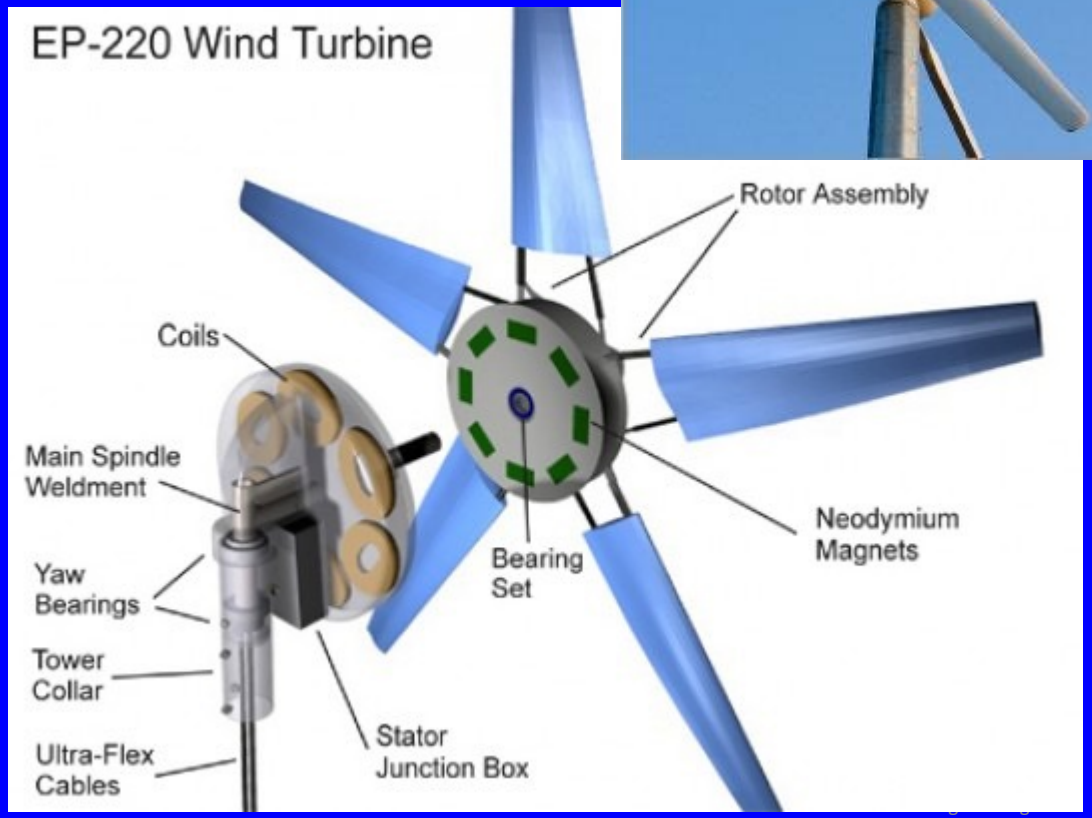
West Valley College
The Blade design
Tutorial

Various Blade designs

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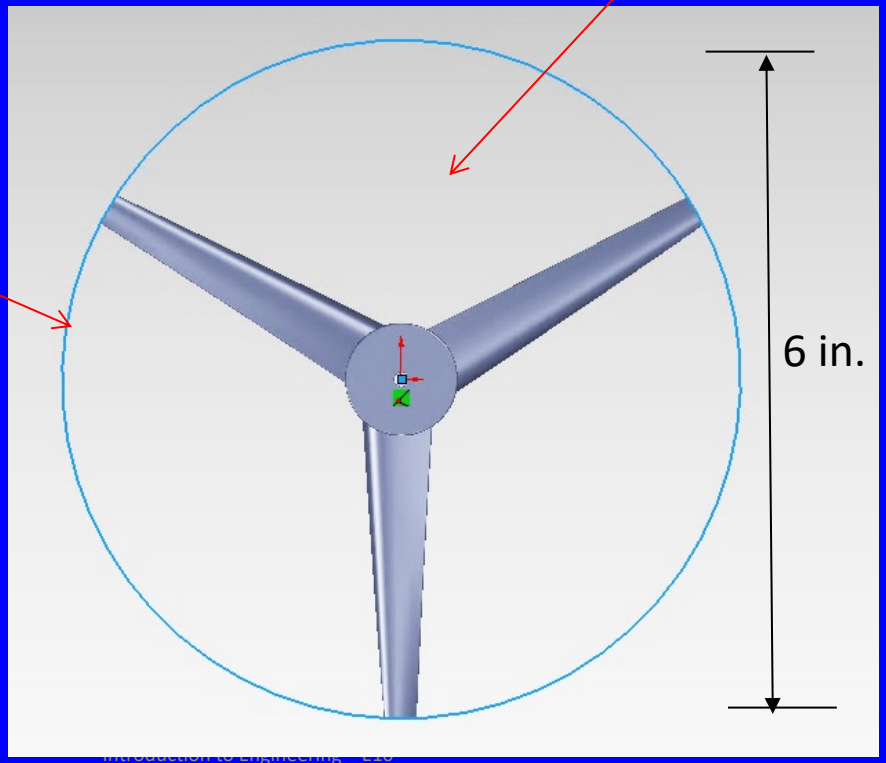
The Blade Shape

Design groups should conduct a thorough search of the Internet to obtain information on wind turbine blade design and efficiency. Based on your research, decide on the number and general shape of the blade.

Constraints:

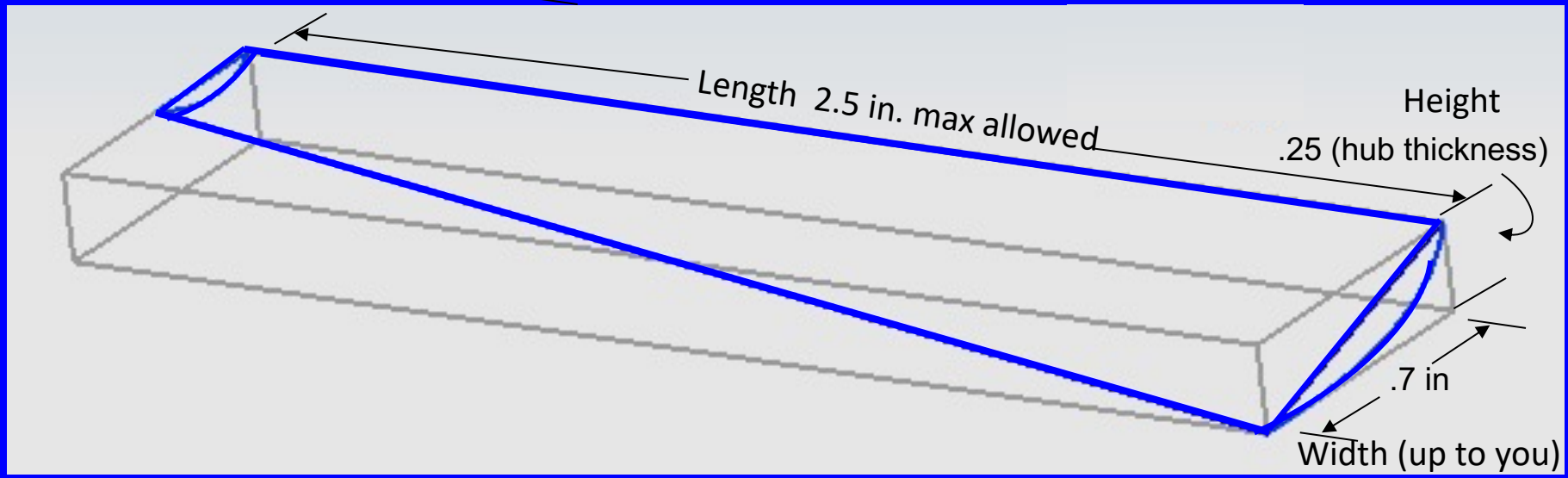
- Total dimension of the turbine blade assembly (diameter of the swept area) should not exceed 6 inches.

The propeller must fit within the 6 inch diameter circle



The Blade Shape

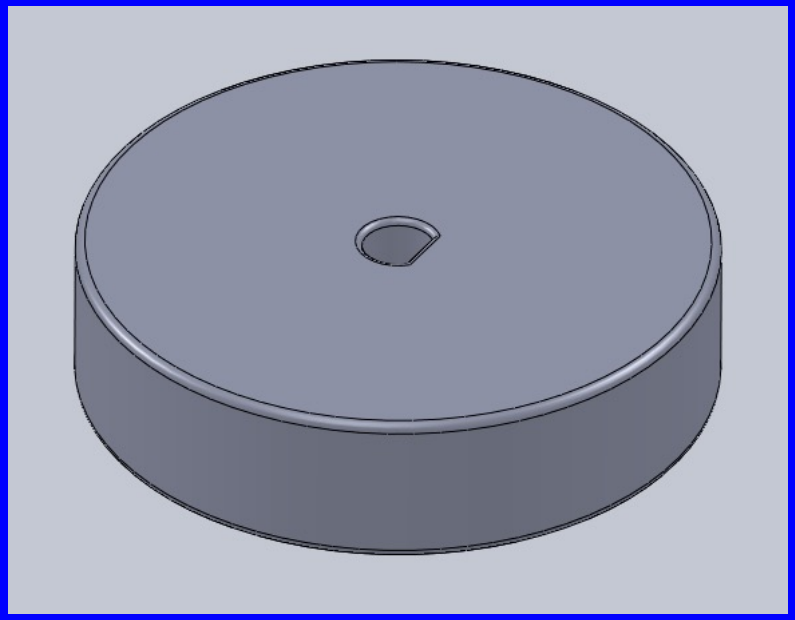
- Blade **length**, measured from the edge of the hub, should not exceed **2.5 inch**.
- Blade **height** is limited by the thickness of the hub. It should not exceed **0.25 inch.**
- **There are no constraints on blade width (.7 dimension)**

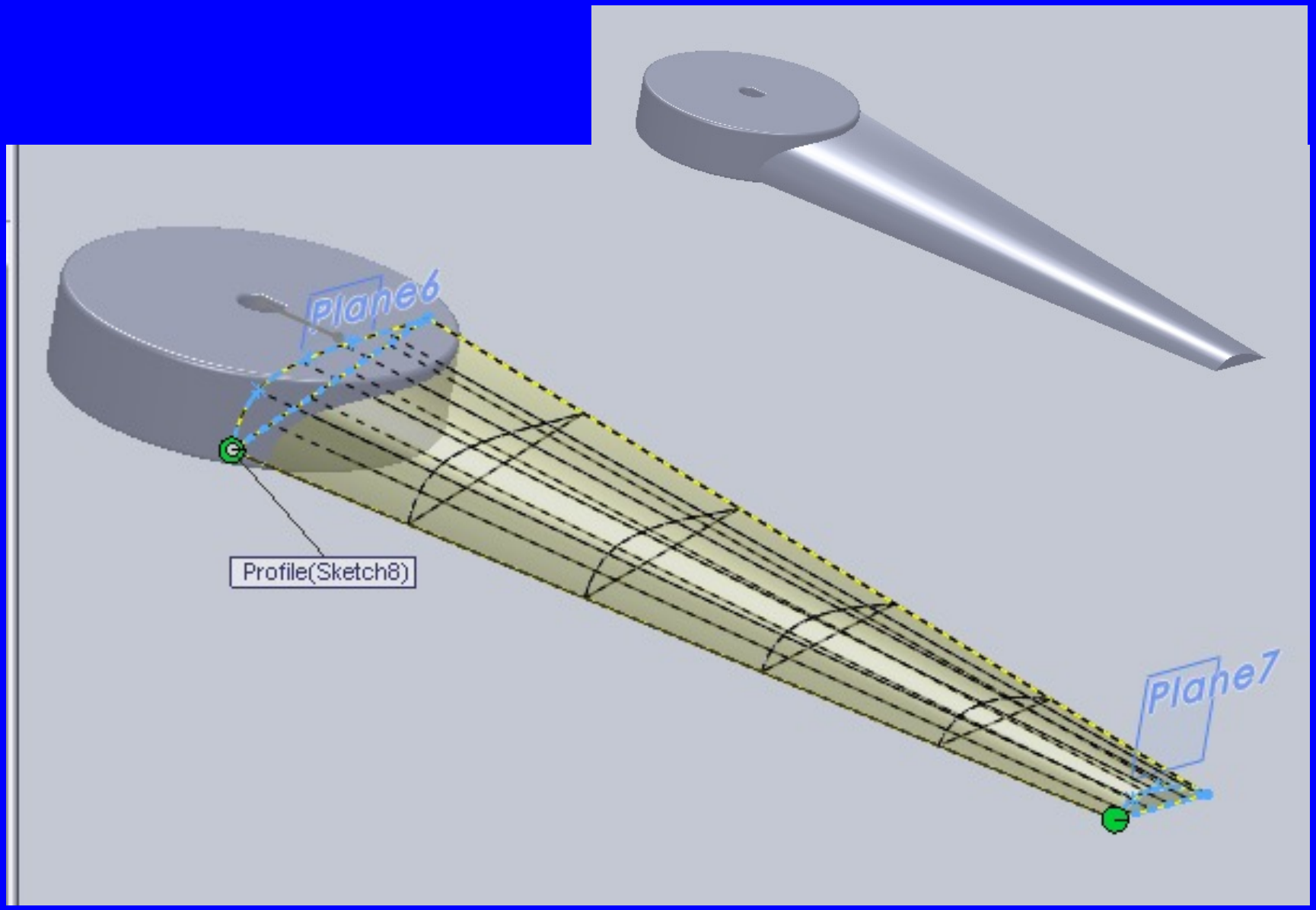


The tutorial in the following slides is based on the blade geometry and shape shown above.

Modeling the Wind Turbine Blade

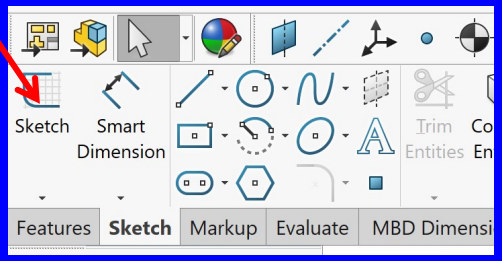
The hub for the wind turbine blade has been modeled using SolidWorks. Use the following link to download the file **"Hub"** from Canvas in Wind Turbine Module

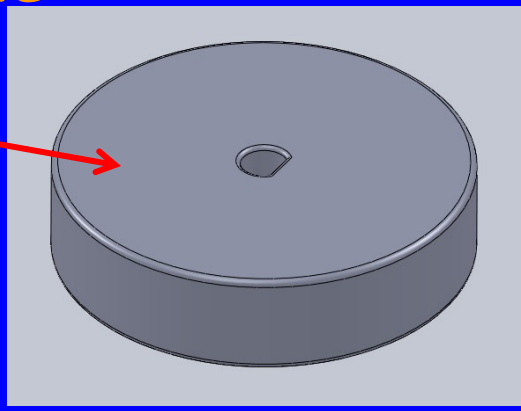



Plane6
Profile(Sketch8)
Plane7

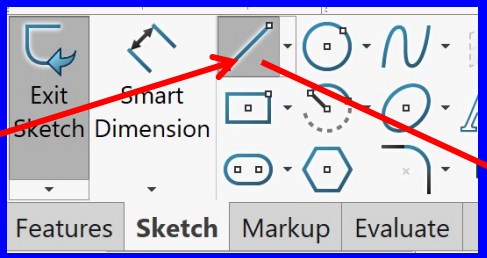
Create the first Reference Plane

1. Select Sketch and choose the top surface of the hub to draw.

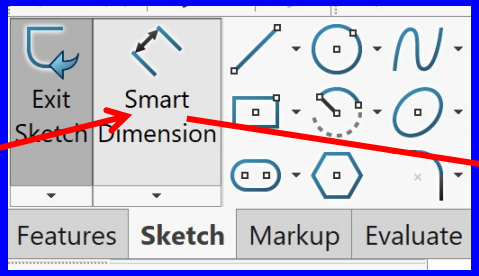


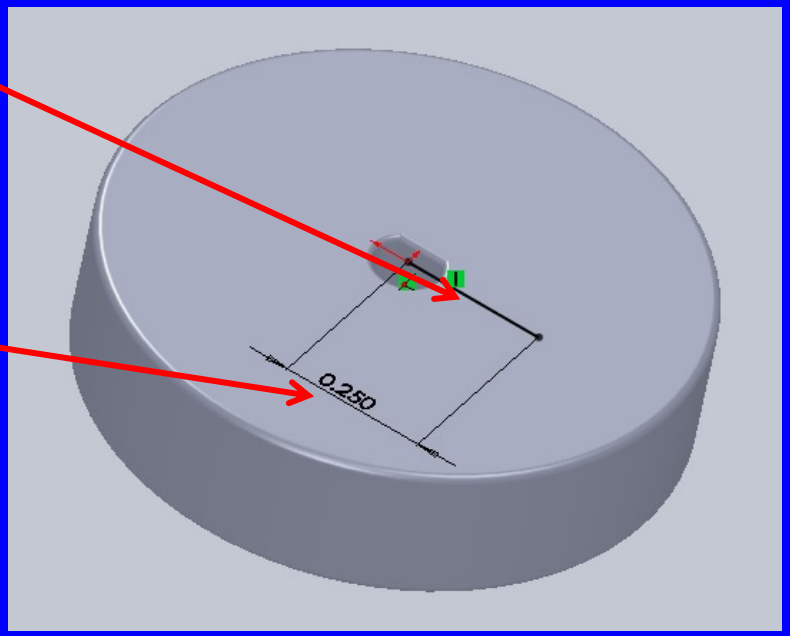


2. Draw a line from the center

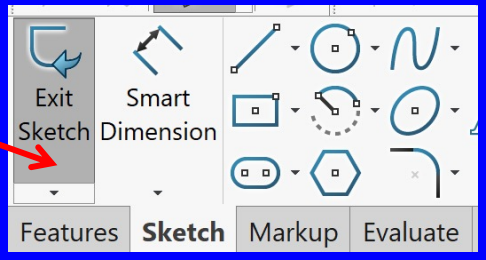


3. Dimension the line 0.25 inch



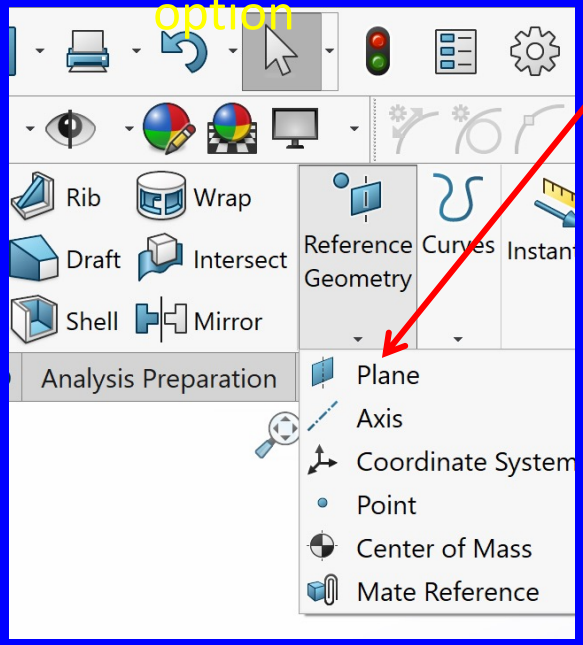


4. Exit the sketch

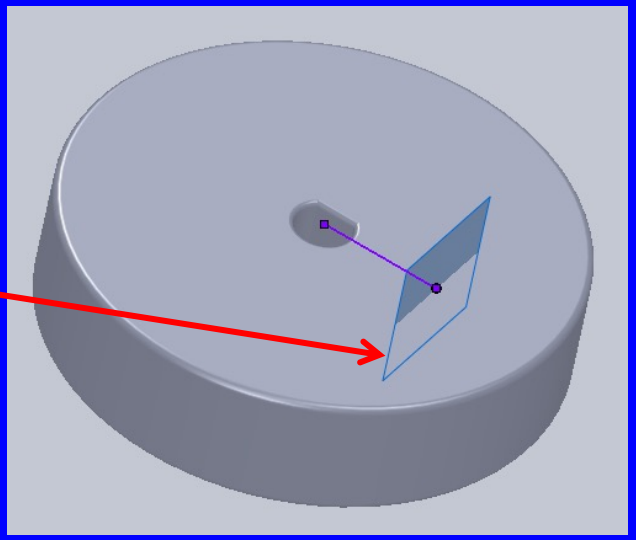


Create the first Reference Plane

1. In Feature tab, select Reference Geometry option to create a plane, choose the Plane option

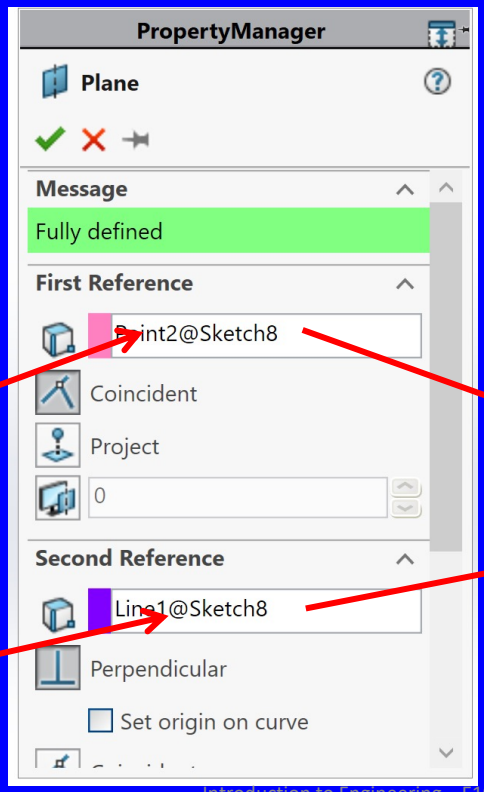


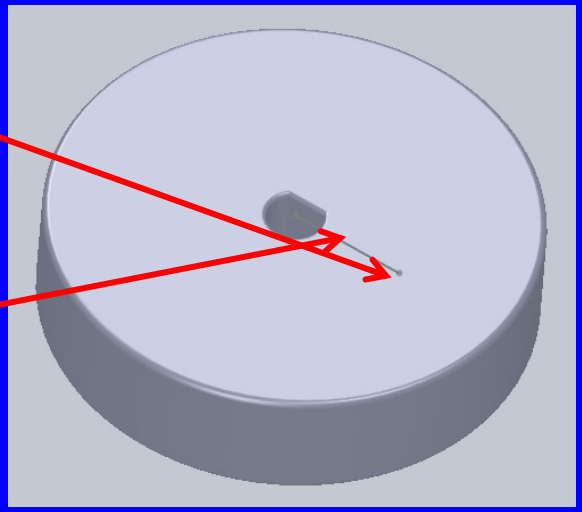
The reference plane passes through the point and is perpendicular to the line



2. Select the end of the line as the first reference.

3. Select the line as the second reference

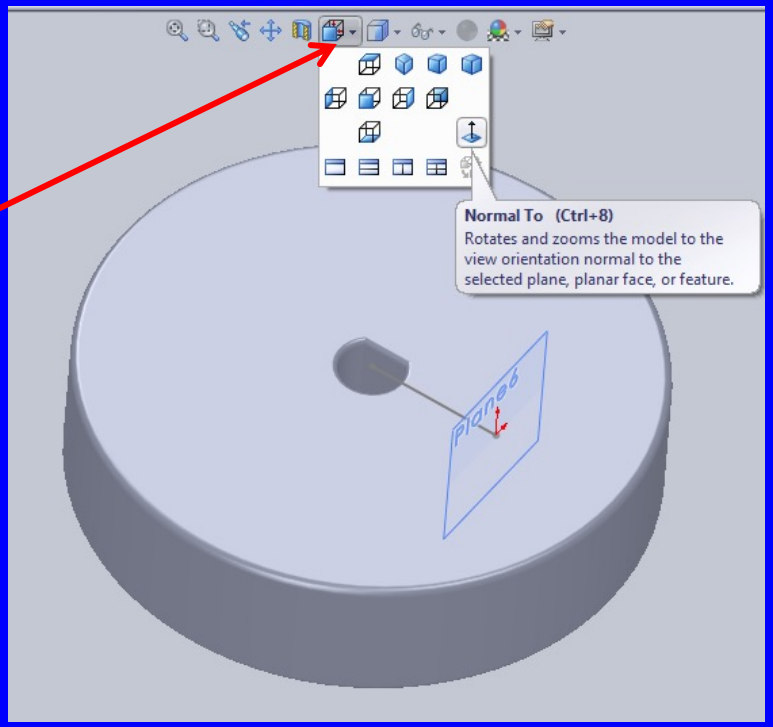




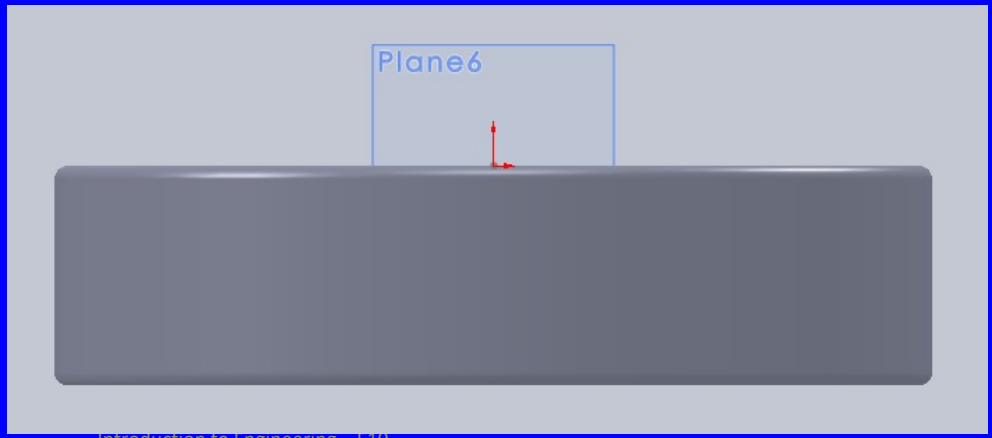
Creating the Blade Profile at the Hub

Select ***Sketch*** and choose the first reference plane to draw the blade profile at the hub

From the ***View Orientation*** menu choose the ***Normal To*** option

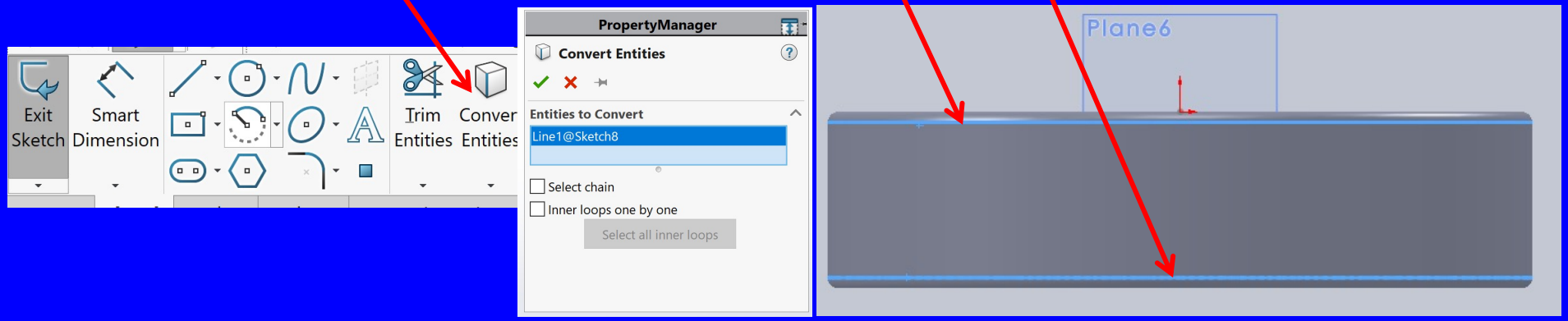


Do not be concerned with the size of the reference plane; it is infinitely large.



Creating the Blade Profile at the Hub

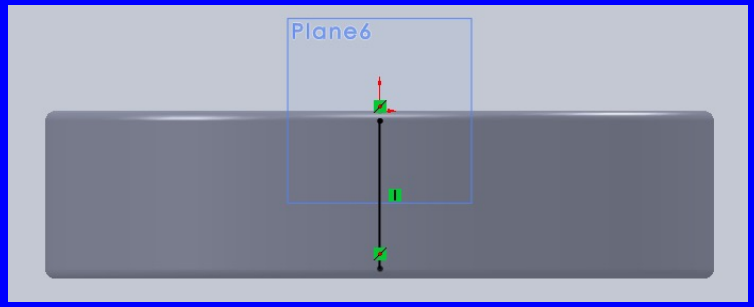
Select ***Convert Entities*** to project the hub onto the sketch plane to use as a reference for drawing the profile. Select the top and bottom edges as shown



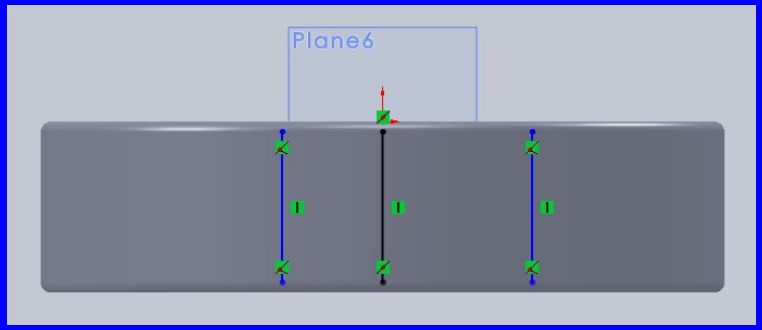
Make sure you select edges and not surfaces. Zoom in for better view

Creating the Blade Profile at the Hub

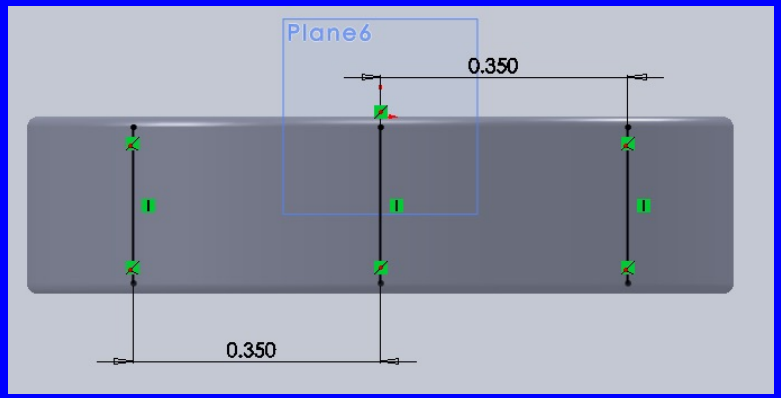
Draw a vertical line from the midpoints of the projected lines



Draw two more vertical lines on either side of the previous line



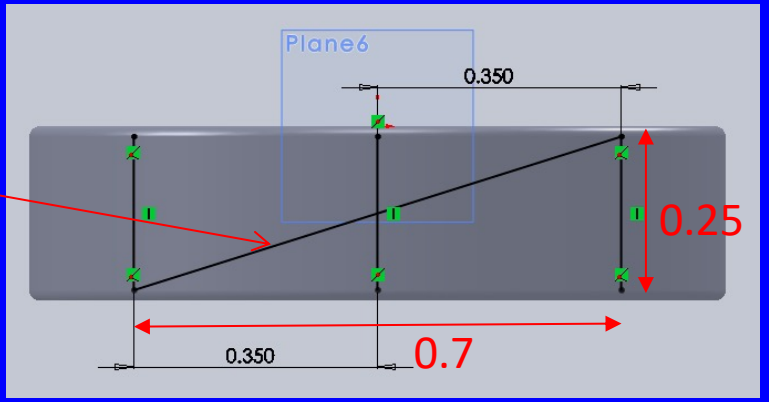
Dimension both lines with respect to the middle line, 0.35 inch. The 0.7 inch distance represents the width of the blade at the hub. (This is a design variable; you may choose a different value.)



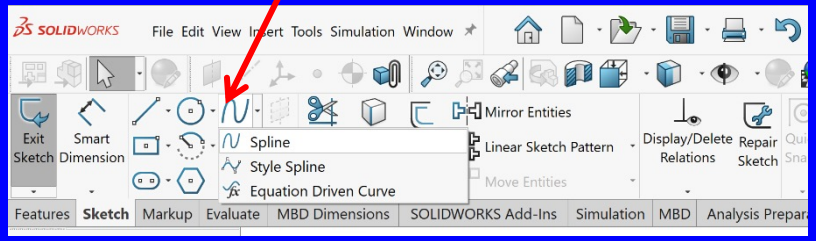
Creating the Blade Profile at the Hub

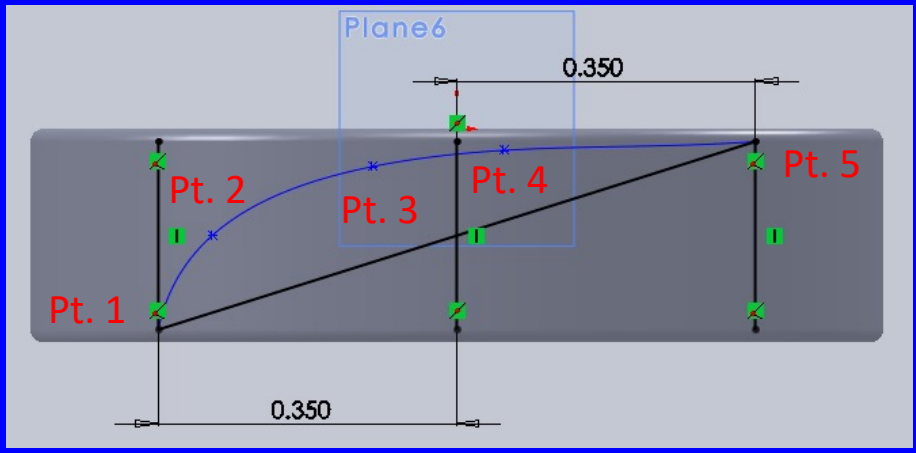
The profile is constructed within the 0.7” by 0.25” envelope

Construct a diagonal line



Use the spline command to draw the curved portion of the profile



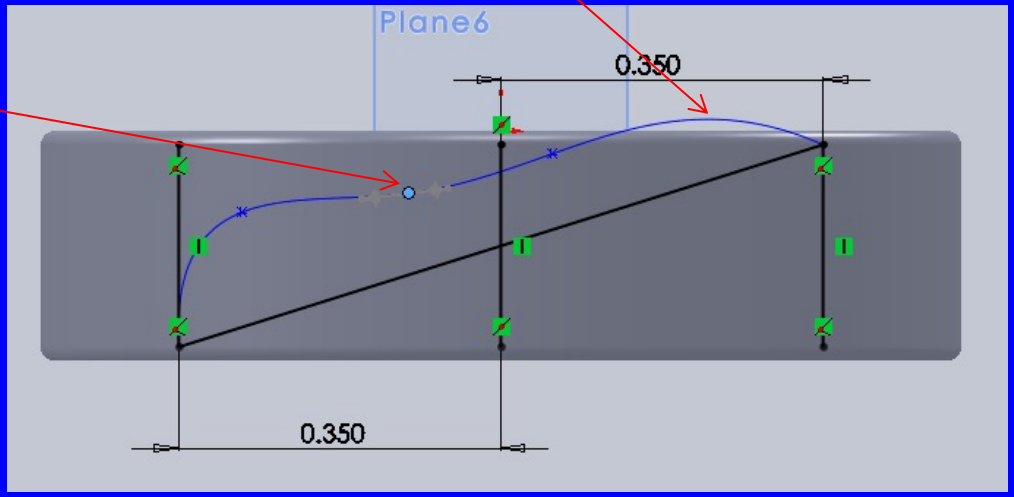


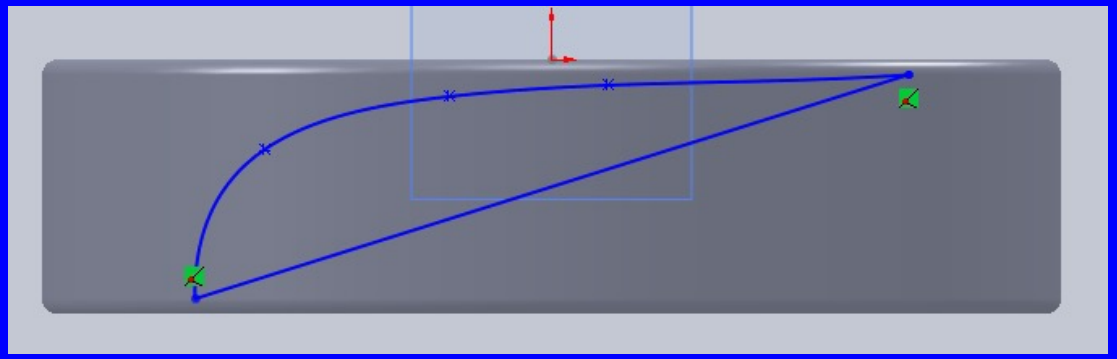
In this design, 5 points are selected. The end points of the diagonal line must be included in your point selection. The location of other points are up to you and will determine shape of the profile, which in turn will affect the power output.

Creating the Blade Profile at the Hub

Make sure the curve does not protrude beyond the surface of the hub

The shape of the spline curve could be modified by selecting any point and dragging it to a new position

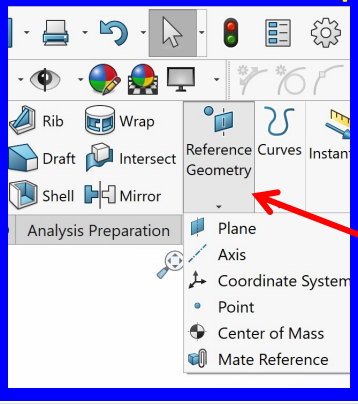


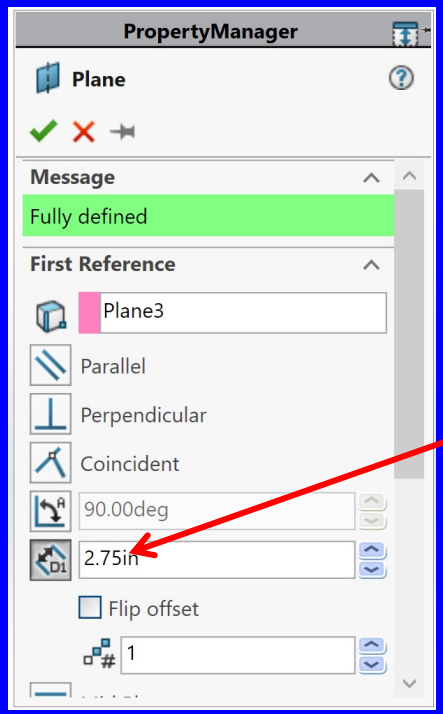


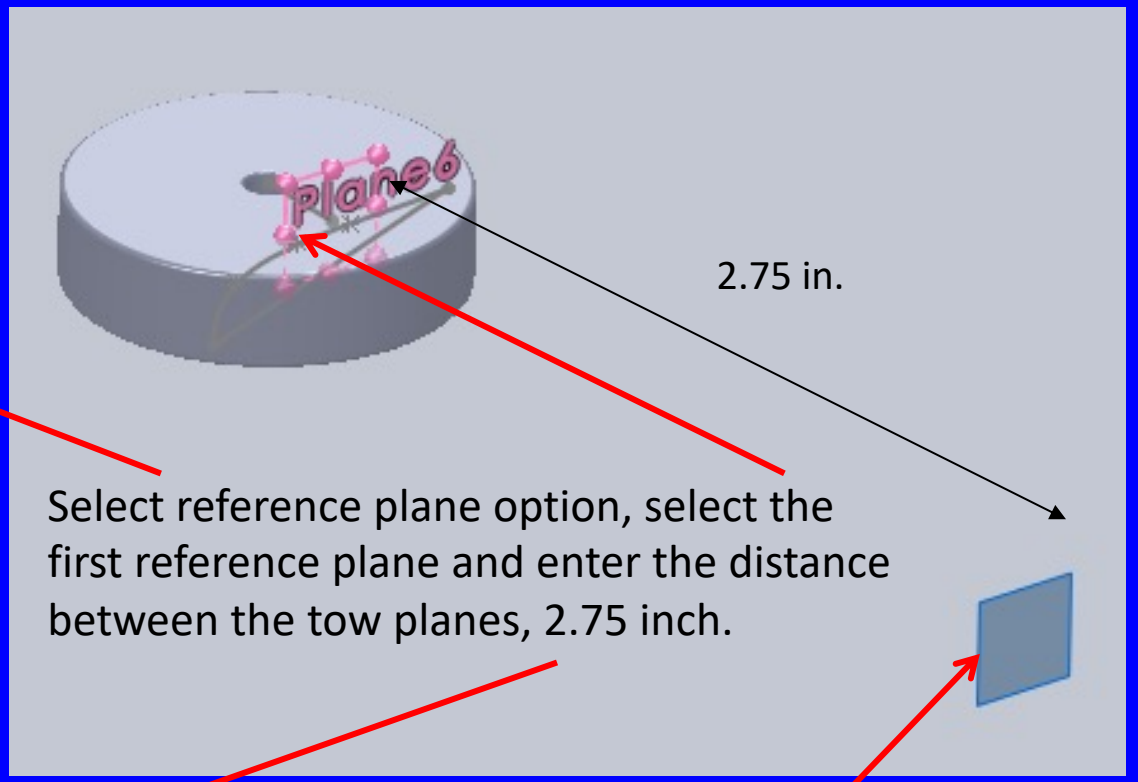
Delete all construction lines before exiting the sketch

Creating the Blade Profile at the Tip

Create the second reference plane to draw the tip profile





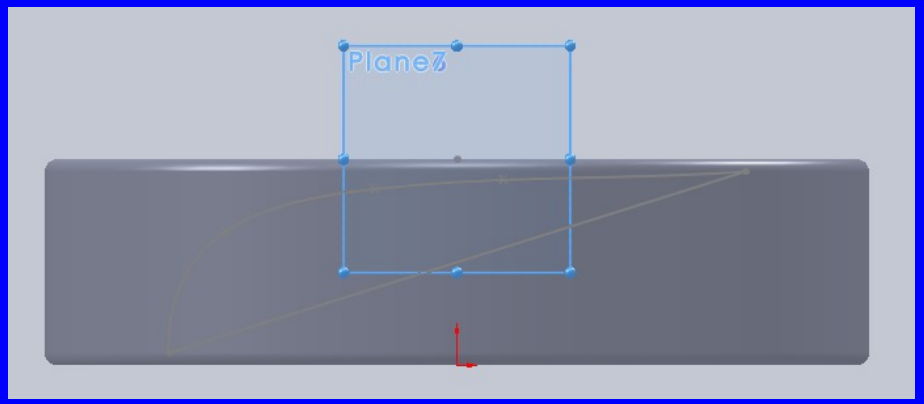


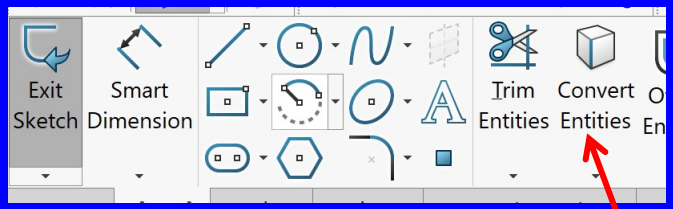
This arrangement will create a blade sweep area of 6 inch in diameter. It is the maximum blade size allowed.

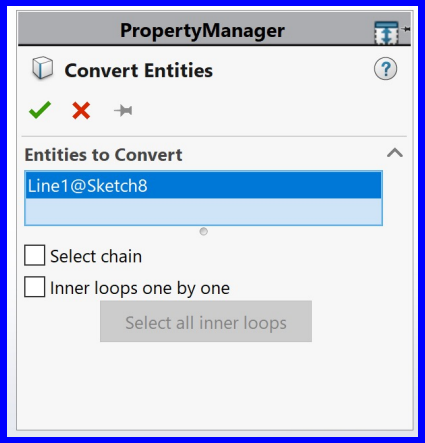
Creating the Blade Profile at the Tip

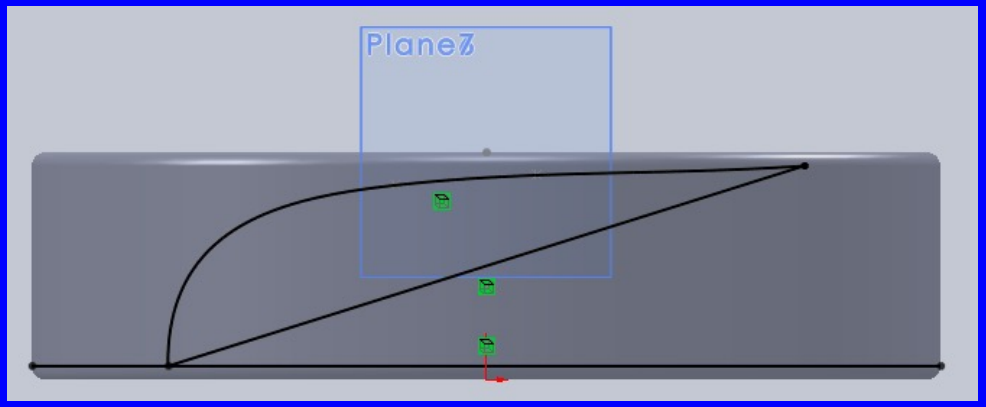
Select ***Sketch*** and choose the second reference plane to draw the tip blade profile.

From the ***View Orientation*** menu choose the ***Normal To*** option.





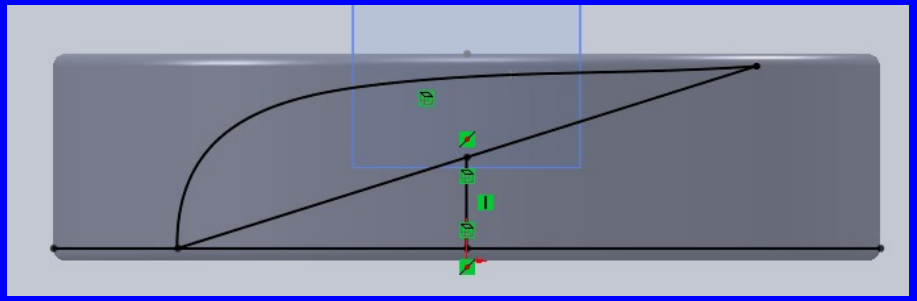




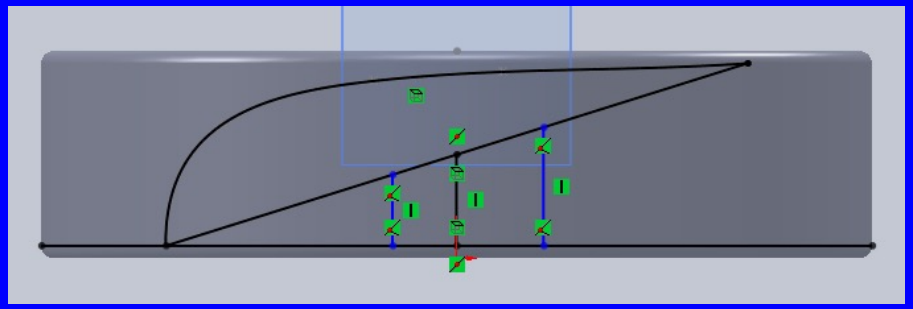
Project the blade profile at the hub and the bottom surface into the current sketch plane. This is used as a reference to draw the tip profile

Creating the Blade Profile at the Tip

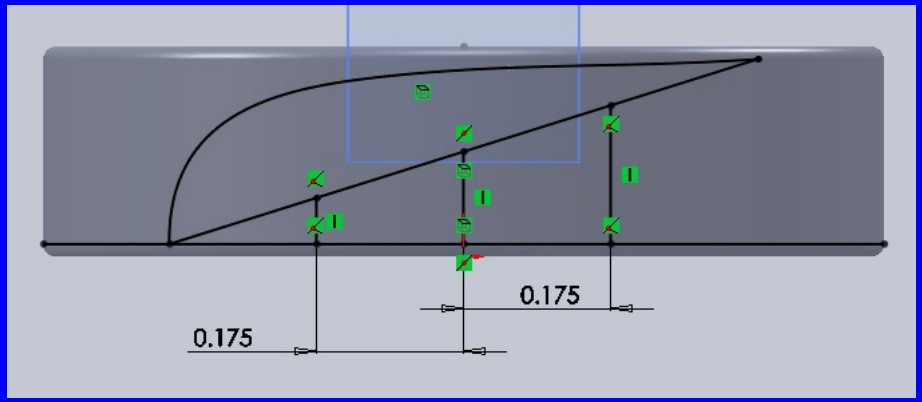
Draw a vertical line from the midpoints of the projected diagonal line.



Draw two more vertical lines on either side of the previous line

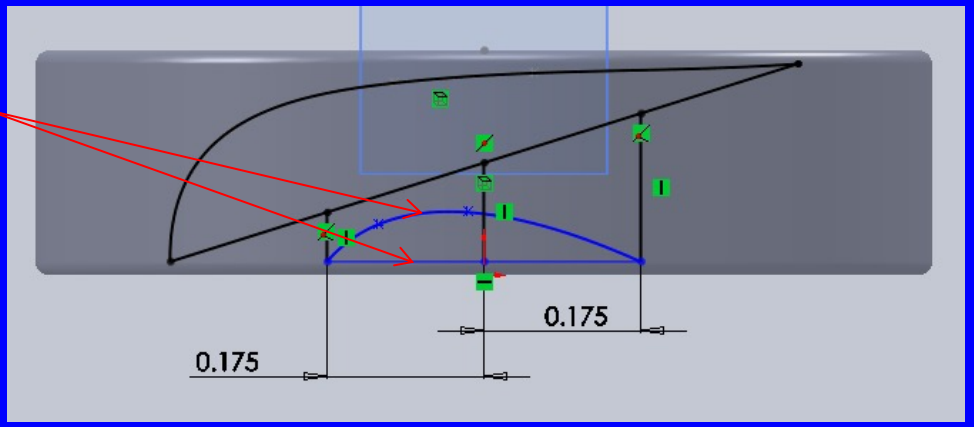


Dimension both lines with respect to the middle line, 0.175 inch. The 0.35 inch represents the width of the blade at the tip, half of the width at the hub. This is also a design variable; you may choose a different value

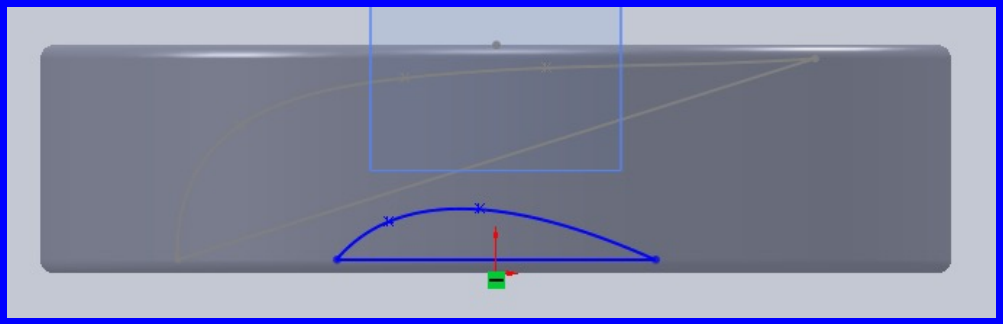


Creating the Blade Profile at the Tip

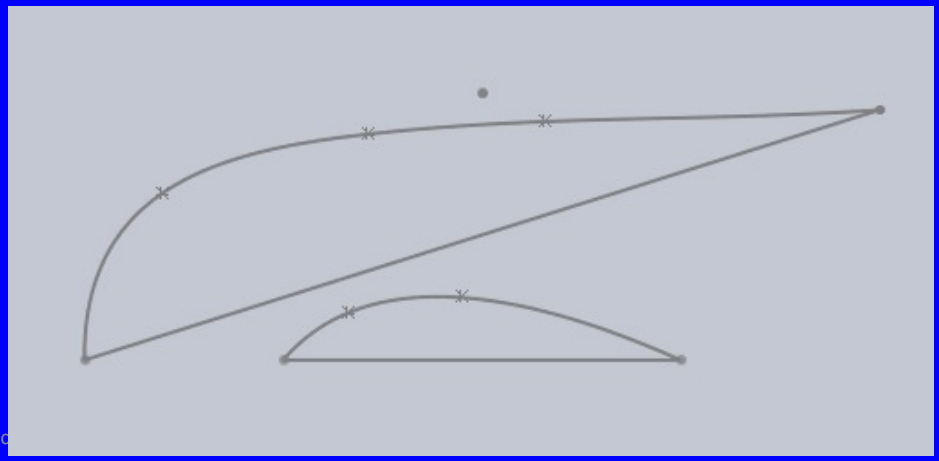
Use the ***Spline*** and ***line*** commands to construct the tip profile. Follow the same procedure as the profile at the hub.



Delete all construction lines before exiting the sketch

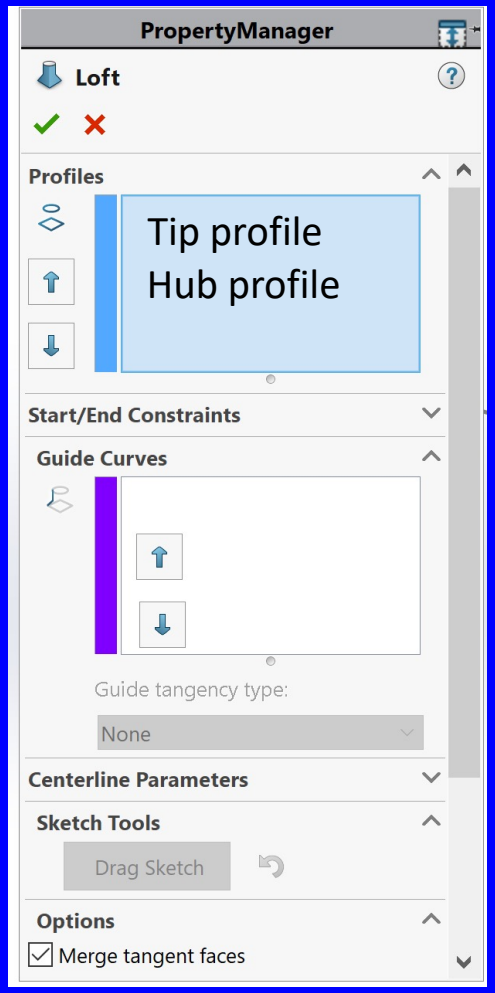


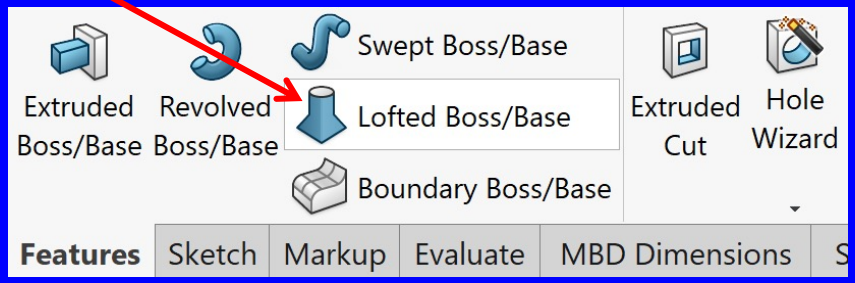
This shows the two blade profiles, at the hub and at the tip

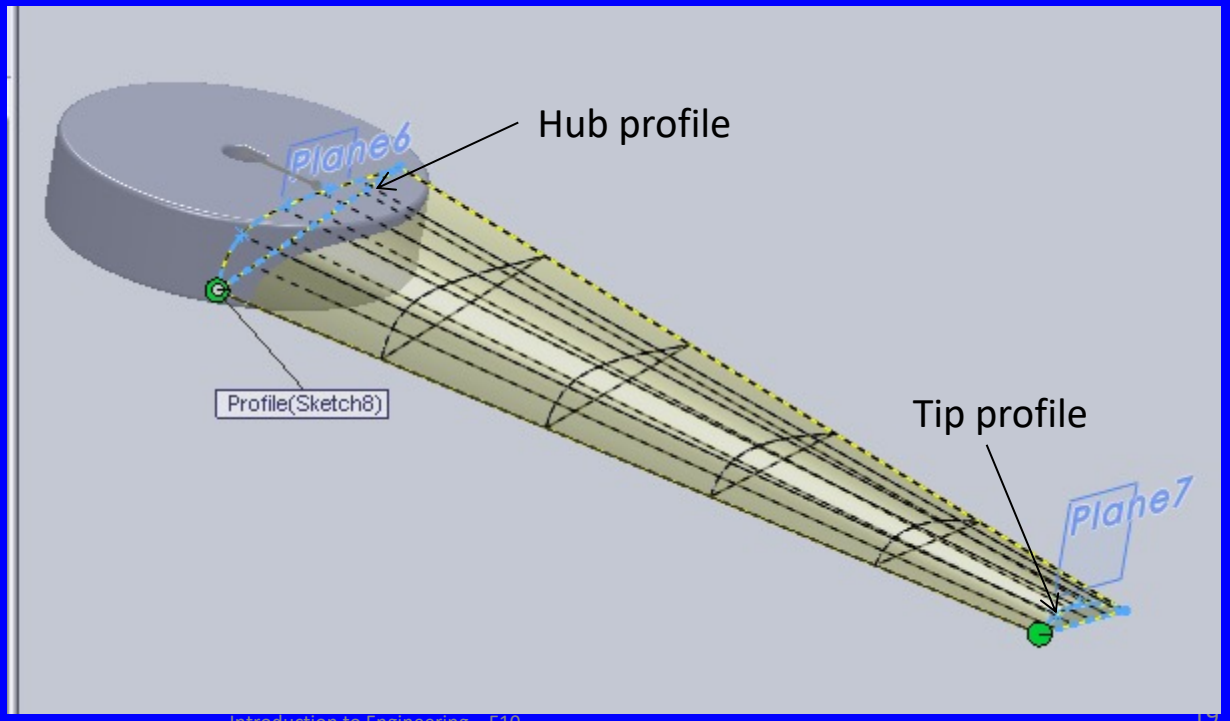


Creating the Blade - Loft

Select the ***Loft*** command and choose the two profile

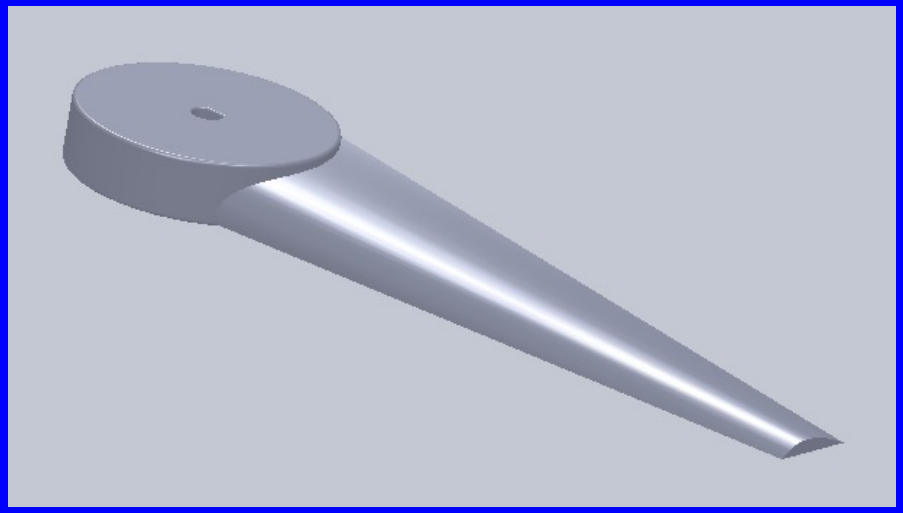




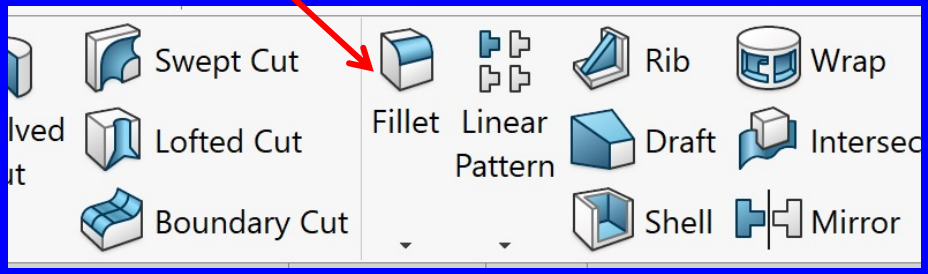


Rounding the Blade Edges – Fillet Command

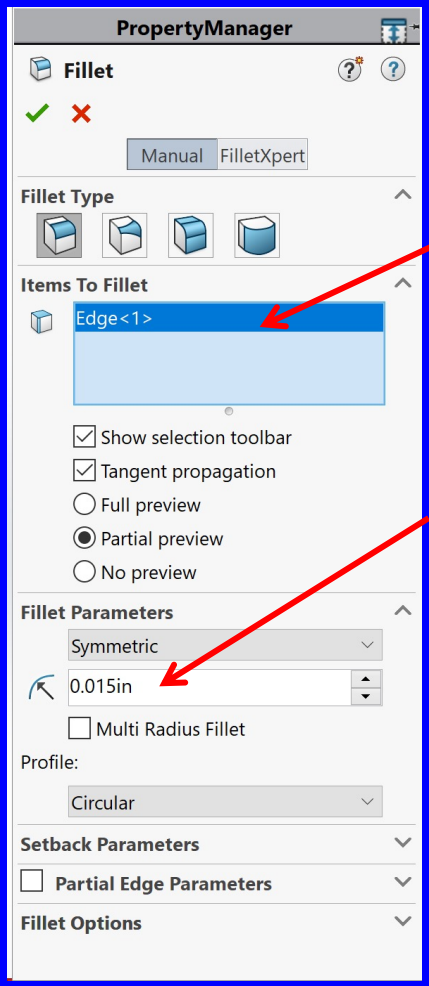
The blade edges should be rounded in order to obtain a model with clean edges. The 3D printer is not capable of modeling sharp edges.



Select the ***Fillet*** command, choose the edges and provide a radius for the fillet.

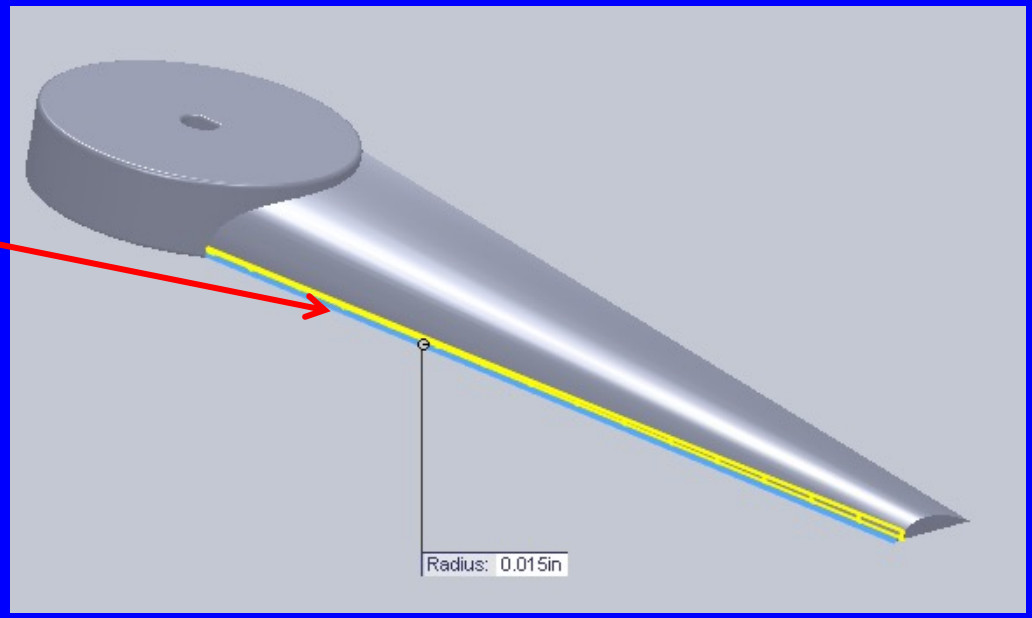


Fillet

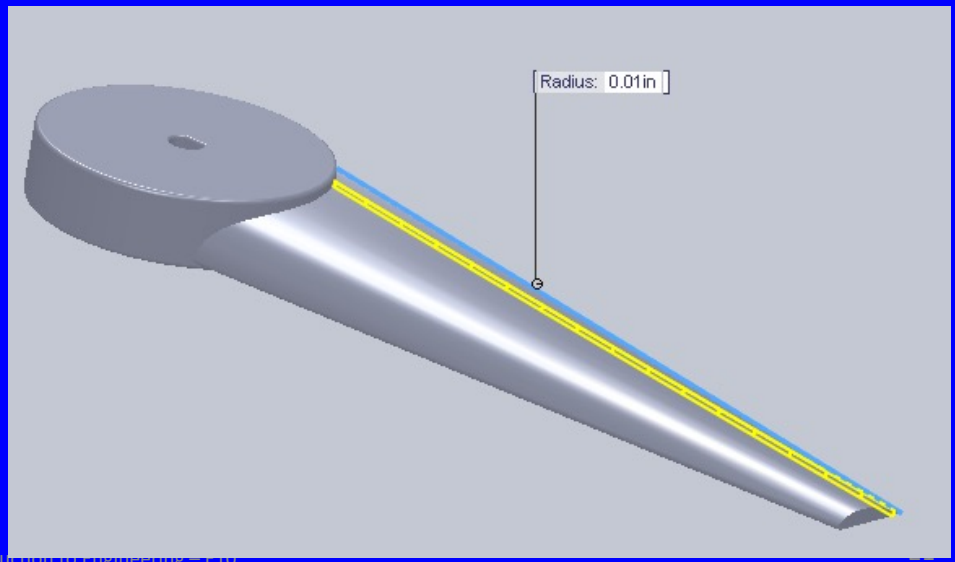


Select the front edge

Assign a value of **0.015 inch** to the **front edge**

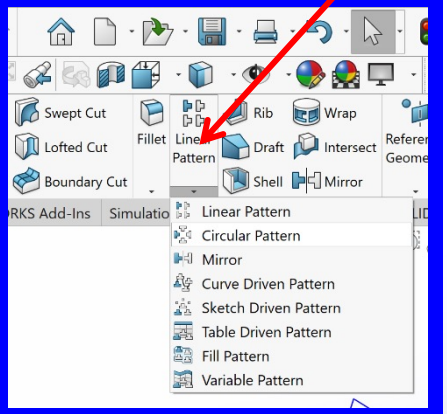


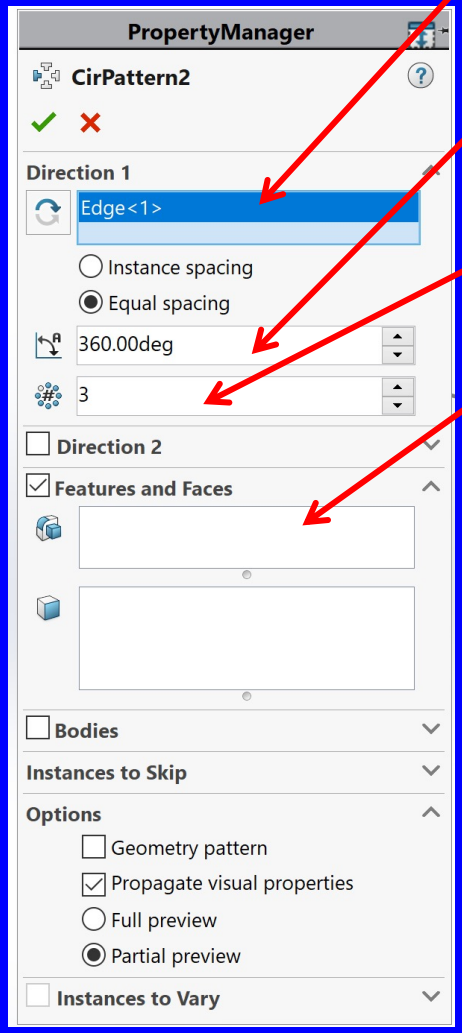
Select ***Fillet*** again and choose the **back edge** and assign a value of **0.01 inch**



Completing the Rotor – Pattern Command

Select the ***Circular Pattern*** command



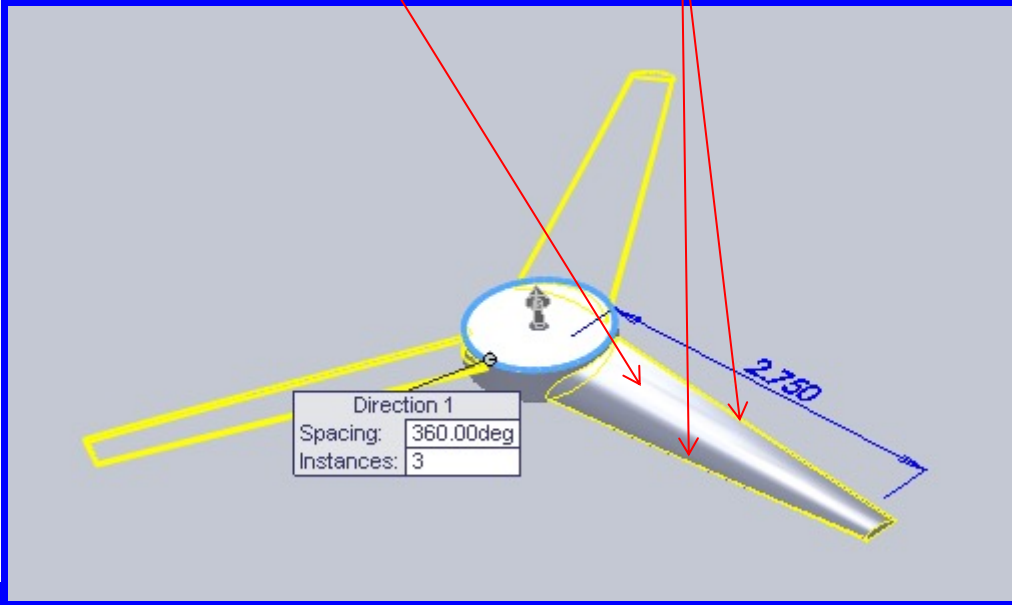


Select the axis of the circular pattern, choose the edge of the circle on the hub

Check the Equal spacing box for full 360 deg.

Choose the number of the blades

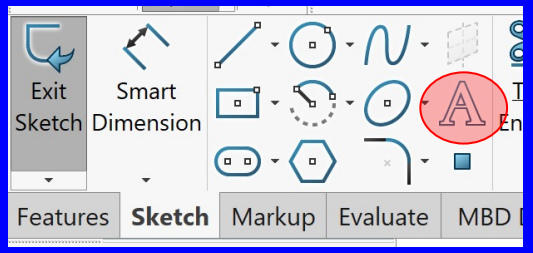
Select the blade and the fillets

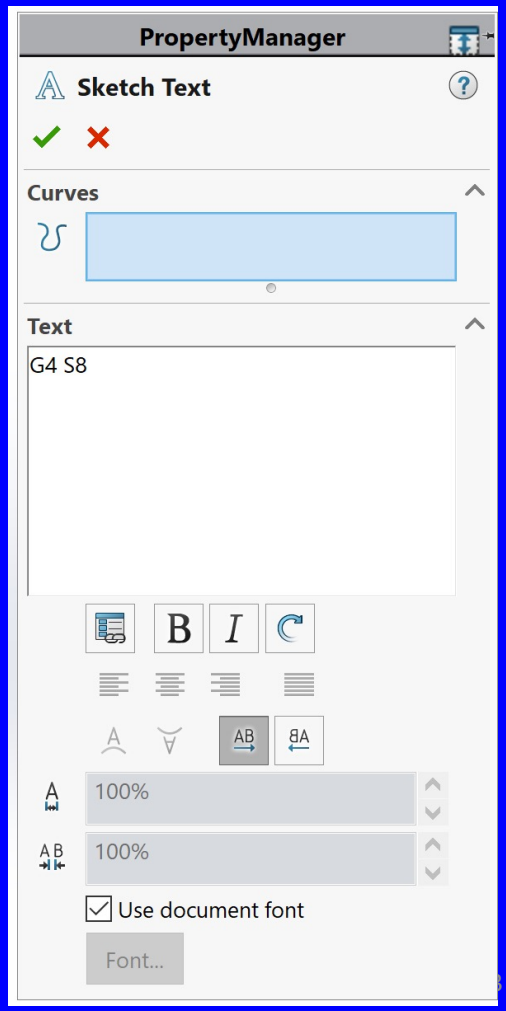


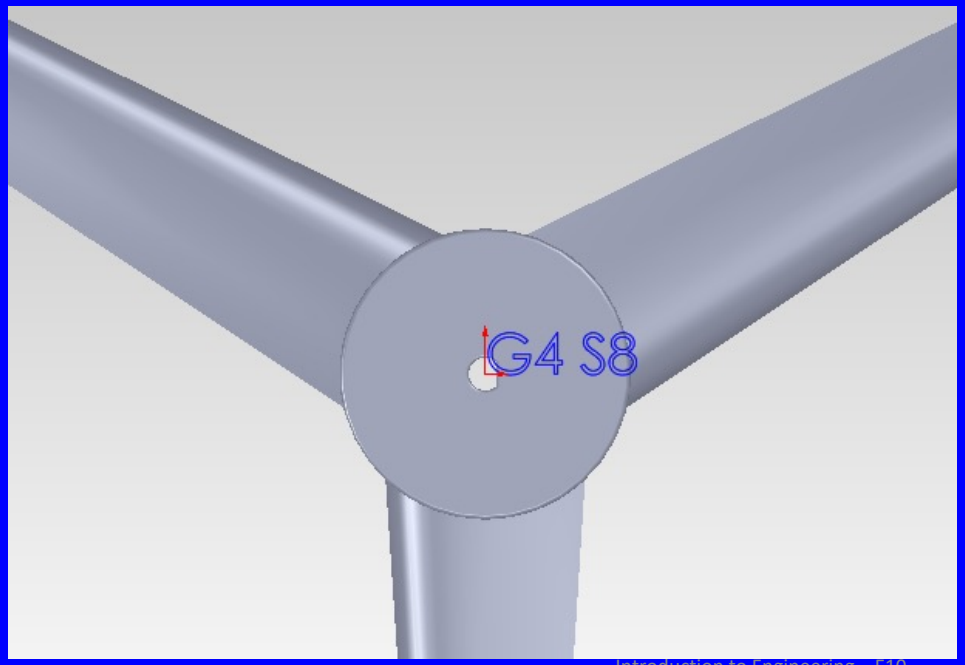
Embossing the Hub

In order to identify your blade it has to be embossed with your group number and lab. section.

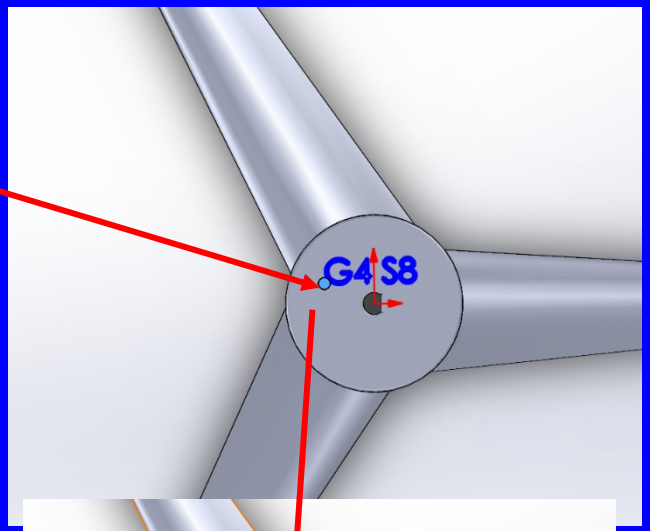
Select ***Sketch*** and choose the top surface of the hub. Use the ***Text*** command and write your group number (G4) and lab. section number (S8). Make sure the text is not too close to the hole.

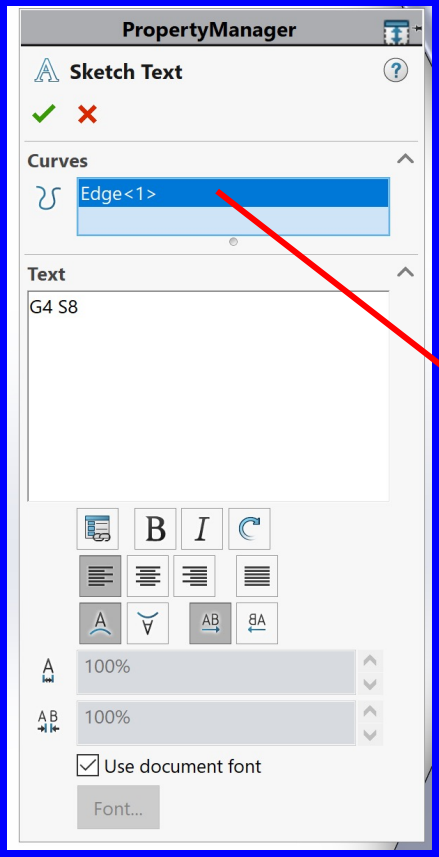




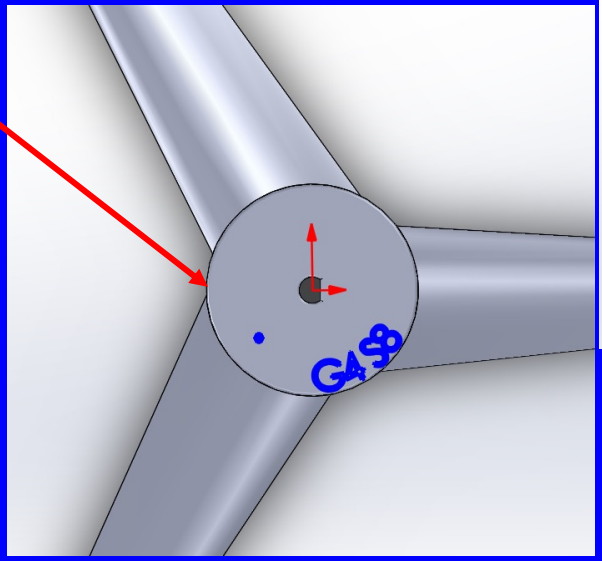


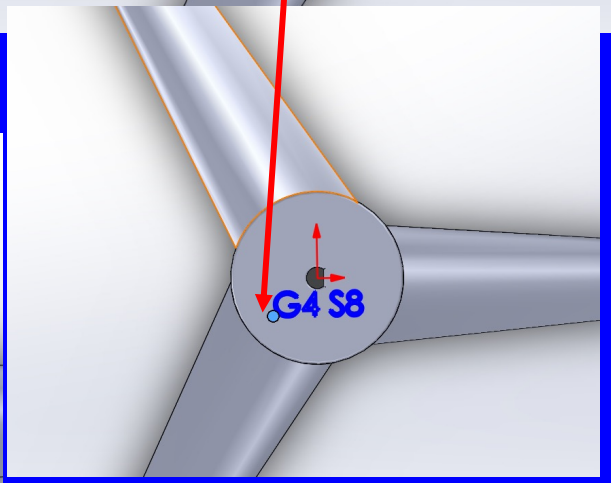
While in the sketch, the text can be moved, select the text, select the point , hold the left mouse key down and move the text



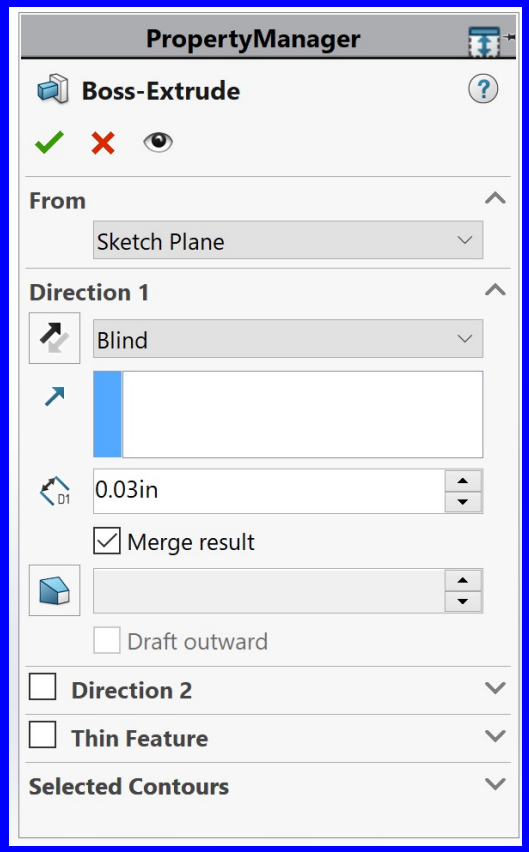


The text could follow any curve (edge of the circle)

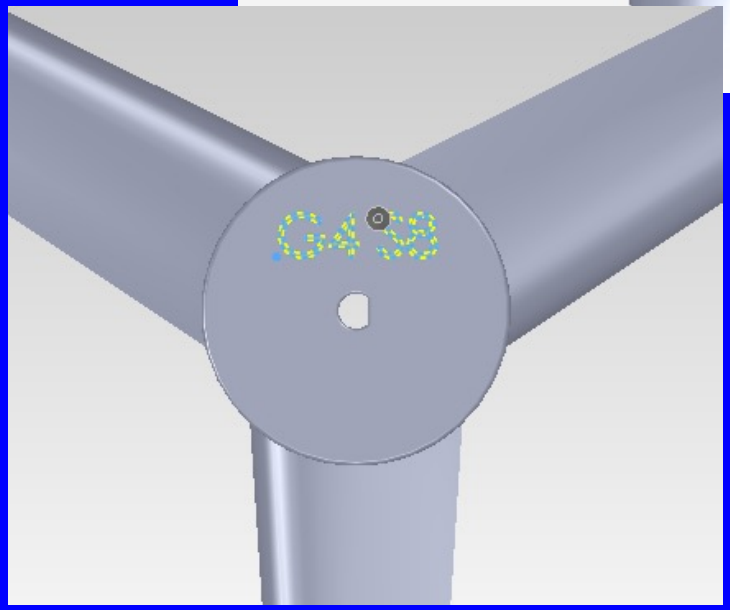




Embossing the Hub



Extrude the text by 0.025 to 0.03 inch



THE ROTOR