FEM Modeling Project #2

Yuhao Dong

ME I 6500

Professor Gollins

4/10/2018

Overview:

In this project, I evaluated the design of a pulley with an attached shaft by using Solidworks Simulation. For this design, I used finite element method to analyze the stress and displacement in a situation that a shaft was connected to a motor, and an obstacle had penetrated one of the holes of the pulley. The pulley system I used for this project was shown in figure 1. The initial boundary conditions were that a torque of 100 N-m was applied to the flat outer face of the shaft and restrained one of the rectangular surfaces of the holes. The three main topics for this project were to find the correct boundary conditions for the physical situation it represented, test if the FEM solution was valid, and modify the design if the FEM solution did not converge.

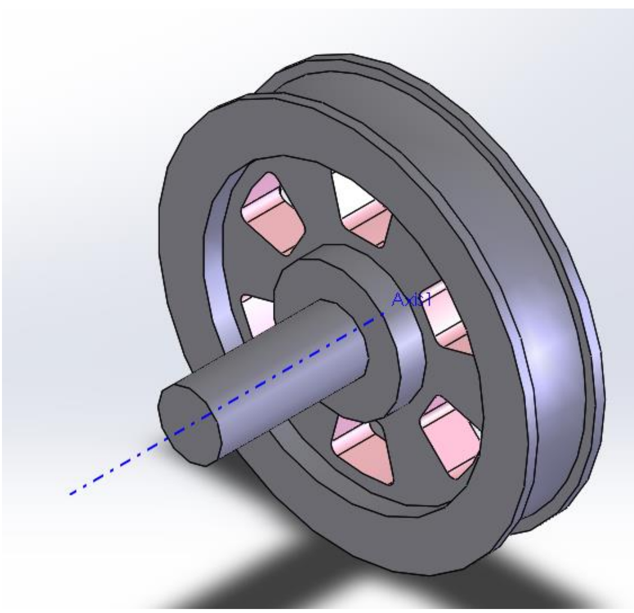


Figure 1: The pulley system used for this project

Background

Before I started to model the pulley, I had made a prediction on where were the maximum stress and displacement located based on the pulley I saw in class and other pulleys I have seen. Figure 2a was one of the pulleys I saw in class. Figure 2b was a 3D printed pulley I had in my house. I attached a pen to this 3D printed pulley to see what would happen if I rotated the pen and retrained the pulley. Since the obstacle had penetrated one of the holes of the pulley (fixed geometry), the shaft edge which was further away from the pulley (closer to the motor) should deform the most (due to rotation). Therefore, the edge of the shaft that connected to the motor should have the maximum displacement, and the maximum stress should be at the edge of the shaft where intersected with the pulley. The figure 2c showed the location where the maximum deformation would take place (circled in red), and figure 2d showed the location where the maximum stress would take place (circled in green).

Figure 2a: pulley in class

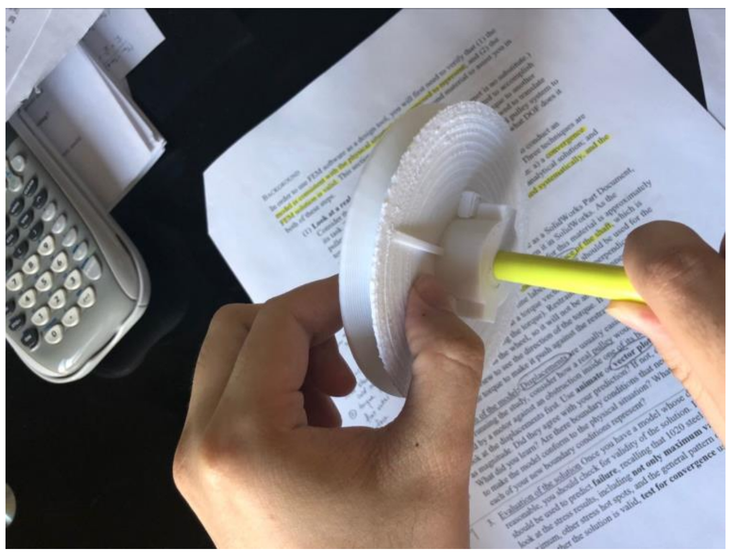


Figure 2b: 3D printed pulley

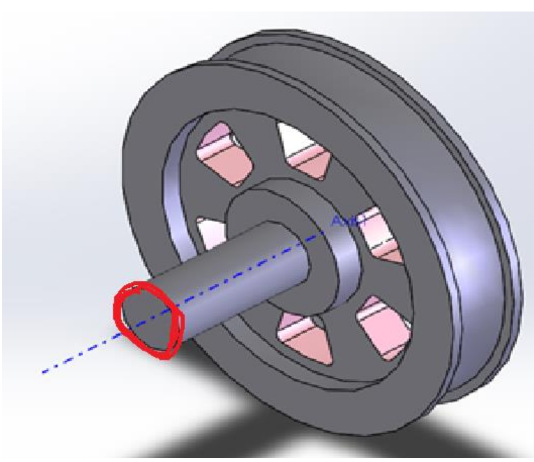


Figure 2c: location for the max. displacement

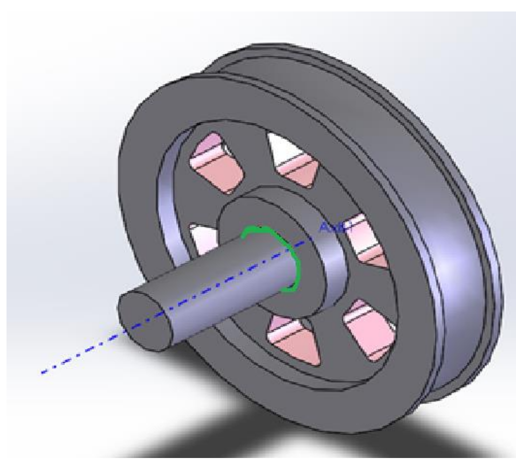


Figure 2d: location for the max. stress

Procedure

After I had made a prediction, I obtained the Solidworks pulley model from blackboard. Then I picked AISI 1020 steel, cold rolled as the material of the pulley and the shaft. The material properties were: Poisson's ratio: 0.29, elastic modulus: $2.05*10^{11}$ N/m^2, density: 7870 kg/m^3, and yield strength: 350000000 N/m^2 (350 MPa). Then I set the initial boundary conditions, which were a torque of 100 N-m was applied to the flat outer face of the shaft and restrained one of the rectangular surfaces of the holes (figure 3).

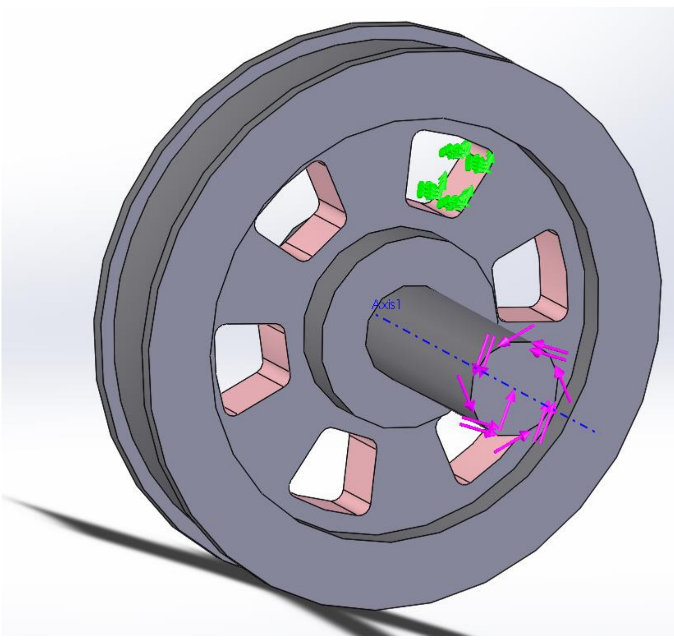


Figure 3: initial boundary conditions for this project

After I chose the material and set up the initial boundary conditions, I was ready to run the FEM simulations. Since in this step, I only needed to find the maximum displacement to see if the initial boundary conditions were correct, the mash size did not matter that much. The mesh size I chose was 4 mm. After I ran the simulation, the displacement graph I obtained was shown below in figure 4, which did not match with my assumption. Based on the graph, the maximum displacement was located on the bottom of the pulley. The entire pulley was shifted to the right, especially the bottom part of the pulley. Moreover, the center of the shaft even shifted to the right as well (The torque applied in purple color did not match with the pulley after it was deformed as shown in figure 4).

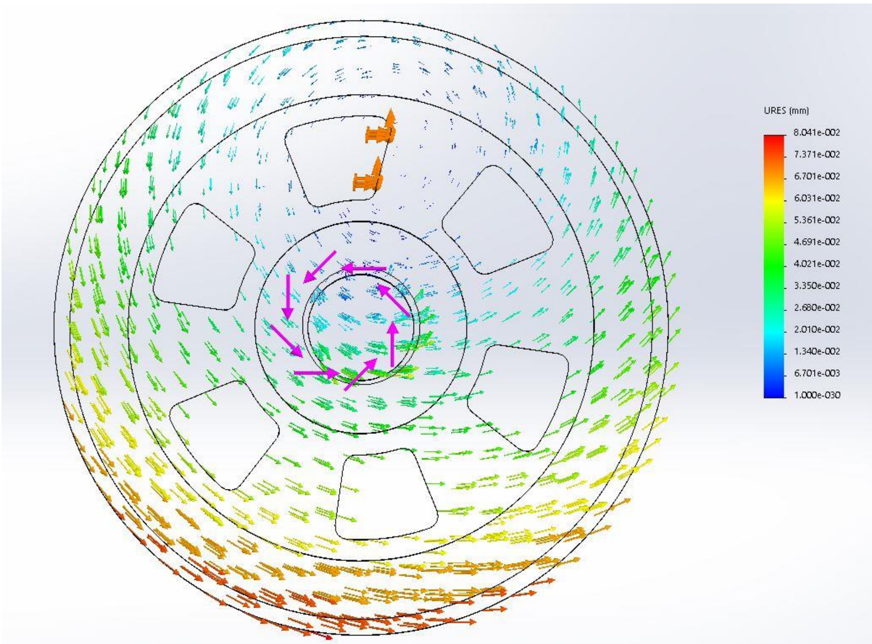
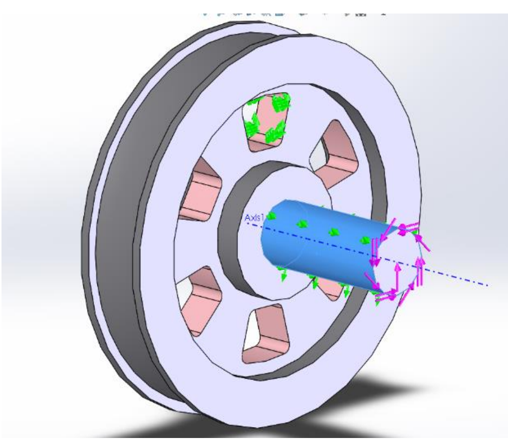


Figure 4: Displacement graph for initial boundary conditions

Based on the information I got above, I thought that the initial boundary conditions did not prevent shaft from shifting, and the boundary conditions should be modified. The shaft should not move or translate to neither x or y direction. Instead, it should only rotate along the z axis (out of the page). Therefore, I treated the entire shaft as a fixed hinge, which only allowed rotation to take place (Figure 5a & 5b).



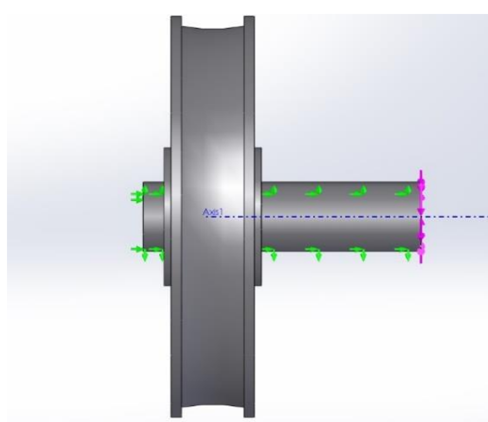


Figure 5a: modified boundary conditions *Figure 5b: side view of the modified b.c.*

After I modified the boundary conditions, I ran the simulation again, and checked if the displacements were correct. The displacement graph I got were shown below. These graphs matched with my assumption. The maximum displacements were occurred at the outer edge of the shaft (connecting to the motor).

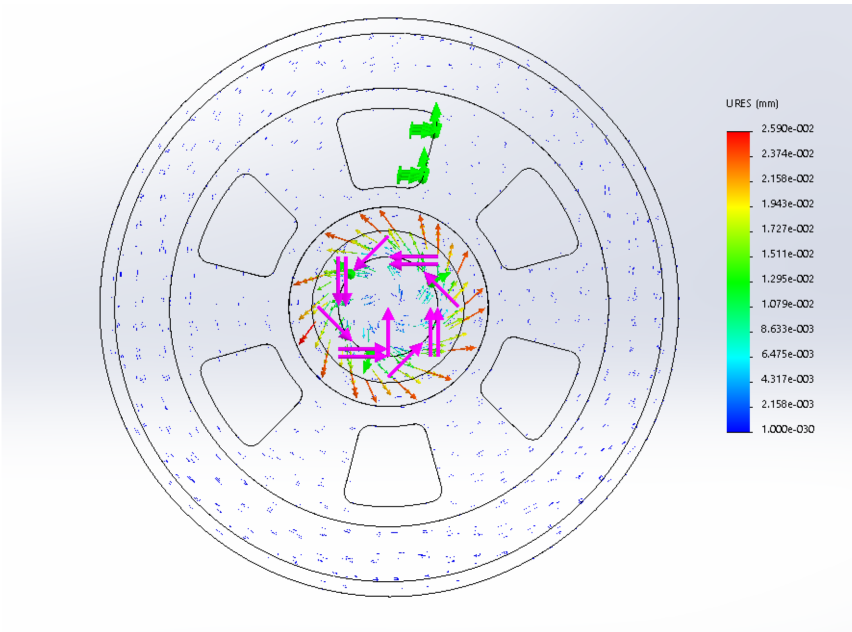


Figure 6a: Front view of the displacement graph for modified boundary conditions

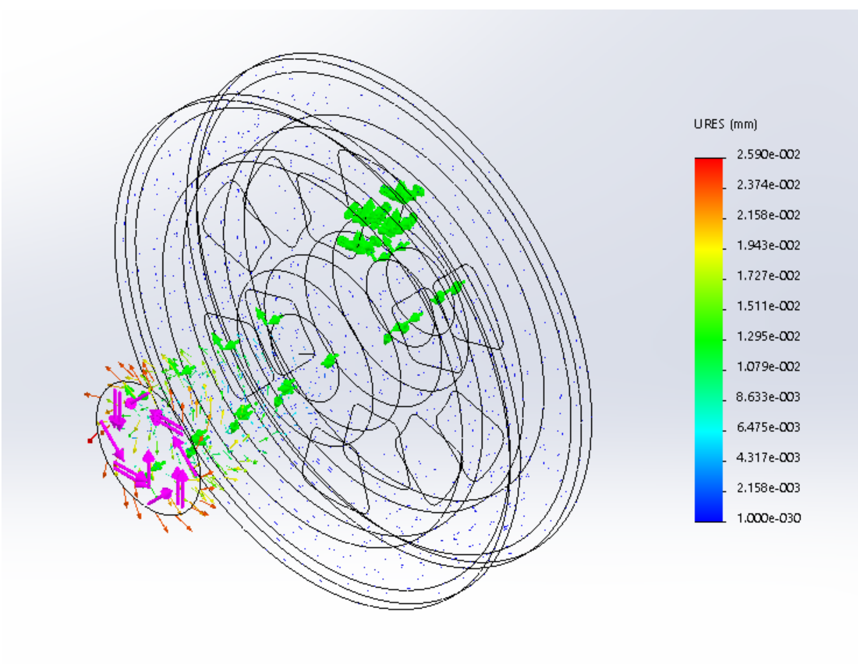


Figure 6b: Isometric view of the displacement graph for modified boundary conditions

After I obtained the correct boundary conditions, the next step was to evaluate the solution. The 1020 steel was a ductile material, so the Von Mises' theory was the best one to predict the failure. Also, the stress I used for this project was Von Mises Stress. The figure 7a was the picture of the major failure theories I learned from Professor Benenson's class. Then I checked the stress graph I obtained before (figure 7b), and it showed that the maximum stress was at the edge of the shaft where the intersected with the pulley. Moreover, I noticed that the amounts of stresses were almost the same along the shaft. It was because that the cross-sectional area of the shaft was constant.

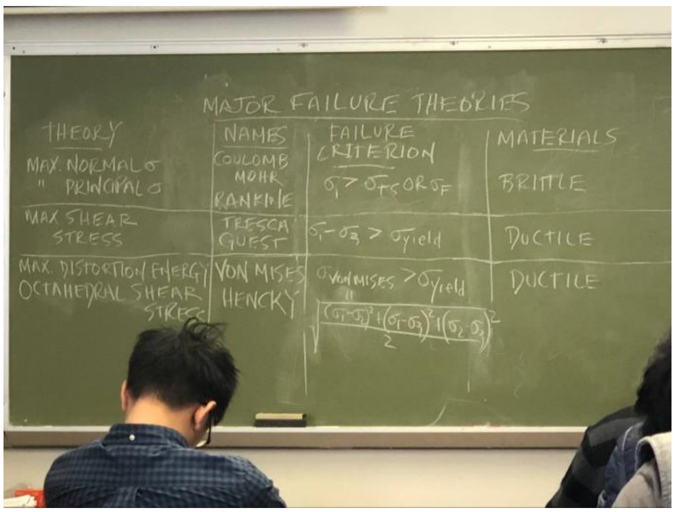


Figure 7a: Major Failure Theories from Professor Benenson's class

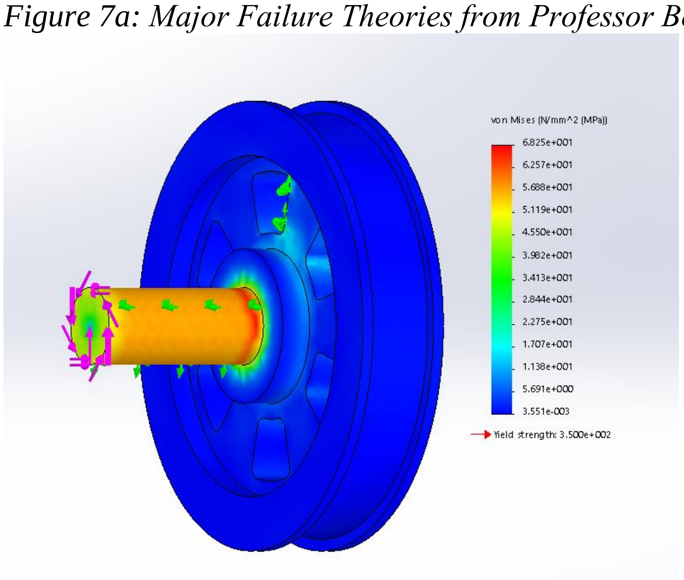


Figure 7b: Stress graph for modified boundary conditions

Since I knew the maximum stress would probably occur at the edge of the shaft that connected to the pulley, I used local refinement method at this place. The global mesh size I used was 4 mm, and I decreased the local element size from 3 mm to 0.5 mm to see if the result converged (The results were shown in table 1). The location for the local element was shown in figure 8a. Also, from figure 7, I saw that the maximum stress did not occur exactly on the edge. Instead, there was a little bit space between the location of the maximum stress and the edge. I did not know if it was because of the inaccuracy of the mesh size, or it was because of any other reasons, so I also used the local refinement method for the entire shaft surface (The results were shown in table 2). The location for the local element was shown in figure 8b.

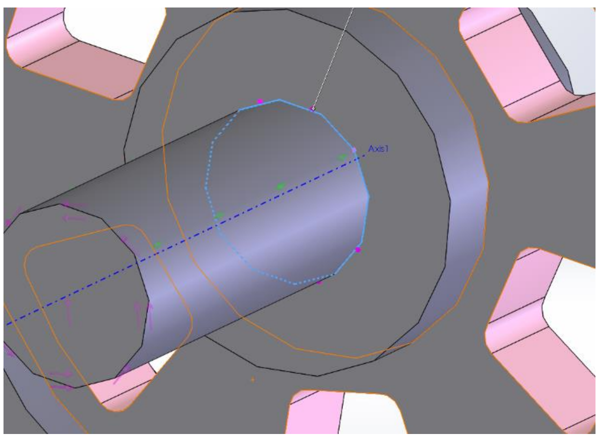


Figure 8a: Edge (blue color) for local refinement

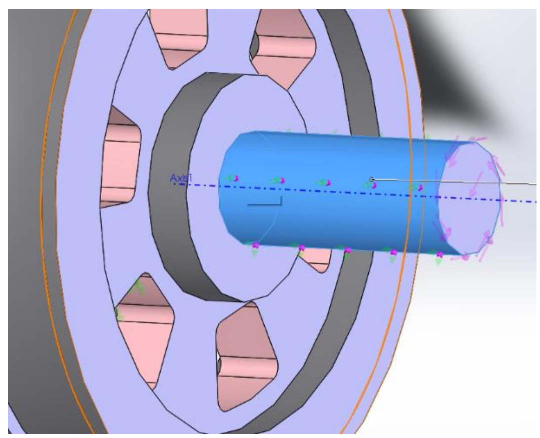


Figure8b: surface for local refinement

The results and the plotted graphs indicated that the results were not converged. As the element size got smaller, the number of nodes got larger, the maximum stress also increased. Therefore, I needed to modify the design a little bit. I thought the reason for the results not converging was the poorly shape element. The edge connecting the shaft and the pully was too sharp (90^o). It could cause some problems. Then I planned to add a fillet to this edge (figure 8c). There were two questions I had with adding a fillet. The first question was whether the result converged or not if adding a fillet. The second question was what size should the fillet be. To answer these two questions, I started to run the Solidworks Simulation again.

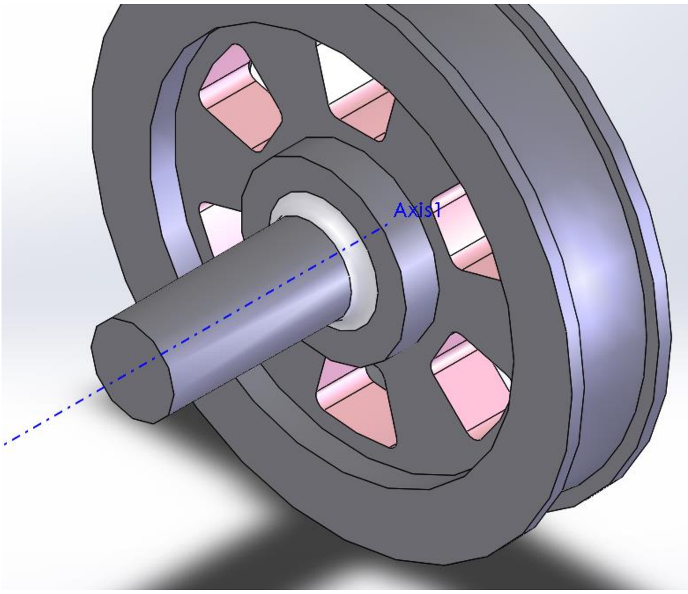


Figure 8c: A 4 mm fillet was added to the system

The first fillet I tested had a radius of 4 mm. The global element size I used was 4 mm, and I decreased the local element size from 3 mm to 0.5 mm to see if the result converged (The results were shown in table 3). The location I chose for the local elements was the fillet area [To make sure if the result converged, I also tried the local element of 0.43 mm]. After obtained the results, I tried another set with global mesh size of 8 mm, and decreased the local element size from 6 mm to 0.5 mm (The results were shown in table 4). I tried the second test because I was not very sure if 4 mm global size was too fine to mesh or not.

I also tried with different fillet sizes. The 2nd one I tried was the fillet with radius of 7 mm. I used the same global mesh size (4 mm), and decreased the local element size from 3 mm to 0.5 mm (The results were shown in table 5). Then I also tested with the fillet with the radius of 8 mm (table 6) and 12 mm. For 12 mm radius fillet, I also test it twice with different global mesh size. The 1st one had the global size of 4 mm (The results were shown in table 7), and the 2nd one had the global size of 6 mm (The results were shown in table 8). Moreover, I tried radius of 15 mm (table 9) and 18 mm (table 10). I then recorded all the information in the table. Also I plotted graphs of maximum stress against total number of nodes for all these cases I did to see if the result converges or not. Also I tried to find the relation between fillet radius and the maximum stress. Then I compared the stresses I obtained from this experiment to analytical solutions so see if the FEM solution worked or not. I also compared the Von Mises Stress I got to the yield stress of the material with a safety factor of 2. The Von Mises Stress could be calculated by using equation $\sqrt{\frac{(\sigma_1-\sigma_2)^2+(\sigma_1-\sigma_3)^2+(\sigma_2-\sigma_3)^2}{2}}$. However, since it was calculated by computer (Solidworks Simulation), I did not actually have to calculate it. I could just obtain it from the result which all the data converged to, but I could use this formula to solve for the analytical solution and compare with the results I obtained from the FEM.

Results

1. Local refinement on the edge (Without the fillet)

Table 1: Data collected for local refinement on the edge (Without the fillet)

Case #	Global element size (mm)	Local element size (mm)	Location for local refine-ment	Total # of elements (10^3)	Total # of nodes (10^3)	DOF (10^3)	Max stress (MPa)	Max displace-ment (mm) [10^{-2}]	Run time (s)
	4			39.589	61.498	1842.51	71.05	2.579	5
1	4	3	edge	40.309	62.455	187.122	72.23	2.580	5
2	4	2.5	edge	41.709	64.377	192.888	74.41	2.582	5
3	4	2	edge	44.137	67.706	202.875	78.20	2.583	6
4	4	1.5	edge	47.266	72.095	216.042	87.20	2.584	6
5	4	1	edge	53.734	80.908	242.481	95.55	2.585	7
6	4	0.5	edge	71.541	105.661	316.740	122.6	2.587	8

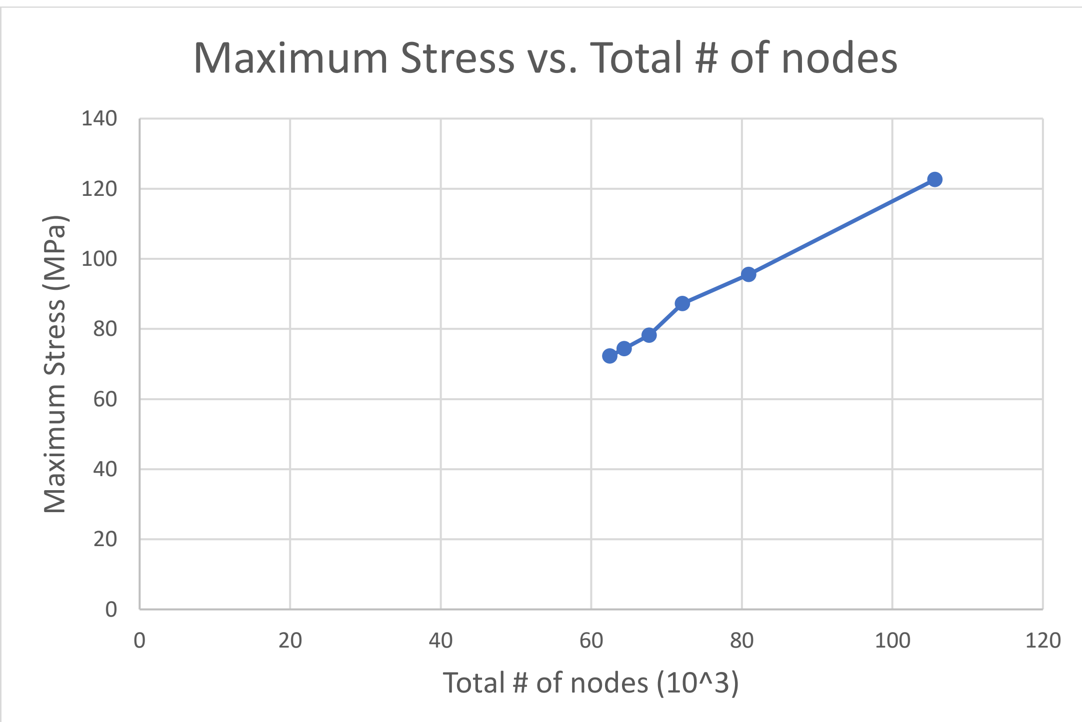


Figure 9: Maximum stress vs total number of nodes for different global element size (edge)

This graph showed that the maximum stress did not converge. As the element size got smaller, the number of nodes got larger, the maximum stress also increased.

2. Local refinement on the surface (Without the fillet)

Table 2: Data collected for local refinement on the edge (Without the fillet)

Case #	Global element size (mm)	Local element size (mm)	Location for local refine-ment	Total # of elements (10^3)	Total # of nodes (10^3)	DOF (10^4)	Max stress (MPa)	Max displace-ment (mm) [10^{-2}]	Run time (s)
1	4	3	surface	44.500	68.529	20.5344	78.57	2.575	6
2	4	2.5	surface	47.635	73.006	21.8775	86.24	2.572	6
3	4	2	surface	57.133	86.591	25.9530	93.20	2.572	7
4	4	1.5	surface	77.260	115.303	34.5666	109.0	2.571	10
5	4	1	surface	137.718	201.114	60.3099	100.2	2.572	16
6	4	0.5	surface	481.633	686.445	205.9092	154.0	2.571	60

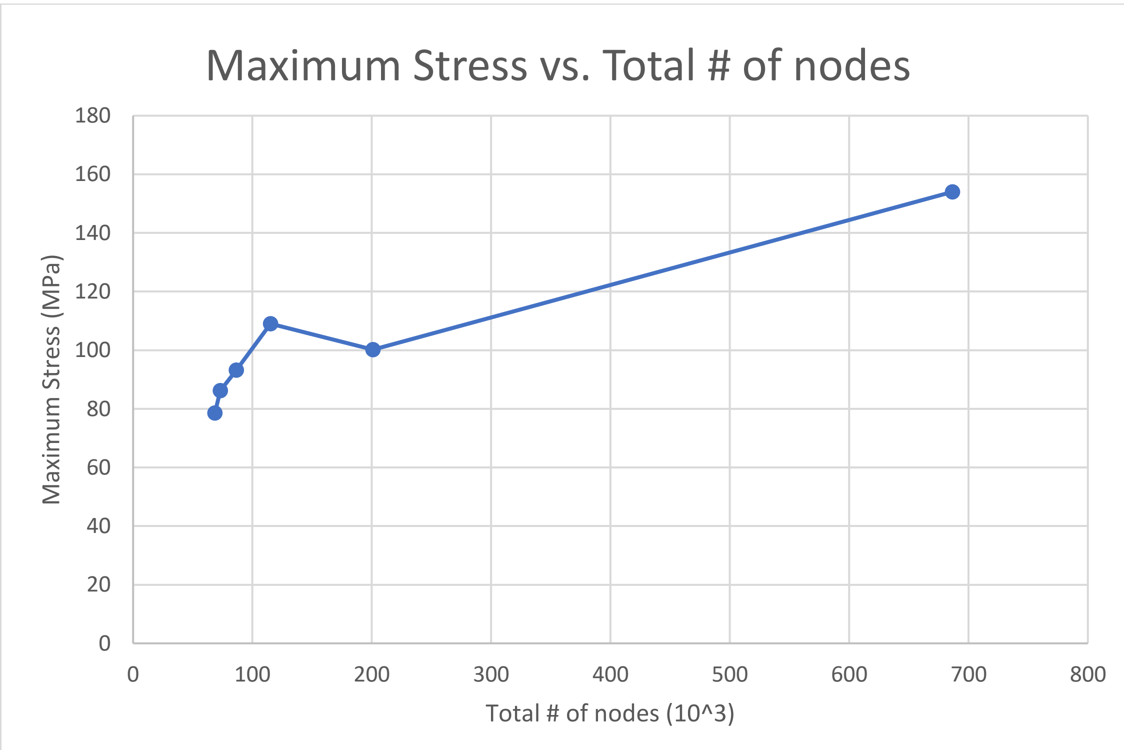


Figure 10: Maximum stress vs total number of nodes for different global element size (surface)

This graph also indicated that the maximum stress did not converge. As the element size got smaller, the number of nodes got larger, the maximum stress also increased. Since both graphs showed that the result did not converge, the design should be modified.

3. Local refinement for fillet of 4 mm, and global mesh size of 4 mm

Table 3: Data for local refinement on the fillet (radius of 4 mm, and global mesh size of 4 mm)

Case #	Global element size (mm)	Local element size (mm)	Location for local refine-ment	Total # of elements (10^3)	Total # of nodes (10^3)	DOF (10^3)	Max stress (MPa)	Max displace-ment (mm) [10^{-2}]	Run time (s)
1	4	3	fillet	42.004	64.835	194.262	74.65	2.522	6
2	4	2.5	fillet	44.114	67.751	203.010	74.91	2.521	5
3	4	2	fillet	44.712	68.663	205.746	75.90	2.522	5
4	4	1.5	fillet	51.696	78.332	234.753	75.96	2.521	6
5	4	1	fillet	62.195	93.236	279.465	76.82	2.522	7
6	4	0.5	fillet	109.033	160.418	481.011	76.79	2.522	13
7	4	0.43	Fillet	125.177	183.818	551.211	76.82	2.522	15

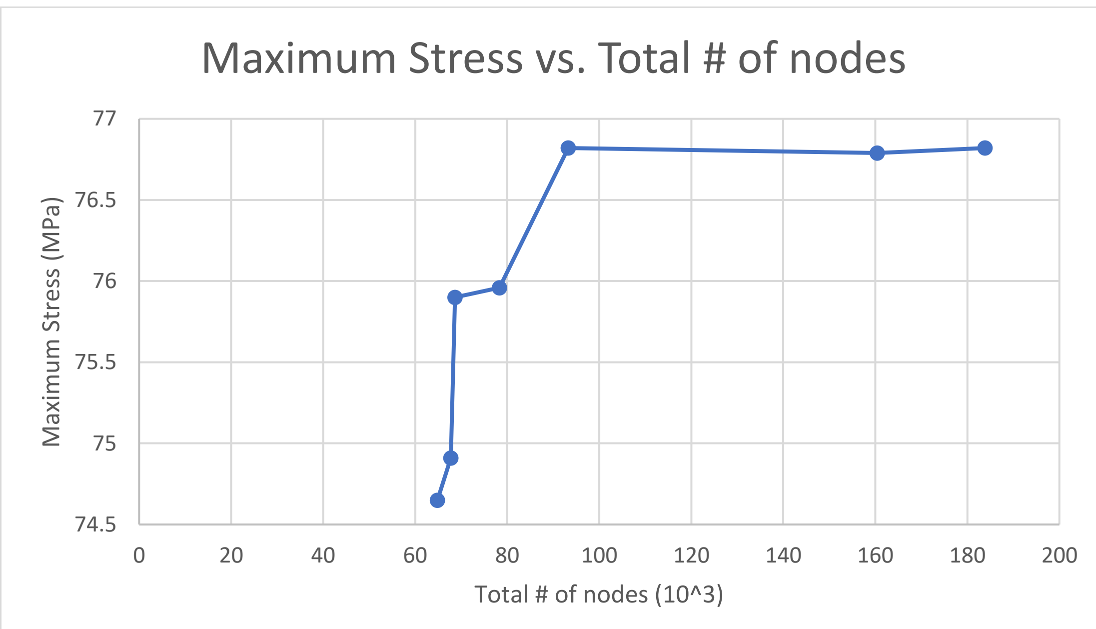


Figure 11: Maximum stress vs total number of nodes for different global element size (r=4 mm)

This graph showed that the result converged when the number of nodes got larger. It proved that adding a fillet could make the design better. But I also wanted to do more tests to support it. Moreover, I also wanted to try more different radius for the fillet to see what was the relation between the radius of the fillet and the maximum stress.

4. Local refinement for fillet with radius of 4 mm, and global mesh size of 8 mm

Table 4: Data for local refinement on the fillet (radius of 4 mm, and global mesh size of 8 mm)

Case #	Global element size (mm)	Local element size (mm)	Location for local refine-ment	Total # of elements (10^3)	Total # of nodes (10^3)	DOF (10^3)	Max stress (MPa)	Max displace-ment (mm) [10^{-2}]	Run time (s)
1	8	6	fillet	7.420	12.725	38.100	74.43	2.55	2
2	8	4	fillet	8.383	14.122	42.291	74.24	2.54	2
3	8	2	fillet	11.586	18.825	56.400	76.77	2.55	2
4	8	1.5	fillet	14.909	23.590	70.695	77.14	2.55	2
5	8	1	fillet	24.350	37.151	111.378	77.69	2.55	3
6	8	0.5	fillet	59.250	88.423	265.194	77.62	2.55	7

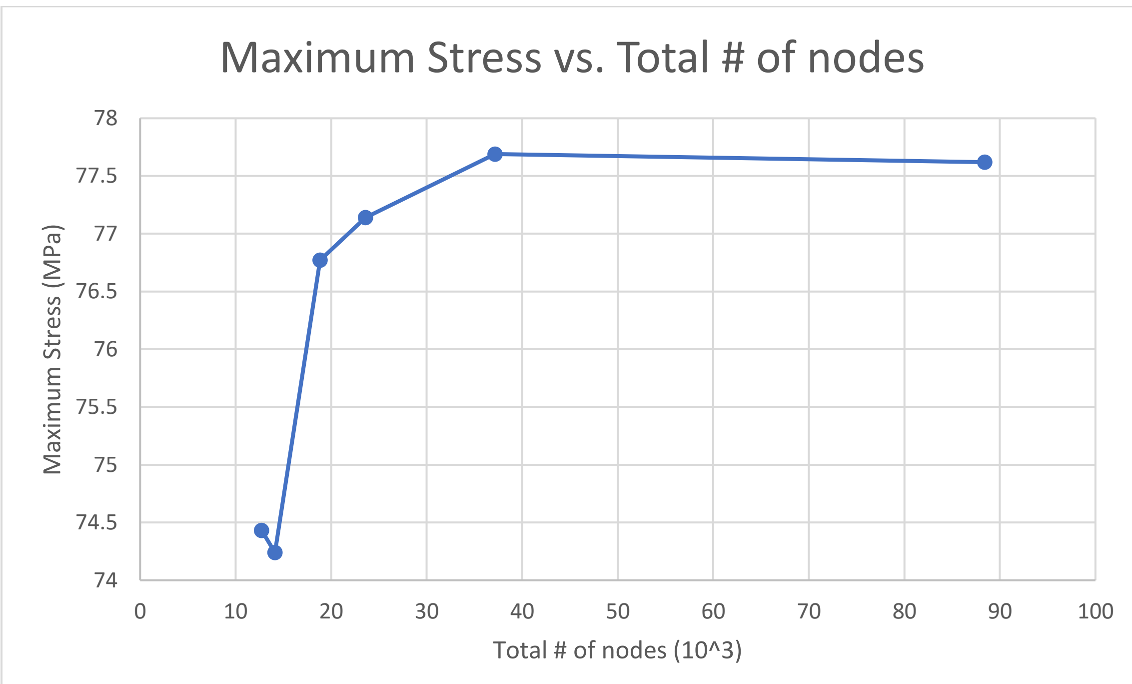


Figure 12: Maximum stress vs total number of nodes for different global element size (r=4 mm)

When I changed the global mesh size to 8 mm, the result also converged when the number of nodes got larger. It required less time to run. It was because the total number of elements, total number of nodes, total number of degrees of freedom for this simulation were all much smaller than the previous one.

5. Local refinement for fillet with radius of 7 mm, and global mesh size of 4 mm

Table 5: Data for local refinement on the fillet (radius of 7 mm, and global mesh size of 4 mm)

Case #	Global element size (mm)	Local element size (mm)	Location for local refine-ment	Total # of elements (10^3)	Total # of nodes (10^3)	DOF (10^3)	Max stress (MPa)	Max displace-ment (mm) [10^{-2}]	Run time (s)
1	4	3	fillet	42.433	65.402	195.963	68.57	2.454	4
2	4	2.5	fillet	44.270	68.013	203.796	68.88	2.454	4
3	4	2	fillet	46.614	71.400	213.957	69.78	2.454	5
4	4	1.5	fillet	52.648	79.966	239.655	70.00	2.454	6
5	4	1	fillet	69.214	103.608	310.581	70.27	2.454	9
6	4	0.5	fillet	125.801	187.867	563.358	70.36	2.454	15

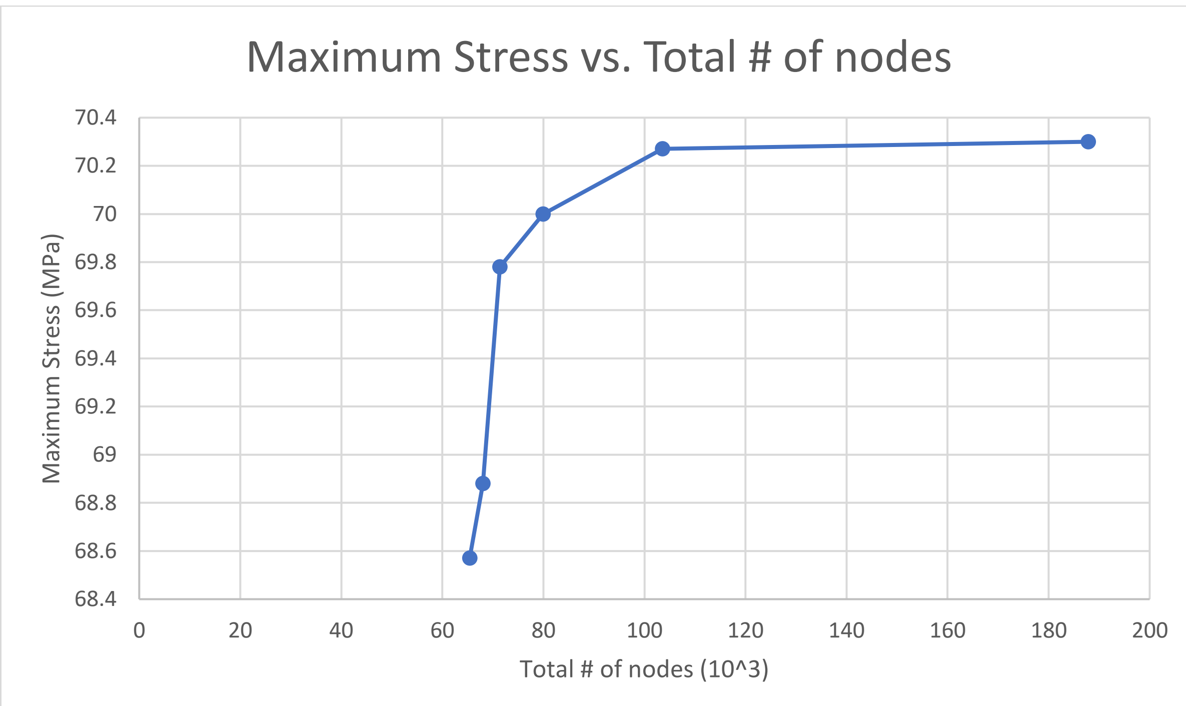


Figure 13: Maximum stress vs total number of nodes for different global element size (r=7 mm)

When I changed the fillet size to 7 mm, the result also converged. However, I noticed that when the fillet size got larger, the maximum stress decreased from 76.8 MPa to 70.3 MPa. I was still not sure if increasing the fillet size would decrease the maximum stress. Therefore, I increased the fillet size to 8 mm.

6. Local refinement fillet with radius of 8 mm, and global mesh size of 4 mm

Table 6: Data for local refinement on the fillet (radius of 8 mm, and global mesh size of 4 mm)

Case #	Global element size (mm)	Local element size (mm)	Location for local refine-ment	Total # of elements (10^3)	Total # of nodes (10^3)	DOF (10^3)	Max stress (MPa)	Max displace-ment (mm) [10^{-2}]	Run time (s)
1	4	3	fillet	42.762	65.863	197.346	68.02	2.429	5
2	4	2.5	fillet	44.804	68.747	205.998	68.04	2.430	6
3	4	2	fillet	47.263	72.299	216.654	68.46	2.429	5
4	4	1.5	fillet	54.709	82.865	248.352	68.66	2.429	6
5	4	1	fillet	69.754	104.753	314.016	69.39	2.429	8
6	4	0.5	fillet	141.813	209.279	627.594	69.30	2.429	17

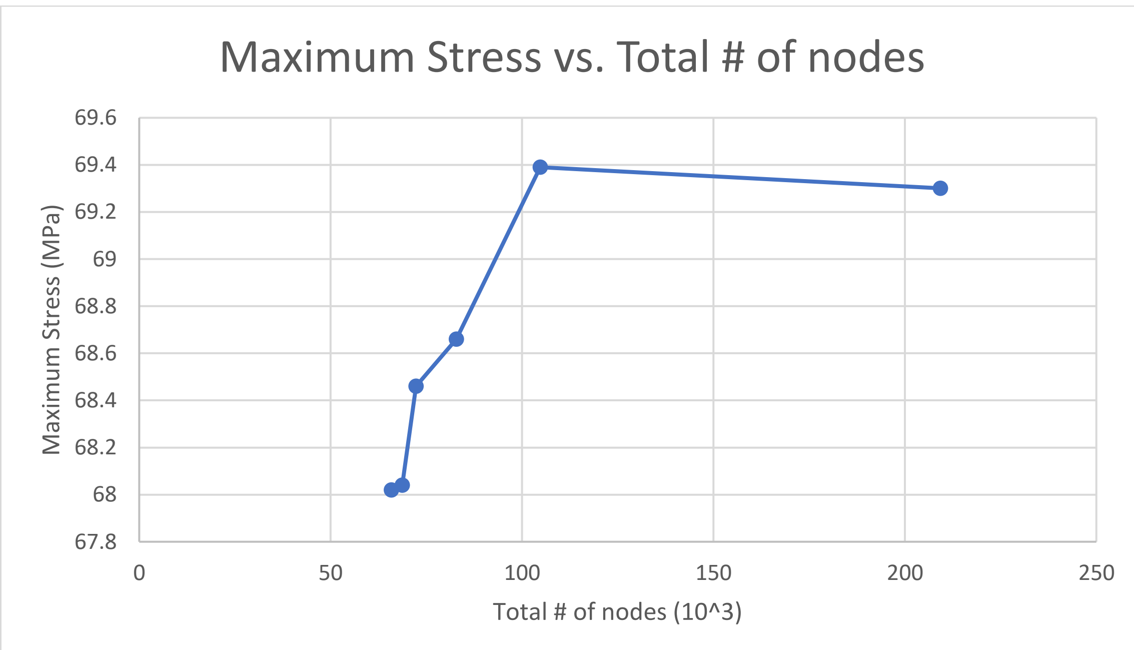


Figure 14: Maximum stress vs total number of nodes for different global element size (r=8 mm)

When I changed the fillet size to 8 mm, the result also converged. The maximum stress also decreased this time from 70.3 to 69.3 MPa.

7. Local refinement for fillet with radius of 12 mm, and global mesh size of 4 mm

Table 7: Data for local refinement on the fillet (radius of 12 mm, and global mesh size of 4 mm)

Case #	Global element size (mm)	Local element size (mm)	Location for local refine-ment	Total # of elements (10^3)	Total # of nodes (10^3)	DOF (10^3)	Max stress (MPa)	Max displace-ment (mm) [10^{-2}]	Run time (s)
1	4	3	fillet	42.935	66.272	198.573	65.31	2.323	5
2	4	2.5	fillet	45.753	70.270	210.567	65.59	2.321	5
3	4	2	fillet	47.536	72.935	218.562	65.81	2.323	6
4	4	1.5	fillet	58.657	88.737	265.968	65.99	2.322	7
5	4	1	fillet	71.658	108.544	325.389	66.20	2.323	9
6	4	0.5	fillet	166.863	248.026	743.835	66.17	2.323	20
7	4	0.45	fillet	193.023	286.794	860.139	66.21	2.323	23
8	4	0.43	fillet	205.882	306.092	918.033	66.16	2.323	26

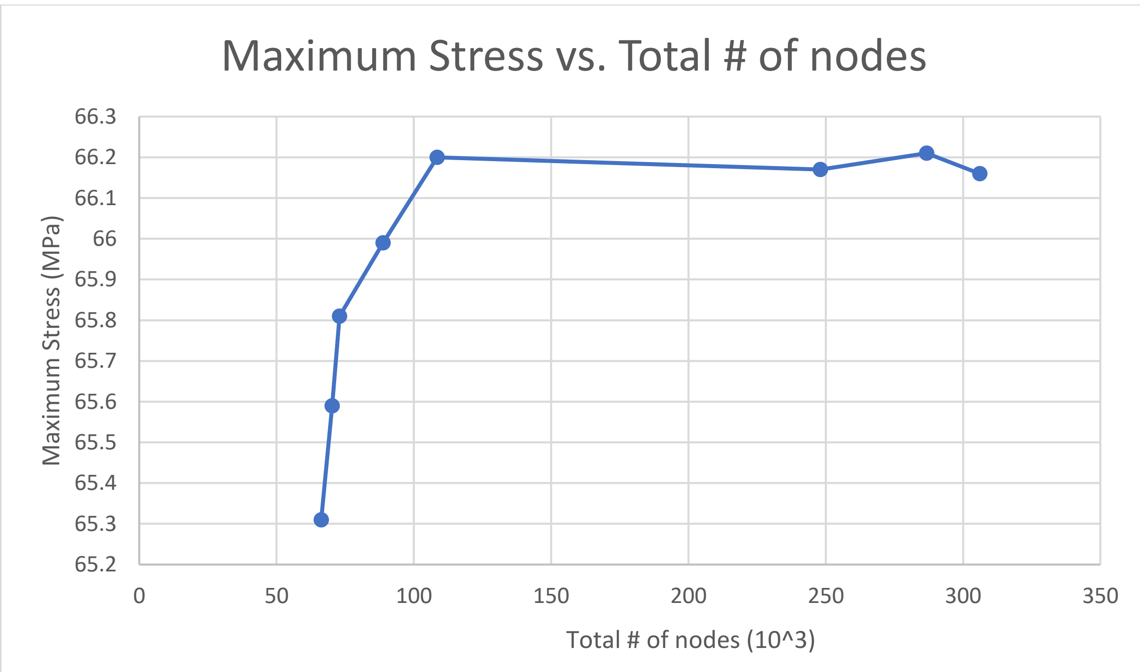


Figure 15: Maximum stress vs total number of nodes for different global element size (r=12 mm)

When I increased the fillet size to 12 mm, the result also converged. The maximum stress for this one was 66.20 MPa, which was also lower than that of fillet size of 8 mm. Then, I tried the global mesh size of 6 mm to see if 4 mm global size was too fine to mesh

8. Local refinement for fillet with radius of 12 mm, and global mesh size of 6 mm

Table 8: Data for local refinement on the fillet (radius of 12 mm, and global mesh size of 6 mm)

Case #	Global element size (mm)	Local element size (mm)	Location for local refine-ment	Total # of elements (10^3)	Total # of nodes (10^3)	DOF (10^3)	Max stress (MPa)	Max displace-ment (mm) [10^{-2}]	Run time (s)
1	6	4	fillet	15.473	25.238	75.591	64.92	2.335	3
2	6	3	fillet	17.357	27.991	83.850	65.44	2.335	2
3	6	2	fillet	21.726	34.418	103.131	66.02	2.335	4
4	6	1.5	fillet	30.613	47.258	141.651	66.47	2.335	5
5	6	1	fillet	50.000	75.530	226.467	66.48	2.335	6

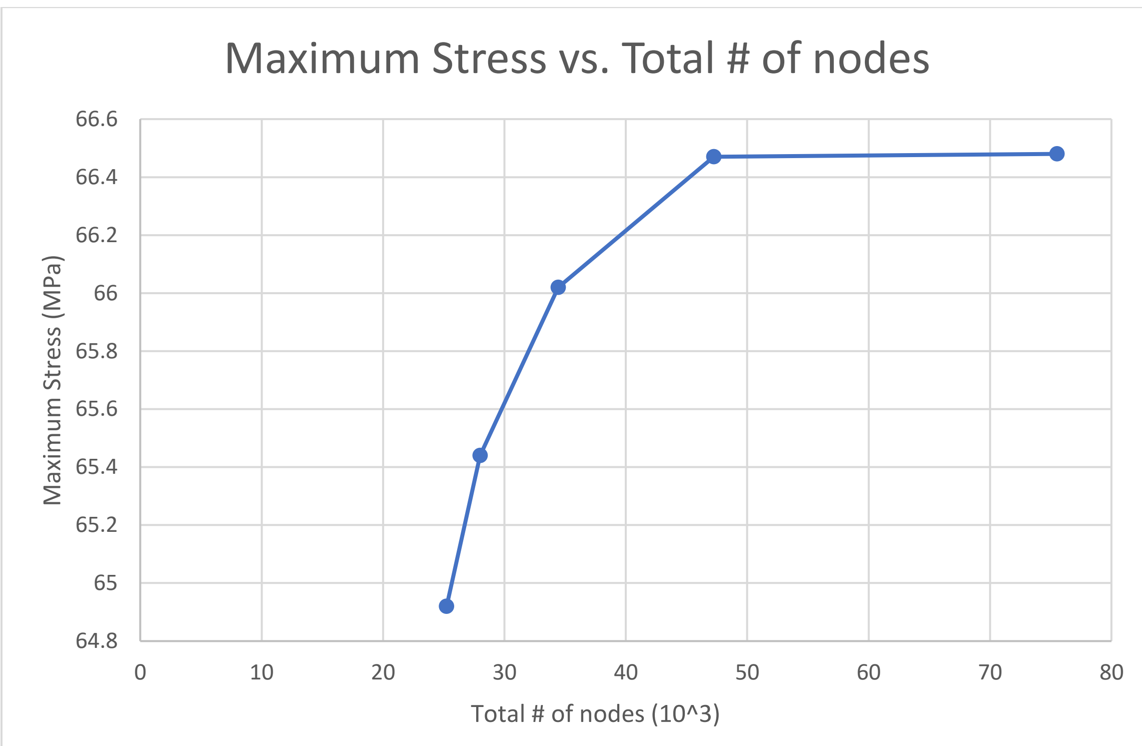


Figure 16: Maximum stress vs total number of nodes for different global element size (r=12 mm)

Graph for the mesh size of 6 mm also indicated that the result was converged.

9. Local refinement fillet with radius of 15 mm, and global mesh size of 4 mm

Table 9: Data for local refinement on the fillet (radius of 15 mm, and global mesh size of 4 mm)

Case #	Global element size (mm)	Local element size (mm)	Location for local refine-ment	Total # of elements (10^3)	Total # of nodes (10^3)	DOF (10^3)	Max stress (MPa)	Max displace-ment (mm) [10^{-2}]	Run time (s)
1	4	3	fillet	42.298	65.313	195.696	64.05	2.248	3
2	4	2.5	fillet	45.204	69.433	208.056	64.15	2.247	5
3	4	2	fillet	50.791	77.475	232.182	64.65	2.248	6
4	4	1.5	fillet	58.810	89.103	267.066	64.71	2.248	8
5	4	1	fillet	73.583	111.905	335.472	64.72	2.248	12
6	4	0.5	fillet	181.441	270.553	811.416	64.69	2.248	22

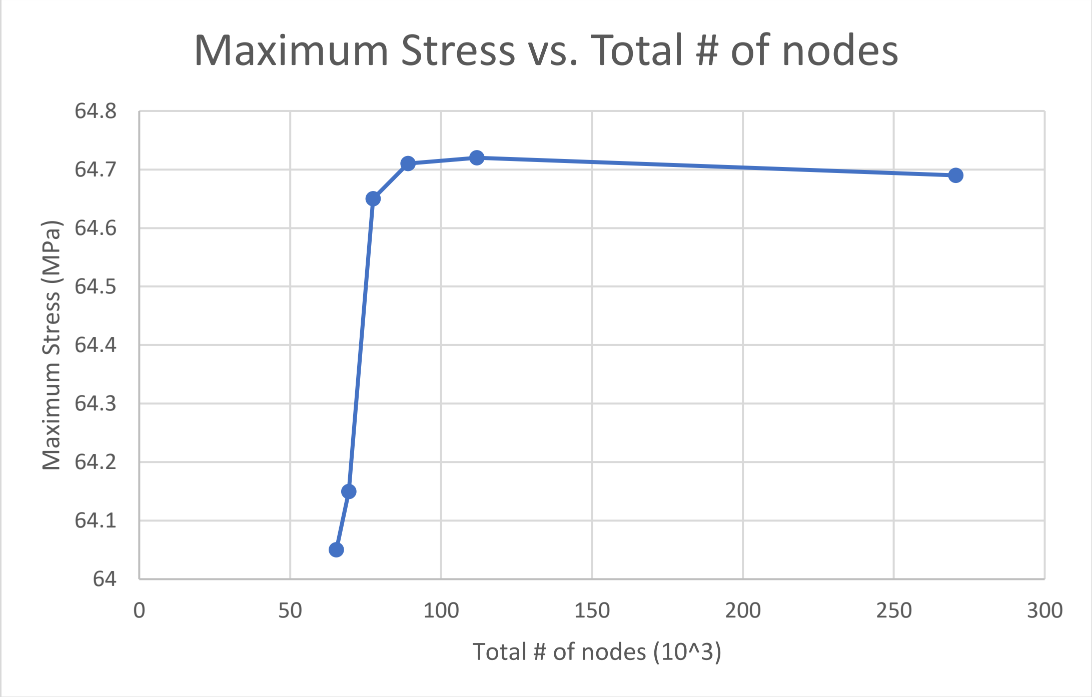


Figure 17: Maximum stress vs total number of nodes for different global element size (r=15 mm)

When I changed the fillet size to 15 mm, the result also converged. The maximum stress also decreased this time from 66.2 to 64.7 MPa.

10. Local refinement fillet with radius of 18 mm, and global mesh size of 4 mm

Table 10: Data for local refinement on the fillet (radius of 18 mm, and global mesh size of 4 mm)

Case #	Global element size (mm)	Local element size (mm)	Location for local refine-ment	Total # of elements (10^3)	Total # of nodes (10^3)	DOF (10^3)	Max stress (MPa)	Max displace-ment (mm) [10^{-2}]	Run time (s)
1	4	3	fillet	43.648	67.139	201.174	63.03	2.187	4
2	4	2.5	fillet	47.483	76.321	213.684	63.18	2.187	4
3	4	2	fillet	53.168	84.168	255.361	63.69	2.187	8
4	4	1.5	fillet	62.369	98.462	289.138	63.85	2.187	11
5	4	1	fillet	86.652	123.742	362.132	63.87	2.187	13
6	4	0.5	fillet	199.646	296.046	887.895	63.86	2.187	23

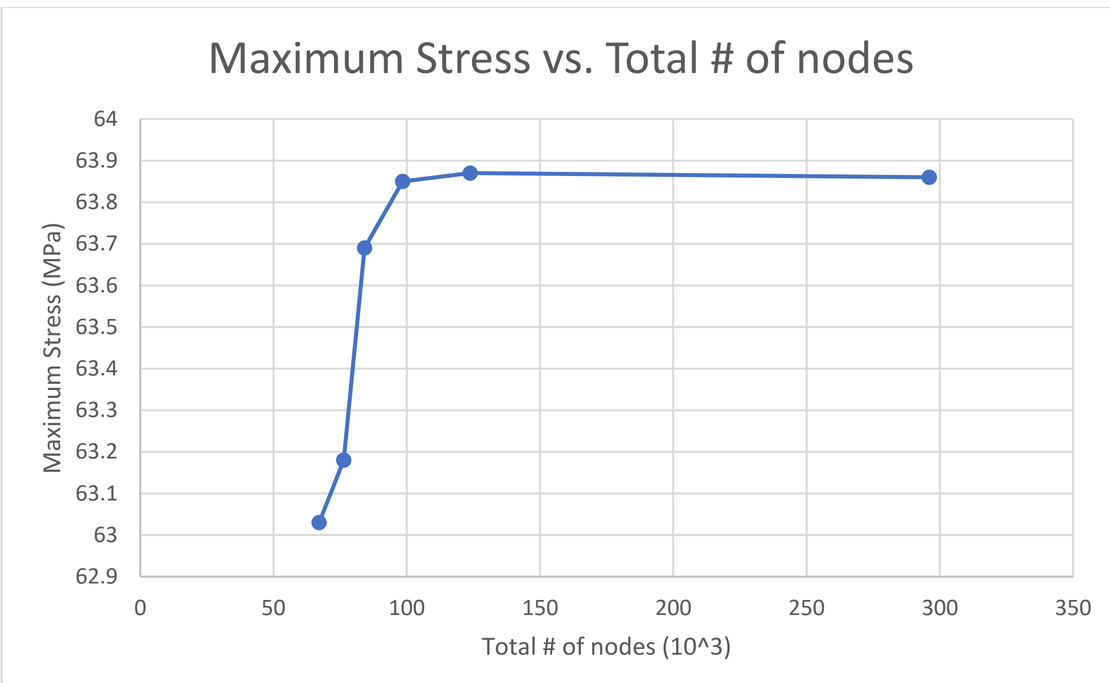


Figure 18: Maximum stress vs total number of nodes for different global element size (r=18mm)

When I changed the fillet size to 18 mm, the result also converged. The maximum stress also decreased this time from 64.7 to 63.85 MPa. Therefore, I could make the conclusion that adding a fillet would improve the design, and make the result converge.

Comparison

For same global mesh size (4mm), comparing the maximum stress for different fillet radius.

Table 11: Comparing maximum stress for different fillet radius

Fillet radius (mm)	Maximum stress (MPa)
4	76.80
7	70.30
8	69.35
12	66.20
15	64.70
18	63.85

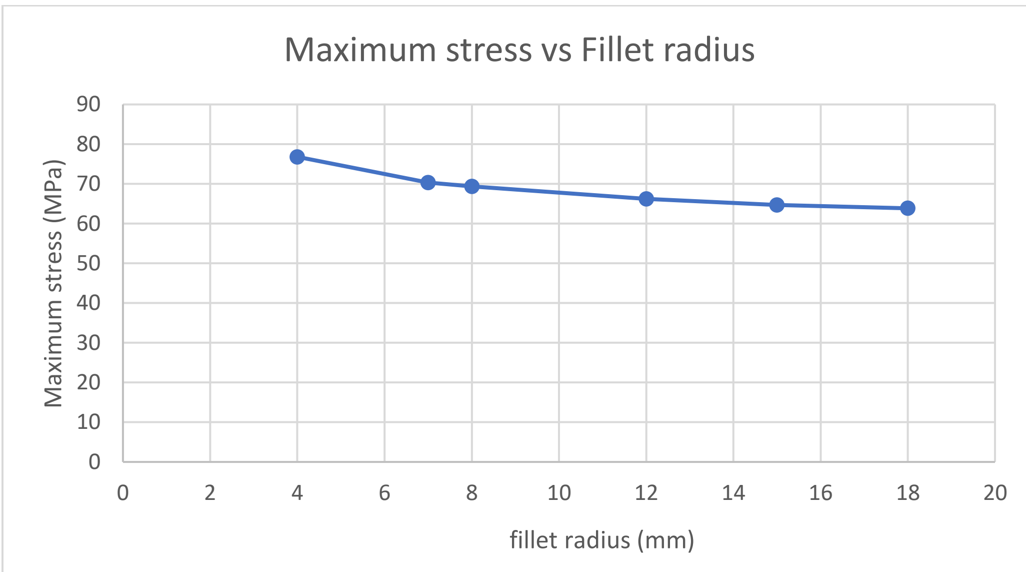


Figure 19: maximum stress vs fillet radius

By comparing the maximum stresses for different fillet radius, I could make the conclusion that increasing the fillet radius could decrease the maximum stresses applied to the pulley system, and decreasing maximum stresses on the pulley could make the design safer. Moreover, when increasing the radius, the maximum stress also converged. The maximum stress it converged to was approximately 64 MPa at 15 mm fillet. Therefore, any radius larger than 15 mm did not have too much impact on maximum stress.

Analytical Solution

The equation I needed to use to calculate the stress was

$$\tau_{max} = K\frac{T\ c}{J}$$

C is radius of the shaft, which is 12.5 mm = 0.0125 m

T is the torque applied, which is 100 N m

J is the moment of inertia. For cylinder is $\frac{\pi d^4}{32} = \frac{\pi 0.025^4}{32} = 3.835 \times 10^{-8}$

K can be obtained by using the graph below

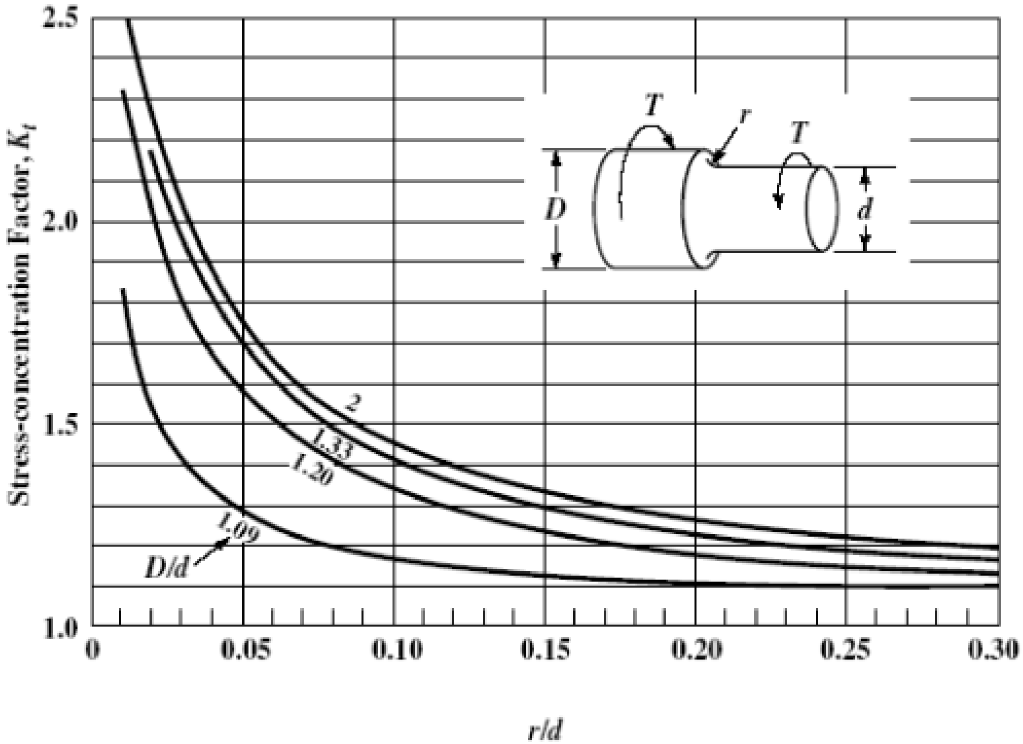


Figure 20: Stress-Concentration factor graph

For our case, D= 50 mm, and d=25 mm. D/d=2

When r (fillet radius)=4, r/d=4/25=0.16. The K value was approximately 1.32

$$\tau_{max} = 1.32\ \frac{100 \times 0.0125}{3.835 \times 10^{-8}} = 43024771Pa \cong 43.025\ MPa$$

Recall $\sigma_{Von\ Mises} = \sqrt{\frac{(\sigma_1-\sigma_2)^2+(\sigma_1-\sigma_3)^2+(\sigma_2-\sigma_3)^2}{2}}$, where σ_1 is the largest value, and σ_3 is the smallest value (negative).

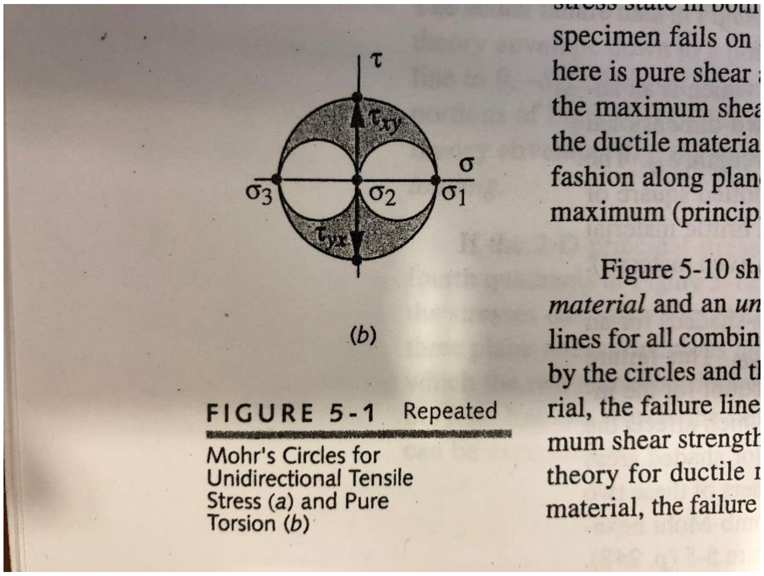


Figure 21: Mohr's Circle from Professor Benenson's handout.

Based on the handout Professor Benenson gave us (figure 21), I obtained the relation below

$$\sigma_1 = -\sigma_3 = \tau_{max}, \qquad \sigma_2 = 0$$

$$\sigma_{Von\ Mises} = \sqrt{\frac{(\sigma_1 - 0)^2 + (\sigma_1 - \sigma_3)^2 + (0 - \sigma_3)^2}{2}}$$

$$\sigma_{Von\ Mises} = \sqrt{\frac{(\tau_{max})^2 + (2\tau_{max})^2 + (\tau_{max})^2}{2}}$$

$$\sigma_{Von\ Mises} = \sqrt{\frac{(43.025\ )^2 + (2 \times 43.025\ )^2 + (43.025\ )^2}{2}} = 74.5215\ MPa$$

$74.5215\ MPa$ was very close to the FEM solution I got, which was 76.80 MPa.

Evaluation of the design

The safety factor was 2, and the yield strength for this material was 350 MPa. The failure occurred when $\sigma_{Von\ Mises} > \sigma_{yield}$. 350 MPa / 2 = 175 MPa. Therefore, if the $\sigma_{Von\ Mises}$ is less than 175 MPa, the result was acceptable. The results I had for fillet of 4 mm, 7 mm, 8 mm, 12 mm, 15 mm, and 18 mm were 76.80 MPa, 70.30 MPa, 69.35 MPa, 66.20 MPa, 64.80 MPa, and 63.8 MPa, which were all less than 175 MPa. Therefore, they were all acceptable.

Discussion

The initial boundary conditions did not work properly. As figure 4 showed, these boundary conditions did not prevent shaft from shifting or translating. I treated the entire shaft as a fixed hinge to allow only rotations. The modified boundary conditions worked because it showed that the maximum displacements took place at the outer edge of the shaft where connected to the motor. However, the FEM results for the modified boundary conditions did not converge. Instead, as the mesh size decreased, the stress continuously increasing (figure 9 & 10). Then I tried to add a fillet to the pulley. The FEM results converged if I add fillets to the system. When the fillet radius was 4 mm, the results converged to 76.80 MPa, when the radius was 7 mm, the results converged to 70.30 MPa, when the fillet radius was 8 mm, the results converged to 69.35 MPa, when the fillet radius was 12 mm, the results converged to 66.20 MPa, when the fillet radius was 15 mm, the results converged to 64.70 MPa, when the fillet radius was 18 mm, the results converged to 63.85 MPa. As the radius of the fillet increased, the maximum stress decreased, and the results got converged as well. Moreover, the analytical solution I calculated was very close to the FEM results (when the radius of fillet was 4 mm, the analytical result for the stress was 74.5215 MPa, and the FEM result for the stress was 76.80 MPa). It also showed that the FEM results were reliable. I also evaluated the design. The safety factor was 2, and the yield strength was 350 MPa. The actual yield strength I need to compare to was 350/2=175 MPa. The design would fail if $\sigma_{Von\ Mises} > \sigma_{yield}$. The stresses I obtained from FEM were all less than this value. Therefore, this design was acceptable.

In my opinion, there was a better way to improve the design. Since the obstacle was very easy to penetrate the holes, we could just simply remove the holes from the pulley. The example I had in figure 2b did not have any hole on the pulley. Also when I searched the pulleys on McMaster.com, they also offered pulleys without holes (figure 22). Therefore, it could be a solution for the situation we had in this project.

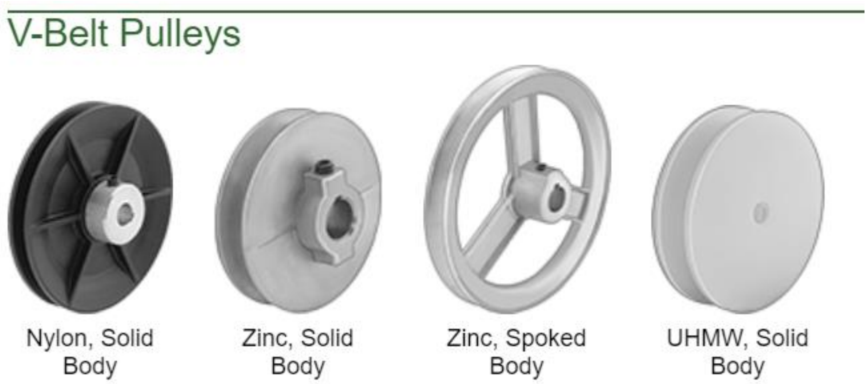


Figure 22: Pulleys offered on McMaster.com