontal** from the Add Relations box.

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



Add dimensions.

220) Click the **Smart Dimension** Sketch tool. Click the **circumference** of the large circle.

221) Click a **position** above the circle. Enter **3.000**in, [**76.20**].

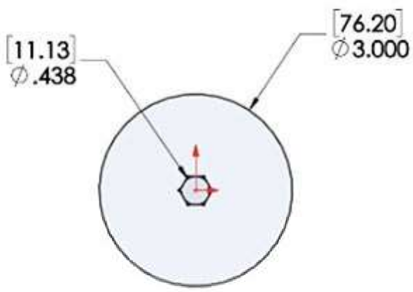
222) Click the **Green Check mark**.

223) Click the **circumference** of the inscribed circle for the Hexagon.

224) Click a **position** above the Hexagon.

225) Enter **.438**in, [**11.13**].

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



Activity: WHEEL Part - Extruded Boss/Base Feature

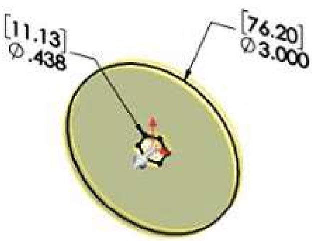
Insert an Extruded Boss/Base feature.

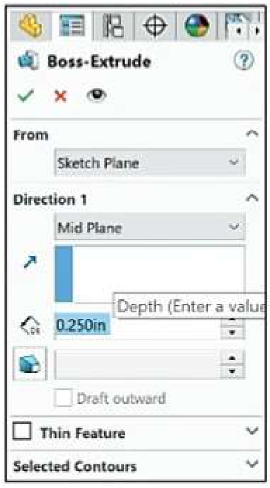
227) Click **Extruded Boss/Base** from the Features toolbar. The Boss-Extrude PropertyManager is displayed.

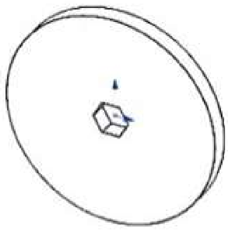
228) Select **Mid Plane** for End Condition in Direction 1.

229) Enter **.250**in, [6.35] for Depth.

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







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

Display Hidden Lines Removed. Save the WHEEL part.
232) Display **Hidden Lines Removed**.

233) Click **Save**.

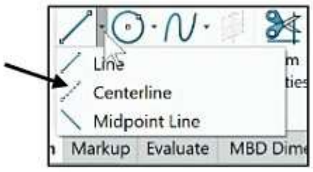
Activity: WHEEL Part - Revolved Cut Feature

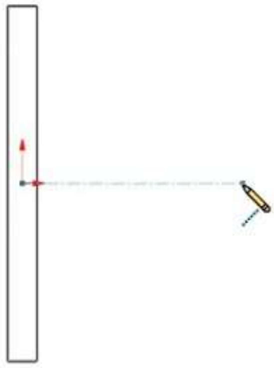
Insert a new sketch for the Revolved Cut feature.
234) Right-click **Right Plane** from the FeatureManager.

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

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

Sketch the axis of revolution.



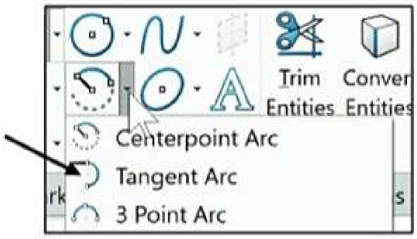


237) Click the **Centerline** Sketch tool from the Consolidated Sketch toolbar. The Insert Line PropertyManager is displayed.

238) Click the **Origin**.

239) Click a **position** horizontally to the right of the Origin as illustrated.

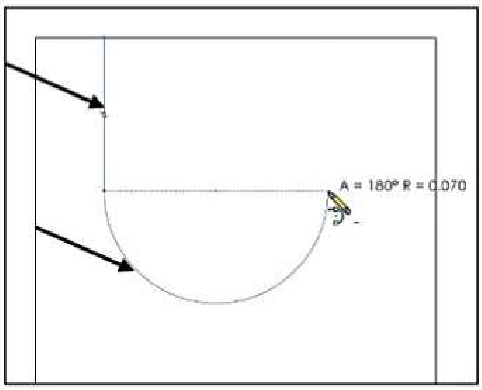
De-select the sketch tool.
240) Right-click **Select**.



241) **Zoom in** on the top edge.

Sketch the profile.
242) Click the **Line** Sketch tool.

243) Sketch the **first vertical line** as illustrated.



244) Click the **Tangent Arc** Sketch tool. The Arc PropertyManager is displayed.

245) Click the **end point** of the vertical line.

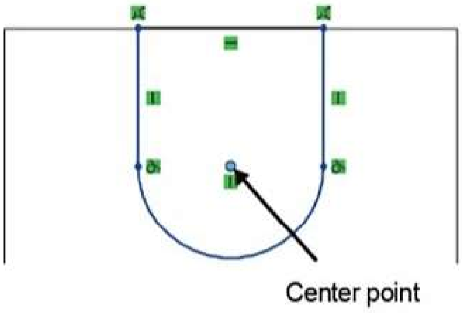
246) Sketch a **180° arc** as illustrated.

De-select the sketch tool.
247) Right-click **Select** in the Graphics window.

248) Click the **Line** Sketch tool.

249) Sketch the **second vertical line** as illustrated. The end point of the line is Coincident with the top horizontal edge of Extrude1.

250) Sketch a **horizontal line** collinear with the top edge to close the profile.



Add a Vertical relation.

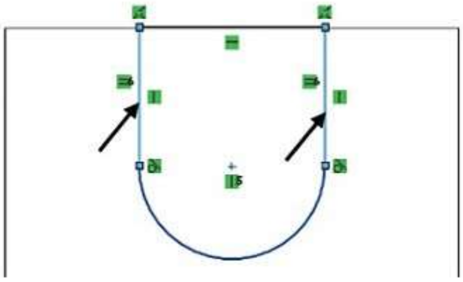
251) Right-click **Select** in the Graphics window.

252) Click the **Origin** from the FeatureManager.

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

254) Click the **center point** of the arc. Release the **Ctrl** key.

255) Click **Vertical** from the Add Relations box.



Add an Equal relation.

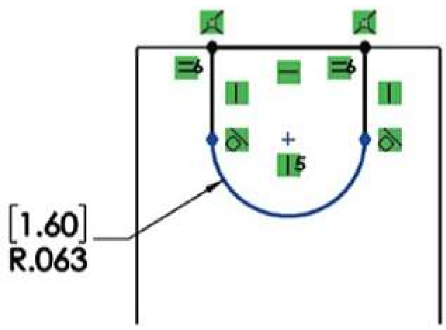
256) Click the **left vertical** line.

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

258) Click the **right vertical** line.

259) Release the **Ctrl** key.

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



Add dimensions.

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

262) Click the **arc**.

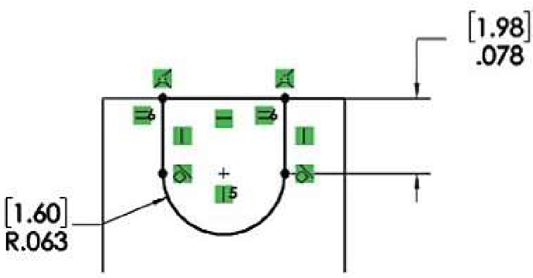
263) Click a position to the **left** of the profile.

264) Enter **.063**in, [**1.6**]. Click the **Green Check mark**.

265) Click the **right vertical** line.

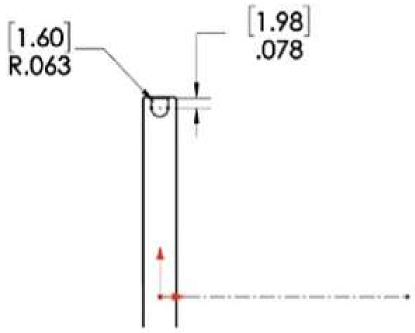
266) Click a position to the **right** of the profile.

267) Enter **.078**in, [**1.98**]. Click the **Green Check mark**. The sketch should be fully defined.



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

De-select the sketch tool.
269) Right-click **Select** in the Graphics window.



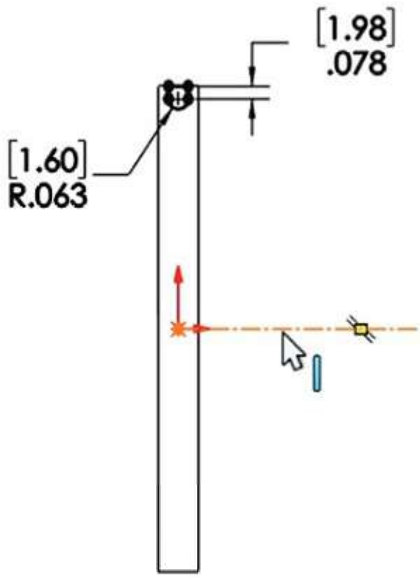
Activity: WHEEL Part - Revolved Cut Feature

Insert a Revolved Cut feature.

270) Select the Axis of Revolution. Click the **centerline** in the Graphics window as illustrated.

271) Click **Revolved Cut** from the Features toolbar. The Cut-Revolve PropertyManager is displayed. The Cut-Revolve PropertyManager displays 360 degrees for Direction 1 Angle.

272) Click **OK** from the Cut-Revolve PropertyManager. Cut-Revolve1 is displayed in the FeatureManager.



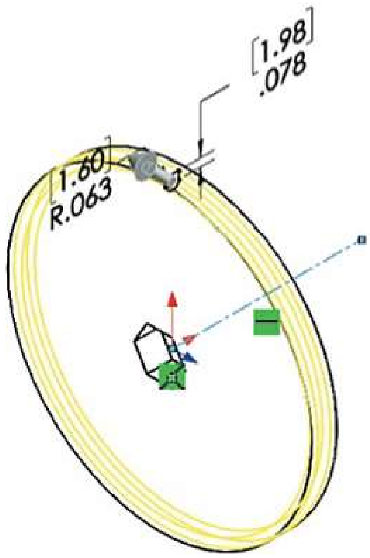
Save the WHEEL part.

273) Click **Save**.

Four bolt circles, spaced 0.5in, [12.7] apart, locate the 8 - ∅.190, [4.83] holes. Simplify the situation. Utilize two Extruded Cut features on each bolt circle.

Position the first Extruded Cut feature hole on the first bolt circle and third bolt circle.

Position the second Extruded Cut feature hole on the second bolt circle and fourth bolt circle.

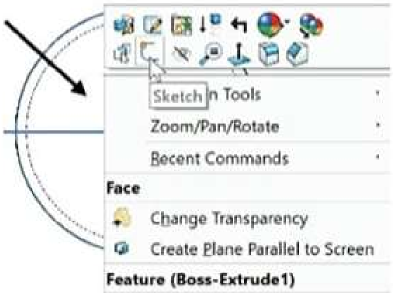


Activity: WHEEL Part - First Extruded Cut Feature

Display the Top Plane.

274) Right-click **Top Plane** from the FeatureManager.

275) Click **Show** from the Context toolbar.



Display a Front view - Hidden Lines Visible.

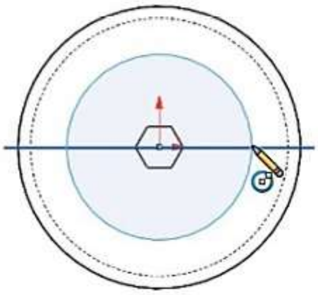
276) Click **Front view** from the Heads-up View toolbar.

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

Insert a new sketch for the first Extruded Cut feature.

278) Right-click the **Boss-Extrude1 front face** as illustrated.

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



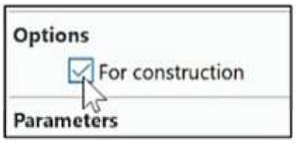
Create the first construction bolt circle.

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

281) Click the **Origin**.

282) Click a **position** to the right of the hexagon as illustrated.

283) Check the **For construction** box.

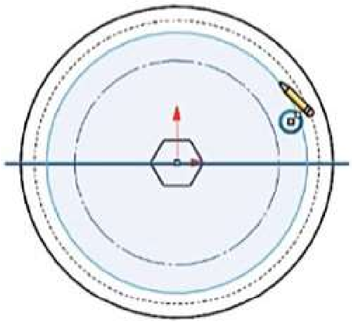


Create the second construction bolt circle.

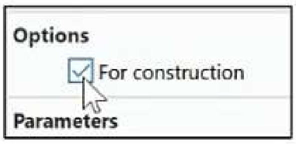
284) Click the **Origin**.

285) Click a **position** to the right of the first construction bolt circle as illustrated.

286) Check the **For construction** box. The two bolt circles are displayed with Construction style lines.



Construction geometry is used only to assist in creating the sketch entities and geometry that are ultimately incorporated into the part. Construction geometry is ignored when the sketch is used to create a feature. Construction geometry uses the same line style as centerlines.



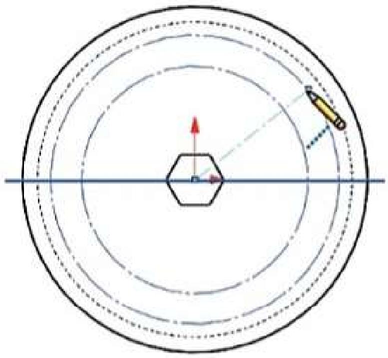
De-select the circle Sketch tool.

287) Right-click **Select**.

Insert a centerline.

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

289) Sketch a **45° centerline** (approximately) from the Origin to the second bolt circle as illustrated.

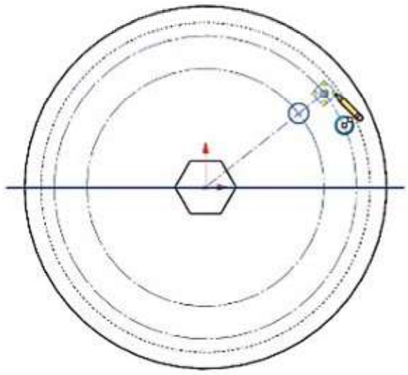


Sketch the two circle profiles.

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

291) Sketch a **circle** at the intersection of the centerline and the first bolt circle.

292) Sketch a **circle** at the intersection of the centerline and the second bolt circle.



De-select the Circle Sketch tool.

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

Note: An Intersection relation is created between three entities: the center point of the small circle, the centerline, and the bolt circle.

Add an Equal relation.

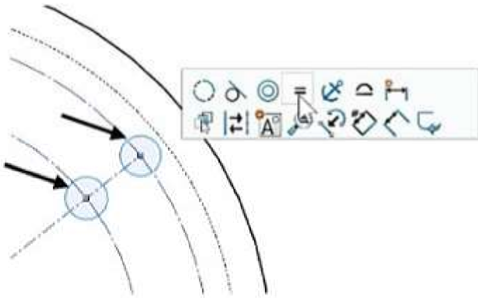
294) Click the **first circle**.

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

296) Click the **second circle**.

297) Release the **Ctrl** key.

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



Add dimensions.

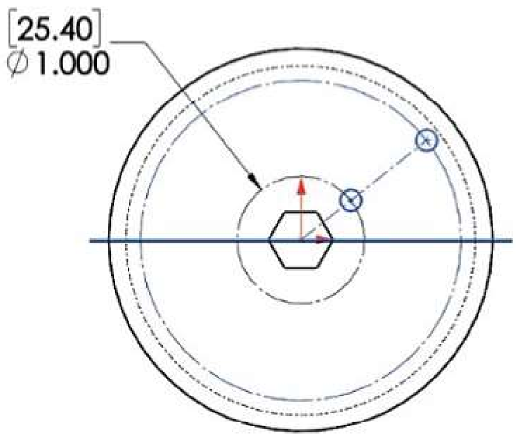
299) Click the **Smart Dimension** Sketch tool.

300) Click the **first construction** circle.

301) Click a **position** above the profile.

302) Enter **1.000**in, [**25.4**].

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



304) Click the **second construction** circle.

305) Click a **position** above the profile.

306) Enter **2.000**in, [**50.80**].

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

308) Click the **second small** circle.

309) Click a **position** above the profile.

310) Enter **.190**in, [**4.83**].

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

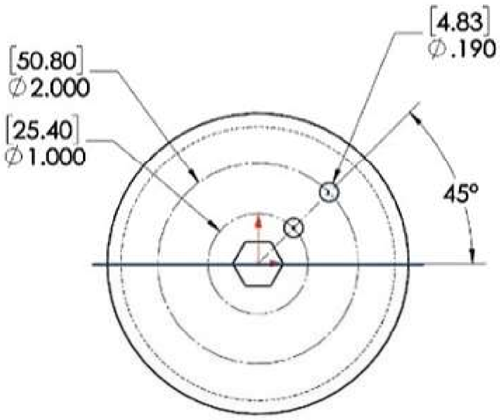
312) Click **Top Plane** from the fly-out FeatureManager.

313) Click the **45° centerline**.

314) Click a **position** between the two lines.

315) Enter **45**deg for angle.

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



Note: If the sketch is not fully defined, you may need to add an Intersection relation between the center point of the small circle, the centerline, and the bolt circle.

Insert an Extruded Cut feature.

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

318) Select **Through All** for the End Condition in Direction 1.

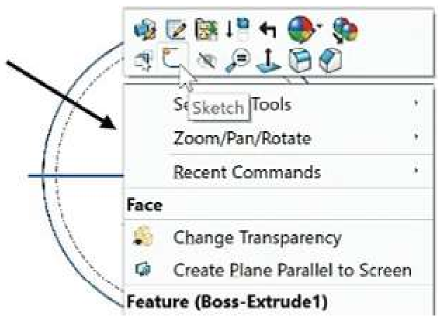
319) Click **OK** from the Cut-Extrude PropertyManager. Cut-Extrude1 is displayed in the FeatureManager.

Activity: WHEEL Part - Second Extruded Cut Feature

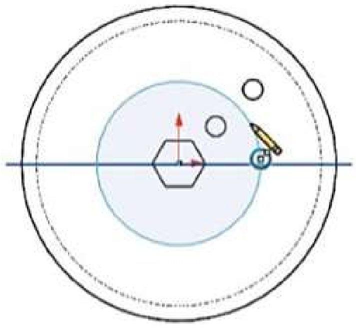
Insert a new sketch for the second Extruded Cut feature.

320) Right-click the **Boss-Extrude1** front face.

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



Sketch two additional Construction line bolt circles, 1.500in, [38.1] and 2.500in, [63.5]. Create the first Construction bolt circle.

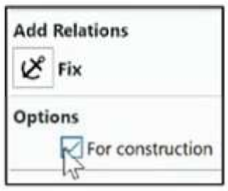


322) Click the **Circle** Sketch tool. The Circle PropertyManager is displayed.

323) Click the **Origin**.

324) Click a **position** between the two small circles.

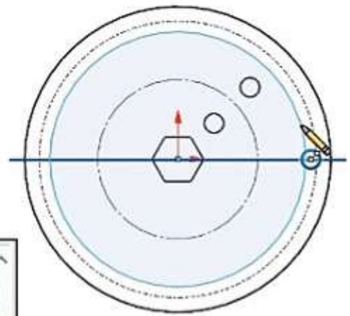
325) Check the **For construction** box.



Create the second additional construction bolt circle.

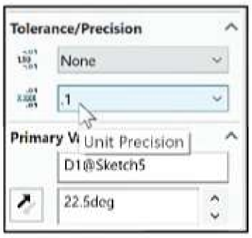
326) Click the **Origin**.

327) Click a **position** to the right of the large construction bolt circle as illustrated.



328) Check the **For construction** box from the Circle PropertyManager. The two bolt circles are displayed with the two construction lines.

Insert a centerline.



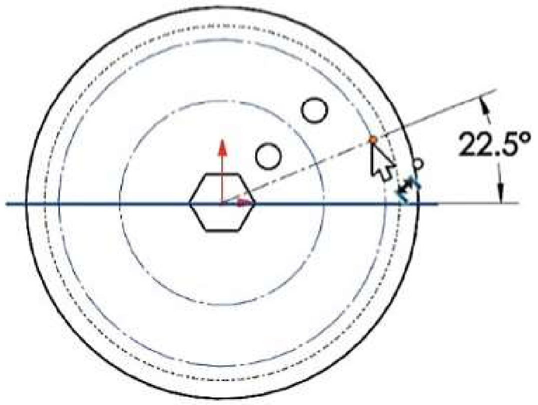
329) Click the **Centerline** Sketch tool. The Insert Line PropertyManager is displayed.

330) Sketch a **22.5° centerline** to the right from the Origin to the second bolt circle as illustrated.

331) Select **.1** from the Unit Precision box.

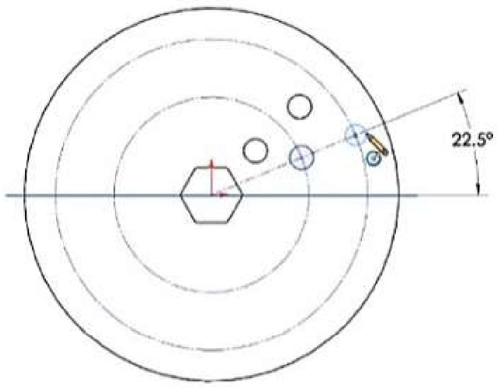
Display Hidden Lines Removed.

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



Sketch the two circle profiles.

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



334) Sketch a **circle** at the intersection of the centerline and the first bolt circle.

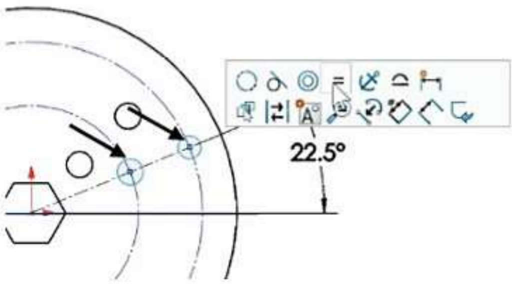
335) Sketch a **circle** at the intersection of the centerline and the second bolt circle.

De-select the Circle Sketch tool.

336) Right-click **Select**.

Add an Equal relation.

337) Click the **first circle**.



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

339) Click the **second circle**.

340) Release the **Ctrl** key.

341) Right-click **Make Equal** = from the shortcut toolbar.

Add dimensions.

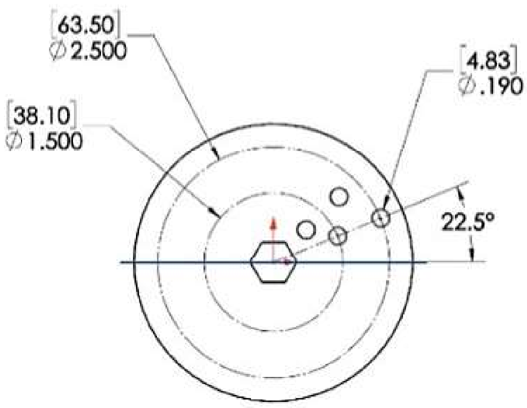
342) Click the **Smart Dimension** Sketch tool. The Smart Dimension icon is displayed.

343) Click the **first construction** circle.

344) Click a **position** above the profile.

345) Enter **1.500**in, **[38.1]**.

346) Click the **second construction** circle.



347) Click a **position** above the profile.

348) Enter **2.500**in, **[63.5]**.

349) Click the **small circle** as illustrated.

350) Click a **position** above the profile.

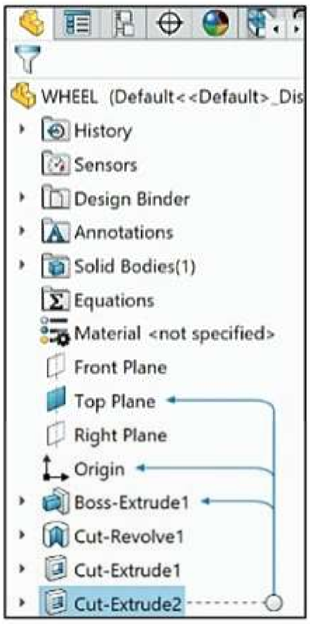
351) Enter **.190**in, **[4.83]**. The sketch should be fully defined.

Insert an Extruded Cut feature.

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

353) Select **Through All** for End Condition in Direction 1.

354) Click **OK** from the Cut-Extrude PropertyManager. Cut-Extrude2 is displayed in the FeatureManager.



Save the model.

355) Click **Save**.

View the Temporary Axes.

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

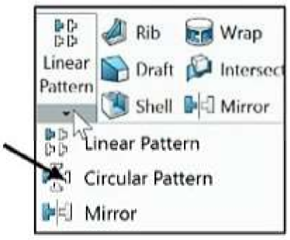
The book is designed to expose the new SOLIDWORKS user to many different tools, techniques and procedures. It may not always use the most direct tool or process.

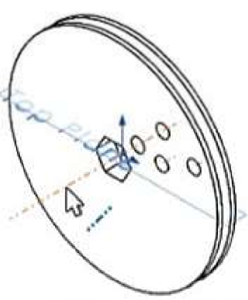
Activity: WHEEL Part - Circular Pattern Feature

Insert a Circular Pattern.

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

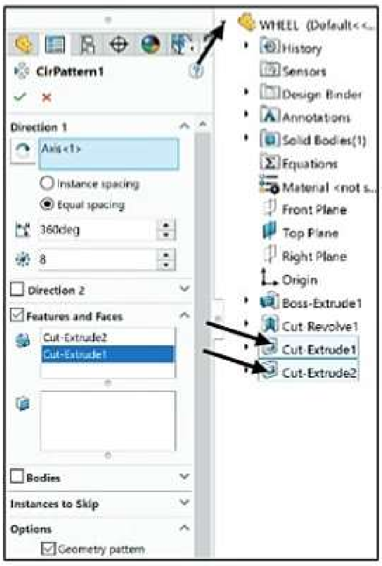
358) Click **Circular Pattern** from the Consolidated Features toolbar. The Circular Pattern PropertyManager is displayed.





359) Click **inside** the Pattern Axis box.

360) Click the **Temporary Axis** in the Graphics window at the center of the Hexagon. Axis<1> is displayed in the Pattern Axis box.



361) Click the **Equal spacing** box.

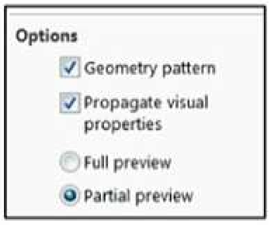
362) Enter **360**deg for Angle.

363) Enter **8** for Number of Instances.

364) Click inside the **Features to Pattern** box.

365) Click **Cut-Extrude1** and **Cut-Extrude2** from the fly-out FeatureManager. Cut-Extrude1 and Cut-Extrude2 are displayed in the Features to Pattern box.

366) Check the **Geometry pattern** box.

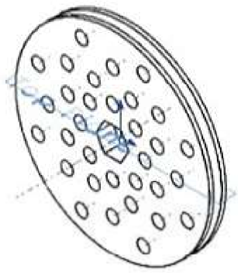


367) Click **OK** from the Circular Pattern PropertyManager. CirPattern1 is displayed in the FeatureManager.

Save the WHEEL part.

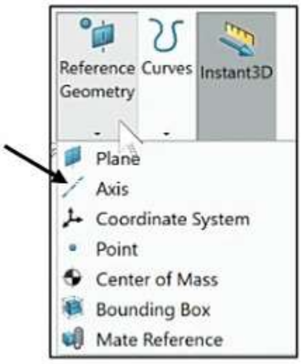
368) Click **Save** .

Utilize a Reference Axis to locate the WHEEL in the PNEUMATIC-TEST-MODULE assembly. The Reference Axis is located in the FeatureManager and Graphics window. The Reference Axis is a construction axis defined between two planes.



Insert a two Plane Reference axis.

369) Click the **Axis** tool from the Reference Geometry Consolidated Features toolbar. The Axis PropertyManager is displayed.

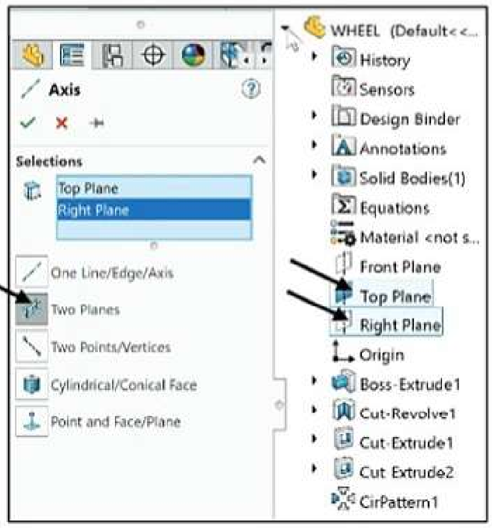


370) Click **Top Plane** from the fly-out FeatureManager.

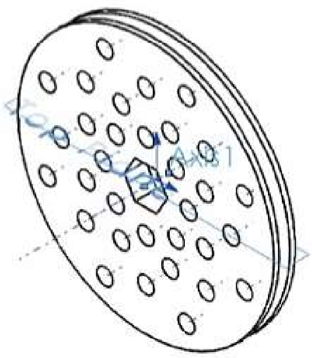
371) Click **Right Plane** from the fly-out FeatureManager. The selected planes are displayed in the Selections box.

372) Click **Two Planes**.

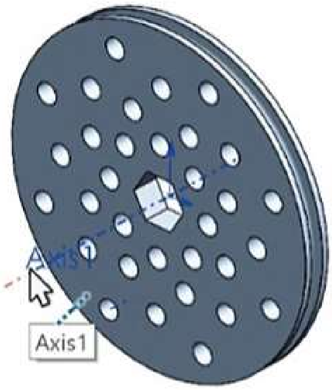
373) Click **OK** from the Axis PropertyManager. Axis1 is displayed in the FeatureManager.



Axis1 is positioned through the Hex Cut centered at the Origin .



374) Click and drag the **Axis1 handles** outward to extend the length on both sides as illustrated.



Display an Isometric view - Shaded With Edges. Clear Temporary Axes. Hide all Planes.

375) Click **Isometric view**.

376) Click **View**, **Hide/Show**, un-check **Temporary Axes** from the Menu bar. **Hide** all Planes.

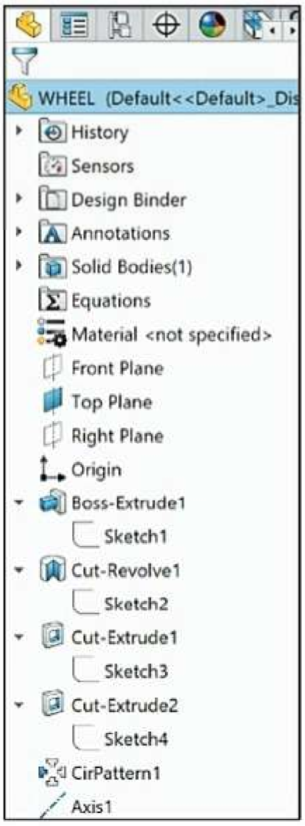
377) Click **Shaded With Edges** from the Heads-up View toolbar.

Save the WHEEL part.

378) Click **Save**.

Sketched lines, arcs or circles are modified from profile geometry to construction geometry. Select the geometry in the sketch. Check the For construction box option.

Review the WHEEL Part



The WHEEL part was created with the Extruded Boss/Base feature. You sketched a circular sketch on the Front Plane and extruded the sketch with the Mid Plane option.

A Revolved Cut feature removed material from the WHEEL and created the groove. The Revolved Cut feature utilized an arc sketched on the Right Plane. A sketched centerline was required to create the Revolved Cut feature.

The WHEEL contained a complex pattern of holes. The first Extruded Cut feature contained two small circles sketched on two bolt circles. The bolt circles utilized construction geometry. Geometric relationships and dimensions were used in the sketch. The second Extruded Cut feature utilized two small circles sketched on two bolt circles.

The two Extruded Cut features were contained in one Circular Pattern and revolved about the Temporary Axis. The Reference Axis was created with two perpendicular planes. Utilize the Reference Axis, Axis1 in the WHEEL-AXLE assembly.