CHECKSHEET FOR PART MINI-PROJECT (Cover page 1 of 2)
NAME: _____

23 /24

Pre-CAD Plan

- ✓✓ Identify Primary & Secondary Features
- ✓✓ Explain Rationale for Location of Origin
- ✓✓ Pick Effective Front/Top/Side Views
- ✓✓ Order of Feature Implementation
- ✓✓ Identify Key Size Dimensions
- ✓✓ Keep track of ALL Assumptions

7 /8

Above and Beyond (Exemplary)

- ✓✓ Exceptional organization and neatness
- XX Analysis of steps/features that could prove difficult
- ___Other:_____

Process Documentation

- ✓✓ Completed Summary and Custom tabs (w/ summary tab overlaid on model)
- ✓✓ Illustration of Modeling Steps
- ✓✓ Explanation of Modeling Steps
- ✓✓ Rationale for Usage of Sketch Tools
- ✓✓ Expanded and Annotated Design Tree
- ✓✓ Compelling Lessons Learned (about this part as well as about SolidWorks)

8 /8

Above and Beyond (Exemplary)

- ✓✓ Exceptional organization and neatness
- ✓✓ Thoughtful use of Reference or Construction Geometry to Simplify Modeling

Other:_____

Finished Products (based on finished model and drawing)

- ✓✓ Fully-Defined Sketches
- ✓✓ Correct/Accurate Model
- ✓✓ Attractive Visualization of Final Part (include at least 1 color image)
- ✓✓ Mass properties shown
- ✓✓ Quality Engineering Drawing(s) on Multiple Sheets (use of part properties, filled out ME template w/ title block items)
- ✓✓ Complete/Non-redundant dimension scheme

6 /8

Above and Beyond (Exemplary)

- ✓✓ Effective use of section view, detail view, or other to assist drawing clarity
- ✓✓ Effective/clean dimension scheme

___

Other:_____

PART MINI-PROJECT SELF ASSESSMENT (Cover page 2 of 2)

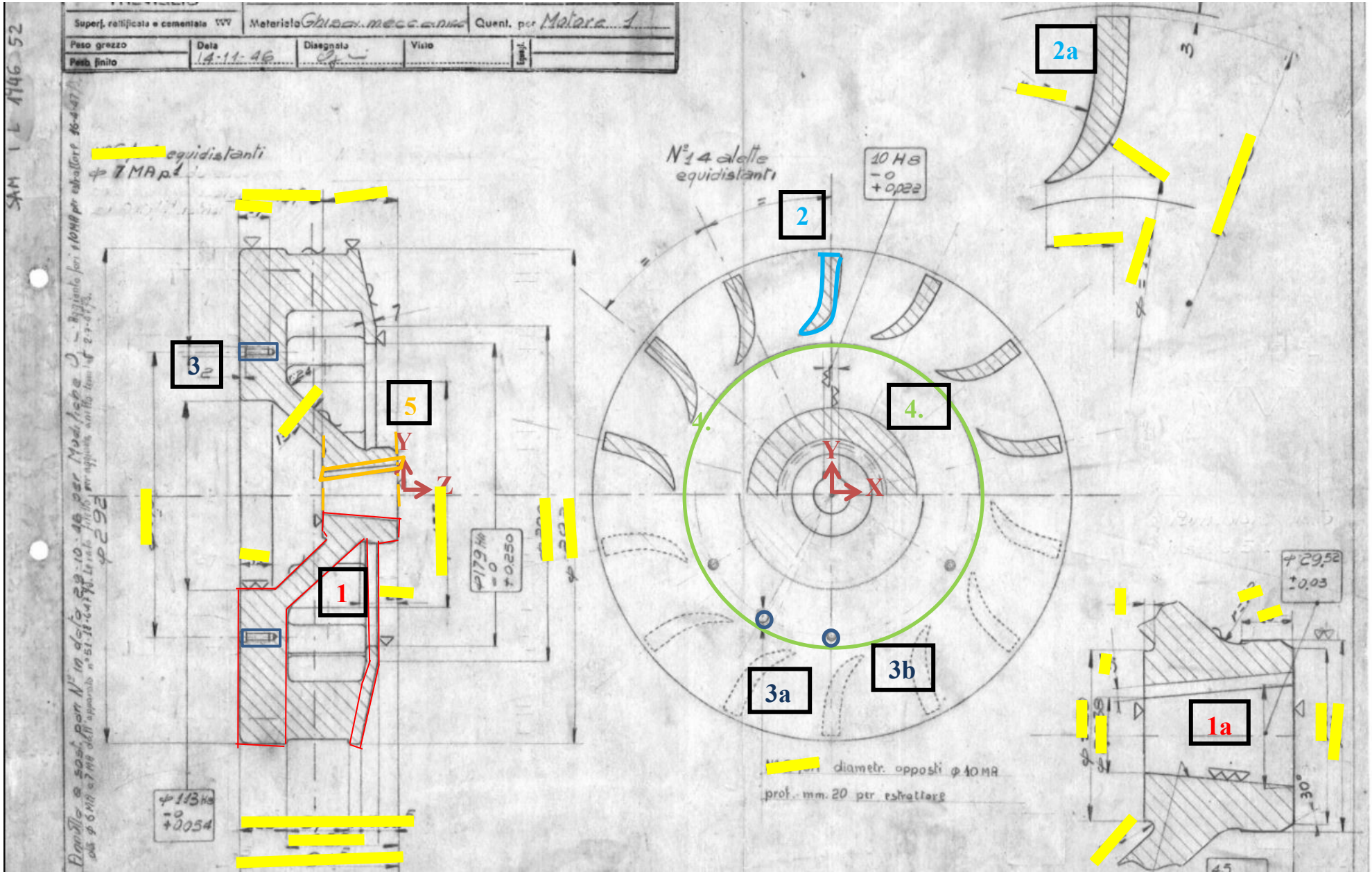
NAME: _____ SECTION: ___1_____ **DATE:** ____3-7-21_____

1. How many total hours did you spend on the part mini-project, including class time? How many in planning? How many in modeling? How many in documentation?

   Planning __**3**_____
   Modeling ____**8**____
   Documenting ____**3**_____ Total ___**14**_____

2. Using the ME 301 grading rubric (1-4), analyze your performance in the following:
   1- incomplete, major deficiencies 3 - *complete, meets expectations*
   2- complete, some deficiencies 4 - *exemplary, exceeds expectations*

Project Component	**Self Rating**	**Rationale**
Pre-CAD Plan - Identify Primary & Secondary Features - Explain Rationale for Location of Origin - Pick Effective Front/Top/Side Views - Order of Feature Implementation - Locate/Calculate Needed Dimensions - Keep track of ALL Assumptions	**4**	I worked closely off the required components listed on the rubric to create my Pre-CAD plan, as well as feedback I have previously received from past assignments. I used a new method (electronic) to annotate my Pre-CAD and I feel it made it much cleaner to work off of.
Process Documentation - Summary Tab Overlaid on Model - Illustration of Modeling Steps - Explanation of Modeling Steps - Rationale for usage of sketch tools - Annotated (i.e., renamed) Design Tree - Lessons/Discoveries (about this part as well as about SolidWorks)	**4**	Similarly, to above, I followed the rubric very closely for this part of the assignment. I worked to take as many screenshots as possible without making the entire document feel too stuffy. Lastly, one strategy I incorporated to aid in clarity was changing the visibility settings of the model.
Finished Products - Fully-Defined Sketches - Correct/Accurate Model - Attractive Visualization of Final Part (include at least 1 color image) - Calculation of Mass & Center of Mass - Quality Engineering Drawing(s) on <u>Multiple Sheets</u> (w/complete set of dimensions, use of part properties, and filled out ME template w/ other title block items)	**4**	Again, I followed the rubric very closely for this part too. I spent a lot of time (and several different methods) figuring out the best way to display my drawing and dimensioning.


Superf. rettificata e cementata
Materiale Ghisa meccanica
Quant. per Motore 1
Peso grezzo
Peso finito
Data 14-11-46
Disegnato
Visto
N° 14 alette equidistanti
10 H8 -0 +0,022
2
2a
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Y
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4.
1
3a
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1a
diametr. opposti φ 10 M8
prof. mm. 20 per estrattore
30°

Pre-CAD Plan:

Note: Highlighted Dimensions are "key dimensions" for creating the sketches and features

1. **(Primary Feature)** Sketch the structural geometry of the fly wheel on the **Right Plane** using dimensions and relations from **1a. Revolve** the sketch around a centerline running through **Origin** (placed in the center of the component to allow for concentric circles constrained to Origin in step 4).
2. **(Secondary Feature)** Sketch a "seed" fin on the **Front Plane**. Dimension according to **2a.** Sketch will be defined off the origin and walls. **Extrude up to next surface** will be used to finish the seed geometry.
3. Use **Hole Wizard** to create seed holes for hole **3a** and **3b**, considering they are two different sizes. **Assume 3b is a tapped screw hole.**
4. **(Secondary Feature)** Use a Circular Pattern around the highlighted pattern on **Front Plane** to create the **14 fins, 6 3b holes, and 2 3a holes.** Bodies will be merged.
5. (**Secondary Feature)** Create a **Lofted Cut** through the inner keyhole to remove the material for the notch. **Two planes will be made** (dashed orange) based off the **Front Plane.** They will have two offset rectangles that I will loft cut between.

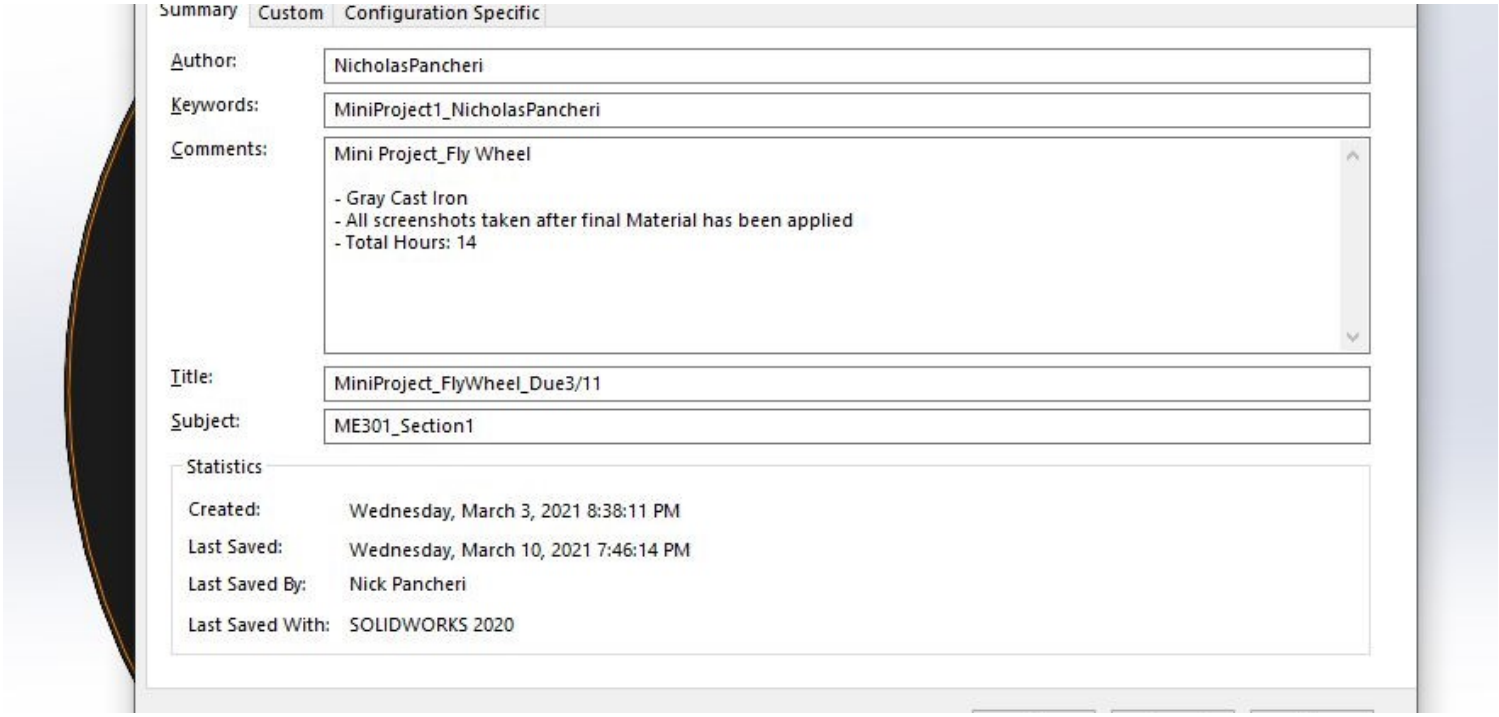
Assumptions:

- there are 6 3b holes but only 2 x 3a holes (based off annotations).
- Extruding "Up to next surface" will work when the terminating face is curved.
- The fins are indeed filler material between the outer and inner casing, and an extrusion up to the surface will create the correct fin depth

Lessons Learned:

1. This was the first part we have had to model that I found the most challenging part was the initial sketches. I think this was due to a few reasons, but notably I found that sketching the curves and arcs were very hard to fully define and lock into place. After spending a while messing around and trying different techniques, I finally began to realize that the tangent relation is very powerful because it can lock in and align both an arc and a line simultaneously.
2. I have found that one of the most valuable aspects of creating a part occurs with the Pre-CAD plan. Taking the time to plan out and consider what features and the order to create the part helped make it a smooth part to begin modelling. Additionally, at the times when I became challenged, having a prepared and organized Pre-CAD plan made it easy to explain to the mentors or my peers the exact step I am stuck on – and in the end, this expedites the entire sketching process.
3. Another very important skill that I feel I have begun to master from this part was the function and implementation of section views within my Part Drawing. It would have been impossible to fully dimension this sketch in the drawing without cutting away part materials to view the central bore and fins. However, in the beginning I found it challenging to select the correct view to cut-away. Later, I realized that the section views cut away material perpendicular to the "section line". Similarly, using the detailed view to clearly focus upon the central bore (my first sketch) made my entire drawing many times cleaner and more legible. In the future I will definitely work to incorporate these views types into future assignment drawings.

Properties Tab: Summaries



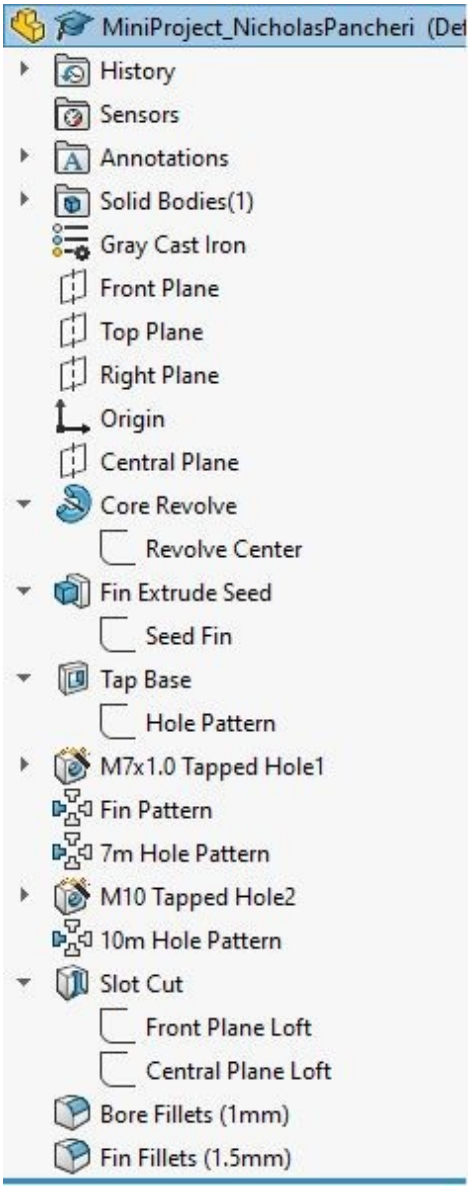
The Part Properties were uploaded within the Properties Tab to reflect relevant information. Properties I wanted imported into the drawing document were added in the "Customs" part of the Properties Tab.

Properties Tab: Custom

	Property Name	Type	Value / Text Expression	Evaluated Value	C
1	PartNo	Text	00-01	00-01	
2	Material	Text	"SW-Material@MiniProject_NicholasPancheri.SL	Gray Cast Iron	
3	Revision	Text	A	A	
4	Author	Text	NicholasPancheri	NicholasPancheri	
5	CheckedBy	Text	KaitlynHarvey	KaitlynHarvey	
6	Description	Text	HW10_NicholasPancheri	HW10_NicholasPancheri	
7	Project Title	Text	Mini Project	Mini Project	
8	<Type a new propert				

Design Tree:

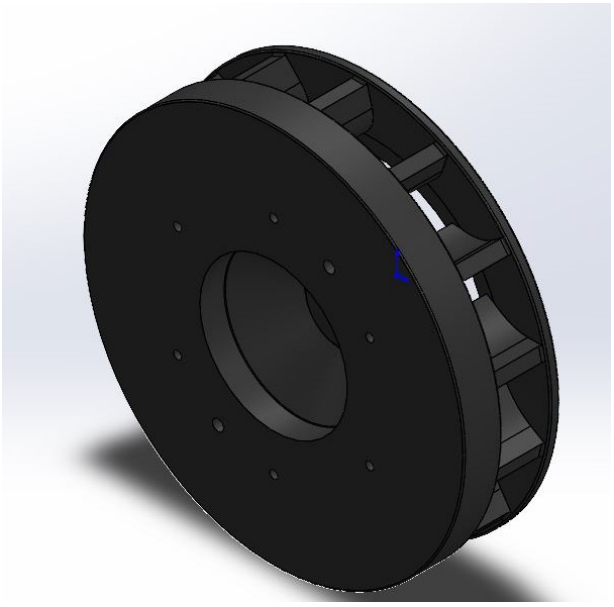
Each Feature and Sketch (except for Reference planes because they were so numerous and multi-use) was given a descriptive name.



Mass Properties:

```
Mass properties of MiniProject_NicholasPancheri
    Configuration: Default
    Coordinate system: -- default --

Density = 0.01 grams per cubic millimeter

Mass = 17793.95 grams

Volume = 2471382.15 cubic millimeters

Surface area = 341394.30  square millimeters

Center of mass: ( millimeters )
     X = 0.00
     Y = -0.01
     Z = 65.00

Principal axes of inertia and principal moments of inertia: ( grams *  square millimeters
Taken at the center of mass.
      Ix = ( 0.87, -0.49,  0.00)     Px = 114296083.08
      Iy = ( 0.49,  0.87,  0.00)     Py = 114420203.84
      Iz = ( 0.00,  0.00,  1.00)     Pz = 207672816.37

Moments of inertia: ( grams *  square millimeters )
Taken at the center of mass and aligned with the output coordinate system.
     Lxx = 114326410.14          Lxy = -53333.73          Lxz = 0.00
     Lyx = -53333.73             Lyy = 114389877.17       Lyz = 6031.09
     Lzx = 0.00                  Lzy = 6031.09            Lzz = 207672815.98

Moments of inertia: ( grams *  square millimeters )
Taken at the output coordinate system.
     Ixx = 189506377.73          Ixy = -53333.73          Ixz = 0.00
     Iyx = -53333.73             Iyy = 189569843.69       Iyz = -2926.38
     Izx = 0.00                  Izy = -2926.38           Izz = 207672817.05
```

Final Model:

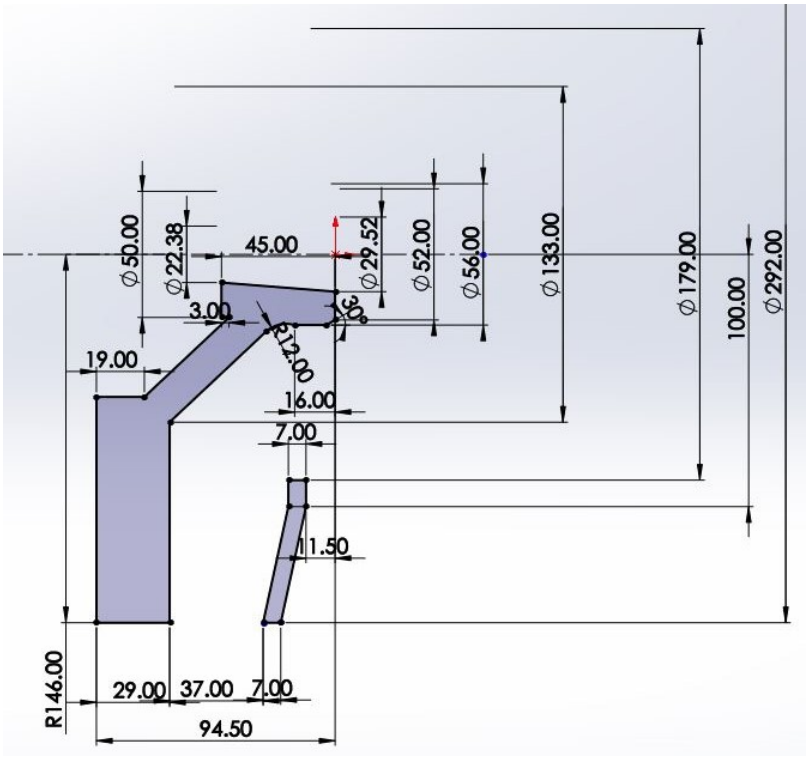


ISO View

Front View

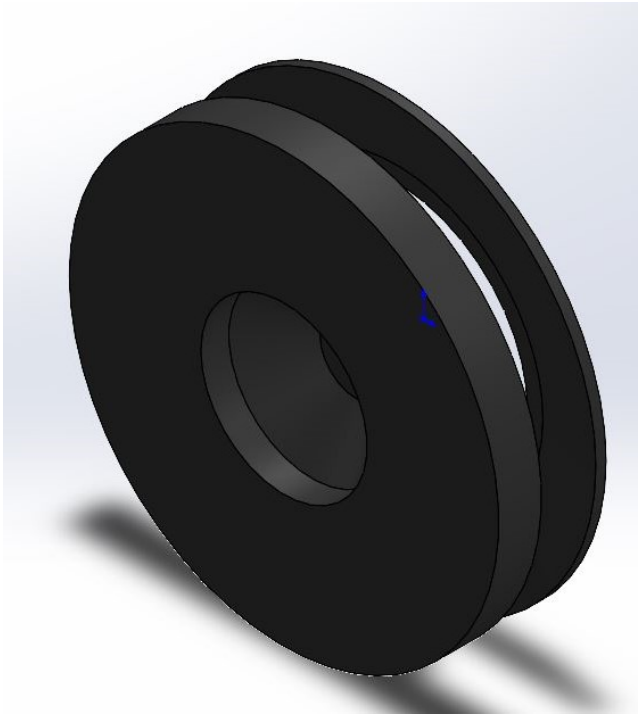
Side View

Note: All my **snips are taken *after* finishing the part** (they are all taken with Gray Cast Iron as the material) using the rollback bar. Also, my part **Visibility** varies throughout the documentation **Depending on how it effects the ability to see new Features and Sketches.**



1. Sketch for Revolve

I began by creating a sketch profile on the **Top Plane** using the dimensions provided by the original schematic. Notably, I set my origin on the **centerline** of the revolve, which allowed for dimensioning scheme to use diameters just like the schematic. The most challenging piece to sketch was the 12 mm radius that connects to the neck between the upper and lower parts. I tried using both the arc tool and feature fillet afterwards, but neither worked properly. Finally, I created a 12 mm radius circle and used **tangent relations** to join it together. That worked perfectly. It was conceptually confusing to sketch the lower right component that was free floating from the rest of the geometry.



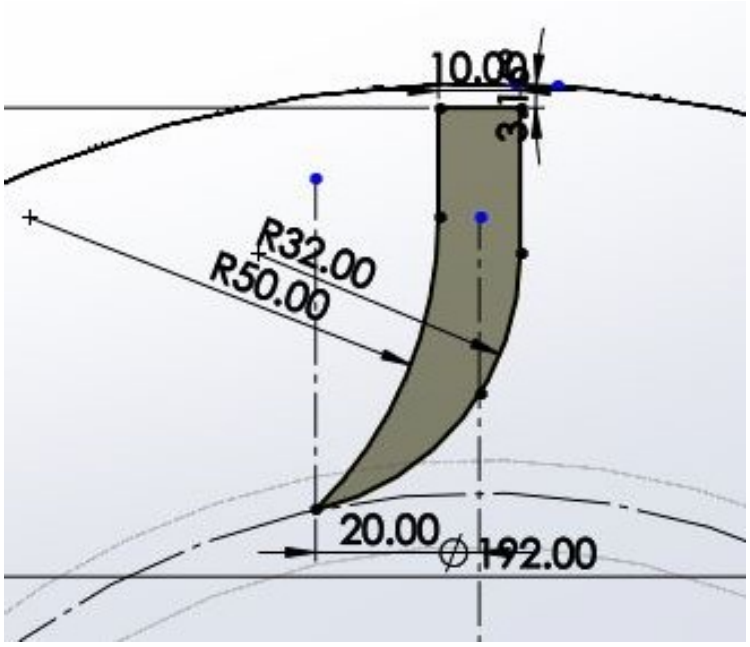
2. Revolving the Sketch

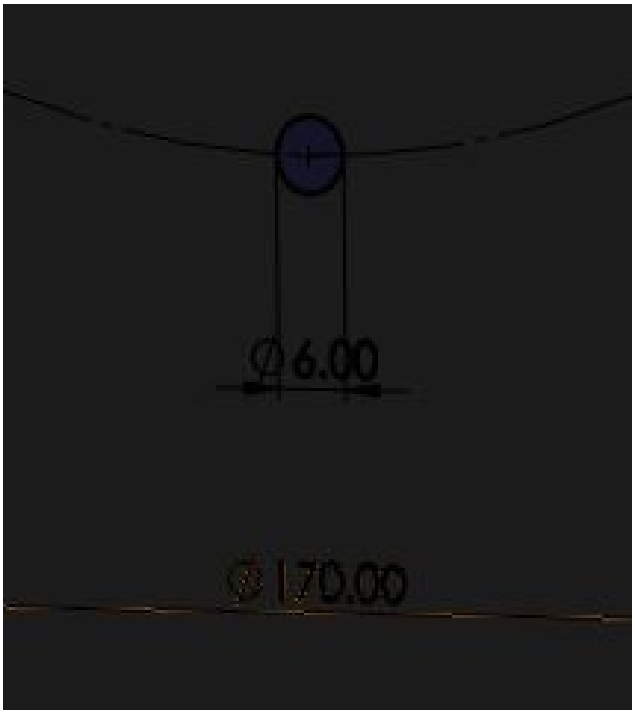
I then used the **Revolve Feature** to create a **Parent Feature**. At this point, it was two separate bodies within the sketch. I specifically chose this feature because it was by far the simplest, most efficient way to model the part, especially compared to lofting or extruding it.

3. Sketching the Seed Fin

I began a sketch of the fin on the interior face of the revolved **Parent Feature**. To define the sketch, I had to create a **reference circle** of 192 mm diameter to make the tip of the fin coincident with. I also made a **centerline** running from the origin vertically up to the center of the fin and added a **vertical relation.** The challenging part of this sketch was correctly adding in the curved arcs. After trying a few different things, I finally made both arcs (R32 and R50) tangent to a centered, 10 mm wide rectangle. I then used the **Power Trim** tool to remove excess lines and close the sketch geometry.

I **Extruded** this fine with the **Up to Surface** end condition to span the gap between the bodies from my **Revolve Sketch.** I made sure to selected **Merge Bodies,** resulting in one single, merged body within my part.



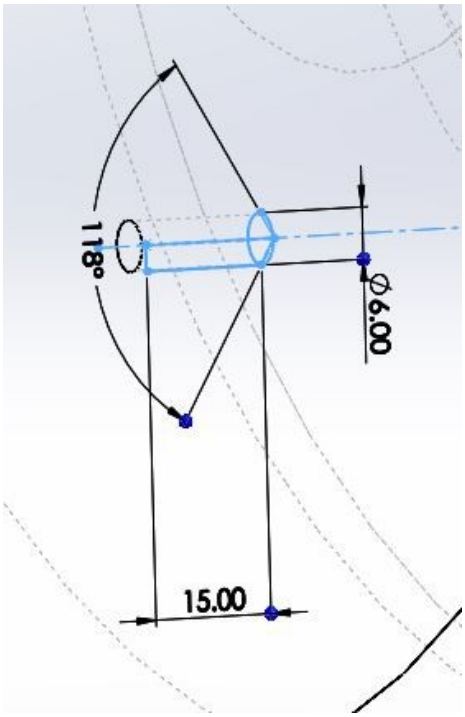


4. Adding Clearance for the 6 x M7 Holes

In order to add the 2 mm clearance hole that each of the 6 M7 holes sits in, I first created a 170 mm **Reference Bolt Circle.** I then sketched a circle on this reference circle and made it **Vertically Related** to the origin. I used a **Cut Extrude** to create the 2 mm **Secondary Feature** that the M7 holes would later be nestled in. I made an **Assumption** that the clearance hole was the same diameter as the hole feature it would contain.

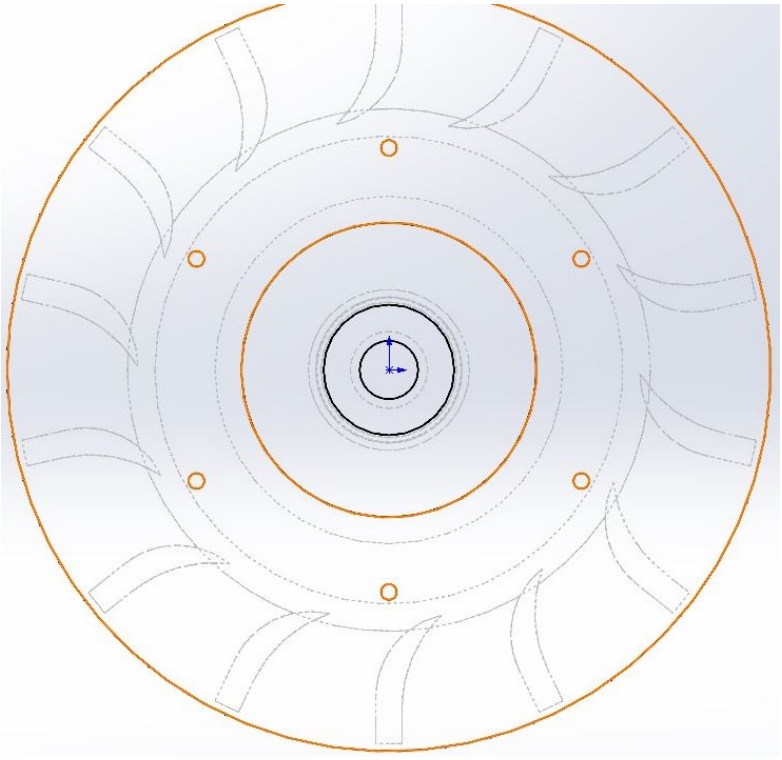
5. Hole Wizard 1: 6 x M7 Holes

I then used the **Hole Wizard** to create another **Secondary Feature** within clearance hole I made in **Step 5 (above)**. I **assumed a Straight Tap w/ the Bottoming Tapped Hole type.** I used an M7 hole as specified in the schematic.



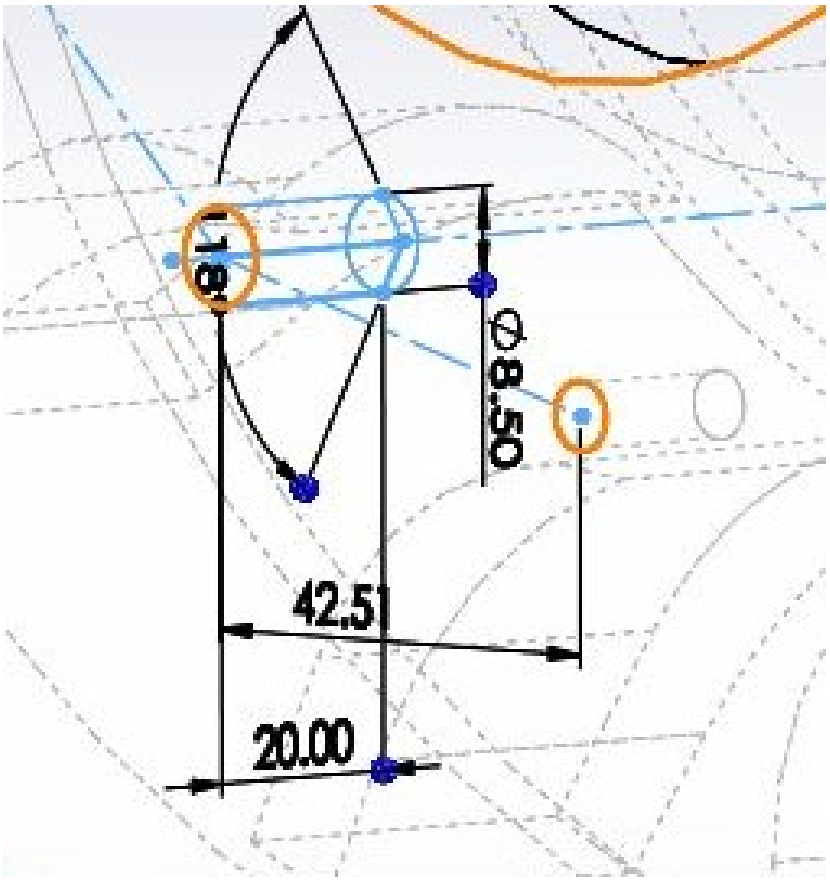
6. Circular Patterning Fins and M7 Holes

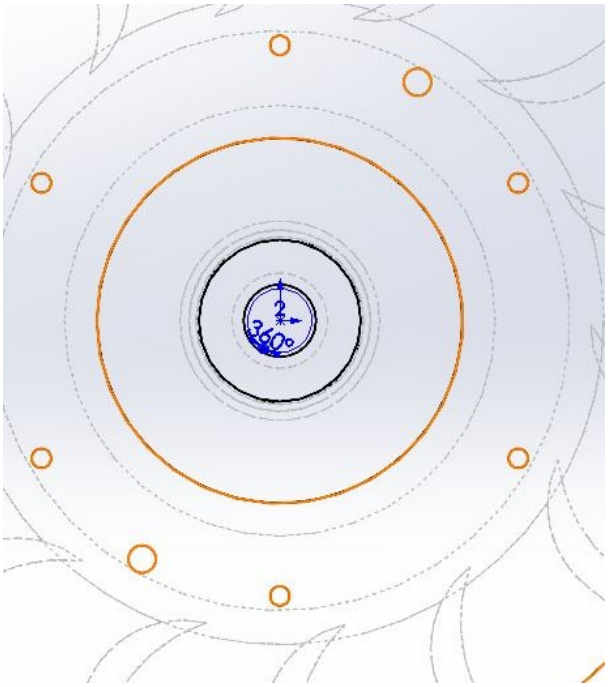
At this point, I created **two Circular Pattern Features** based off my **Seed Fin Sketch (Step 3) and Clearance Hole/Hole Wizard Sketches (Steps 4 & 5).** For both patterns, I used the outer edges of the part (highlighted Orange) as my **Direction 1.** I repeated the Fins 14 times and Both holes 6 times. I specifically chose to use this feature because it is the most efficient and accurate way to create these equally spaced components of the part.



8. Hole Wizard 2:

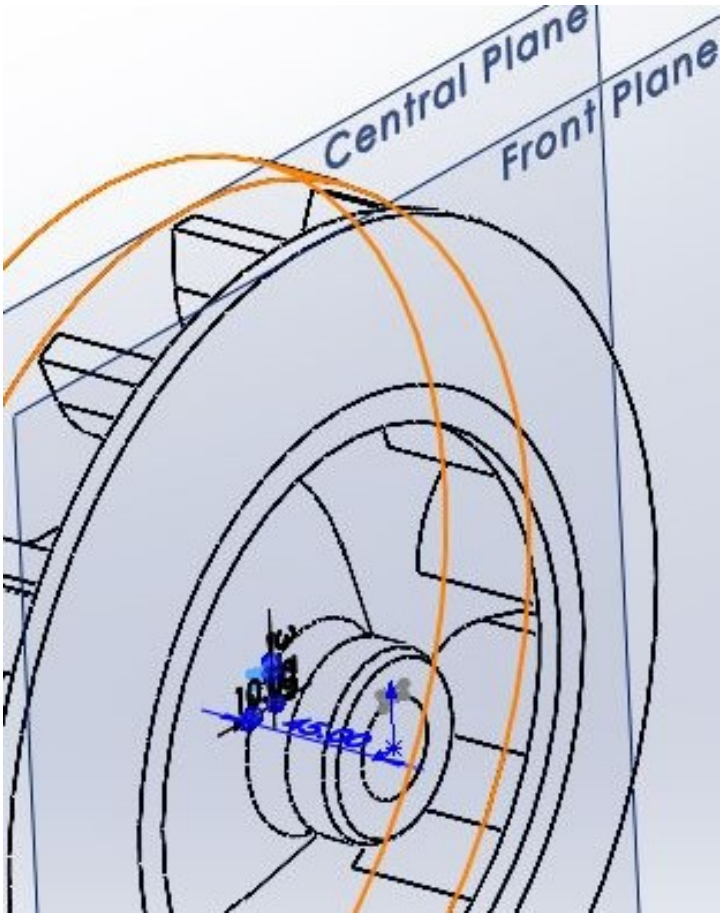
Then, I added second hole type, **assuming it is a Straight Tap w/ the Tapped Hole type.** I did have a tough time fully define the placement of the hole. I ended up creating conjoining line segments from between two holes of **7m Circular Hole Pattern** and added a **Midpoint Relation** and made it **Concentric** with my original **Reference Bolt Circle.** I **assumed it was a 20 mm** depth based off the schematic.

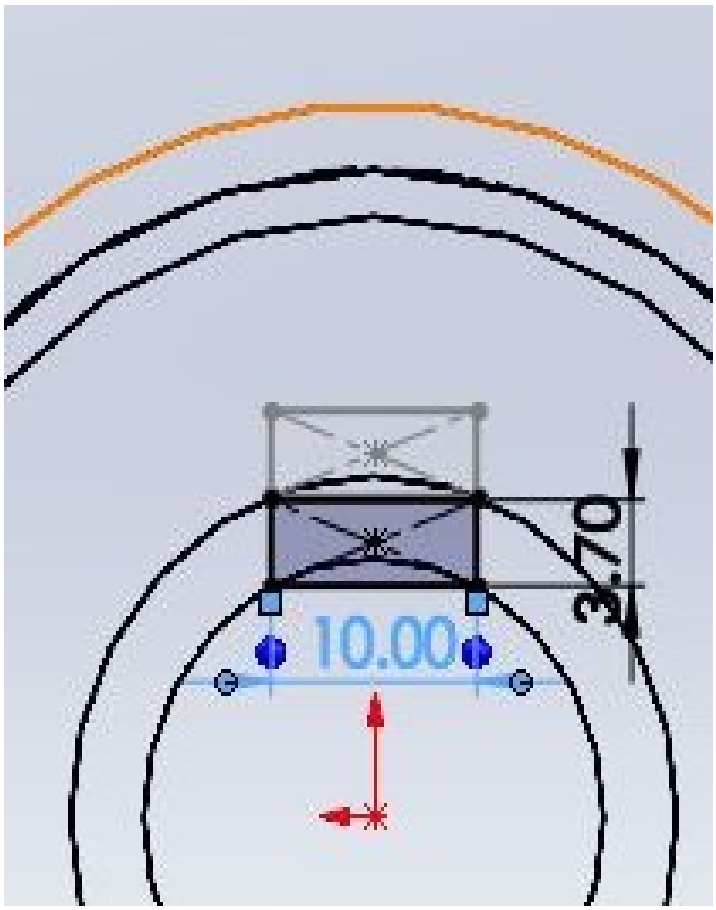




7. Circular Patterning M10 Holes

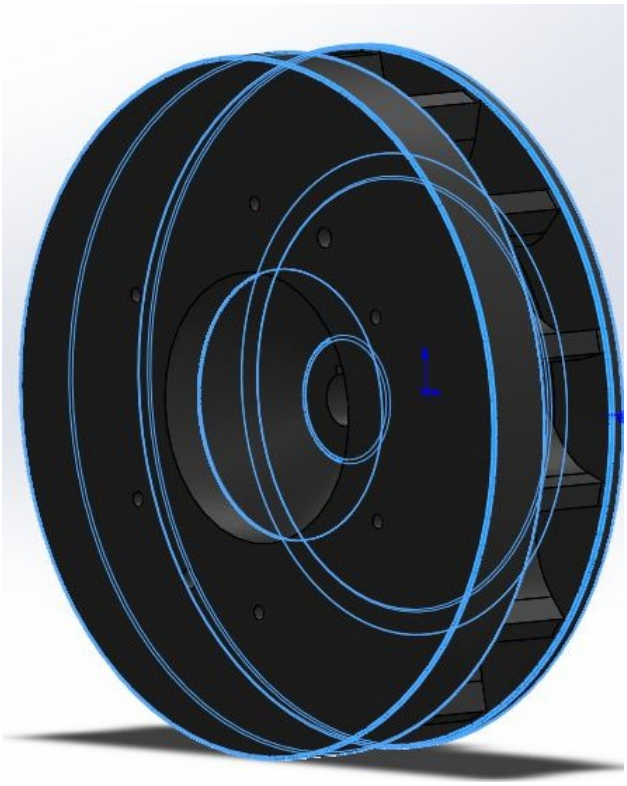
I completed the Hole Features by adding the second M10 hole by creating a third **Circular Pattern Feature.** I made it to occur 2 times, using the inner rim (highlighted Orange) as **Direction 1.**

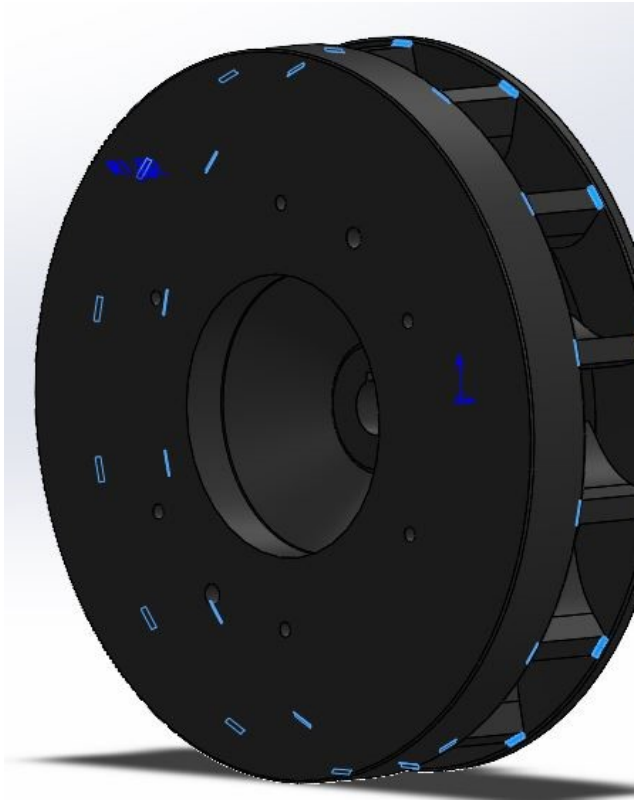




9. Lofting the Keyhole

To add the keyhole running through the central bore, I made a **Central Reference Plane** based off the **Front Plane (Above Left).** I sketched two rectangles on each plane that were **Horizontally and Vertically Related** to the front and back perimeters, respectively, of the central bore **(Above Right)**. I then used the **Lofted Cut Feature** to remove the material and create the keyhole geometry. I specifically used the Lofted Cut Feature because it allowed me to create two similar rectangles based on whatever reference geometry I desired.



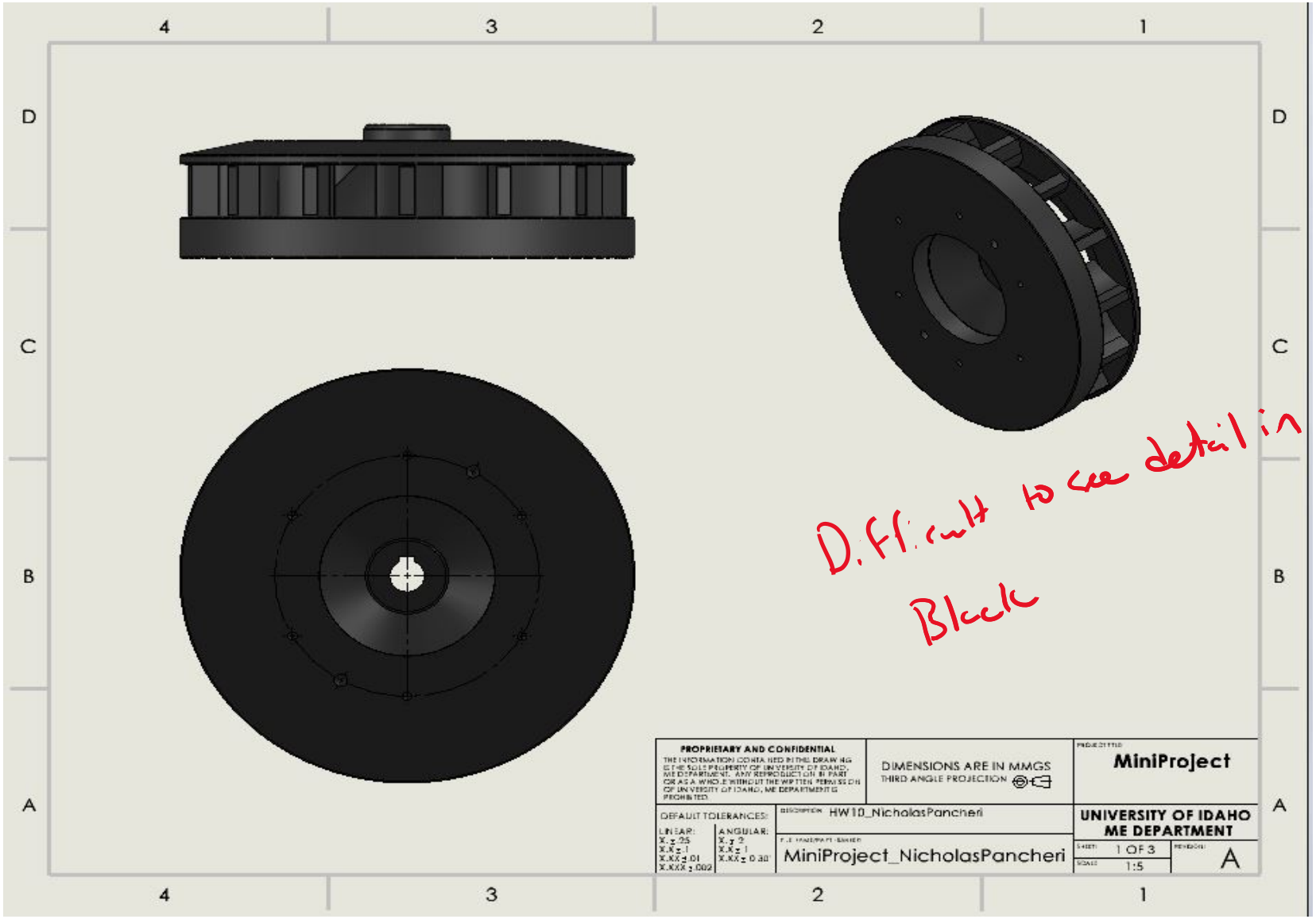


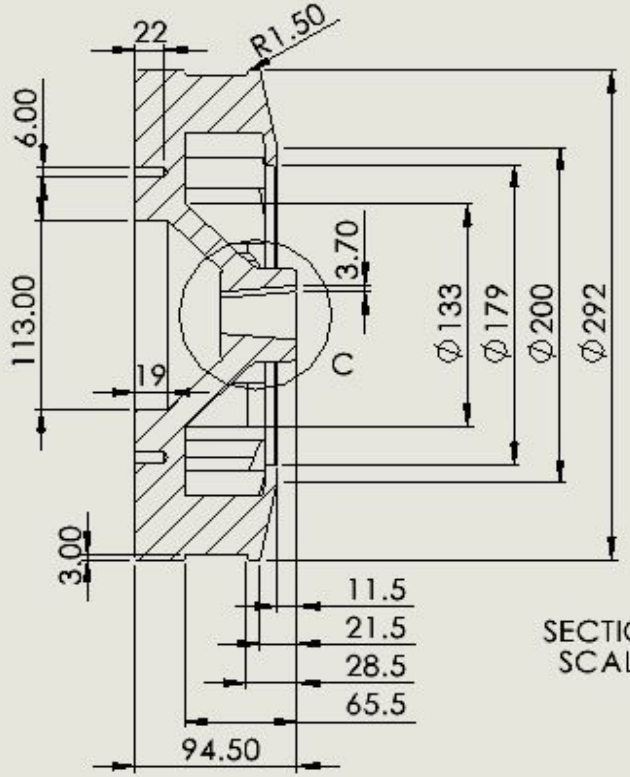
10. Adding the Fillets

Later, I went back and added all remaining fillets for the part. I selected all the necessary cylindrical edges and filleted them, a**ssuming they were a 1 mm Fillet Feature (Above Left).** I repeated this process with the corners of edge Fin, but this time **Assumed they were 1.5 mm Fillets** because they looked slightly wider than the cylindrical fillets **(Above Right).**

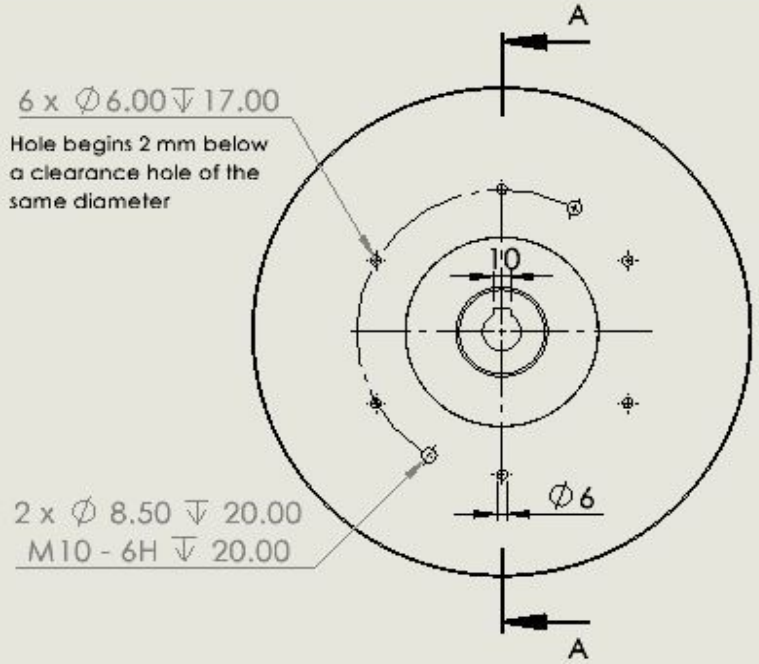
11. Drawing and Finishing Touches

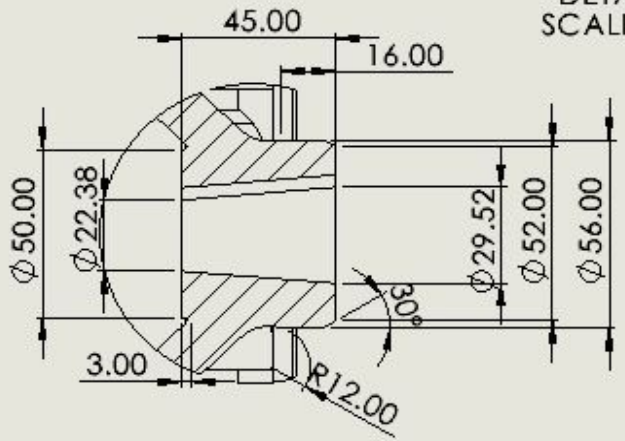
Lastly, I changed the material of the part to Gray Cast Iron. Afterwards, I converted the part into a drawing on the ME Template and used the Smart Dimensions tool to annotate the essential part dimensions. I selected a **Front, Top, Section View, Isometric view, and two detailed views.** I decided to add Section and Detail views to make it easier to dimension the internal components and provide a cut-away to better show the geometry of the Fins and Central Bore.





SECTION A-A
SCALE 1 : 4



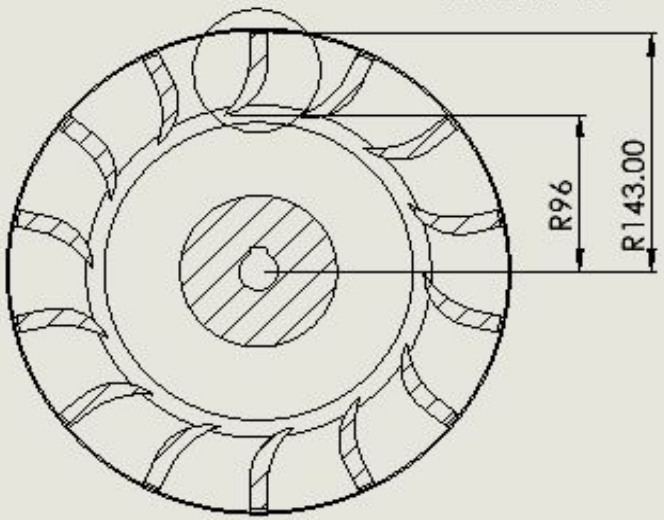


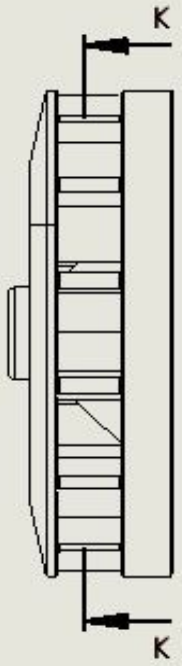
DETAIL C
SCALE 1 : 2

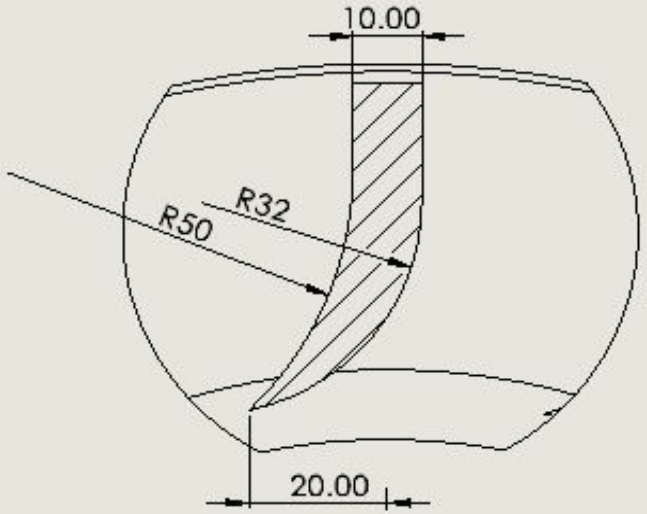
All fillets are 1 mm unless otherwise noted

PROPRIETARY AND CONFIDENTIAL THE INFORMATION CONTAINED I4 THIS DRAWING IS THE SOLE PROPERTY OF UNIVERSITY OF IDAHO, ME DEPARTMENT. A 4Y REPRODUCT ON IN PART OF AS A W HOLE W THOUT THE WRITTEN PERMISS ON OF UN VERSITY OF IDAHO, ME DEPARTMENT S PROHIBITED.	DIMENSIONS ARE IN MMGS THIRD ANGLE PROJECTION	PROJECT(S): **MiniProject**
DEFAULT TOLERANCES: LINEAR: X. ± .25, X.X ± .1, X.XX ± .01, X.XXX ± .002 ANGULAR: X.± 2, X.X ± 1, X.XX ± 0.30°	DESCRIPTION: HW10_NicholasPancheri FILE NAME(S)/PART NUMBER: MiniProject_NicholasPancheri	**UNIVERSITY OF IDAHO ME DEPARTMENT** SHEET: 2 OF 3 SCALE: 1:5 REV(S): A

SECTION K-K
SCALE 1 : 4







DETAIL L
SCALE 1 : 1

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DIMENSIONS ARE IN MMGS
THIRD ANGLE PROJECTION

PROJECT TITLE: **MiniProject**

DEFAULT TOLERANCES:

LINEAR:	ANGULAR:
X ± 0.25	X ± 2
X.X ± 0.1	X.X ± 1
X.XX ± 0.01	X.XX ± 0.30'
X.XXX ± 0.002	

DESCRIPTION: HW10_NicholasPancheri

P. DRAWING PART NUMBER: MiniProject_NicholasPancheri

UNIVERSITY OF IDAHO
ME DEPARTMENT

SHEET: 3 OF 3
SCALE: 1:5
REVISION: A