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

24/24

NAME: **Kaitlyn Harvey**

Pre-CAD Plan

_1_ Identify Primary & Secondary Features
_1_ Explain Rationale for Location of Origin
_1_ Pick Effective Front/Top/Side Views
_1_ Order of Feature Implementation
_1_ Identify Key Size Dimensions
_1_ Keep track of ALL Assumptions

8/8

Above and Beyond (Exemplary)

_1_ Exceptional organization and neatness
_1_ Analysis of steps/features that could prove difficult
___ Other:____

Process Documentation

_1_ Completed Summary and Custom tabs (w/ summary tab overlaid on model)
_1_ Illustration of Modeling Steps
_1_ Explanation of Modeling Steps
_1_ Rationale for Usage of Sketch Tools
_1_ Expanded and Annotated Design Tree
_1_ Compelling Lessons Learned (about this part as well as about SolidWorks)

8/8

Above and Beyond (Exemplary)

_1_ Exceptional organization and neatness
_1_ Thoughtful use of Reference or Construction Geometry to Simplify Modeling
___ Other:____

Finished Products (based on finished model and drawing)

_1_ Fully-Defined Sketches
_1_ Correct/Accurate Model (with use of at least 1 loft)
_1_ Attractive Visualization of Final Part (include at least 1 color image)
_1_ Mass properties shown
_1_ Quality Engineering Drawing(s) on Multiple Sheets (use of part properties, filled out ME template w/ title block items)
_1_ Complete/Non-redundant dimension scheme

8/8

Above and Beyond (Exemplary)

_1_ Effective use of section view, detail view, or other to assist drawing clarity
_1_ Effective/clean dimension scheme
___ Other:____

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

NAME: _Kaitlyn Harvey________ **SECTION:** _02_____ **DATE:** _03/11/21____

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 ____3h______

Modeling ____7h______

Documenting ____5h______ Total ___15h____

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 feel like I earned full points for this section because I followed the requirements in the rubric and put a lot of thought into the most efficient way to model this part.
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	I feel like I earned full points for this section because I thoroughly explained my modeling process, as well as my rationale and some of the insights I gained from doing this project.
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 Multiple Sheets (w/complete set of dimensions, use of part properties, and filled out ME template w/ other title block items)	4	I feel like I earned full points for this section because my final model matches the original part drawing very closely based on the different assumptions I made, and my drawings are well thought-out and organized.

PART MINI-PROJECT SCHEDULE

Day 1 - Kick-Off (JEB 331)

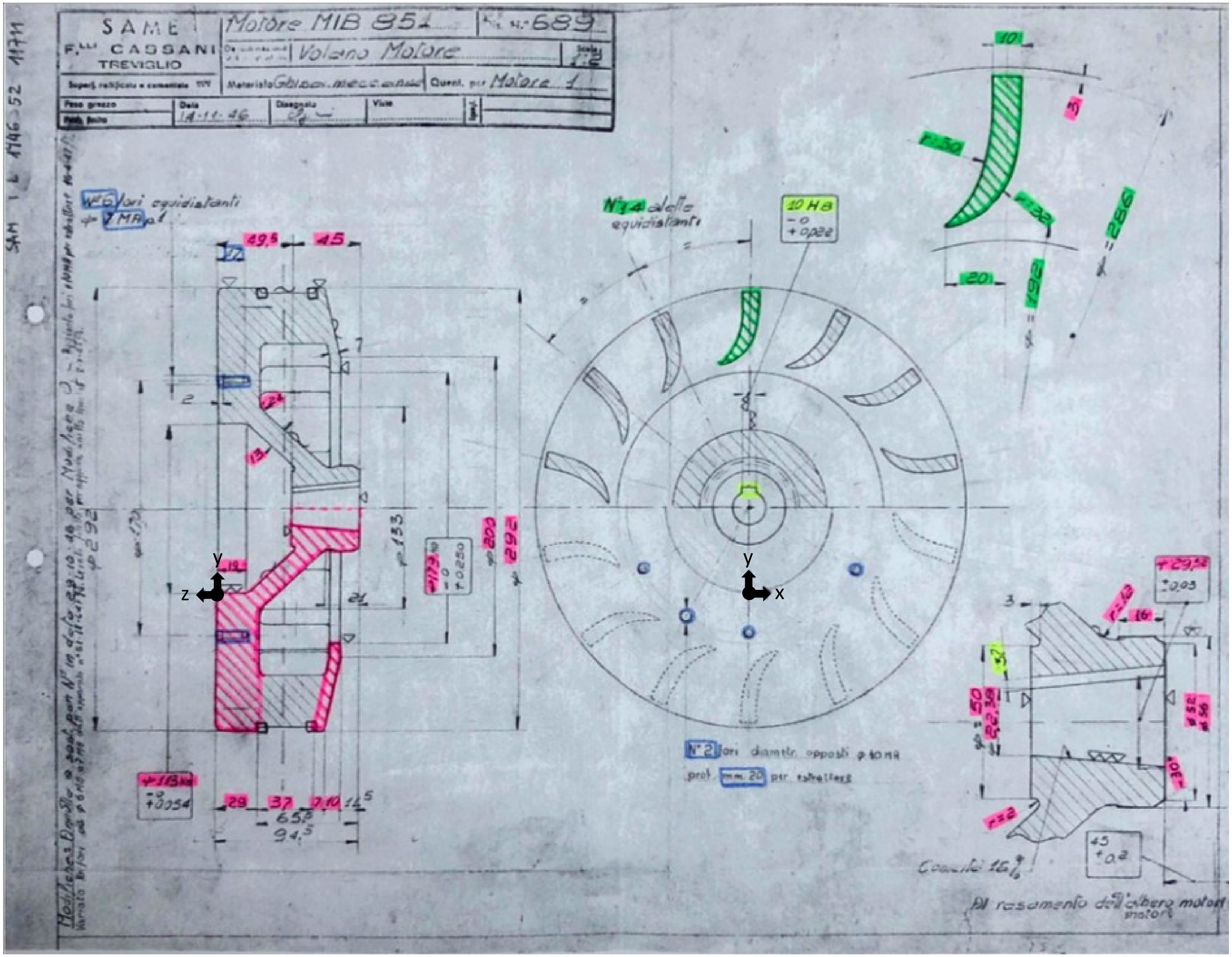
1. Part Mini Project introduction and assignment review.
2. Analyze legacy drawing and ask questions.
3. Pre-CAD: specify planes, origin, axes, reference geometry, and modeling approach. (homework)
4. Begin modeling part. (homework)
5. Log notes/assumptions you make about your part. (homework)
6. Inventory additional consulting questions you would like answered. (homework)

Days 2 & 3 - Computer Lab Consulting (JEB 331)

7. Bring hard copy and electronic documents/files to class on flash drives.
8. Share modeling/drawing rationale and progress to date.
9. Actively participate in individual/group consultations and problem solving.
10. Finish SW model and mass/center-of-mass calculations. (homework)
11. Finish SW drawing(s) w/dimensions and annotations. (homework)
12. Have someone check your drawing and sign off in the title block. (homework)
13. Prepare a complete documentation package. (homework)
    a. pre-CAD plan
    b. process documentation
    c. finished products

Day 4 - Submit Entire Package at beginning of class March 11

Pre-CAD Plan


SAME
F.lli CASSANI
TREVIGLIO
Motore MIB 851
Volano Motore
Materiale Ghisa meccanica
Quant. per Motore 1
14-11-46
N°6 fori equidistanti
N°14 alette equidistanti
10 H8
N°2 fori diametr. opposti
prot. mm 20 per estrattore
Al rasamento dell'albero motore
y
z
x

Order of Feature Implementation:

1a: Sketch the bottom half of the right sectional view onto the right plane in Solidworks.

- I think this will be one of the hardest steps because of the unnatural shape of the right section profile. I'm guessing this step will take the longest because of all of the dimensioning and relationing between all of the different lines.

1b: Revolve the sketch around the centerline at the top right of the sketch

2a: Sketch the fin on the back face of the front of the part (note: sketch will be drawn in reverse)

- I think this step will be challenging because there aren't enough dimensions to fully define the sketch, so I will have to figure out what relations to add to fully define the sketch and make sure it is drawn correctly.

2b: Extrude the sketch to the front face of the back of the part

2c: Use the circular pattern tool to create 14 fins around the whole part (use the front face of the part as the first direction and the thickness of the part as the second direction)

3a: Sketch a rectangle on the innermost circle on the front face of the part and a second rectangle of the same size on the innermost circle on the back of the part

3b: Use the loft tool to make a lofted cut between the two sketches to finish the keyhole feature

4a: Use the hole wizard on the front face of the part to make two separate hole features for each hole type

- I think this step will be challenging because I still don't have a lot of experience with the hole-wizard. It will take a long time for me to figure out what types of holes to use to mimic the sketch profile based on the little information that was given.

4b: Use the circular pattern feature to create 6 holes of the first type and 2 holes of the second type around the front face of the part

5: Add 2mm fillets to the outer edges (unless otherwise stated) and 1mm fillets to the fins

Assumptions:

- All outer fillets are 2mm unless otherwise stated because the depth of the clearance hole seems to match the fillet radius
- Fins don't have a uniform depth between front and back of part (fins extrude to back part at an angle)
- The fillets on the diagonal bars on the right section sketch all have a radius of 24mm
- Assume fins are filleted to the primary feature and have a radius of 1mm (because outer fillets are 2mm and clearance between the fins and the edge of the part is 3mm)

Rationale for Origin Location:

I chose to place the origin in the upper left corner of the leftmost rectangle in the right section view sketch. This will allow me to have one of my sketch entities already directly related to and constrained by the origin. The other sketch entity can easily be constrained by relating it to the first sketch entity with the given dimensions.

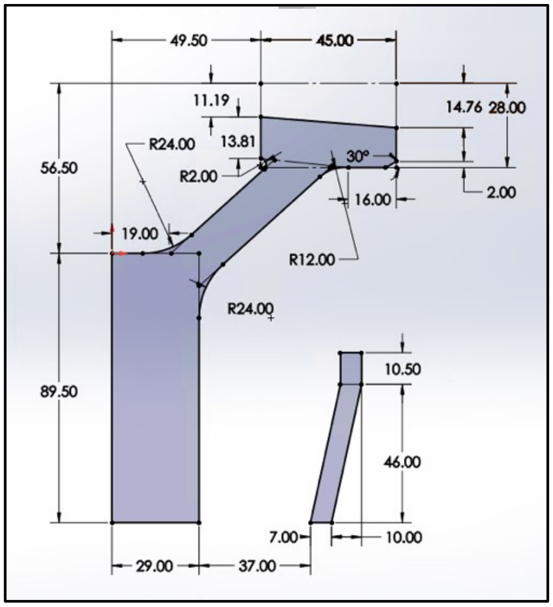
Primary Feature:

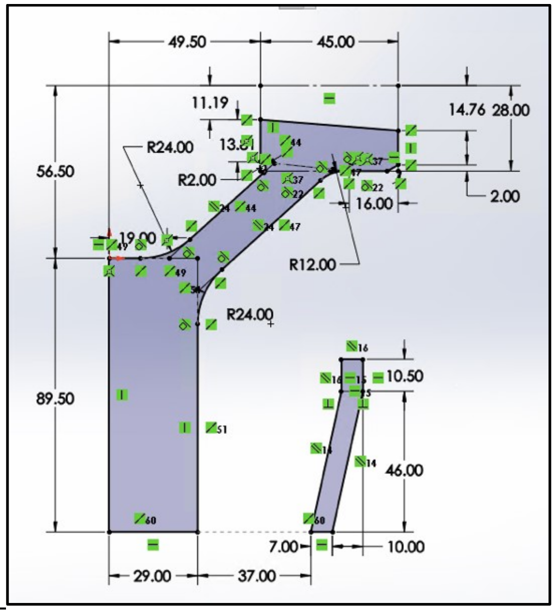
- Right Sectional View Revolve

Secondary Features:

- Fins
- Keyhole Loft
- Hole-Wizard Holes
- Fillets

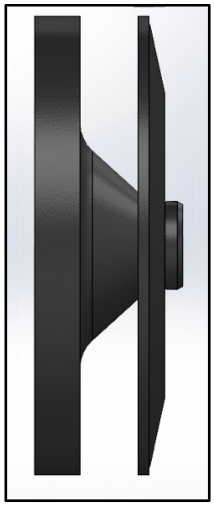
Creating the Right Section View Sketch

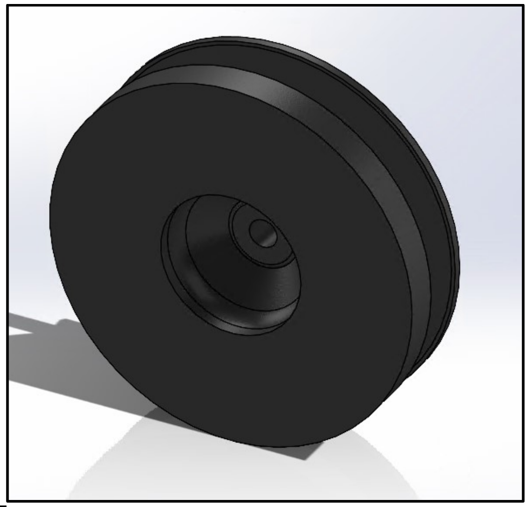




To begin making this part, I sketched the bottom half of the right section view included in the part drawing. I chose to place this sketch on the right plane in Solidworks for ease later when I needed to revolve the sketch. Many of the dimensions in the part drawing were redundant, so I only selected the ones I thought were necessary to complete the sketch (these can be seen in the Pre-CAD plan). I originally thought about placing the origin above the upper right rectangle where the centerline is because then the origin would be in the center of my whole part once it was revolved. However, this would mean that I would have to relate all of my sketches to the origin, instead of the origin being directly on one of the sketches to begin with. So, instead, I ended up choosing to place the origin in the upper left corner of the leftmost rectangle. This made it so one of my sketches was already constrained by the origin, and the other sketch could easily be constrained by relating it to the first sketch. The images above show my initial sketch with dimensions and relations.

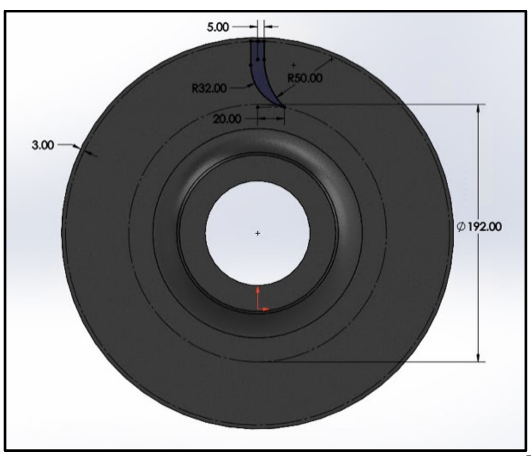
Revolving the Right Section View Sketch

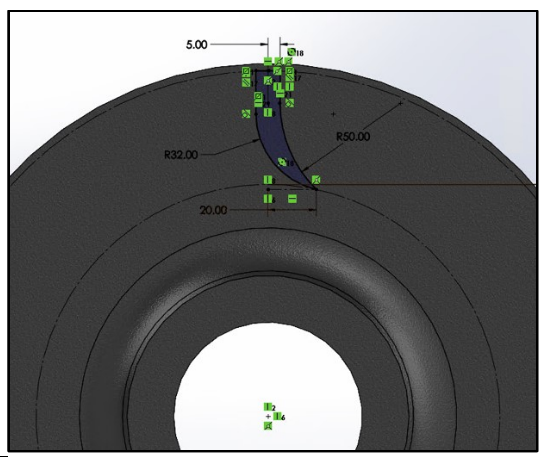




After the sketch was completed, I revolved it around the centerline at the top right of the sketch. This created my basic primary feature. I chose to use the revolve tool mainly because the part is circular, but also because even though the initial sketch was complex, it made completing and defining the bulk of the feature very easy. The above images show the primary feature after the sketch was revolved.

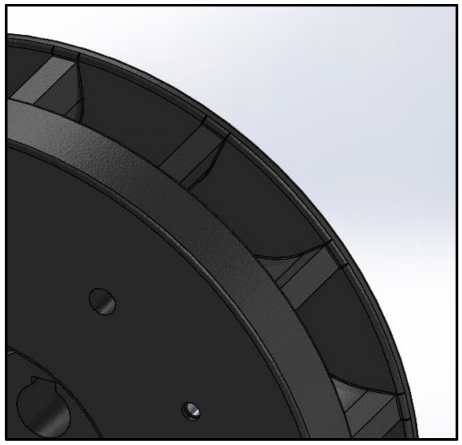
Sketching the Fins





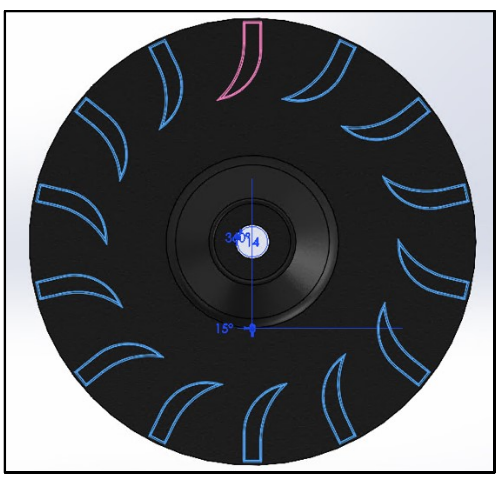
After the primary feature was created, I made a sketch for the fins, which make up one of the secondary features. I chose to sketch the fin on the back face of the front of the part so I could later extrude between the front and back of the part. However, this meant I needed to sketch the fin backwards from the orientation given in the part drawing. At first, I was concerned that we weren't given enough dimensions to make this sketch. However, after playing around with some of the relations on, I determined that the key was to make the arcs tangent to the vertical lines and to utilize circular construction geometry around the rest of the part. Using relations made it possible to make this sketch without needing the vertical fin dimensions. The above sketches show the completed fin sketch.

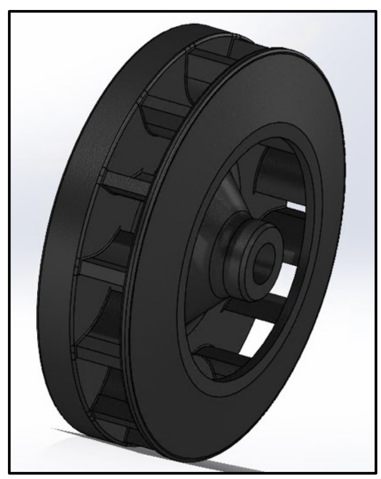
Adding Fillets to the Part



Before using a circular pattern to revolve the fins, I added fillets, which are a secondary feature, to both the fin and the outer edges of the primary feature. Since the part drawing didn’t have dimensions for all of the fillets, I had to make some assumptions. For one, it looked like the 2mm clearance hole depth matched the radius of the corner fillets, so I assumed all of the outer fillets to have a 2mm radius. The second assumption I had to make was that the fins also had fillets and that their radius was 1mm, as that was the difference between the 2mm edge fillets and the 3mm clearance between the fin and the edge of the part. The above image shows both the edge and fin fillets.

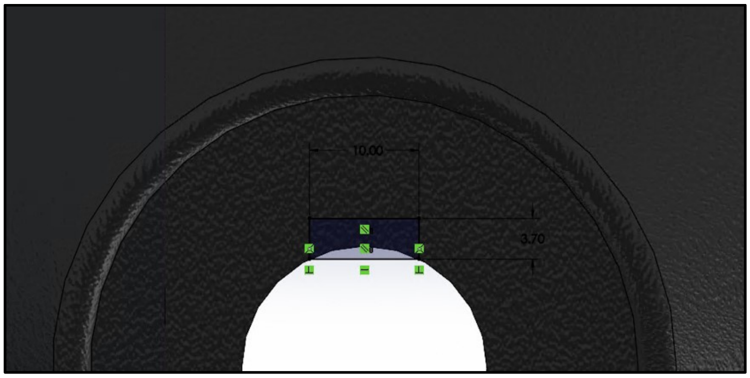
Creating a Circular Pattern for the Fins

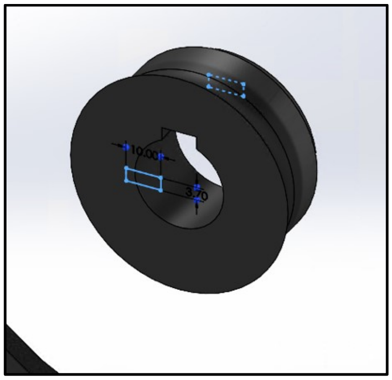




After sketching the fin, I extruded between the back face of the front of the part and the front face of the back of the part. I chose to use the extrude tool rather than make a second sketch and use the loft tool because I thought this would give better control over how the sketch extruded between each of the faces. To create the remaining 13 fins, I used the circular pattern tool and selected the front face as my first direction and the part’s thickness as my second direction to ensure the fins would pattern in the correct orientation. The above images show the circular pattern and the resulting secondary features.

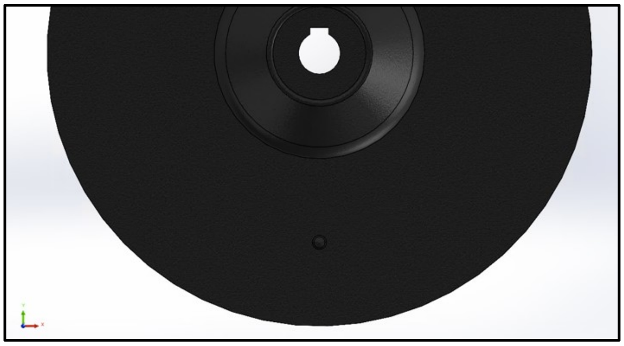
Sketching and Lofting the Keyhole

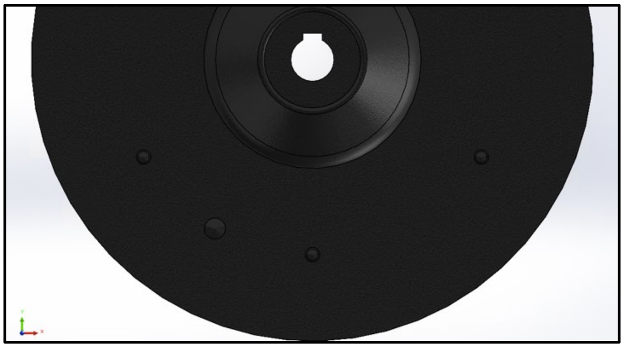




The next secondary feature I created was the keyhole. Because a loft was required for this assignment, I decided that this would be the most appropriate place to apply that technique. To begin, I sketched a 10 by 3.7 mm rectangle on the center circle on the front face of the part. I then sketched a rectangle of the same size on the center circle on the back face. The key to implementing these sketches was to make sure the rectangle was coincident with the circle at both of the bottom corners. Once the sketches were completed, I used the lofted cut tool to make the loft between the two sketches. The above images show the sketch dimensions and relations, as well as their placement on the part, and the completed loft.

Creating the Holes



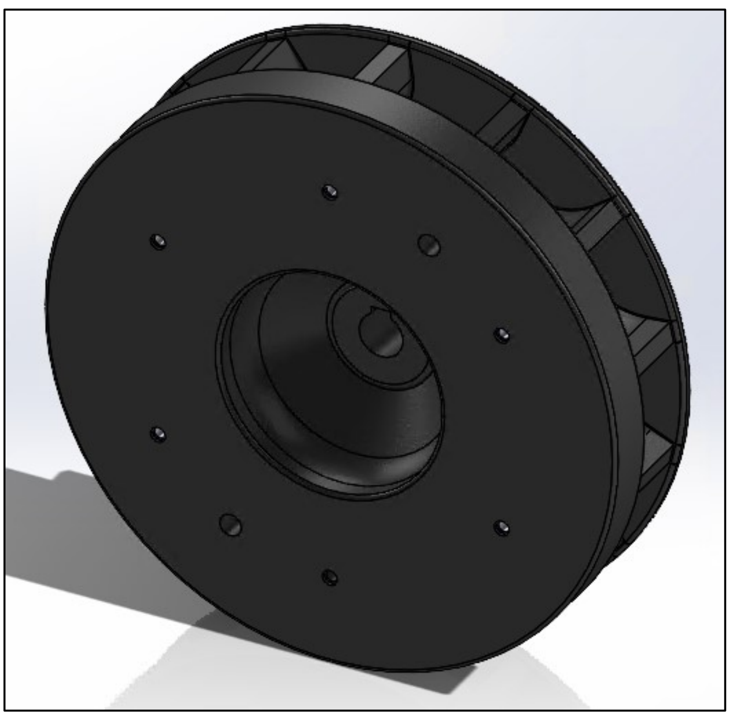


Next, I created the holes, which serve as another secondary feature. Since the dimensions and hole types were given, I used the combination of an M7x1.0 tapped hole and an M7 clearance hole to create the first hole type, and an M10 clearance hole to create the second hole type. I used the hole-wizard tool instead of doing just a cut extrude so that the holes would be designed and labeled with the information for real screw types. The above images show both hole types.

Creating a Circular Pattern for the Holes

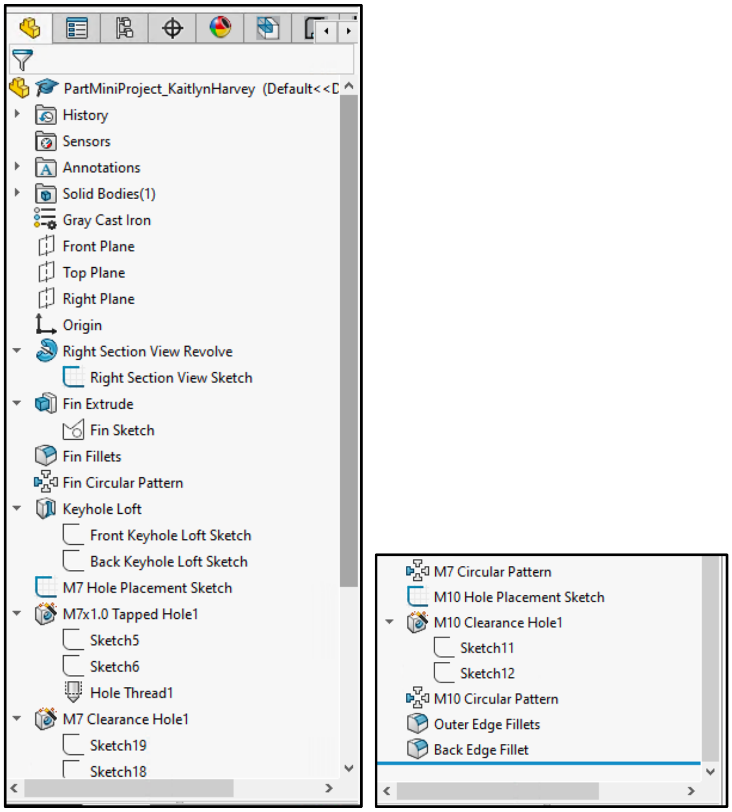
To complete the secondary hole features, I used the circular pattern tool to create six evenly space holes of the first type, and two evenly spaced holes of the second time on the front face of the part. I selected the front face as my first direction and the part's thickness as my second direction to ensure the holes would pattern in the correct orientation.

Final Product



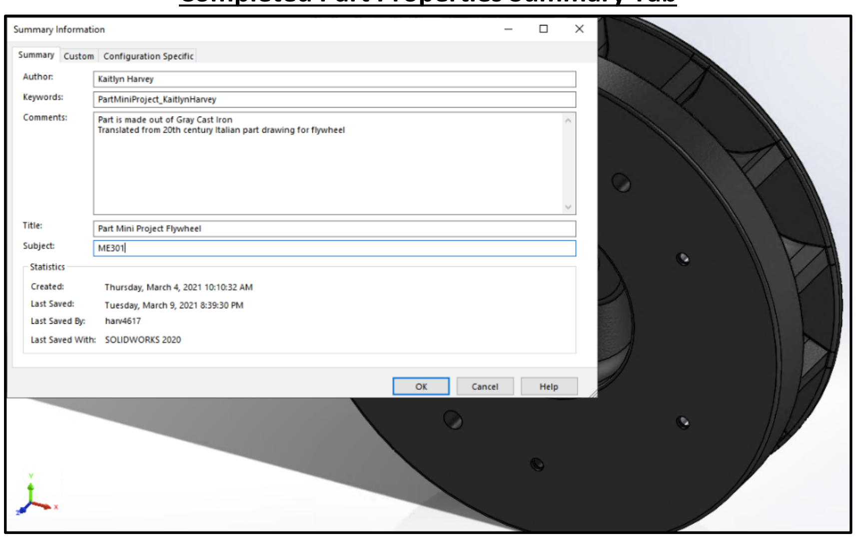
For the final product, I assigned Gray Cast Iron as the material, which is the material mentioned in the original part drawing.

Completed Feature Design Tree



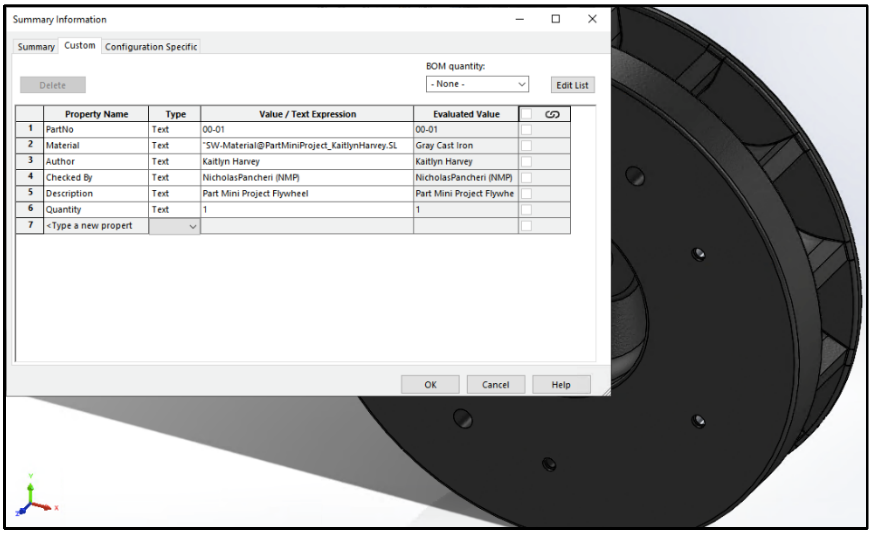
The above image shows the completed feature design tree for my part, which includes fully defined sketches, the assigned material, and annotated feature names.

Completed Part Properties Summary Tab



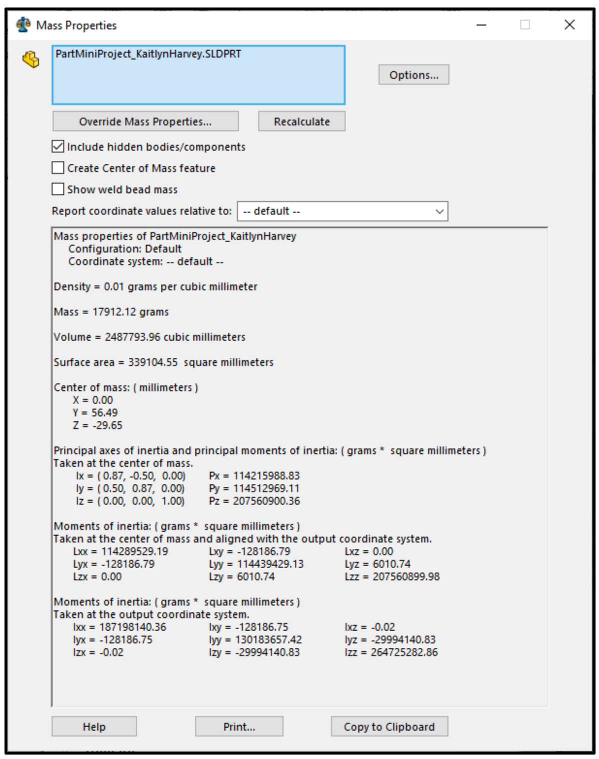
The image above shows the Part Properties Summary, which includes the author, homework number, important project notes, part title, and course name, overlaid on the completed part model.

Completed Custom Parts Tab



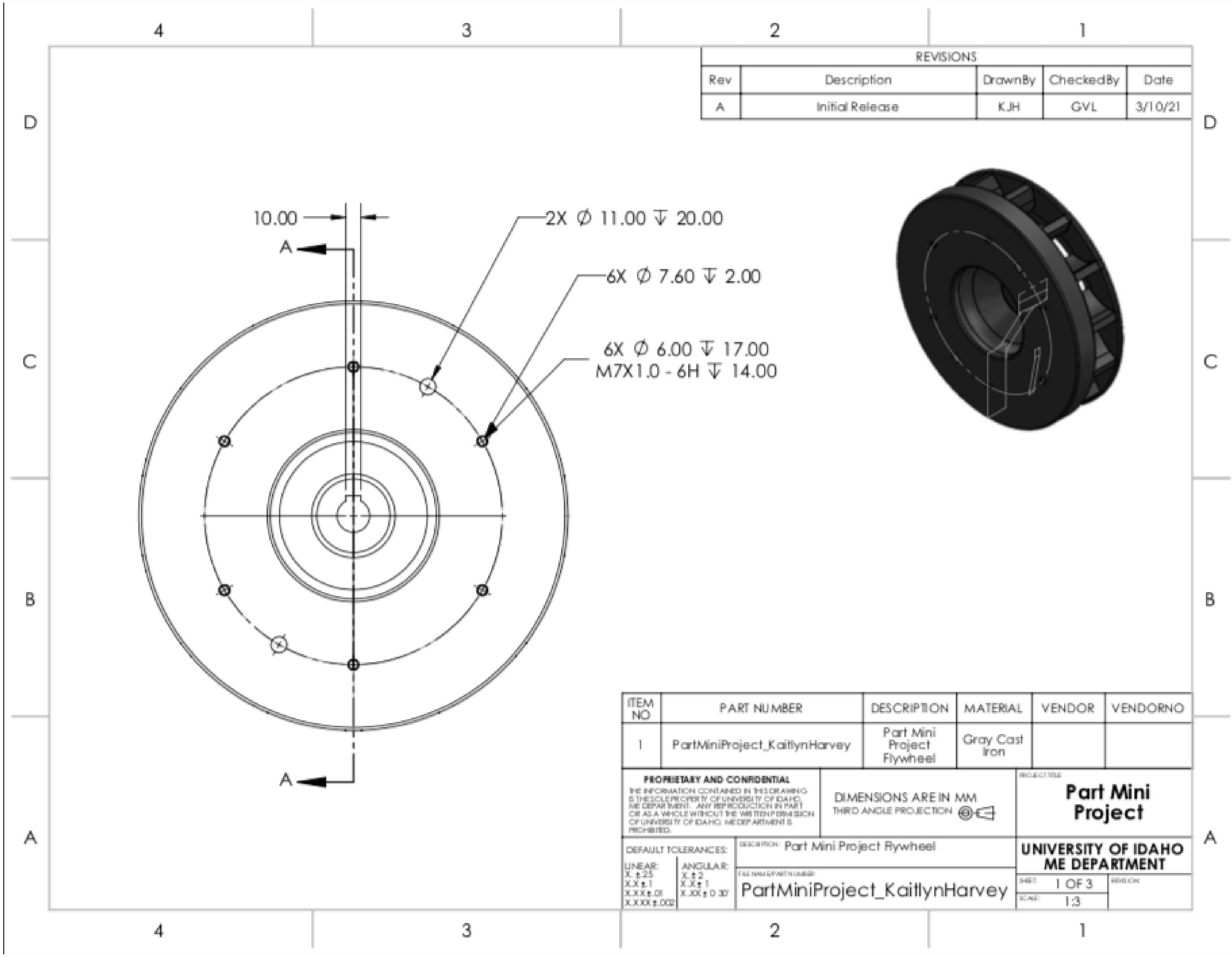
The image above shows the Custom Part Properties table, which includes the material, part number, description, and quantity of the part, as well as the author and who the part was checked by.

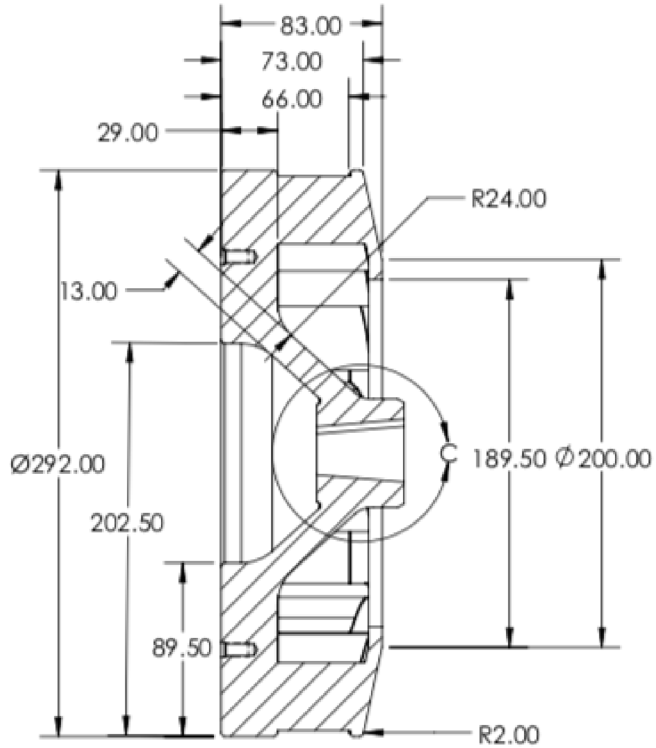
Mass Properties



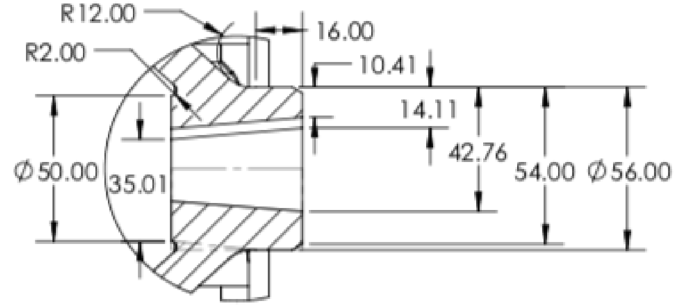
The above image shows the mass properties for the completed flywheel part, with Gray Cast Iron as the assigned material.

Shop Drawings





SECTION A-A
SCALE 1 : 3



DETAIL C
SCALE 1 : 2

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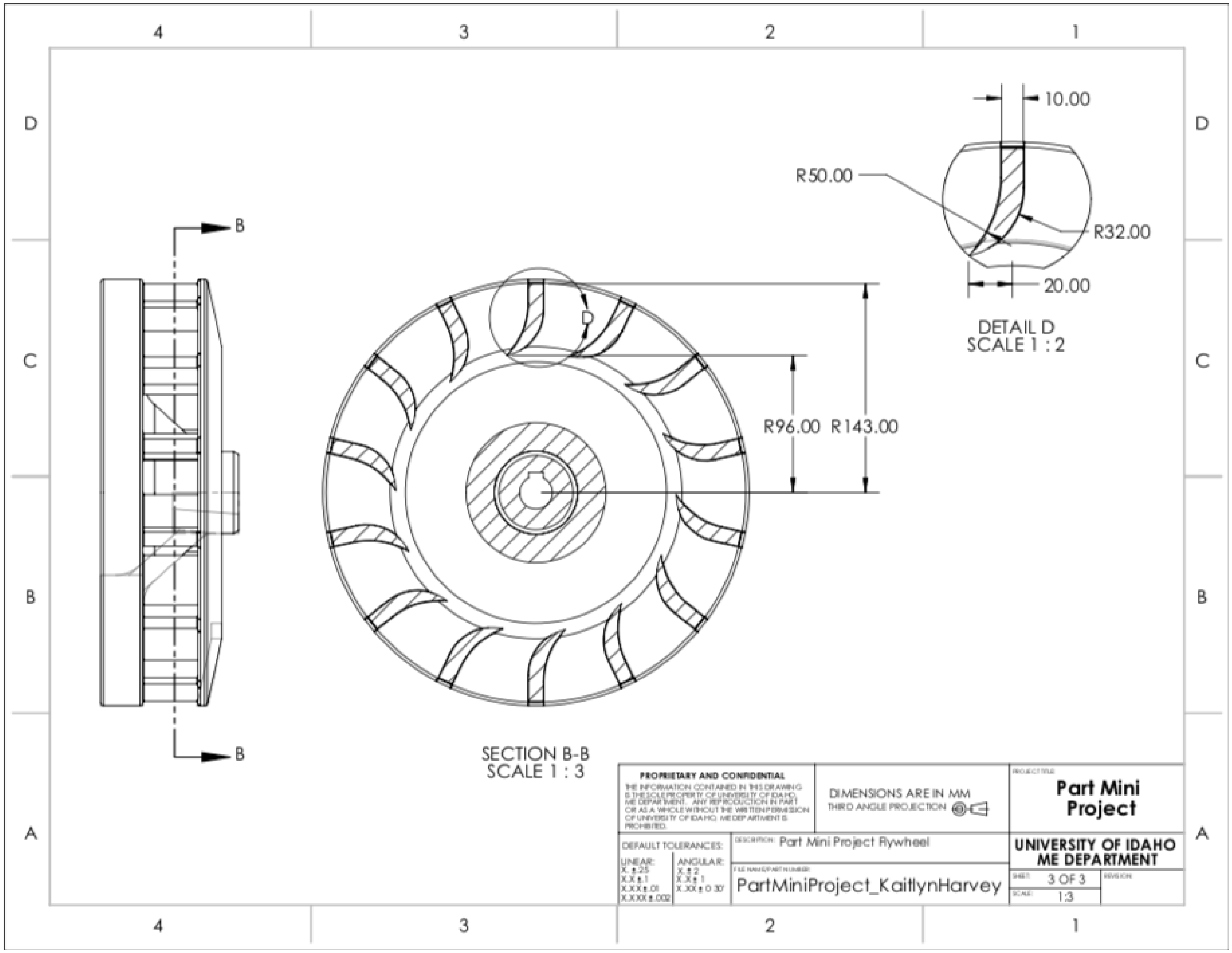
DEFAULT TOLERANCES: LINEAR: X. ±.25, X.X ±.1, X.XX ±.01, X.XXX ±.002 ANGULAR: X. ±2, X.X ±1, X.XX ± 0 30'	DESCRIPTION: Part Mini Project Flywheel FILE NAME/PART NUMBER: PartMiniProject_KaitlynHarvey	**UNIVERSITY OF IDAHO ME DEPARTMENT** SHEET: 2 OF 3 SCALE: 1:3 REVISION:



The images above show the detailed part drawing for my Flywheel Part Mini Project. The part drawings include a front and right view, as well as an isometric view to show the completed final part. I also included several section views to showcase the inner workings of the part, specifically concerning the hole features, fin geometry, and the sectional part geometry. Unlike the original part drawing, I generally used baseline dimensioning, rather than just dimensioning the diameters, to ensure there would be no tolerance stacking.

Lessons Learned

1. The first lesson I learned is that sometimes you have to make assumptions when going from a part drawing to a solid model, particularly when there isn't enough information given. The best way to do this is to look at the other given dimensions and geometries and estimate what the most likely or best-fitting dimensions would be. For example, I made the assumption that my outer fillets would be 2mm, so because the clearance between the edge of the part and the fins was 3mm, I was able to determine the fin fillets would have to be 1mm. You can also use Solidworks to figure out some of the missing dimensions for future reference by using relations until the part snaps together properly and is fully defined. This will be useful in the future because I'm sure I will have to deal with part drawings that are missing information and will have to make my best educated guess.
2. Aside from making assumptions, I also learned that sometimes part drawings will have extraneous or conflicting information. In these cases, it is best to first consider the main function or importance of the part. From there, you can see what features are the most important and require the most care with dimensioning. Then you can decide which dimensions to use and which dimensions to leave out because they might over-define the part. I think this is an important skill to develop because it requires you to think critically and actually understand the part you are modeling, instead of just blindly using dimensions for a sketch.
3. I also learned that you can use two hole-wizard features to make one hole. In this project, the part required a hole that had a clearance hole depth as well as an M7 hole, which meant that I could only get the correct profile by making two separate holes. This was more useful than making a cut extrude and then creating a hole inside of it because the hole-wizard is designed to make holes that match the specifications for real screws. So, both depths of the hole were fitted and dimensioned to include real screw types. This is useful for the future in case I encounter any other holes that require a clearance depth above the actual screw depth.