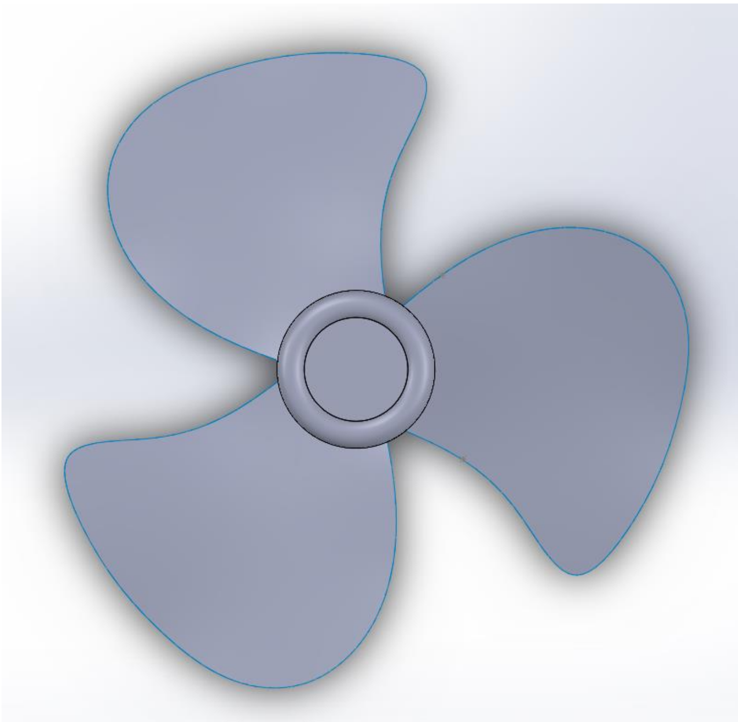
SOLID MODELING PROJECT: REVERSE ENGINEERING A PROPELLER



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OVERVIEW

For my solid modeling project, I attempted to reverse engineer a propeller on SolidWorks. This propeller was removed from a metallic rod of an electric fan, and there were two, major components of the object: the central hub and the 3 identical blades. This can be shown in the following figure:

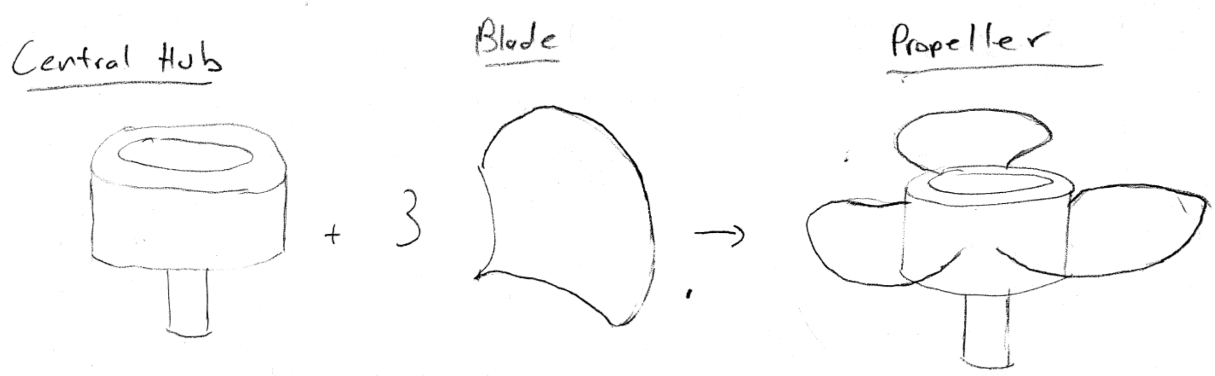


Figure 1: Major Components of Propeller

The central hub has the follow minor components: a radially symmetric shell, a cylinder with a hole, and 3 identical ridges. This can be shown in the following figure:

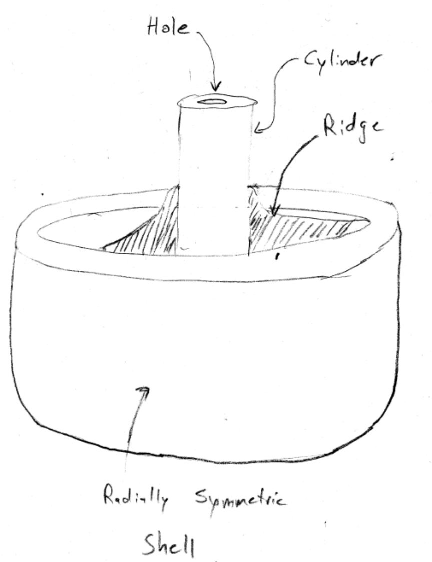


Figure 2: Minor Components of Central Hub

By observing the object, I noticed that the blades had to be almost identical to one another. Thus, I decided to only get dimensions of only one of the blades and then create a circular pattern on SolidWorks of the blade around the outer surface of the radially symmetric shell of the central hub.

For this project, I hope to increase my analytical skills of observing parts of the object and to learn how to reverse engineer them. I also hope to learn to use advanced SolidWorks features, such as creating surface lofts, boundary surface, and filling surfaces.

PROCEDURE

Measurements of Central Hub

To take accurate measurements of the central hub, I observed its top and bottom views in order to analyze and determine their measurements as accurately as possible. The follow images show the actual object labeled with the individual, minor components of the central hub:

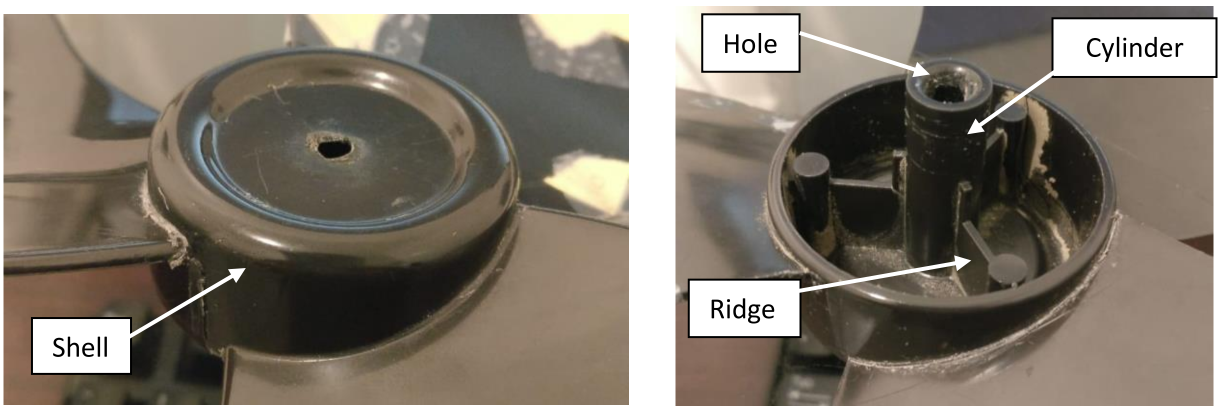


Figure 3: Top (left picture) and bottom (right picture) views of the central hub

The central hub comprises of a shell, a cylinder with a hole, and a ridge to support each of the 3 blades. I categorized one of the parts of the central hub as a cylinder because an external, metallic rod from a fan is supposed to go through the cylinder's hole, but not all the way through. When I found the propeller, the top surface was punctured. This puncture was probably created because the depth of the hole in the cylinder was too large; there was probably very little plastic material between the top surface of the central hub's shell and the bottom of the cylinder's hole. I neglected this puncture for this reason and did not plan to dimension nor model it on SolidWorks.

Measuring Different Heights of the Central Hub:

I began taking measurements by first getting the maximum height of the object. To get this overall distance, I added 2 heights: the distance from the top of the shell to the bottom of the shell and the distance from the bottom of the shell to the bottom of the cylinder. For the first distance, I used a digital caliper as shown:

Figure 4: Measuring distance between top and bottom shell

This distance was 20.52 mm. I measured the second distance (from the bottom of the shell to the bottom of the cylinder) using a ruler as shown in the following figure:

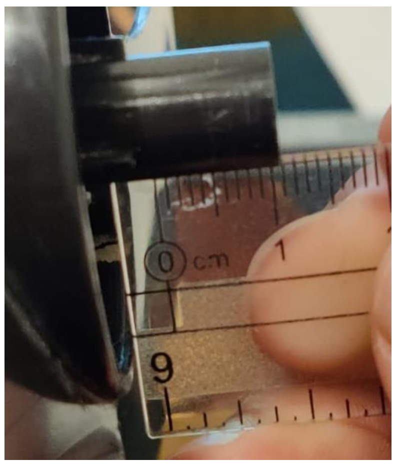


Figure 5: Measuring distance between bottom shell and bottom cylinder

The ruler showed 10.5 mm, but this had to offset by 4.0 mm because of the distance between the edge of the ruler and the 0 mm line mark. As a result, the distance between the bottom of the shell and the bottom of the cylinder was 14.5 mm. This gave an overall height of 35 mm. These measurements can be represented in the figure below:

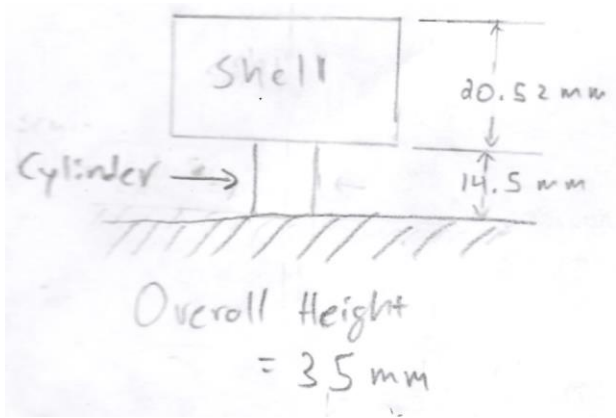


Figure 6: Schematic of height measurements

Measuring Dimension of Cylinder and Hole:

Because the cylinder's outer diameter was small, I decided to use a digital caliper and tried to get the largest chord possible by turning the digital caliper around the cylinder. The cylinder's diameter was approximately 10.55 mm. I then took measurements of the hole, which was not a circle. Instead, the hole had a cross section of an arc that was connected by a chord. Using a digital caliper, I calculated two different types of distances. One measurement was between a point from the outer surface of the cylinder to a point on the arc, which was 2.25 mm. The other measurement was from a point from the outer surface of the cylinder to a point on the chord, which was 3.11 mm. These dimensions can be illustrated in the following figure:

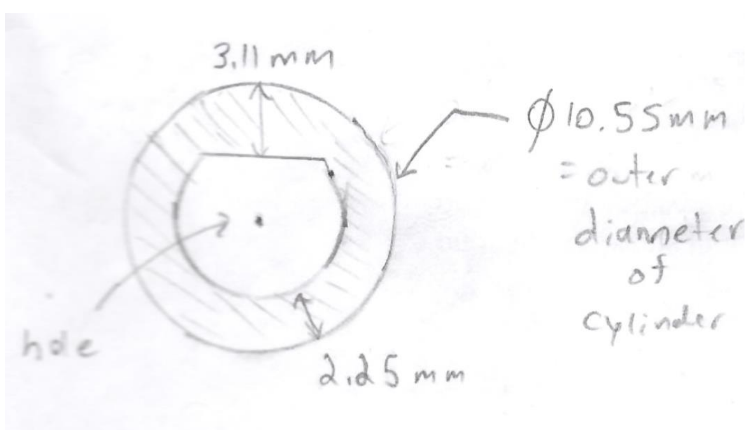


Figure 7: Schematic of cylinder and hole with dimensions

I did not take a measurement of the depth of the hole because I believe the hole's original depth was too deep; this probably allowed the metallic rod to puncture through the top surface of the central hub when the rod was inserted through the hole. On SolidWorks, I plan to make the depth of the hole 20 mm from the bottom of the cylinder.

Measuring the Shell's Diameter:

I then tried to get the diameter of the outer shell of the central hub. I initially wanted to use a digital caliper to measure its diameter, but I would risk taking measurements of a chord, which can be a length smaller than the circle's diameter. As a result, I placed the object on a flat piece of cardboard and tried to keep it centered. Using a straightedge, I aligned its vertical edge against the outer shell and the horizontal edge on the cardboard platform. I then used a pin to prick the point where the vertical edge met the platform. An example of this can be shown in the following figure:

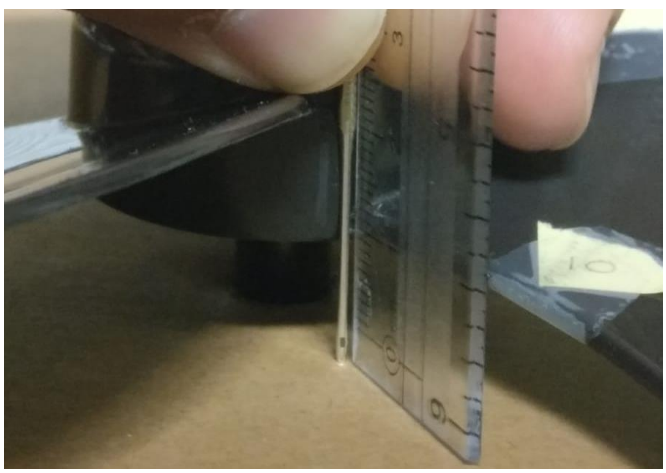


Figure 8: Using a needle to prick onto a piece of carboard to get a point of the shell's outer diameter

I repeated this process 2 more times. These 3 pricks were created to represent points that were projected from the outer surface of the shell. I labeled these points as x, y, and z, which can be shown in the following figure:

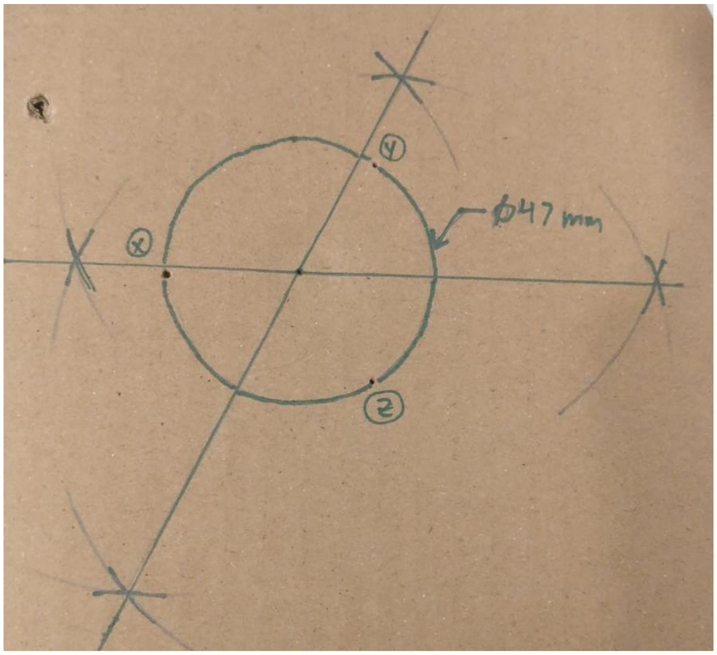


Figure 9: Determining the diameter of the circle of the outer shell

As shown on Figure 9, I used a compass to construct two sets of perpendicular bisectors: one bisector for the points x and y, and another bisector for the points x and z. The point where the 2 perpendicular bisectors met was the center of the circle. I used this center to then trace out the circle. I then measured the radius between the center and a point on the circle using a ruler. This distance was approximately 23.5 mm, which gave an overall diameter of 47 mm of the outer shell.

Measuring Shell Thickness:

After determining the heights and the dimensions of the cylinder, I took measurements of the dimensions of the central hub's shell's thickness. The shell had varying thicknesses because I believe it was plastically cast molded. As a result, I decided to take 3 different measurements around the bottom of the shell. This can be illustrated in the following figure:

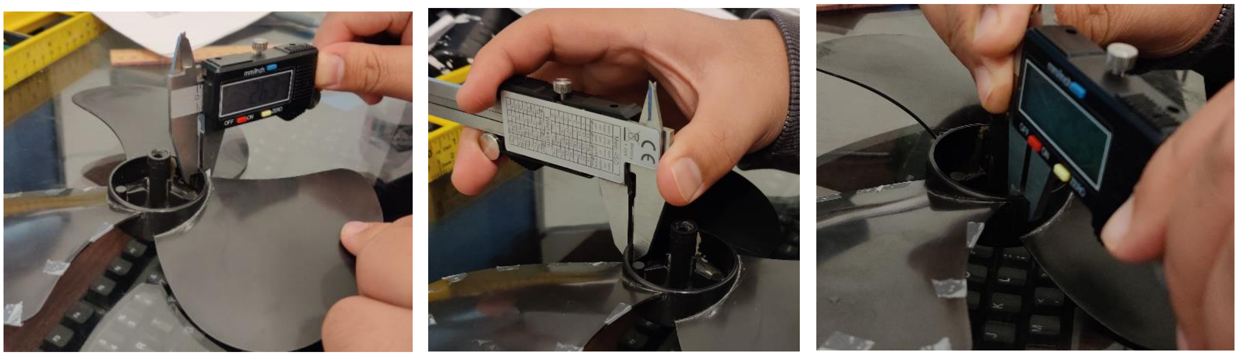


Figure 10: Taking 3 different measurement around the shell to get an average thickness.

The thickness measurements were: 2.40 mm, 2.42 mm, and 2.50 mm, which gave an average thickness of 2.44 mm.

Measurement of Protrusion on the Shell's Top Surface:

The top surface of the shell of the central hub was not completely flat. In fact, there was a protrusion from the flat part of the surface as shown in the figure below:

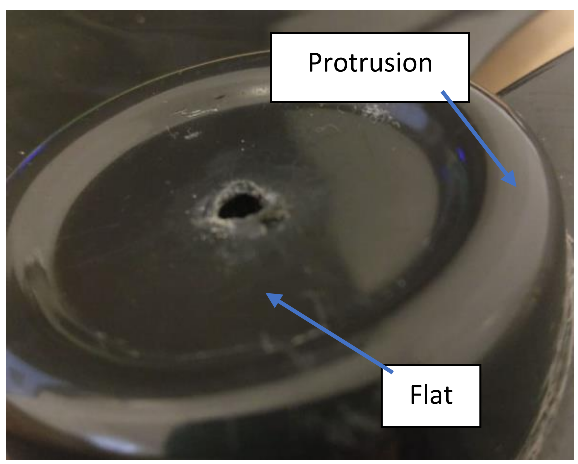


Figure 11: Image of the top surface of the shell of the central hub

To simplify the reverse engineered design and by looking at the shape of the protrusion, I considered the top surface to have an outer arc. This idea can be illustrated in the following figure:

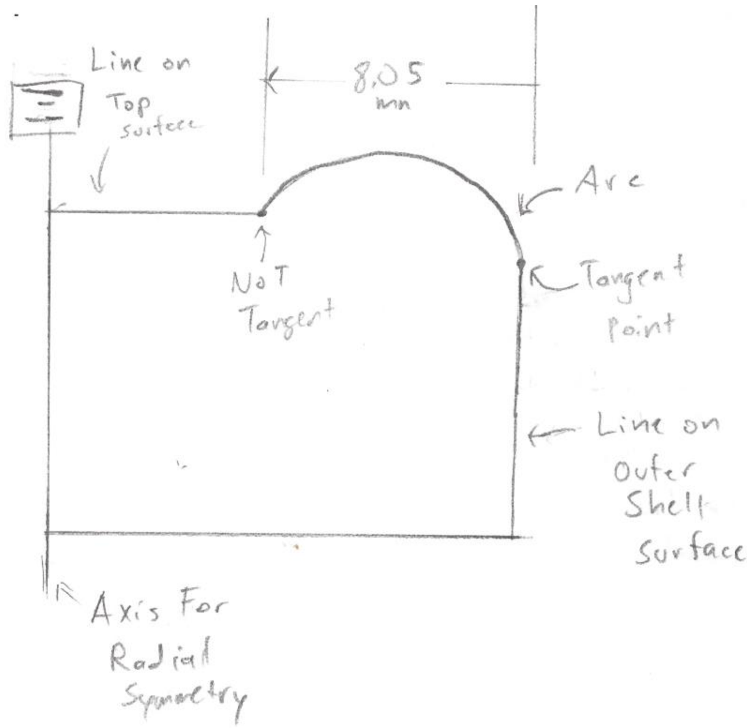


Figure 12: Schematic front view of shell showing dimensions and relations for the arc

This arc did not seem tangent to the top, flat surface because when I felt the groove between the surface and the curved surface, it was not smooth. However, I knew this arc had to be tangent to a vertical line on the surface of the outer shell because the groove connecting the curved surface and the outer shell's surface was smooth. I also knew that I can determine the distance between the endpoints of the arc by using a caliper, as shown in the following figure:

Figure 13: Measuring the distance between endpoints of the arc

The horizontal distance between the endpoints of the arc was approximately 8.05 mm, which was shown in Figure 12.

Because the shell and the cylinder had radial symmetry, I drew a sketch by hand that I used as a guide to replicate on SolidWorks by revolving a digital version of this sketch about an axis of revolution to create the central hub's shell and cylinder. The shell had a specified thickness, which meant that the outline of the sketch representing the outer surface had to be offset by the thickness of the shell. This can be represented by the following hand drawn sketch showing a summary of all the measurements I had taken:

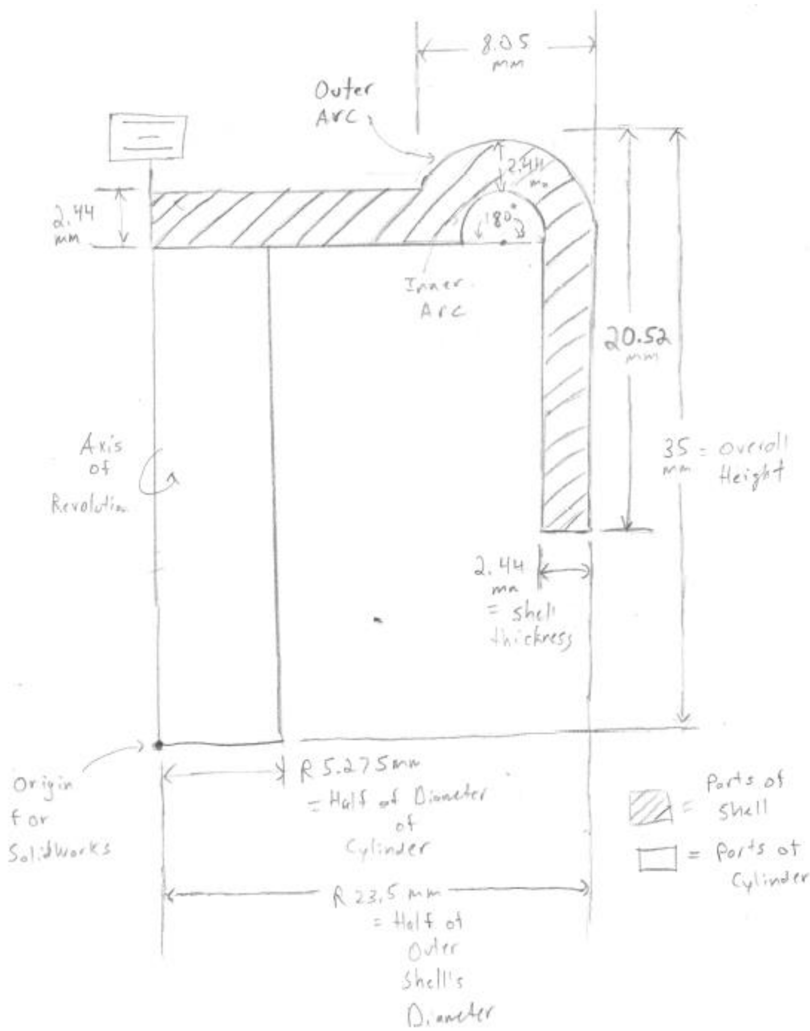


Figure 14: Hand-Drawn sketch to replicate on SolidWorks

It was hard to determine the angles of the outer and inner arcs (as shown in the sketch from the previous figure) from the actual object. To simplify the design, I decided to make the inner arc 180° and concentric with the outer arc. The outer arc was fully defined because its end and center points were fixed.

My Thoughts on the Ridges:

After inspecting on how the ridges of the propeller were connected, I saw that it was complicated to reverse engineer them. Here is a magnified image of one of the 3 ridges on the propeller:

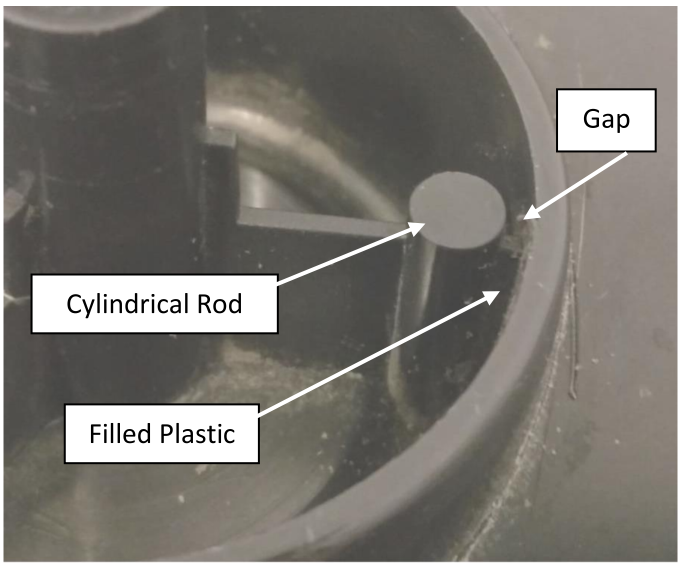


Figure 15: Magnified photograph of ridge

As shown in the above photo, there was a gap between top of the cylindrical rod shown in the previous photograph and the inner surface of the central hub's shell. I also saw some plastic material that was filled below the gap and between the cylindrical rod and the inner surface of the shell. Thus, I decided it was too complicated to reverse engineer these ridges and created my own ridges on SolidWorks to connect the cylinder to the inner surface of the shell. These ridges were needed in order to provide support between the cylinder and the shell of the central hub.

Measurements of Blade

As mentioned previously, I planned to only create one reverse engineered model of the blade and make a circular pattern of this blade three times around the cylindrical surface of the outer shell.

After looking at the original model of the propeller, I decided that one way of creating a SolidWorks model of the object was by determining the dimensions of different cross sections on the blade of the propeller that I later lofted on SolidWorks using guide curves. The following figure explains my rough idea of how I planned to make the blade on SolidWorks:

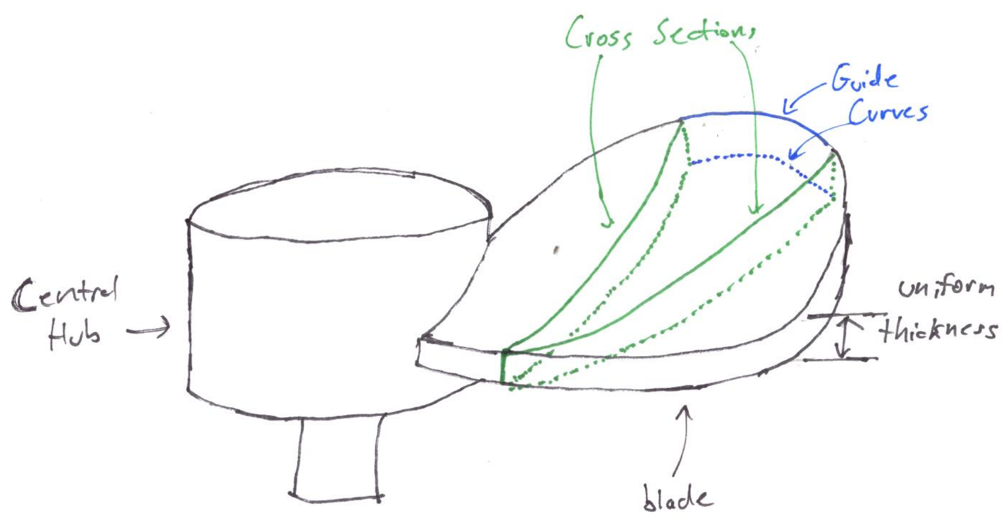


Figure 16: Rough idea of creating blade on SolidWorks

Getting Average Thickness of Blade:

I decided that the blade needed a uniform thickness because I believe the original propeller was plastically cast molded and had varying thicknesses between the blades top and bottom surfaces. Also, having a uniform thickness would help simplify the reverse engineered model of the blade. I planned on getting an average thickness by taking the average of multiple distances; each distance would be the measurement between a point on the top surface of the blade and a point vertically below on the bottom surface of the blade. The following figure shows an image of measuring this distance:

Figure 17: Measuring thickness between 2 points on the blade

The following are the dimensions of the different thicknesses I measured: 1.98 mm, 1.84 mm, 1.67 mm, 1.43 mm, 1.76, 1.67, 2.03, 2.19, 2.14, and 1.44 mm. Thus, I determined an average thickness of the blade to be approximately 1.82 mm.

Picking Points on the Blade:

To get cross section of the blade to loft on SolidWorks, I picked points on the edge of the top surface of one of the blades and used a contour gauge to get the curves connecting some of these points. This approach can be illustrated as follows:

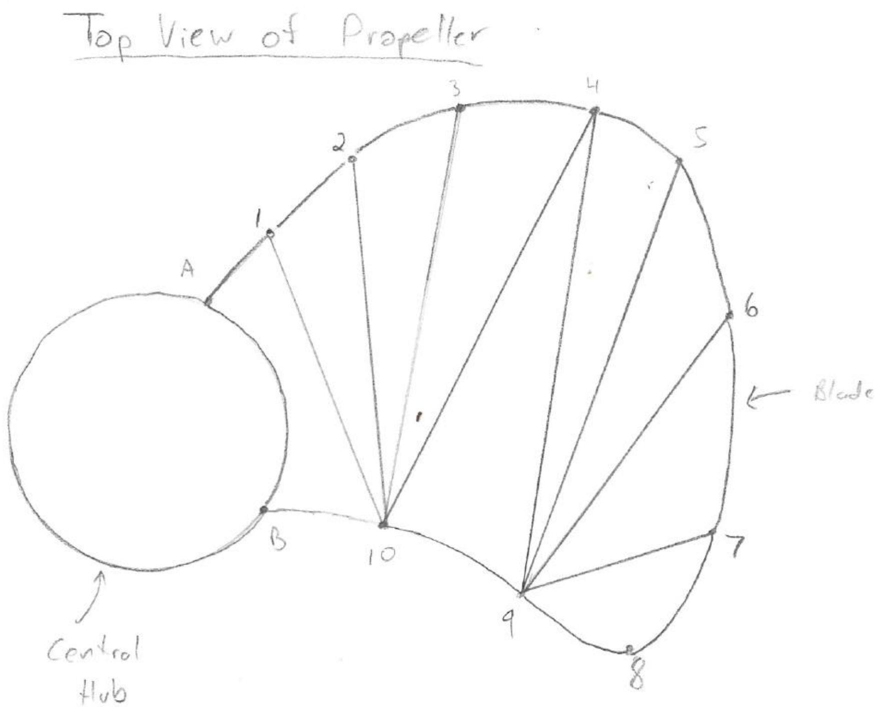


Figure 18: Schematic of points and contours on propeller

As shown in the previous schematic, I chose 12 points on the propeller. A and B were points that were both on the edge of the top surface of the fan blade and on the central hub. Points 1 through 10 were only located on the edge of the top surface of the fan blade. Because a contour is defined as a curve between two of its endpoints on a plane, I used the following notation to represent the contours I had sketched: 10-1, 10-2, 10-3, 10-4, 9-4, 9-5, 9-6, and 9-7. Note that 10-1 and 1-10 both represent the same contour. These contours were represented as the lines shown in the previous schematic because the drawing was a top view of the propeller

To simplify my explanation for the Measurement section of the report, I will call the bottom plane of my object as the flat surface of the bottom of the cylinder. Each of the contours that I sketched lid on a plane is perpendicular to the bottom plane and that passes through the contour's endpoints.

I represented points 1 through 10 on my physical object by ripping out the corners of post-it notes and letting those corners point at specific points on the edge of the top surface of the fan blade. Points A and B are both on the surface of the top of the fan blade and on the surface of the central hub. This can be shown in the following figure:

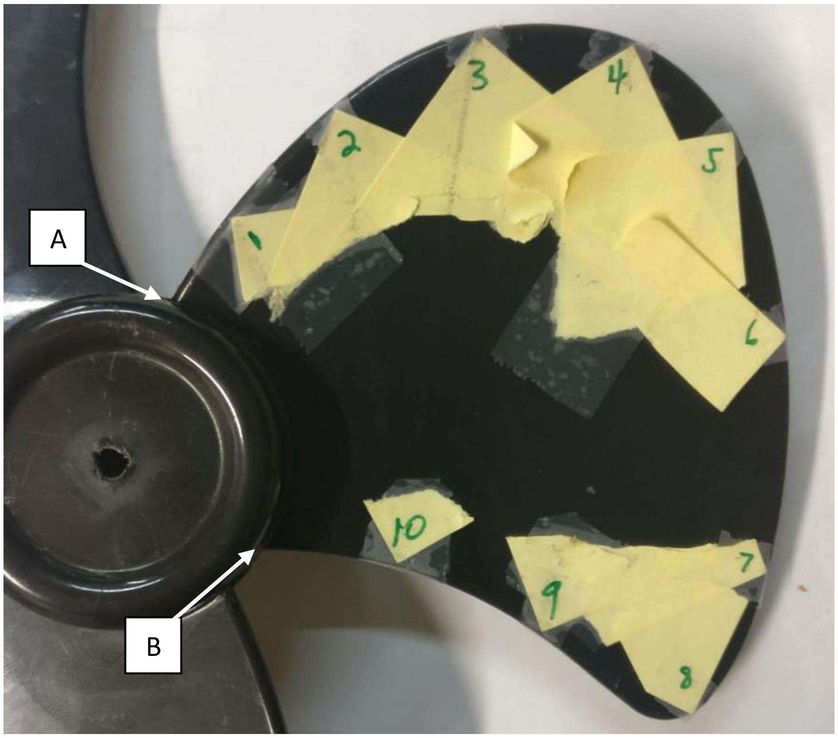


Figure 19: Photo of labeled points using sticky notes on the blade of the propeller

Because I was scared the adhesion of the sticky notes wasn't strong enough, I also taped the post-its onto the blade. I didn't want to use a marker to mark points on the blade because I was afraid that they would rub off; it would also be hard to see the color of the marks because the blade was painted black

I also planned on getting the approximate coordinates of these points. These points were placed with respect to the origin, which was a point fixed on the bottom plane and located along the radially symmetric axis of the central hub. If this was confusing to understand, this origin was also shown on Figure 14. Because SolidWorks' default axis for height is the y-axis (as shown in the following figure), I wanted to keep the same consistency.

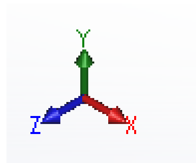


Figure 20: Default axes representation on SolidWorks

Thus, I represented the height for each of the 12 points as the y-coordinate.

Failed Attempt to Get Cylindrical Coordinates (r & θ):

I initially wanted to get cylindrical coordinates for the 12 points I've picked along the blade. To get these coordinates, I found a block of wood and hammered a piece of sturdy cardboard onto it. I placed a screw through the opening of the propeller and used an electric screw driver to fix it in place. Because the propeller would still rotate, I used two additional screws to fix the entire object on the platform. This can be illustrated in the following figure:

Figure 21: Photo of fixed propeller on platform

I then began to plot points on the cardboard by using a pin, as shown in the following figure:

Figure 22: Photo of attempt to get the points representing “r” and “θ” coordinates

By using a ruler, I also measured the perpendicular distance from the cardboard platform, which represents the bottom plane of my object, to each point on the top surface of the blade, as illustrated in the following figure:

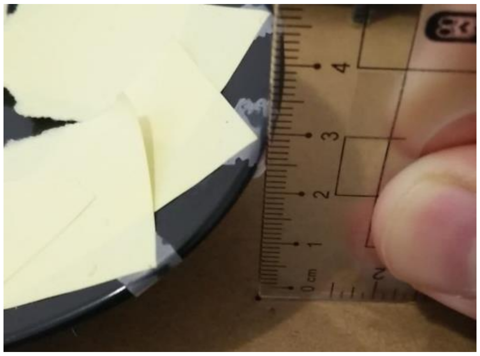


Figure 23: Photo of attempt to get heights for “y” coordinate

However, after removing the fan blade, I realize two problems. By using a screw to prevent the propeller from moving up and down, too much pressure was applied to push the propeller onto the cardboard; this created a dent on the surface of the cardboard, as shown in the following figure:

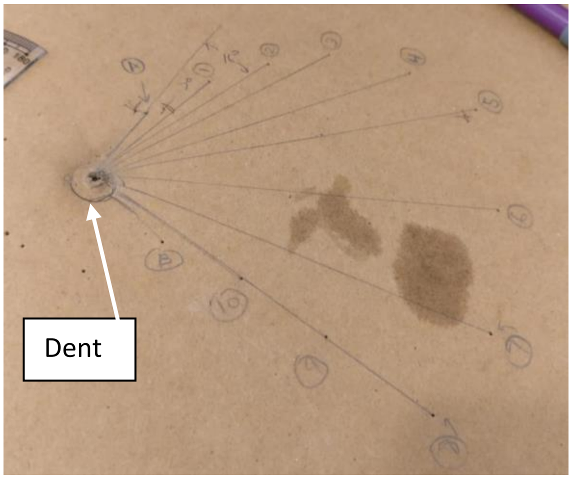


Figure 24: Photo of dent on cardboard surface

I knew this created inaccuracies for the measurements of the heights of points A, B, and 1 through 10 from the cardboard surface. As a result, I had to disregard these measurements.

Another problem that I discovered was using θ as a cylindrical coordinate. To explain this issue, I called the origin as point O. The difference of angles AO2 and AO1 were very small. Additionally, angles BO10, BO9, and BO8 all represent the same angle. I did not trust this measurement process of getting different angle measurements for cylindrical coordinates and decided to not use cylindrical coordinates on SolidWorks whatsoever.

Tracing the Contours:

Even though using the wooden platform wasn't a good idea to get the cylindrical coordinates, I was still able to get the contours. Even though there was a dent in the cardboard, this did not have any effect when determining the contours between the points I needed. The following figure shows how I determined these contours:

Figure 25: Getting the contour between 2 points on the blade

As I used the contour gauge, I made sure that metallic pins on the gauge were normal to the surface of the cardboard. This was important because I wanted to get a contour for the plane that is perpendicular to the surface of the cardboard, which is on the same plane as the surface of the bottom of the cylinder of the central hub. When I create the contour on SolidWorks using the spline tool, the sketch will be on a plane that is fully defined by being perpendicular to the bottom plane of the object and the endpoints of the contour.

After getting a contour between two points on the blade, I used a pencil to trace the physical representation of the contour on the gauge onto a piece of paper to scan. I made sure that I correctly labeled the end points of the contour with the respective points that I used for the blade. The following figure shows the scans of the contours I have traced out in pencil:

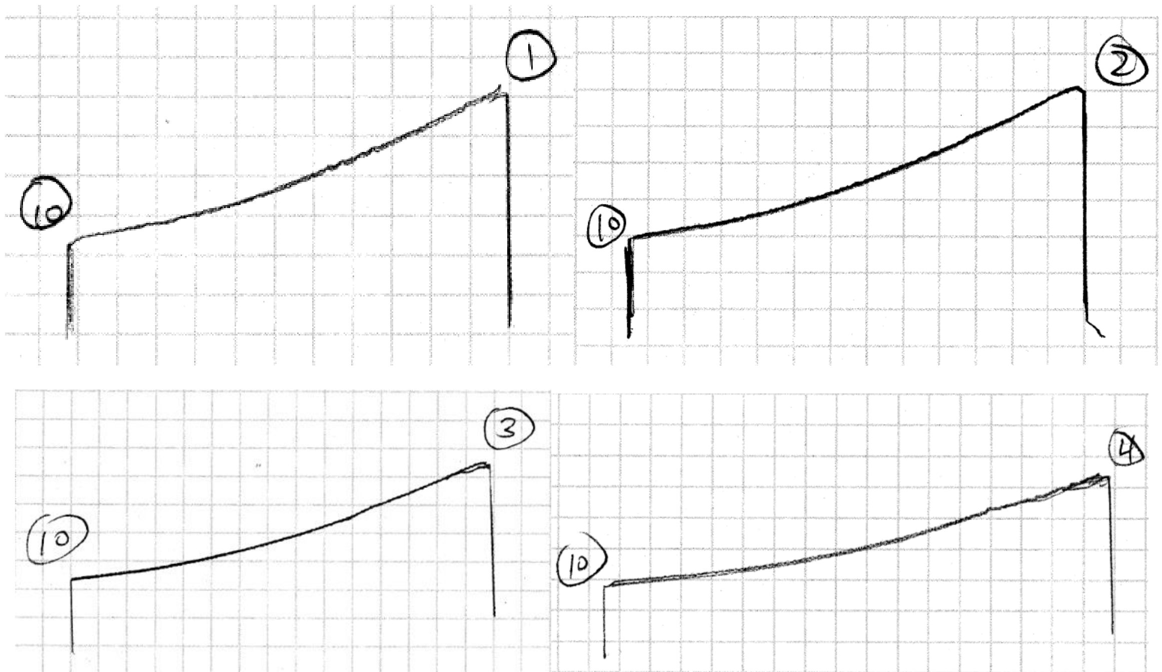


Figure 26: Contours 1-10, 2-10, 3-10, and 4-10

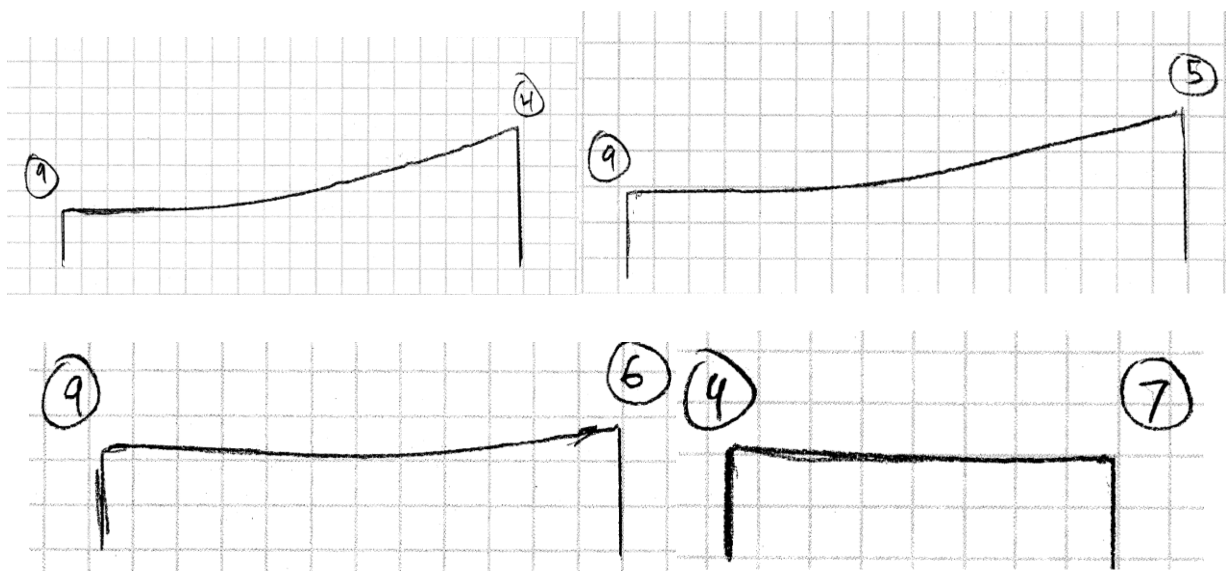


Figure 27: Contours 4-9, 5-9, 6-9, and 7-9

I did not need to put a scale when scanning these contours because each of the contours' endpoints were fixed. When I imported these images to SolidWorks, the points A, B, and 1 through 10 represented on SolidWorks were already fixed, and I positioned each of these images on the plane containing the set of the two points so that the endpoints of the contour would match with the set of points. This idea can be further explained in the modeling section of this report.

Measuring the Height for Each Point on the Top Surface of the Blade

Instead of using the platform with the wood and cardboard to get height measurements, I removed the cardboard and the screws from the previous measurement setup. I then re-screwed a screw through a plastic bottle cap. The screw went through the puncture holed of the central hub to fix the fan blade in place. The cap was used so that an extra force applied by the screw would deform the cap first, since the cap was made of more pliable plastic than the plastic material of the propeller. This can be shown in the following figure:

Figure 28: Photo of new setup of propeller

By using this setup, I then used a ruler to determine the heights of the points on the blade from the wooden platform, as shown in the following figure:

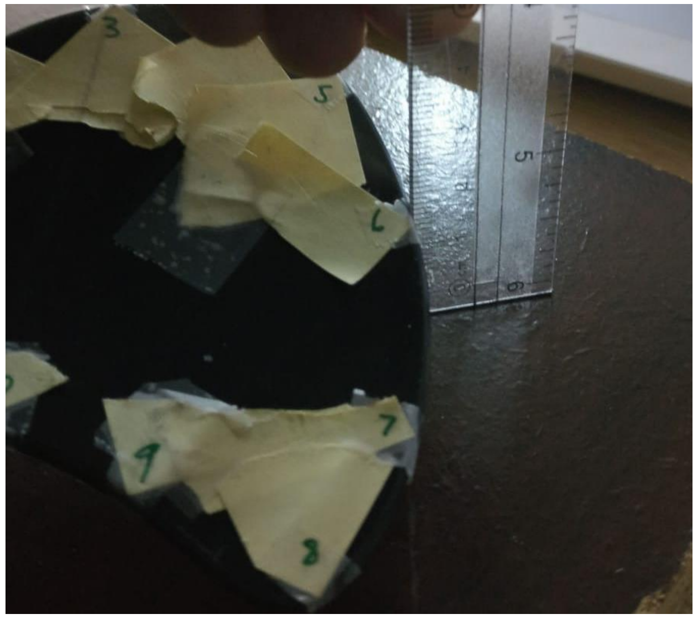


Figure 29: Measuring height of points on blade

As mentioned previously, the dimensions of the height of the points from the wooden platform had to be offset by 4.0 mm because of the distance between the edge of the ruler and the 0mm line mark. The following table shows the dimensions of the height for each point from the wooden surface of the platform:

Points	Heights (mm)
A	30
10	15.5
9	13.5
8	9
7	11
6	14
5	24
4	31

3	36
2	35
1	33
B	19

Table 1: Heights (y-coordinates) of the 12 points

Getting x and z Coordinates of the 12 Points Instead

To get the x and z coordinates of the 12 points, I tried to get a very accurate, leveled photograph of the object by doing the following setup:

Figure 30: Setup of getting photograph of propeller

I used a leveling app on my sister's phone and put napkins underneath it to level the phone as much as possible. I repositioned the ruler and took a photograph of my object, as shown in the following figure:

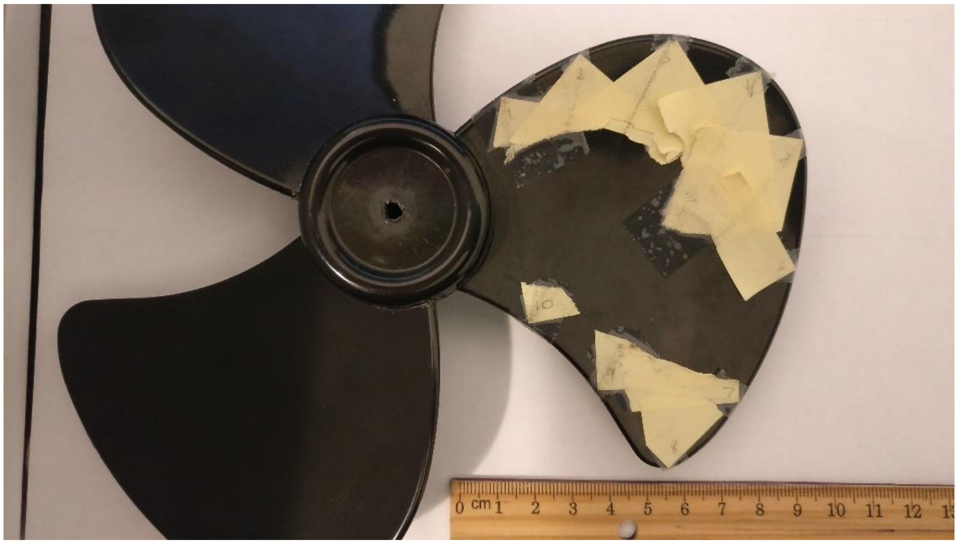


Figure 31: Leveled photograph of propeller

I planned to import this photo onto the bottom plane of my object on SolidWorks to get the x and z coordinates of points A, B, and 1 through 10. I discussed how I used this photograph to create the 3D model of the blade on SolidWorks in the Modeling section of this report.

MODELING

MODELING THE CENTRAL HUB:

I began to create my 3D model of the propeller on SolidWorks by creating the central hub component of the propeller. Using the hand-drawn sketch shown in Figure 14 as a guide, I created the following sketch on SolidWorks using lines and arcs on the Front Plane:

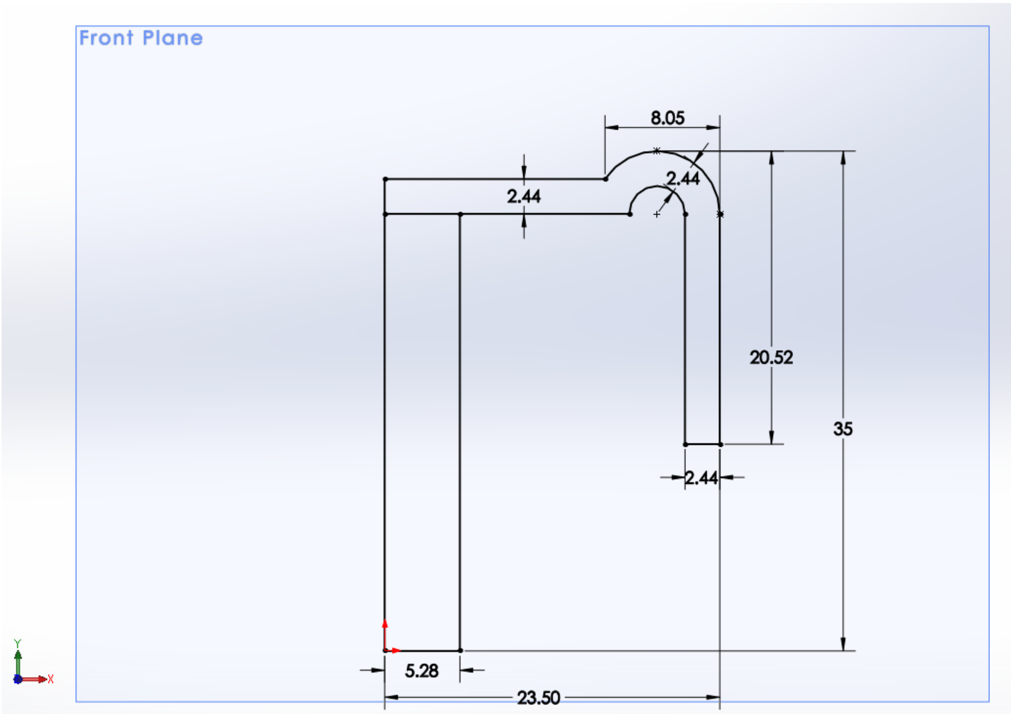


Figure 32: Heights (y-coordinates) of the 12 points

I chose my origin as the center point of the circular cross section on the bottom of the cylinder. I made the two arcs concentric and the thickness of the shell 2.44 mm. I created a 180° arc by making its the end points of the arc horizontal; the top and bottom arcs concentric.

Using the Revolve feature, I then revolved this sketch about the y-axis, which passes through the origin:

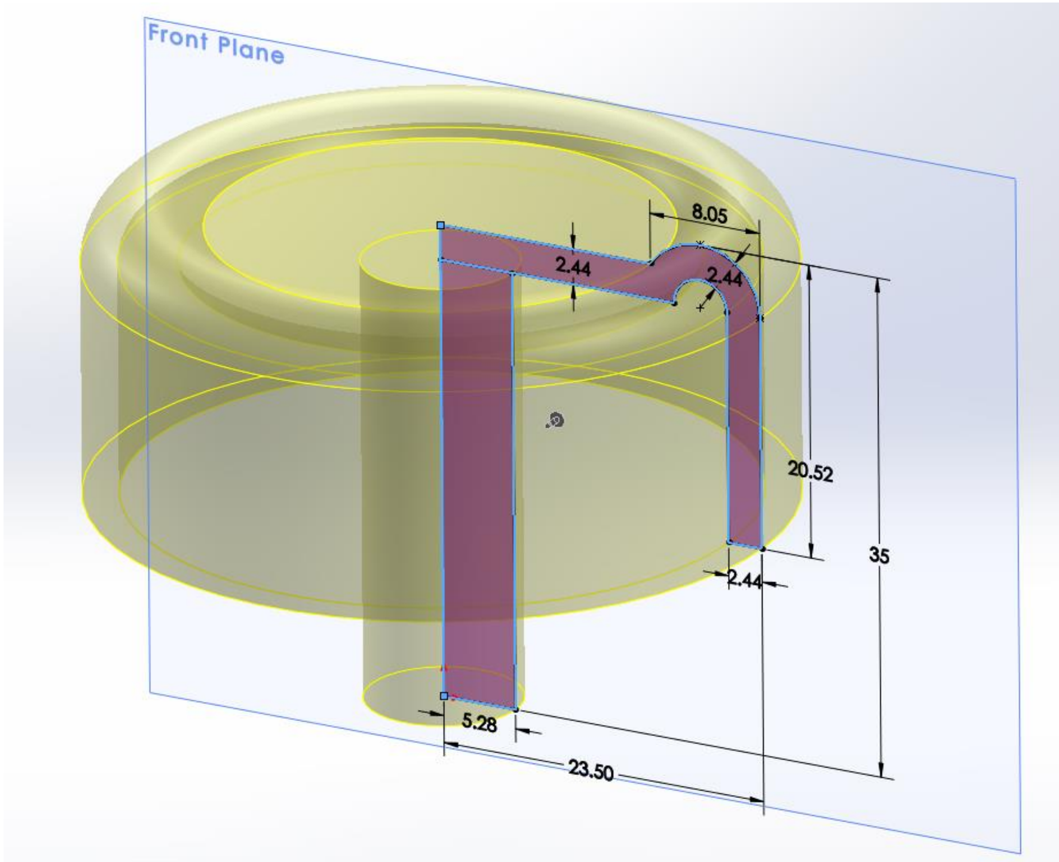


Figure 33: Revolving Sketch about y-axis

This gave the following 3D model:

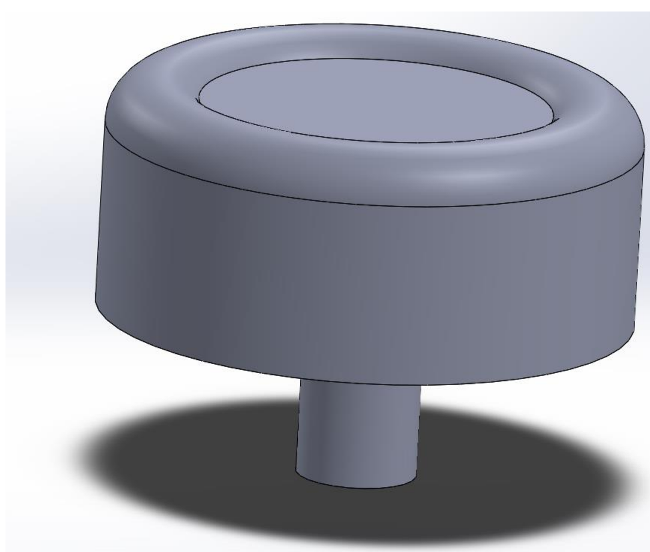


Figure 34: Model after Revolving

I then created the hole in the cylinder by first creating the sketch on the bottom surface of the cylinder as shown in the following figure:

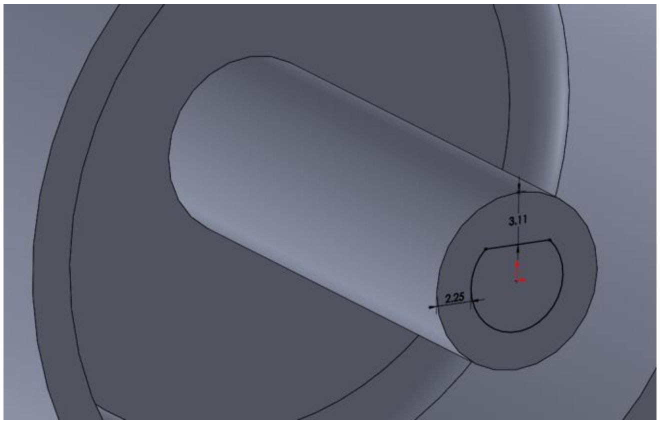


Figure 35: Sketch for hole on bottom surface of cylinder

I then used the Extruded Cut feature to create the hole with a depth of 20 mm:

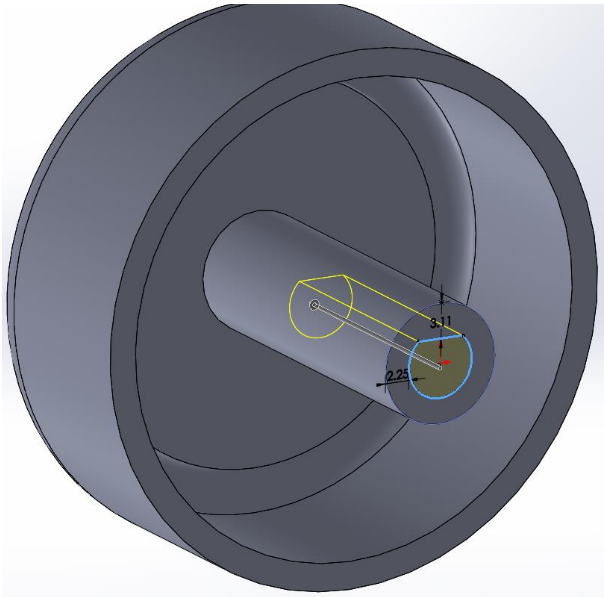


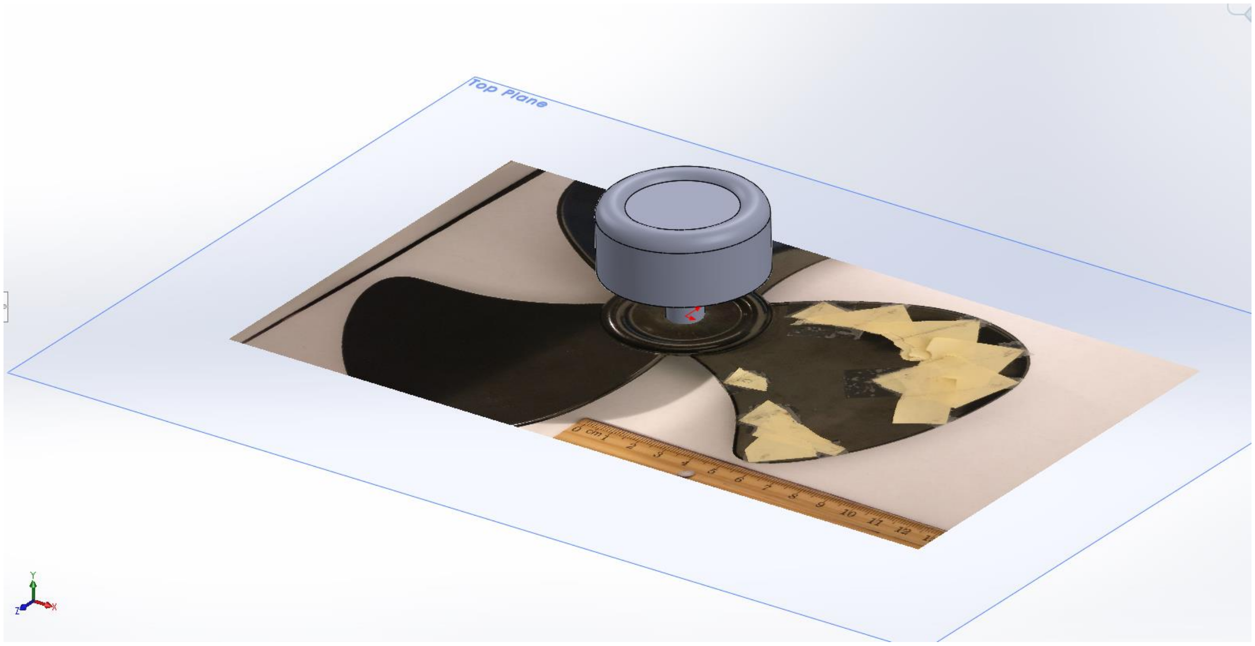
Figure 36: Making the hole inside the cylinder

Thus, the central hub was complete.

<u>MODELING THE BLADE</u>:

<u>Importing Leveled Photograph</u>:

I began modeling the blade by importing the image of the leveled photograph of the propeller onto a sketch located on the top plane. The top plane was the same plane as the flat surface of the bottom of the cylinder. This can be shown in the following figure:



<u>Figure 37</u>: Importing leveled photograph onto top plane

Putting the image on a separate sketch allowed me to make my work easier by having the ability to turn its visibility on when I needed it and off when I didn't need it. I then centered this image so that the image of the outer surface of the central hub lines up with the circular edge of the modeled central hub:

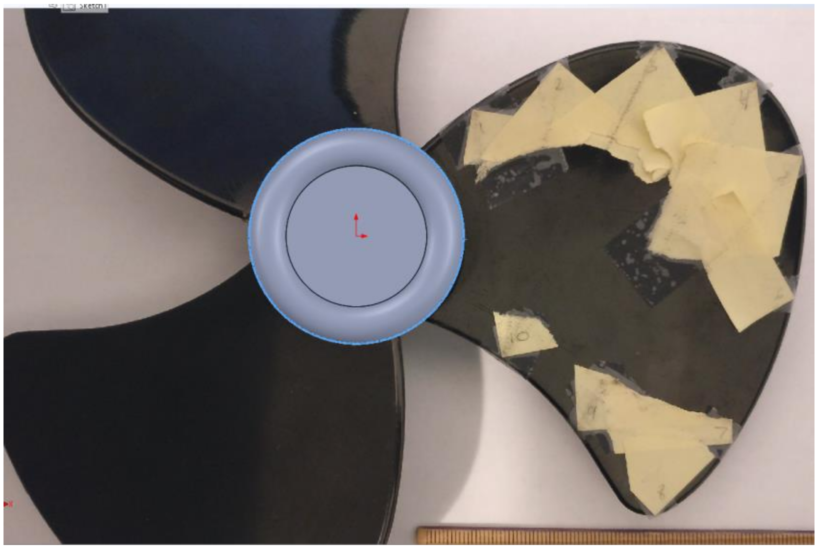


Figure 38: Centering leveled photograph

I realized that using the ruler from the image as a scale was not necessary, since I already aligned the image with the 3D modeled central hub.

Creating Projection Curve Containing Points A and B:

I created a 3D sketch and positioned points A and B with perpendicular distances of 19 mm and 30 mm, respectively, from the top plane. I added a relation that allowed both points to lie on the surface of the cylindrical shell:

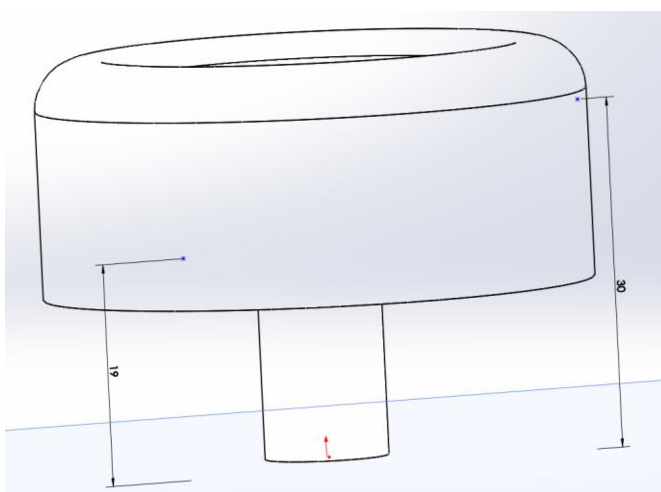


Figure 39: Creating points A and B

I then repositioned the x and z coordinates for both points A and B by using the leveled image I imported into the 3D model:

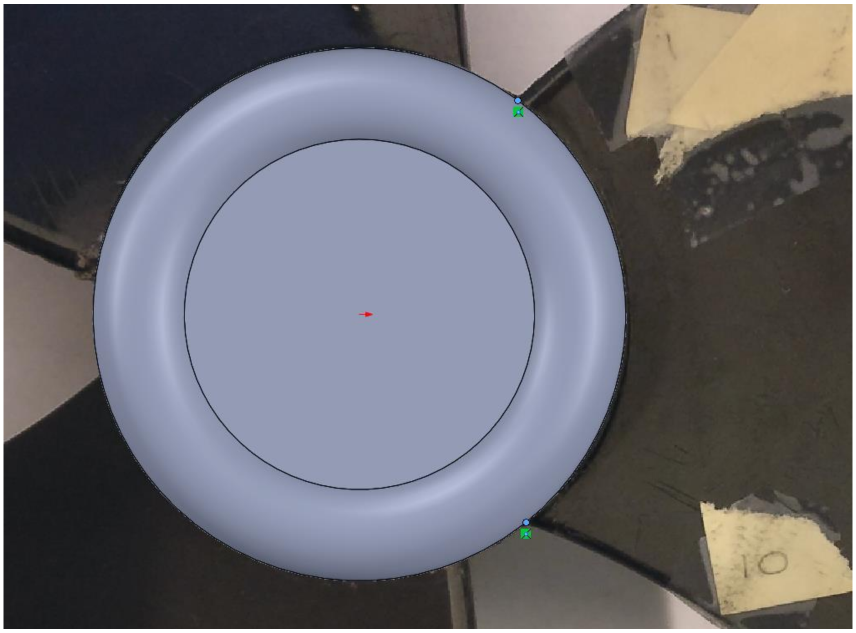


Figure 40: Positioning x and z coordinates for points A and B

I then created a plane that intersects these two points and is perpendicular to the top plane:

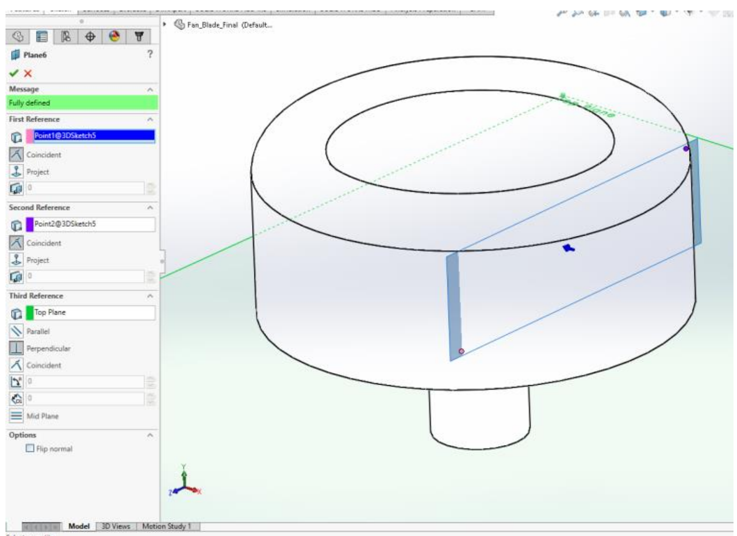


Figure 41: Creating a plane between points and perpendicular to the top plane

I then created a sketch on this plane that contains points A and B by connecting a line between them. I then created a parallelogram with a height of 1.82 mm. This can be shown in the following image:

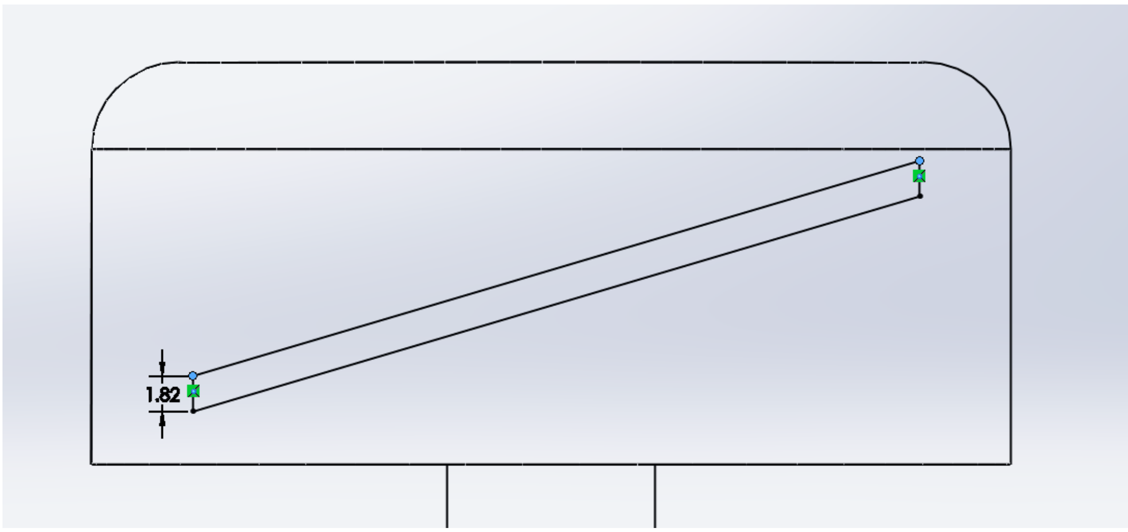


Figure 42: Creating sketch on the previously created plane

I then created a projection curve by projecting this sketch onto the cylindrical shell of the central hub:

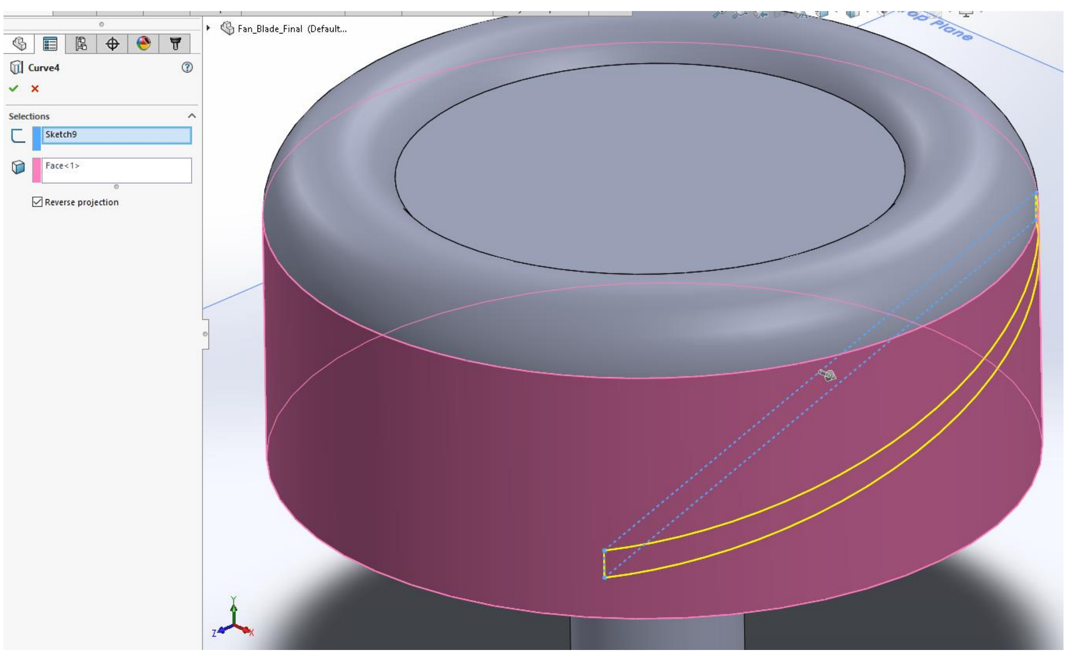


Figure 43: Creating a projection curve

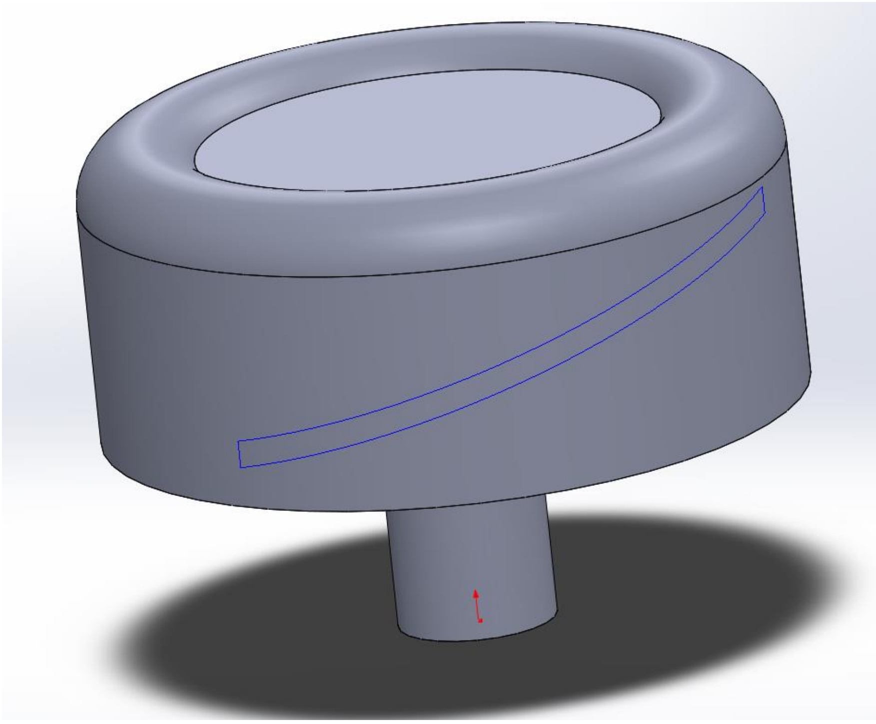


Figure 44: Projected sketch on the central hub

Positioning Points 1 through 10:

To plot the points 1 through 10, I first created points on a sketch on the top plane on SolidWorks using the imported, leveled image of the blade:

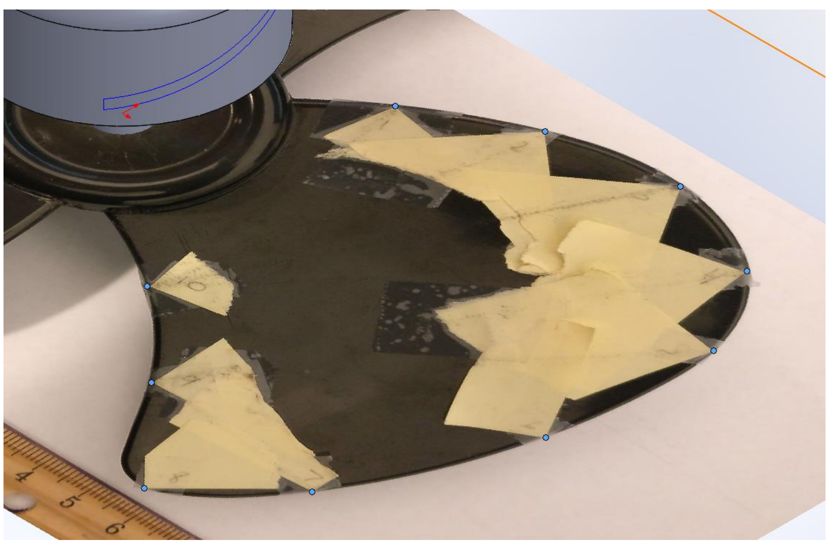


Figure 45: Positioning x and z coordinates of points 1 through 10

I then created vertical construction lines from these points on the top plane. These lines were dimensioned using their respective heights of points 1 through 10. I also converted entities of points A and B from the previous projection curve. I circled and labeled the points that represented points A, B, and 1 through 10 in the following figure:

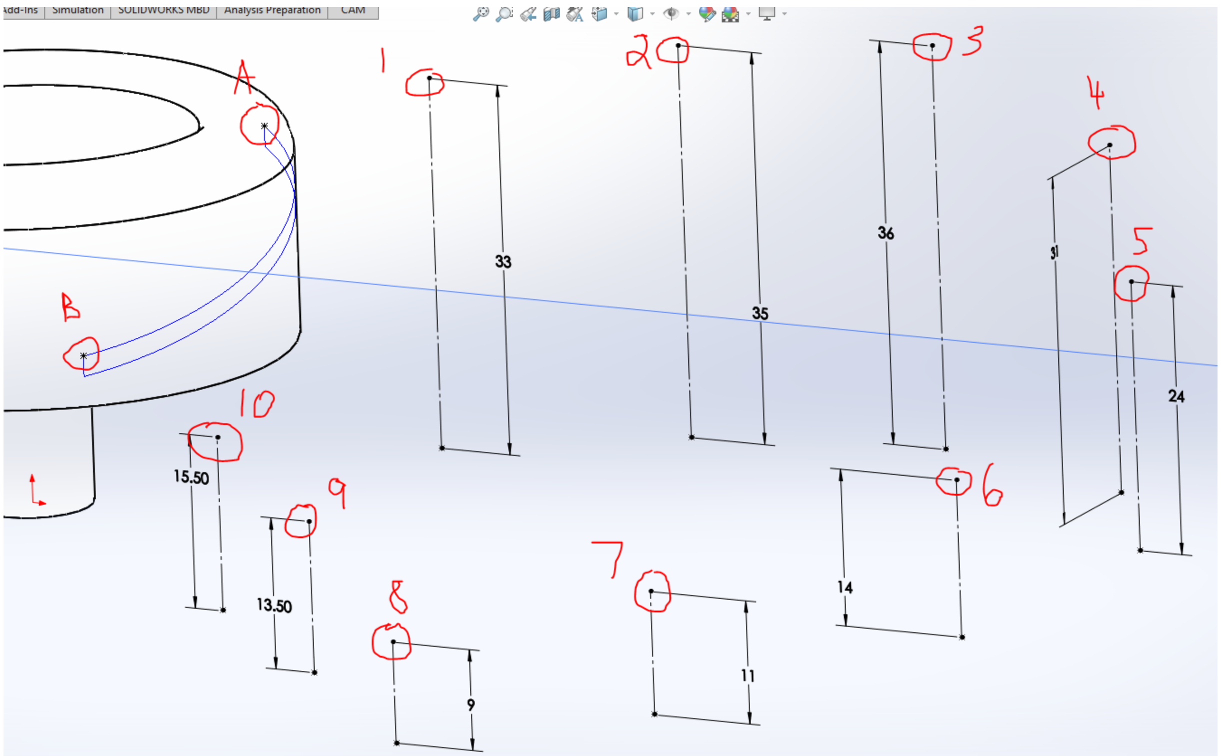


Figure 46: Creating points 1 through 10

Creating Guide Curve for Points A, B, and 1 through 10:

On the same 3D sketch I used to create points A, B, and 1 through 10, I connected adjacent points using the spline tool. I also converted entity of the top curve connecting points A and B. This can be shown in the following figure:

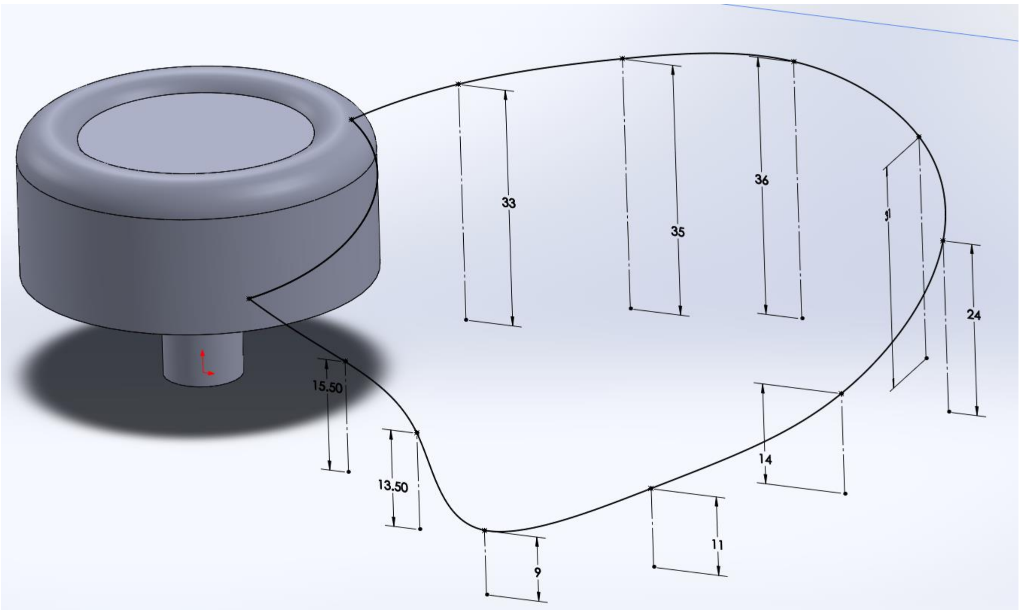


Figure 47: Using a spline tool connecting points A, 1 through 10, and B; converted entity of top curve connecting points A and B

Created a Surface Fill Using Contour From Previous Figure:

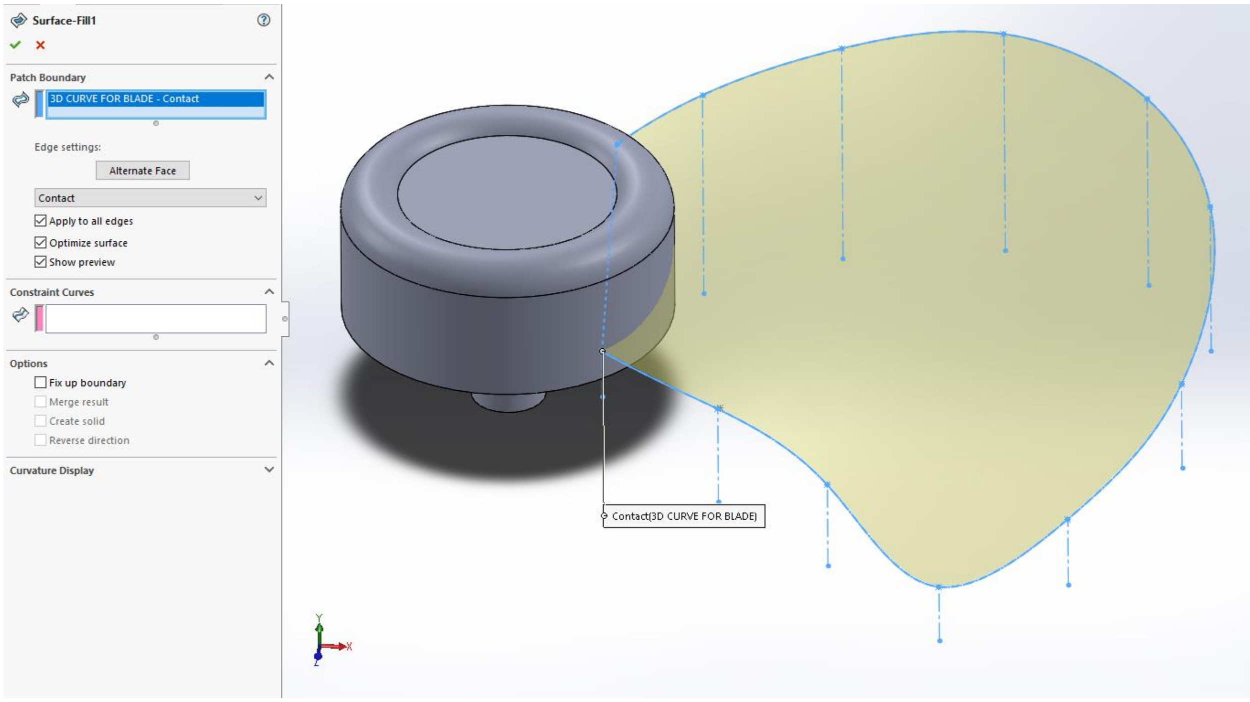


Figure 48: Using Surface-Fill to Create Top Surface of Fan Blade

Extruded Surface by Average Blade Thickness:

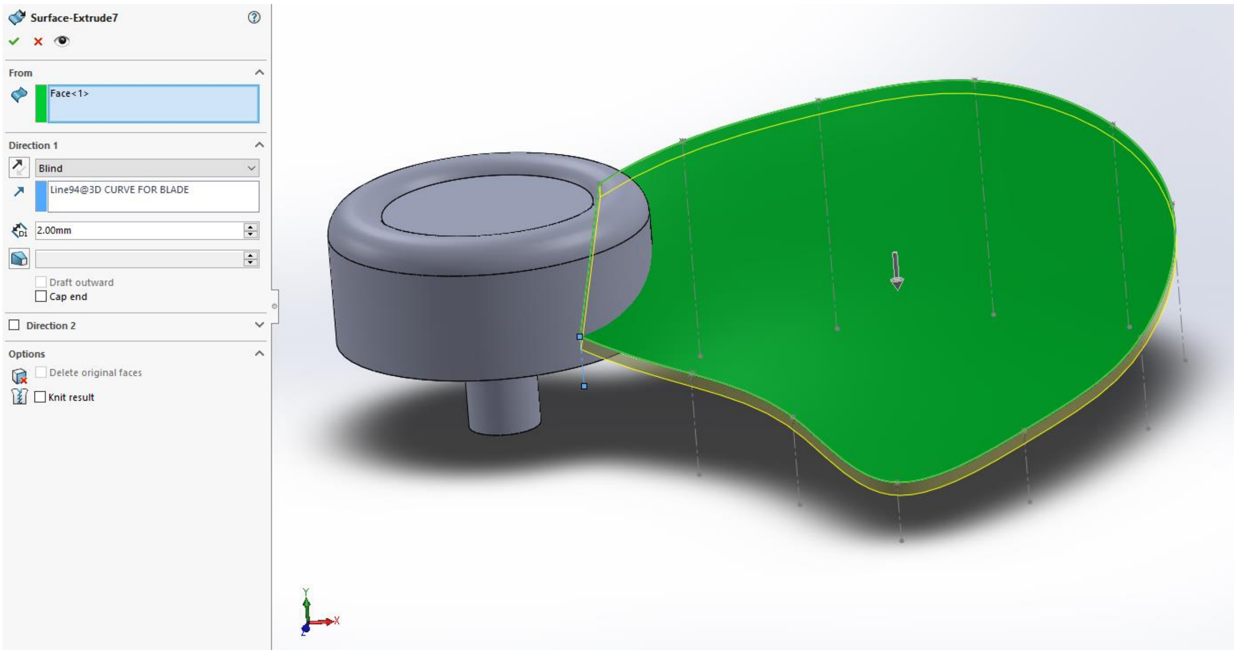


Figure 49: Extruding the Blade's Top Surface

Filling Bottom Surface of Blade to Complete the Solid Model of a Blade:

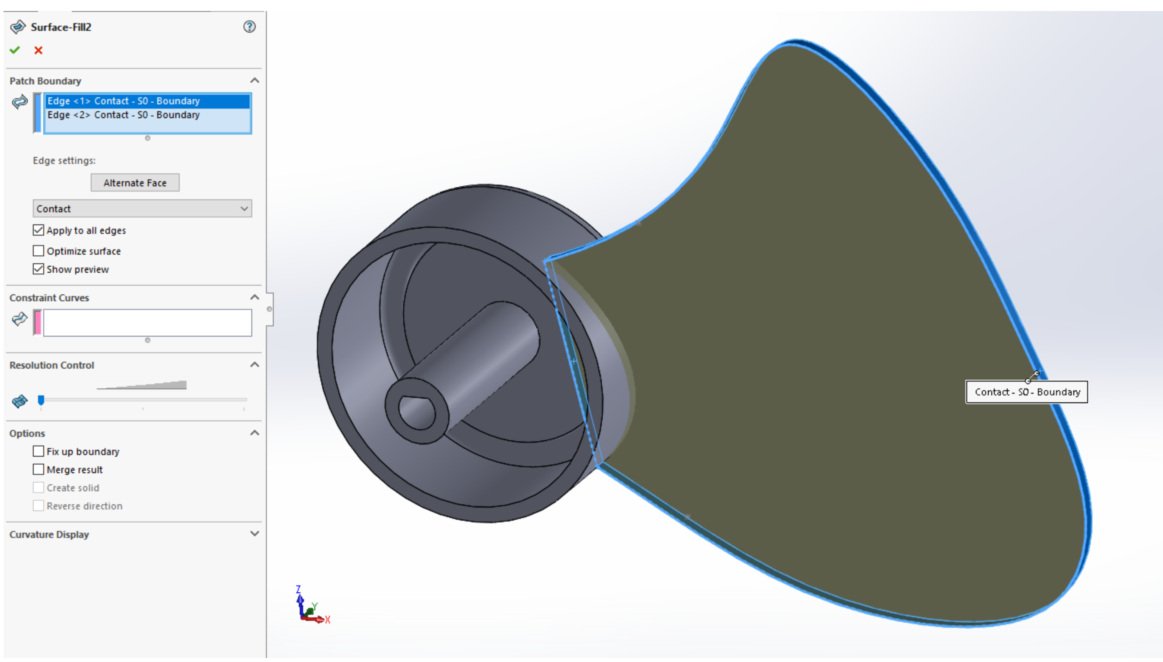


Figure 50: Completing Blade by Filling Bottom Surface

Utilizing Circular Pattern to Make 3 Blades:

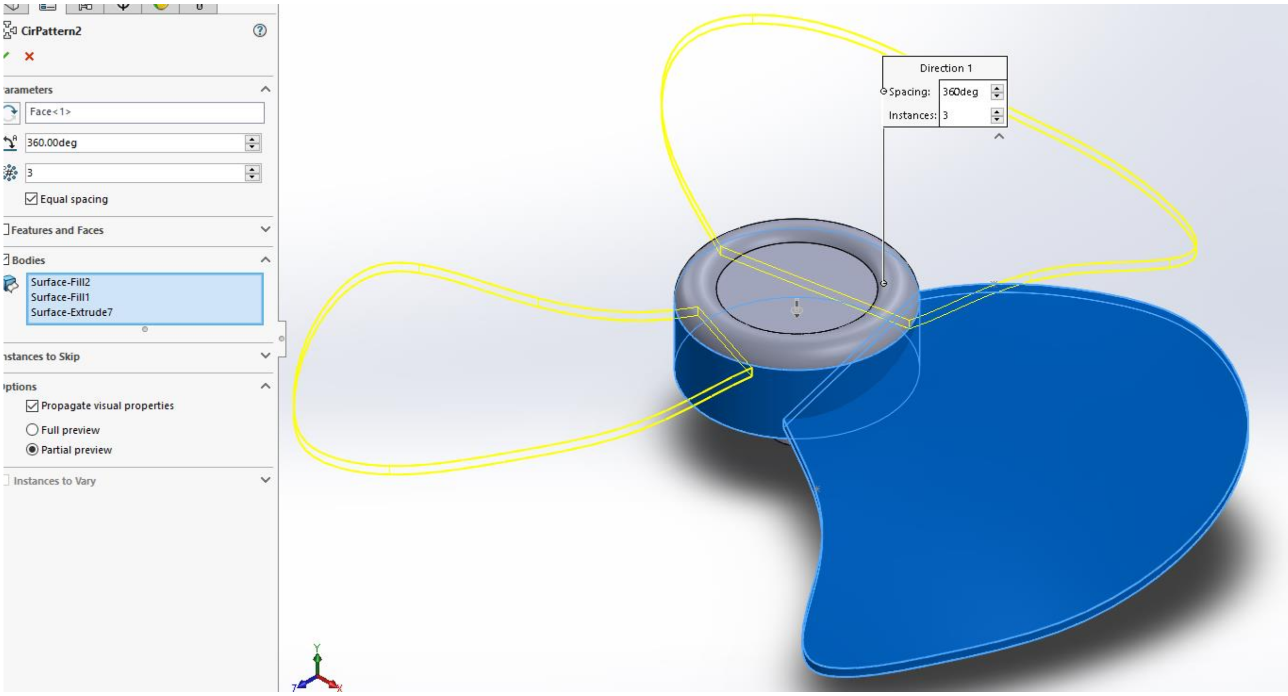


Figure 50: Creating Circular Pattern of Blade

RESULTS

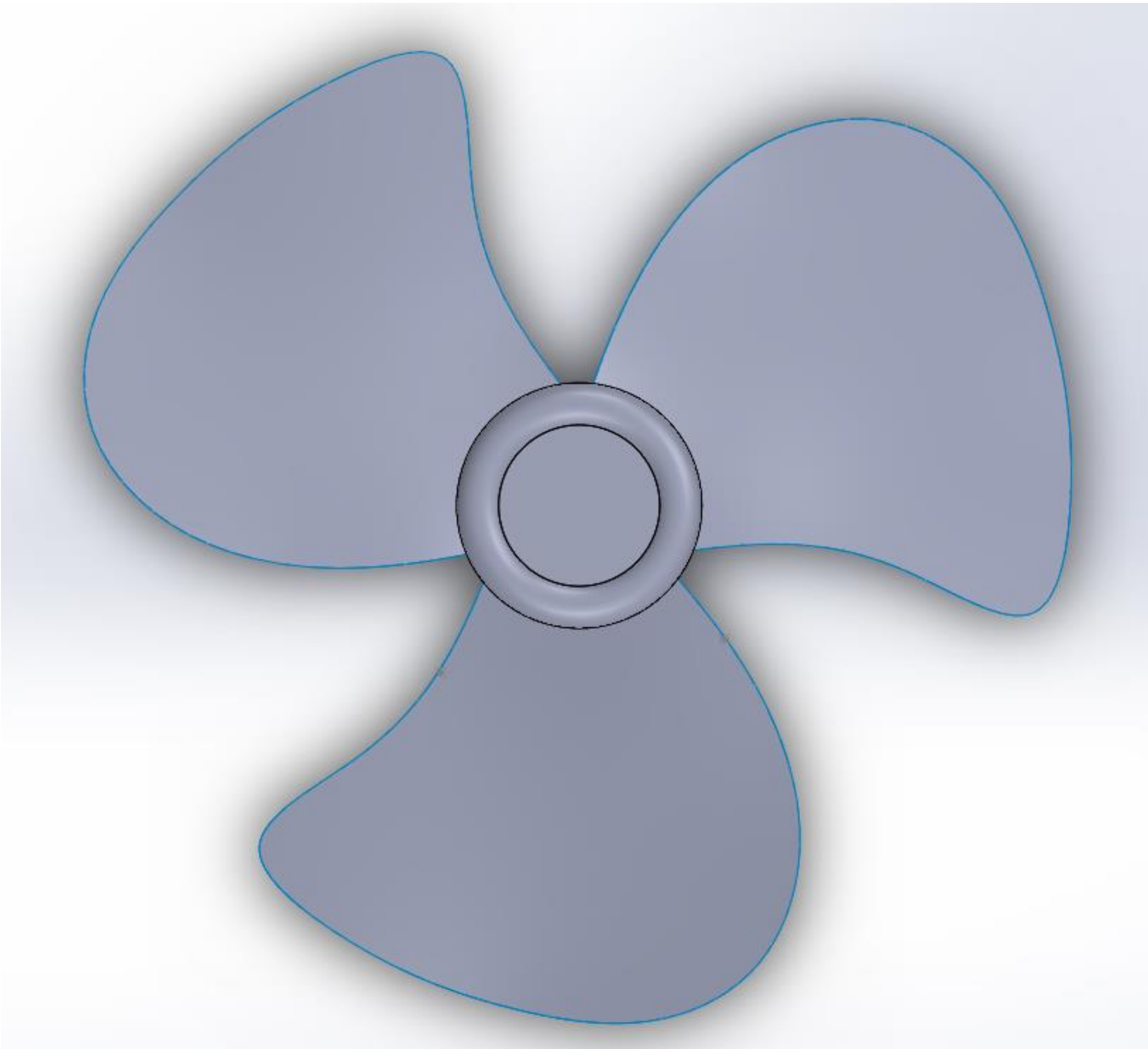


Figure 76: Top View of Reverse Engineered Model of Propeller

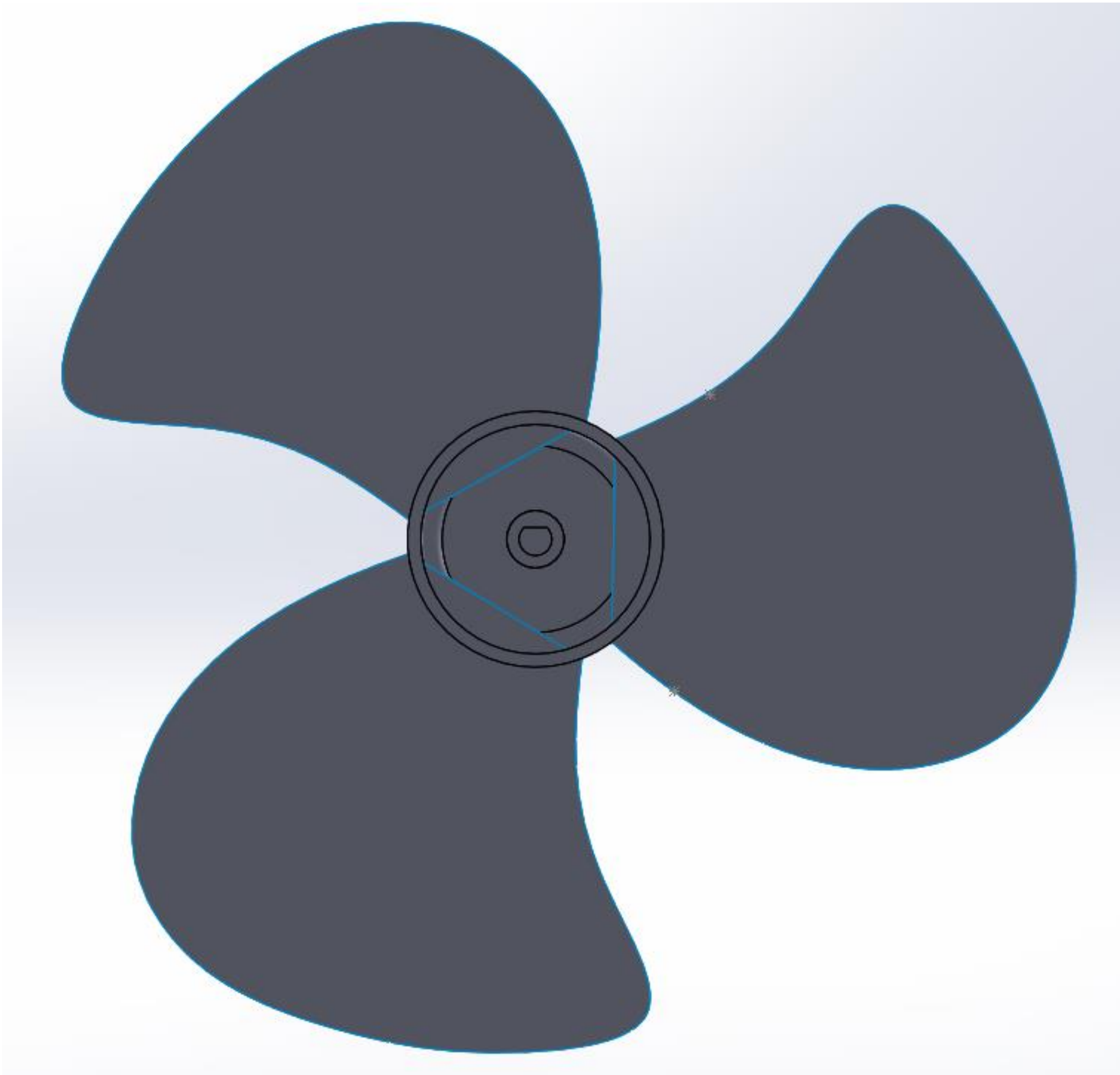


Figure 77: Bottom View of Reverse Engineered Model of Propeller

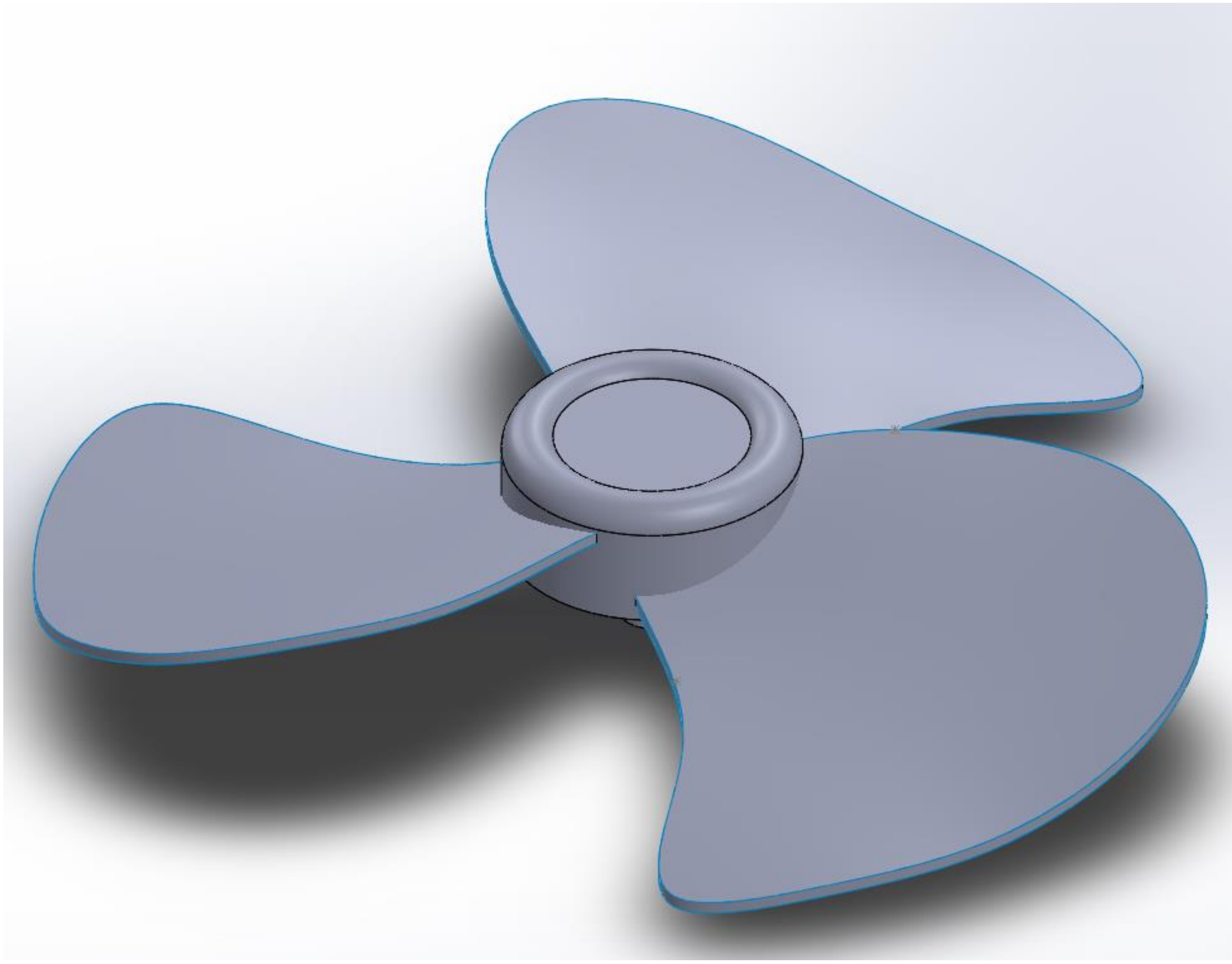


Figure 78: Isometric View of Reverse Engineered Model of Propeller