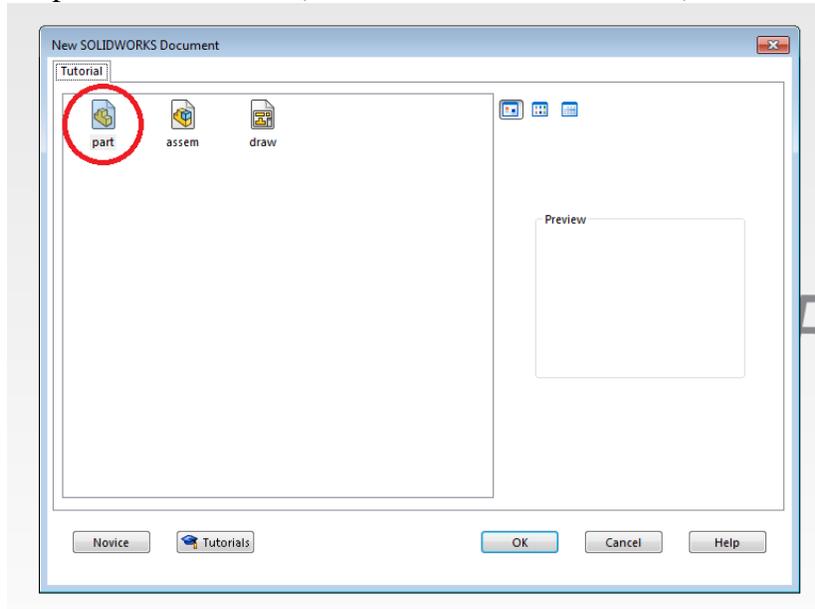


CHEG 3128

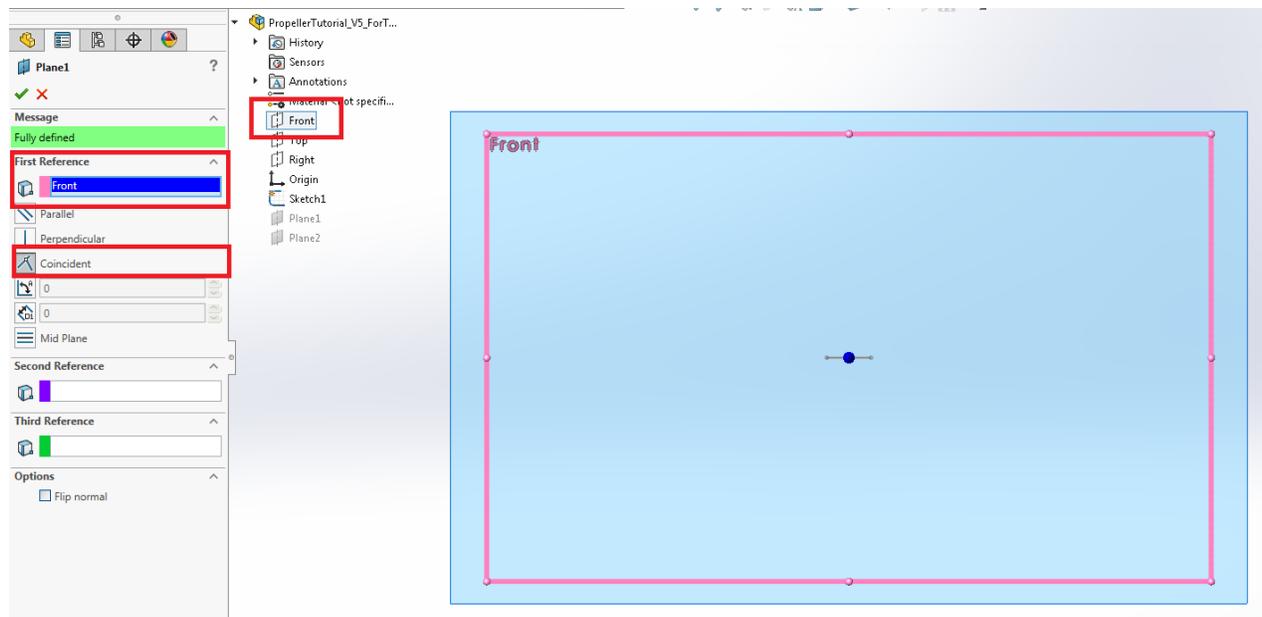
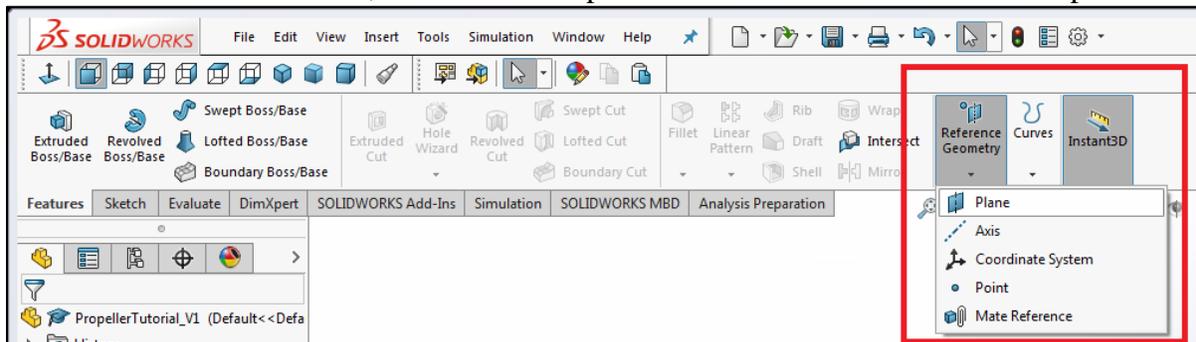
Solidworks Tutorial: Propeller Design

I. Creating the propeller blade

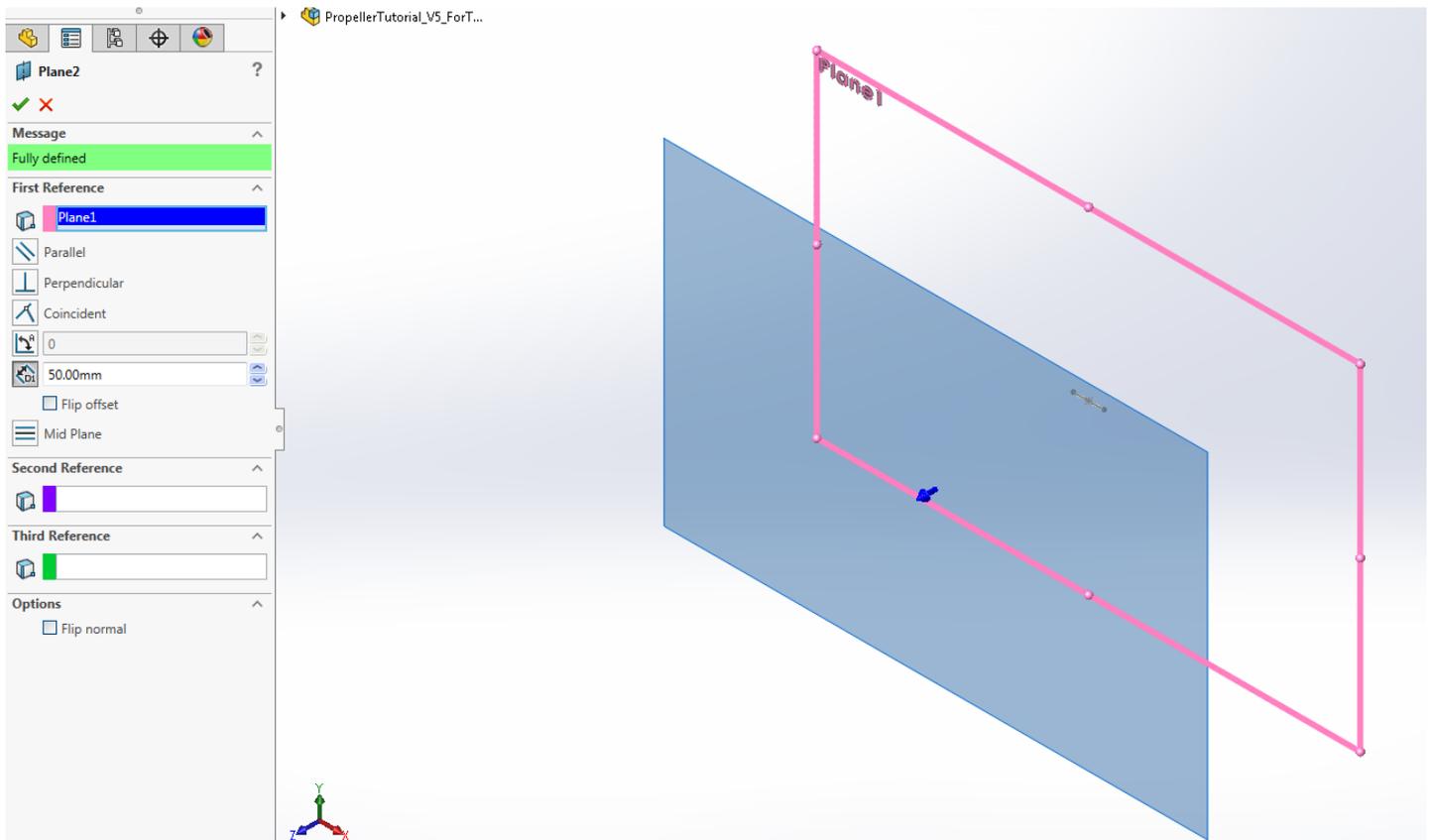
1. Open a new part in Solidworks (File → New → “Part” → OK)



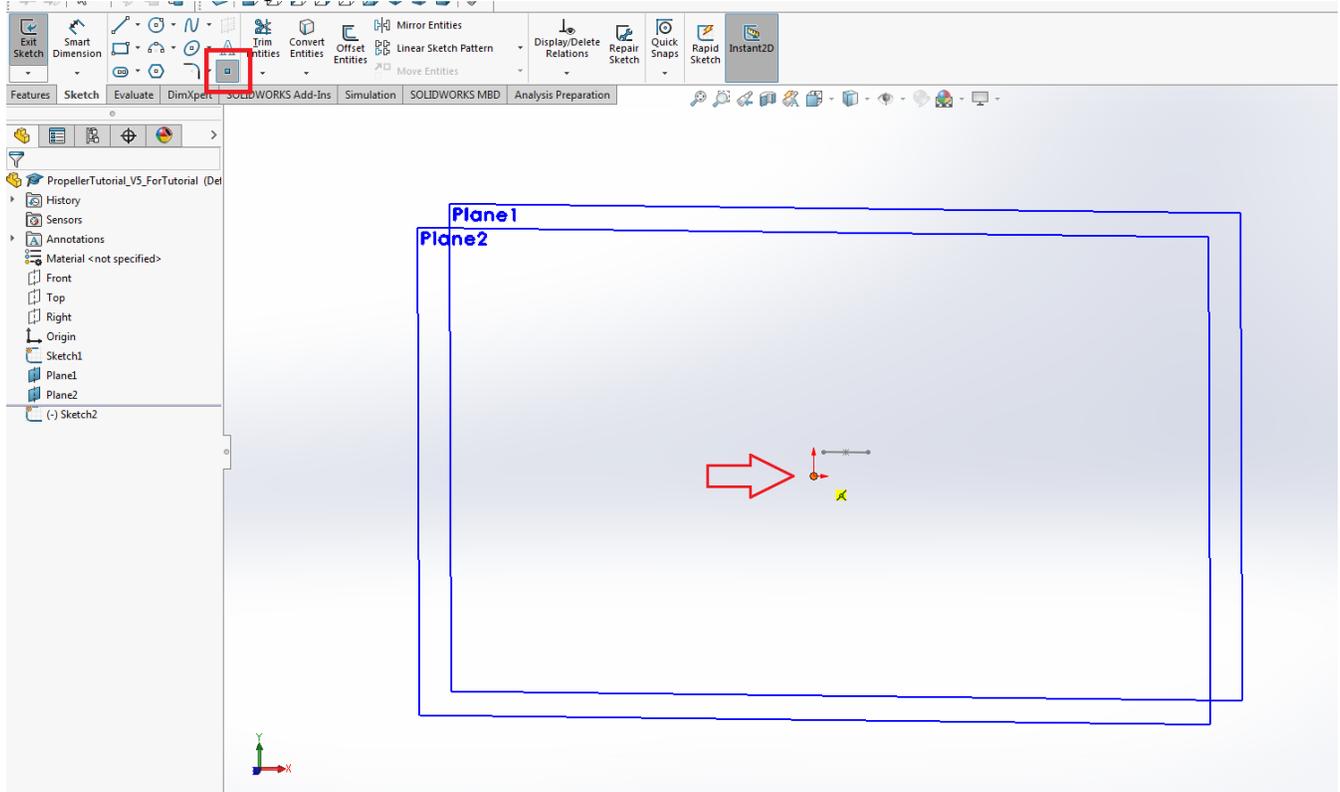
2. Under the Features tab, add a reference plane that is coincident with the front plane.



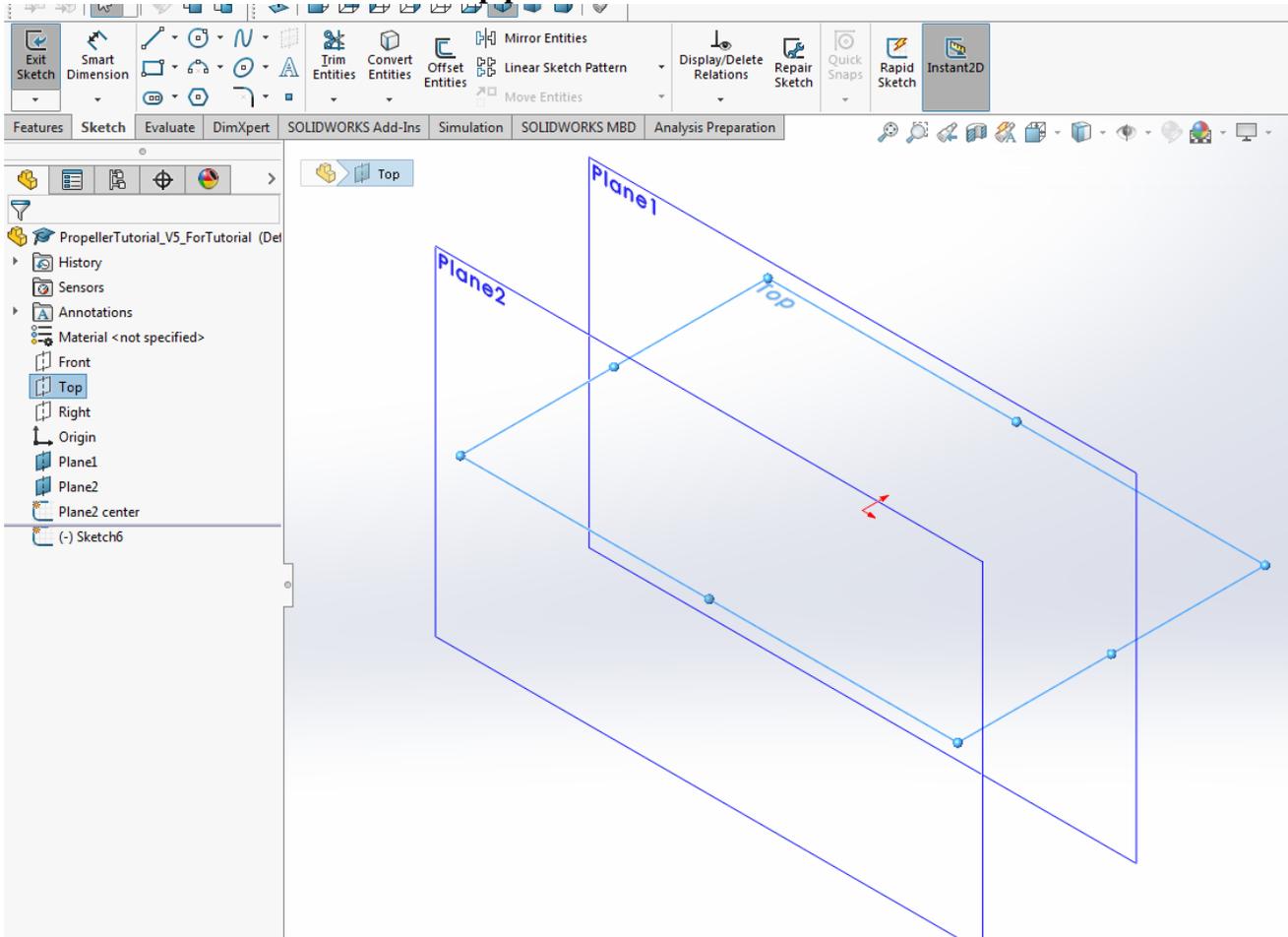
3. Add a second reference plane that is parallel to Plane 1 with 50mm between the two planes. This will be the length of the propeller blade. Isometric View gives a good perspective.



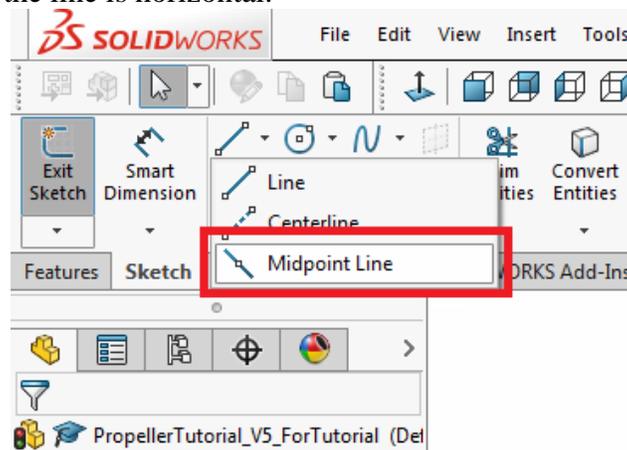
4. Sketch a point at the center of Plane 2, then Exit Sketch.

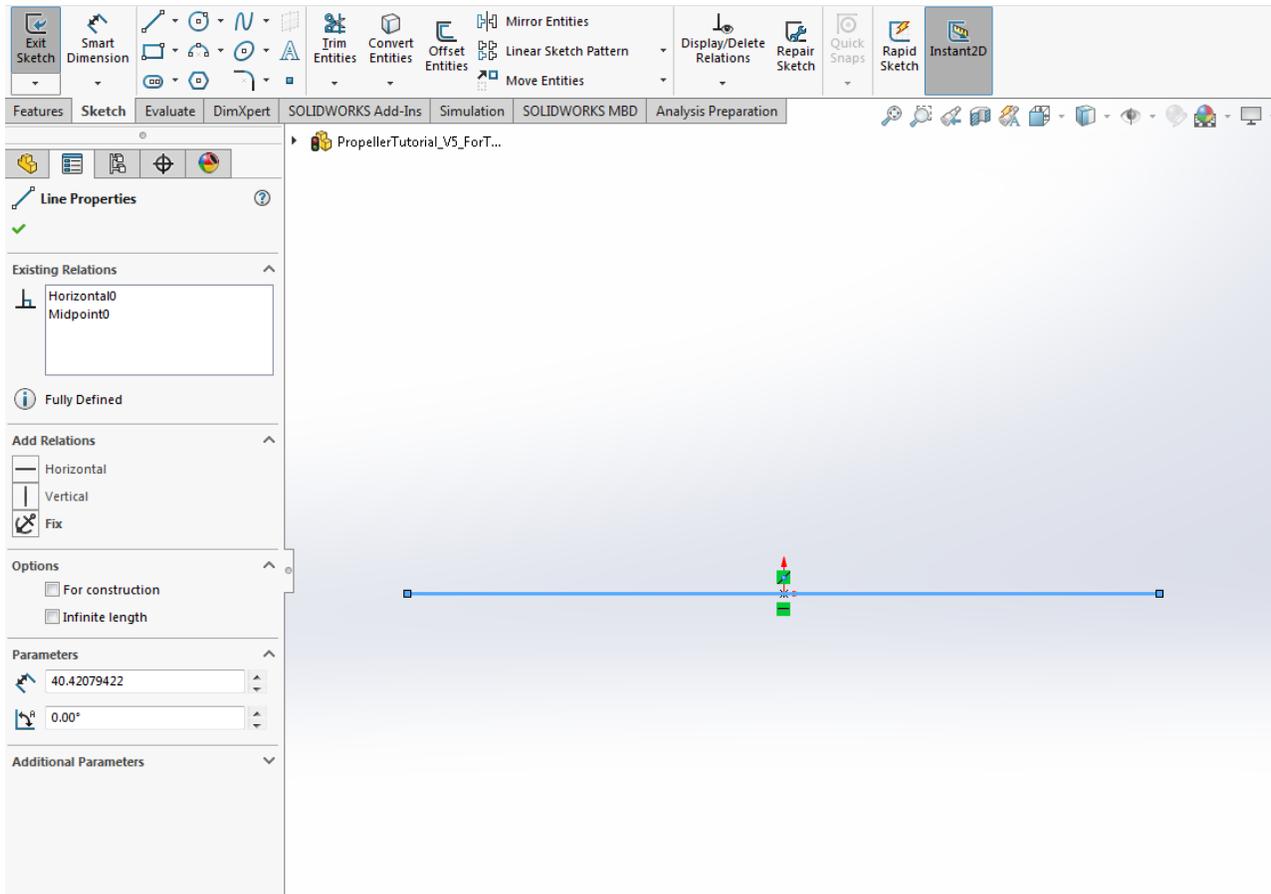


5. Start a new sketch on the top plane

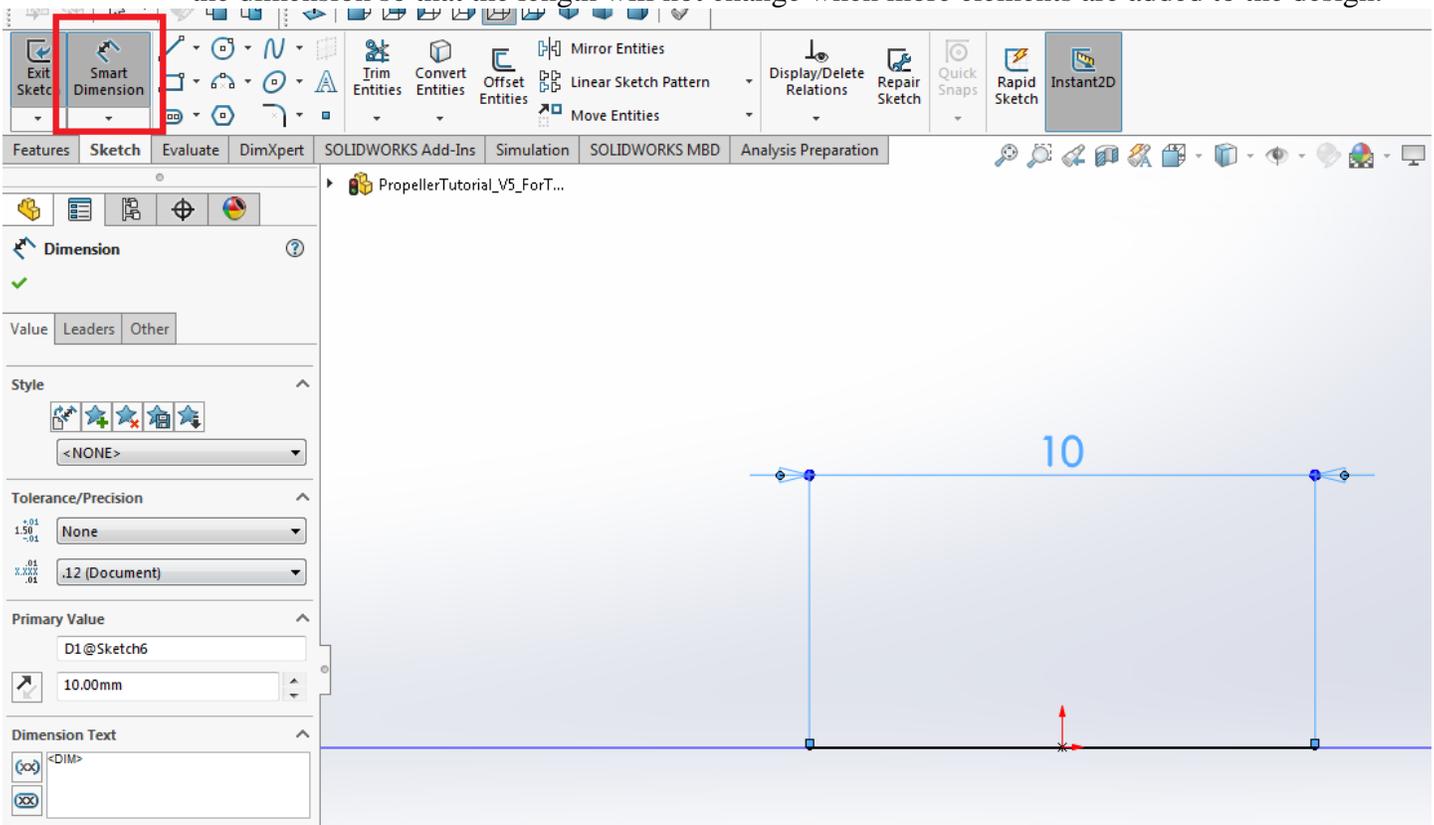


6. Sketch a line with the center at the origin. To do this, select Midpoint Line under the Line dropdown menu. Then, select the origin and move the cursor to the right and click to create a line. Make sure the line is horizontal.

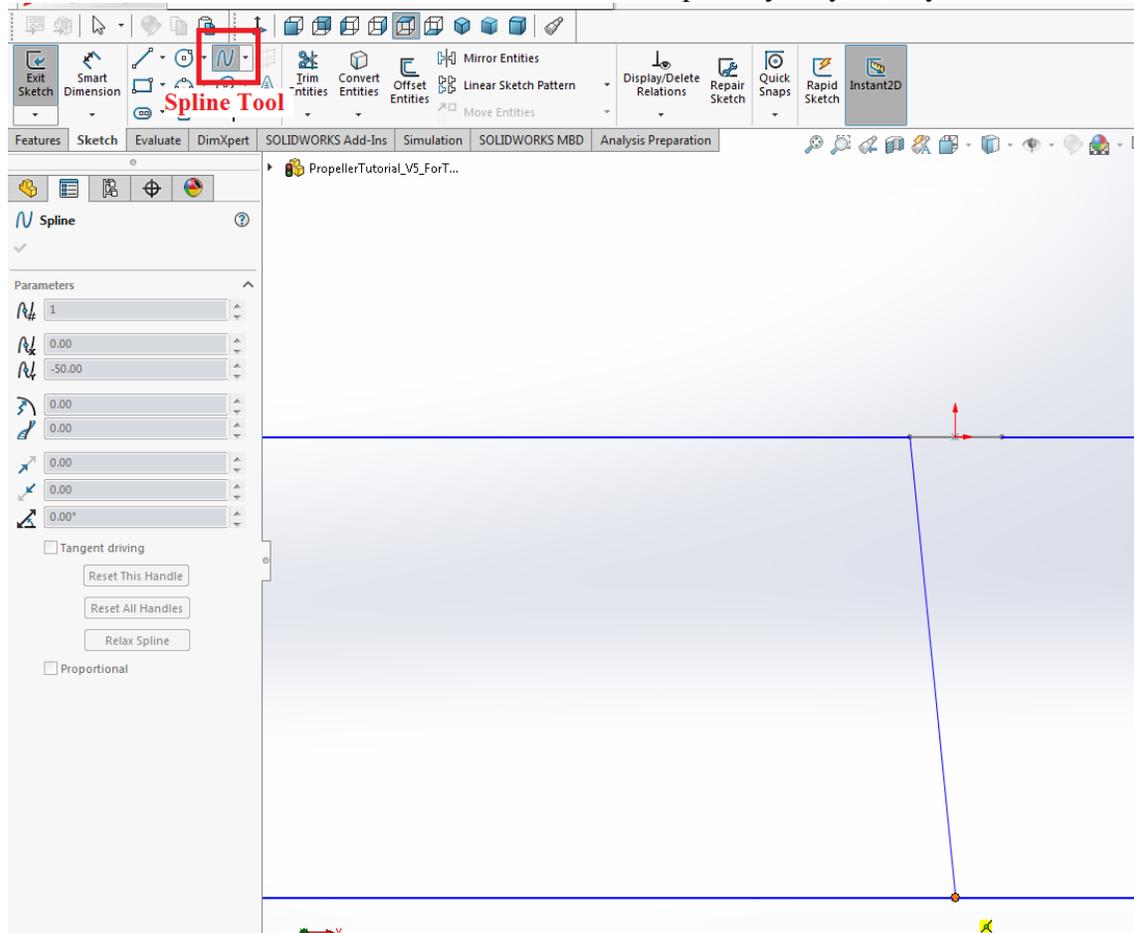




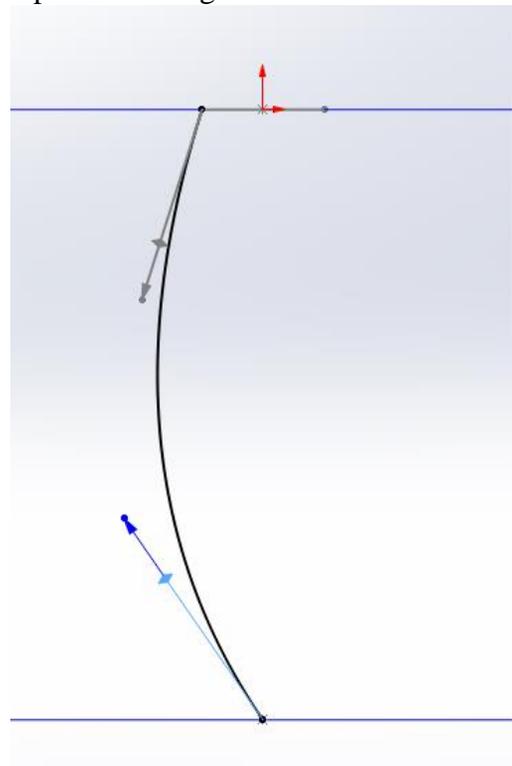
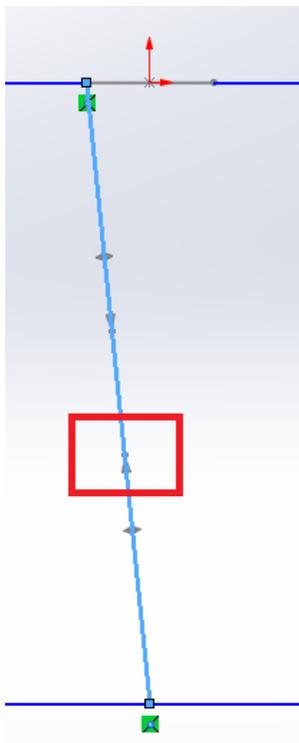
- Using the Smart Dimension tool, set the line length to be 10mm. Smart Dimension will lock in the dimension so that the length will not change when more elements are added to the design.



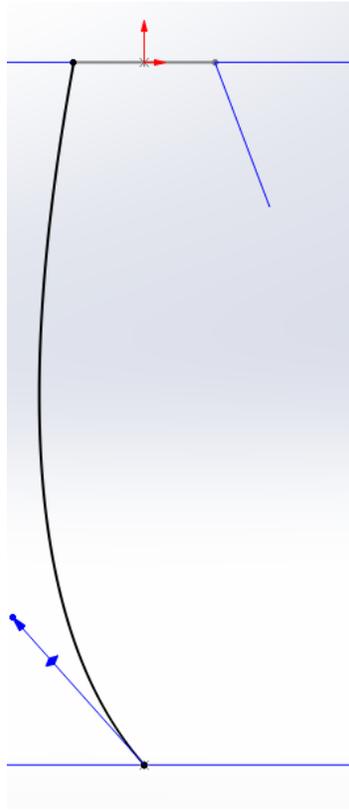
8. In the same sketch, draw a spline, starting at the leftmost point of the line and ending at the point drawn in the center of Plane 2. Then hit the “Escape” key on your keyboard.



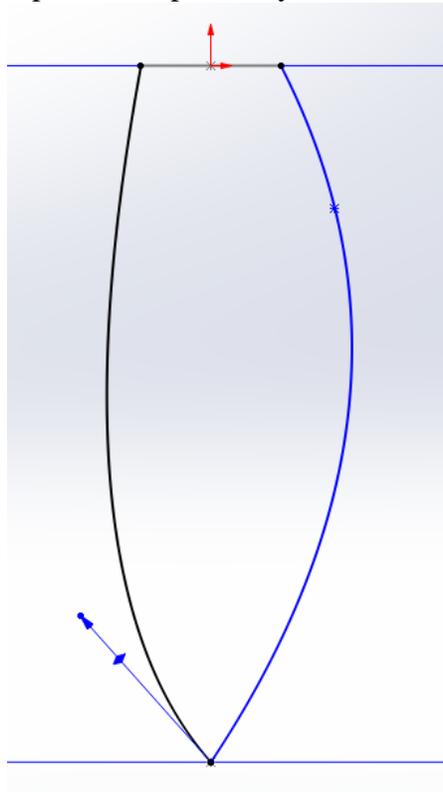
Select the spline, then select the bottom point of the spline and drag it down and to the left:



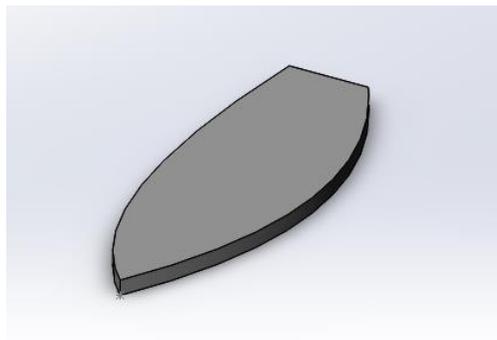
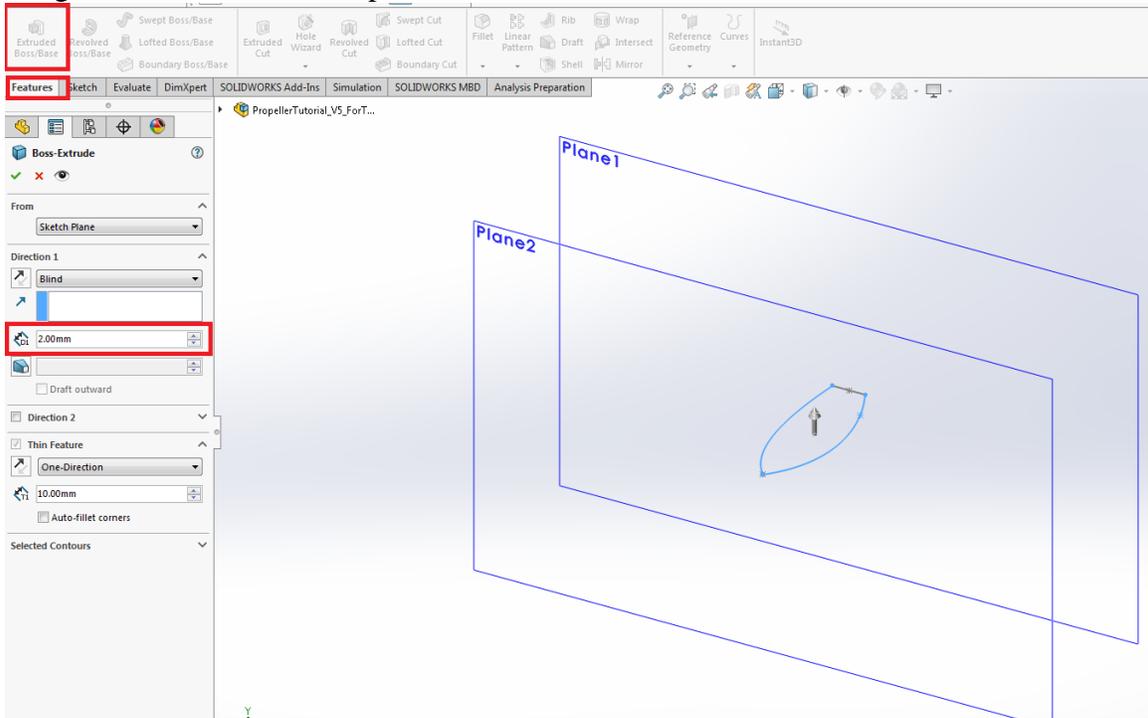
9. In the same sketch, create a second spline; there will be two segments in this spline. First, start at the rightmost point of the line and end this first segment roughly at the point shown in the picture below.



Second, end the spline at the point drawn in the center of Plane 2 then hit the “Escape” key. Feel free to manipulate the shape of this spline as you did with the first spline. Then, Exit Sketch.

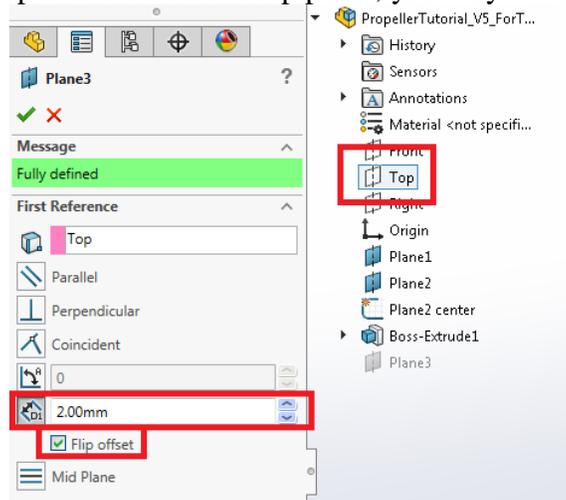


10. Under the “Features” tab, select “Extruded Boss/Base.” Set the depth to 2mm, then click on the green checkmark to complete the feature.

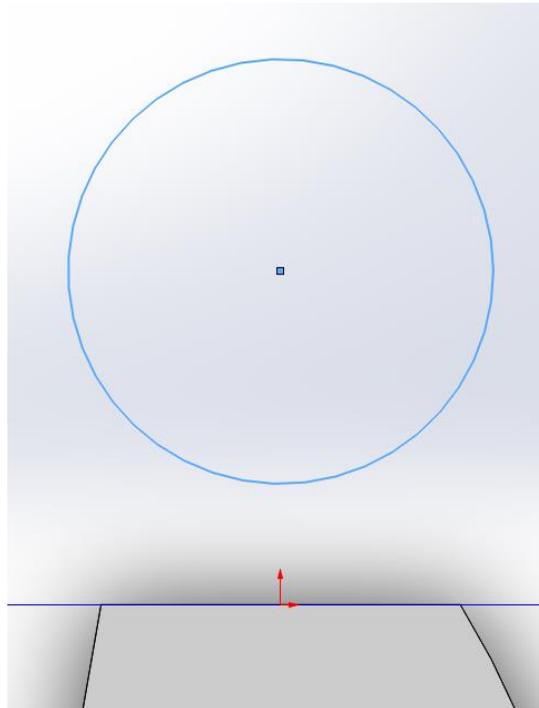
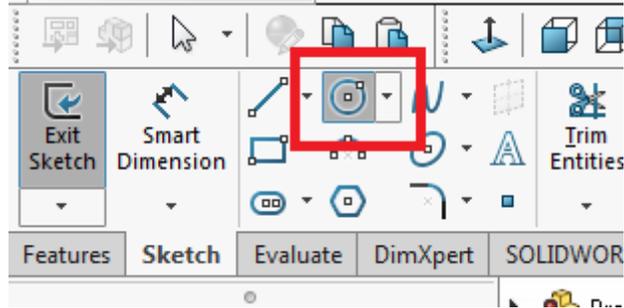


II. Creating the base

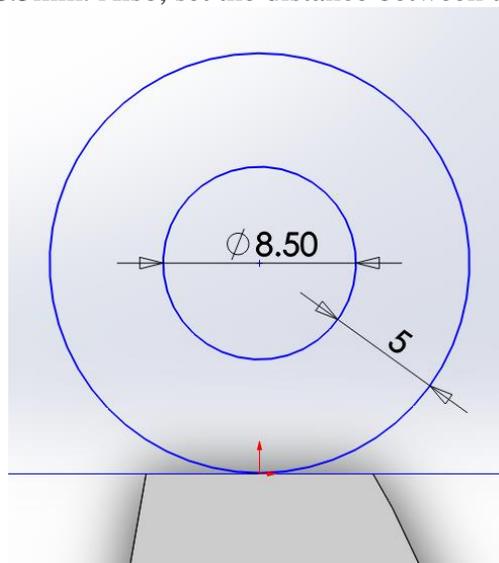
1. Add a new reference plane. The “First Reference” should be the top plane; set the depth to 2mm. Make sure the new plane is below the top plane; you may have to check the “Flip offset” box.



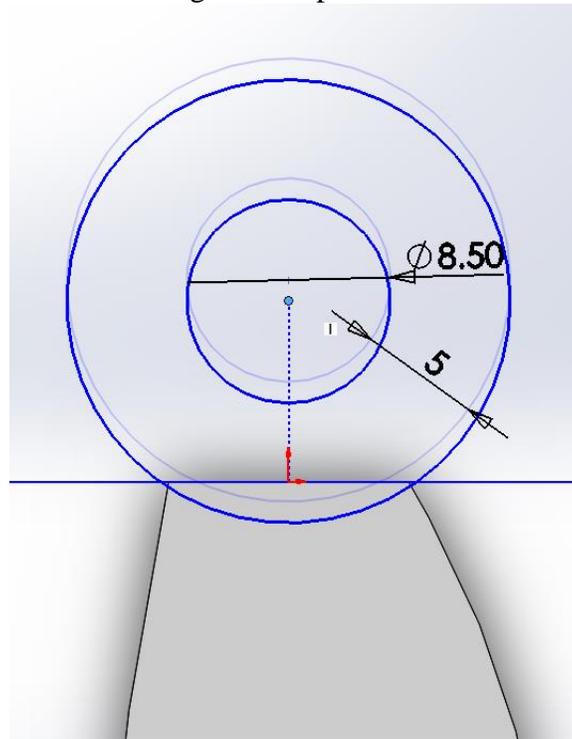
2. Create a new sketch on the new plane. Sketch a circle whose center is above the origin:



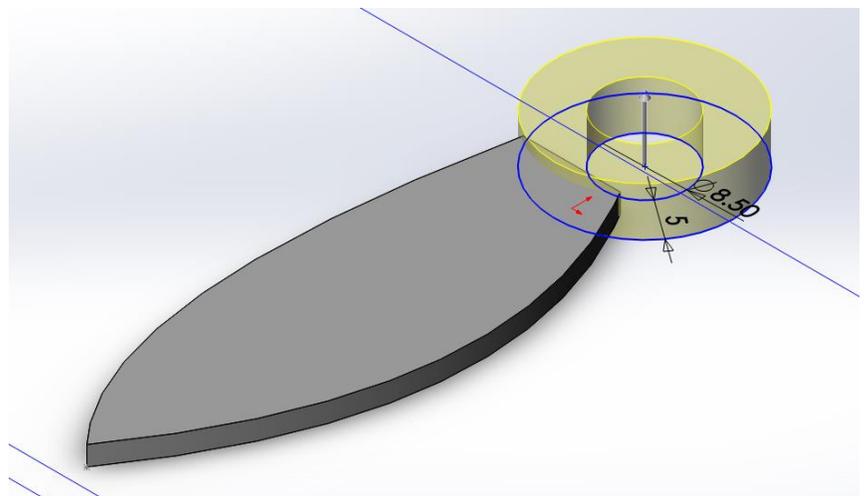
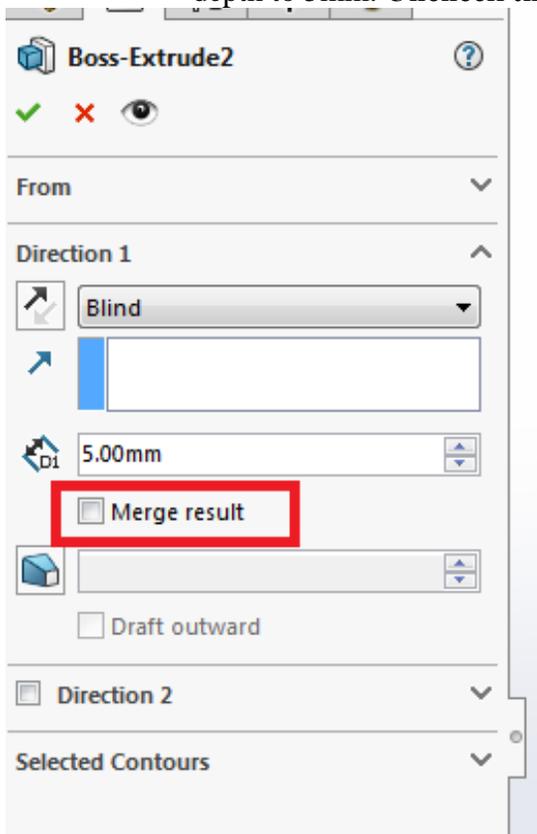
3. Draw a second circle that is concentric with the first. Use Smart Dimension to set the diameter of the smaller circle to 8.5mm. Also, set the distance between the two circles to 5mm.



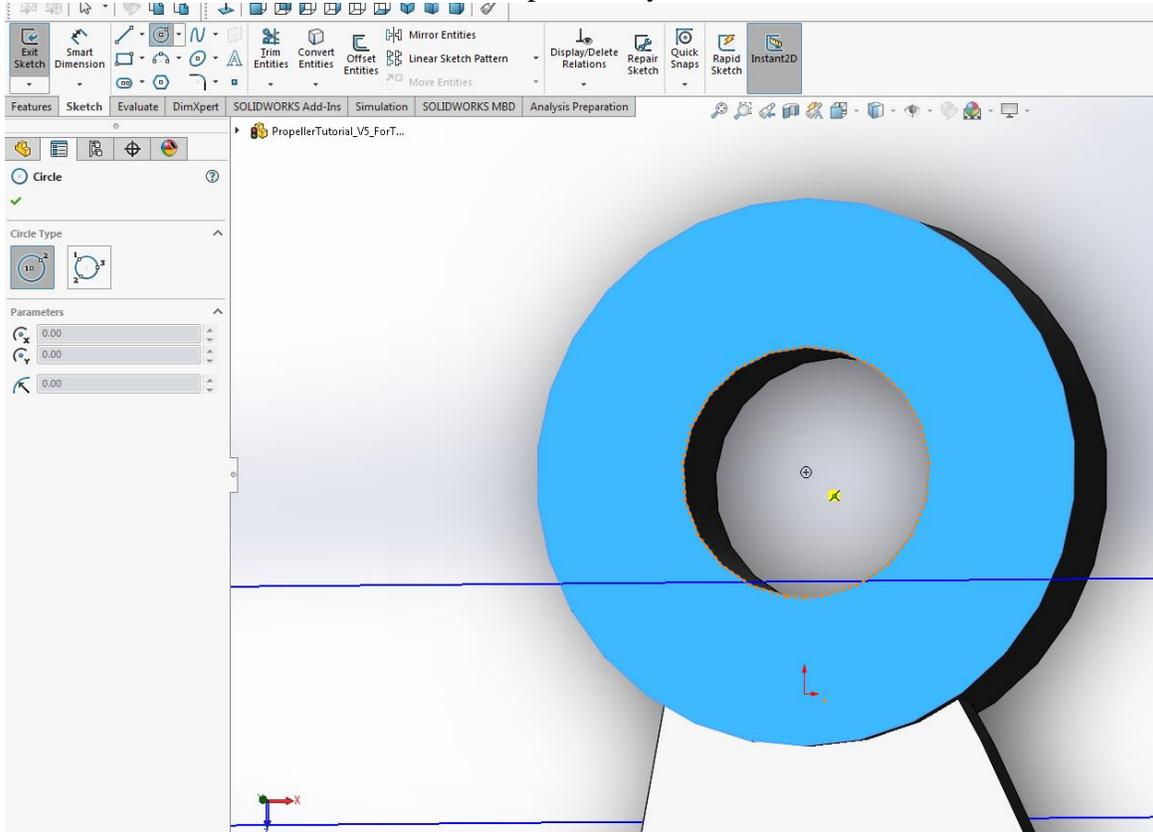
4. Select the center of the circles, then drag this point and drop the circles to a point where the larger circle intersects the entire line. Make sure the circle center is still above the origin; there should be a dotted line connecting the two points:



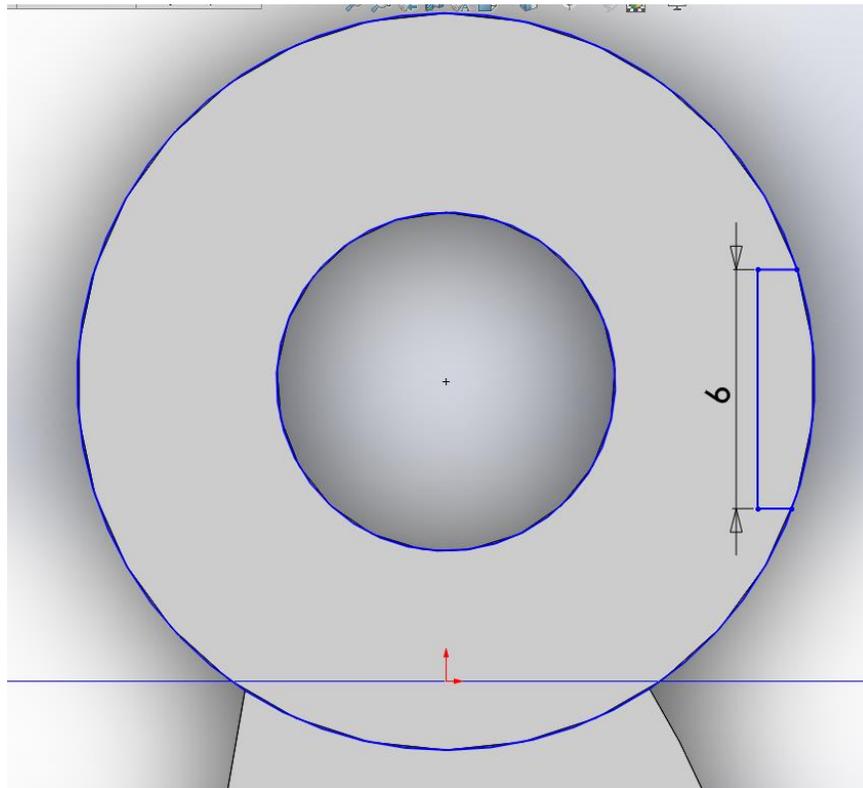
5. Under the “Features” tab, select “Extruded Boss/Base.” Direction 1 should be “Blind” and set the depth to 5mm. **Uncheck the “Merge result” box!**



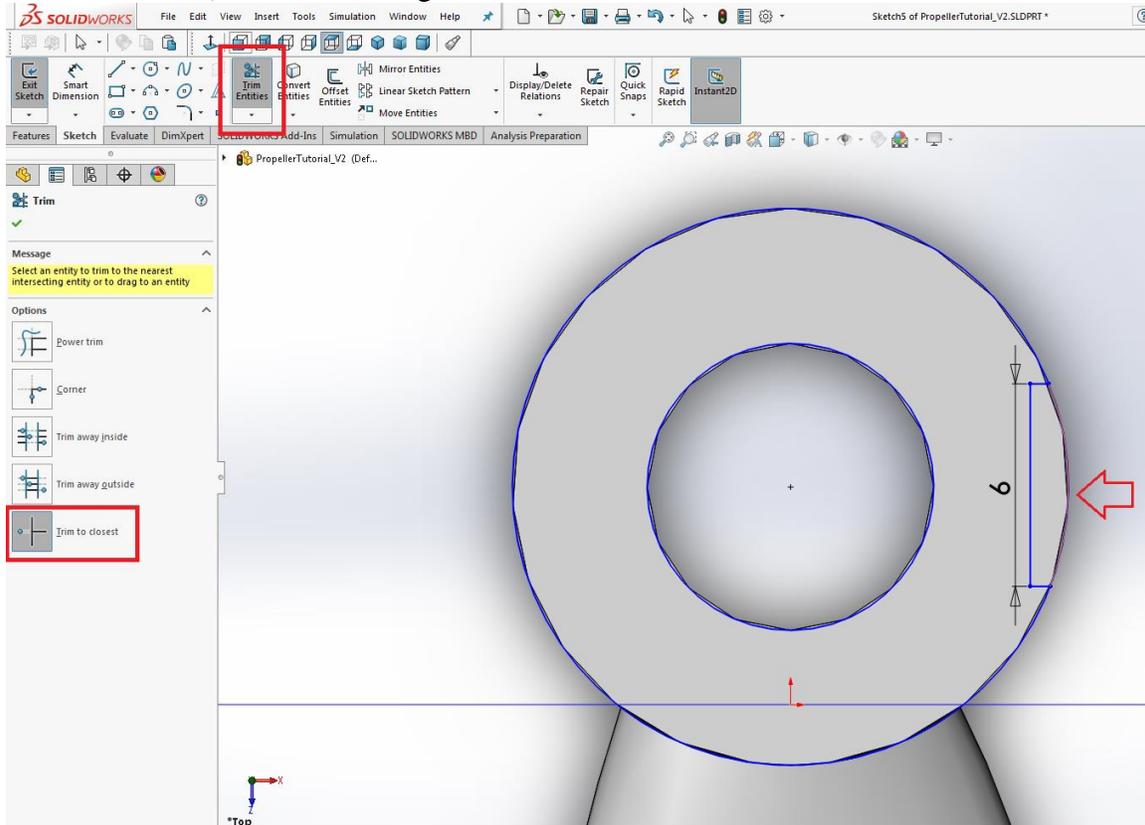
6. Create a new sketch on the face of the circle you just extruded. Draw two concentric circles that are the same diameters as the circles previously drawn.



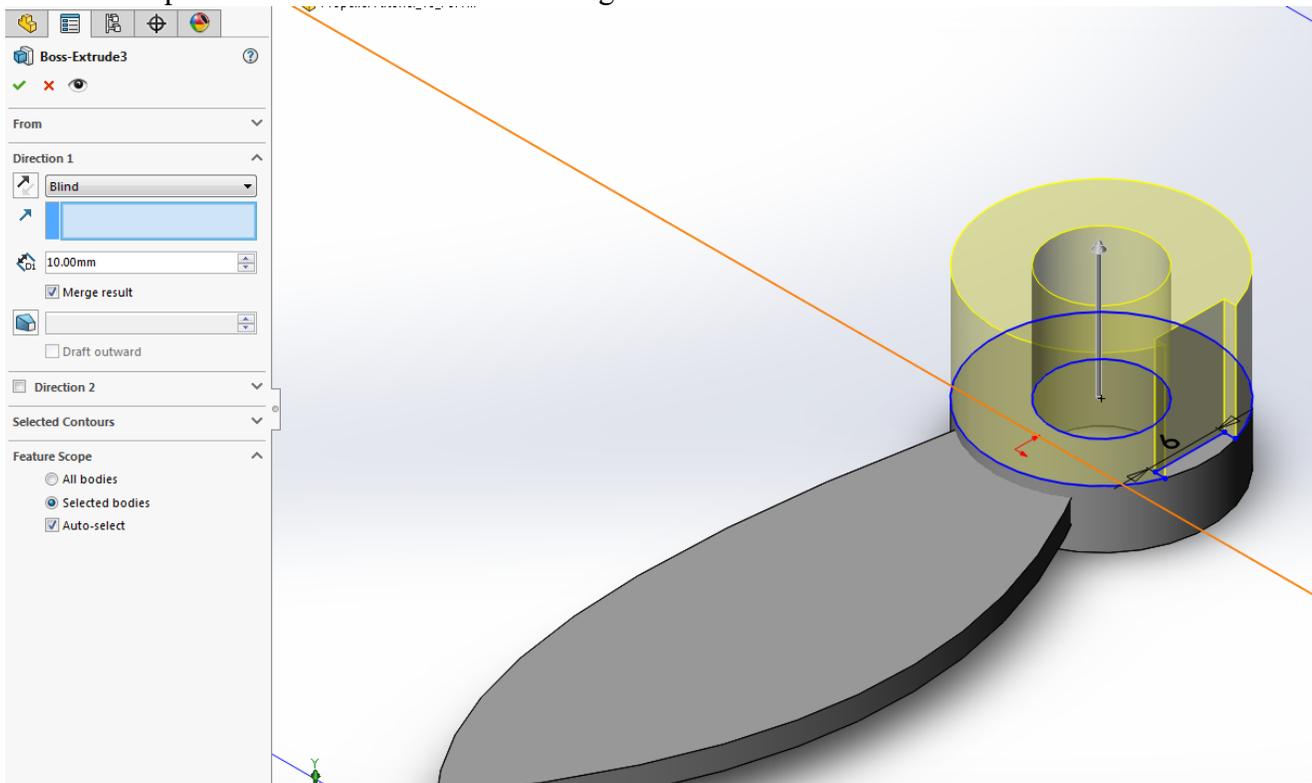
7. Draw lines outlining an inset in the circle. Use the Smart Dimension tool to set the length to 6mm:



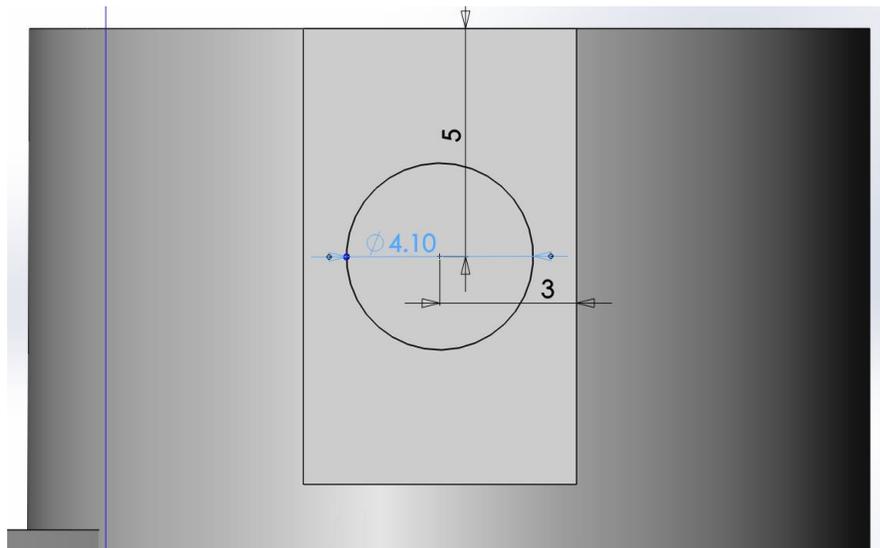
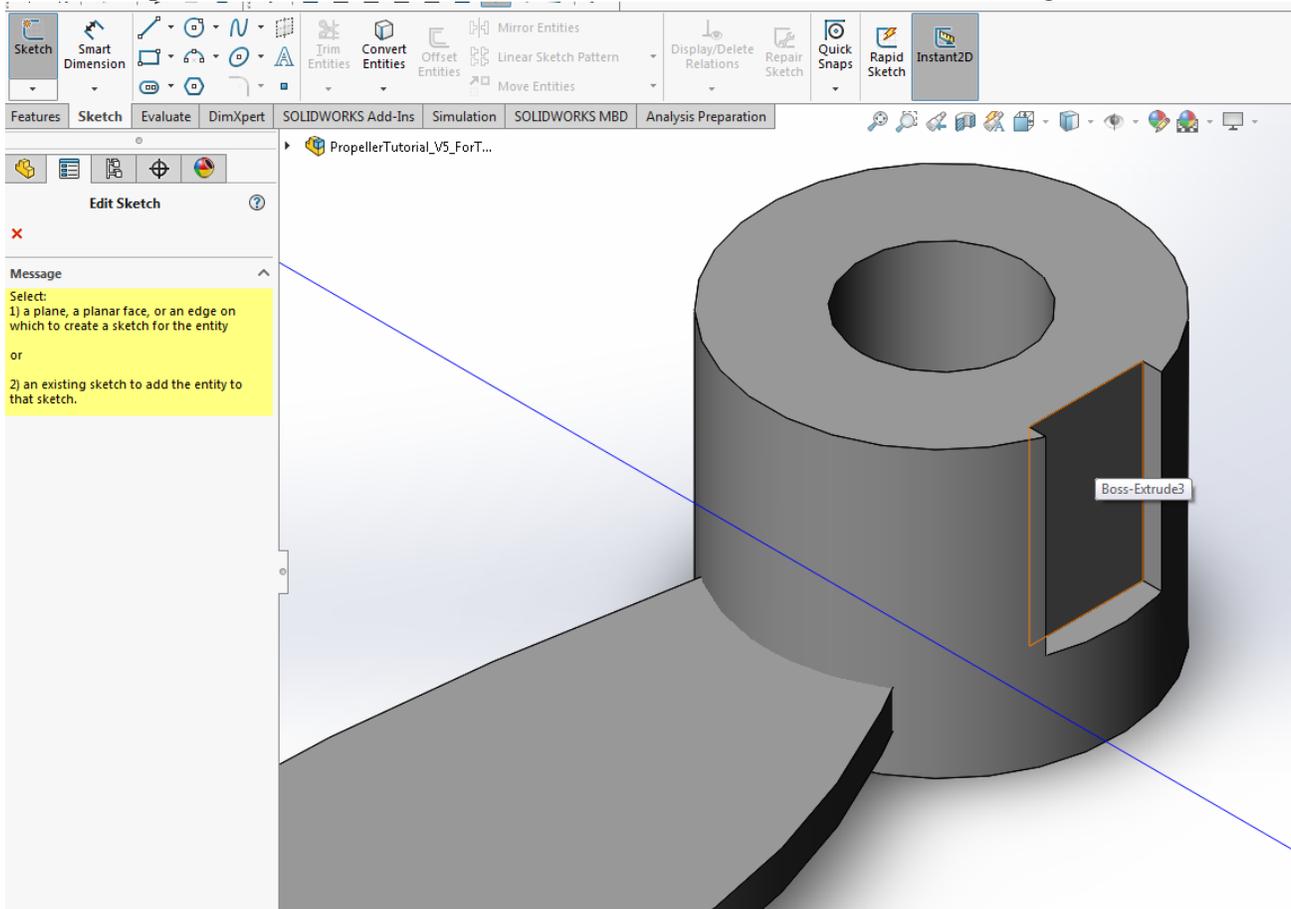
8. Using the Trim Entities tool, remove the segment of the circle that is connected to the inset (see arrow below), then click the green checkmark to confirm.



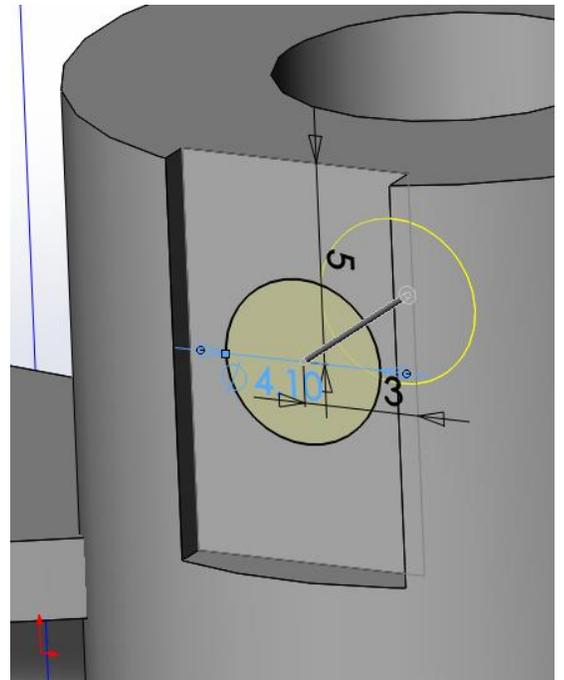
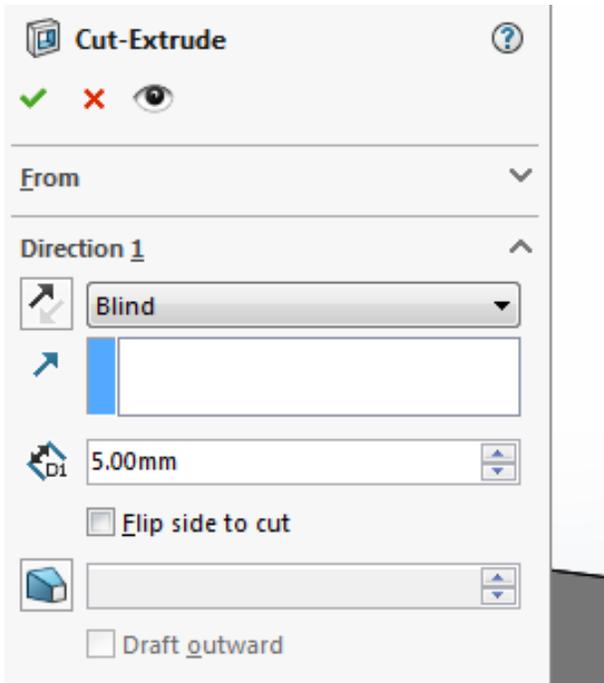
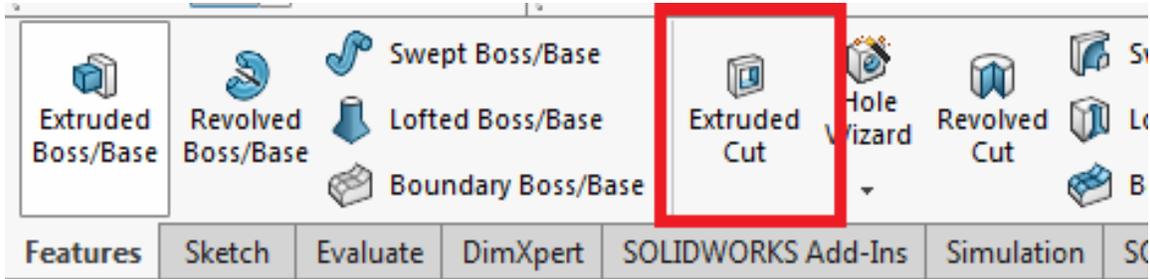
9. Under the “Features” tab, select “Extruded Boss/Base.” Direction 1 should be “Blind” and set the depth to 10mm. Make sure the “Merge result” box is checked.



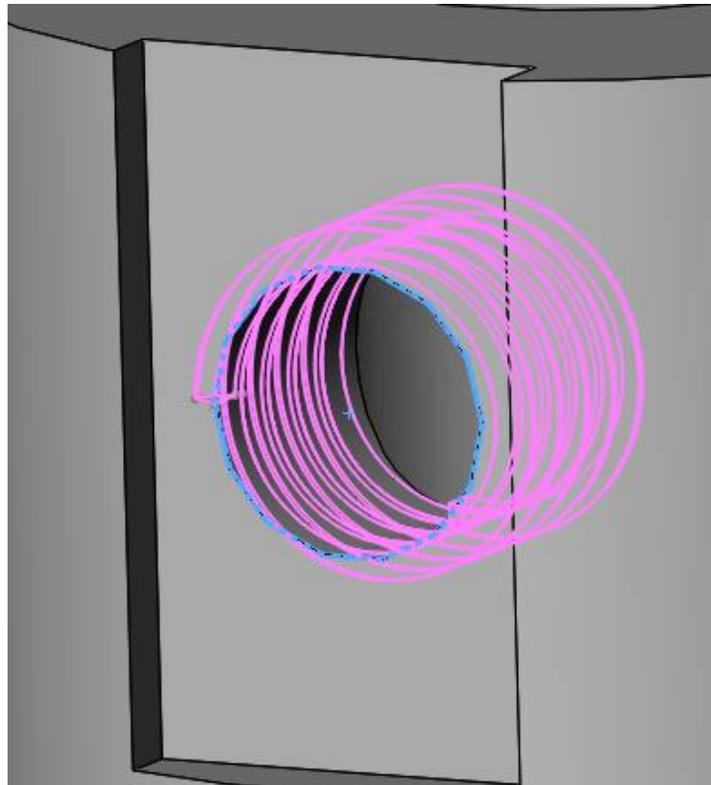
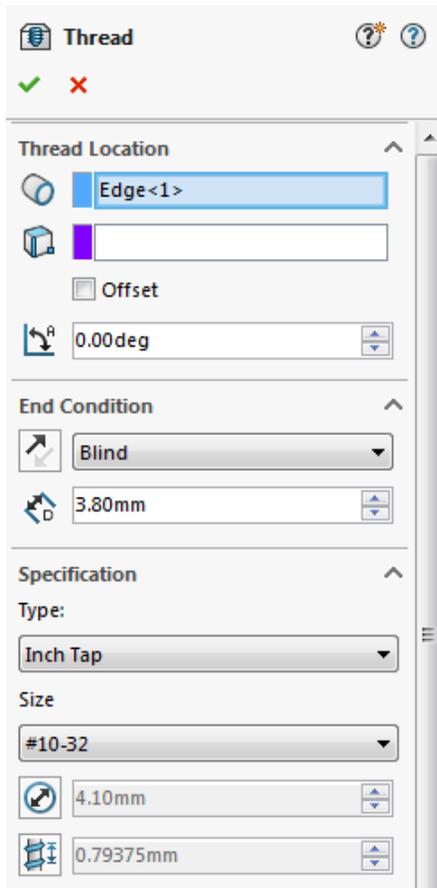
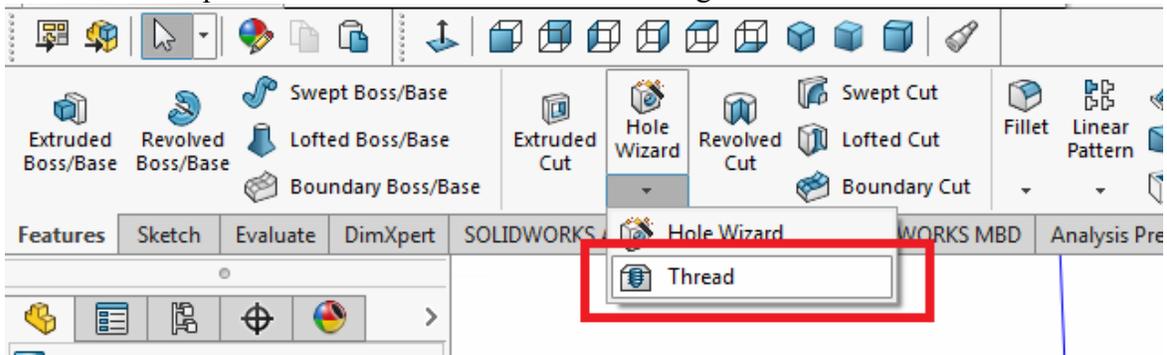
10. Create a new sketch on the face of the inset. Sketch a circle on this face. Using Smart Dimension, set the diameter of the circle to 4.10mm, set the center of the circle 5.0mm above the top of the base, and set the distance from the circle center to the inset edge to 3mm.



11. Under the “Features” tab, select “Extruded Cut.” Set the depth to 5mm. Click the green checkmark to confirm.

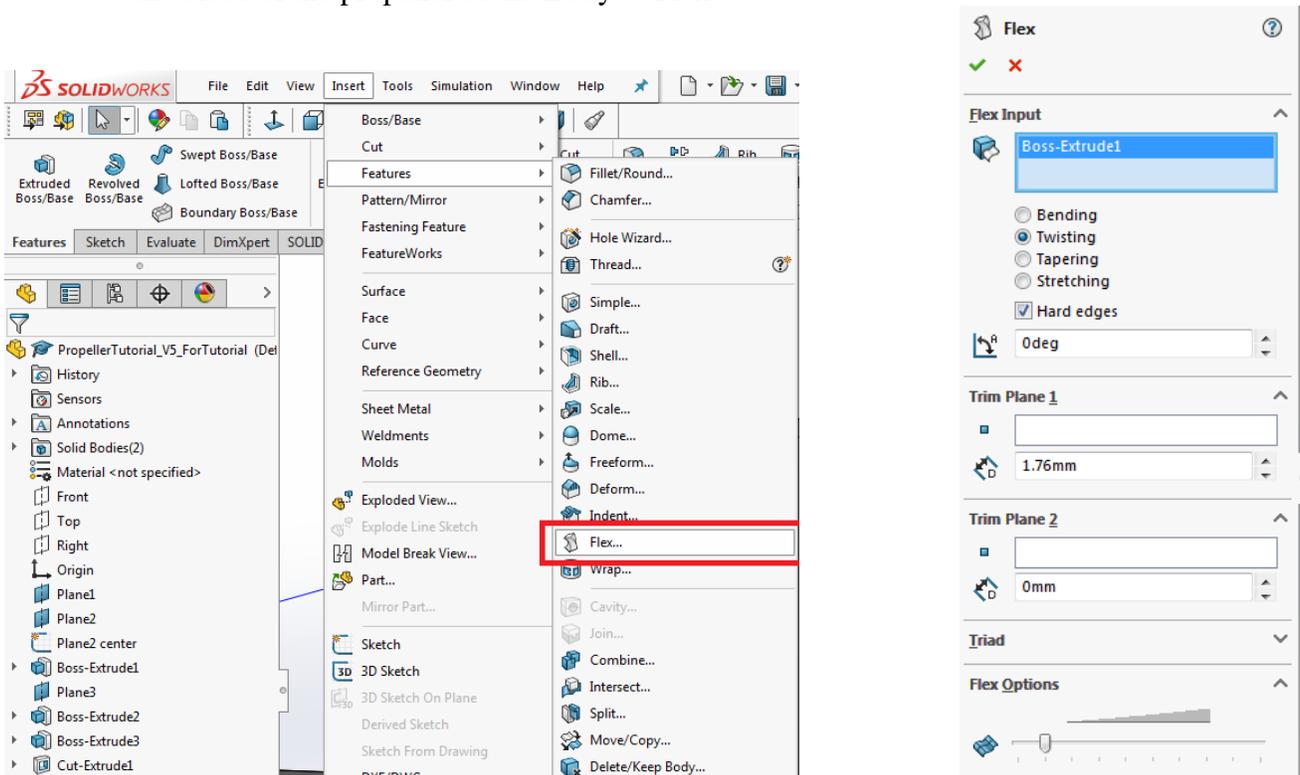


12. Under the “Features” tab, select “Thread” (in the dropdown menu under “Hole Wizard”). The Thread Location is the 4.1mm circle; set the depth to 3.8mm. Under Specifications, set the Type to “Inch Tap” and the Size to “#10-32”. Click the green checkmark.

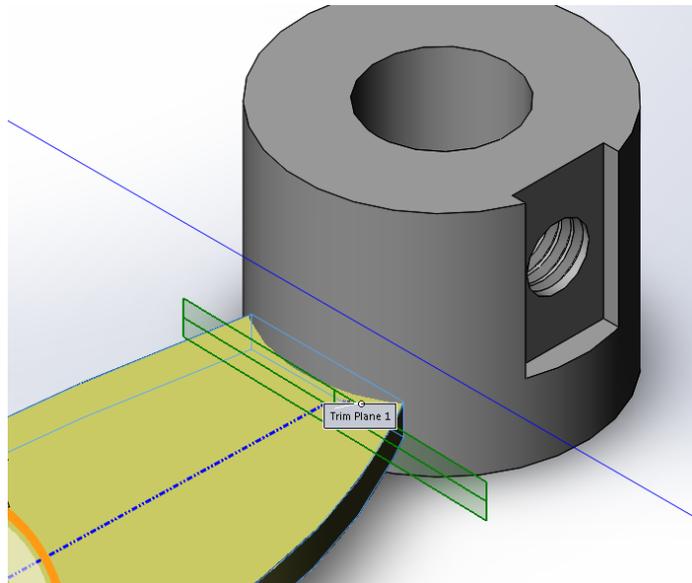


III. Complete the model

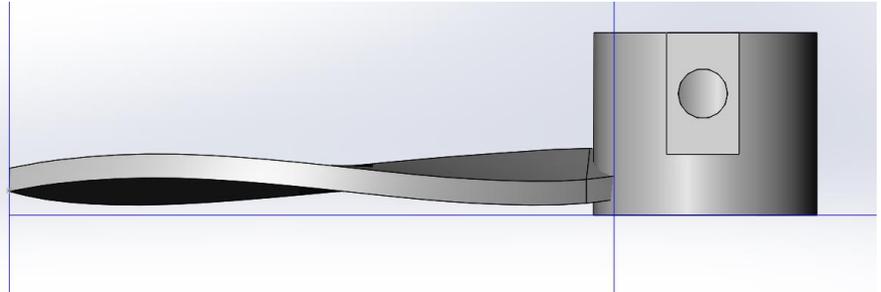
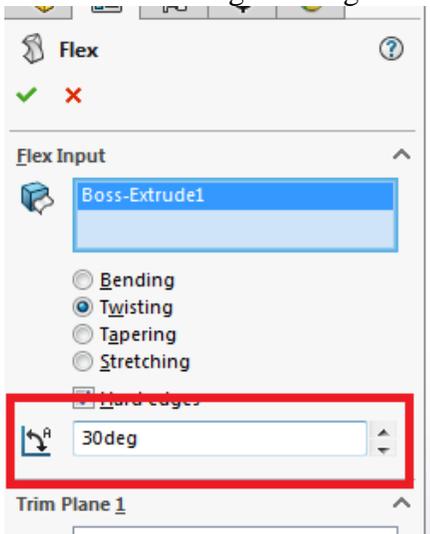
1. Under the “Insert” toolbar, go to Features and then select “Flex.” Select “Twisting” and choose the blade of the propeller as the Body to Flex.



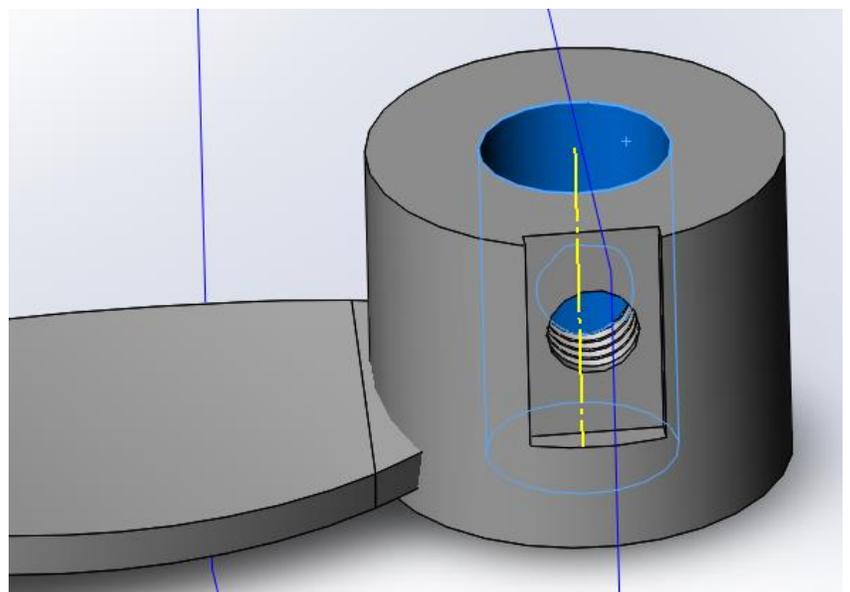
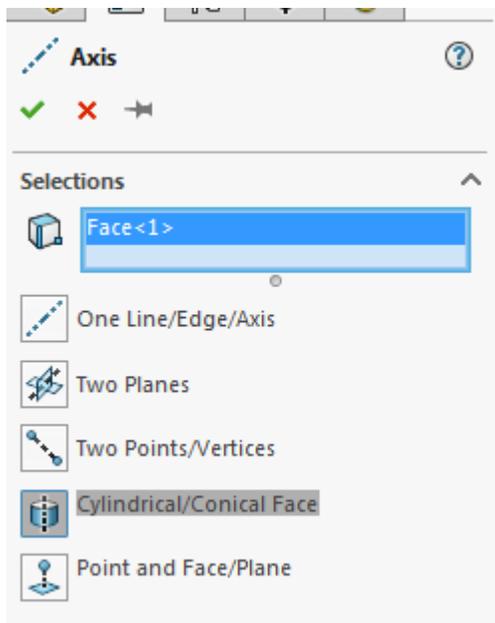
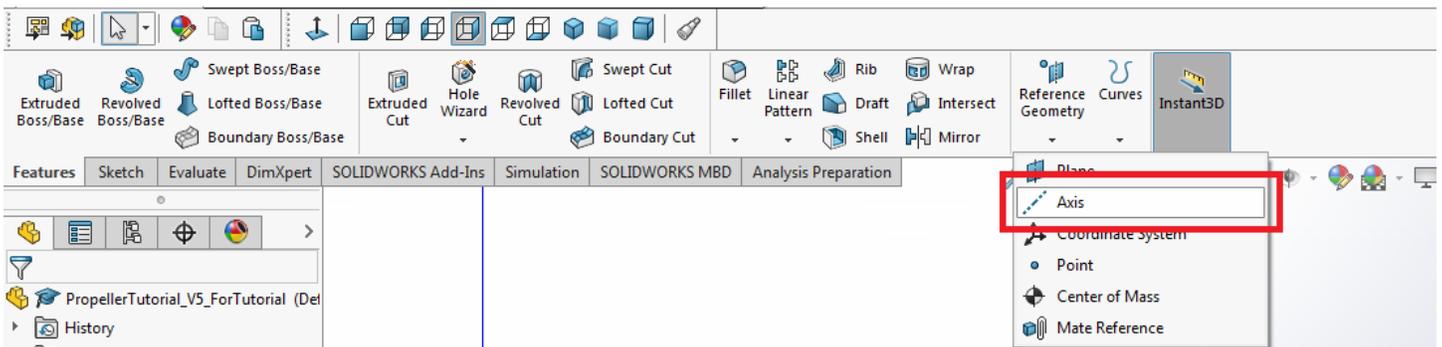
2. Hover the mouse over Trim Plane 1 (green) until the 4-directional arrows appear; click and drag the plane until it does not touch the cylindrical base.



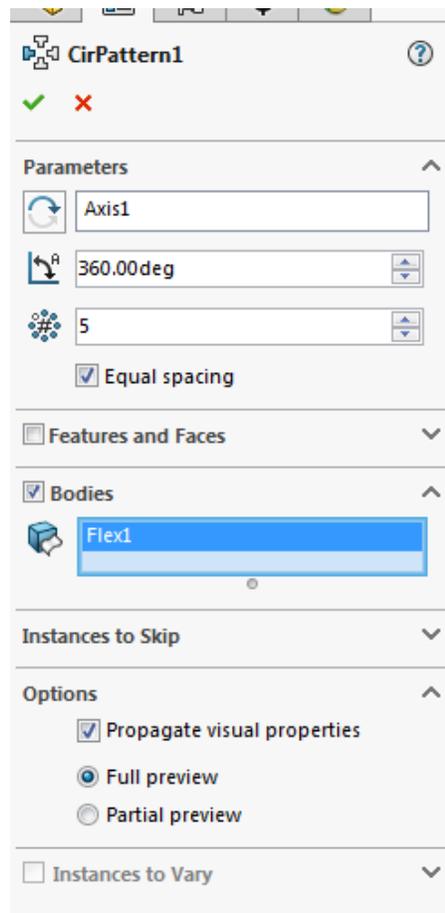
3. Change the degrees to the desired twist. The picture below has a 30deg twist:



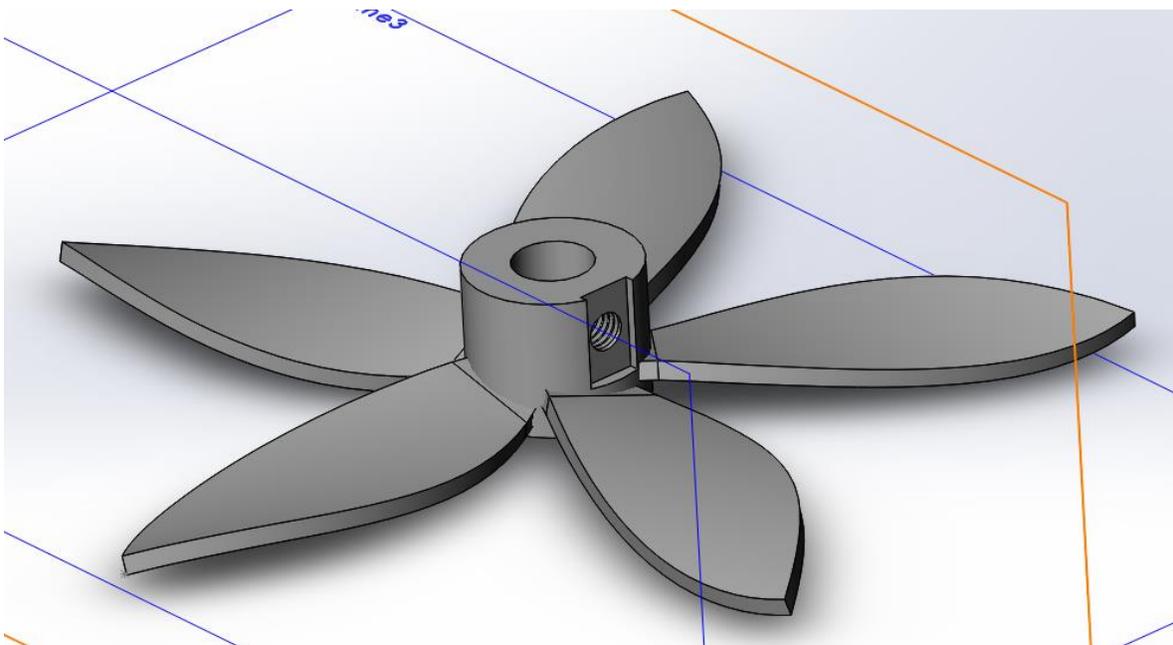
4. Add a reference axis to the center of the base; choose the inner cylinder of the base as the reference entity. Click the green checkmark to confirm.



5. Under the “Features” tab, select “Circular Pattern” (may be under the Linear Pattern dropdown menu). Choose the reference axis that you just made as the Pattern Axis, and choose the propeller blade as the Body to Pattern. **Make sure you are selecting a Body and not a Feature or Face.** Check the “Equal Spacing” box, and add 5 instances.



Hit the green checkmark:



Your design is complete!