CITY COLLEGE
CITY COLLEGE OF NEW YORK

HOMEWORK 7:
FEM Project 2.0

ME 371 Computer Aided Design
Section: 3362
Dr. Savvas Xanthos

Submitted by: Pradip Thapa
Number # 23
Date: April 15, 2011

1. Introduction of FEM

The finite element method (FEM), sometimes referred to as *finite element analysis* (FEA), is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Simply stated, a boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of the domain. Boundary value problems are also sometimes called *field* problems. The field is the domain of interest and most often represents a physical structure.

The *field variables* are the dependent variables of interest governed by the differential equation. The *boundary conditions* are the specified values of the field variables (or related variables such as derivatives) on the boundaries of the field. Depending on the type of physical problem being analyzed, the field variables may include physical displacement, temperature, heat flux, and fluid velocity to name only a few.

A GENERAL PROCEDURE FOR FINITE ELEMENT ANALYSIS

- Define the geometric domain of the problem.
- Define the material properties of the elements.
- Define the physical constraints (boundary conditions).
- Define the physical constraints (boundary conditions).
- Define the element connectivity (mesh the model).
- Define the element type(s) to be used.
- Define the geometric properties of the elements.
- Run the model
- Post processing the result.

2. Overview of FEM in Stress Analysis:

The field variables for stress analysis is an elemental nodal displacement and this objective of this project is to understand the application of FEM and its procedures to use in Solidwork 2006 SP3 COSMOS for linear analyses of solid models whose analytical solution is unknown. The part selected for this project is a steel pulley with is attached to a shaft, and the shaft is connected to a motor, which continues to turn the shaft, even though an obstacle has penetrated one of the holes, preventing an inside face from moving. The primary purpose of the analysis is to determine whether the pulley-shaft system would deform permanently that is the elastic body will pass beyond it yield strength and create a permanent changes in the structure. And if the analysis shows that under these circumstances, if it does then the model is subjected to redesign to prevent it from failure. Since the boundary conditions are not predefined each study should be analyzed with more real life situation in respect to constrain and loading to obtain the optimum solution and tested by convergence test to validate the model nd its stress solution.

3. Geometric and Material Parameter of Model:

Material Parameter	Value
Assign Material	Steel 1020 cold rolled
Young Modulus	205 GPa
Poisson ratio	0.29
Yield Strength	350 MPa

4. Boundary Condition:

Model	Value
Restrain	Yes
Torque	0.25 N.m

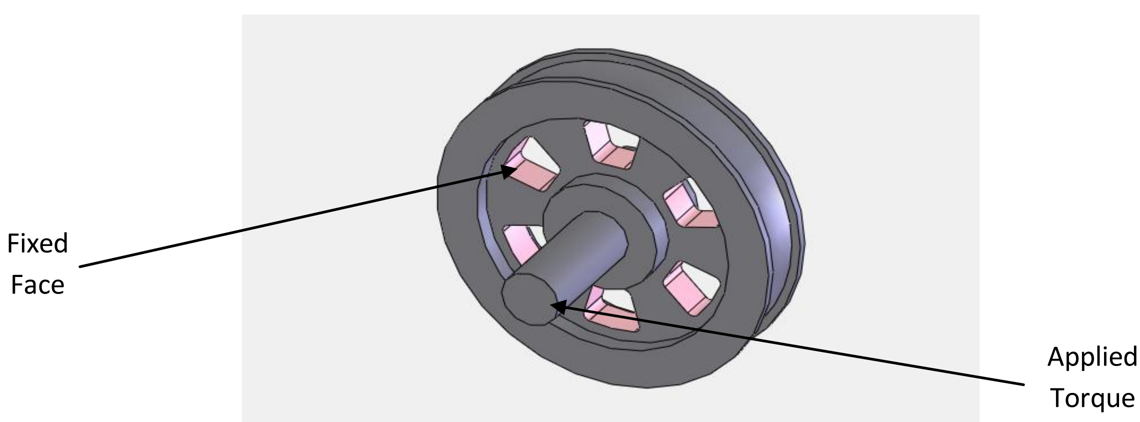


Figure 1 Pulley

5. Intuitive Case Study with global mesh control :

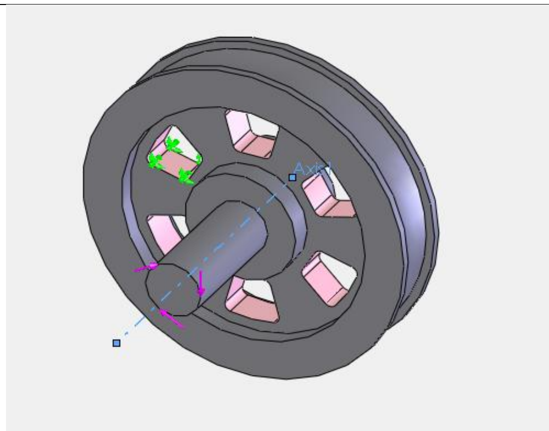


Figure2 .0 Constrain and loading

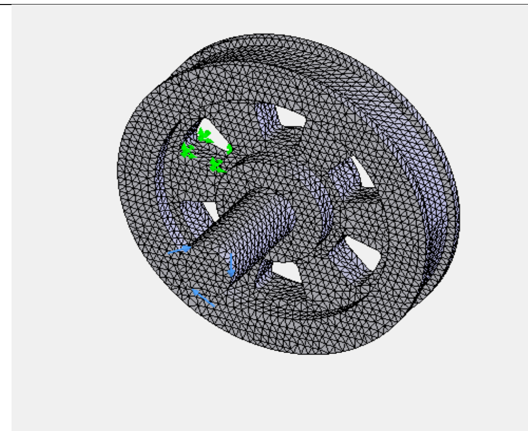


Figure 3.0 High quality Fine Mesh

Boundary condition and loading is critical in finite element solver as the discrimination of the model and field variable is used to determine the resultant variables as prescribed by the user. As the first approach to the problem, the initial constrain is applied on pink rectangular face shown in figure 2 indicated by green arrow and torque is applied on the circular face of extruded cylindrical shaft indicated by purple arrow and the direction if the torque is clockwise with respect to the central axis indicated by the blue line. As found from FEM project 1 it was found that Mesh type and element size also governs the resultant value of the stress and displacement. Therefore a high quality mesh (P=2), was selected and fine mesh size was selected to generate the mesh.

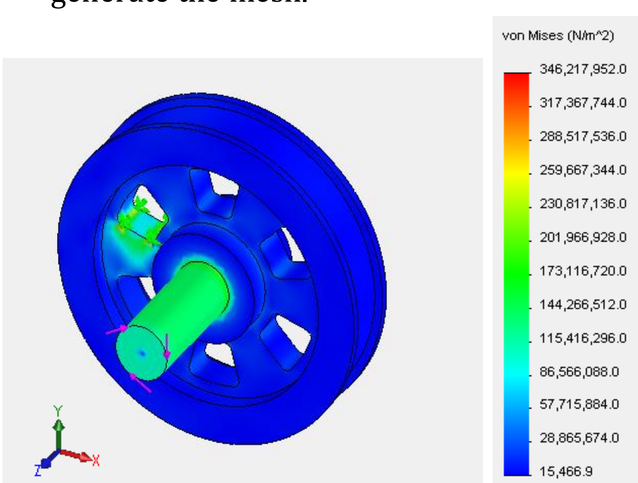


Figure 4 .0 Von-Mises Stress

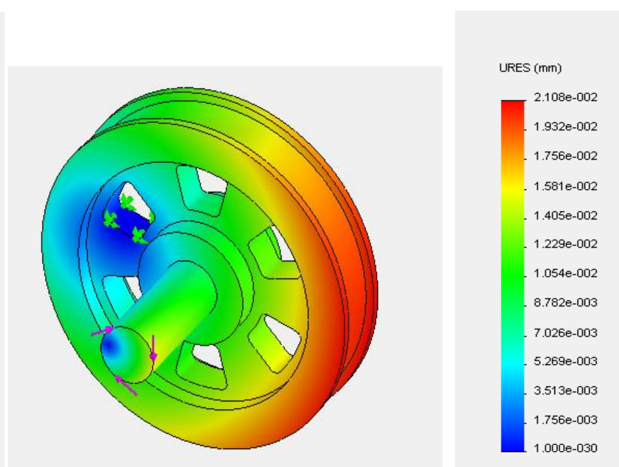
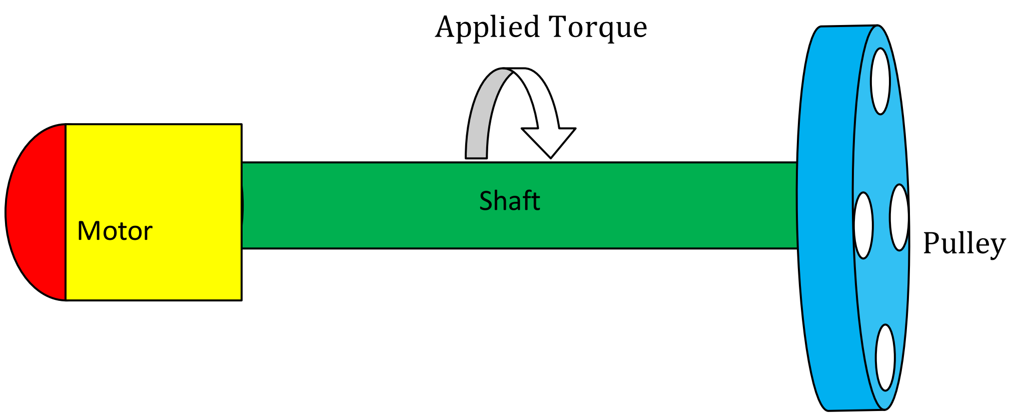


Figure5.0 Resultant Displacement

Figure 4.0 represent the Von-Mises Stress which shows that the stress in uniformly distributed along the shaft and maximum stress is produced at the restrain and the z-direction over the surface of the shaft and also displacement in the pulley disk is maximum in the direction of the applied torque.

6. Modified Case Study local mesh control:

The stress concentration theory says that the stress should be maximum at the place where there is sudden change in the cross section area of the elastic body which is under loading because this sudden change in the cross section disrupt the shear flow inside the body creating concentration of nodal displacement. Hence in new study mesh control was applied in the edge of the joint between the shaft and pulley.



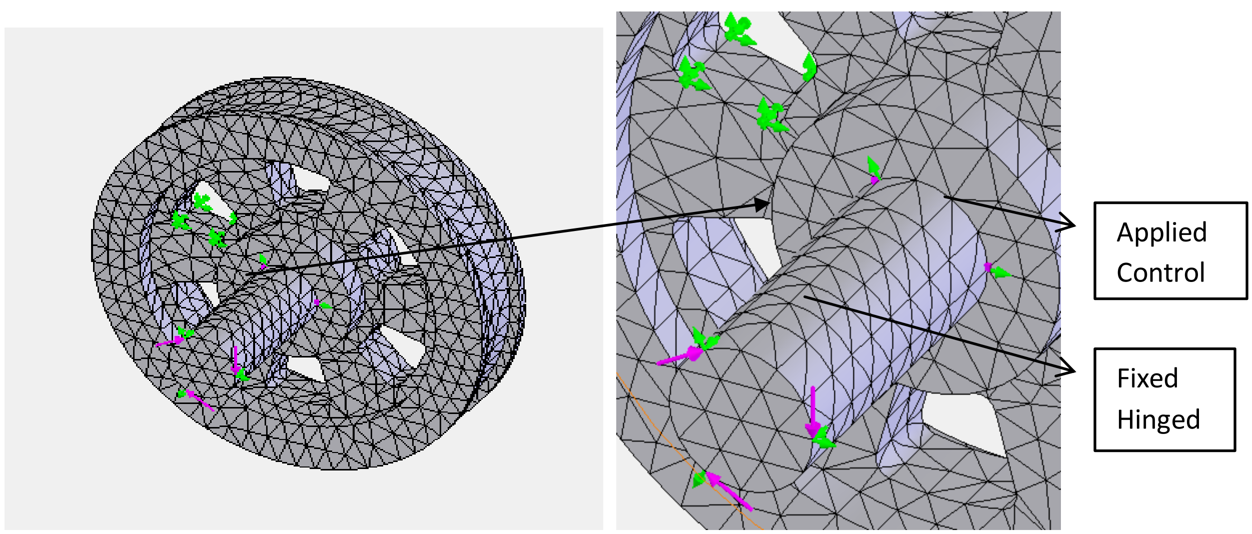


Figure6 .0 High quality Medium Global Mesh and Coarse Local Mesh

Also we know the pulley is attached to the motor and therefore the shaft is like a fixed hinged support between motor and pulley disk preventing it lateral movement of the shaft and torsion force that rotating the shaft will deform the body angularly rather than translating in shaft and pulley in the normal direction of applied torque.

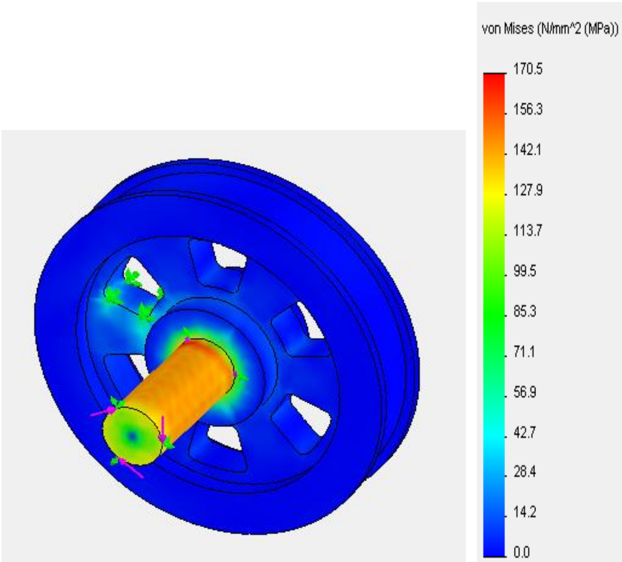


Figure 7 .0 Von-Mises Stress

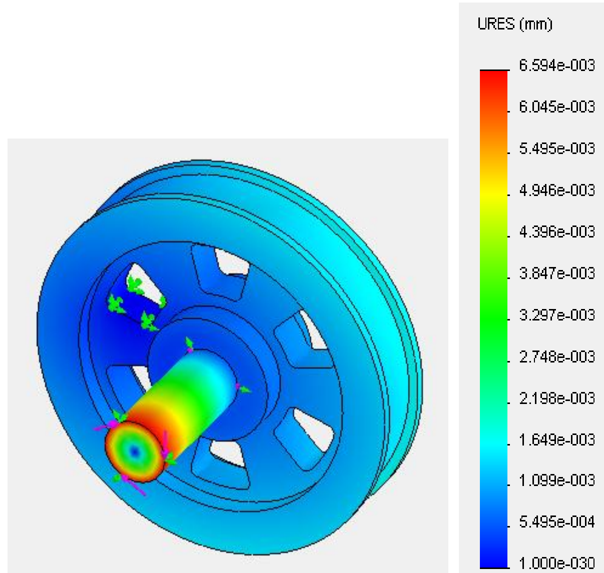


Figure8.0 Resultant Displacement

Figure 7.0 represent the Von-Mises Stress which shows that the stress and figure 8.0 represent the resultant displacement which shows as completely different result than from figure 5.0 and 6.0. The stress is maximum at the intersection of the shaft and pulley which is also predicated by stress concentration theory. The displacement in the pulley disk is maximum at the surface face where torque is applied which was also observed during the torsion test lab for ME 462 Material Science when a straight line drawn on the surface was twisted most on the direction which was hold and twisted.

7. Modified Case Study Change in the Loading position:

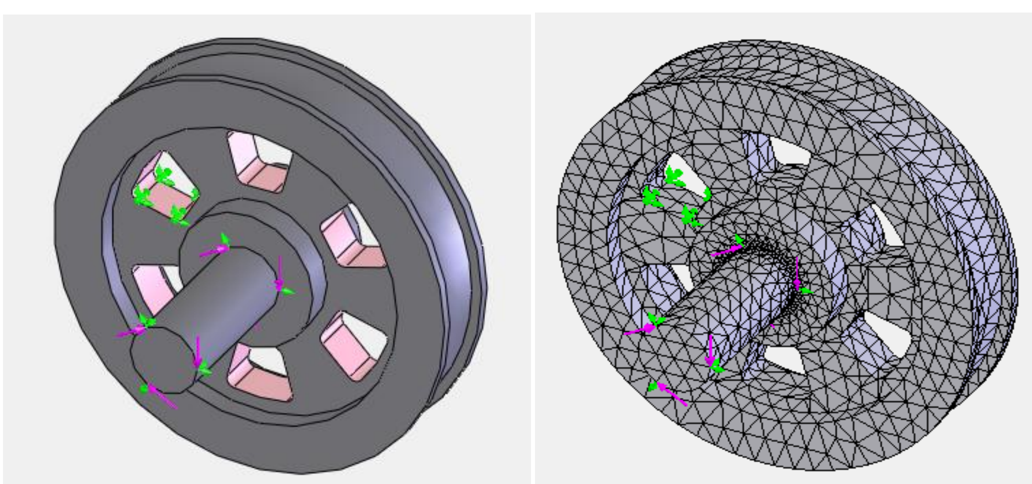


Figure9 .0 High Quality Medium Global Mesh and Fine Local Mesh with Torque applied over the surface of shaft

In the figure above the loading case is modified such a way the now the torque is applied over the entire surface of the shaft and the restrain however remain the same. Fine mesh was applied in the zone of interest and medium global mesh was chosen for the entire body.

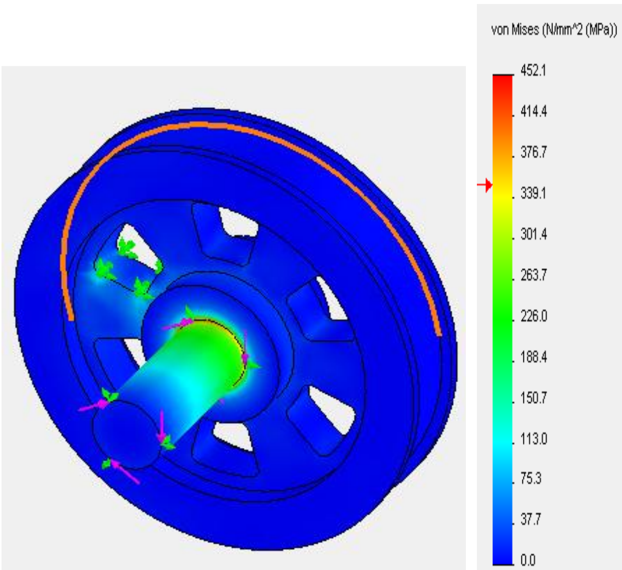


Figure 10 .0 Von-Mises Stress

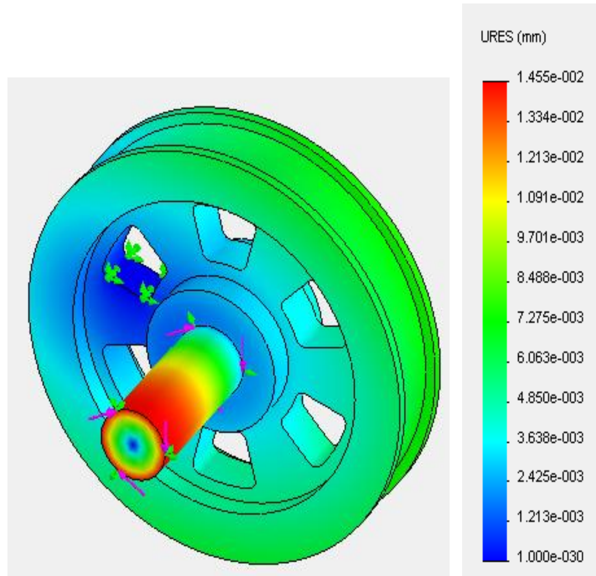


Figure11.0 Resultant Displacement

From the figure 10, we can say that the maximum stress has passed way beyond the Yield stress and which means the deformation is now in plastic region which is not predicated accurately by SolidWorks COSMOS, because it is linear solver and the stress strain behaves nonlinearly in plastics region. The resultant stress is double the value predicated by the loading face region torque.

8. Modified Case Study Change in the Restrain position over the shaft surface:

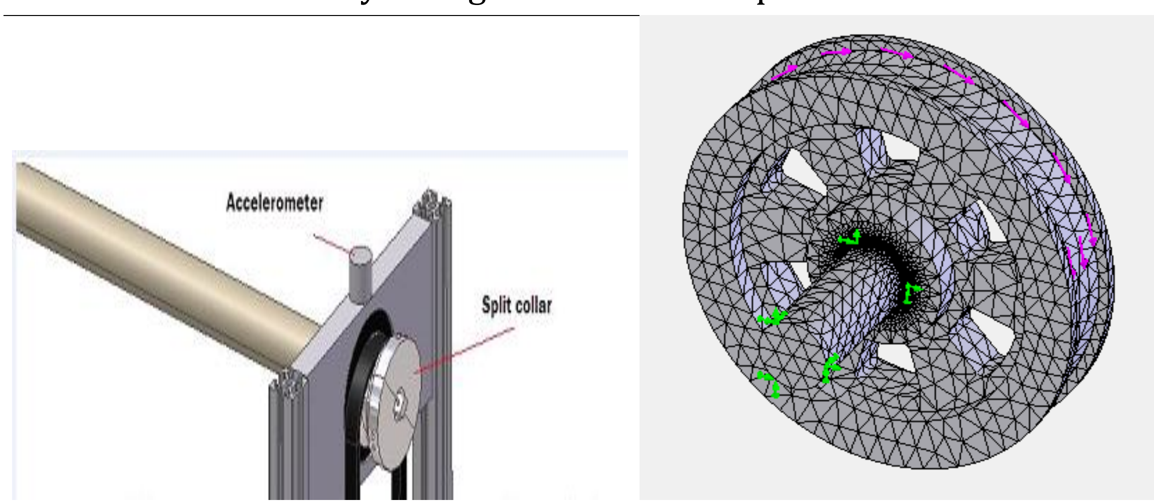


Figure 12 .0 High Quality Medium Global Mesh and Fine Local Mesh with Torque applied over the surface of pulley

The assumption is that in the pulley belt system generally for pulley system for which radius ratio of the system if is equally to 1, then the surface over the belt the belt apply force is half of the circumference and the force is applied tangentially over the contact surface by the belt.

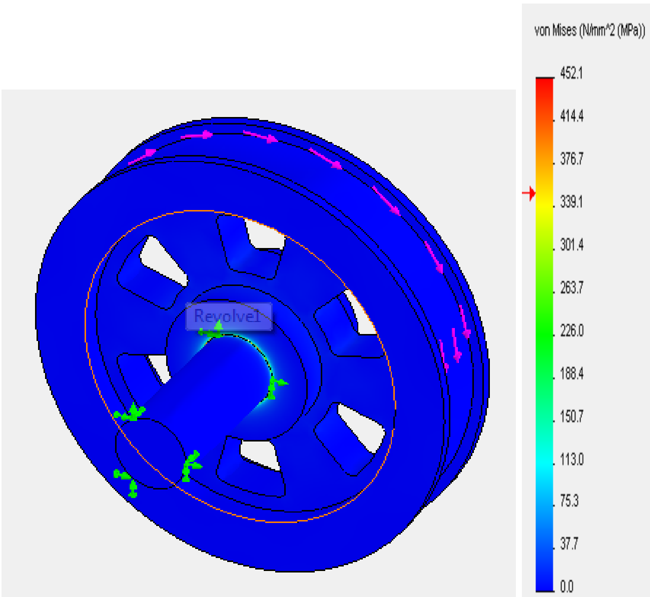


Figure 10 .0 Von-Mises Stress

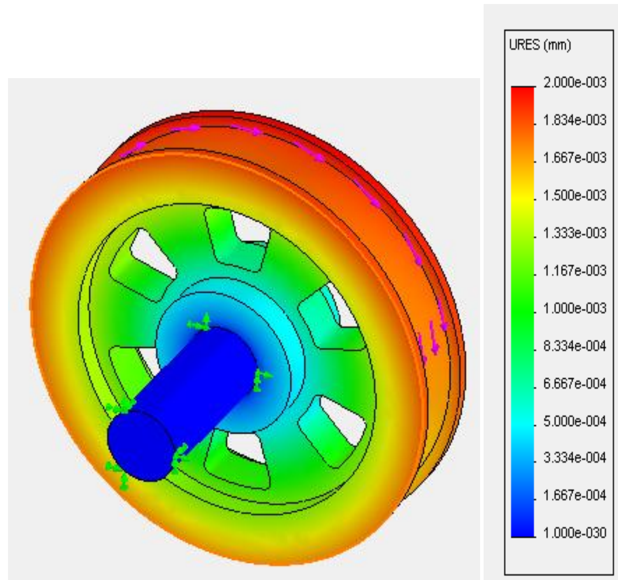


Figure11.0 Resultant Displacement

Above figure also show that when the control is applied on the intersection of the pulley and shaft and the torque the shaft is constraint as the fixed support over its surface and the torque is applied on the half surface of the pulley. Then again from the analysis we can see that the stress is above the yield stress.

9. Modified Case Study Change in the Restrain position over the shaft face:

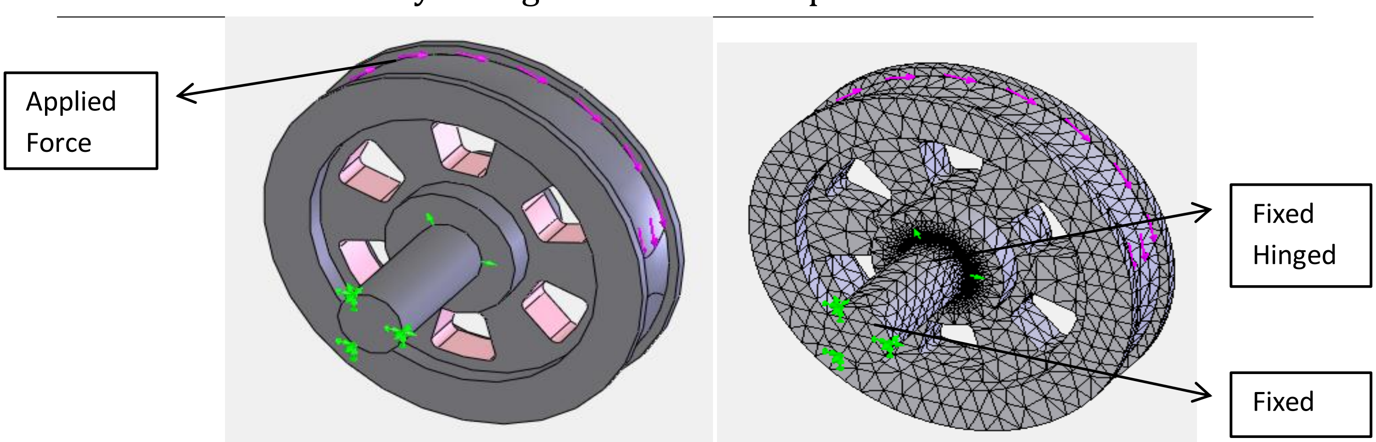
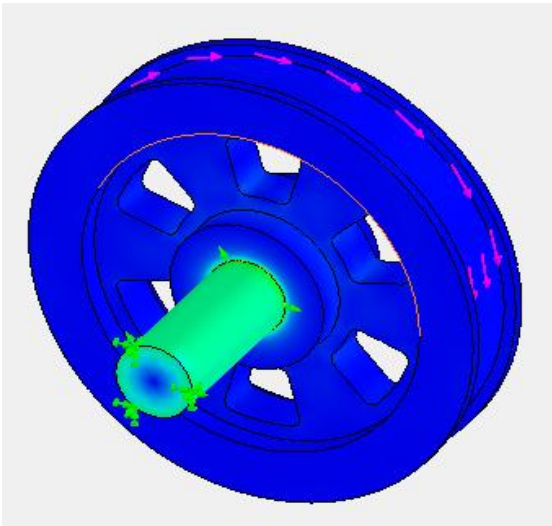
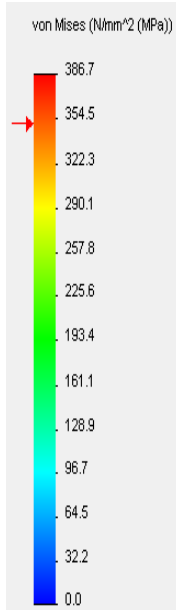


Figure 12 .0 High Quality Medium Global Mesh and Fine Local Mesh with Torque applied over the surface of pulley





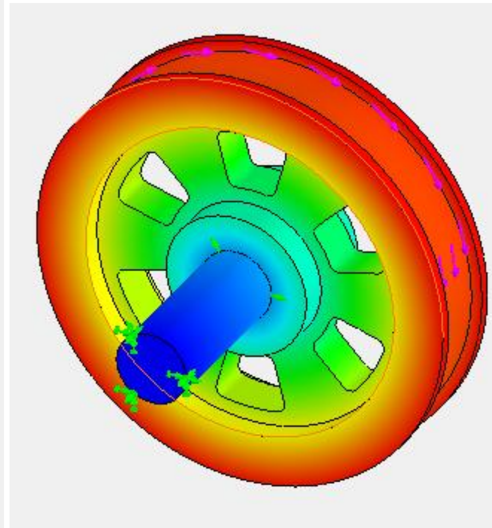


Figure 13 .0 Von-Mises Stress **Figure14.0 Resultant Displacement**

Above figure also show that when the control is applied on the intersection of the pulley and shaft and the torque the shaft is constraint as the fixed support at the face of shaft and hinged support at its surface and the torque is applied on the half surface of the pulley. Then again from the analysis we can see that the stress is above the yield stress.

10. H-adaptive :

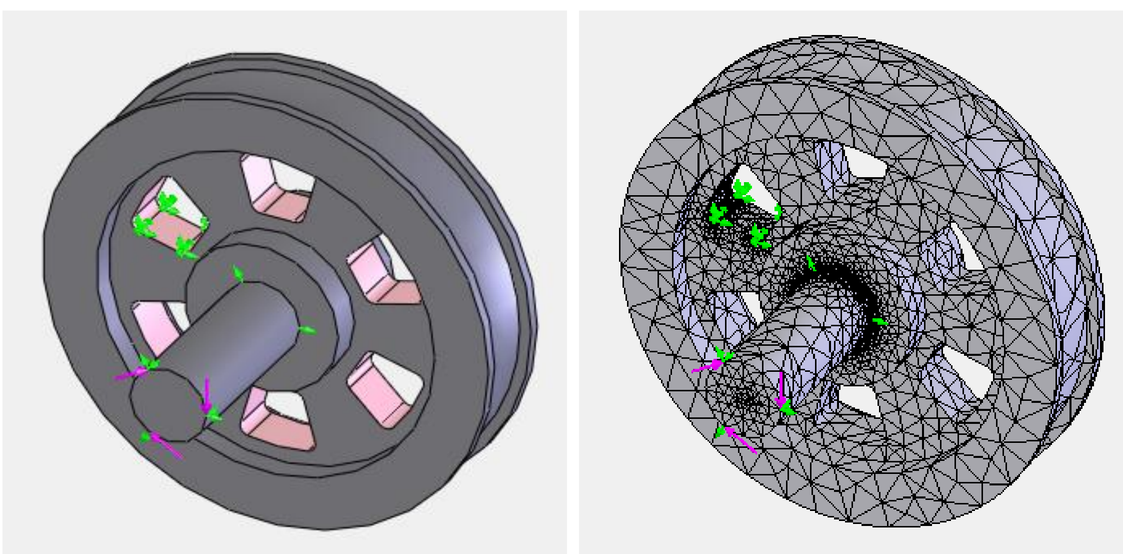


Figure 15 .0 Torque applied at the face of the shaft and restrain fixed at the rectangular face in pulley and hinged over the surface of shaft

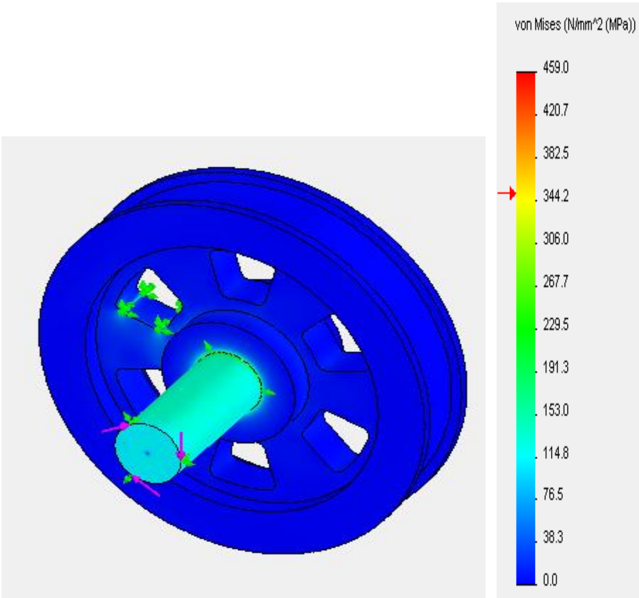


Figure 13 .0 Von-Mises Stress

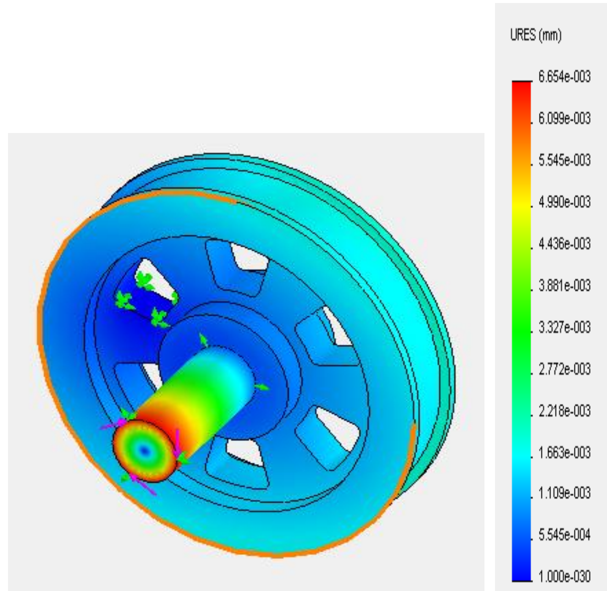
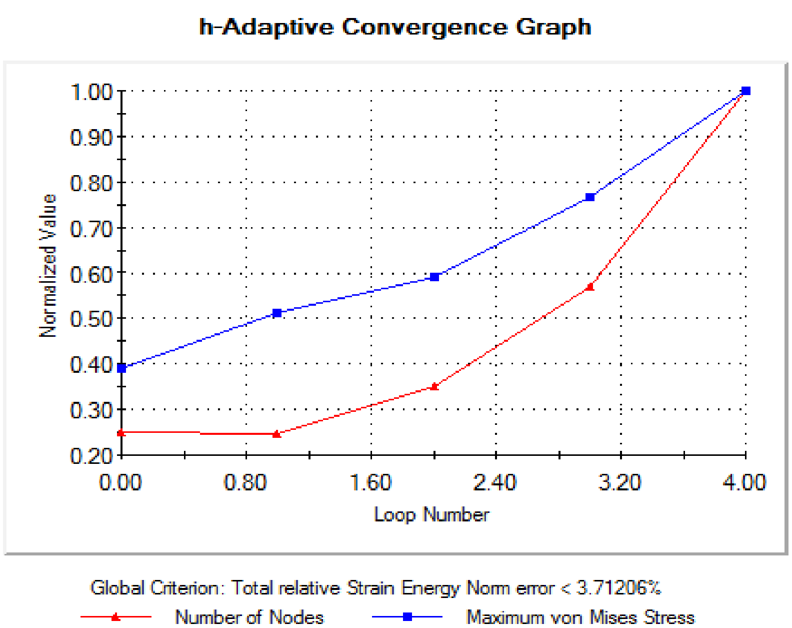


Figure14.0 Resultant Displacement



Quality (P)	No. of nodes	No. of elements	No. of DOF	Total solution time	Von-Mises Stress (MPa)	Ures (mm)	Global Element size (mm)	Local Element Size (mm)	Yield Strength (MPa)
2	60314	39583	178290	0:00:12	434.7	0.006	0.70014	NA	350

11. Results:

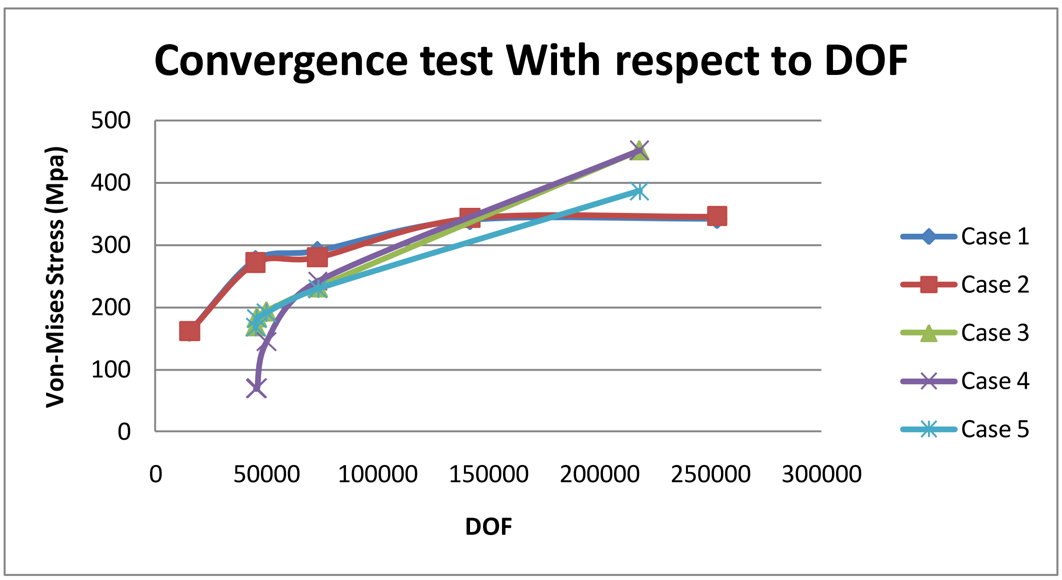
Case 1					
Study No.	1	2	3	4	5
Quality (P)	2	2	2	2	2
No. of nodes	5141	15074	24379	47335	84470
No. of elements	2743	8792	14783	29987	55263
No. of DOF	15378	45129	73014	141798	253149
Total solution time	0:00:02	0:00:03	0:00:03	0:00:04	0:00:11
Von-Mises Stress (MPa)	160.5	275.8	290.5	340.3	342.2
Ures (mm)	0.0218	0.0218	0.0218	0.0218	0.0218
Global Element size (mm)	1.40028	0.70014	0.568864	0.437588	0.035007
Local Element Size (mm)	1.40028	0.70014	0.568864	0.437588	0.035007
Yield Strength (MPa)	350	350	350	350	350

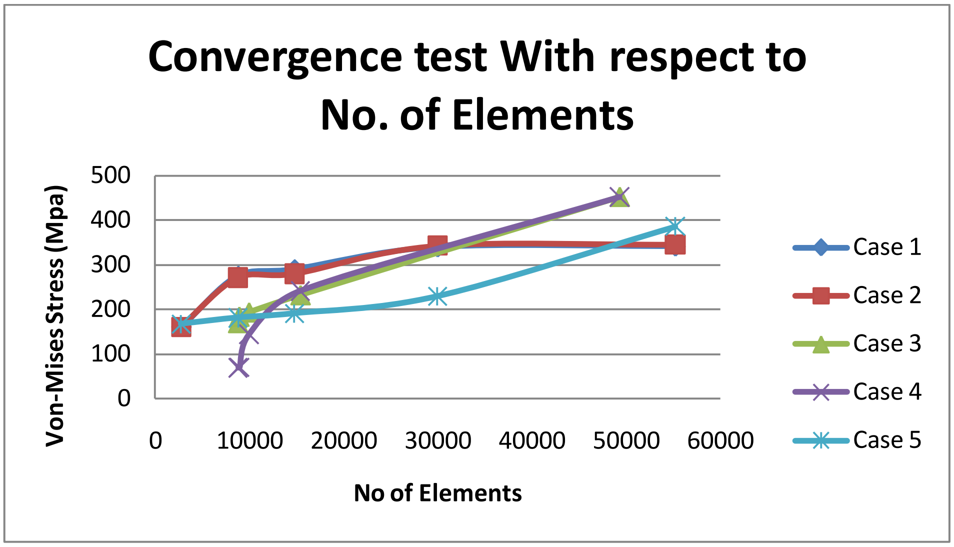
Case 2					
Study No.	1	2	3	4	5
Quality (P)	2	2	2	2	2
No. of nodes	5141	15074	24379	47335	84470
No. of elements	2743	8792	14783	29987	55263
No. of DOF	15378	45129	73014	141798	253149
Total solution time	0:00:02	0:00:03	0:00:03	0:00:04	0:00:11
Von-Mises Stress (MPa)	160.5	275.8	290.5	340.3	342.2
Ures (mm)	0.0218	0.0218	0.0218	0.0218	0.0218
Global Element size (mm)	1.40028	0.70014	0.568864	0.437588	0.035007
Local Element Size (mm)	1.40028	0.70014	0.568864	0.437588	0.035007
Yield Strength (MPa)	350	350	350	350	350

Case 3					
Study No.	1	2	3	4	5
Quality (P)	2	2	2	2	2
No. of nodes	5141	15074	24379	47335	84470
No. of elements	2743	8792	14783	29987	55263
No. of DOF	15378	45129	73014	141798	253149
Total solution time	0:00:02	0:00:03	0:00:03	0:00:04	0:00:11
Von-Mises Stress (MPa)	160.5	275.8	290.5	340.3	342.2
Ures (mm)	0.0218	0.0218	0.0218	0.0218	0.0218
Global Element size (mm)	1.40028	0.70014	0.568864	0.437588	0.035007
Local Element Size (mm)	1.40028	0.70014	0.568864	0.437588	0.035007
Yield Strength (MPa)	350	350	350	350	350

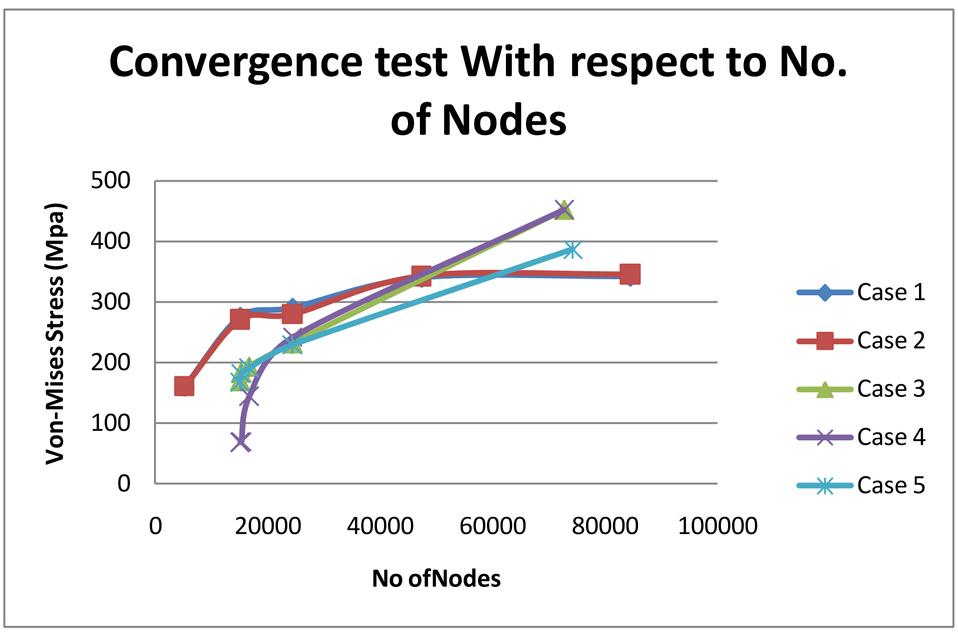
Case 4					
Study No.	1	2	3	4	5
Quality (P)	2	2	2	2	2
No. of nodes	5141	15074	24379	47335	84470
No. of elements	2743	8792	14783	29987	55263
No. of DOF	15378	45129	73014	141798	253149
Total solution time	0:00:02	0:00:03	0:00:03	0:00:04	0:00:11
Von-Mises Stress (MPa)	160.5	275.8	290.5	340.3	342.2
Ures (mm)	0.0218	0.0218	0.0218	0.0218	0.0218
Global Element size (mm)	1.40028	0.70014	0.568864	0.437588	0.035007
Local Element Size (mm)	1.40028	0.70014	0.568864	0.437588	0.035007
Yield Strength (MPa)	350	350	350	350	350

Case 5					
Study No.	1	2	3	4	5
Quality (P)	2	2	2	2	2
No. of nodes	5141	15074	24379	47335	84470
No. of elements	2743	8792	14783	29987	55263
No. of DOF	15378	45129	73014	141798	253149
Total solution time	0:00:02	0:00:03	0:00:03	0:00:04	0:00:11
Von-Mises Stress (MPa)	160.5	275.8	290.5	340.3	342.2
Ures (mm)	0.0218	0.0218	0.0218	0.0218	0.0218
Global Element size (mm)	1.40028	0.70014	0.568864	0.437588	0.035007
Local Element Size (mm)	1.40028	0.70014	0.568864	0.437588	0.035007
Yield Strength (MPa)	350	350	350	350	350





Convergence test With respect to No. of Elements
Von-Mises Stress (Mpa)
500
400
300
200
100
0
0
10000
20000
30000
40000
50000
60000
No of Elements
Case 1
Case 2
Case 3
Case 4
Case 5



Convergence test With respect to No. of Nodes
Von-Mises Stress (Mpa)
500
400
300
200
100
0
0
20000
40000
60000
80000
100000
No ofNodes
Case 1
Case 2
Case 3
Case 4
Case 5

12. Analytical Solution for Modified Geometry:

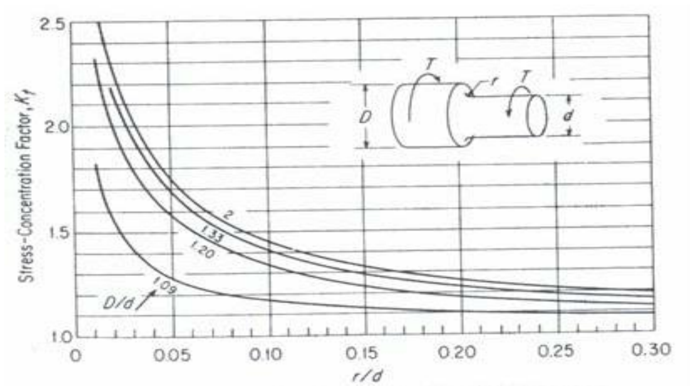
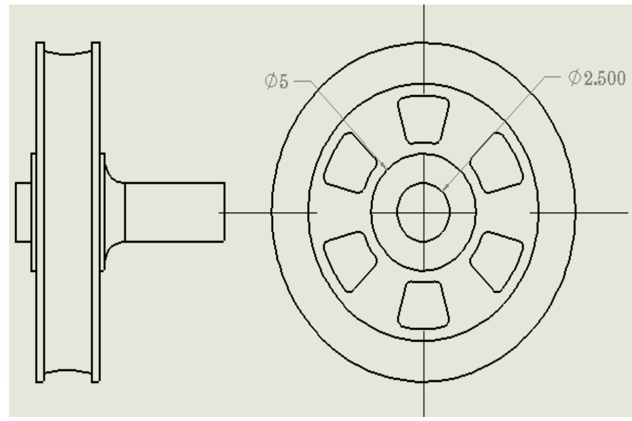


Fig. 4. Stress-concentration factor, K_t, for a filleted shaft in torsion*



We have

$$Torque\ on\ shaft: T = 0.25\ N.m$$

$$c = radius\ of\ shaft = 1.25mm = 0.00125m$$

$$Polar\ Moment\ of\ inertia\ (J) = \frac{1}{2} * \pi * 0.00125^4 = 3.83 * 10^{-12} m^4$$

$$\sigma_{nom} = \frac{Tc}{J} = \frac{.25 * .00125}{3.83 * 10^{-12}} = 81.6\ MPa$$

$$For\ small\ shaft\ Small\ shaft\ diameter\ (d) = 2.5 * 10^{-3}\ m$$

$$For\ big\ shaft\ Big\ shaft\ diamter\ (D) = 5 * 10^{-3}\ m$$

$$\frac{D}{d} = 2\ and\ \frac{r}{d} = a \rightarrow r = 2.5 * 10^{-3} * a\ m$$

13. Modified Geometry to meet Optimization with factor of safety (2):

$$Safety\ factor\ (FS) = \frac{Yield\ Stress}{Design\ Stress}$$

We have Yield Stress for Steel 1020 Cold Rolled = 350 MPa

$$2 = \frac{350}{Design\ Stress}$$

$$:\rightarrow Design\ Stress = \frac{350}{2}$$

$$Maximum\ \ allowable\ Strees = 175\ MPa$$

$$K = \frac{Design\ Stress}{Nominal\ Stress}$$

$$K = \frac{175}{81.6}$$

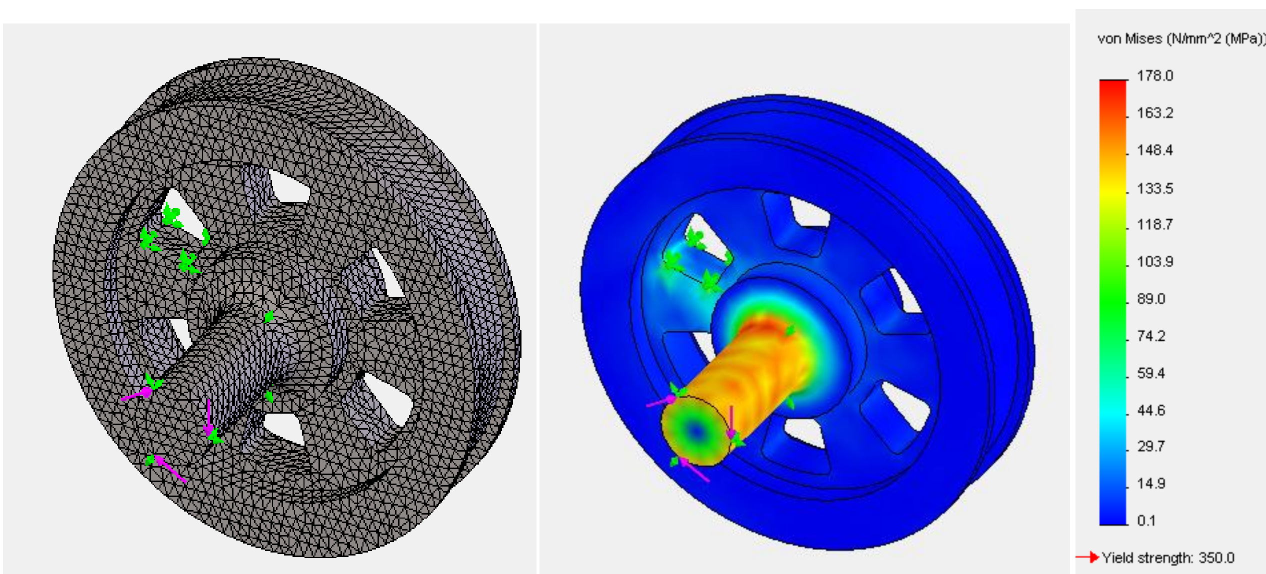
$$K = 2.144$$

Therefore from the stress concentration graph the ratio of $\frac{r}{d}$ for

$$\frac{D}{d} = 2\ and\ K = 2.144\ \rightarrow\ \mathrm{a} =\ 0.35$$

Solving for r for fillet

$$r = 2.5 * 10^{-3} * a\ = 2.5 * 10^{-3} * 0.35 = 0.875\ mm$$



Quality (P)	No. of nodes	No. of elements	No. of DOF	Total solution time	Von-Mises Stress (MPa)	Ures (mm)	Global Element size (mm)	Local Element Size (mm)	Yield Strength (MPa)
2	83411	54500	250233	0:00:07	178	0.006	0.350577	0.350577	350

14. Conclusion:

As we mentioned earlier, although restraining the shaft from moving in the radial and axial direction gave us more realistic locations for both maximum stress and displacement this was not enough to give us the correct mathematical model for us to use in our analysis. As is shown in table 1 on results, increasing the mesh density brought above higher values for maximum stress. Adding a fillet is also more accurately representative of manufacturing capabilities as sharp corner are hard to machine. Even after adding a small fillet to the shaft, the stress concentration still caused stresses that were higher than the maximum allowable stress. The fillet radius had to be increased to 0.75 radius to lower the stress concentration enough to bring it below the maximum allowable stress. The results obtained from the FEM analysis was very similar to that estimated using approximate analytical techniques. Even though no exact solutions are available for the given part geometry, analysis of the stresses in the areas of high stress can give an insight into whether FEM results in the correct range. In all the study using the original design with 0.1 mm fillet radius, the stress at the junction between the pulley and shaft was almost two times the stress on the main section of the A lot was learned during the course of this project. Although FEM # 1 taught us a lot about the effect of a particular mesh refinement and location has on our results, this project focused more on the particular boundary condition needed to recreate the intended purpose of our design. We have learned that sharp corners in the geometry of a mathematical model create, what is Kurowski calls, stress singularity which create infinite stress in that portion of the model. To avoid having any stress singularities in our geometry one must either fillet the edge by a significant amount or remove such an edge entirely. I learned how much the solution improved when using a curvature based mesh instead of a standard mesh.