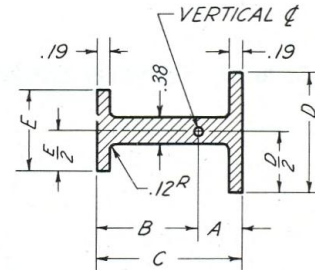


SECTION NUMBER	A	B	C	D	E
1	1.00	2.00	3.00	1.88	1.62
2	.94	1.88	2.82	1.75	1.38
3	.62	1.82	2.44	1.62	1.12
4	0.00	1.88	1.88	1.50	.88
5	.68	2.12	1.44	1.38	.62
6	1.00	-	-	1.25	-



ASSIGNMENT

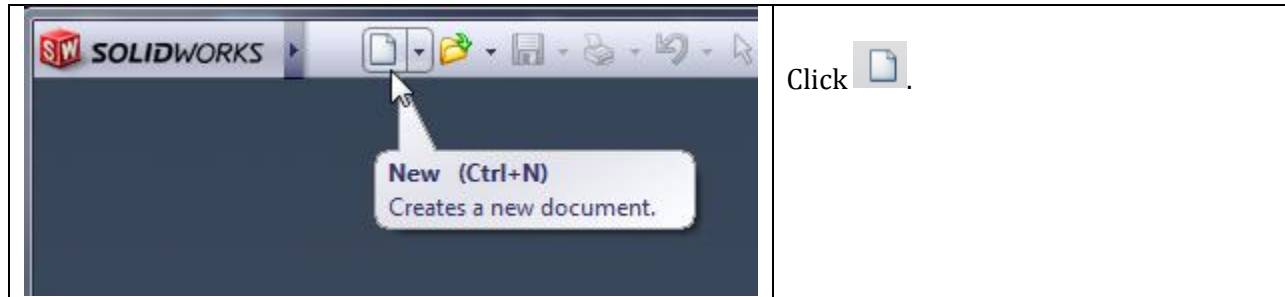
Submit TWO isometric views of the Shaft Hanger with your report,

1. Shaded view of the **trimetric** front.
2. Hidden lines removed display of the isometric front.

Be sure to turn-off ALL datum features such as reference planes and axes.

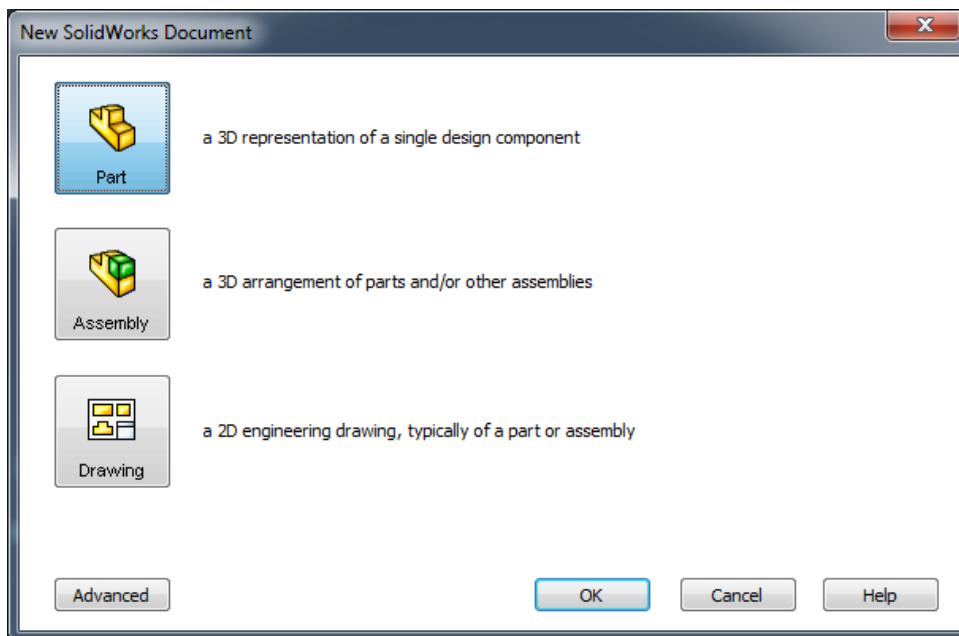
CREATE A NEW PART: SHAFT HANGER

Start SolidWorks and

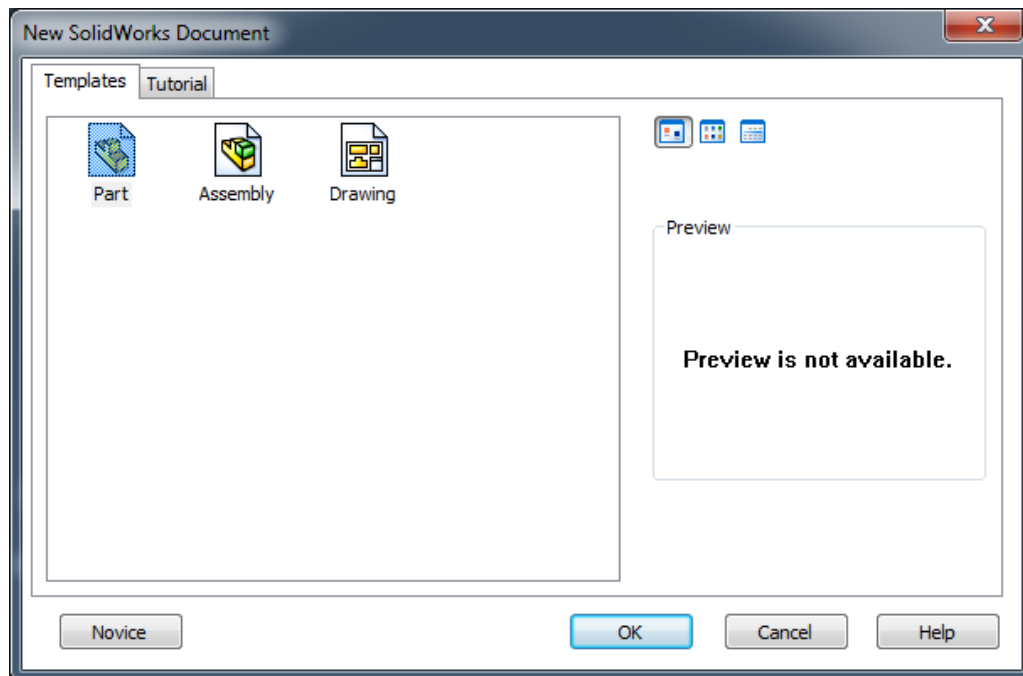


SolidWorks has two modes in its dialog box:

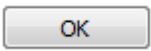
1. **Novice** mode – this is default mode with three default templates – **Part**, **Assembly** and **Drawing**.
2. **Advanced** mode provides access to additional templates and tabs created in System options.



Select **Advanced** mode,

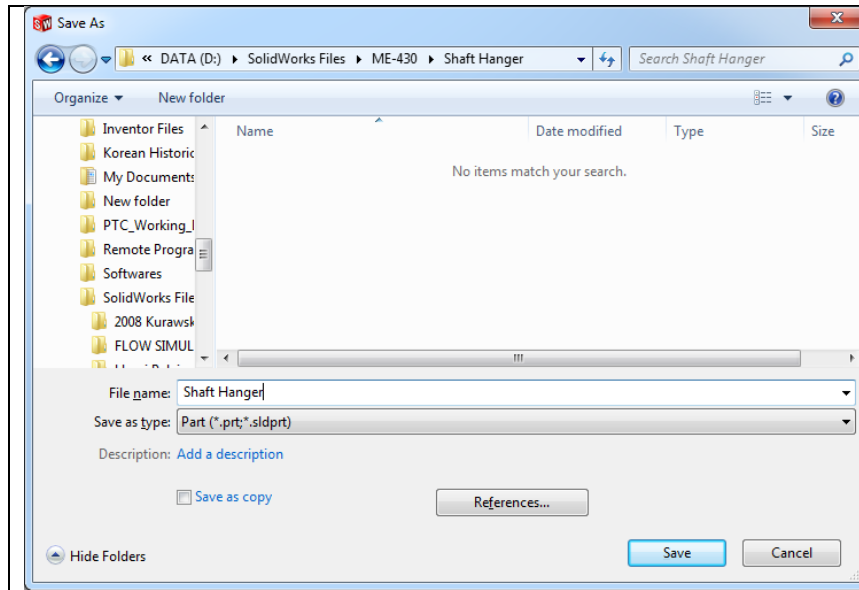


Note that **Part** is the default template in the dialog box.

Click  .



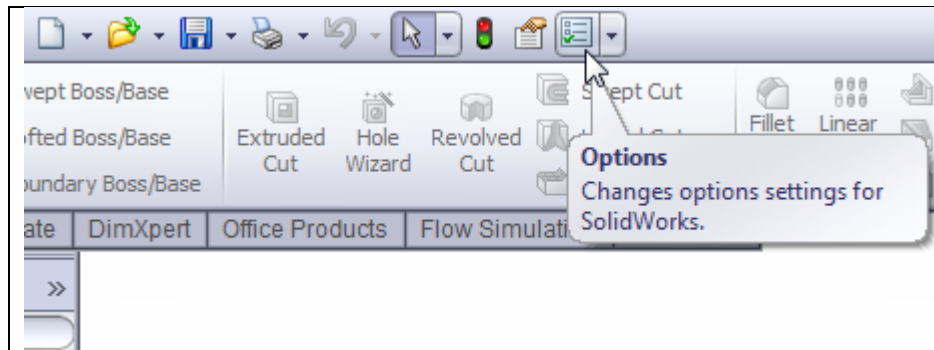
Create a folder for your class and part such as ...:\ME-430\Shaft Hanger\



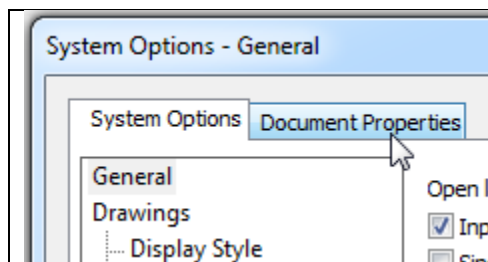
Enter **Shaft Hanger** for the name of the part.

Click **Save**.

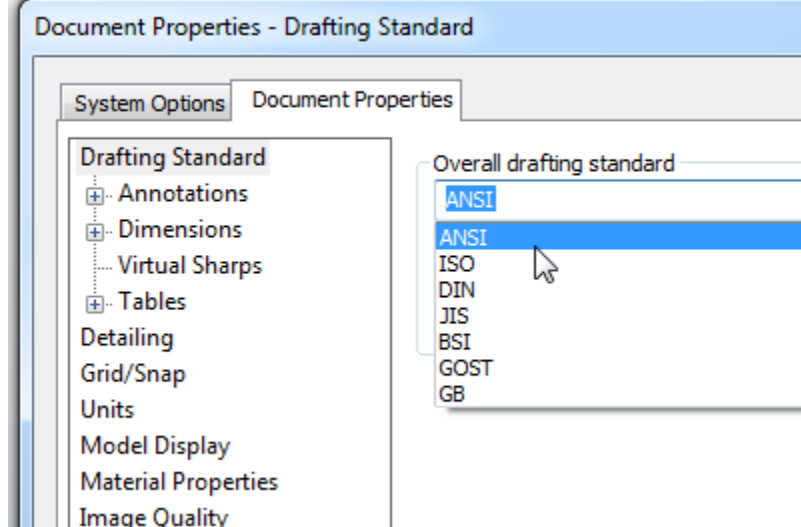
SET OVERALL DRAFTING STANDARD, UNITS SYSTEM AND PRECISION

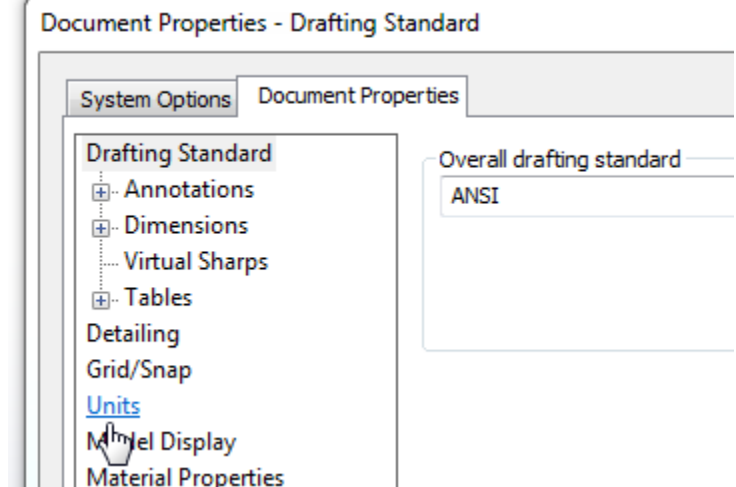


Click **Options**.

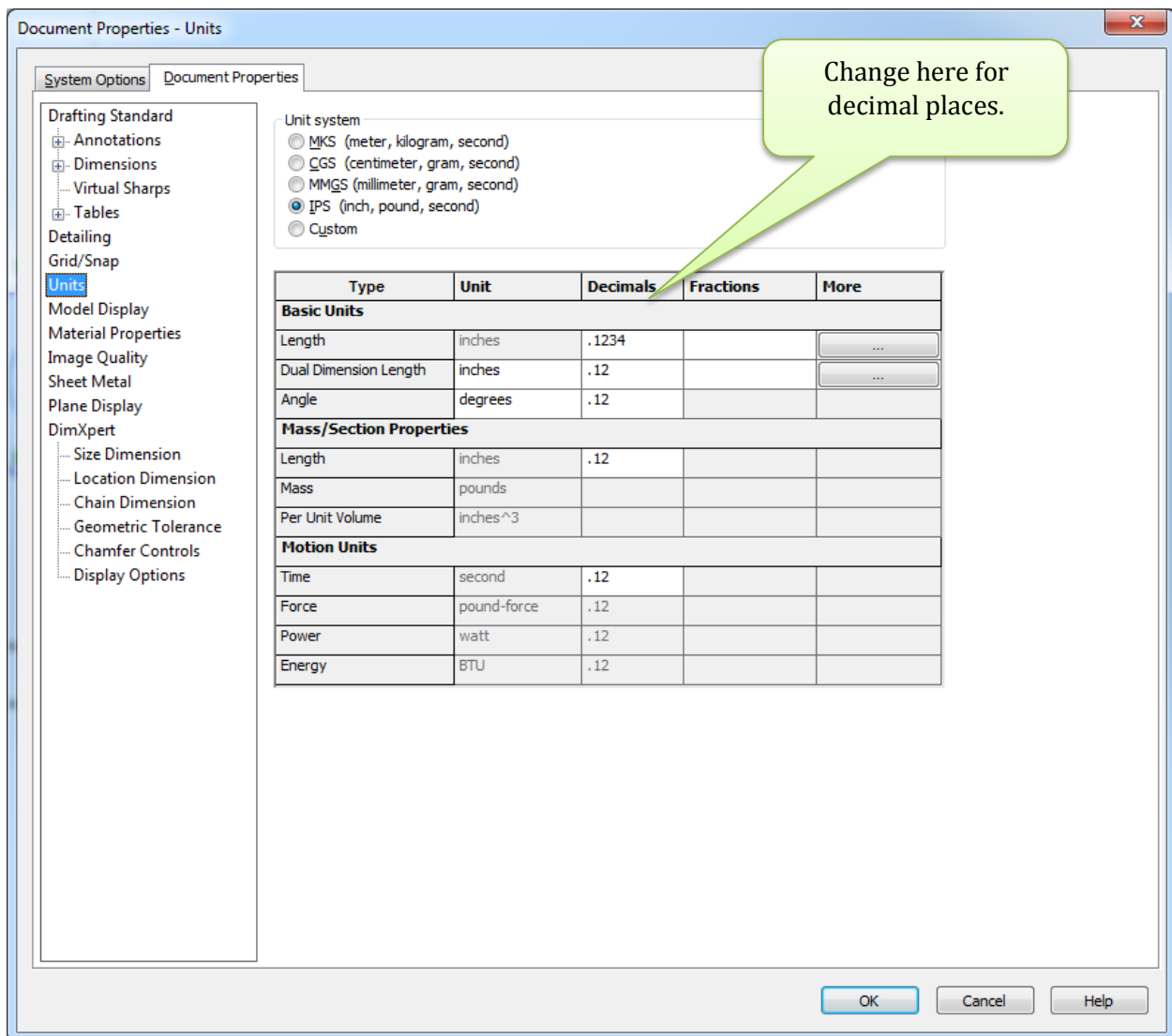


Select **Document Properties** tab.

	<p>Select ANSI for Overall drafting standard.</p> <p>Note that ANSI is an US drafting standard and uses Third Angle Projection.</p>
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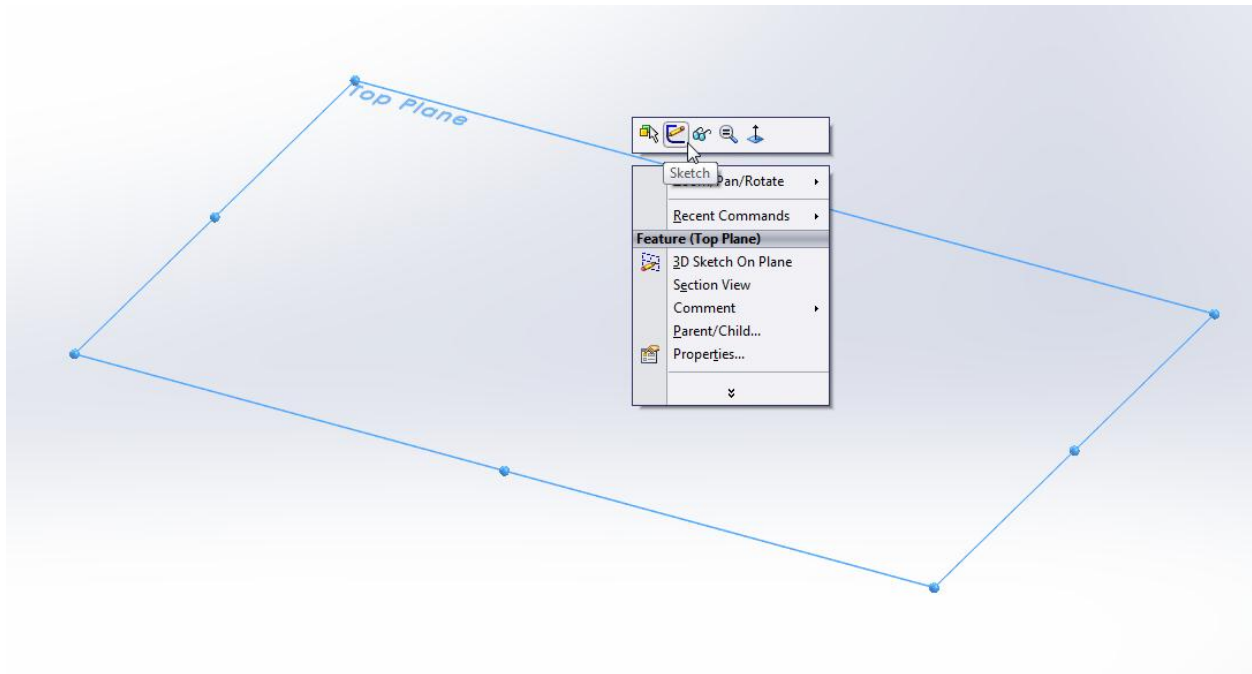
	<p>Click Units.</p>
--	----------------------------


Click **IPS** for the **Units** and select **.1234** decimal places for **Length** of **Basic Units**.




Click **OK**.

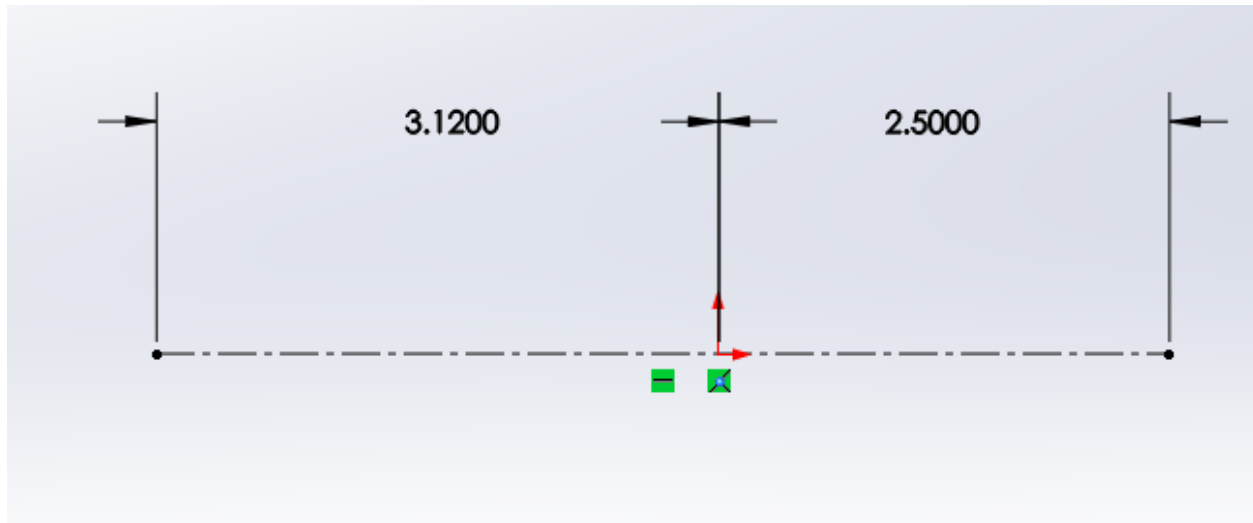
Pick the **Top Plane** and select **Sketch**.




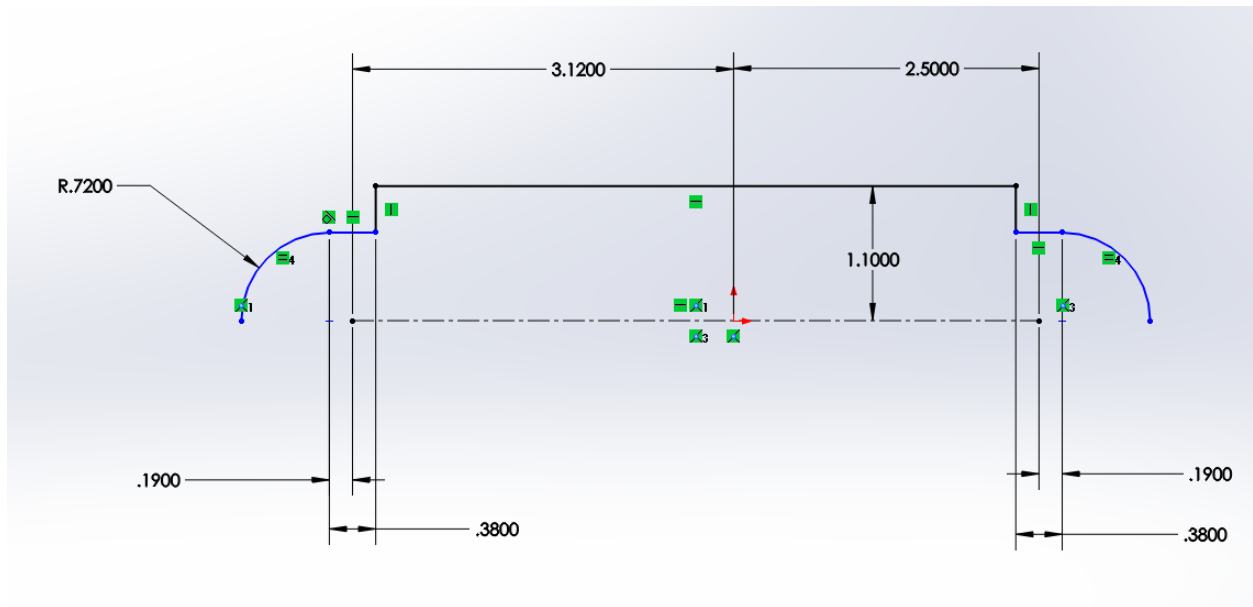
	<p>Select Top view for sketching the section.</p> <p>Click  .</p>
--	---

	<p>Select Centerline  Centerline and draw a centerline. The centerline passes through the origin.</p>
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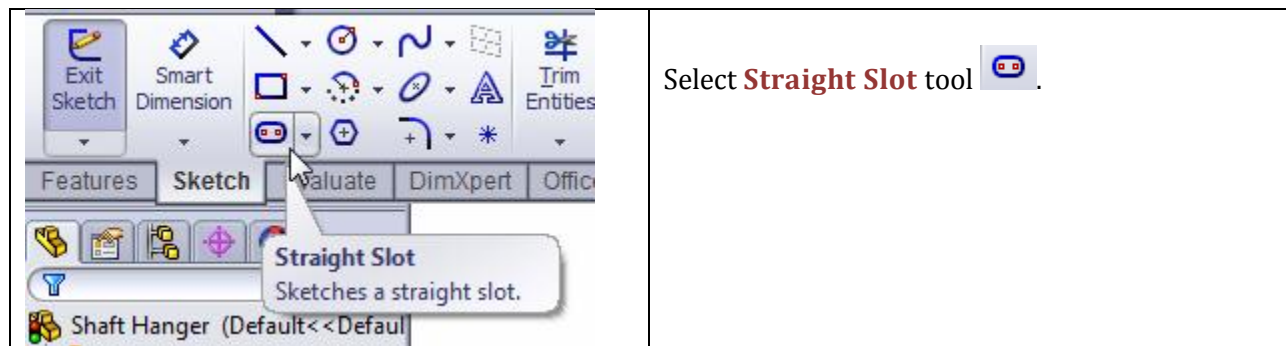
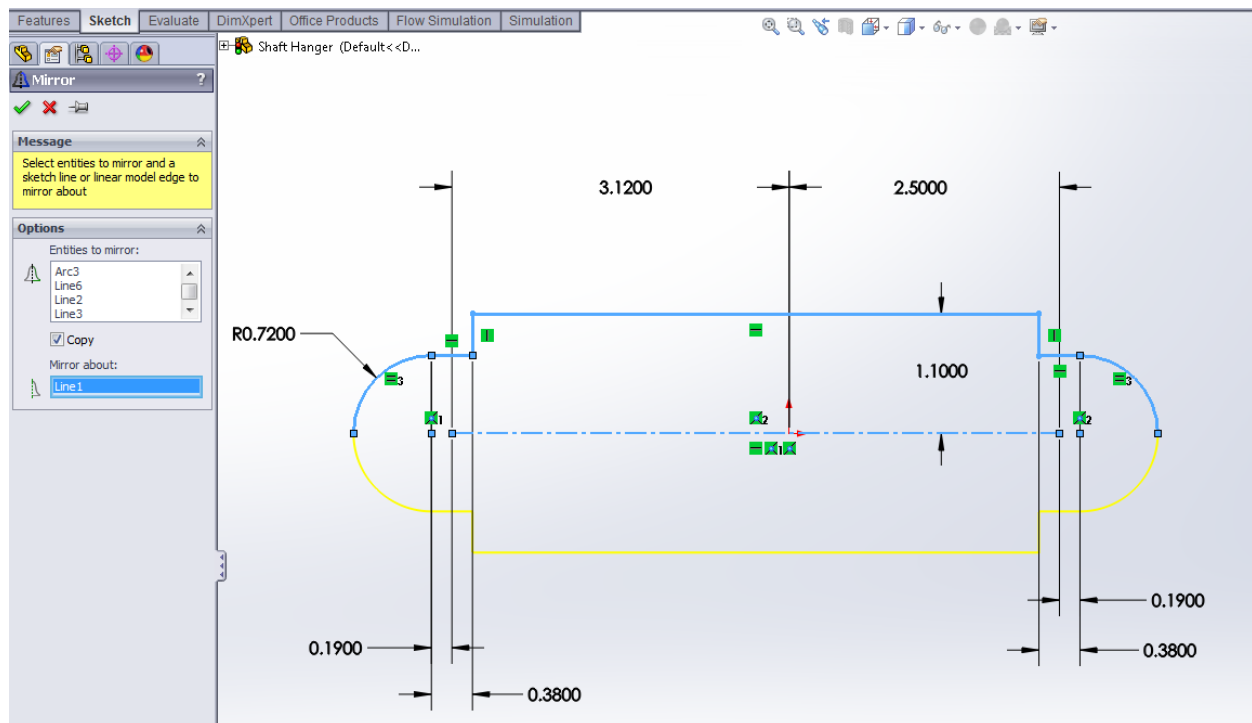
Dimension the center line as shown below.



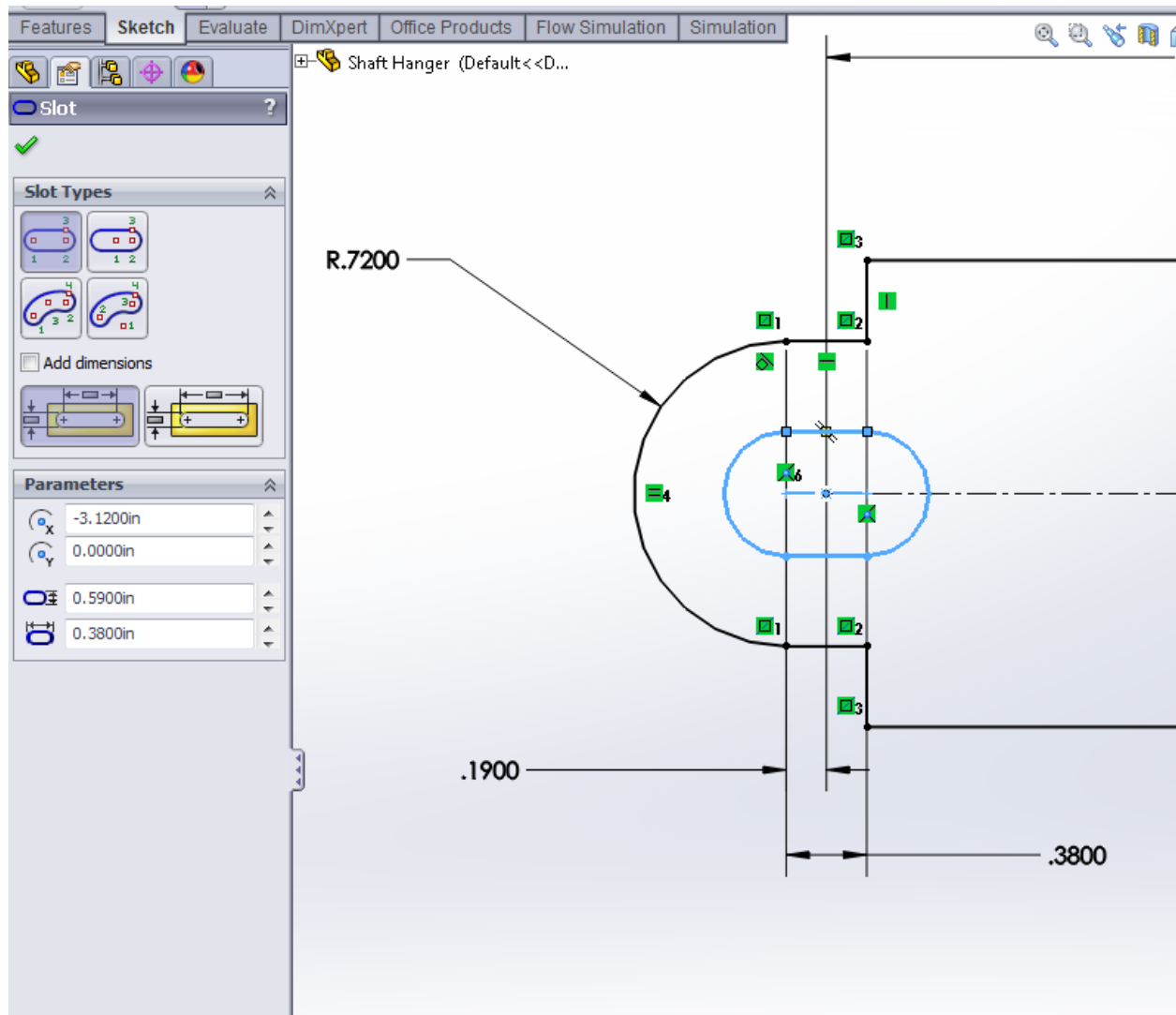
Use **Line** tool  and **Smart Dimension** to sketch and dimension the following section. Be sure to constrain your sketch as shown below.



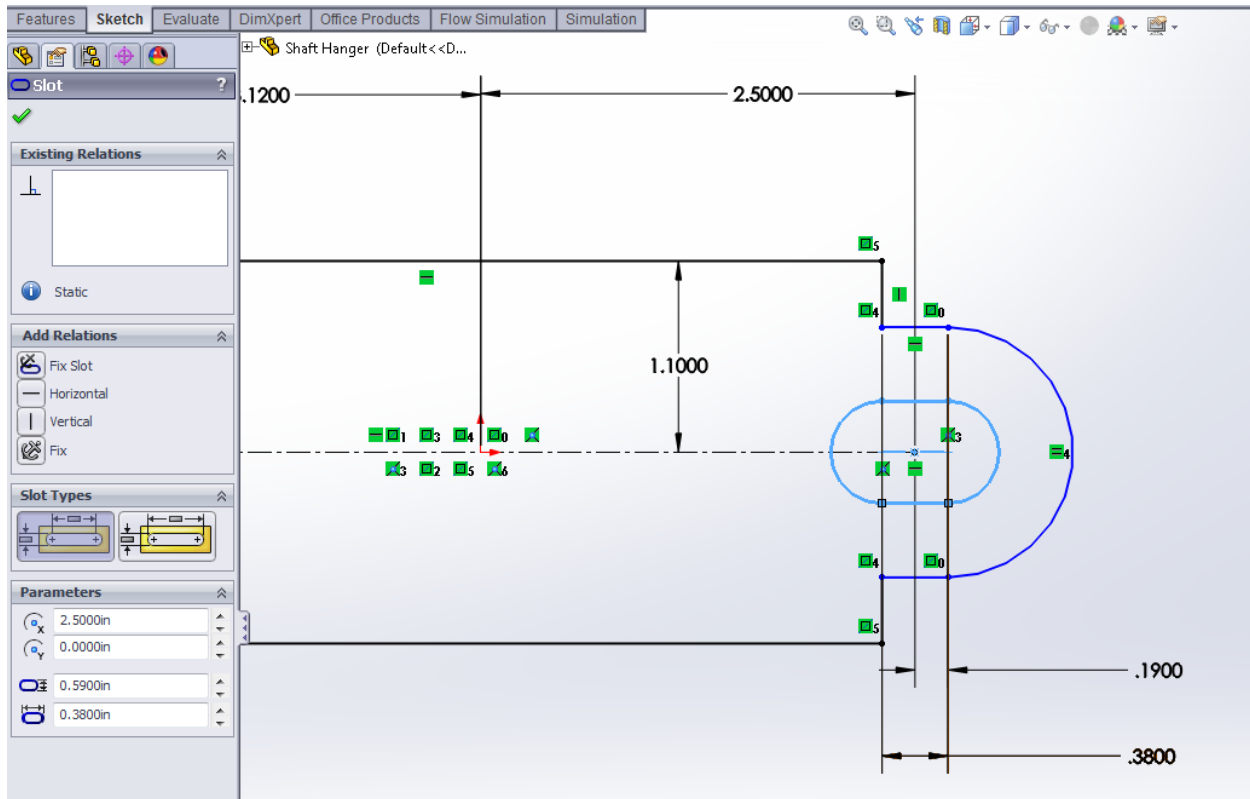
Mirror the sketch as shown below.



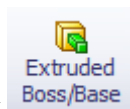
Enter the dimension of the slot as show in the **Slot** Properties dialog box.



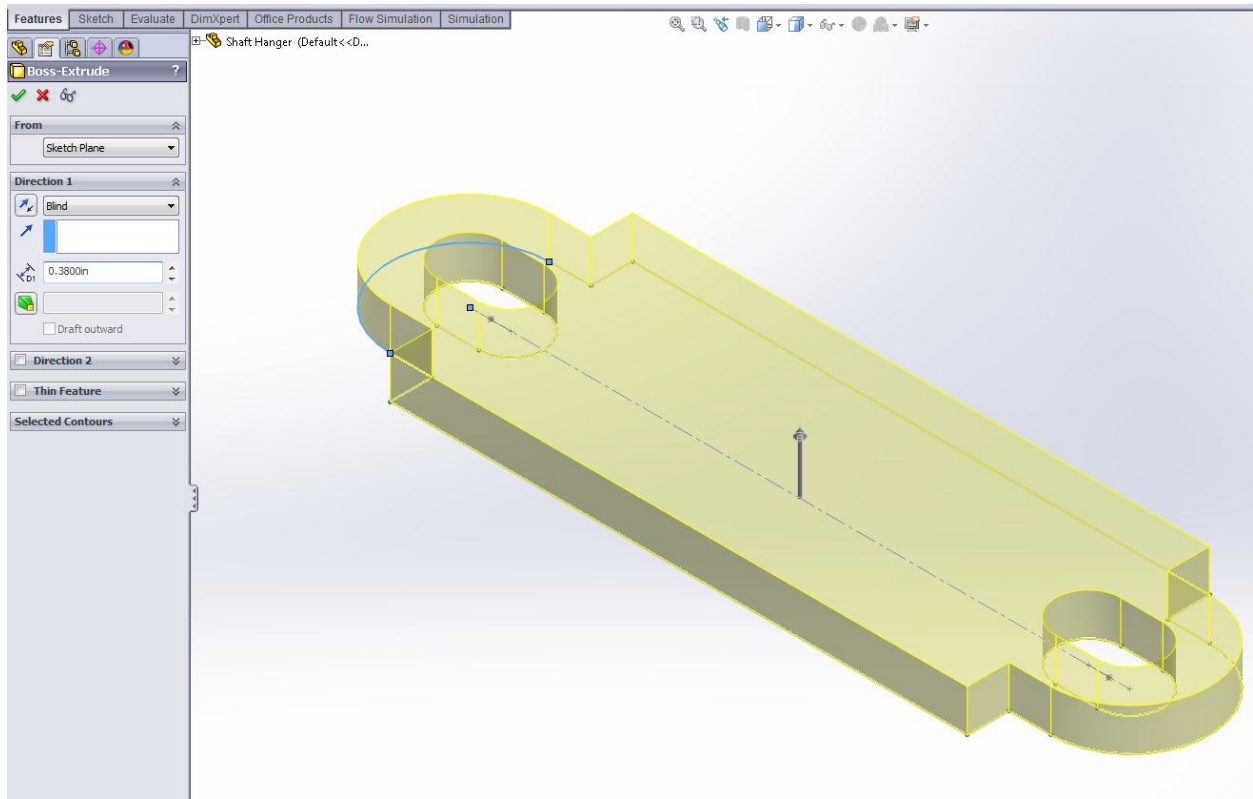
Create another slot on the right end of the section as shown below.



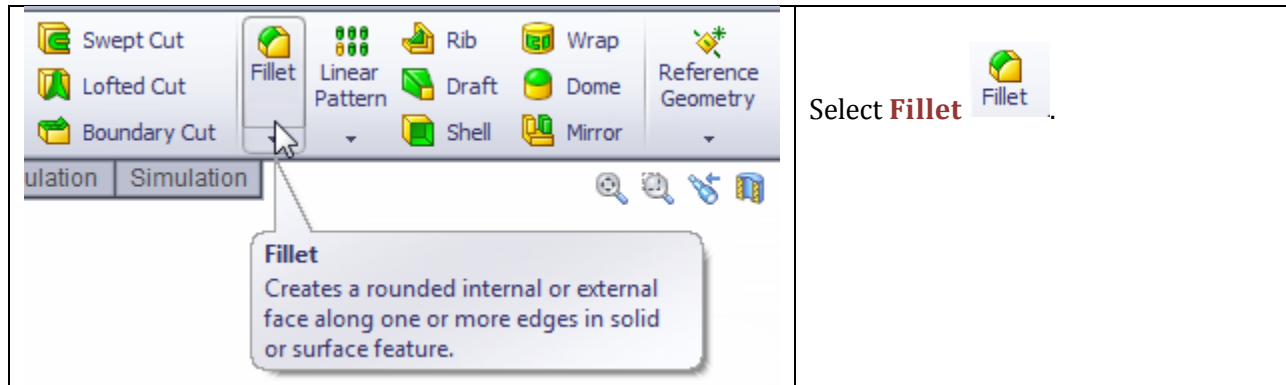
Click **Exit Sketch** to exit the sketch mode.



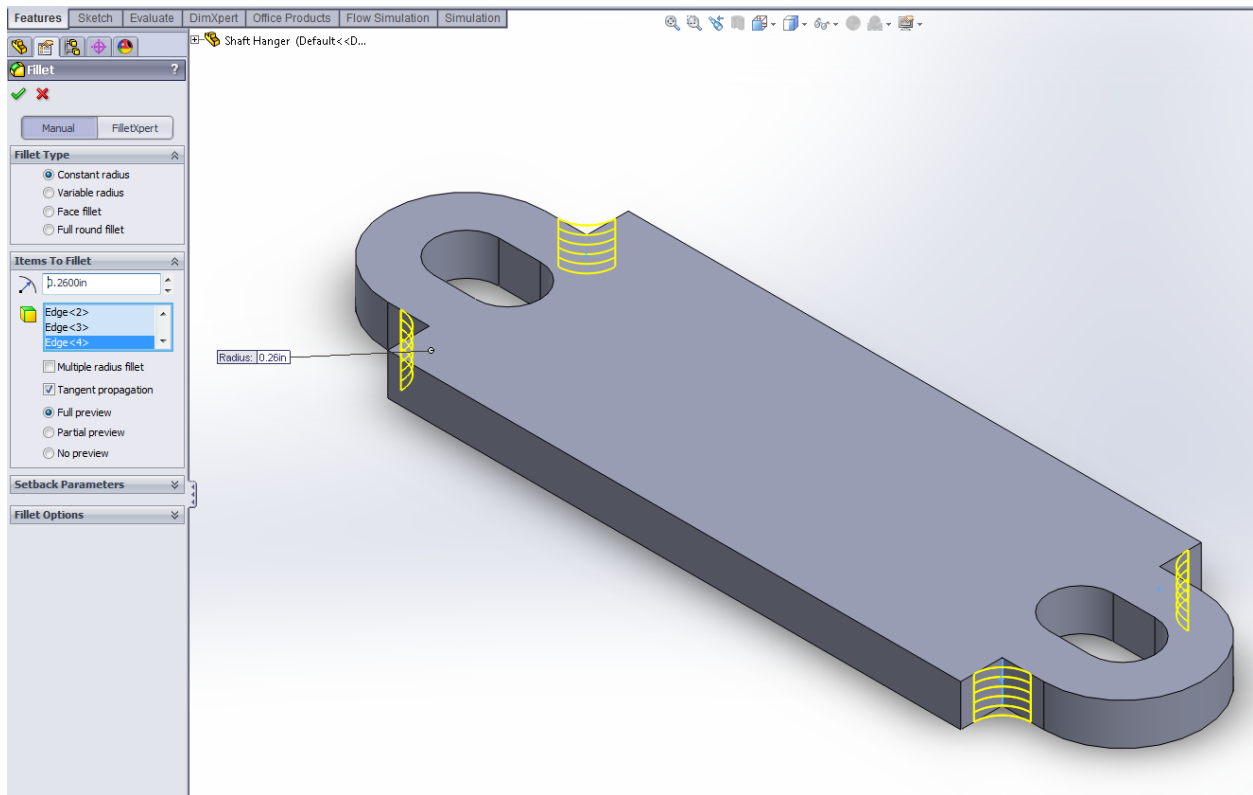
Select **Features** tab and click **Extruded Boss/Base**. Pick the section and enter **0.38** inch for the thickness.



Click .

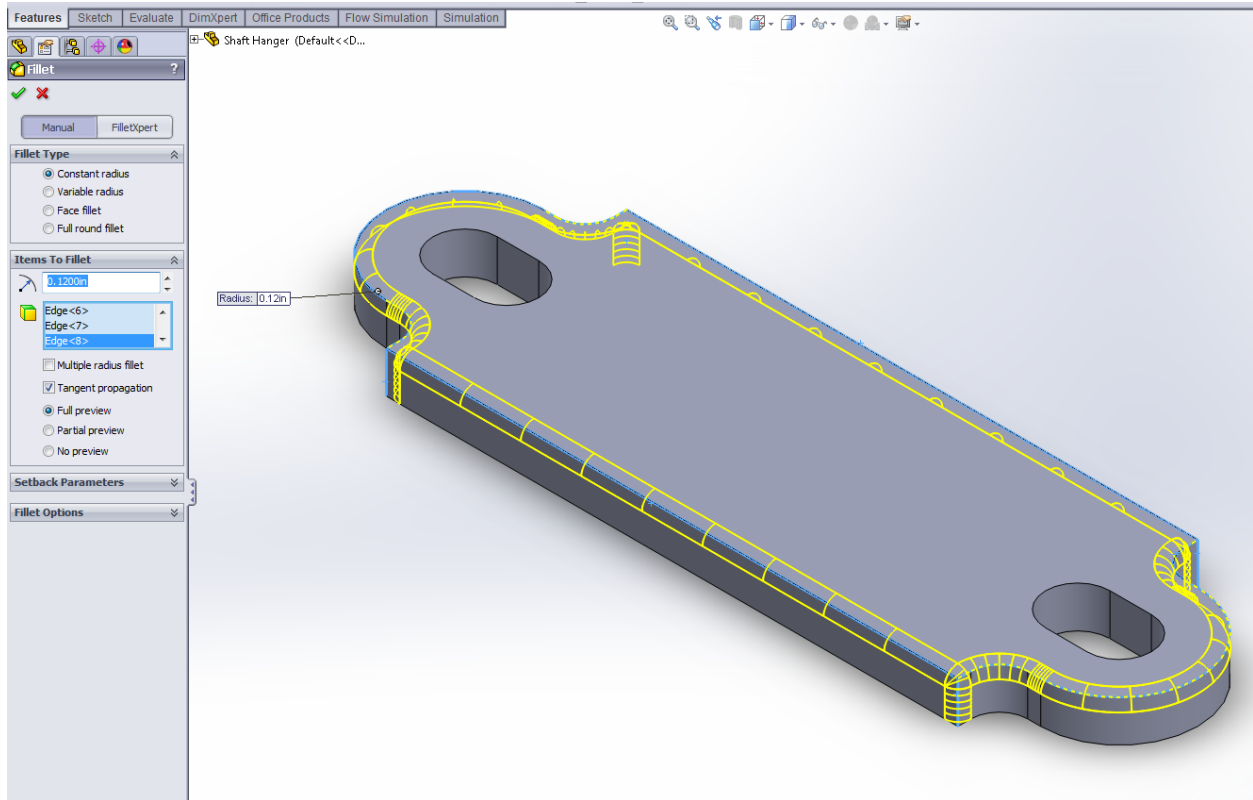


Create **0.26** inch radius of fillets at the location shown below.



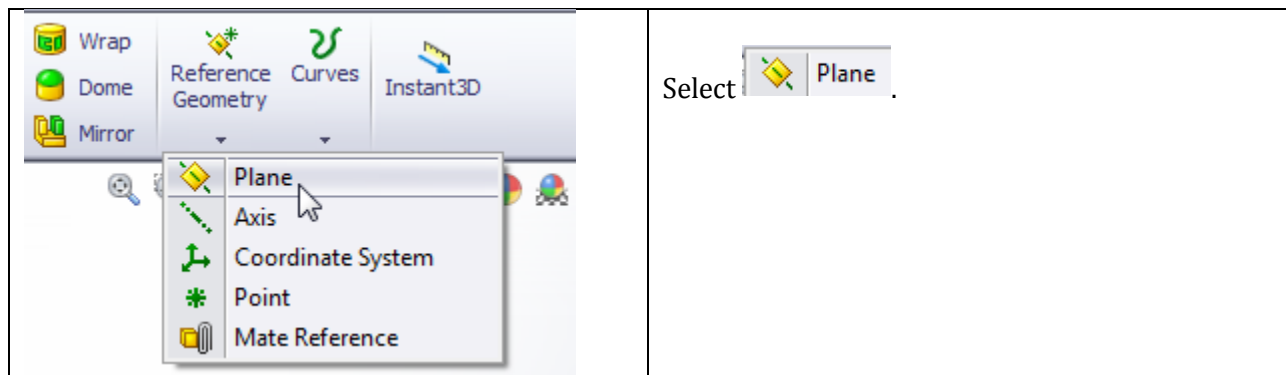
Click  .

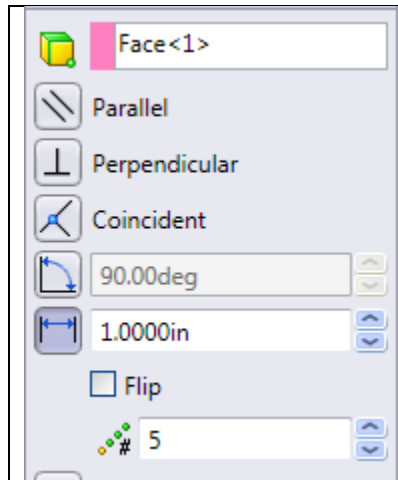
Select **Fillet**  and create **0.12** inch radius as shown below.



Click .

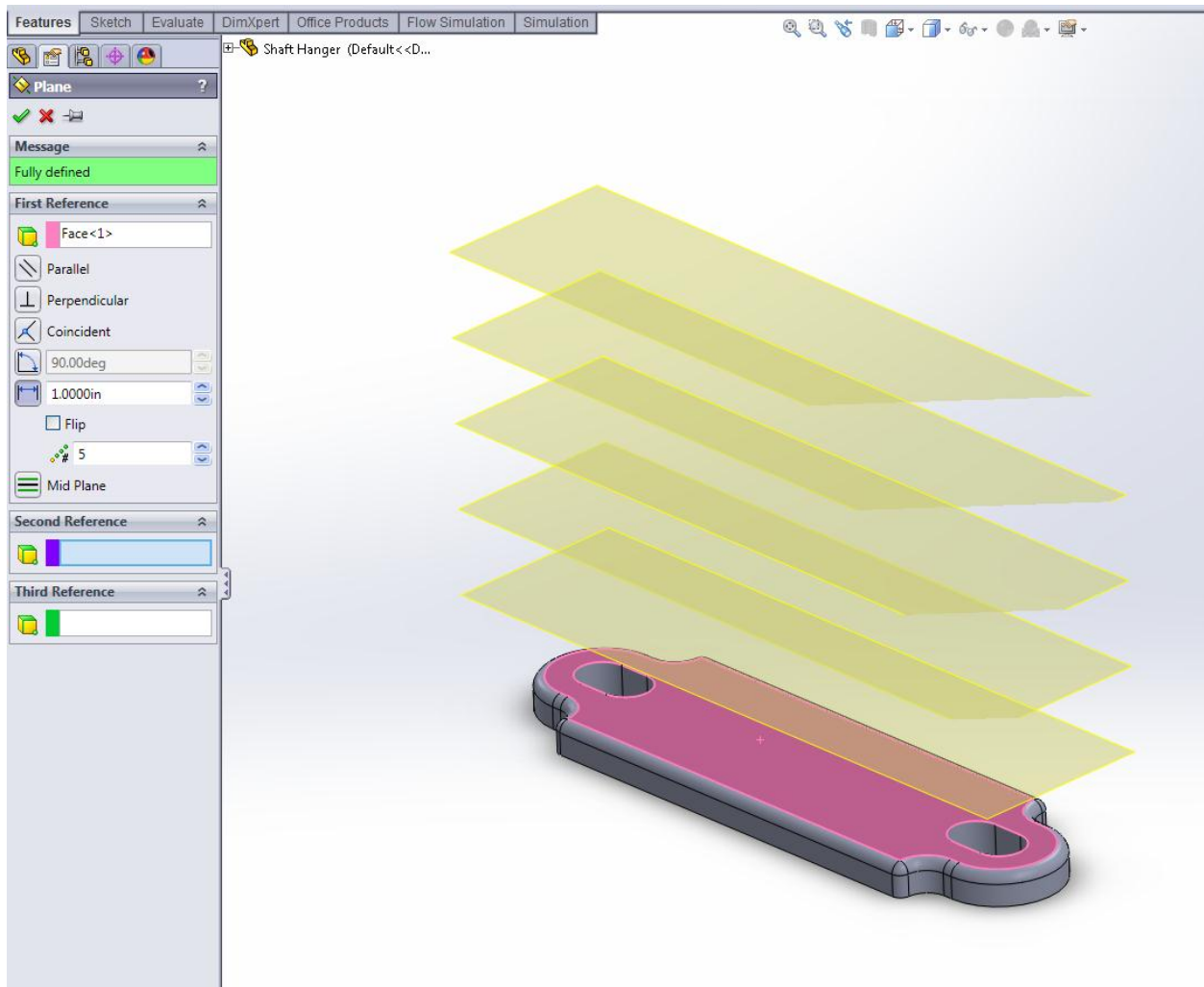
Create **5** reference planes from the top face of the base feature. The offset distance of each plane is **1** inch. The reference planes will be used to sketch **5** sections for **Lofted Boss/Base**.



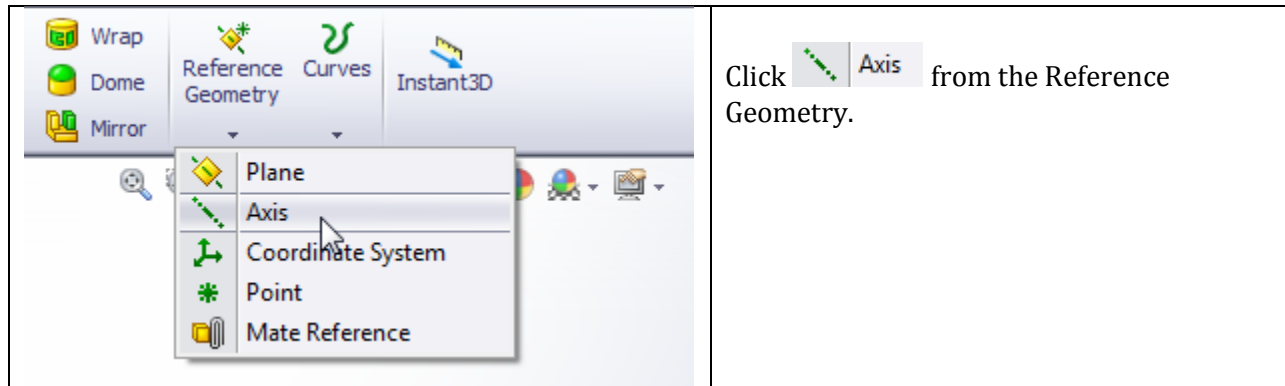


Pick top face of base feature and enter **1** inch for the offset distance.

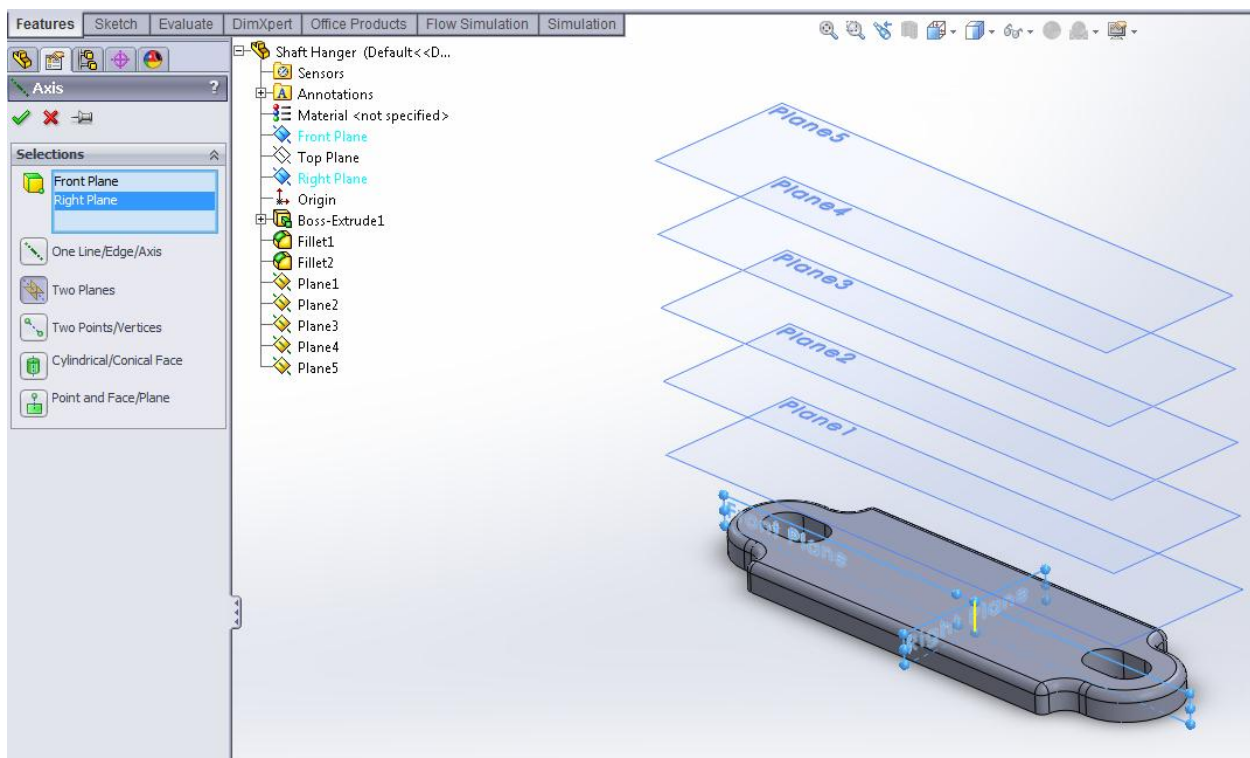
Enter **5** for the **Number of planes to create**.




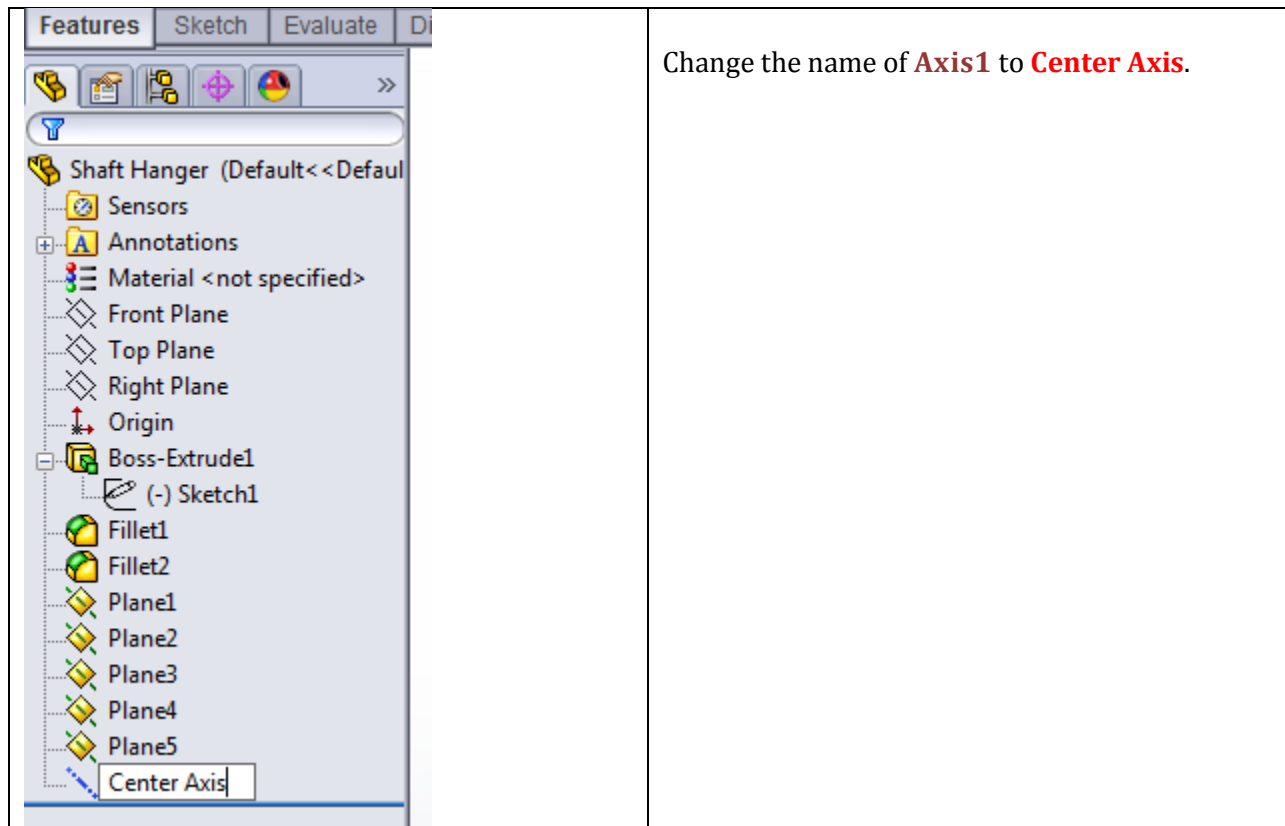
Click .




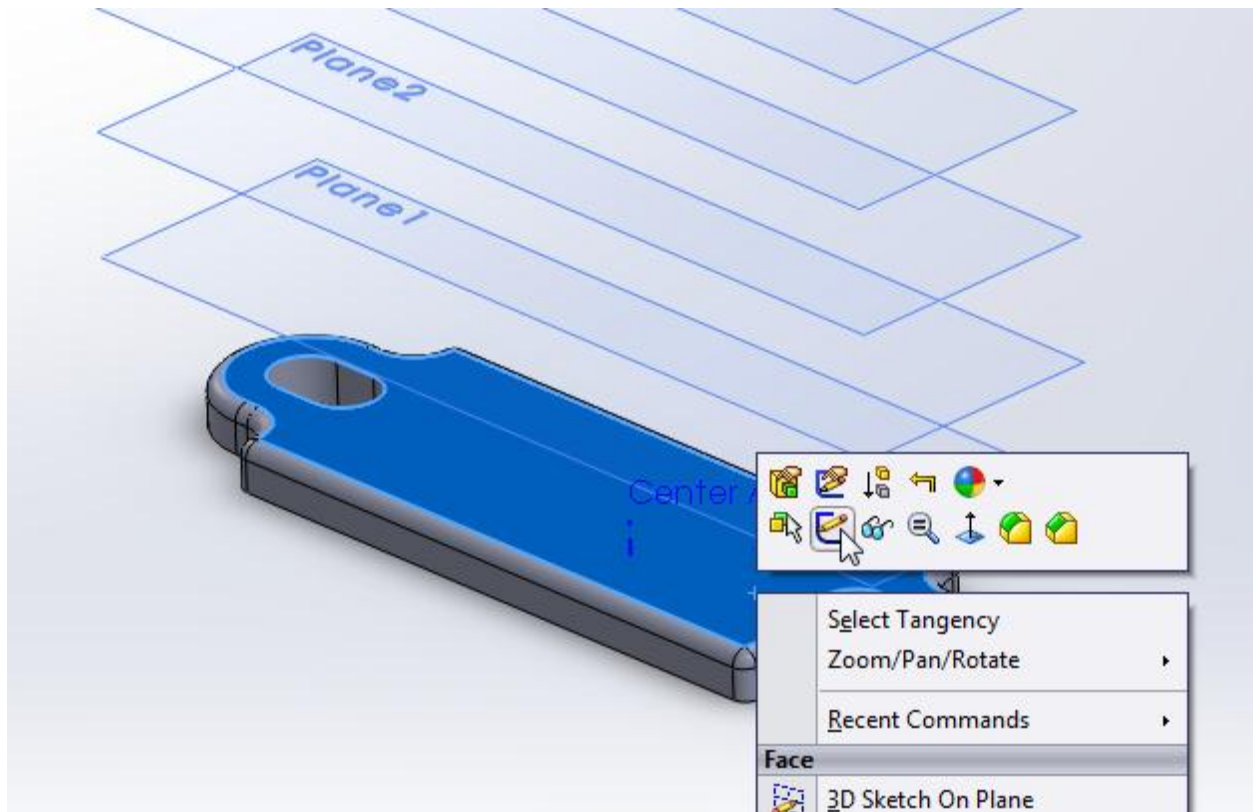
As **Front Plane** and **Right Plane** from features tree as reference for axis creation.



Click . Note that reference axis created above is optional. You can proceed without this axis.

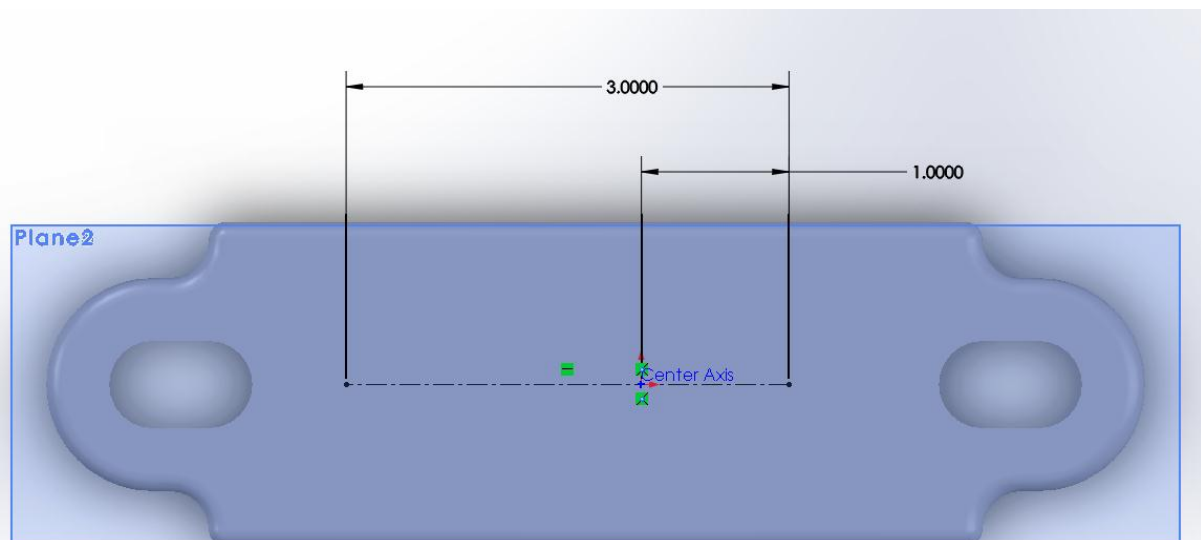


Right-click top face of base feature and select **Sketch** .

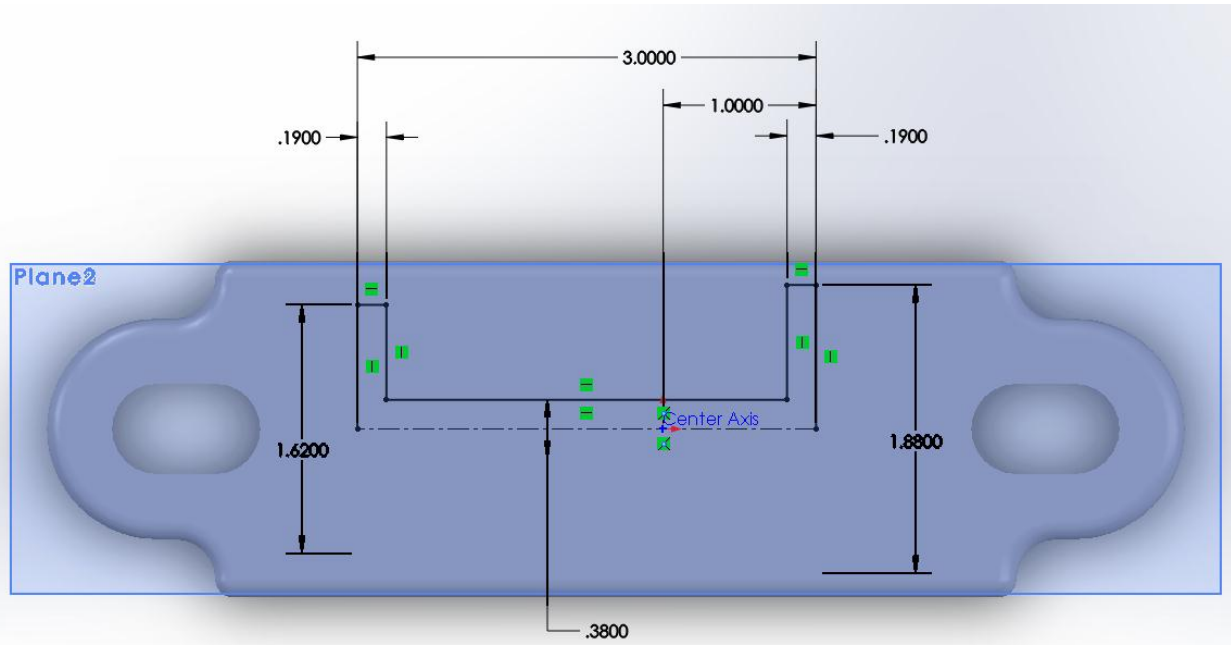


Set the display to **Top** view .


Draw and dimension a horizontal centerline as shown below.

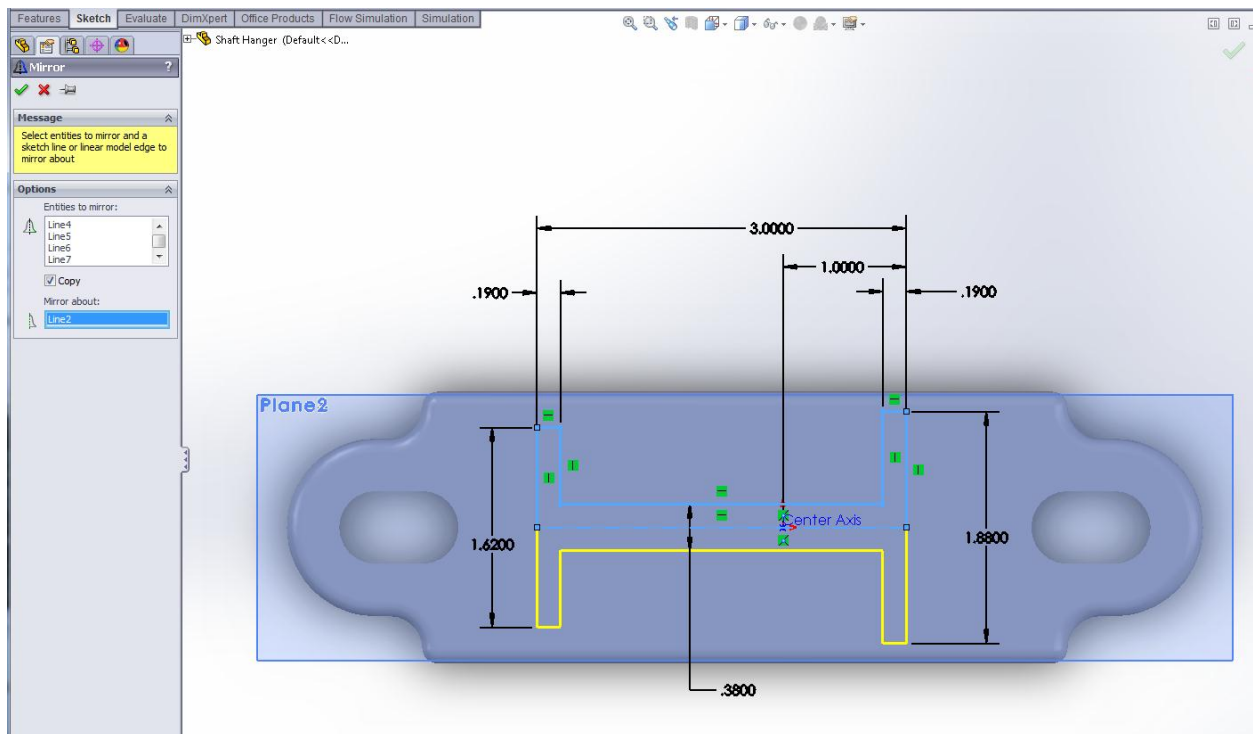


Next sketch the following entities.



Click .

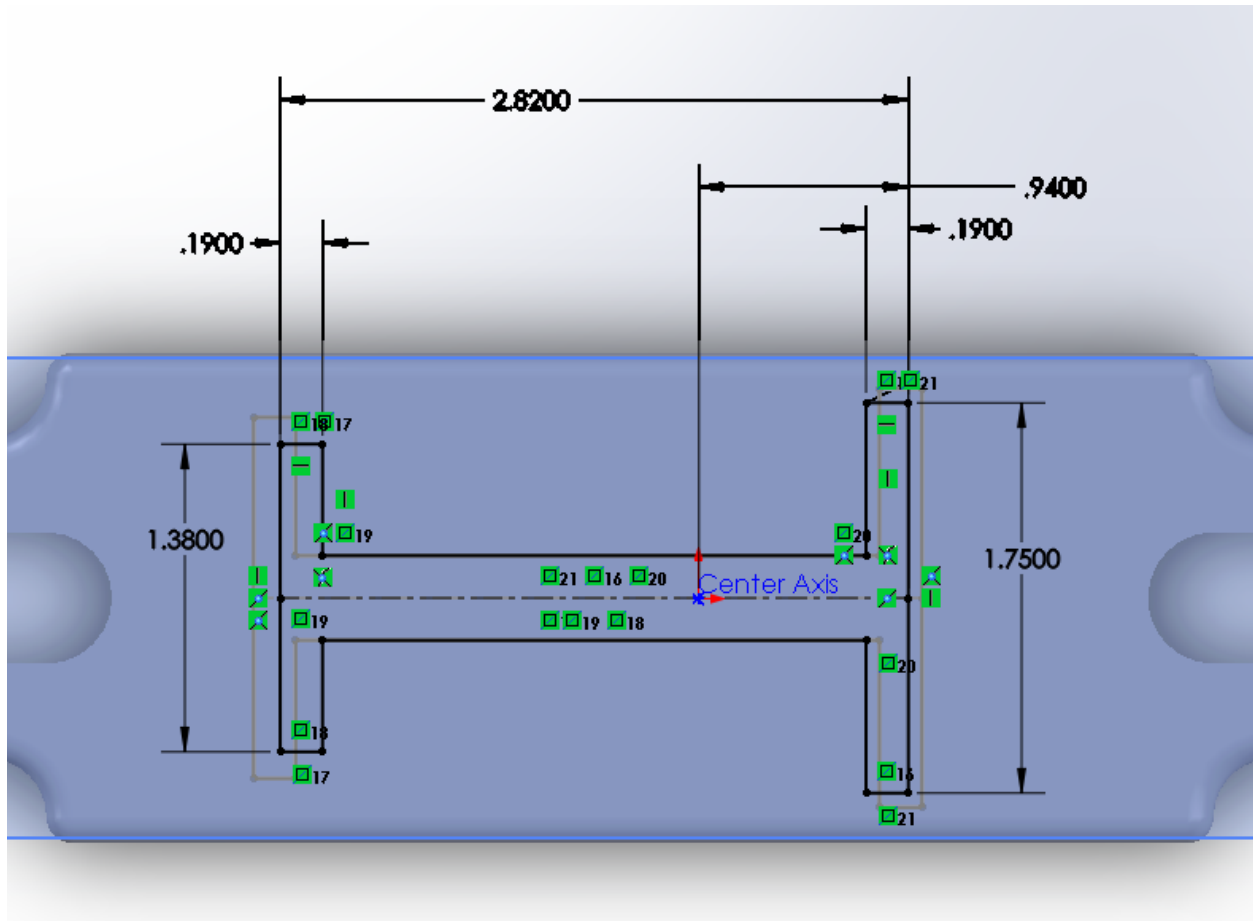
Select  **Mirror Entities** and mirror the entities with respect to the centerline.



Click .

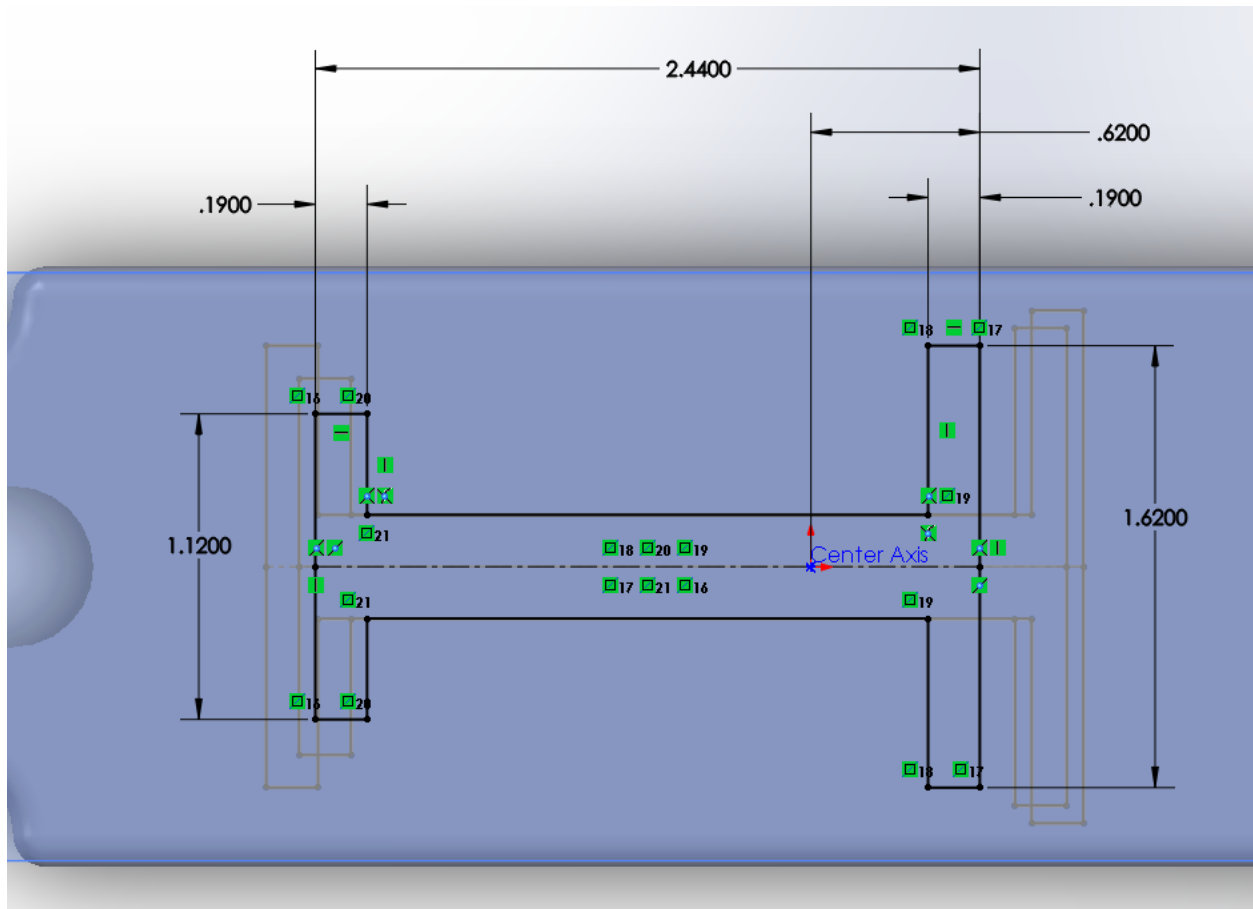
Right-click **Plane1** and select . Set the display to **Top** view .

Sketch the following section. Similar to above, create a half section and then mirror the upper half of the entities.



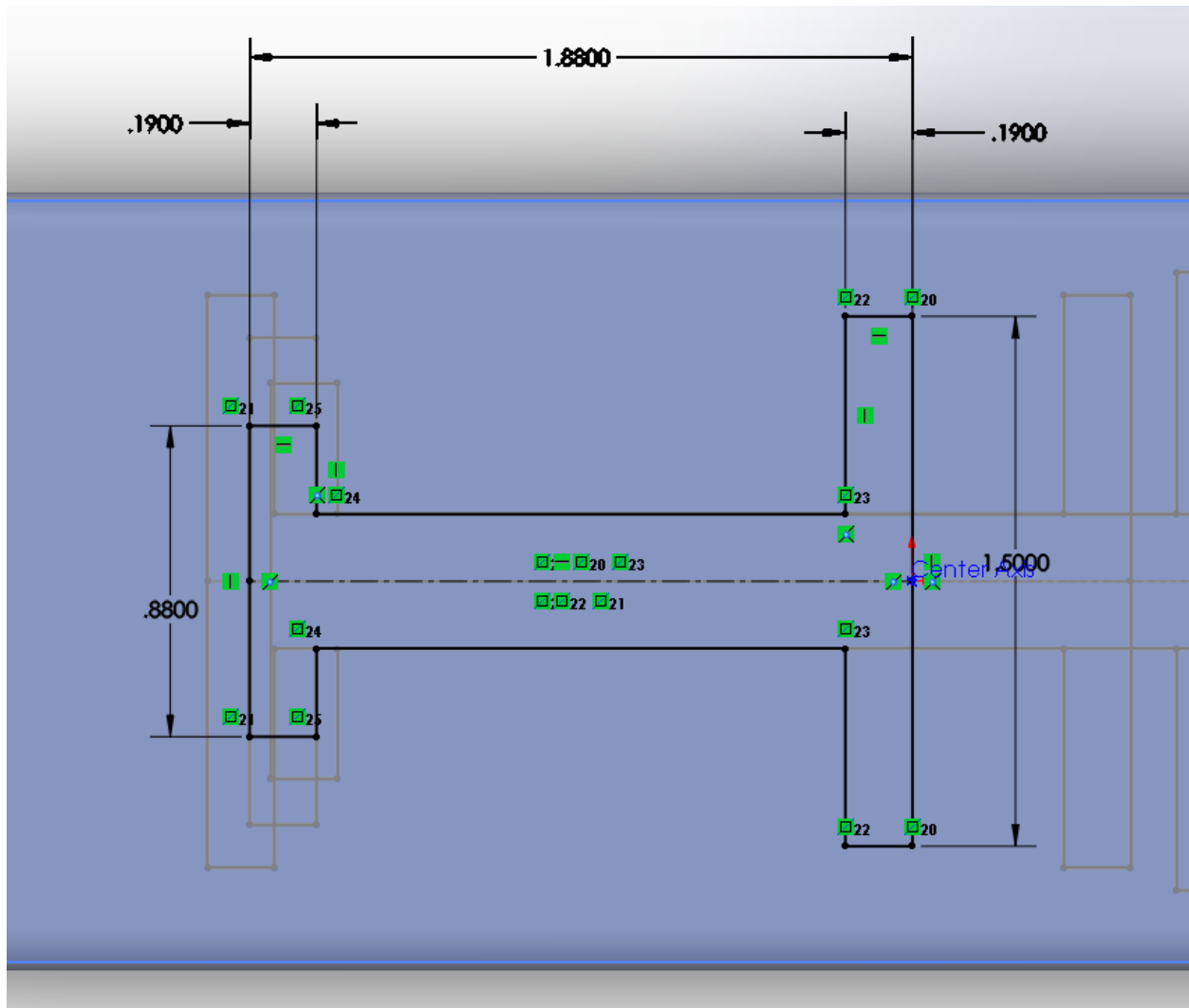
Right-click **Plane2** and select . Set the display to **Top** view .

Sketch the section shown below.



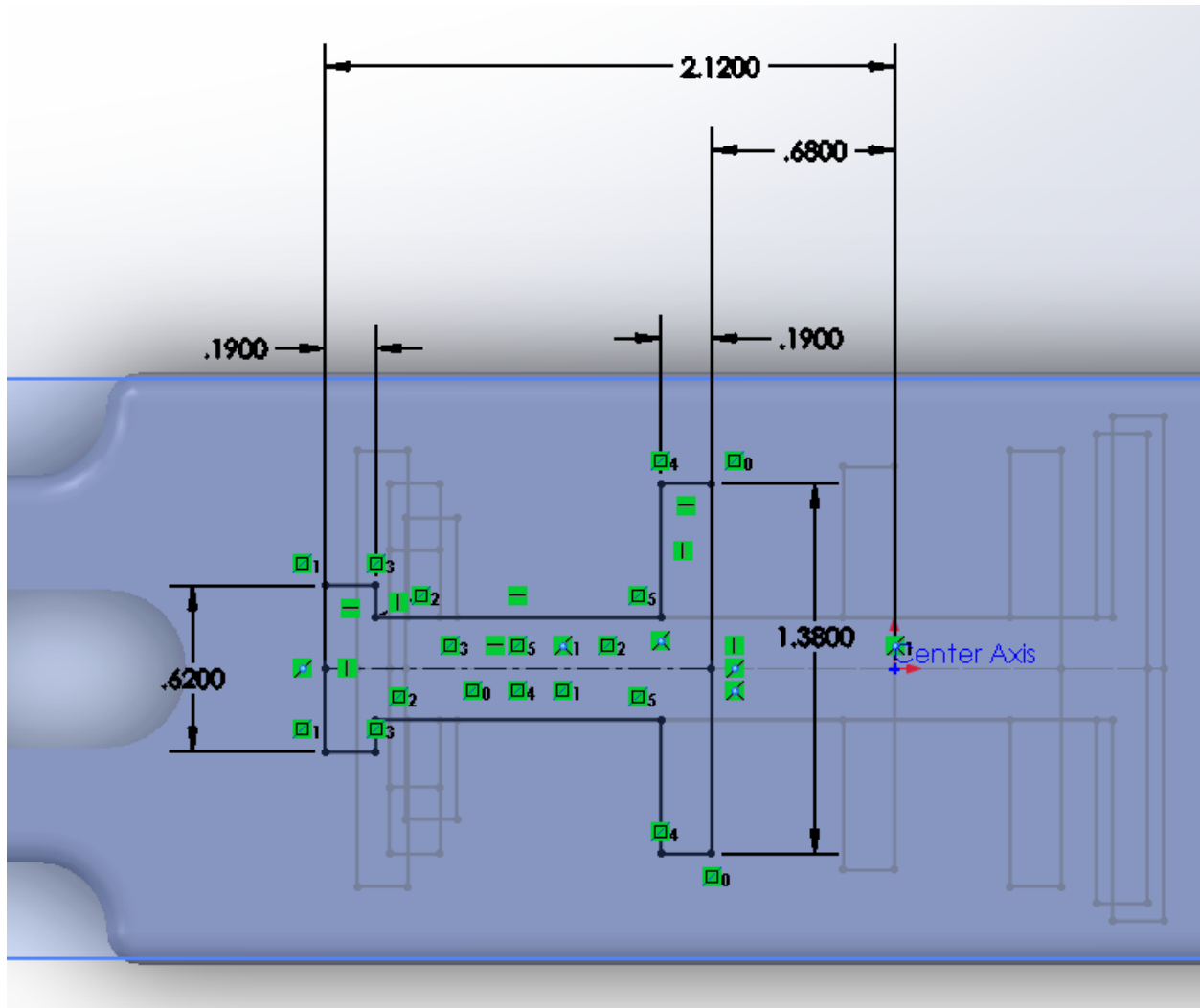
Right-click **Plane3** and select  . Set the display to **Top** view  .

Sketch the fourth section as shown below. Note that the most left vertical line pass through the origin.



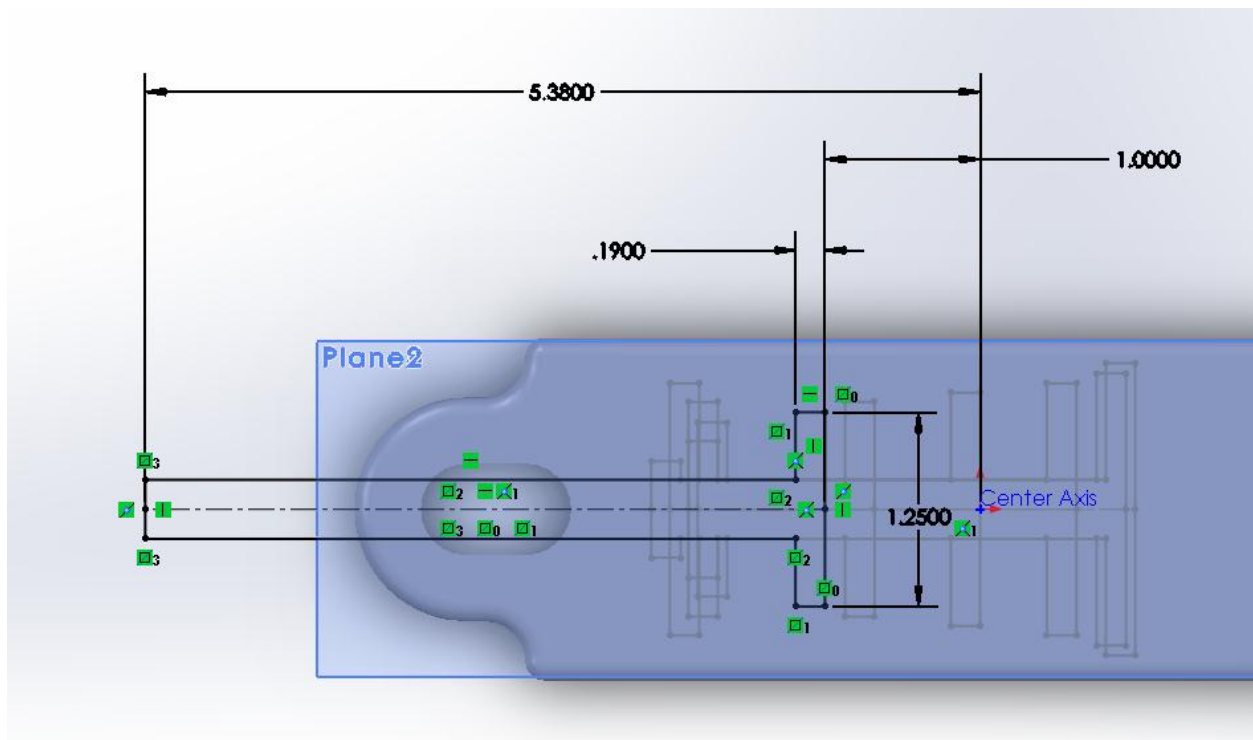
Right-click **Plane4** and select . Set the display to **Top** view .

Sketch the fifth section as shown below.

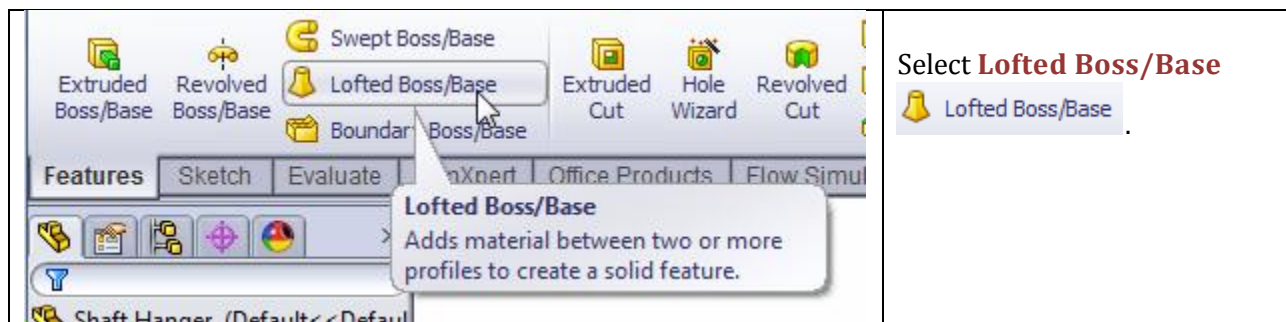
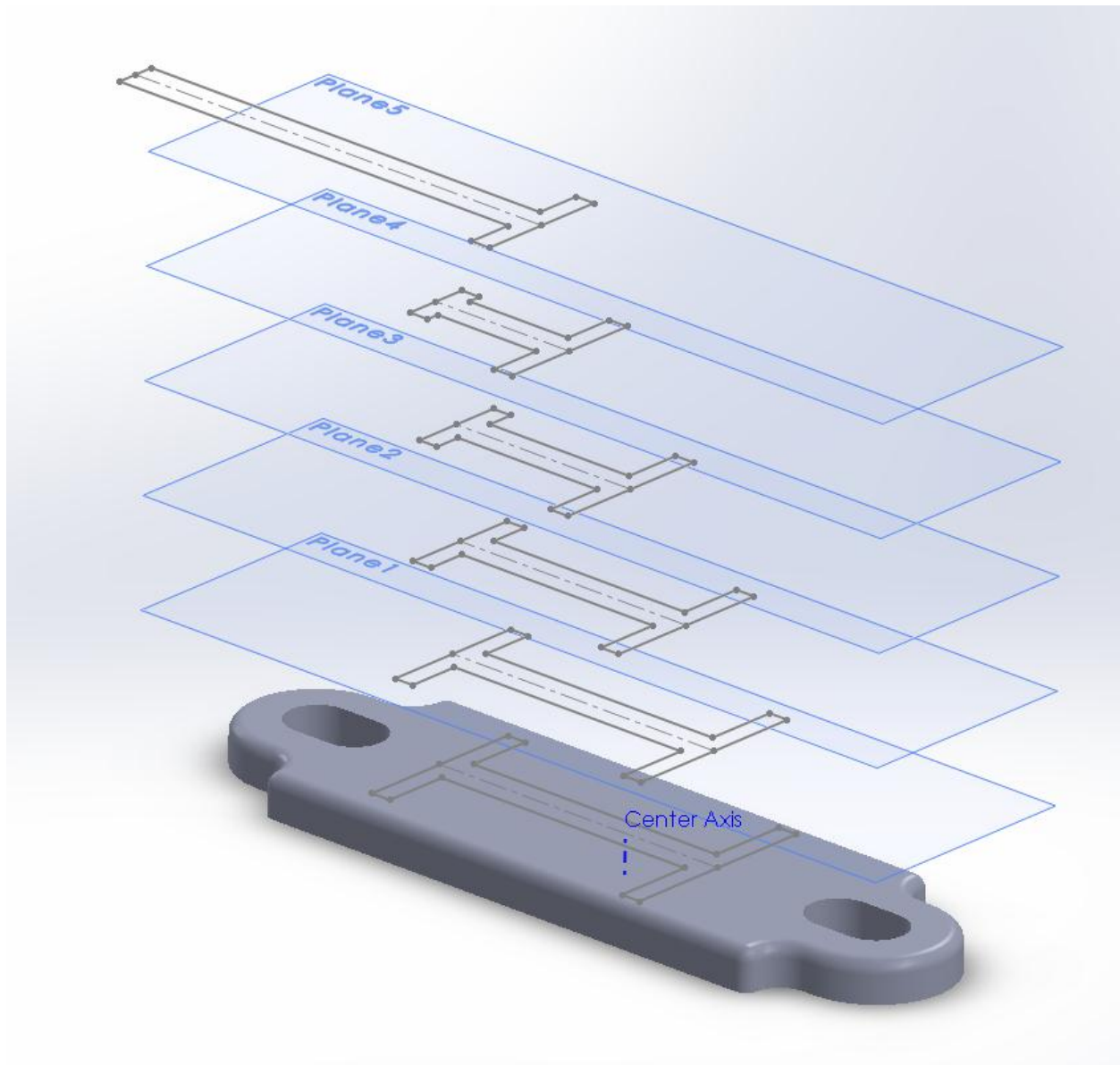


Right-click **Plane5** and select  . Set the display to **Top** view  .

Sketch the sixth section (last) as shown below.

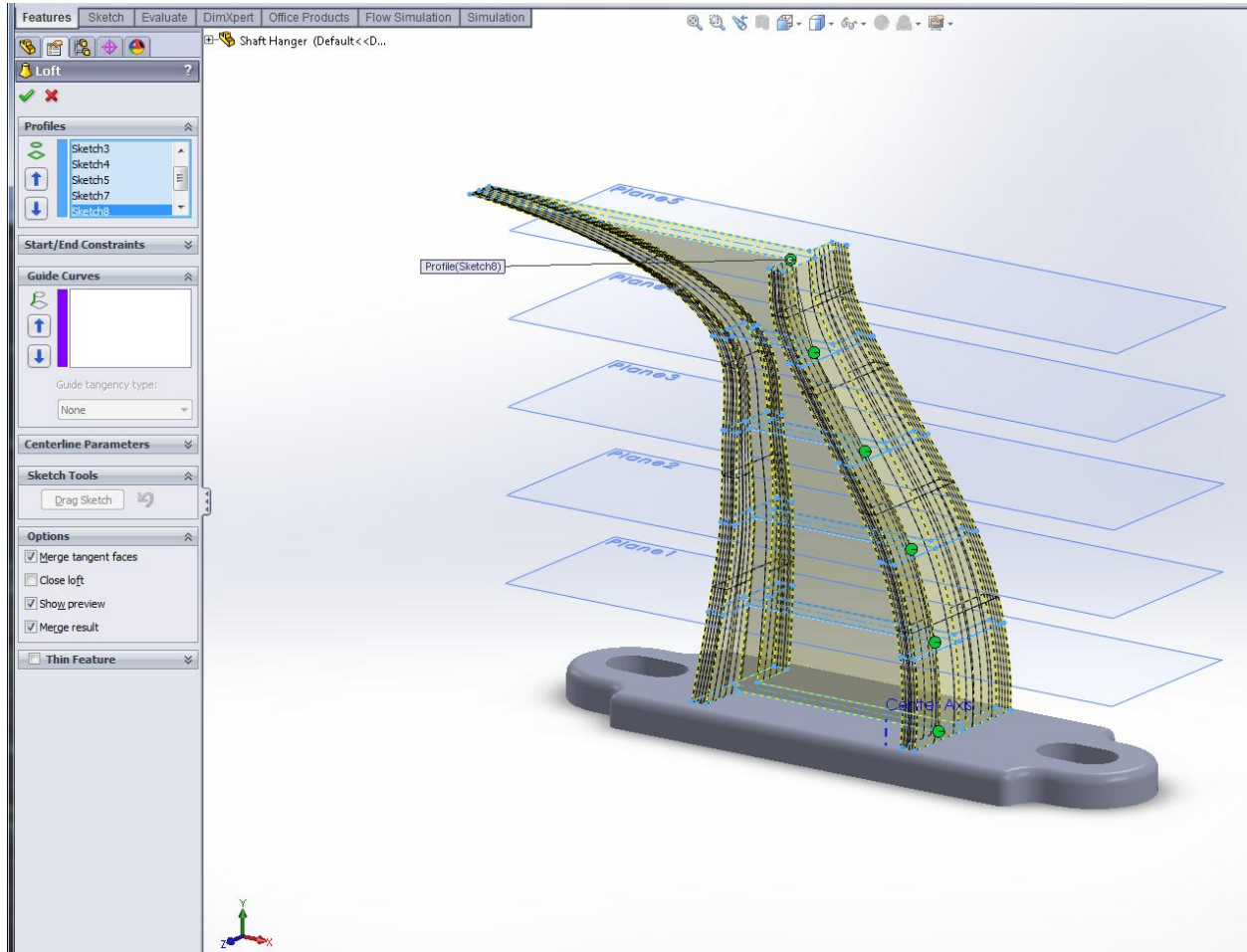


The resulted sections for lofting or parallel blend are shown below.

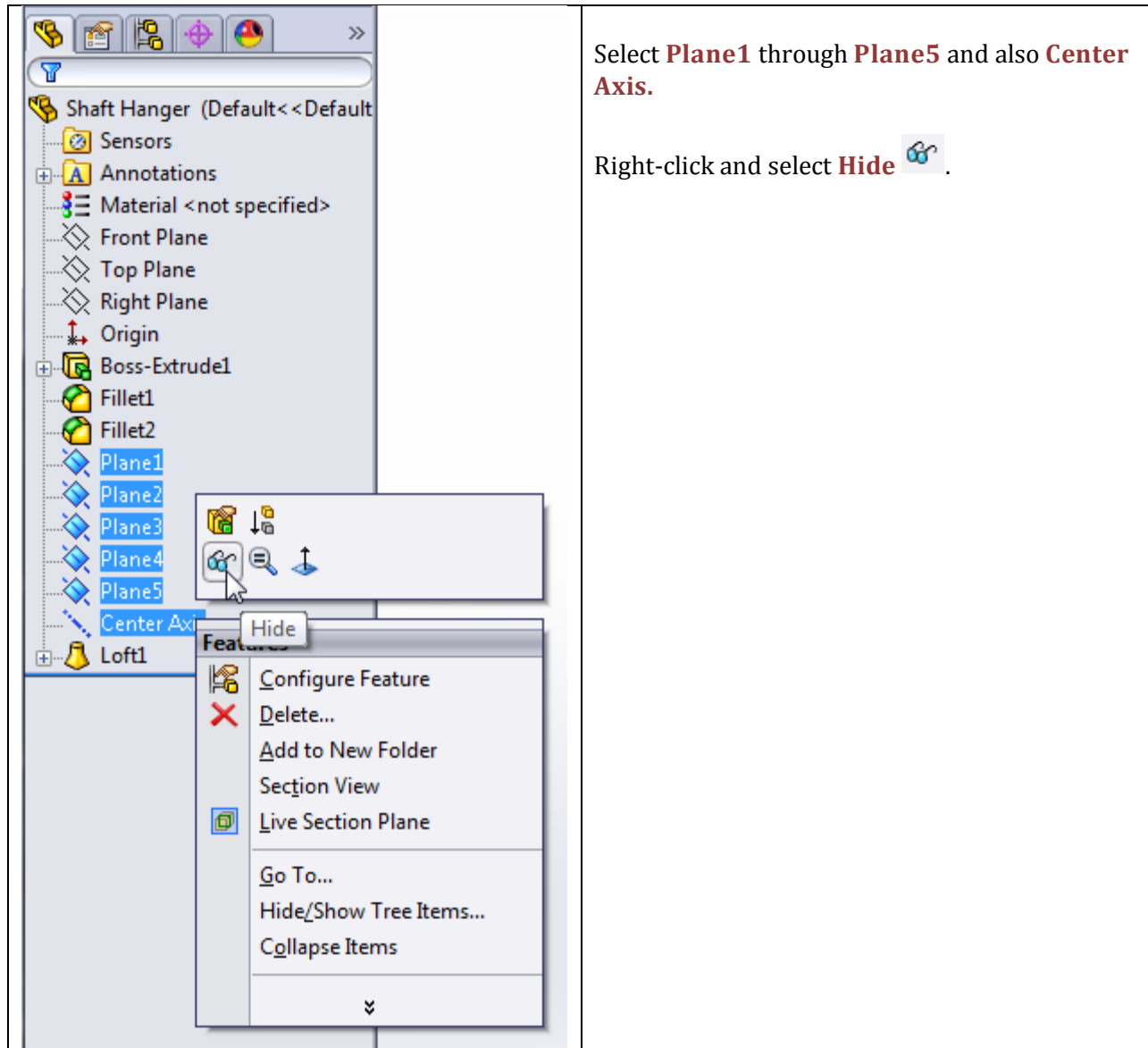



Pick the profiles starting from the bottom. For each profile, select the point from which you want the path of the loft to travel – note the green points shown below.

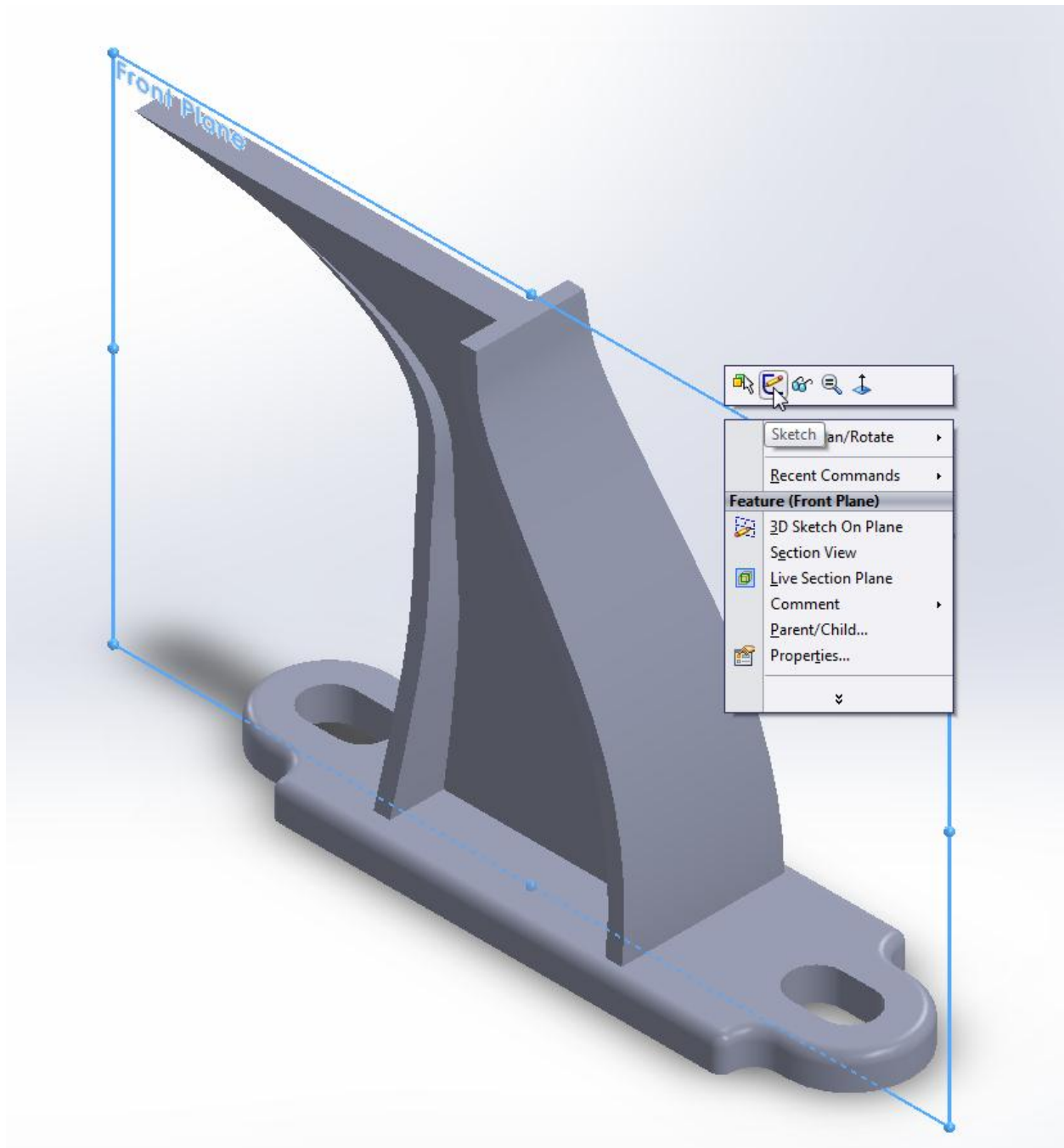
If the loft preview shows an undesirable loft, re-select or reorder the sketches to connect different points on the profiles.



Click .

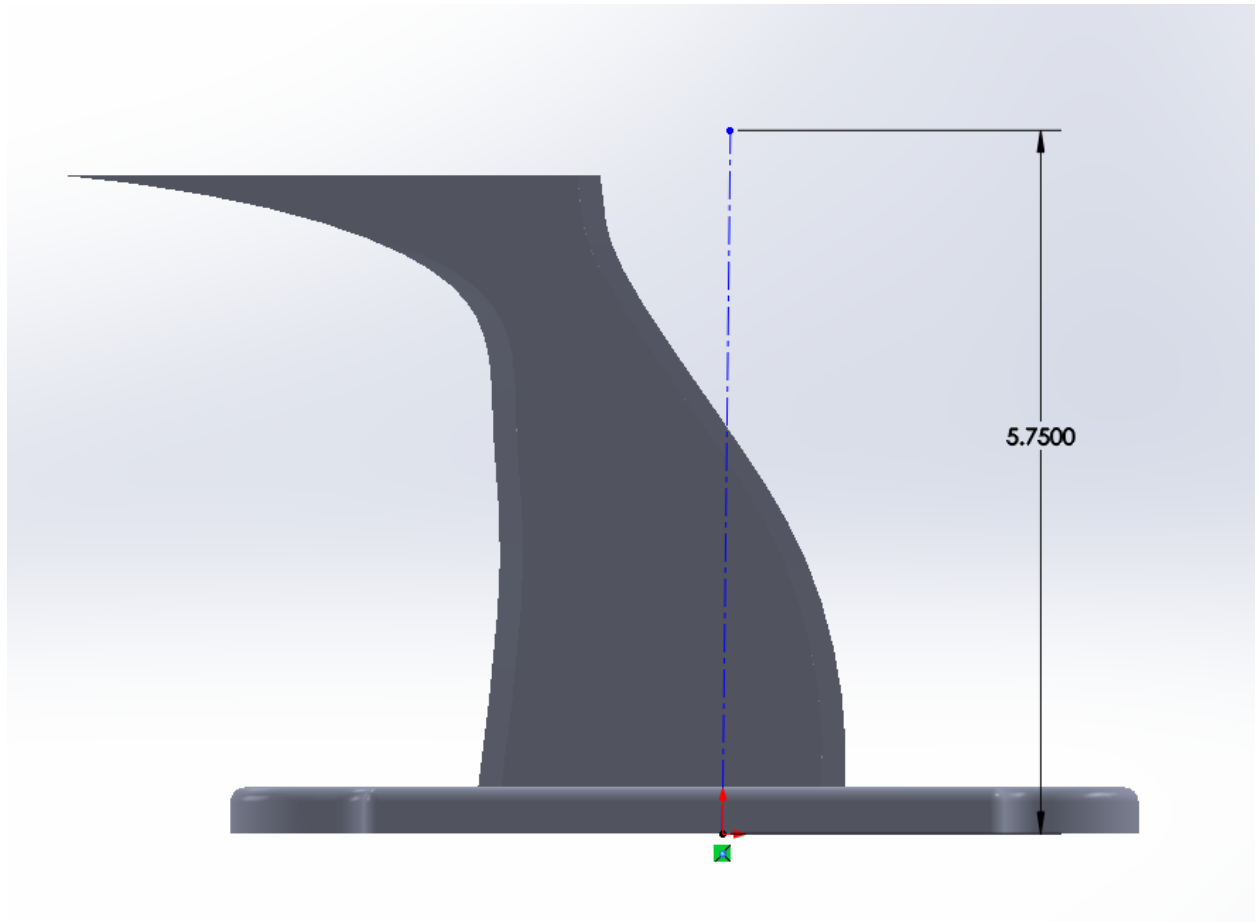


Highlight **Front Plane** from **FeatureManager** tree. Right-click **Front Plane** in the graphics area and select **Sketch** .

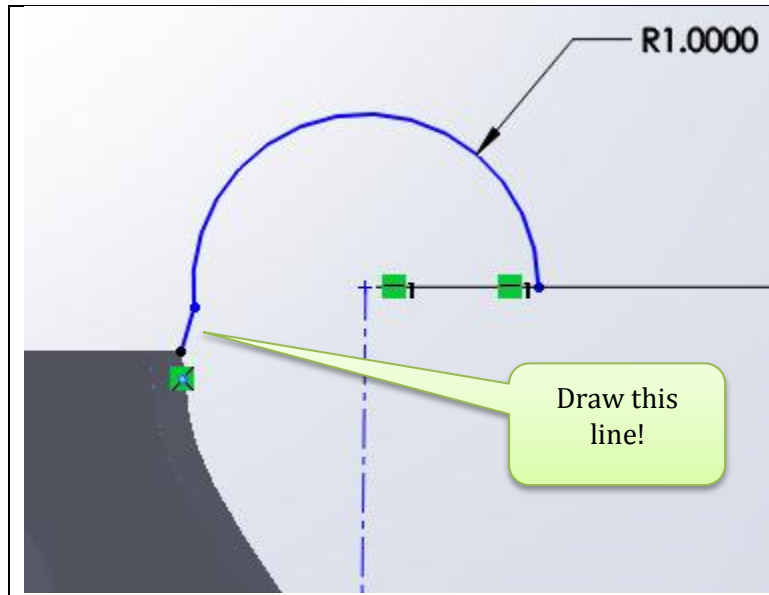


Set the display to **Front** view .

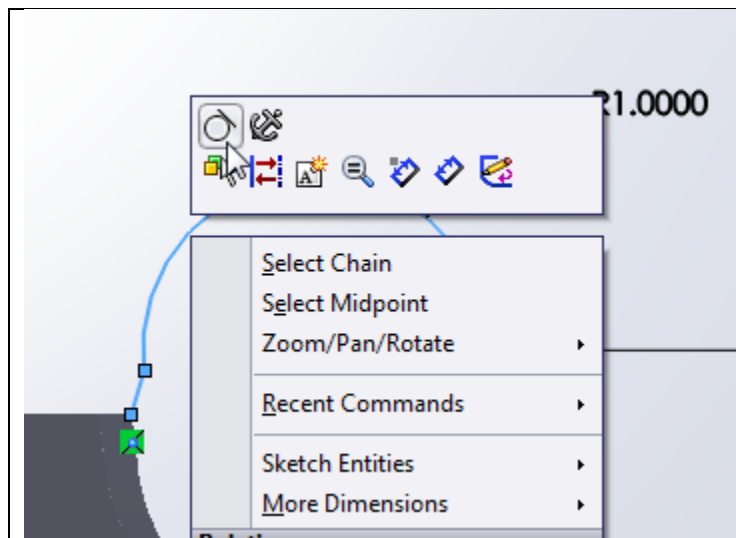
Draw a vertical centerline through the origin and dimension it to be **5.75** inches.




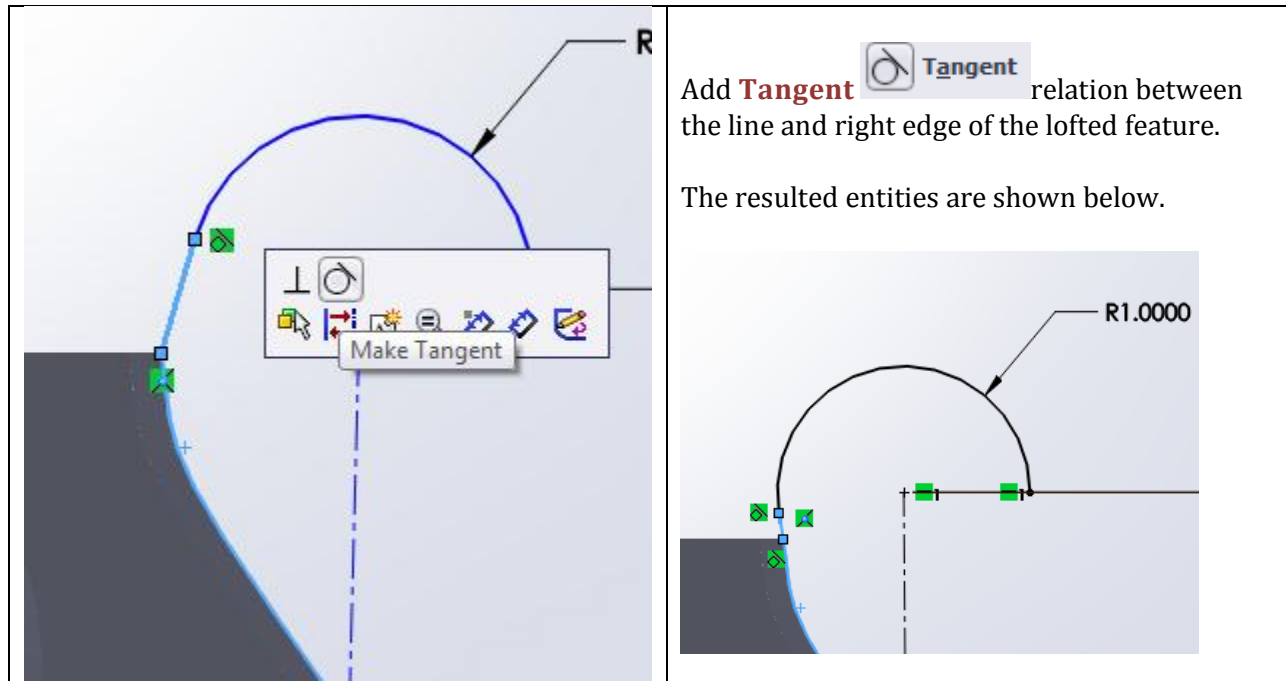
<p>A 2D CAD sketch showing a horizontal line with a center point and an arc with a radius of R1.0000. The center point is marked with a green square and a blue dot. The arc is drawn from the center point to the right end point of the horizontal line. The radius is labeled R1.0000.</p>	<p>Using the top endpoint of the centerline as the center, draw an arc with a radius of 1 inch as illustrated.</p> <p>Constrain (Add Relations) the center and the right end point of the arc so that both are horizontal</p> <p><input type="checkbox"/> Horizontal</p>
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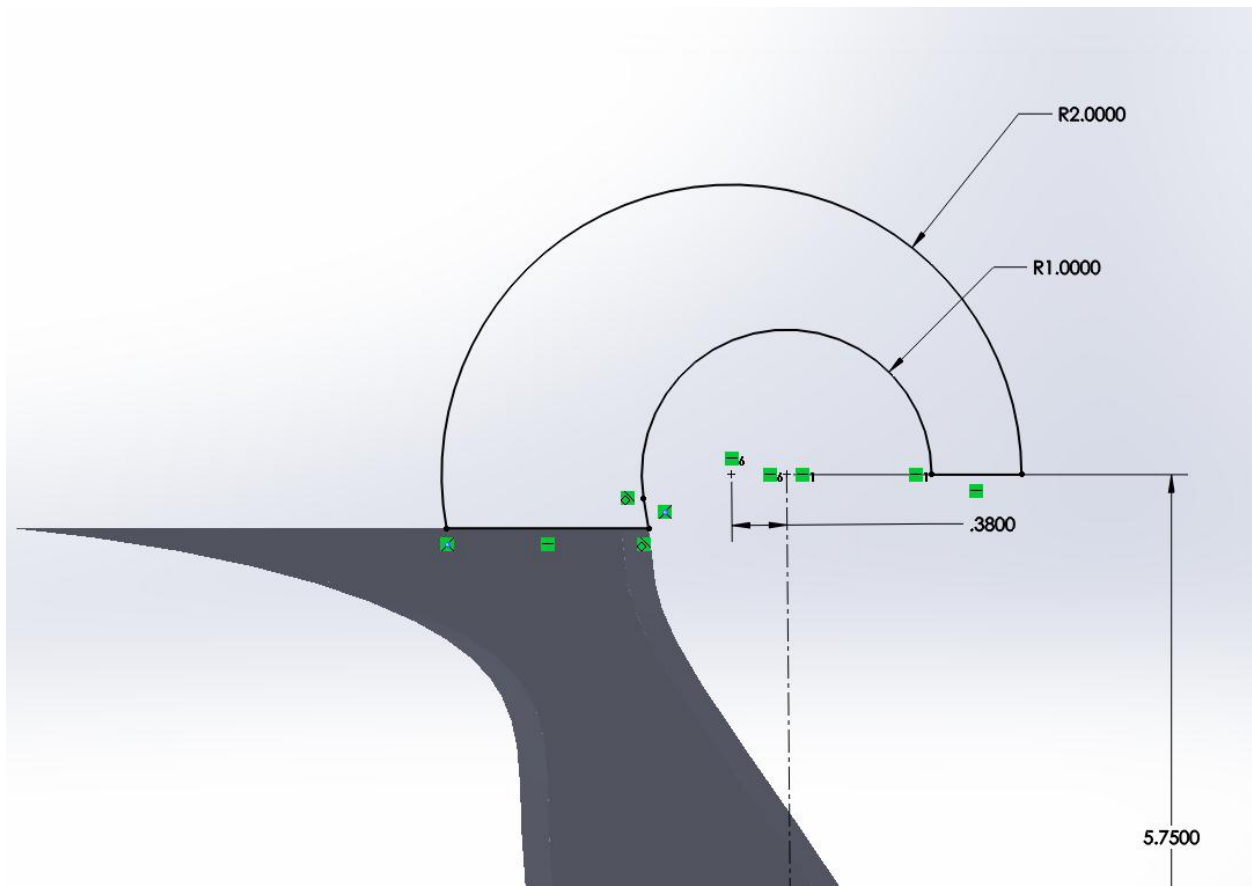
Draw a line connecting the left end of the arc and upper right vertex of the lofted feature.



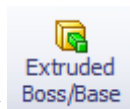
Add **Tangent**  **Tangent** relation between the line and the arc.

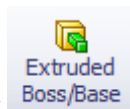


Draw a **2** inch radius of arc centered at a point located on the left of **1** inch radius arc. The distance between the two center points is **0.38** inch. Be sure to apply horizontal and other constraints as illustrated in the figure below. Close the section.



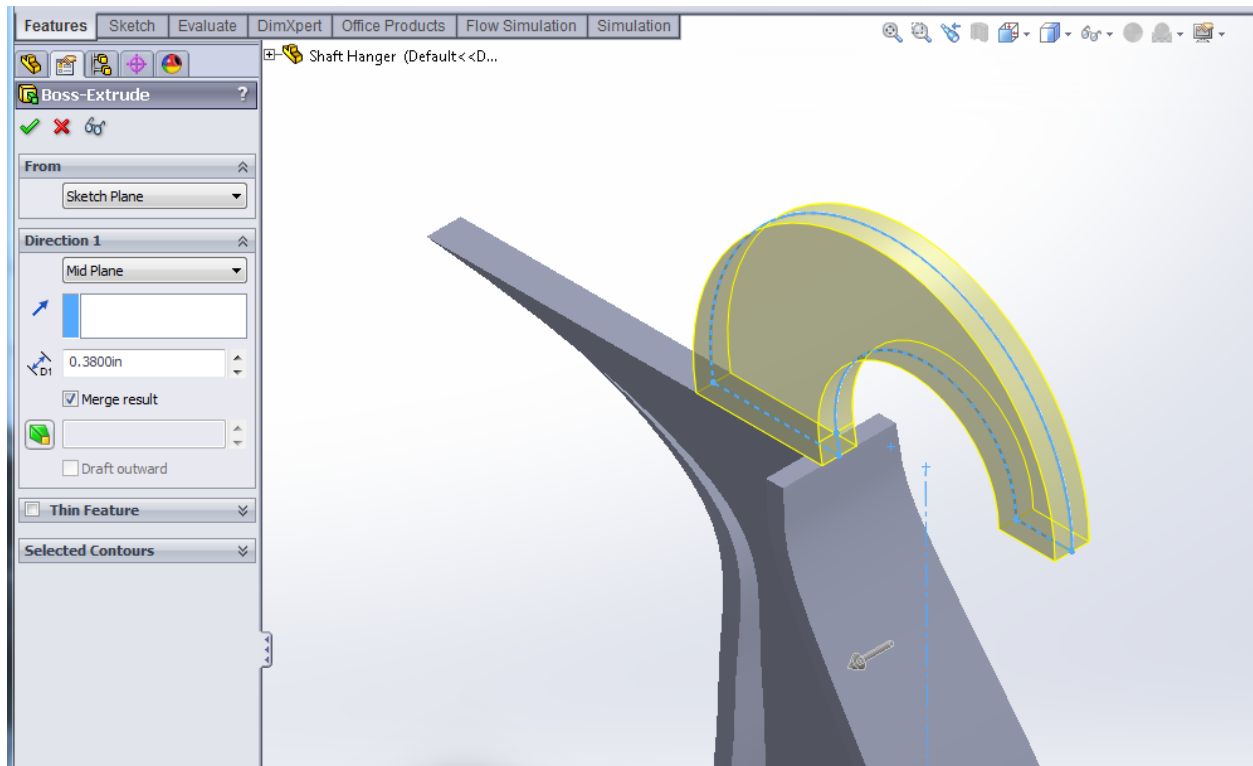
Click .



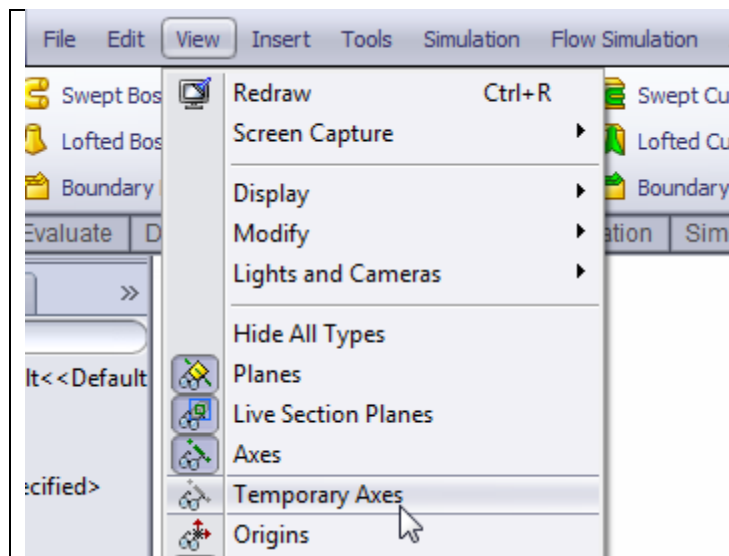
Select **Features** tab and click . Pick the section just created.

Select **Mid Plane** for **Direction 1**.

Enter **0.38** inch for the depth.



Click .

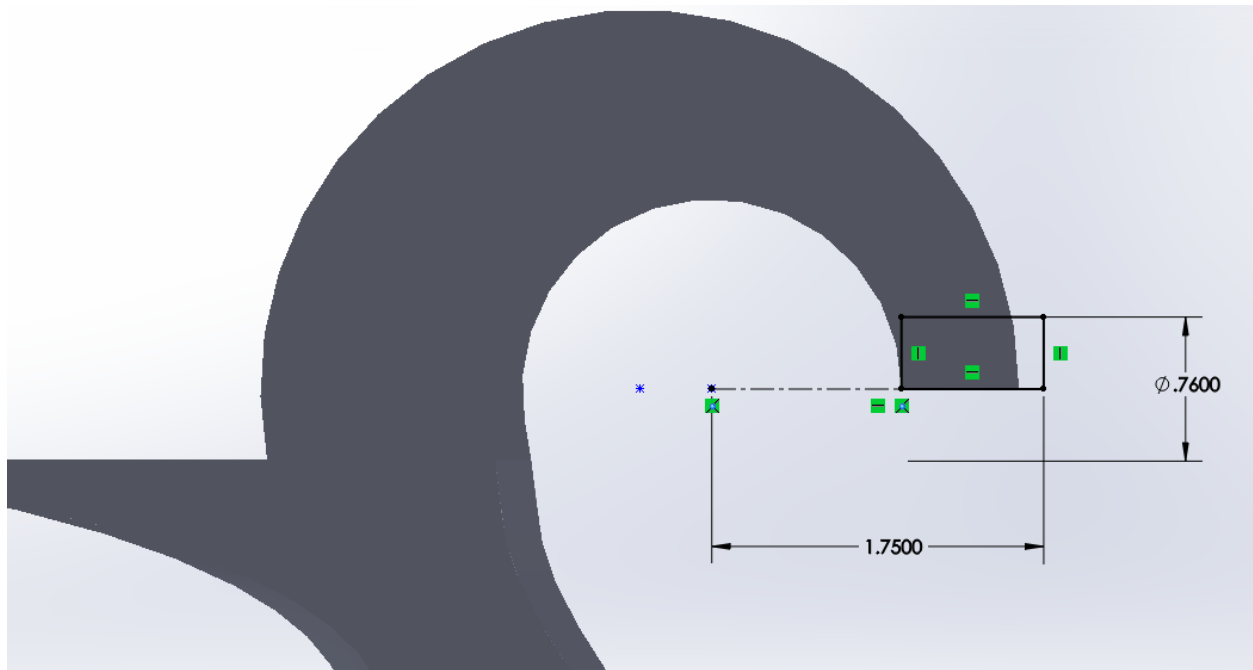



Select **View -> Temporary Axes**.

Right-click **Front Plane** and select **Sketch** .

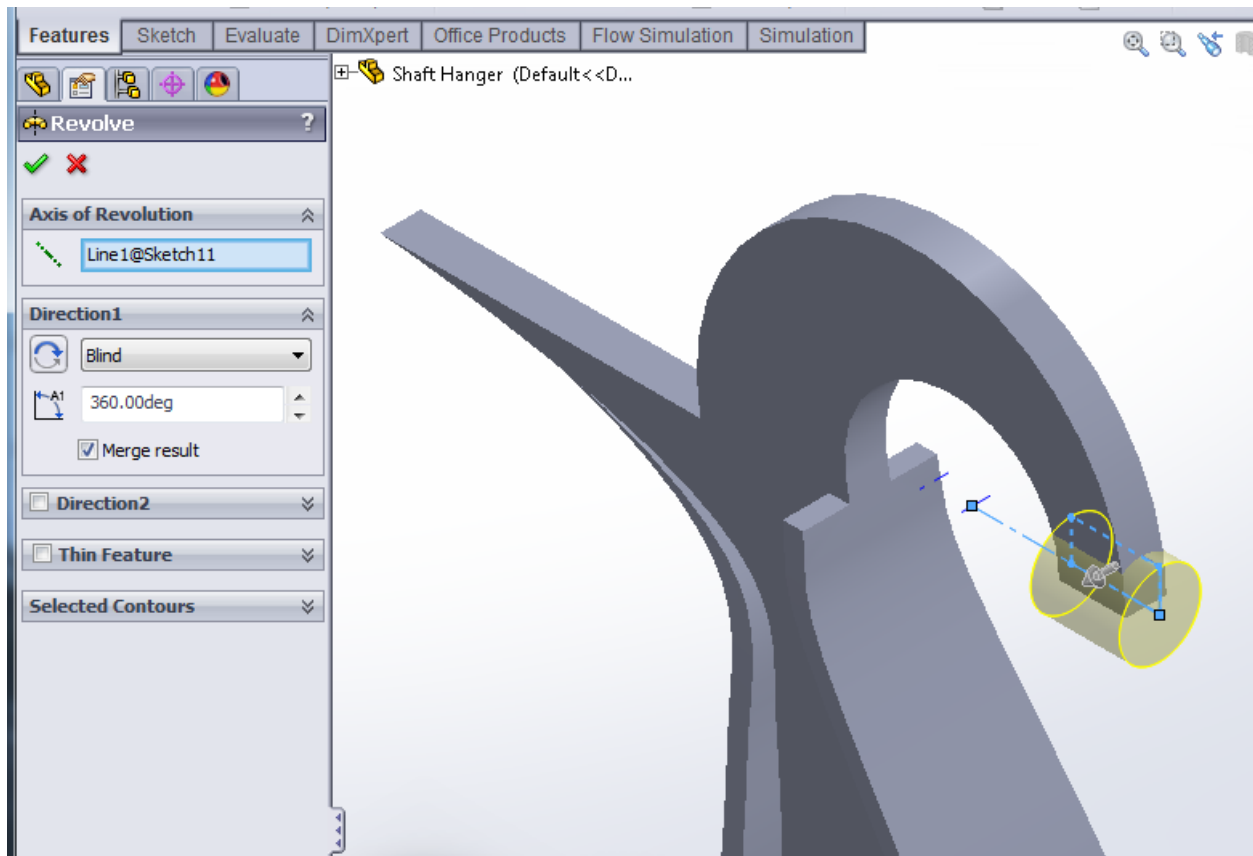
Set the display to **Front** view .

Sketch the following centerline and rectangular section.



Click  and click  to exit the sketch mode.

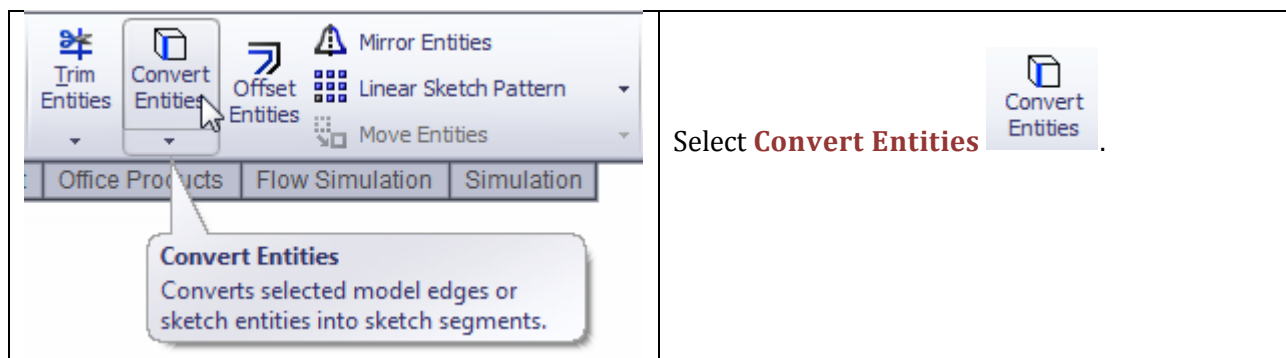
Select **Features** tab and click . Create a revolved feature as shown below.



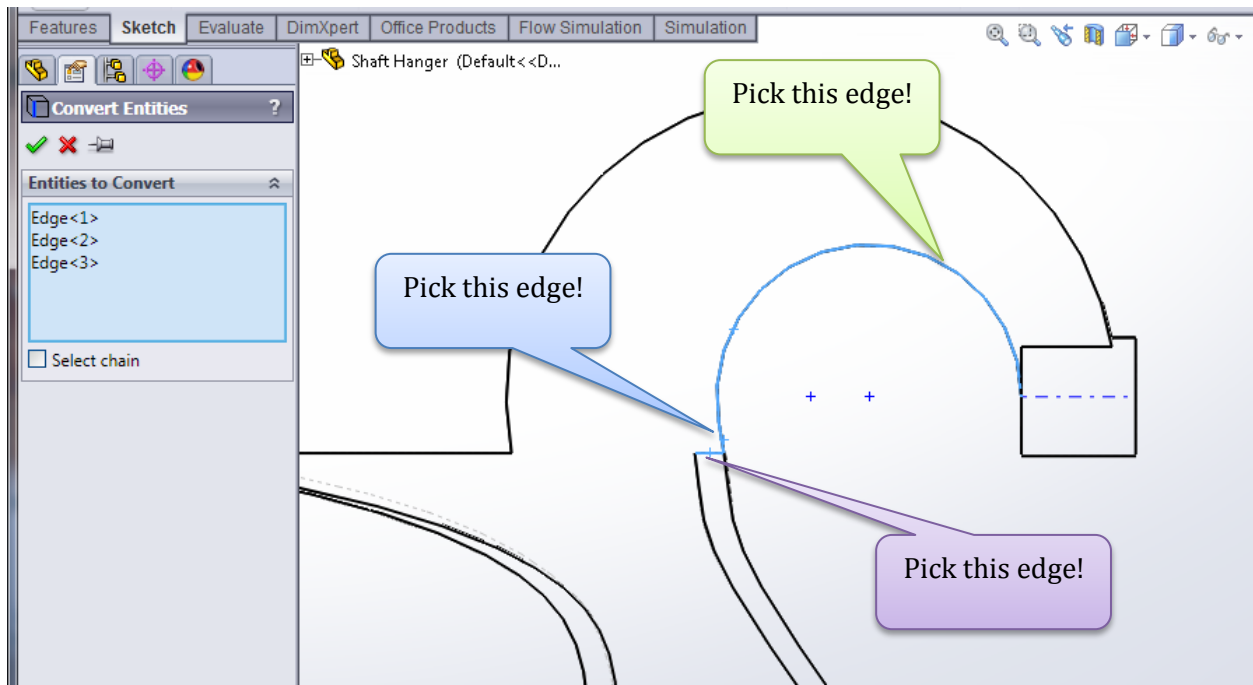
Click  .

Right-click **Front Plane** and select **Sketch**  .

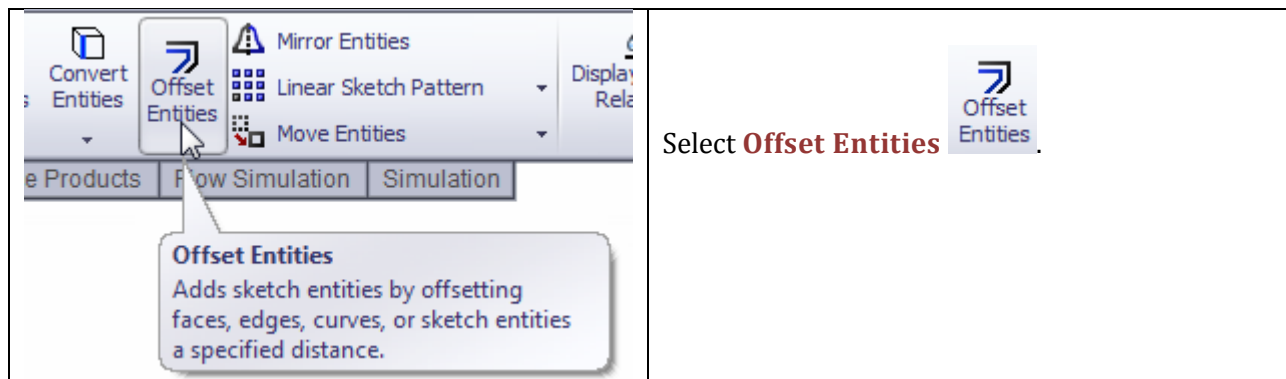
Set the display to **Front** view  .



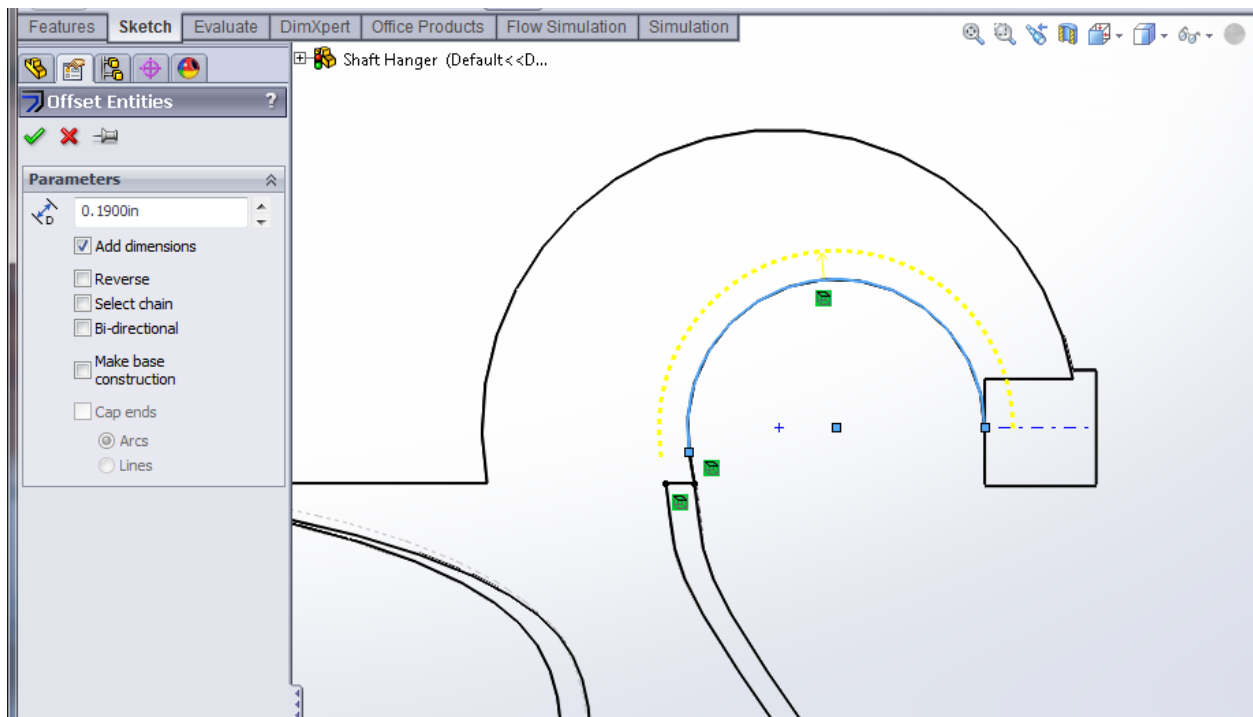
Select 3 edges as shown below.



Click .

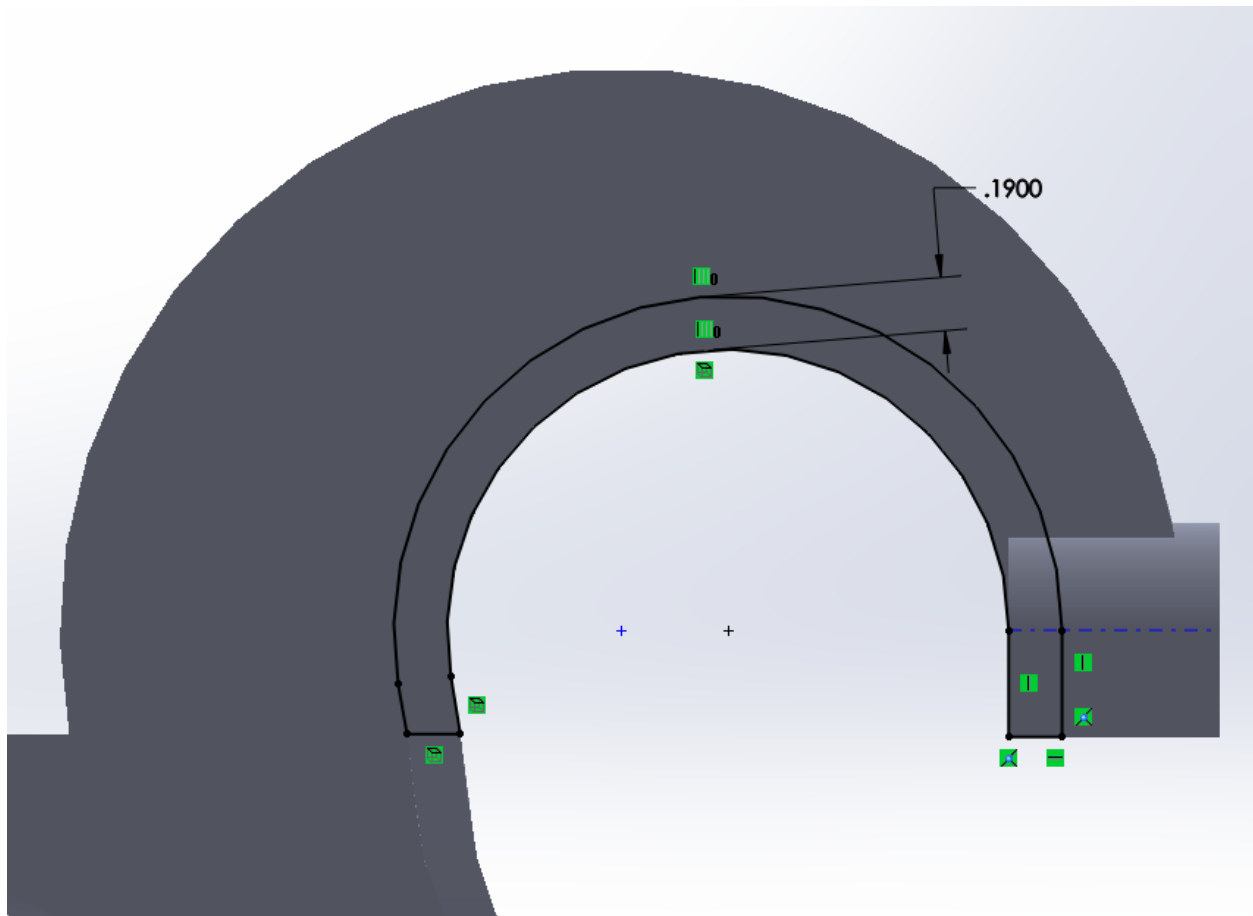


Pick the arc edge to be offset. Enter the offset distance of **0.19** inch.

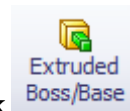


Click  .

Complete the sketch so that the following section is created.



Click **Exit Sketch** to exit the sketch mode. Select **Features** tab and click

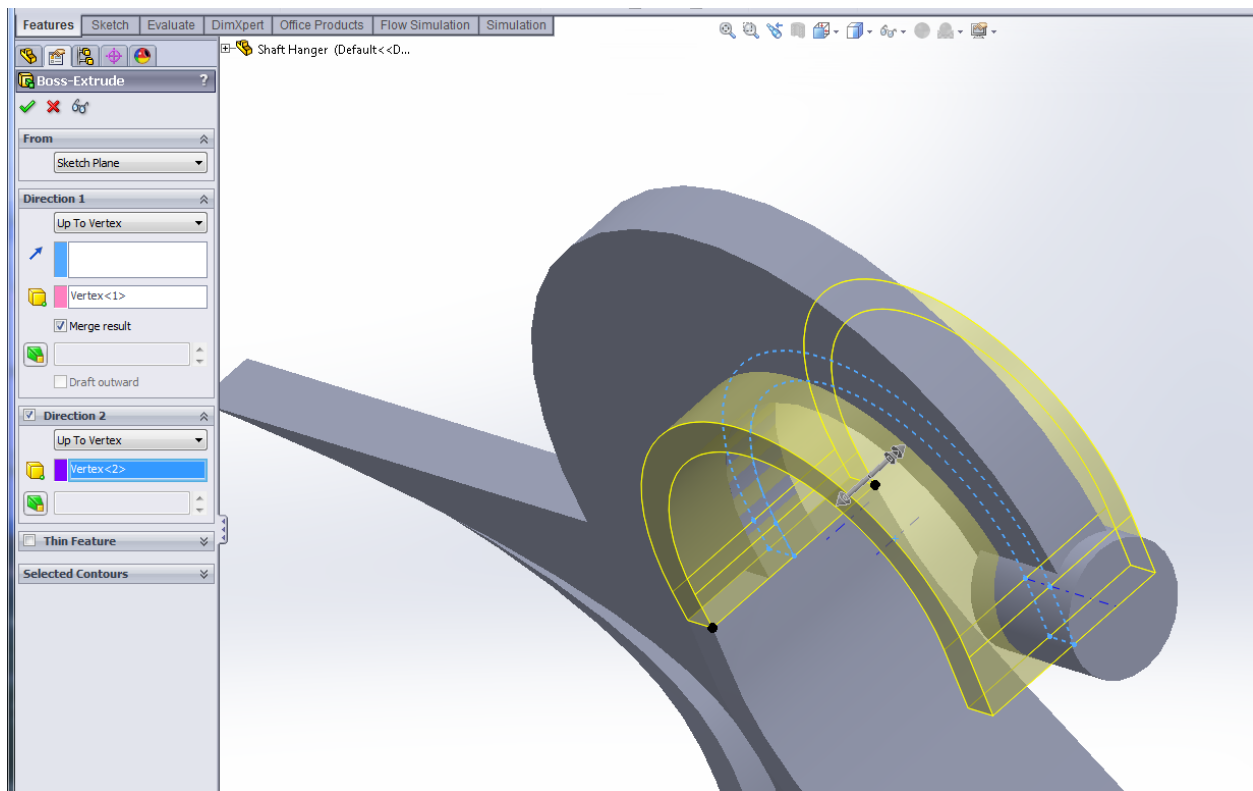


Select **Up To Vertex** for **Direction 1**.


Pick top front vertex of the lofted feature.

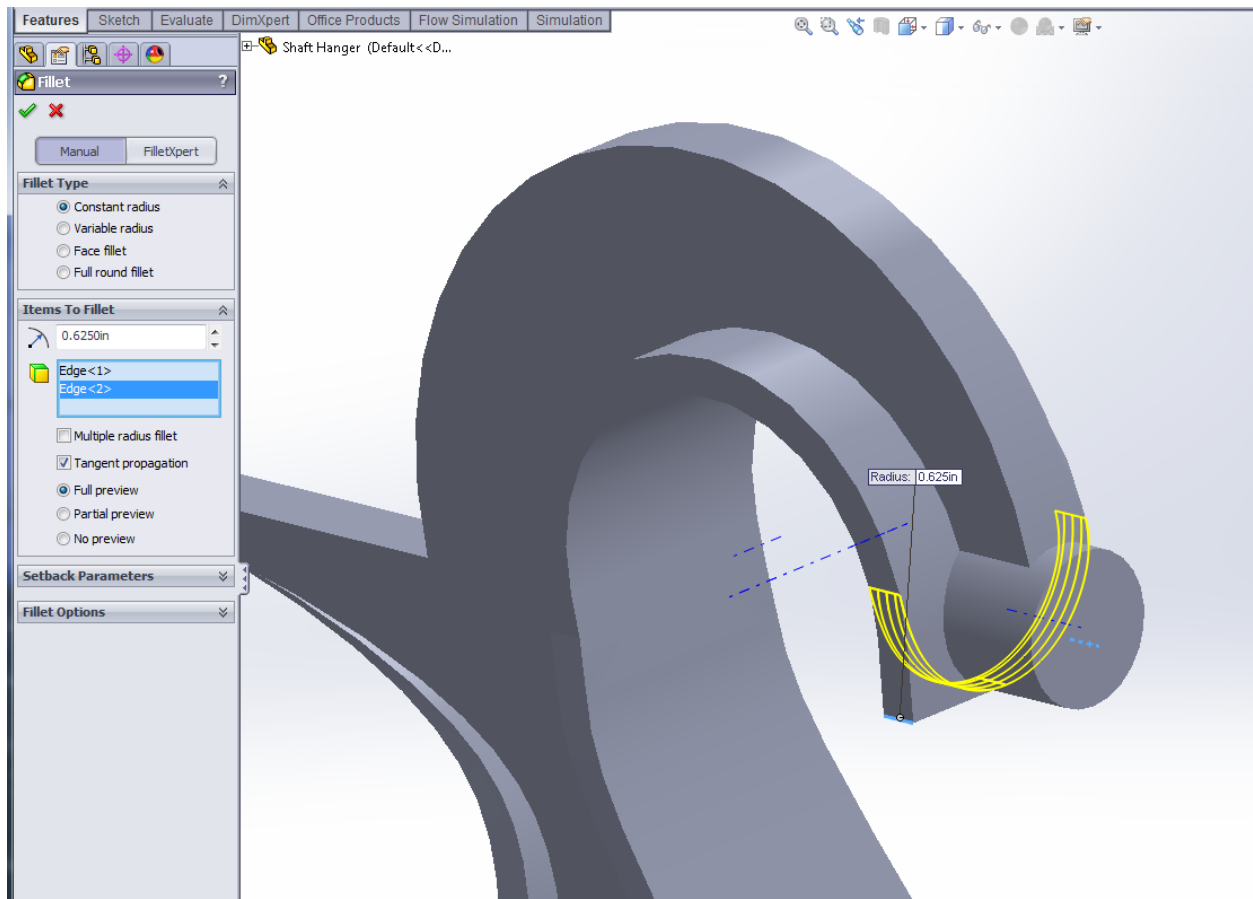
Select **Up To Vertex** for **Direction 2**.

Pick top rear vertex of the lofted feature.

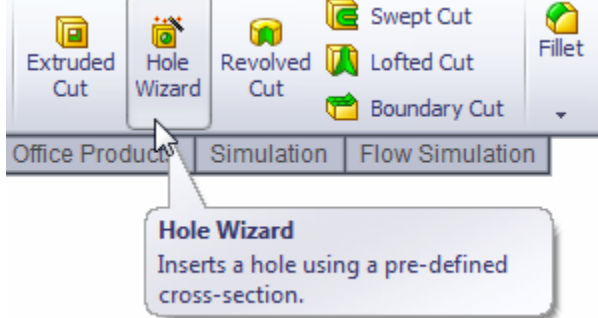
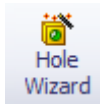


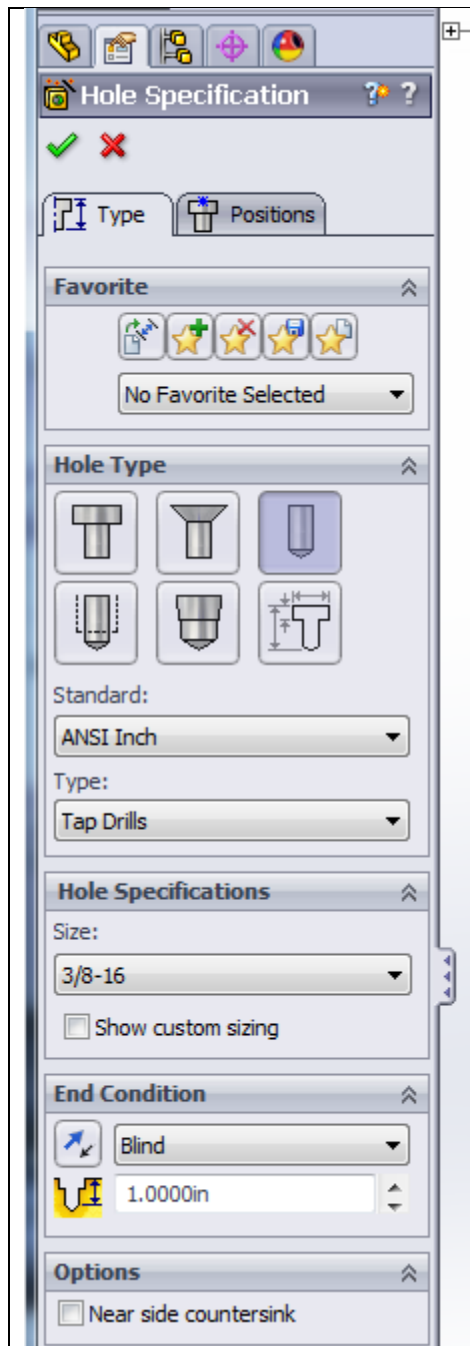
Click  .

Select **Fillet**  and pick two edges as shown below. Enter **0.625** inch for the radius.



Click .

	<p>Click Hole Wizard .</p> <p>The Hole Specification PropertyManager is displayed.</p>
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Select **ANSI Inch** for **Standard**.

Be sure **Hole**  is selected.

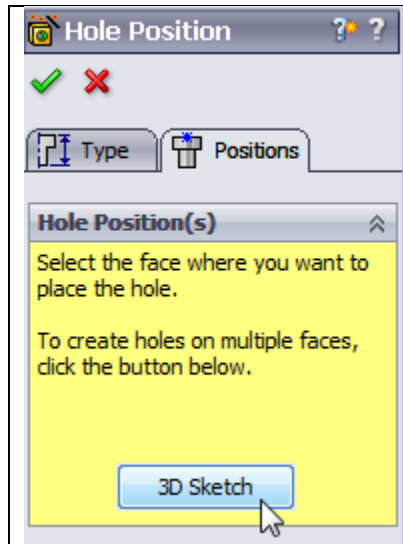
Select **Tap Drills** for **Type** of the hole.

Select **3/8-16** for **Size**.

Select **Blind** for **End Condition**.

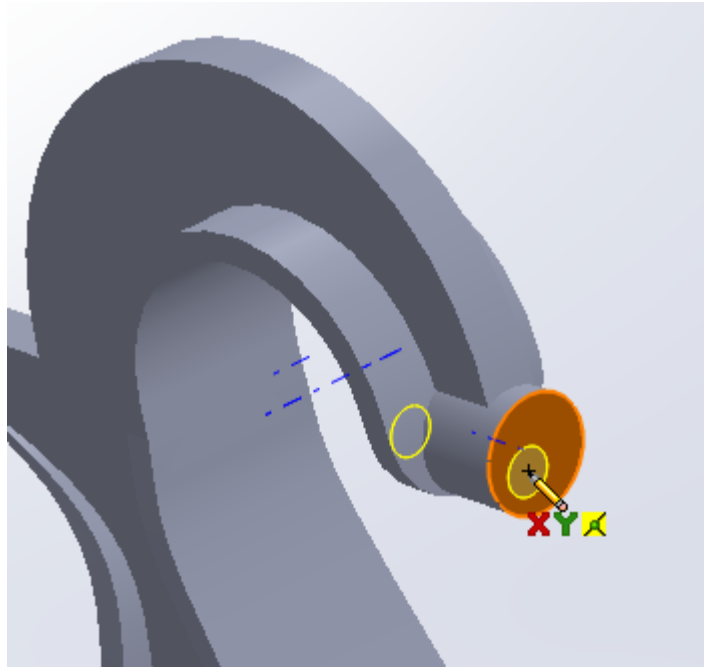
Enter **1** inch for the depth.

Click the **Position** tab.



Click **3D Sketch**.

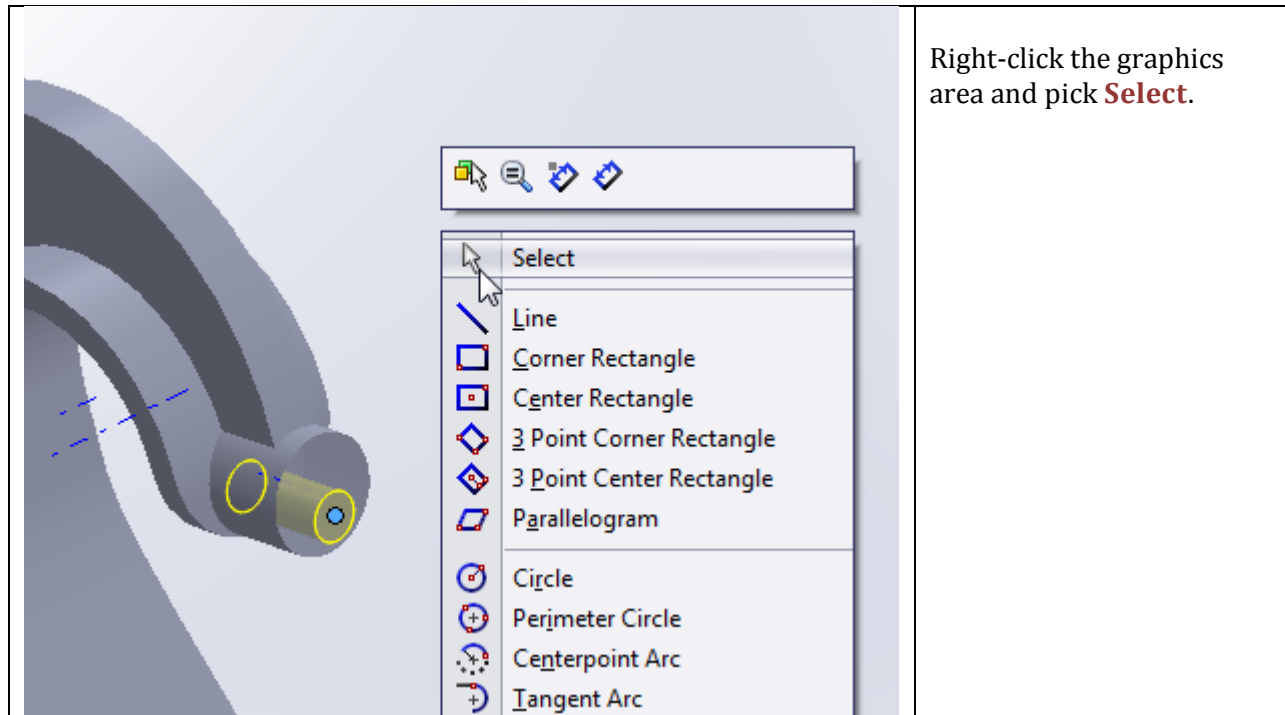
Click on the face of revolved feature as shown.



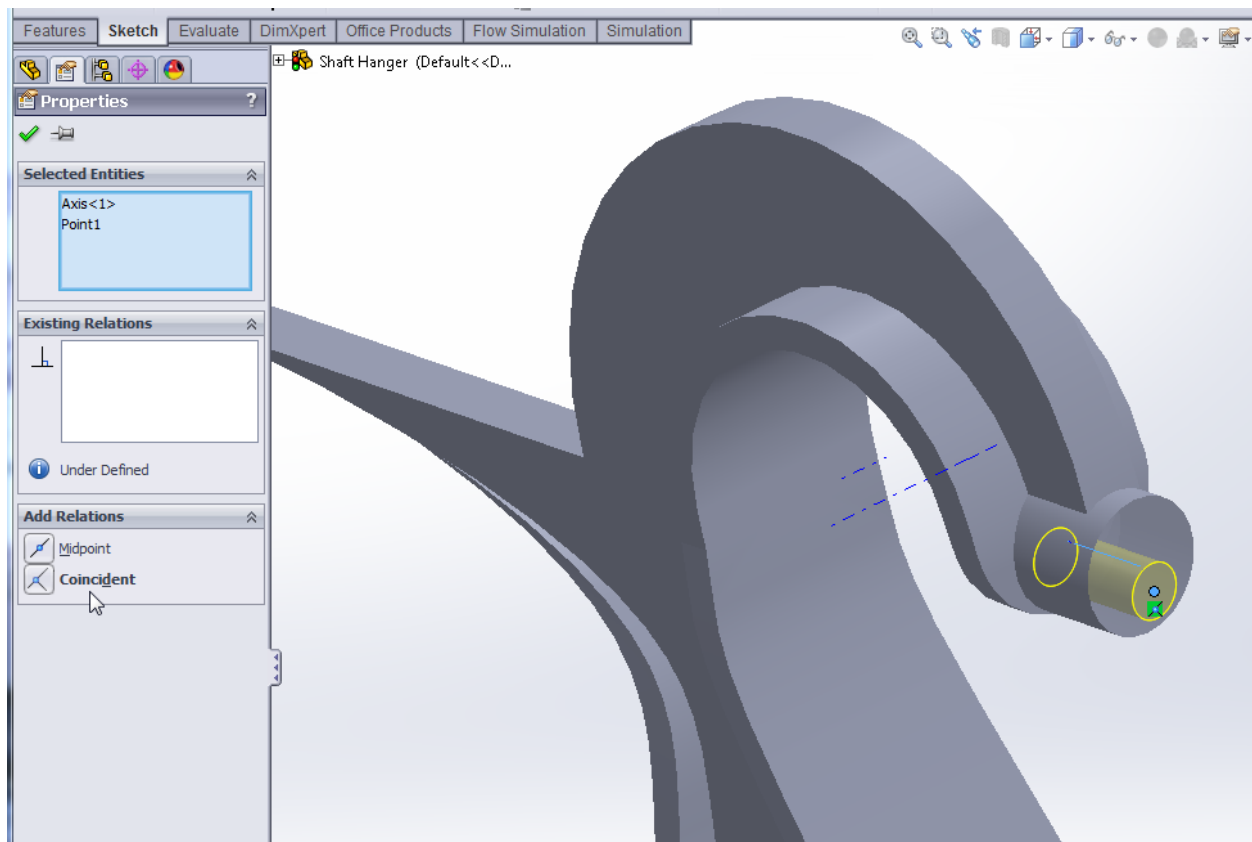
When point tool




is active, wherever you pick, a sketch point will be created.



Pick the center of the hole and hold **CTRL**-key and pick the nearby axis.



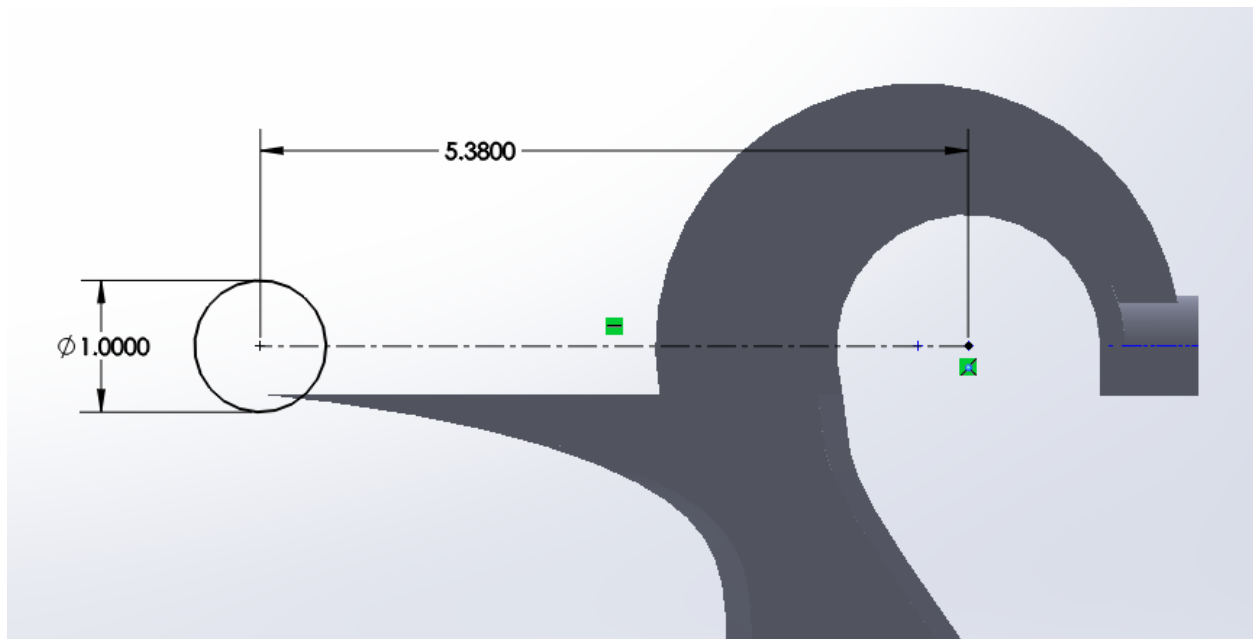
Select **Coincident**.


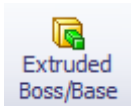
Click  twice.

Highlight **Front Plane** from **FeatureManager** tree. Right-click **Front Plane** in the graphics area and select **Sketch** .

Set the display to **Front** view .

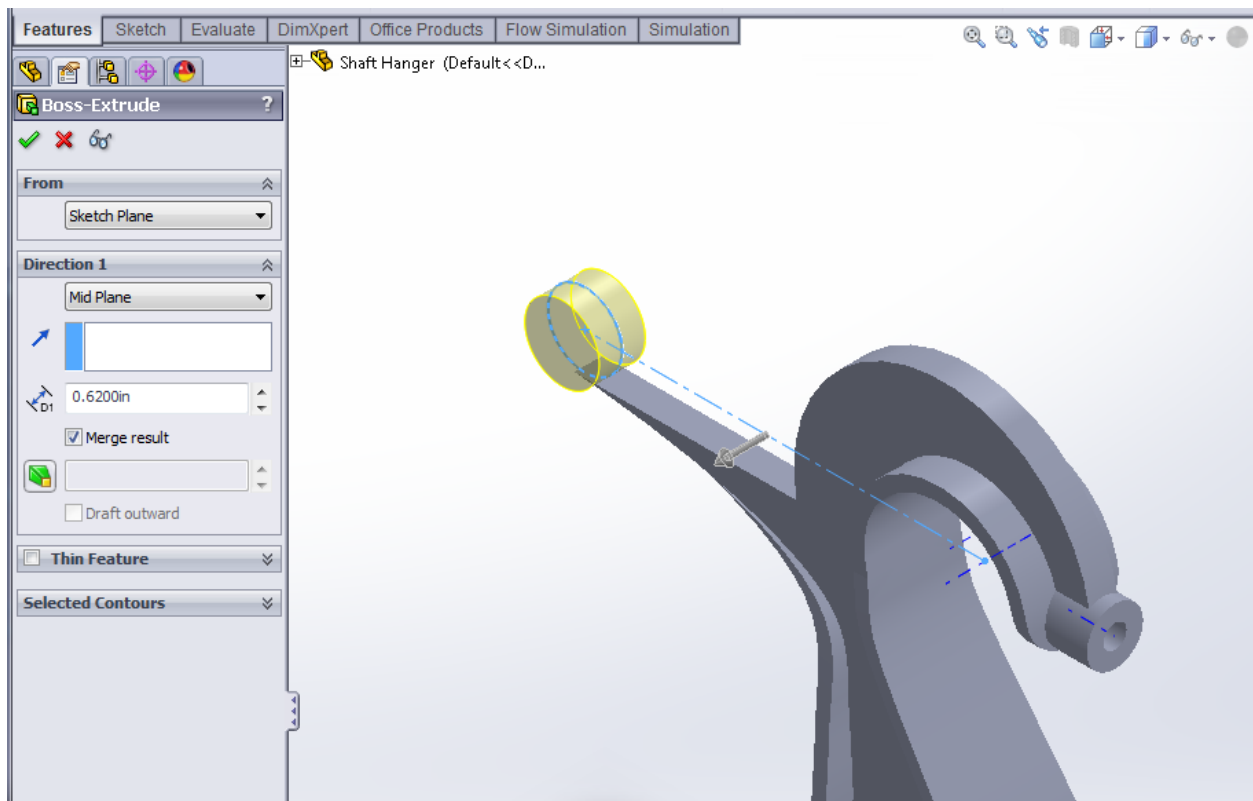
Sketch and dimension the following horizontal centerline and a circle at the left end of the centerline.



Click  to exit the sketch mode. Click .

Select **Mid Plane** for **Direction 1**.

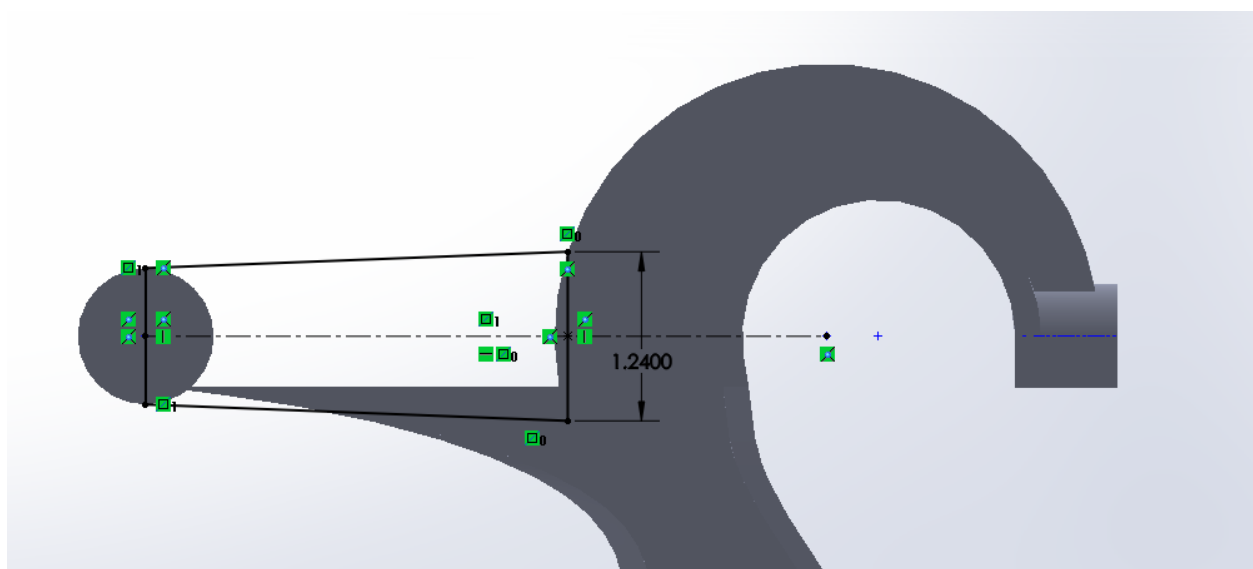
Enter **0.62** in. for the depth.



Click  from the **Properties PropertyManager**.

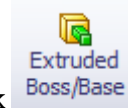
Highlight **Front Plane** from **FeatureManager** design tree. Right-click **Front Plane** in the graphics area and select **Sketch** .

Sketch the following section. Be sure you have the correct constraints shown below.



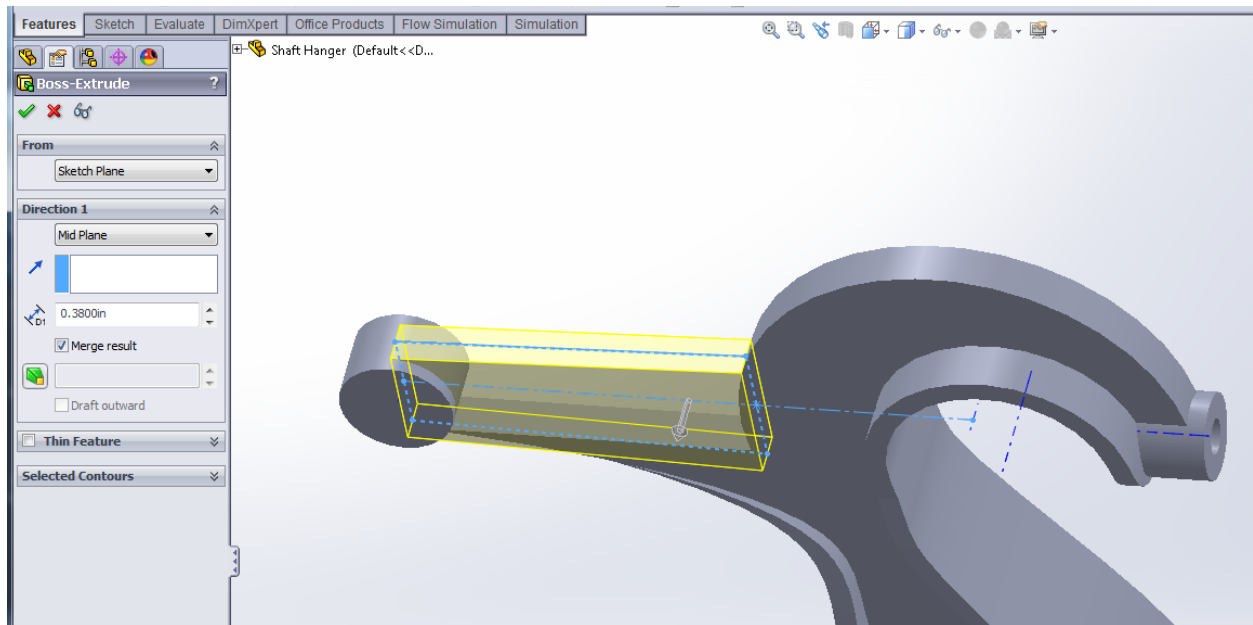


Click to exit the sketch mode. Select **Features** tab and click



Select **Mid Plane** for **Direction 1**.

Enter **0.38** inch for the depth.

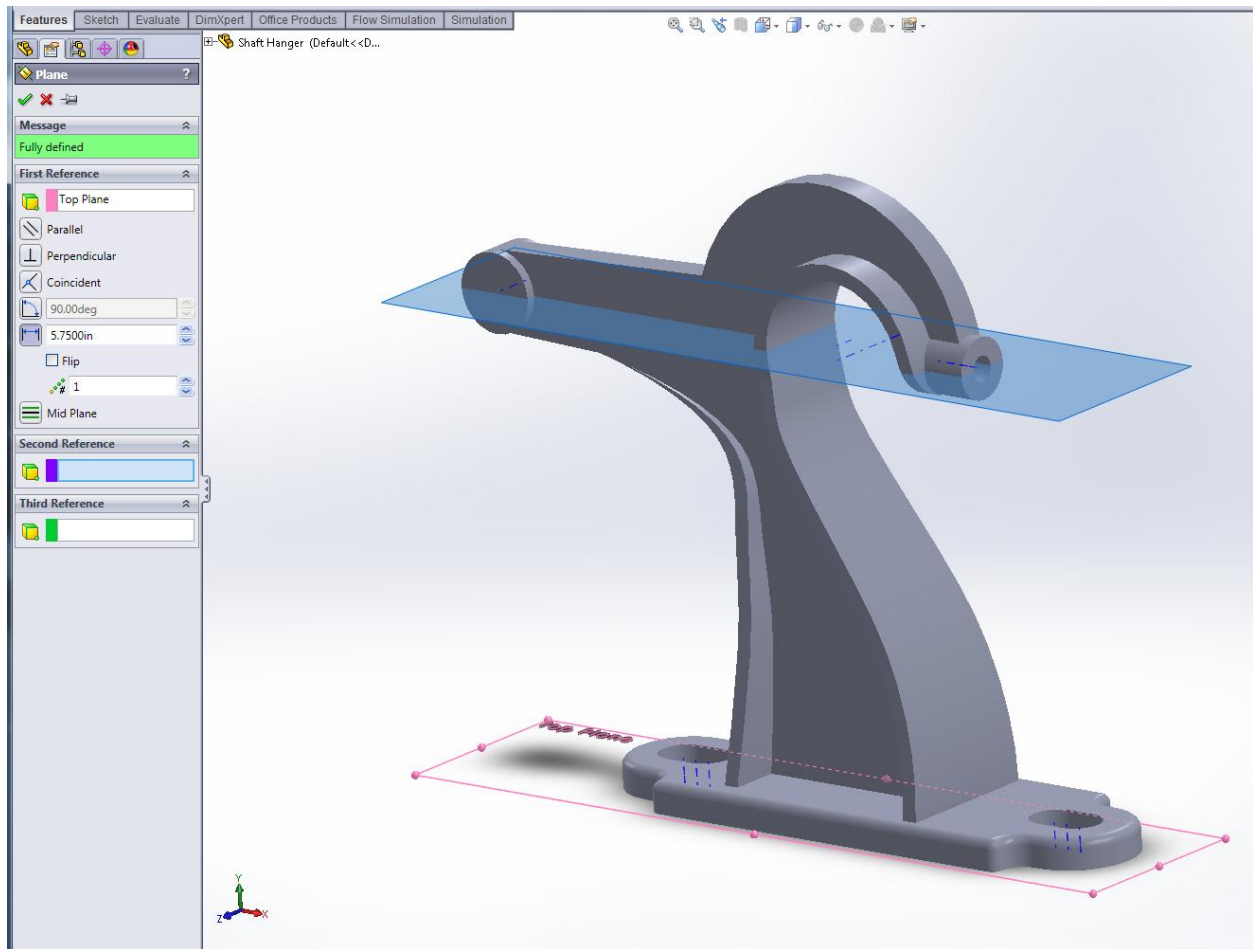


Click


Create a reference plane at the distance of 5.75 inches from the bottom of the base feature or **Top Plane**.

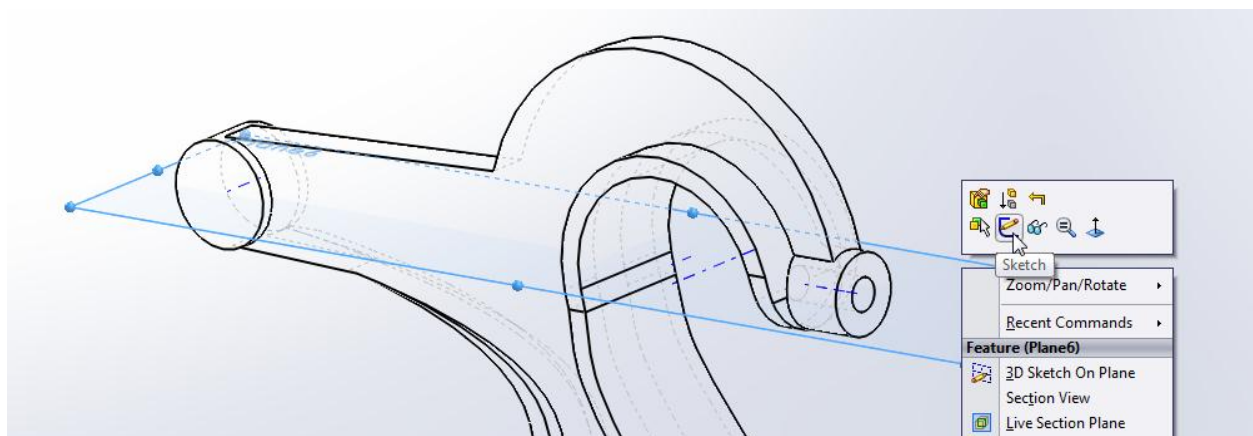
Hold the **CTRL** key down. Click and drag the boundary of the **Top Plane** upward in the graphics area.

Set the offset distance of **5.75** inches.



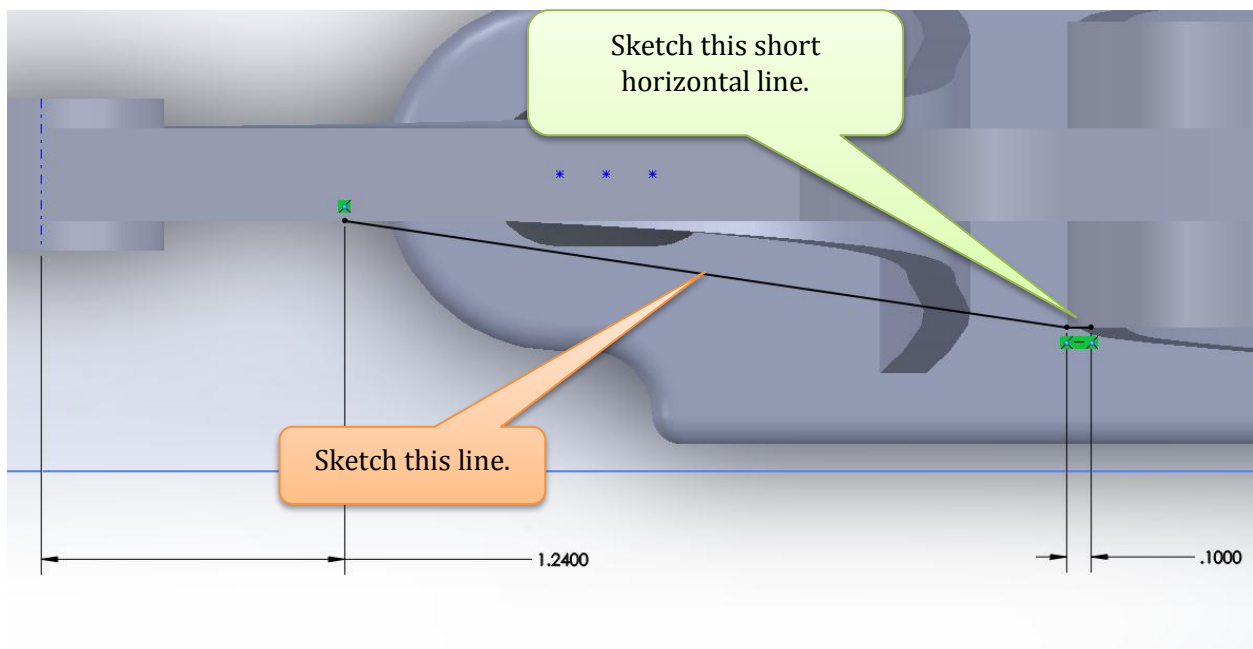
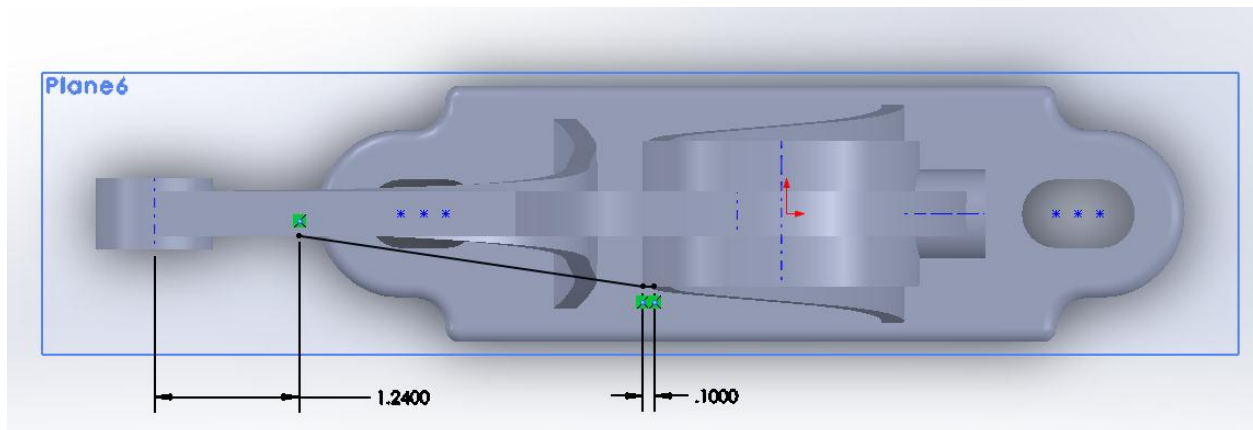
Click .


Right-click **Plane6** you just created and select **Sketch** .

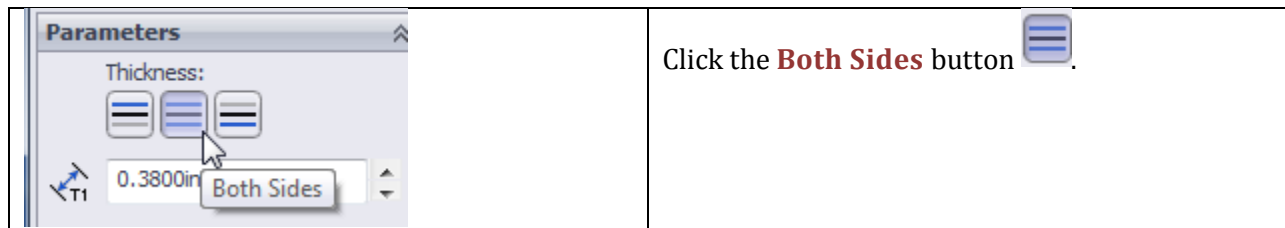
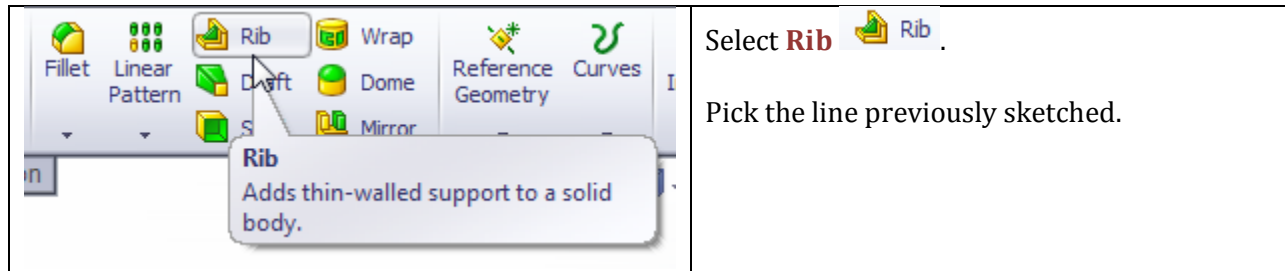


Set the display to **Top** view .

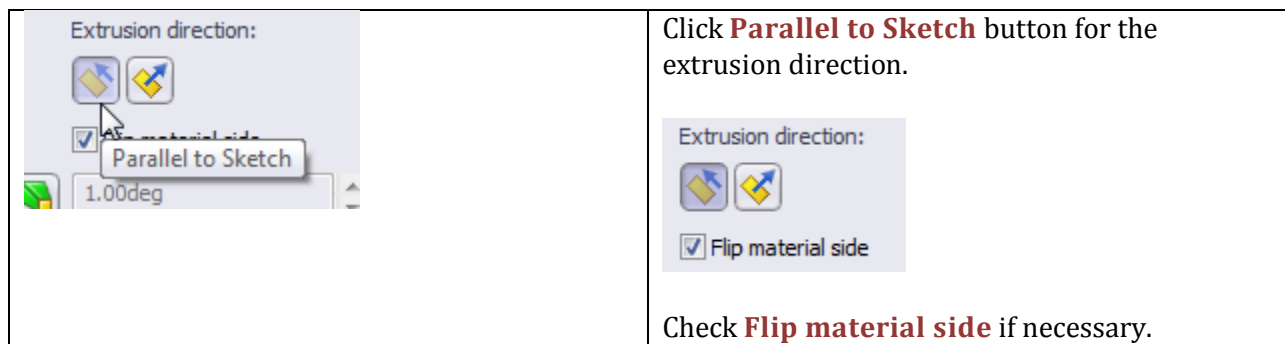
Sketch the following inclined and short horizontal lines – 2 lines altogether. Be sure to dimension the left endpoint of the line to the center axis of left cylinder. Note the coincident constraint points (green) as the end points of the line.

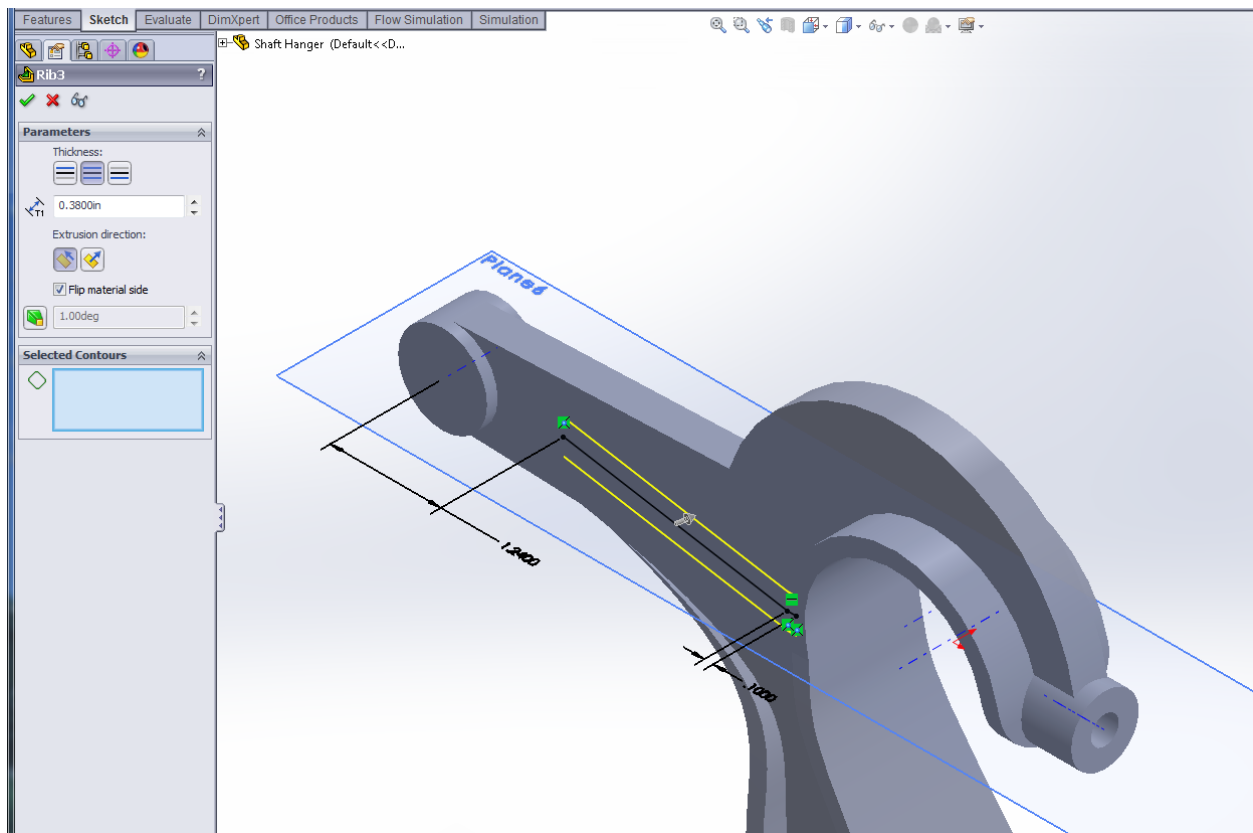


Click  to exit the sketch mode.

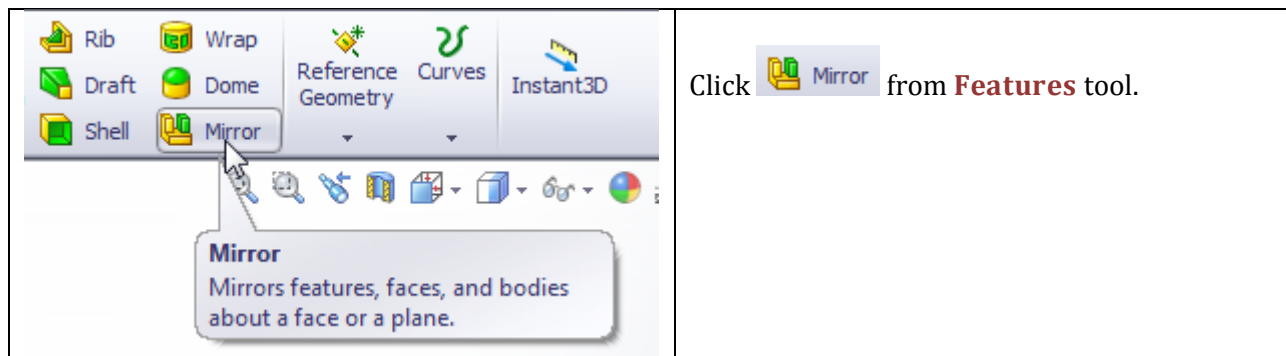


Enter **0.38** inch for the rib thickness.



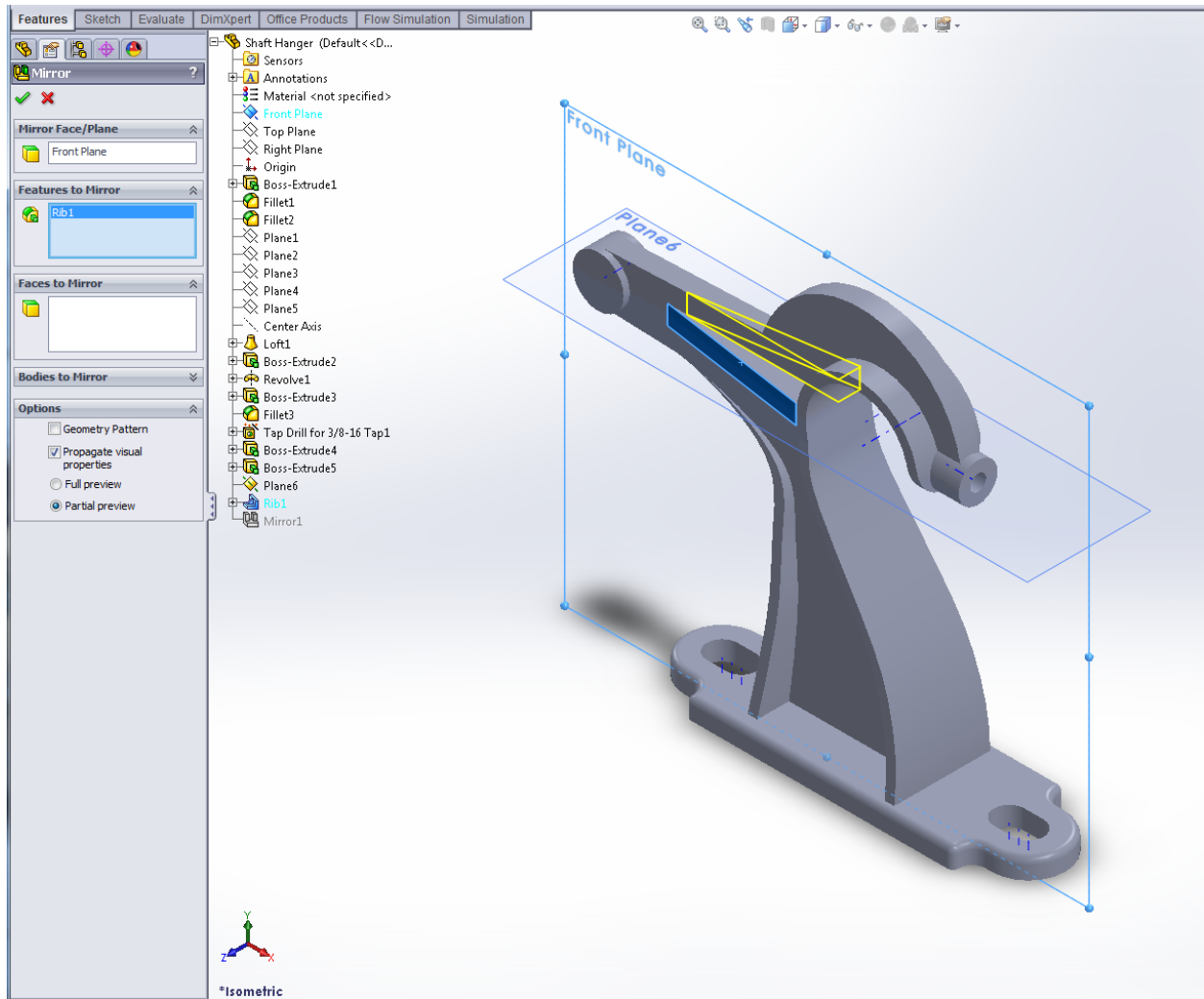


Click .

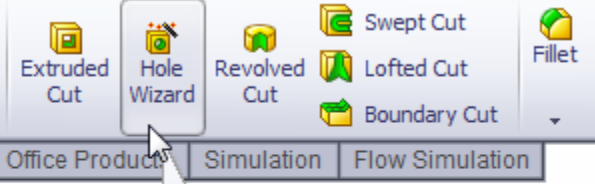
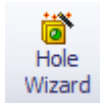


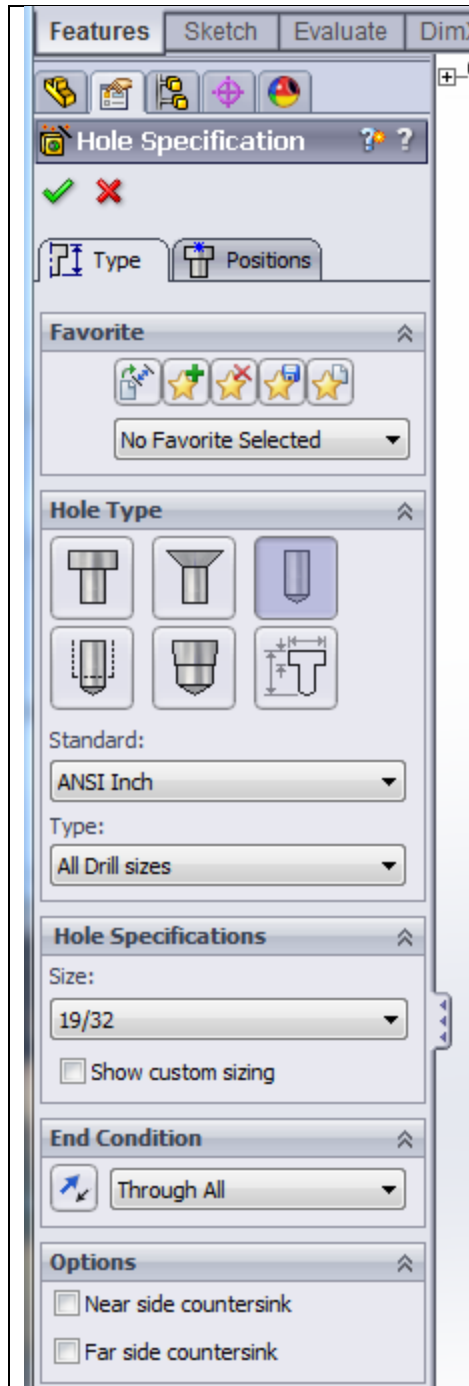
Select **Front Plane** for **Mirror Face/Plane**.

Select **Rib1** as **Features to Mirror**.



Click .

	<p>Click Hole Wizard .</p> <p>The Hole Specification PropertyManager is displayed.</p>
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Select **ANSI Inch** for **Standard**.

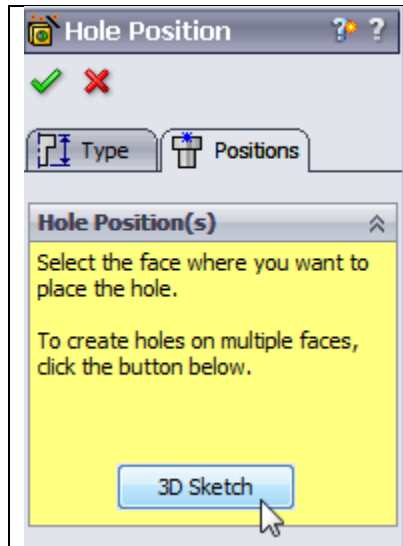
Be sure **Hole**  is selected.

Select **All Drill Size** for **Type** of the hole.

Select **19/32** for **Size**.

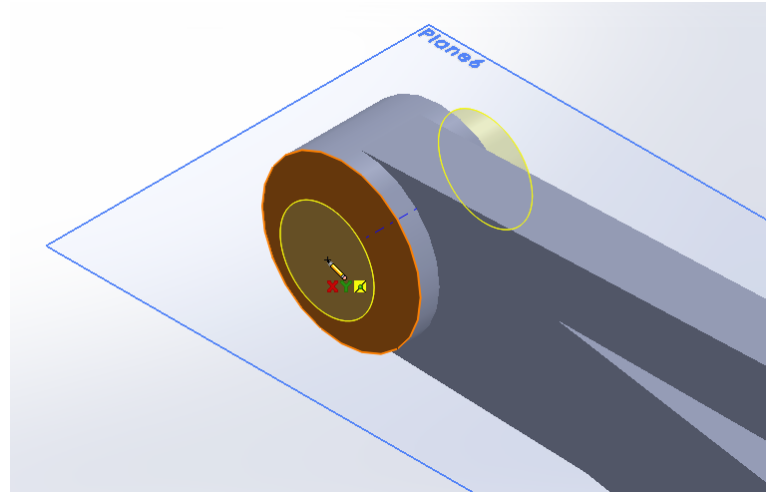
Select **Through All** for **End Condition**.

Click the **Position** tab.

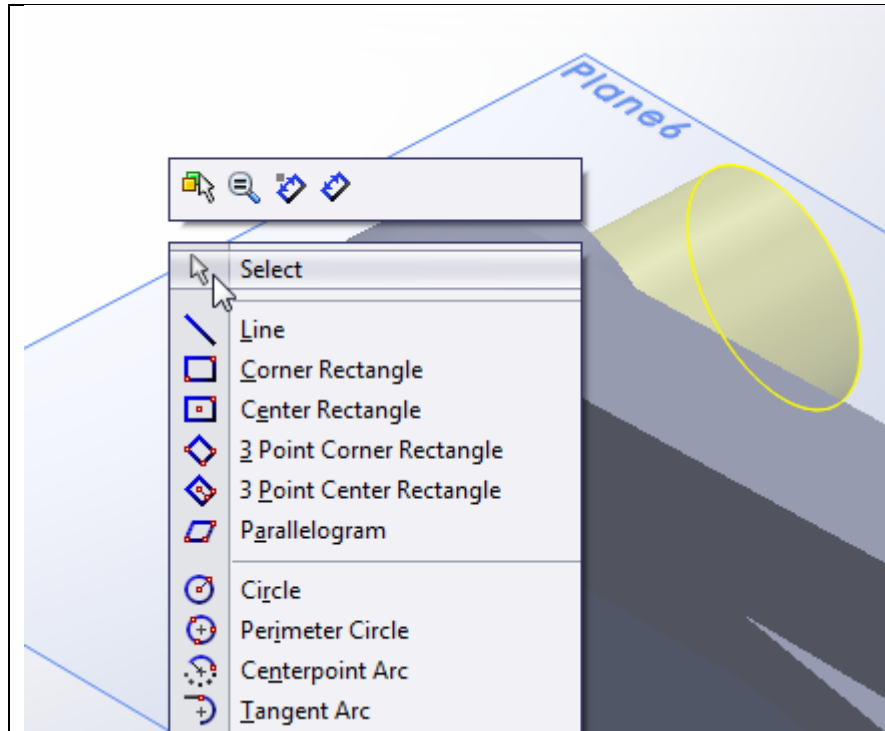


Click **3D Sketch**.

Click on the face of revolved feature as shown.

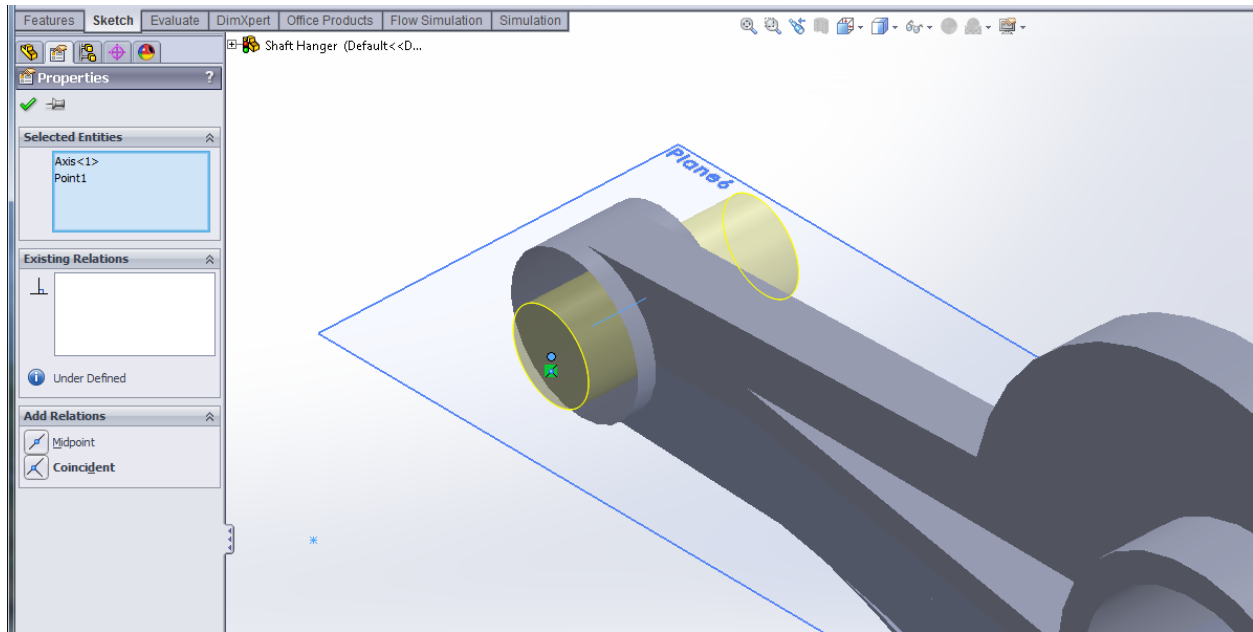


When point tool is active, wherever you pick, a sketch point will be created.




Right-click the graphics area and pick **Select**.

Hold the **CTRL** key down. Pick the center of the hole and pick the nearby axis.

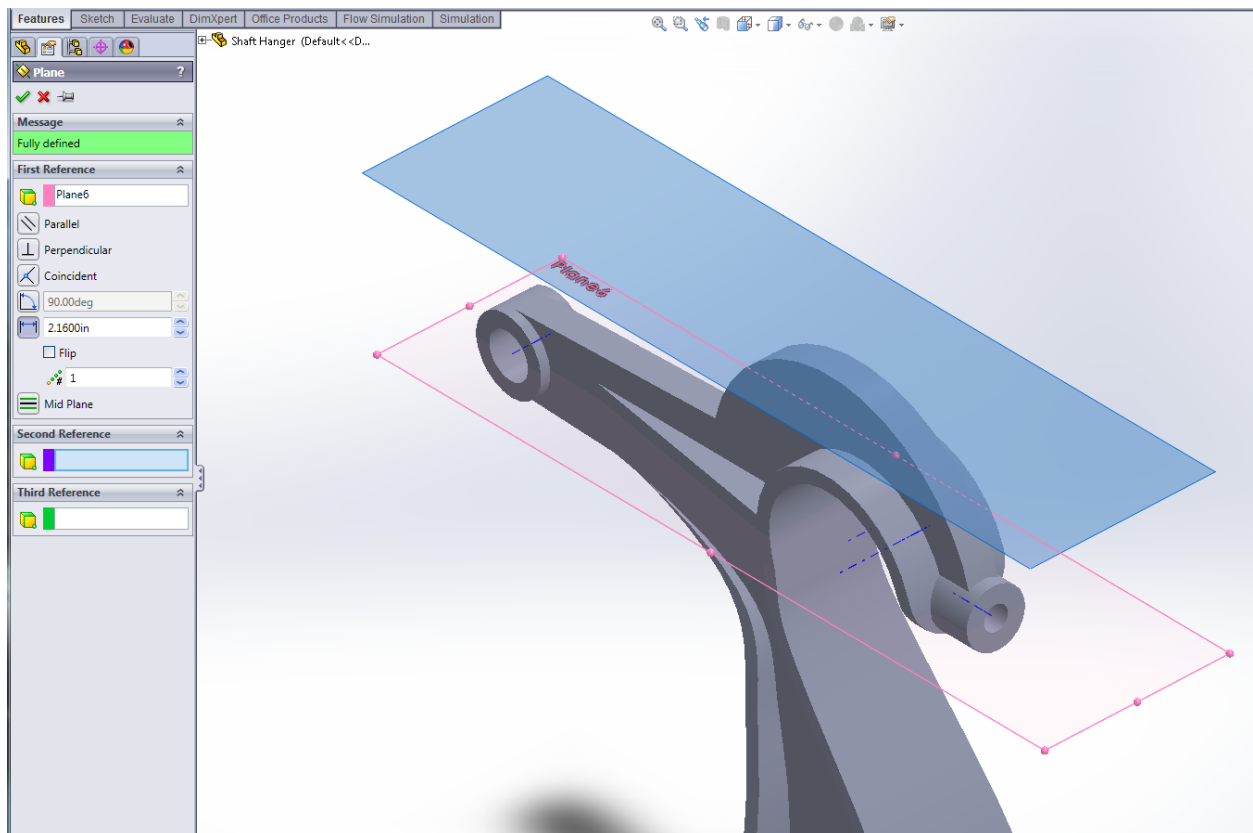


Select **Coincident**.


Click  twice.

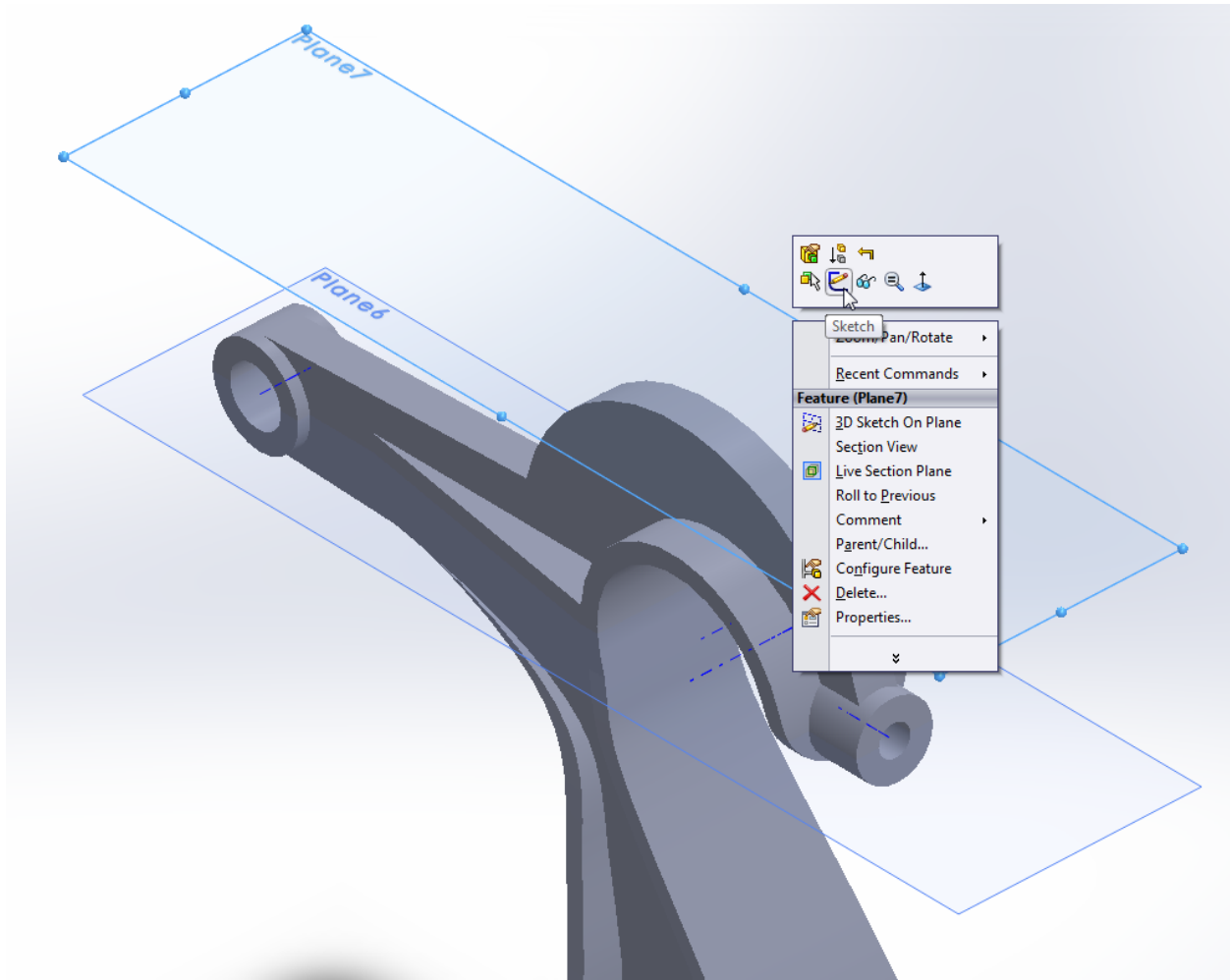
Pick the **Plane6** and hold the **CTRL** key down. Click and drag the boundary of the **Plane6** upward in the graphics area.

Set the offset distance of **2.16** inches.



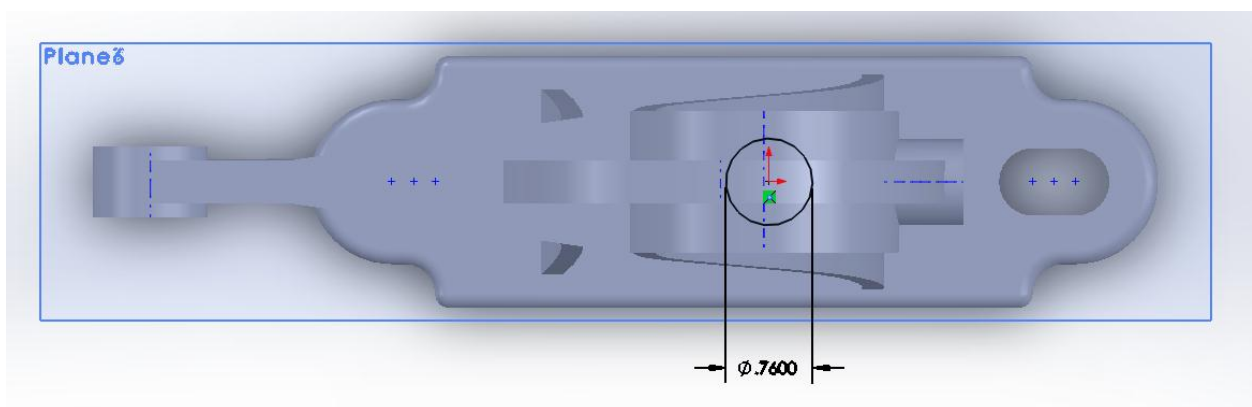
Click .


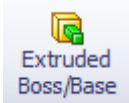
Right-click **Plane7** you just created and select **Sketch** .



Set the display to **Top** view .

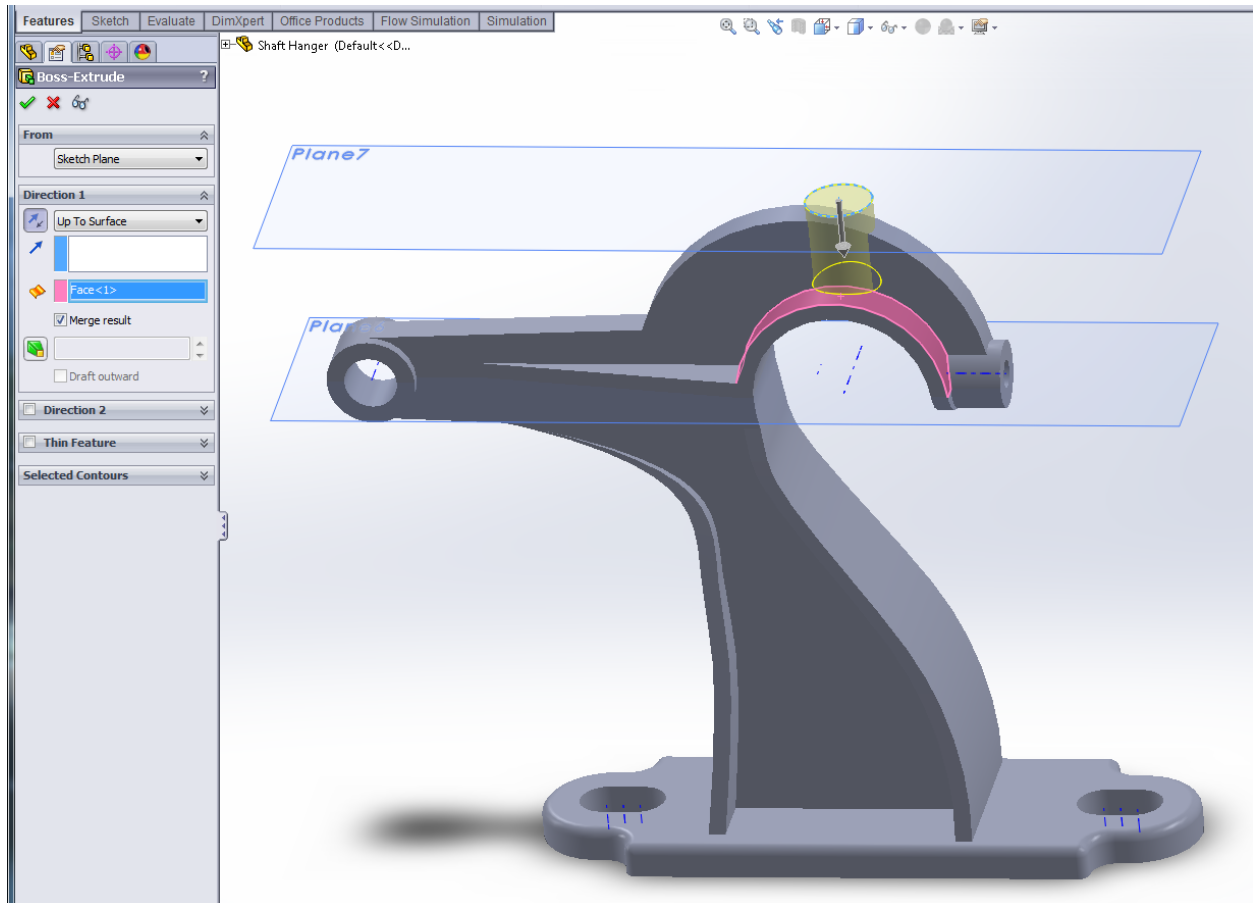
Draw a circle centered at the origin. The diameter of the circle is **0.76** inch.



Click  to exit the sketch mode. Select **Features** tab and click .

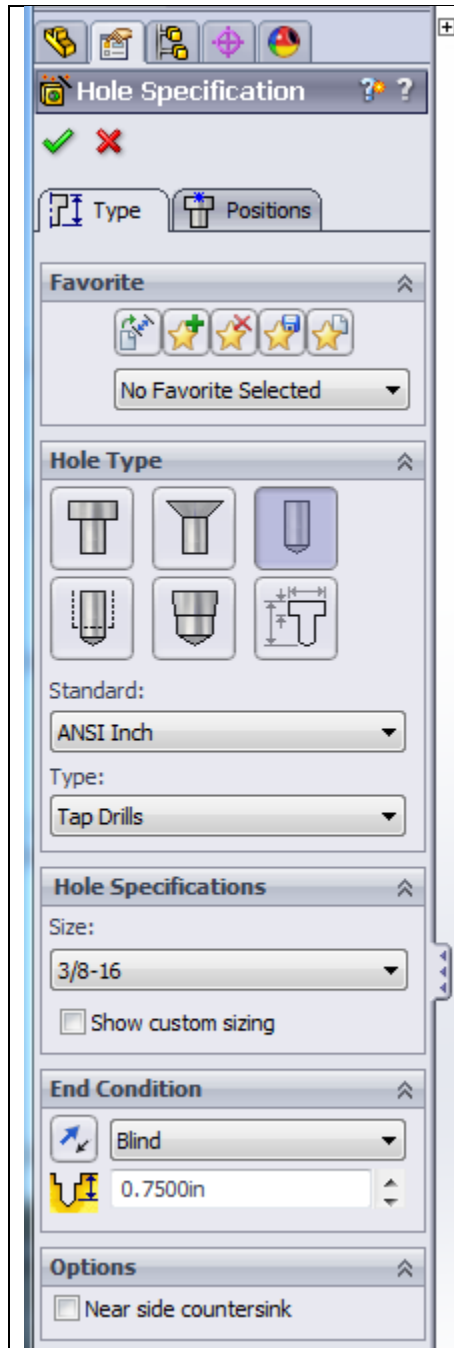
Select **Up To Surface** for **Direction 1**.

Pick the surface as illustrated below – show in pink color.



Click .

Click **Hole Wizard** . The **Hole Specification PropertyManager** is displayed.



Select **ANSI Inch** for **Standard**.

Be sure **Hole**  is selected.

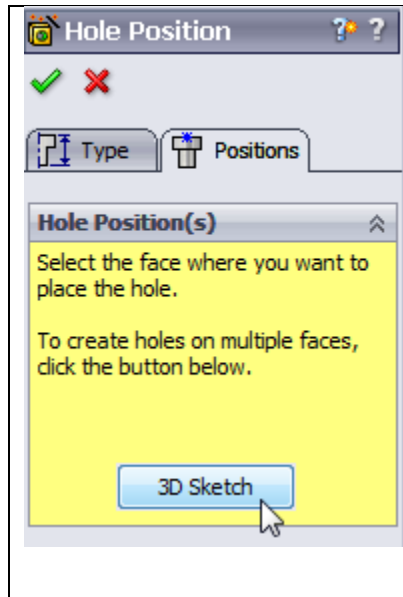
Select **Tap Drills** for **Type** of the hole.

Select **3/8-16** for **Size**.

Select **Blind** for **End Condition**.

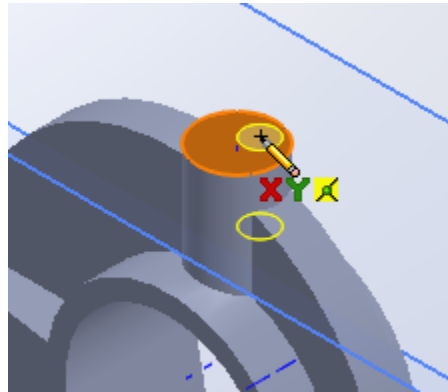
Enter **0.75** inch for the depth.

Click the **Position** tab.



Click **3D Sketch**.

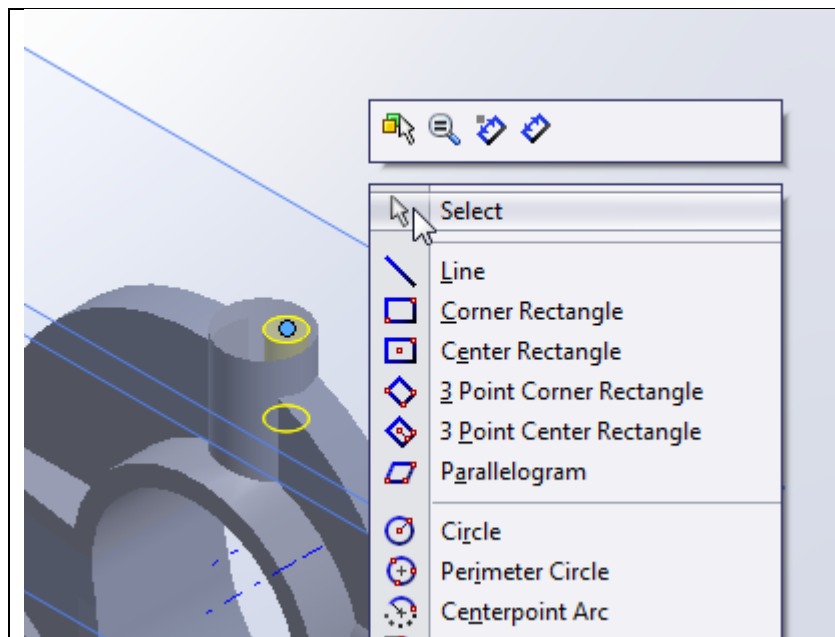
Click on the face of cylindrical feature as shown. The exact location is not important.



When point tool

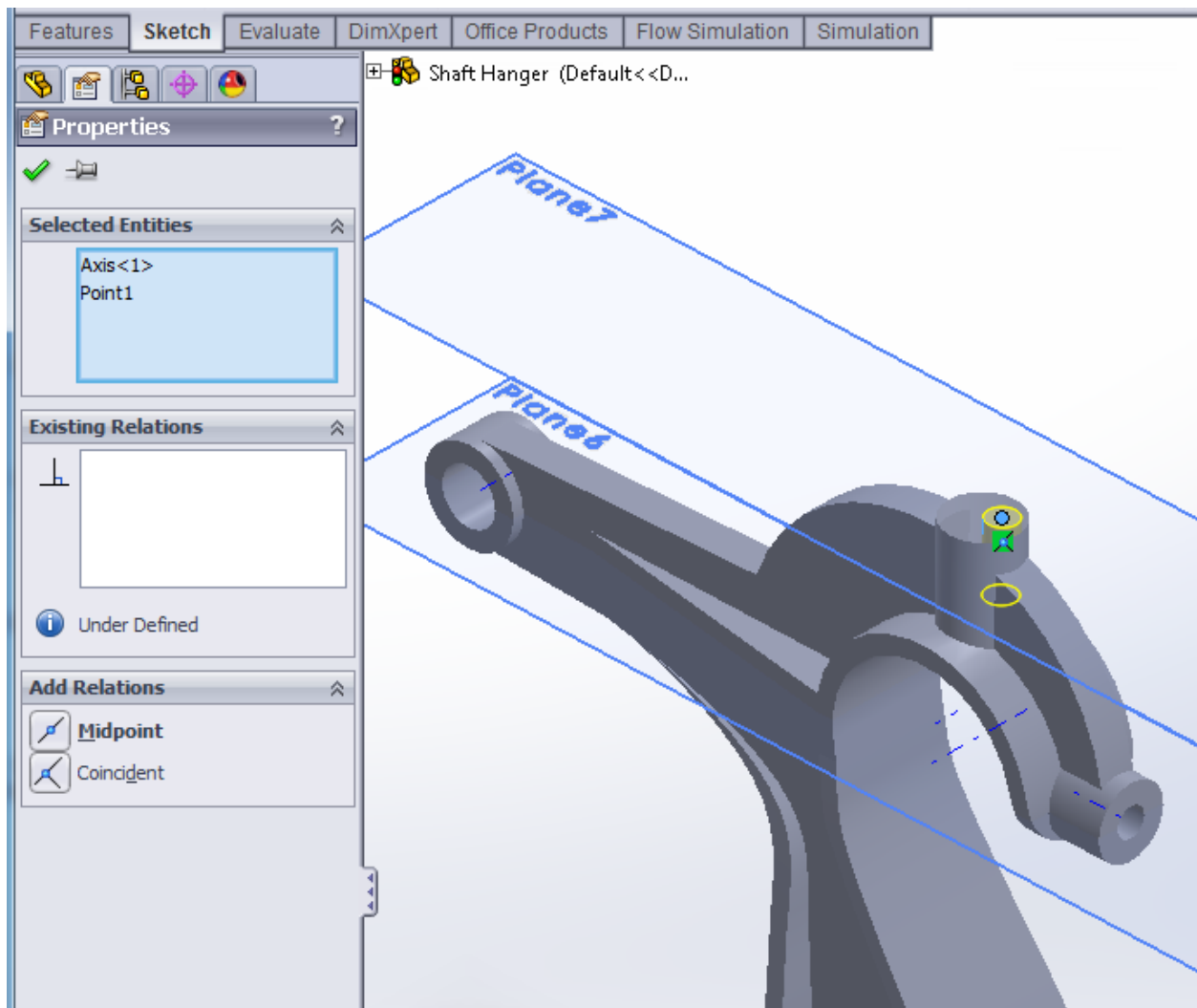


is active, wherever you pick, a sketch point will be created.



Right-click the graphics area and pick **Select**.

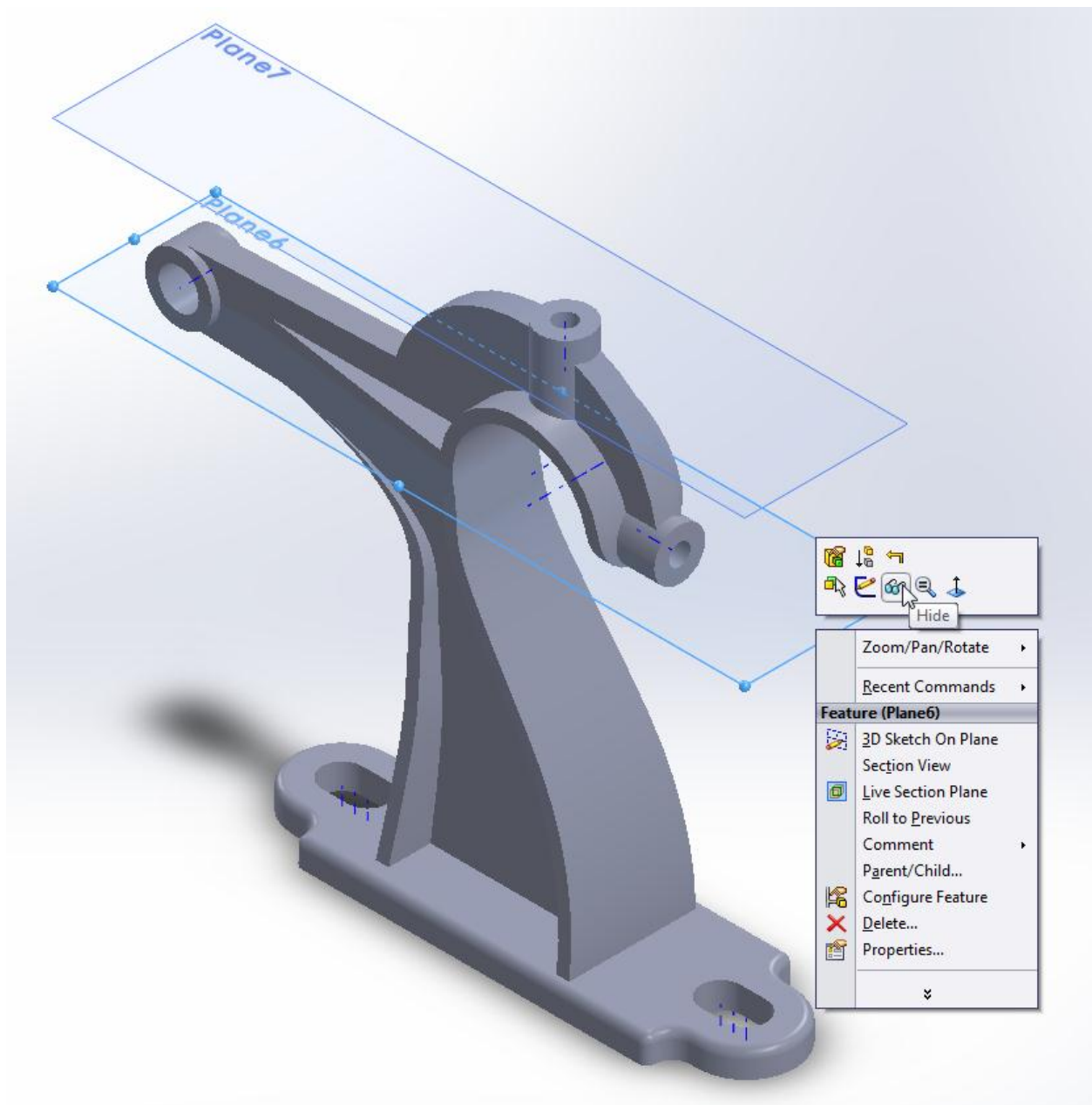
Pick the center of the hole and hold **CTRL**-key and pick the nearby axis.

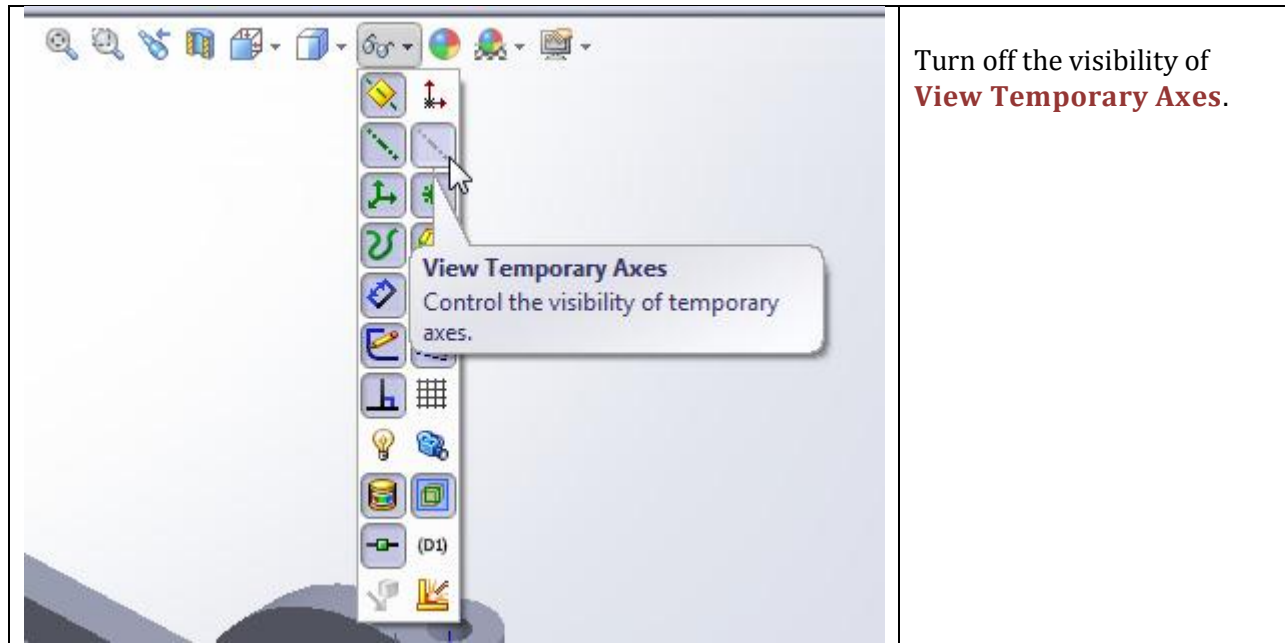


Select **Coincident**.

Click  twice.

Select **Plane6** and **Plane7** and right click the graphics area – select **Hide**.

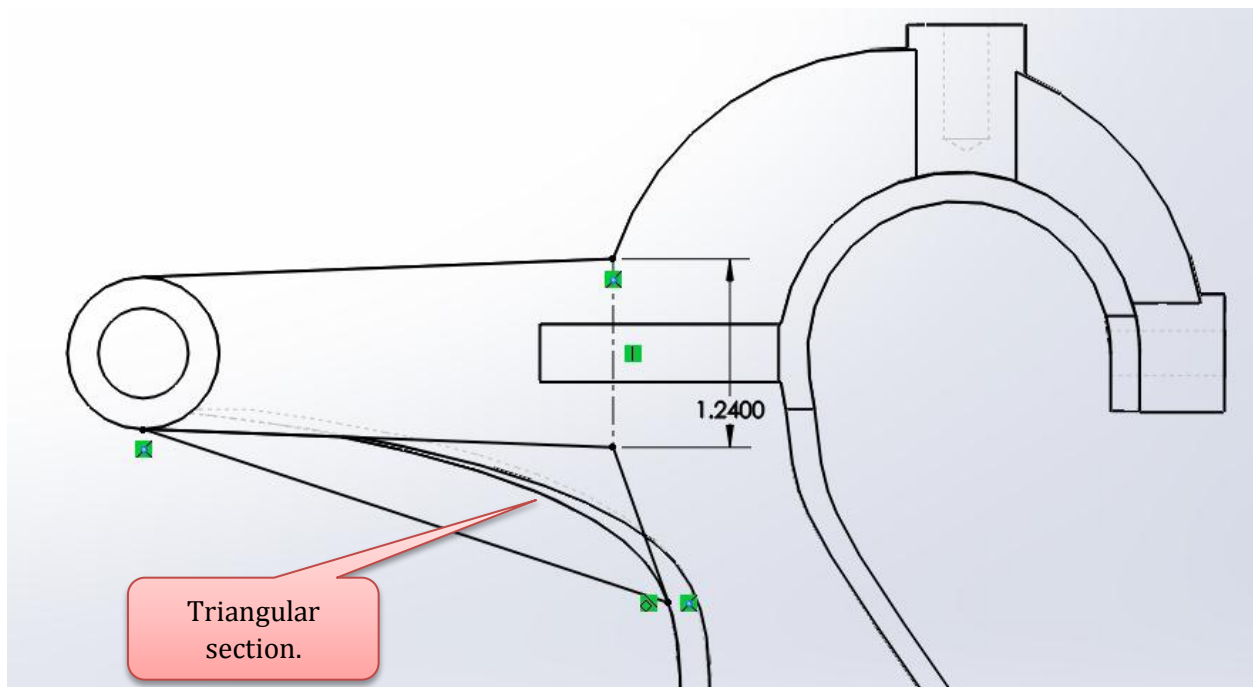






Pick **Front Plane** from **FeatureManager** design tree, right click it in the graphics area and select **Sketch**.

Set the display to **Front** view. For clarity, set the display to **Hidden Lines Visible** view.

Draw and dimension a vertical centerline as illustrated and then sketch the following triangular section. Note the constraints (**Coincident**, **Tangent** and **Vertical**) indicated by green.

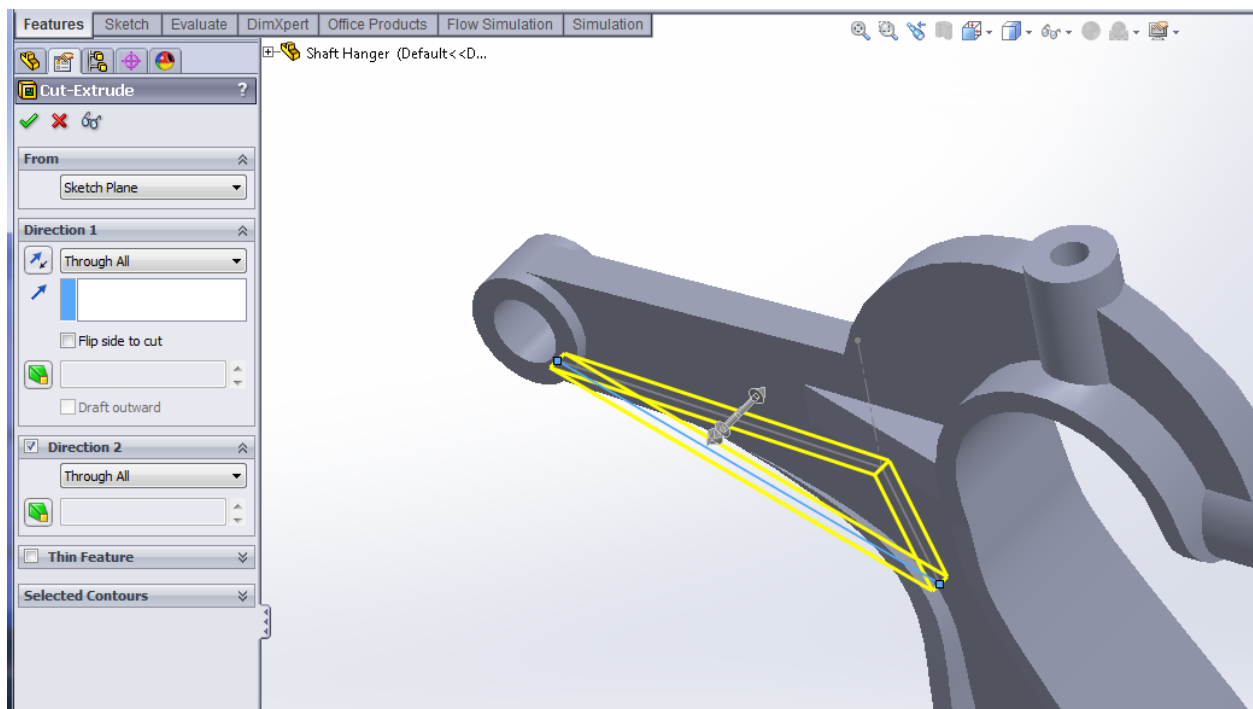


Click  and click  to exit the sketch mode.

Select **Features** tab and click **Extruded Cut** . Pick the section.

Select **Through All** for **Direction 1**.

Select **Through All** for **Direction 2**.



Click .


The front face as indicated by blue color is perfect.

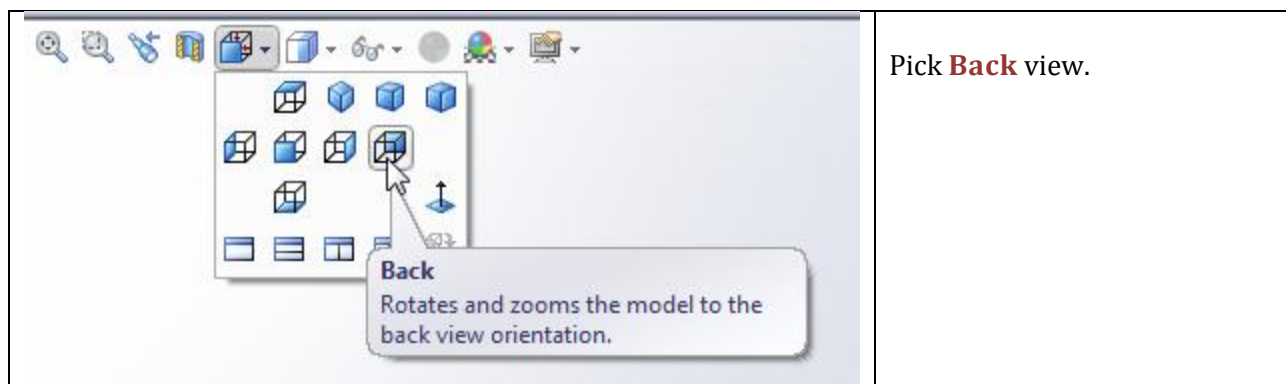
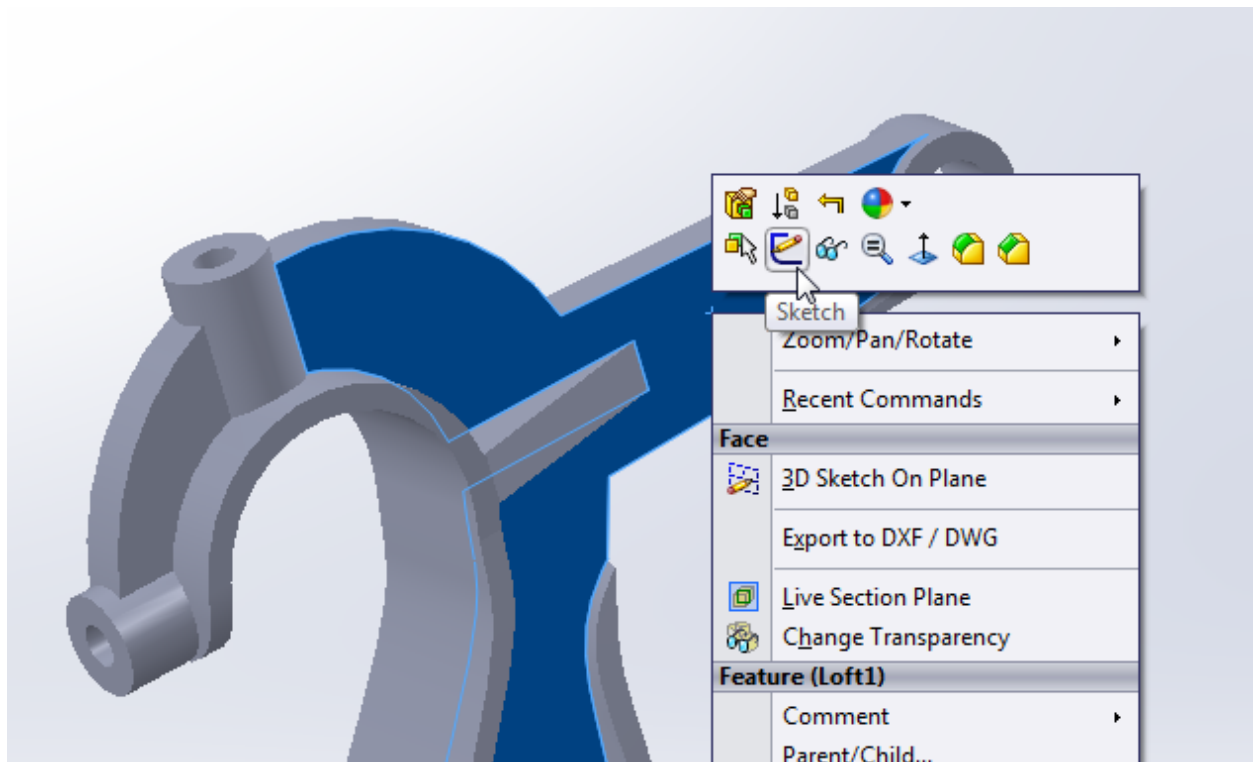


But there is excess material on the rear flat face (blue color).



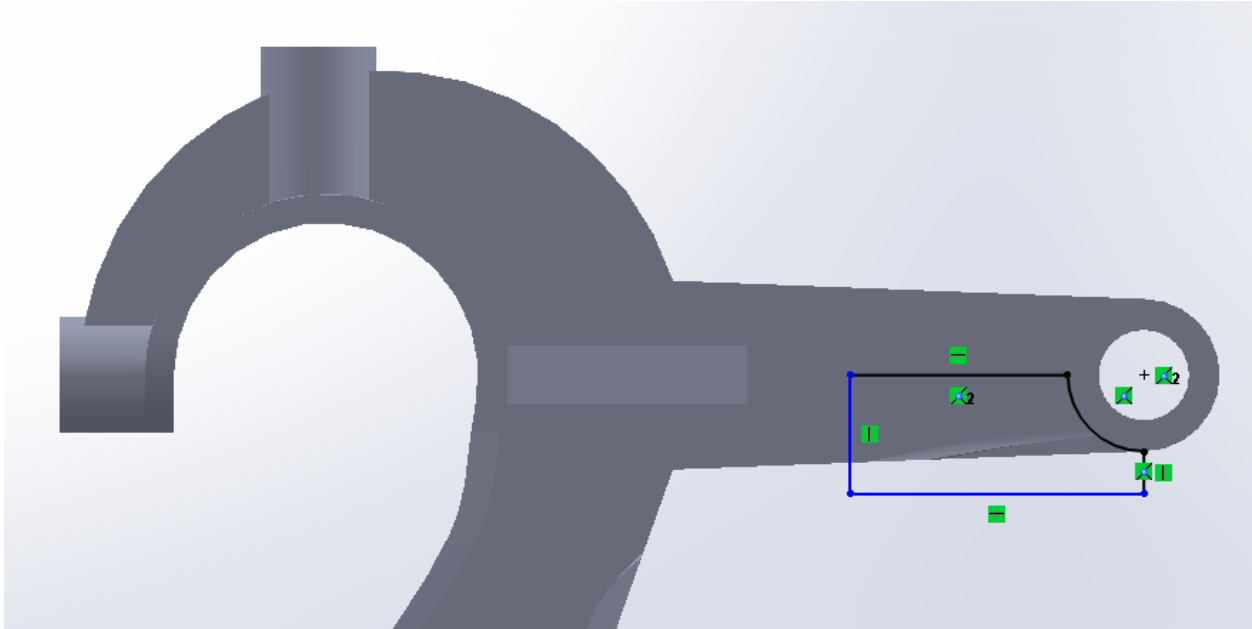
This area of excess material must be clean up using **Extruded Cut**.



Right click the face and select **Sketch** .



Sketch the following section. Note that the size of the section must cover the area of excess material that has to be removed – dimensions are not important.

Make sure the arc of the section and outer edge of the small cylinder are coincident.

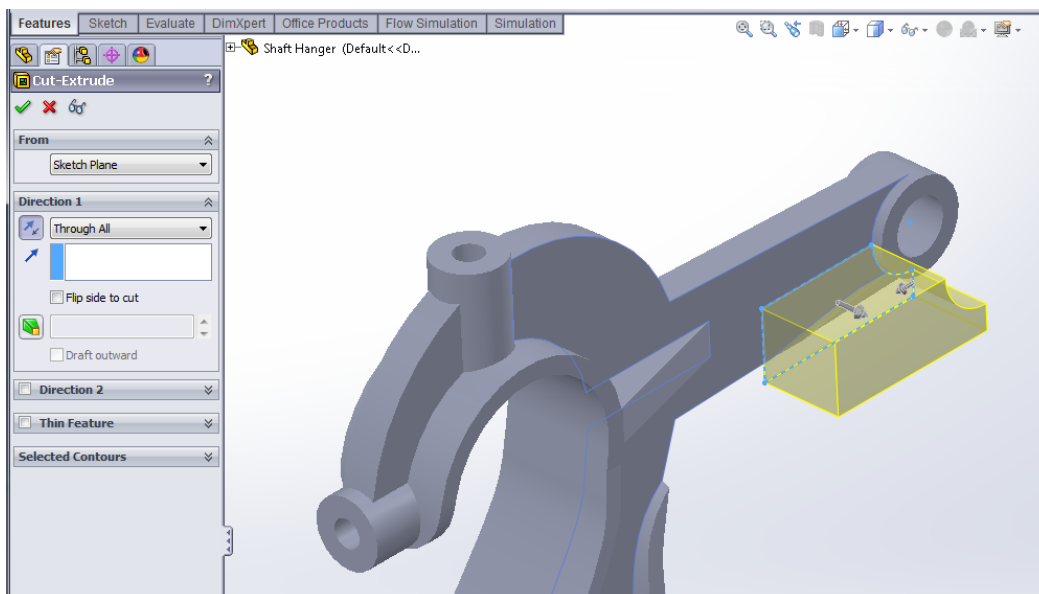


Click  and click  to exit the sketch mode.

Select **Features** tab and click **Extruded Cut**

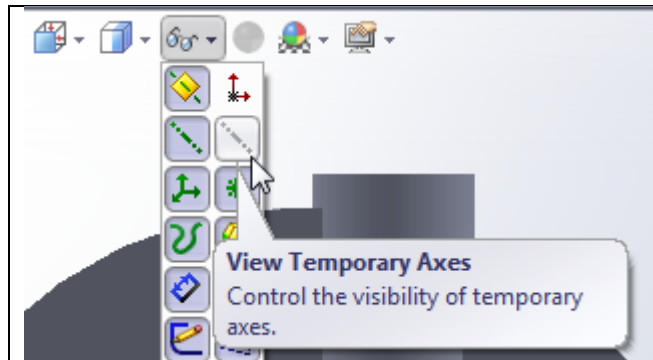
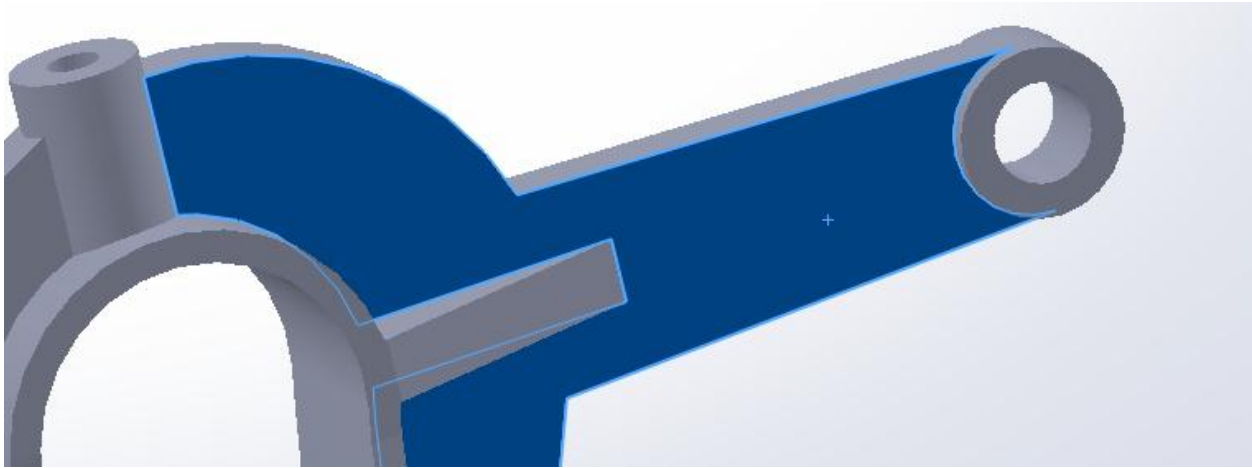


Select **Through All** for **Direction 1**. Click **Reverse Direction** button



Click .

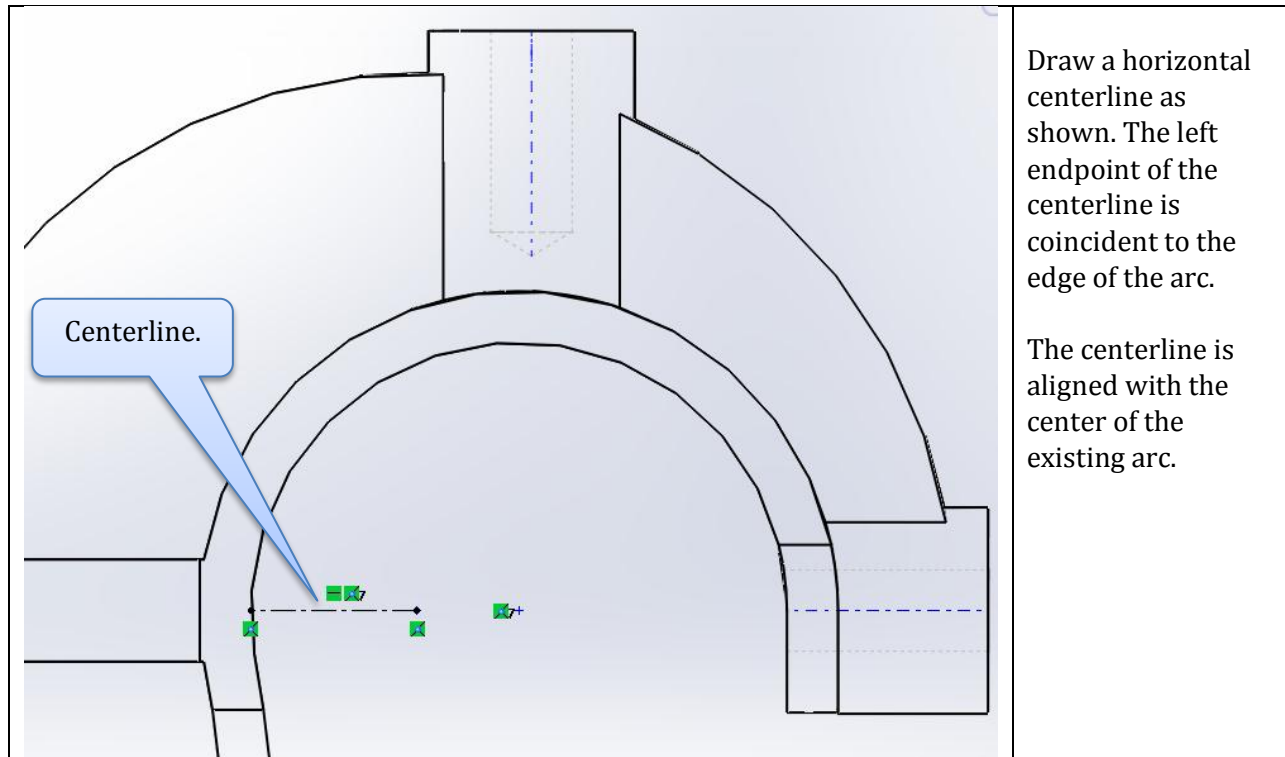
The excess material has been removed.




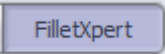
If your **Temporary Axes** is off, turn it on.

Pick **Front Plane** from **FeatureManager** design tree, right click it in the graphics area and select **Sketch** .

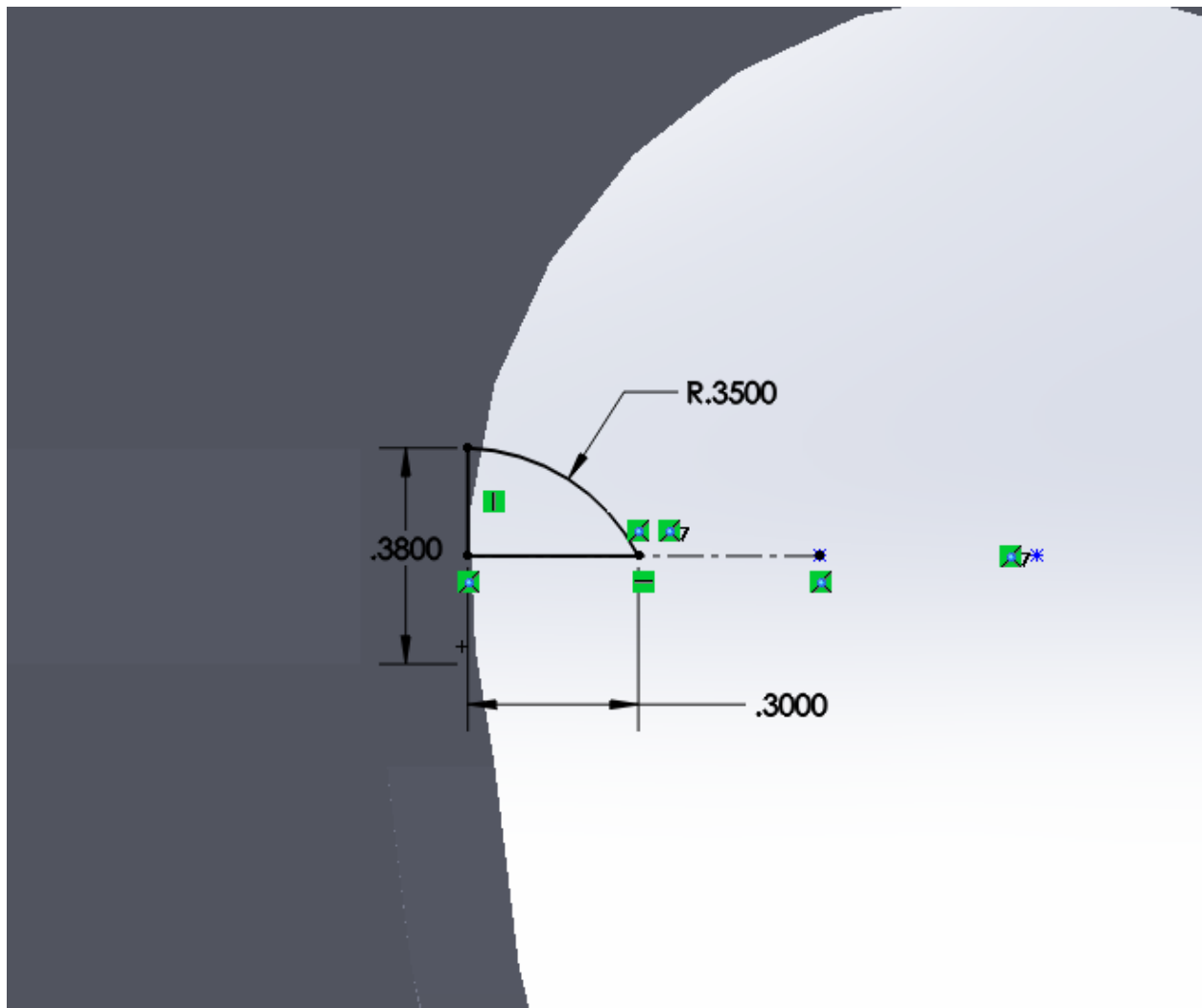
Set the display to **Front** view .



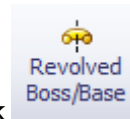
Sketch the following section to be revolved.

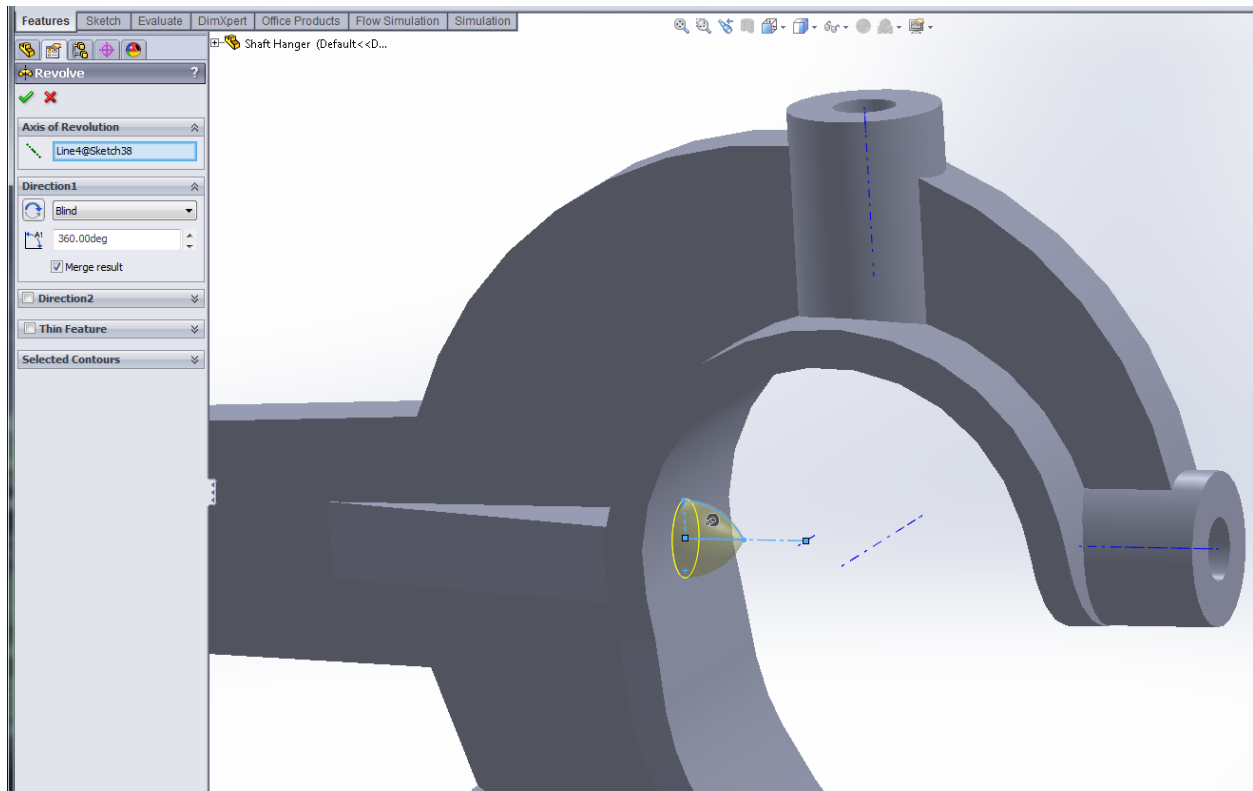
Click  to add rounds of **0.12** inch radius on your own. You might want to use **FilletXpert** by selecting  tab.

The resulted Shaft Support model is shown below.

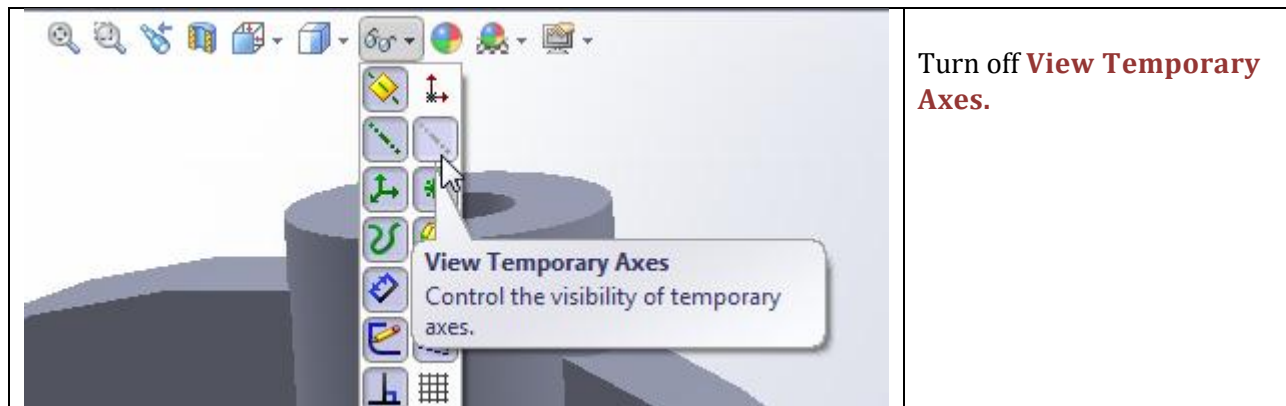


Click to exit the sketch mode. Select **Features** tab and click

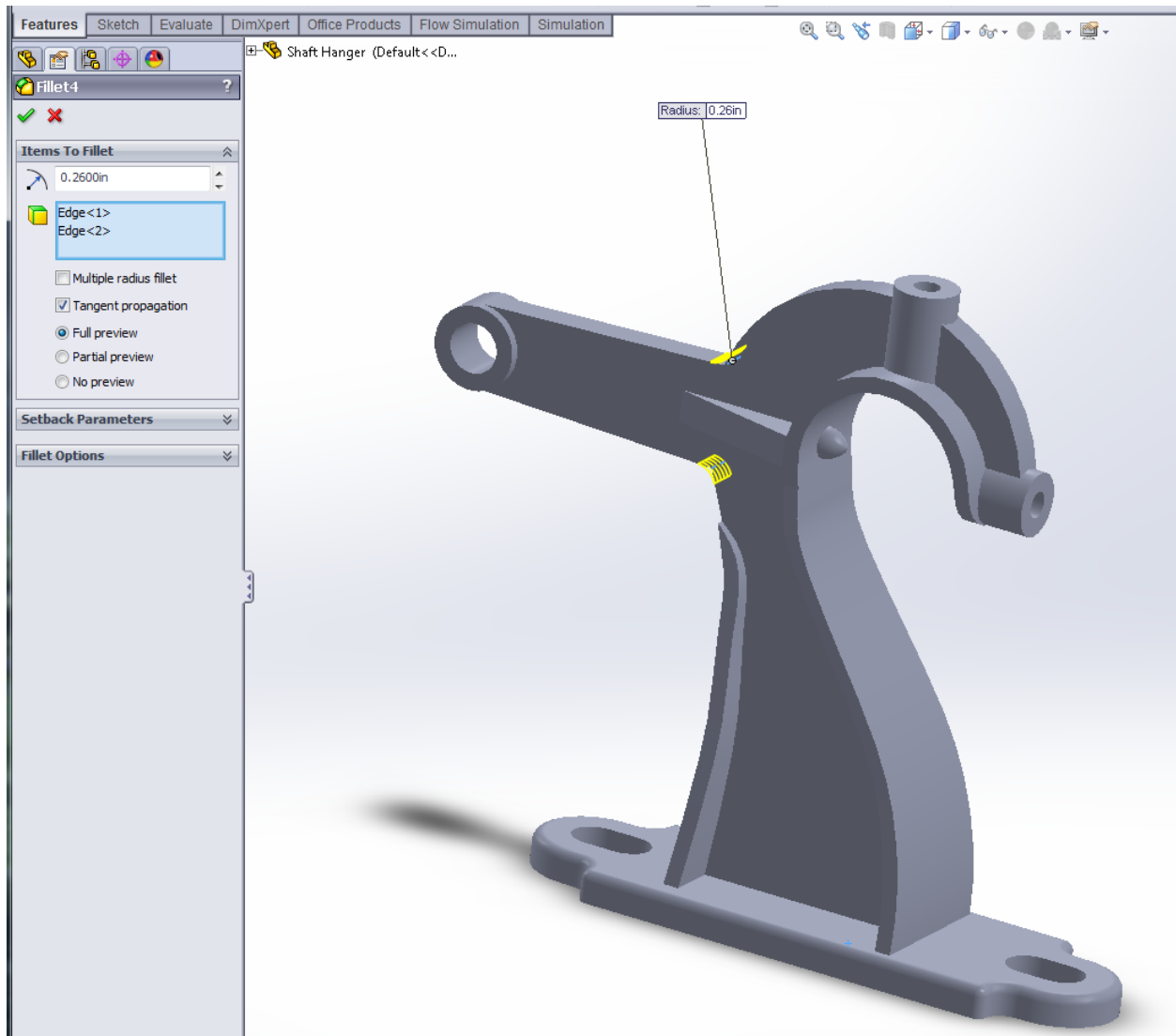




Click  .



Click  and create **0.26** inch fillets at locations below.



The last step is to create fillets and rounds with **0.12** inch radius on your own. You might want to change the radius of the fillets e.g. **0.08** inch especially in thinner area if failure occurs.

	<p>Hint: Use FilletXpert to manage, organize, and reorder constant radius fillets for you.</p> <p>In addition to pick edges, you might want to pick face to be filleted.</p> <p>The FilletXpert automatically calls the FeatureXpert when it has trouble placing a fillet on the specified geometry.</p>
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The resulted Shaft Hanger model is shown below.

