Computer Aided Product Design

FINITE ELEMENT ANALYSIS

Author:

Mohammad Foroughi

Finite Element Analysis

Table of Contents

Finite Element Analysis

1. Introduction

The aim of this project is to identify and understand the outline of finite element analysis as well as familiarising with NX6 (Engineering simulation software).
There are five different sections for this project, which will contain a total of hand calculations and simulations for all sections.

The five sections of the assignment are as follow:

1 Stress concentration factor of three round bars with shoulder fillets in tension, torsion and bending using the graphs and hand calculation with various radiuses of 30, 20, 15 and 10mm.

2 Advanced simulation using NX6, considering simulation of same bars and comparing the hand calculation with the computer simulation.

3 Using the mesh refinement method in order to achieve accurate results that will be close or same as hand calculations.

4 Model and analysis of the steel shaft with nylon pulley is presses-fitted onto the shaft and applying 20Kn torque to one end of the shaft to find out the maximums von-mises stress as well as factor of safety of the design.

5 Discuss the results and recommend the new design of the model in order to achieve and improve the better factor of safety.

2. Literature review

2.1 Definition of finite element method

The finite element (FEA) method is a numerical analysis technique, which is used by engineers, scientists and mathematicians to achieve solutions as well as integral equations. This method approximately describes a wide range of physical and non-physical problems. Physical problems range in diversity from solid, fluid and soil mechanics, to electromagnetism or dynamics. The principle of the method states that a complicated domain can be sub-divided into a series of smaller regions in which the differential equations are roughly solved. By assembling some equations for each of the region, the behavior over the whole problem domain is determined. Each region is referred to as an element and the method of subdividing a domain into a finite number of elements is referred to as discretisation. At specific points called nodes, the elements will be connected, and the assembly process requires that the solution be continuous along common boundaries of adjacent elements (Salford University, 2010).

2.2 How does finite element analysis work

Finite element uses a complex system of points call nodes, which make a grid called a mesh. This mesh is designed to contain the material and structural properties, which define how the structure will responds to certain loading conditions. Nodes are assigned at a certain density throughout the material depending on the expected stress levels of specific area. The higher nodes density can be seen around regions, which receive large amount of stress. Fracture point of previously tested material, fillets, corners, complex detail, and high stress areas could consist of point of interest.

Mesh acts like the spider web from one node to other, this web of vectors is what carries the material properties to the object, creating many elements (Laboratory for Scientific Visual Analysis, 1997).

Application Problem	State (DOF) vector u represent	Forcing vector f represents
Structures and solid mechanics	Displacement	Mechanical force
Heat conduction	Temperature	Heat flux
Acoustic fluid	Displacement potential	Particle velocity
Potential flows		Particle velocity
General flows	Velocity	Fluxes
Electrostatics	Electric potential	Charge density
Magnetostatics	Magnetic potential	Magnetic intensity

Table 1: Physical Significance of Vectors u and f for Miscellaneous FEM Applications
Source: *Technische Universitat Munchen, 2010*

There are several loading conditions which may be possibly applied to the system named as below:

- Point, pressure (Figure 2), thermal, gravity, and centrifugal static loads
- Thermal loads from solution of heat transfer analysis
- Enforced displacements
- Heat flux and convection
- Point, pressure and gravity dynamic loads

2.3 Type of Engineering Analysis

- Structural analysis includes both linear and non-linear models. Linear models apply simple parameters and suppose that the material is not plastically deformed. Non-linear models consist of stressing the material past its elastic capabilities. The stresses in the material then differ with the amount of deformation

- Fatigue analysis: the number of engineering failures is caused by fatigue. Fatigue failures are defined as the tendency of a material to fracture by means of progressive brittle cracking under repeated alternating or cyclic stress of intensity considerable below the normal stress.

http://www.engineersedge.com/material_science/fatique_failure.htm
engineers use this type of analysis to calculate the life of a material or structure by showing the effects of cyclic loading on the specimen. Failure due to fatigue could also demonstrate the damage tolerance of the material (Laboratory for Scientific Visual Analysis, 1997).

- Heat transfer: According to the kinetic theory, the internal energy of a substance is generated from the motion of individual atoms or molecules. Heat energy is the form of energy, which transfers this energy from one body or system to another. This heat transfer can take place in a number of ways such as conduction, convection and radiation (About.com, 2010). Heat transfer analysis models the conductivity or thermal fluid dynamics of the material or structure. This includes the steady-state or transient transfer. Steady-state transfer is the constant thermo properties in the material that yield linear heat diffusion (Laboratory for Scientific Visual Analysis, 1997).

- Vibration: analysis is used when testing a material against random vibrations, shock, and impact. Any of these incidences could operate on the natural vibration frequency of the material which may lead in to resonance and subsequent failure.

3. Hand calculations (Part 1)

3.1 Round bars with shoulder fillets in tension

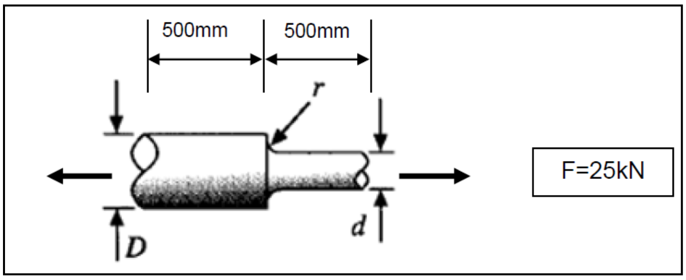


Figure 1: Round bars with shoulder fillet in tension

The Stress concentration factor in the above geometry can be calculated using the following formulas:

$$K = SCF = \frac{\sigma_{\max}}{\sigma_{nom}}$$

Therefore

$$\sigma_{\max} = K * \sigma_{nom}$$

From the given graph and given information (r, D and d) the value of K can be calculated for each radius. The same method as below was applied to different values of “r” and the results are as follow:

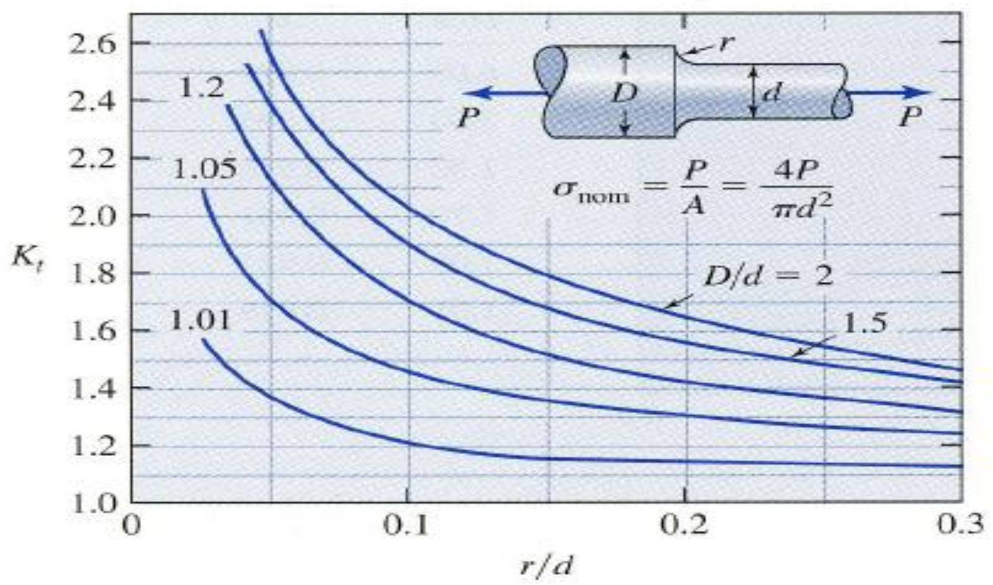


Figure 2: Theoretical stress-concentration factor K_t for a shaft with a shoulder fillet in axial tension.

Approximate formula

$K_t \approx B\left(\frac{r}{d}\right)^a$, where:

D/d	B	a
2.00	1.015	−0.300
1.50	1.000	−0.282
1.20	0.963	−0.255
1.05	1.005	−0.171
1.01	0.984	−0.105

<u>If r = 30:</u>

F = 25 kN → 25,000N

D = 150 mm

d = 100 mm

k = ?

$\frac{r}{d} = ?$

$\frac{D}{d} = ?$

k = SCF = $\frac{\sigma_{max}}{\sigma_{nom}}$

$\sigma_{max} = k\ \sigma_{nom}$

If r = 30 then $\frac{r}{d} = \frac{30}{100}$ = 0.3

$\frac{D}{d} = \frac{150}{100} = 1.5$

To find value of k we can use $\frac{r}{d}$ and $\frac{D}{d}$ to find the value of k from the above graph:

Therefore, $\boxed{\mathbf{k = 1.45}}$

In the order to find nominal stress $\sigma_{nom} = \frac{P}{A} = \frac{4P}{\pi d^2}$

P = F = 25,000N

$$\boxed{\sigma_{nom} = \frac{4\ x\ 25{,}000}{3.14\ x\ 100^2} = \mathbf{3.184\ N/mm^2}}$$

$\sigma_{max} = k\ \sigma_{nom}$ → σ_{max} = 1.45 x 3.184 = 4.616 N/mm^2

$\boxed{\sigma_{max} = \mathbf{4.616\ N/mm^2}}$

<u>If r = 20:</u>

$\frac{r}{d} = \frac{20}{100} = 0.2$

$$\frac{D}{d} = \frac{150}{100} = 1.5$$

From the graph $\boxed{\mathbf{k = 1.55}}$

$$\boxed{\sigma_{max} = 1.55 \times 3.184 = 4.935\ N/mm^2}$$

<u>If r = 15:</u>

$$\frac{r}{d} = \frac{15}{100} = 0.15$$

$$\frac{D}{d} = \frac{150}{100} = 1.5$$

From the graph $\boxed{\mathbf{k = 1.65}}$

$$\boxed{\sigma_{max} = 1.65 \times 3.184 = 5.253\ N/mm^2}$$

<u>If r = 10:</u>

$$\frac{r}{d} = \frac{10}{100} = 0.1$$

$$\frac{D}{d} = \frac{150}{100} = 1.5$$

$$\boxed{\mathbf{k = 1.85}}$$

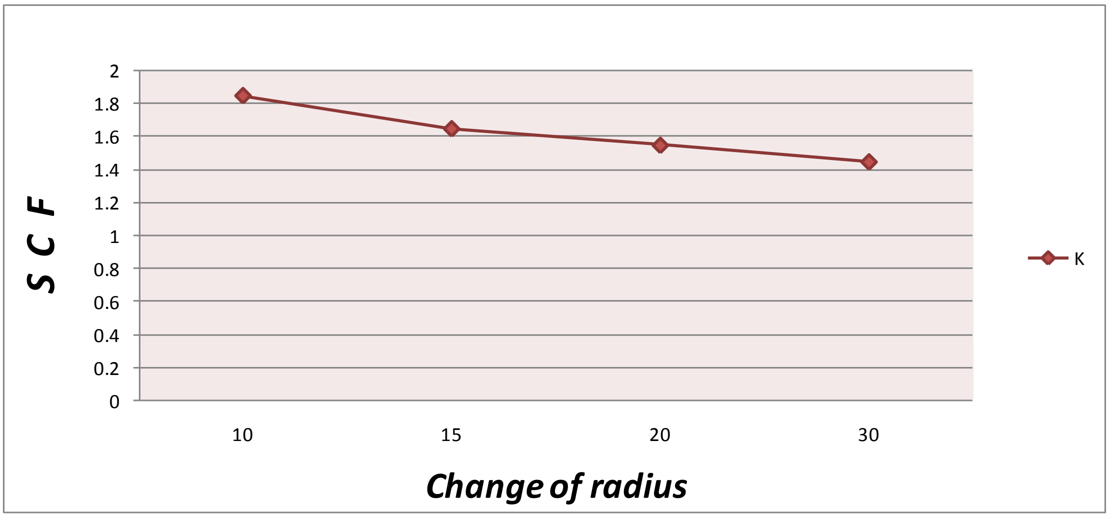
$$\boxed{\sigma_{max} = 1.85 \times 3.184 = 5.890\ N/mm^2}$$

radius r (mm)	10	15	20	30
r/d	0.1	0.15	0.2	0.3
D/d	1.5	1.5	1.5	1.5
K	1.85	1.65	1.55	1.45
nominal stress (N/ mm^2)	3.184	3.184	3.184	3.184
Max stress (N/mm^2)	5.890	5.253	4.935	4.616

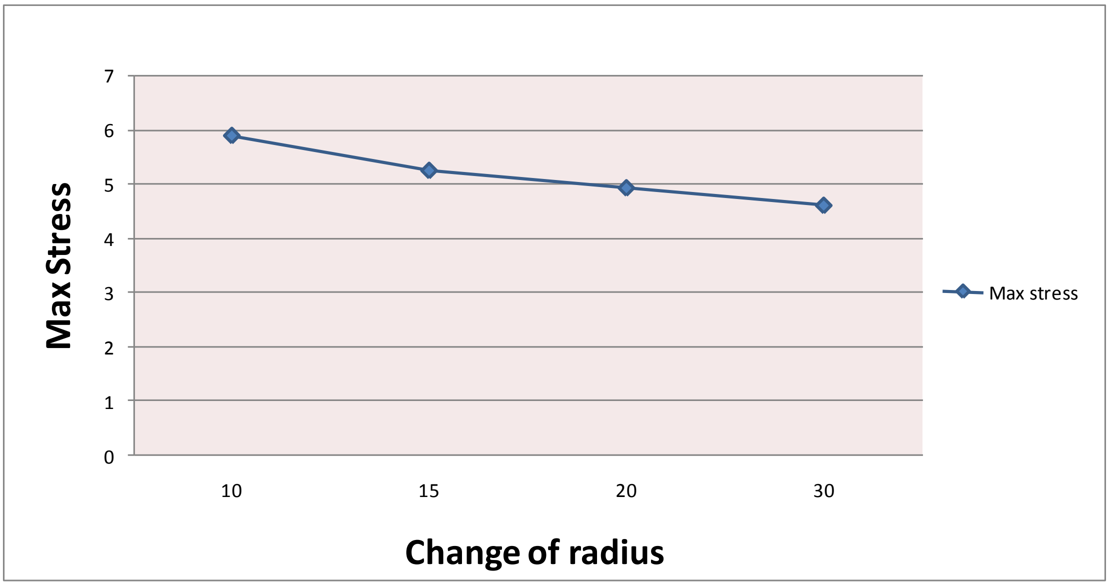
Table 2

The Graph 1 provided below shows that by increasing the radius of the fillet the stress concentration factor (SCF) and maximum stress decrease.

From this result it could be concluded that in order to reduce the maximum stress and SCF in the design decreasing the filler radiuses and avoiding sharp edges is one of the solutions.



Graph 1



Graph 2

3.2 Round bars with shoulder fillets in torsion

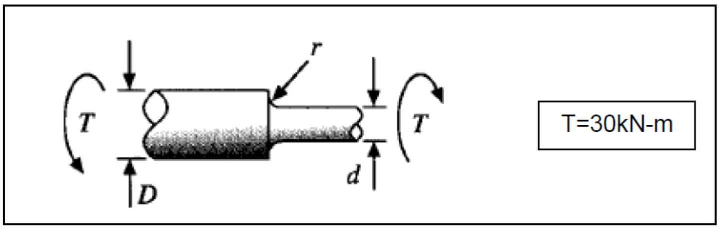


Figure 3: Round bars with shoulder fillet in torsion

The Stress concentration factor in the above geometry can be calculated using the following formulas:

$K = SCF = \dfrac{\sigma_{max}}{\sigma_{nom}}$ Therefore: $\sigma_{max} = K * \sigma_{nom}$

Form the given graph (Figure 4) and given information (r, D and d) the value of K can be calculated for each radius.

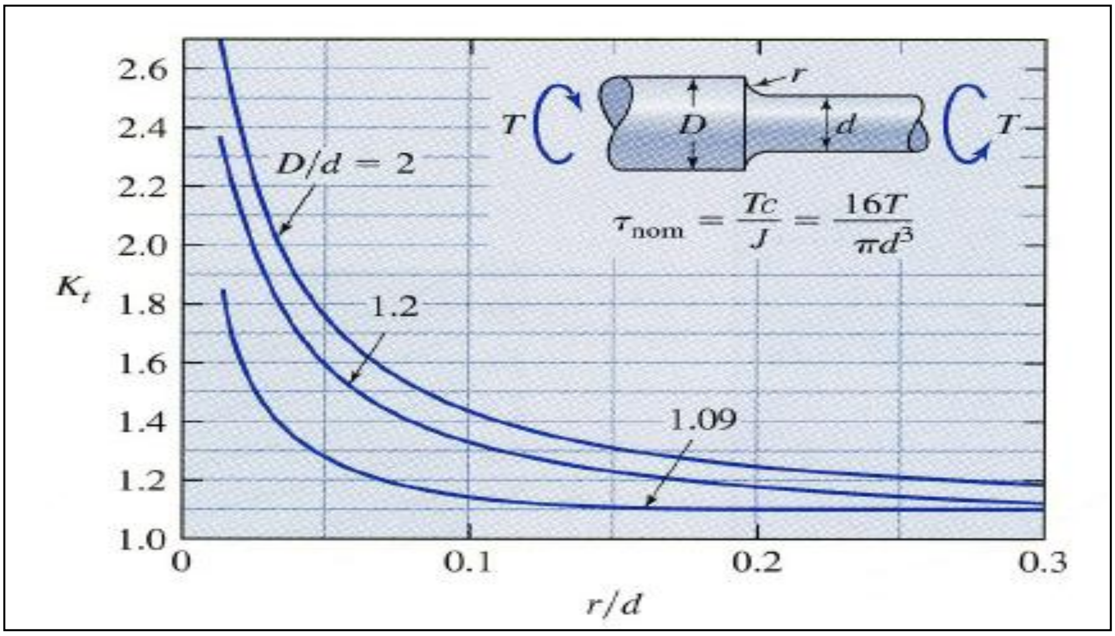


Figure 4

Approximate formula

$K_t \approx B\left(\frac{r}{d}\right)^a$, where:

D/d	B	a
2.00	0.863	−0.239
1.20	0.833	−0.216
1.09	0.903	−0.127

T = 30 kN_m $\qquad \frac{r}{d} = ?$

D = 150 $\qquad \frac{D}{d} = ?$

d = 100 $\qquad K_t = ?$

<u>If r = 30:</u>

$$\frac{r}{d} = \frac{30}{100} = 0.3$$

$$\frac{D}{d} = \frac{150}{100} = 1.5$$ from the graph $K_t = 1.075$

In order to find the nominal stress could use the nominal stress formula for a round bar in tension which is as follow:

$$T_{nom} = \frac{TC}{J} = \frac{16T}{\pi d^3}$$

$$\sigma_{nom} = \frac{16 x 30 x 1000 x 1000}{3.14(100)^3} = 152.866 N/mm^2$$

$\boxed{\boldsymbol{\sigma_{max} = K_t \text{ x } \sigma_{nom} = 1.075 \text{ x } 152.866 = 164.330 \text{ N/mm}^2}}$

<u>If r = 20:</u>

$$\frac{r}{d} = \frac{20}{100} = 0.2$$

$$\frac{D}{d} = \frac{150}{100} = 1.5$$ $K_t = 1.25$

$\boxed{\boldsymbol{\sigma_{max} = 1.25 \text{ x } 152.866 = 191.082 \text{ N/mm}^2}}$

<u>If r = 15:</u>

$$\frac{r}{d} = \frac{15}{100} = 0.15$$

$$\frac{D}{d} = \frac{150}{100} = 1.5$$ $K_t = 1.3$

$$\boxed{\sigma_{max} = 1.3 \times 152.866 = 198.335\ N/mm^2}$$

<u>If r = 10:</u>

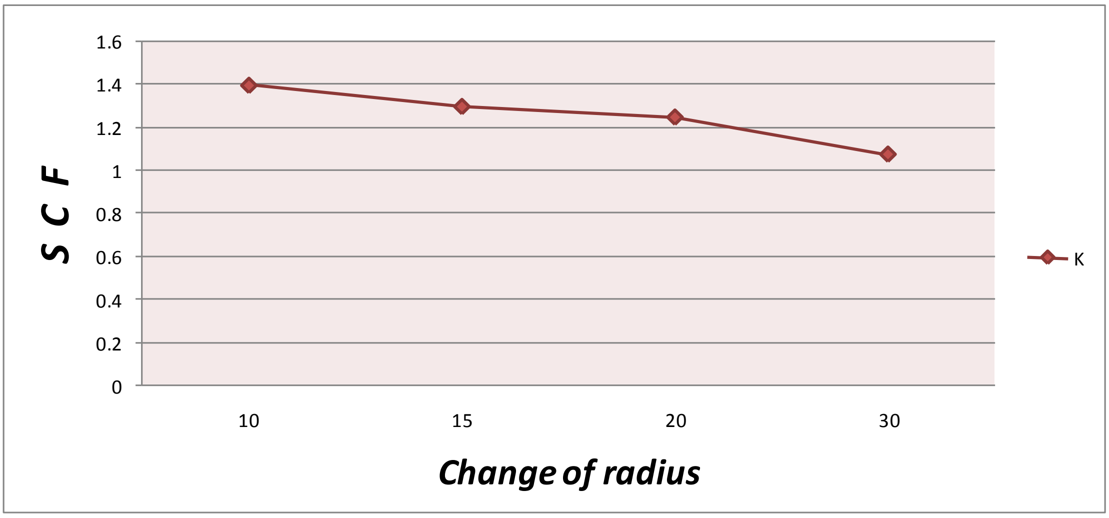
$$\frac{r}{d} = \frac{10}{100} = 0.1$$

$$\frac{D}{d} = \frac{150}{100} = 1.5 \qquad K_t = 1.4$$

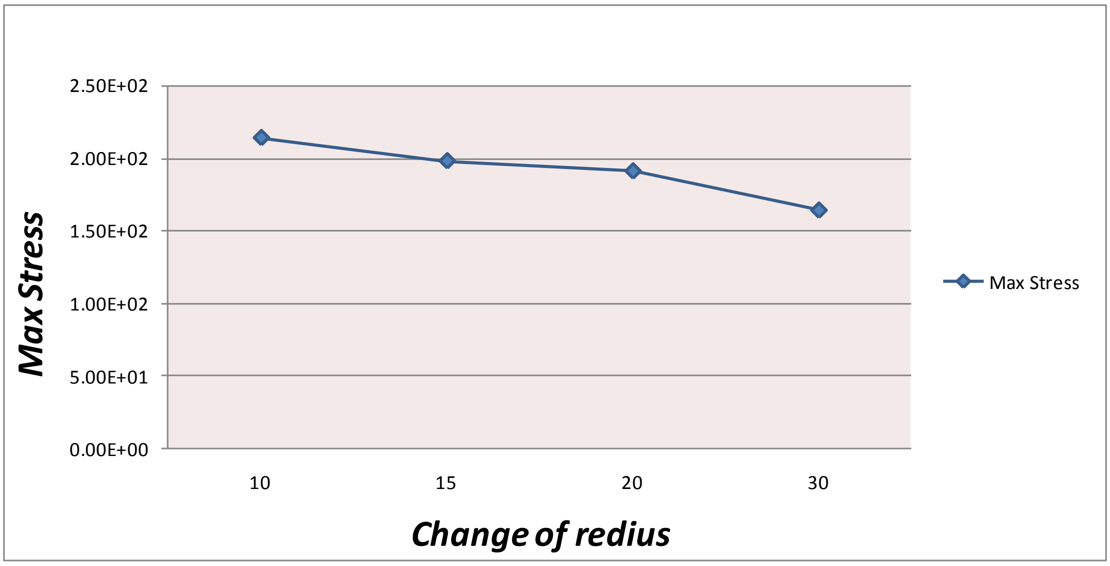
$$\boxed{\sigma_{max} = 1.4 \times 152.866 = 214.012\ N/mm^2}$$

radius r (mm)	10	15	20	30
r/d	0.1	0.15	0.2	0.3
D/d	1.5	1.5	1.5	1.5
K	1.4	1.3	1.25	1.075
nominal stress (N/ mm^2)	152.866	152.866	152.866	152.866
Max stress (N/mm^2)	2.140E+002	1.983E+002	1.910E+002	1.643E+002

Table 3



Graph 3



Graph 4

3.3 Round bars with shoulder fillets in bending

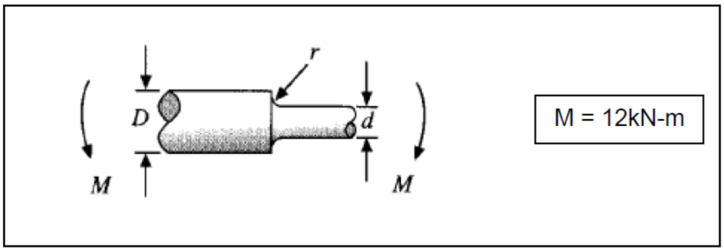


Figure 5

The Stress concentration factor in the above geometry can be calculated using the following formulas:

$$K = SCF = \frac{\sigma Max}{\sigma\ Nom}$$

Therefore

$$\sigma Max = K * \ \sigma\ Nom$$

Form the given graph and given information (r, D and d) the value of K can be calculated for each radius.

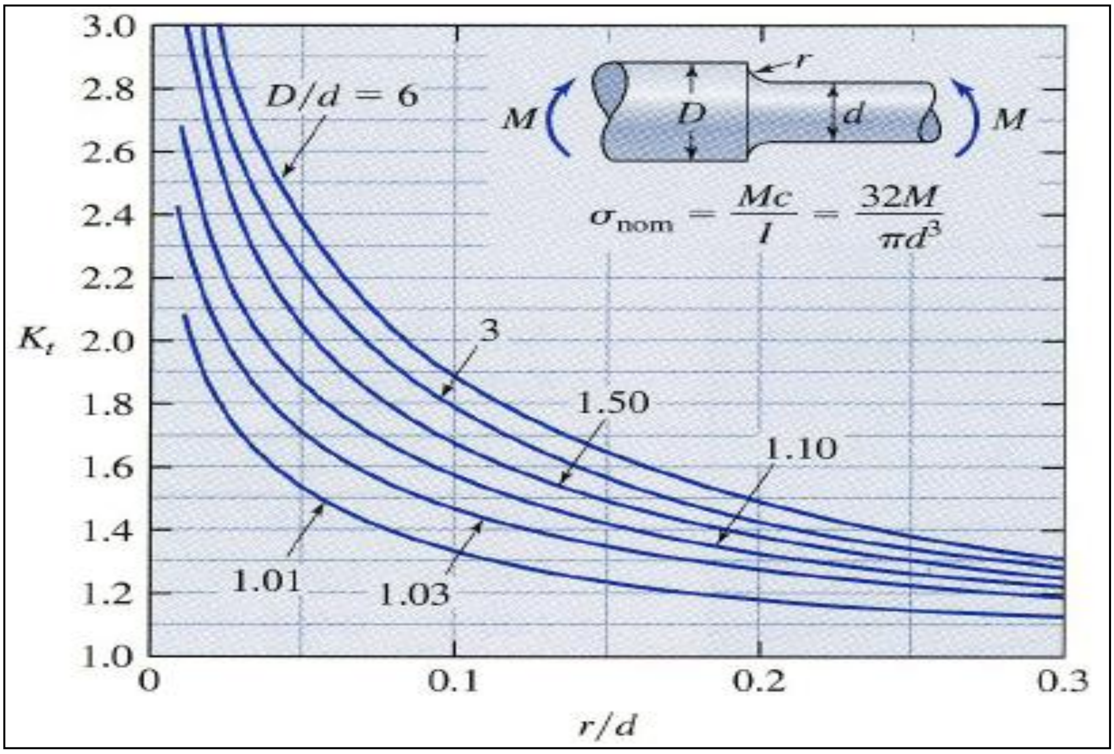


Figure 6

Approximate formula
$K_t \approx B\left(\frac{r}{d}\right)^a$, where:

D/d	B	a
6.00	0.879	−0.332
3.00	0.893	−0.309
1.50	0.938	−0.258
1.10	0.951	−0.238
1.03	0.981	−0.184
1.01	0.919	−0.170

<u>If r = 30:</u>

$\frac{r}{d} = \frac{30}{100} = 0.3$ M = 12 kN_m

$\frac{D}{d} = \frac{150}{100} = 1.5$ $K_t = 1.25$

K = S C F = $\frac{\sigma_{\max}}{\sigma_{nom}}$ $\sigma_{max} = K \times \sigma_{nom}$

$\sigma_{nom} = \frac{MC}{I} = \frac{32 x M}{\pi x d^3}$

$$\sigma_{nom} = \frac{32\,x12\,x1000\,x1000}{3.14\,x100^3} = 122.292\,N/mm^2$$

$\boxed{\sigma_{max} = 1.25 \text{ x } 122.292 = 152.862 \text{ N/mm}^2}$

If r = 20:

$$\frac{r}{d} = \frac{20}{100} = 0.2$$

$$\frac{D}{d} = \frac{150}{100} = 1.5 \qquad K_t = 1.38$$

$\boxed{\sigma_{max} = 1.38 \text{ x } 122.292 = 168.762 \text{ N/mm}^2}$

If r = 15:

$$\frac{r}{d} = \frac{15}{100} = 0.15$$

$$\frac{D}{d} = \frac{150}{100} = 1.5 \qquad K_t = 1.5$$

$\boxed{\sigma_{max} = 1.5 \text{ x } 122.292 = 183.438 \text{ N/mm}^2}$

If r = 10:

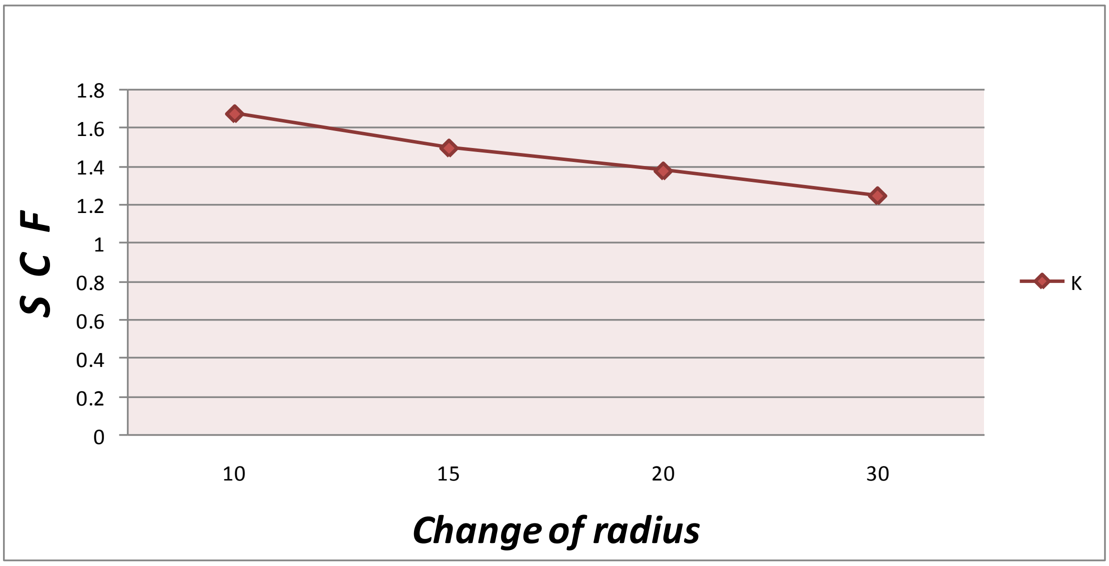
$$\frac{r}{d} = \frac{10}{100} = 0.1$$

$$\frac{D}{d} = \frac{150}{100} = 1.5 \qquad K_t = 1.68$$

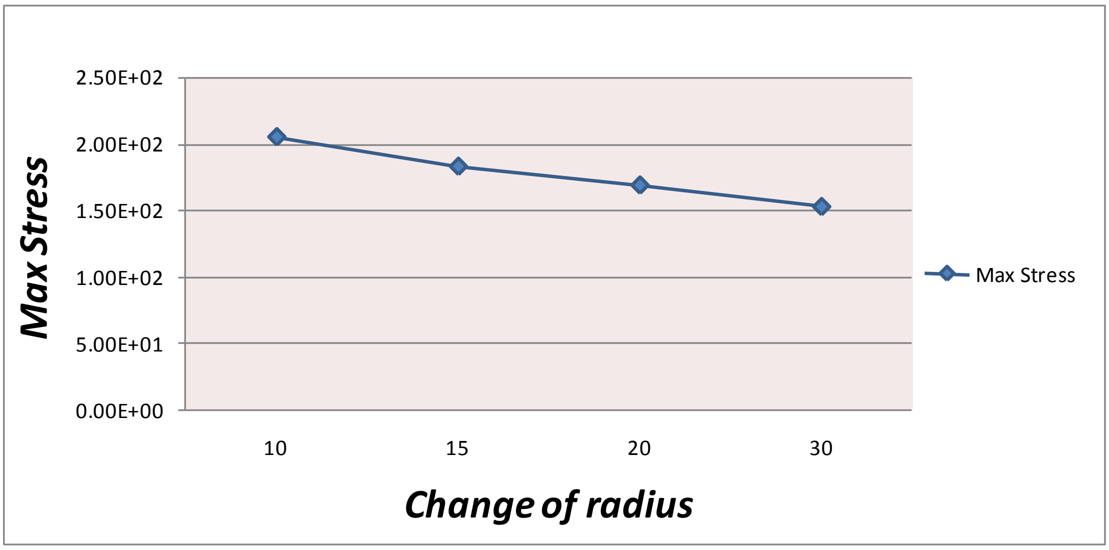
$\boxed{\sigma_{max} = 1.68 \text{ x } 122.292 = 205.450 \text{ N/mm}^2}$

radius r (mm)	10	15	20	30
r/d	0.1	0.15	0.2	0.3
D/d	1.5	1.5	1.5	1.5
K	1.68	1.5	1.38	1.25
nominal stress (N/ mm^2)	122.292	122.292	122.292	122.292
Max stress (N/mm^2)	2.054E+002	1.834E+002	1.687E+002	1.528E+002

Table 4



Graph 5



Graph 6

4. Simulations of round bar in 3 different geometries

4.1 Round bars with shoulder fillets in tension

Before carrying out the simulation the round bar needs to be modeled with four different shoulder fillet radiuses (10, 15, 20, and 30). To avoid any geometric error for the simulation the round bar was created based on one sketch only and by using the revolve command. Geometry of the round was created and at the end by adding the different fillet radiuses at the shoulder the model was ready for the FEA simulation.
Table below indicates the material of the model which was applied for this simulation

Material	Mass Density	Young's Modulus	Poison Ratio
Aluminum A356	2.67e-00	70000 N/mm (MPa)	0.3

Table 5

The next stage was to select the mesh type and size for the round bar geometry. The 3D tetrahedral mesh type was selected with 10mm mesh size for this simulation. The next step was to fix one end of the bar and apply force to the other end. By using fixed constrain for one end of the round bar 25 KN (25000N) force could apply to the other end.
The figure below (Figure 9) indicates the fixed constrain and force which were applied to each end of the bar.

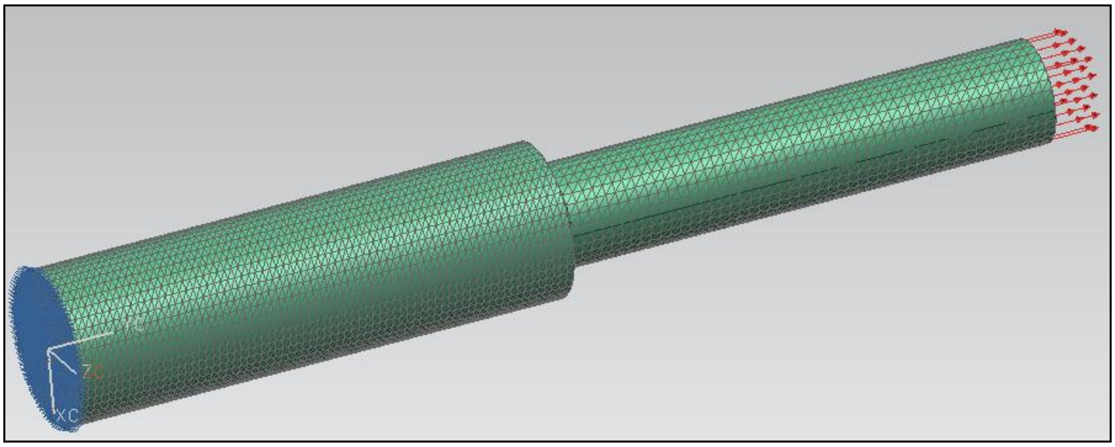


Figure 9

Finite Element Analysis

The same method was carried out for four different fillet radius of round bar in tension simulation. Figures below show the result of this simulation with different radiuses.

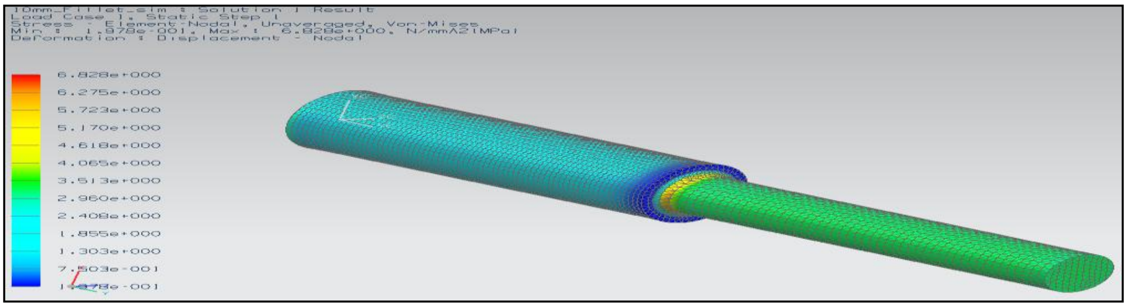


Figure 10: Radius = 10 (Max stress: 6.82 N/mm^2)

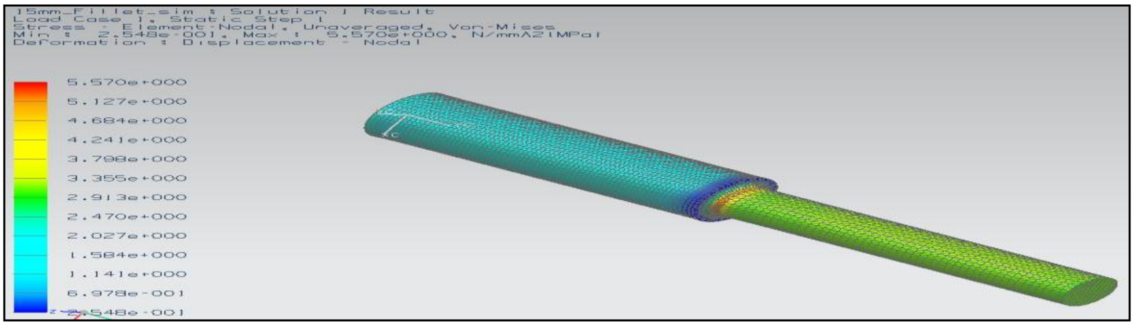


Figure 11: Radius = 15 (Max stress: 5.57 N/mm^2)

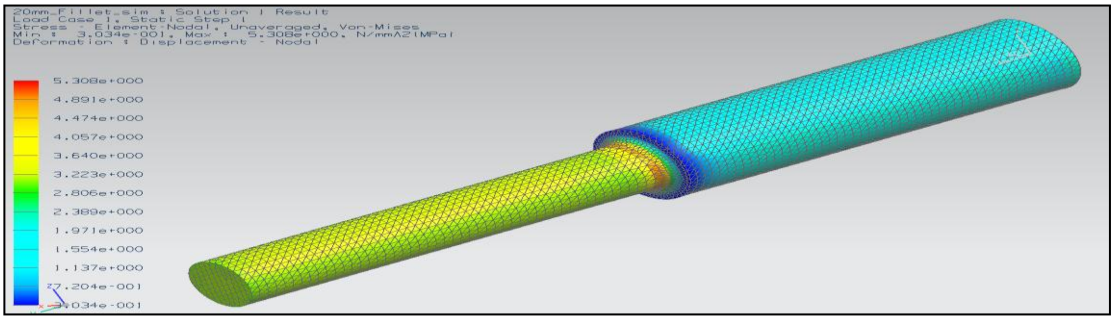


Figure 12: Radius = 20 (Max stress: 5.30 N/mm^2)

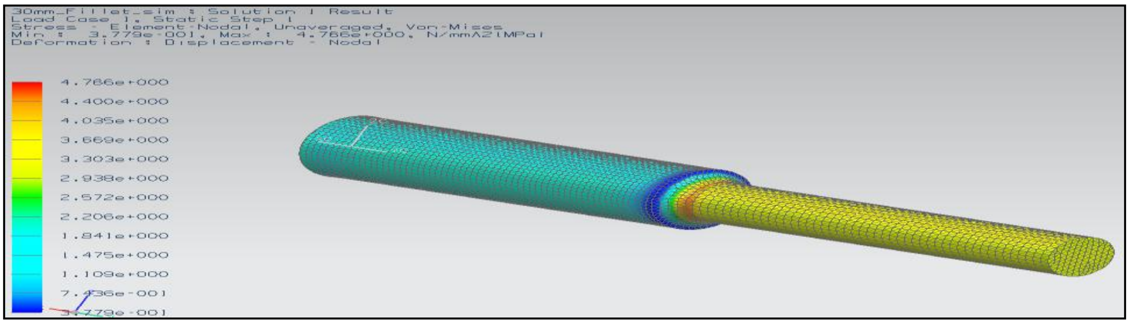


Figure 13: Radius = 30 (Max stress: 4.76 N/mm^2)

4.2 Round bars with shoulder fillets in torsion

At this stage the same method as above was repeated, only exception was that torque was selected as a type of load for the torsion simulation (T=30KN-m). The Figure 14 below indicates the fixed constrain and torque which was applied to the bar.

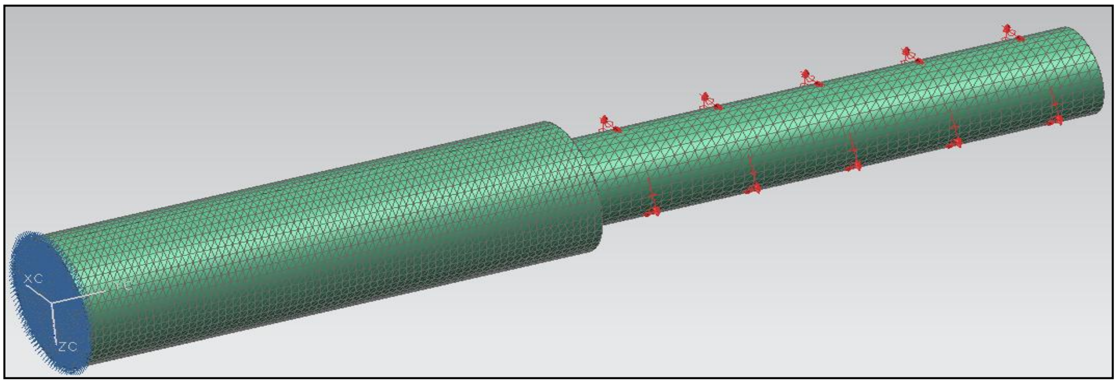


Figure 14

The figures below show the result of round bar with different radius of fillets in torsion simulation.

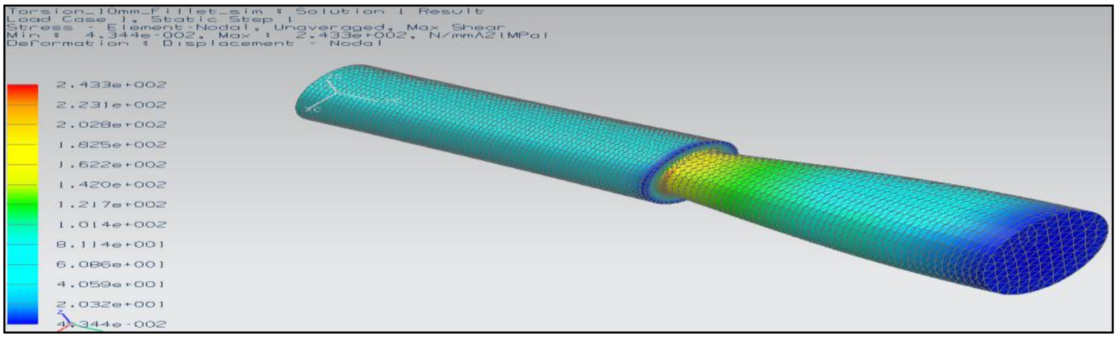


Figure 15: Radius of 10 (Max stress: 2.43e+002 N/mm^2)

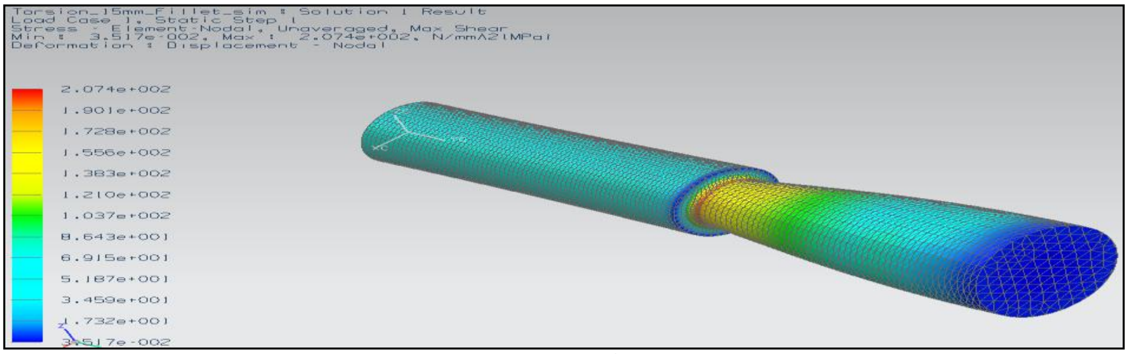


Figure 16: Radius of 15 (Max stress: 2.07e+002 N/mm^2)

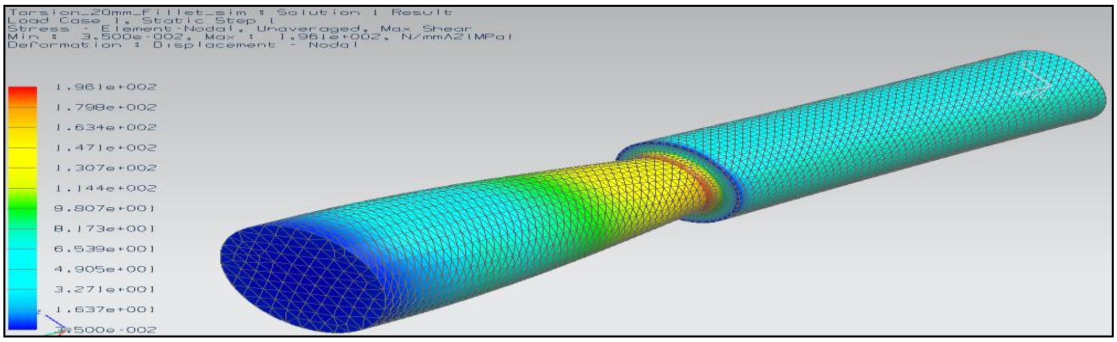


Figure 17: Radius of 20 (Max stress: 1.96e+002 N/mm^2)

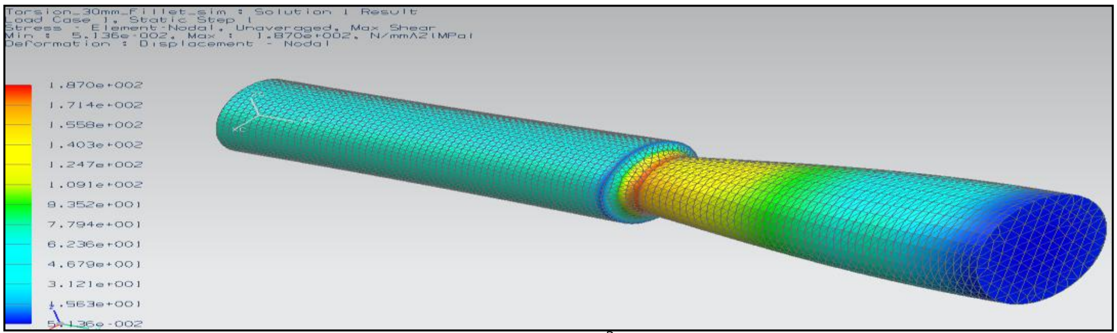


Figure 18: Radius of 30 (Max stress: 1.87e+002 N/mm^2)

4.3 Round bars with shoulder fillets in bending

For bending simulation all above has been repeated with the same material specification as well as the above methods. The type of load was selected as the moment in Z direction (M=12KN-m).

The Figure 19 below indicates the fixed constrain and the type of load which is moment and applied to the solid model.

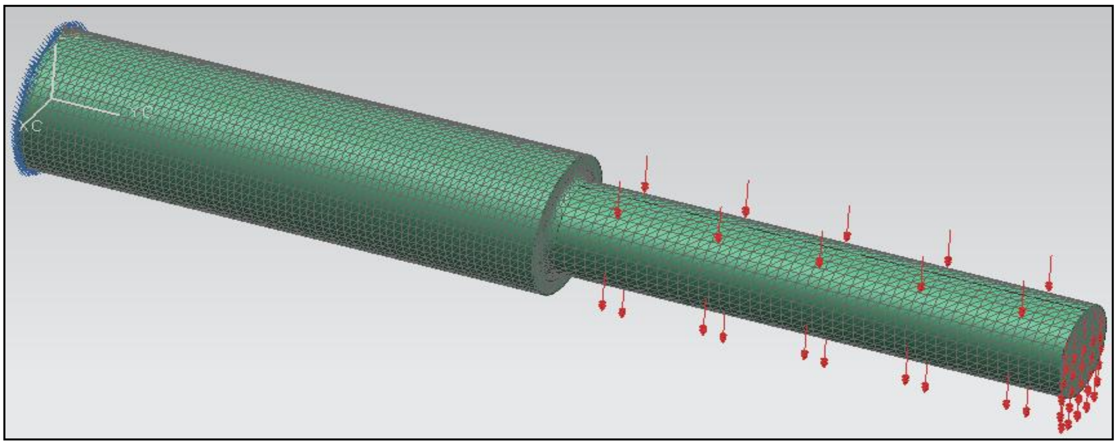


Figure 19

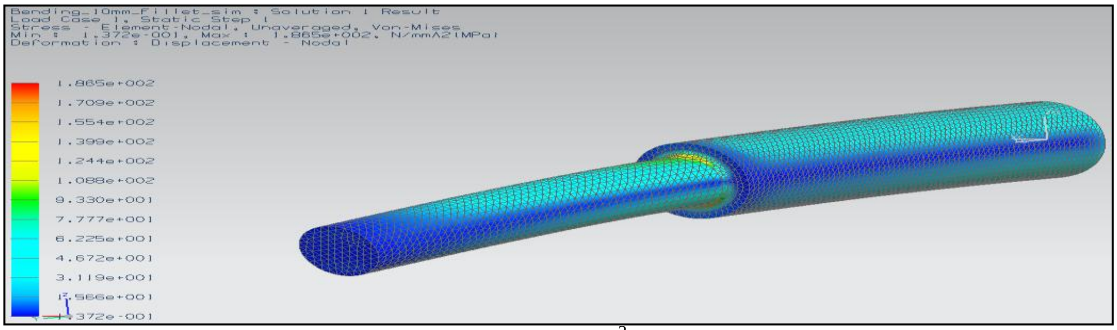


Figure 20: Radius of 10 (Max stress: 1.86e+002 N/mm^2)

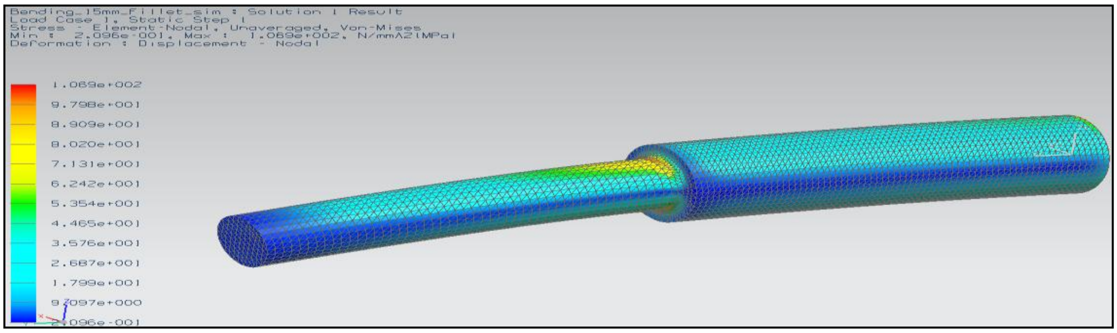


Figure 21: Radius of 15 (Max stress: 1.07e+002 N/mm^2)

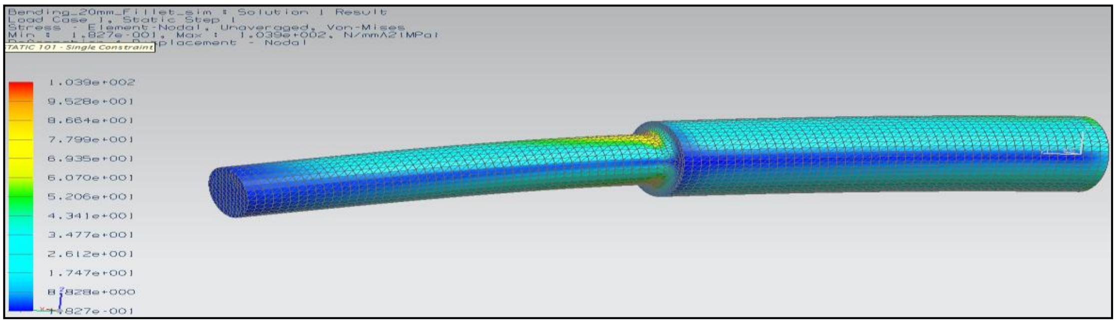


Figure 22: Radius of 20 (Max stress: 1.04e+002 N/mm^2)

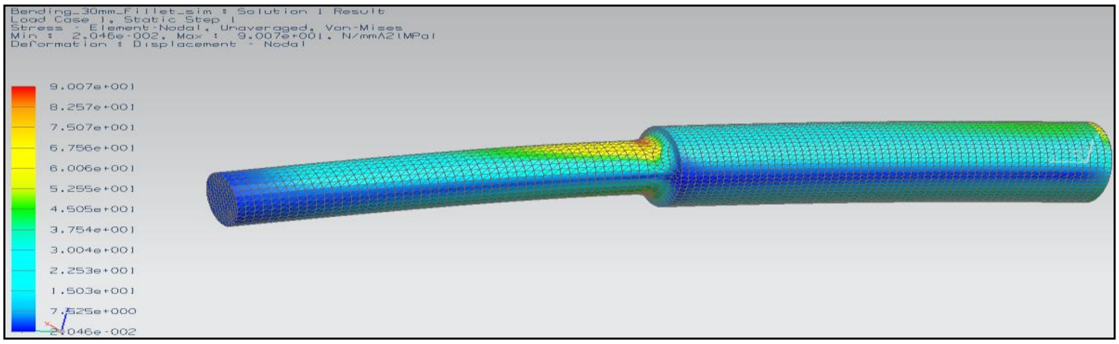
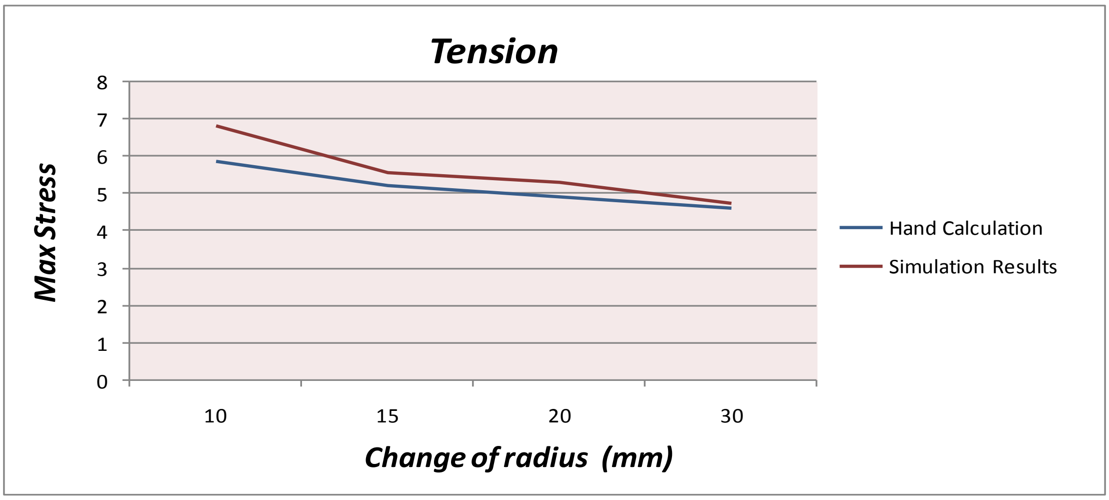


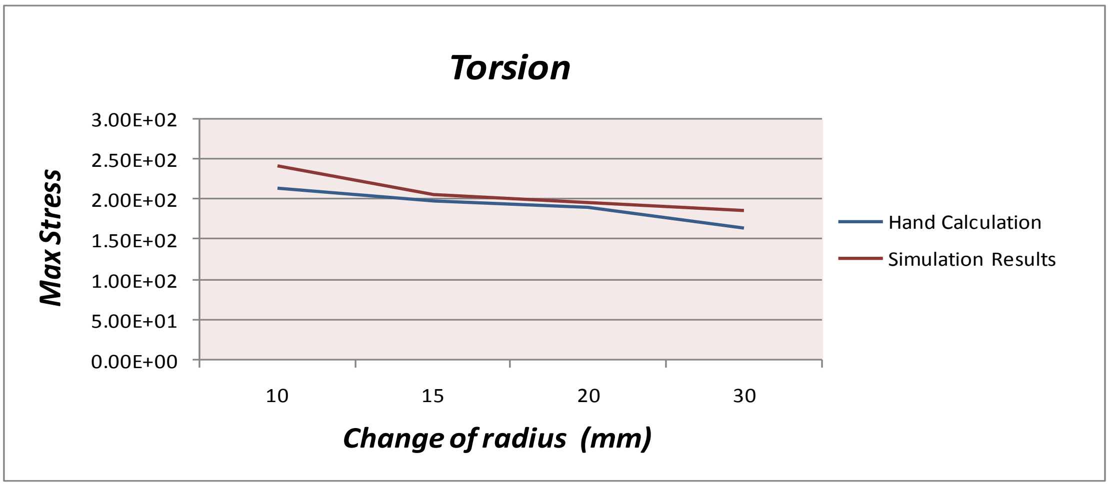
Figure: 23: Radius of 30 (Max stress: 9.01e+002 N/mm^2)

	r(mm)	Hand Calculation Max Stress	Mesh Size	Simulation Max Stress	Type of Load
Tension	10	5.890	10	6.82	F=25KN
	15	5.253	10	5.57	F=25KN
	20	4.935	10	5.30	F=25KN
	30	4.616	10	4.76	F=25KN
Torsion	10	2.140e+002	10	2.43e+002	T=30KN-m
	15	1.983e+002	10	2.07e+002	T=30KN-m
	20	1.910e+002	10	1.96e+002	T=30KN-m
	30	1.643e+002	10	1.87e+002	T=30KN-m
Bending	10	2.054e+002	10	1.86e+002	M=12KN-m
	15	1.834e+002	10	1.07e+002	M=12KN-m
	20	1.687e+002	10	1.04e+002	M=12KN-m
	30	1.528e+002	10	9.01e+002	M=12KN-m

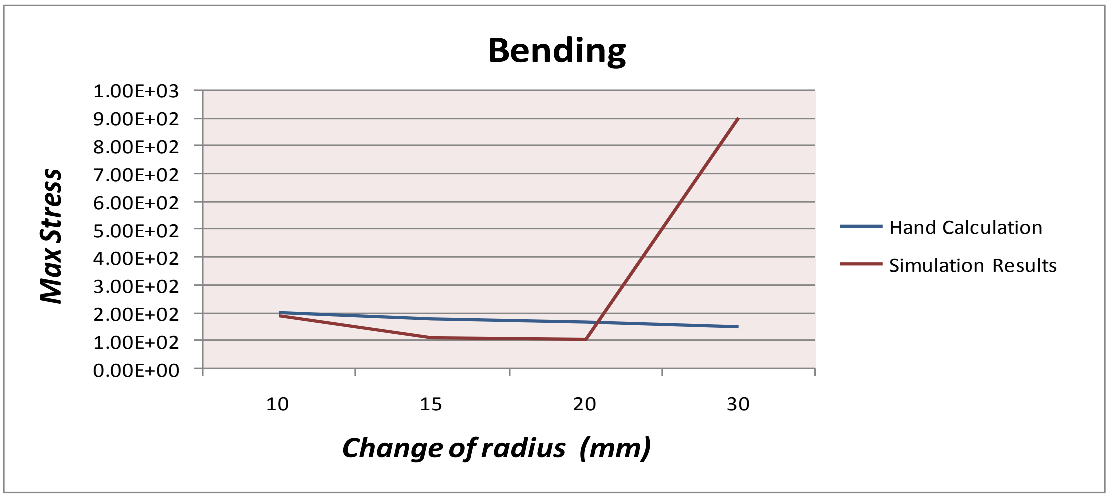
Table 6



Graph 7



Graph 8



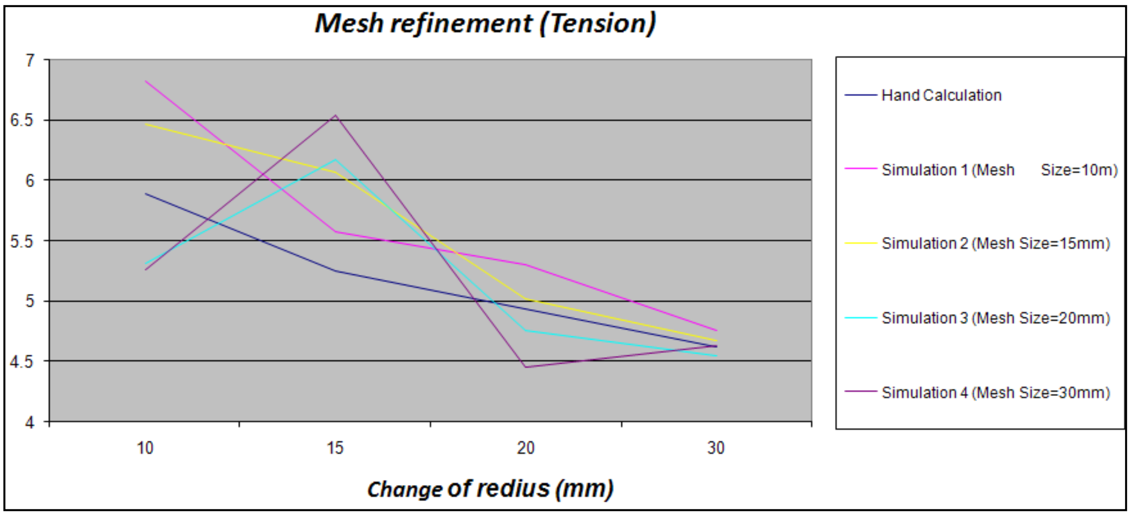
Graph 9

5. Mesh refinement

The very method carried out in the pervious part has been repeated but using different element sizes (10, 15, 20 and 30mm). The table 7 and graphs provided below demonstrate the comparison of different element sizes with hand calculations.

	r(mm)	Hand Calculation	Simulation 1 (Mesh Size=10m)	Simulation 2 (Mesh Size=15mm)	Simulation 3 (Mesh Size=20mm)	Simulation 4 (Mesh Size=30mm)
Tension	10	5.890	6.82	6.47	5.31	5.26
	15	5.253	5.57	6.07	6.17	6.54
	20	4.935	5.30	5.02	4.76	4.45
	30	4.616	4.76	4.67	4.55	4.63

Table 7



Graph 10

Conclusion (part A):

A few important points can be concluded by comparison of the mesh refinement with the hand calculation according to the data of the tables and the graphs provided above:

1. By increasing the size of radius the maximum stress on the part will decrease. Therefore avoiding very sharp edges in the design can decrease stress concentration factor and also the maximum stress.
2. According to the above graphs reducing the element size in meshing process can increase the accuracy of the result. But there is a stage that refining the mesh does not have any effect in the result. Therefore decreasing the size of the element would not be necessary as it increase the process of solving the simulation.

6. FEA simulation of wheel shaft assembly

At this stage it is required to model and analyse the steel shaft with a nylon pulley press-fitted onto the shaft and it is required to calculate the factor of safety and von-mises value while 20KN-m torque been applied to one end of shaft and the nylon pulley is fixed.

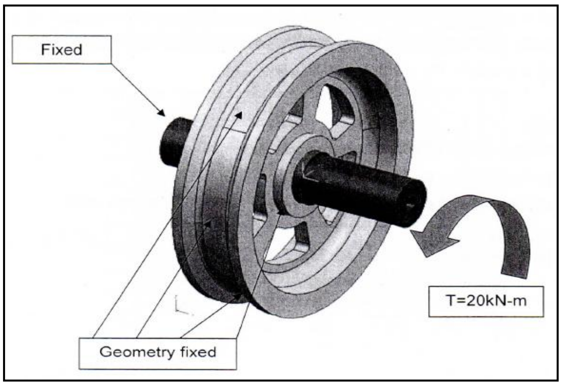


Figure 24

The first step to approach this simulation is to define the materials of the shaft and pulley which are chosen to be steel for the shaft and nylon for the pulley. The second step is to mesh the geometry, according to the above simulation the element type for the shaft and pulley been select as 3D tetrahedral and mesh size has been set to 10mm for the both steel shaft and nylon pulley figure 25 demonstrates the geometry after meshing process.

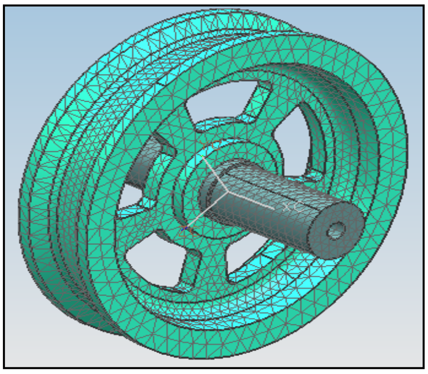


Figure 25

Next stage of this process is to set up the contact parameters for the simulation. Table below show the contact parameters which been set at this stage.

Max iterations force lope	10
Max iteration status loop	20
Contact force tolerance	0.01
Contact change for convergence	Number of contact
Allowable number of contact change	20
Contact status	Start from previous
Initial penetration gap	Set to zero

Table 8

Defining a contact between the shaft and the pulley is the next stage of the process. It aids the pulley and the shaft to move together when applying the torque to one end of the shaft. Surface to surface contact method has been applied between the middle of the shaft and inside of the pulley with target offset of 0.5mm. The following image illustrates the contact between the two parts.

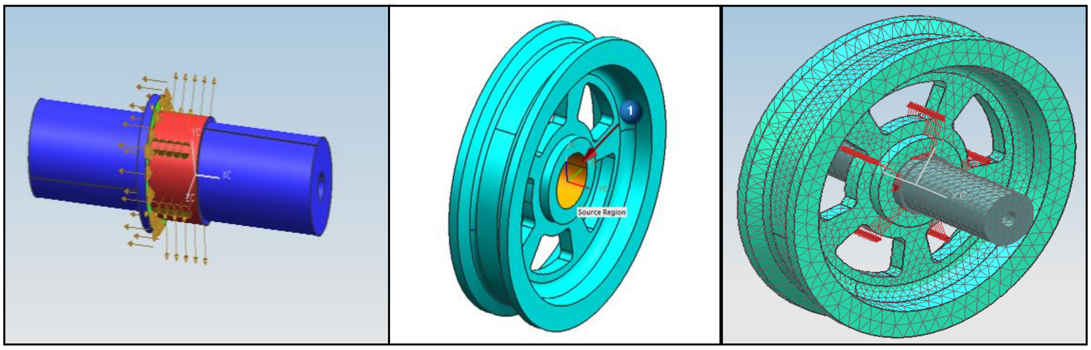


Figure 26

The next stage before starting the simulation is to define the constraint and apply the load.

In order to stop pulley when the load is applying to the shaft, it must be fixed so it could stay still due to the load. And 20KN-m torque is applied to one end of the shaft and fixing the other end of the shaft as shown in the below figure.

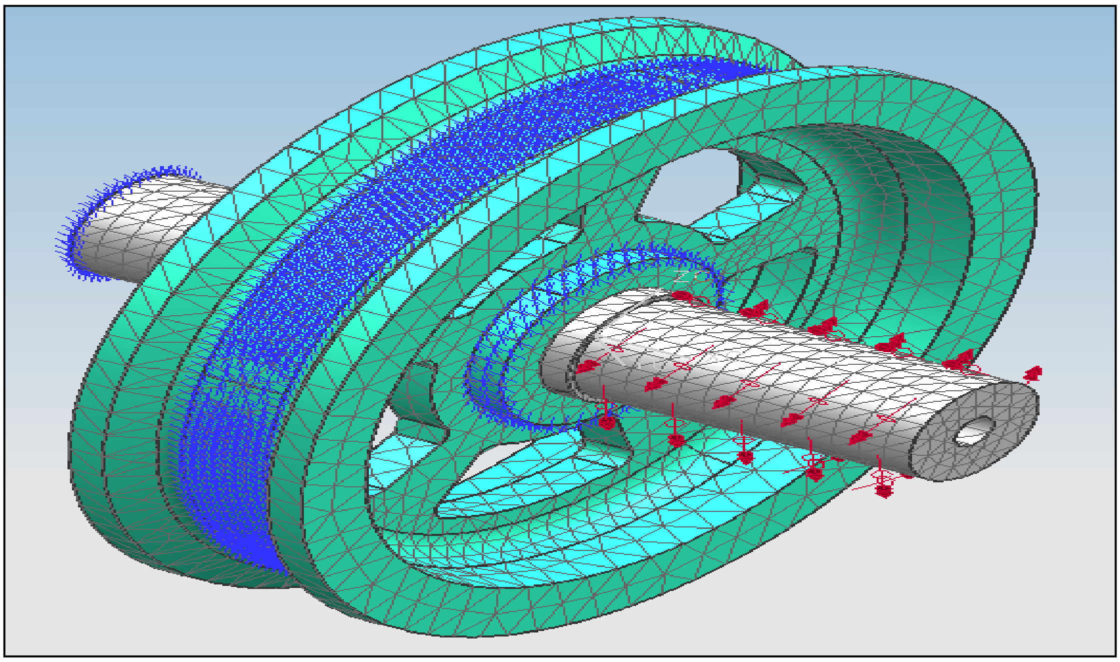
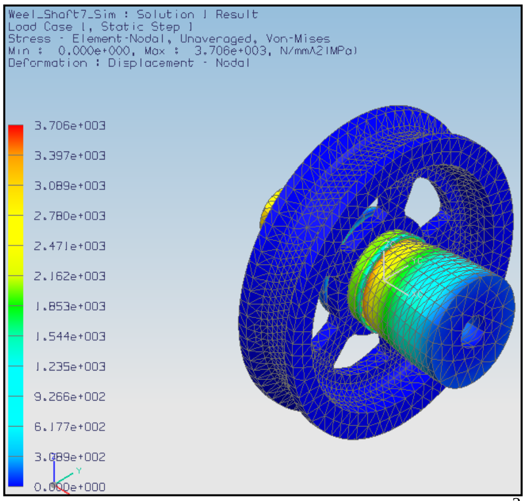


Figure 27

Finite Element Analysis

The next stage of this process is to solve the simulation in the order find the maximum stress on the wheel shaft (pulley). The below figure shows the maximum stress of the wheel shaft when 20KN-m torque been applied to one end of the shaft and the other side as well as wheel been fixed.



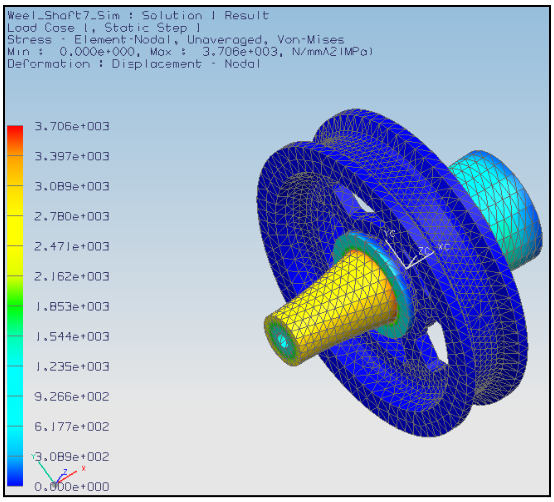


Figure 28 Max stress: 3.70e+003 N/mm^2

Figure 29

In order to calculate the factor of safety it is required to find the von-mises stress (max stress) and the yield strength of the material.

The von-mises stress is found to be 3.706e+003:

Yield strength of steel in normal condition is: 137895 (found from material properties NX6)

$$\textbf{FOS} = \frac{\textit{yield strenght}}{\textit{maximum stress}} = \frac{137895}{370} = 0.372$$

7. FEA simulation of re-design wheel shaft assembly

At this stage geometry of the wheel shaft has been re-design to improve the factor of safety of the model. All above process been repeated step by step for this stage. The main dimension of the original wheel shaft kept same as it was.

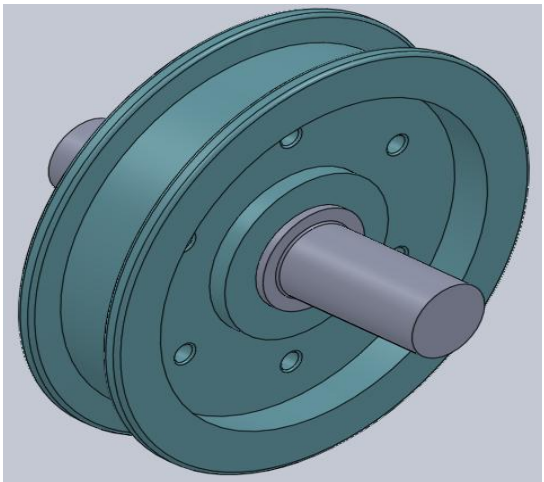


Figure 30

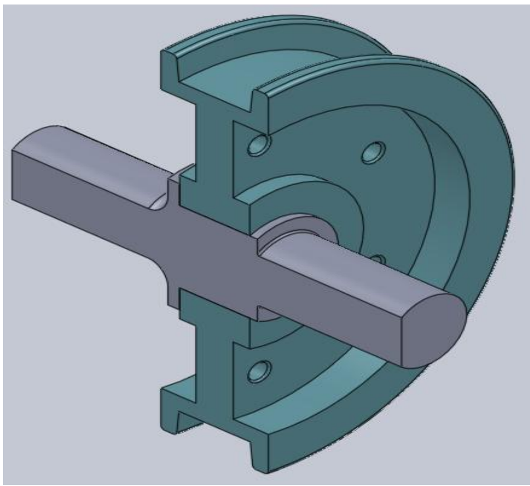


Figure 31

In the order to improve the factor of the safety the holes in the wheel shaft been modified to the smaller round holes. Second, as can seen from the section view of the wheel shaft (pulley) from above figure the flat section over the wheel been re-sized to the bigger area in order to distribute the load over the geometry of the wheel. Additional to that some fillet been applied to both edge of middle section of the shaft.

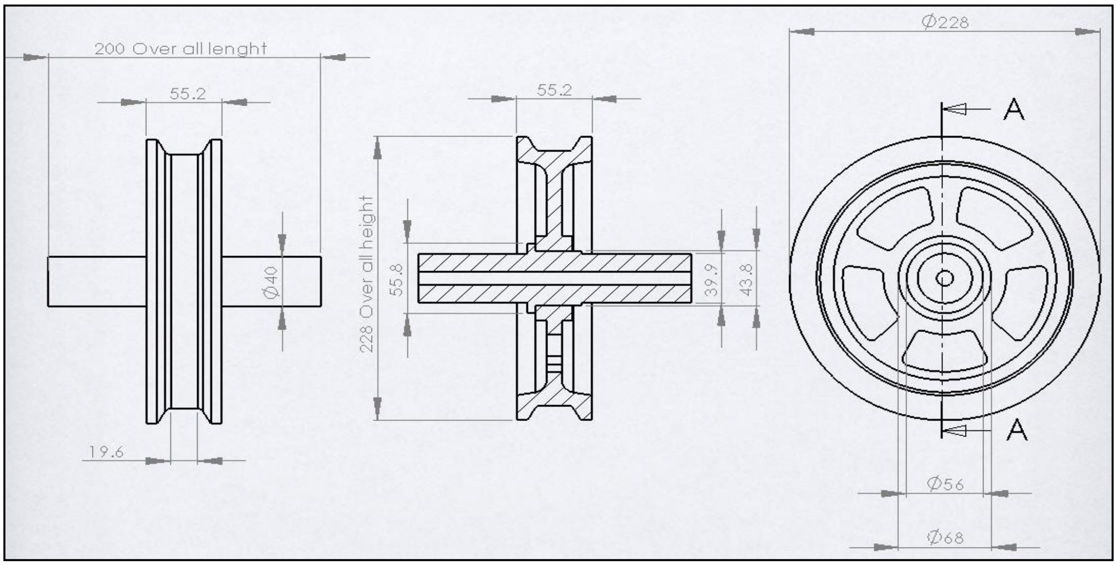


Figure 31 Original Pulley

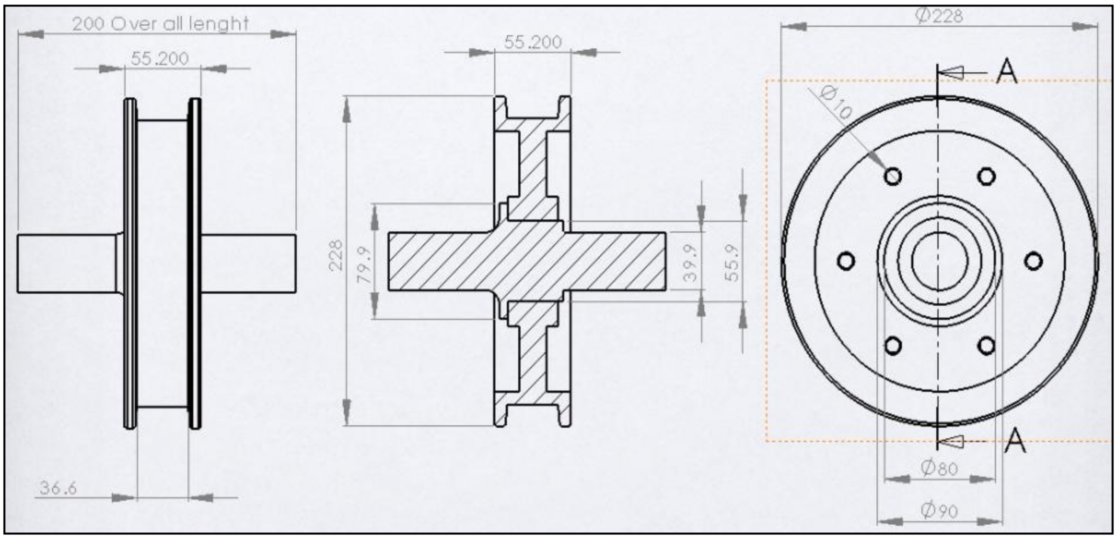
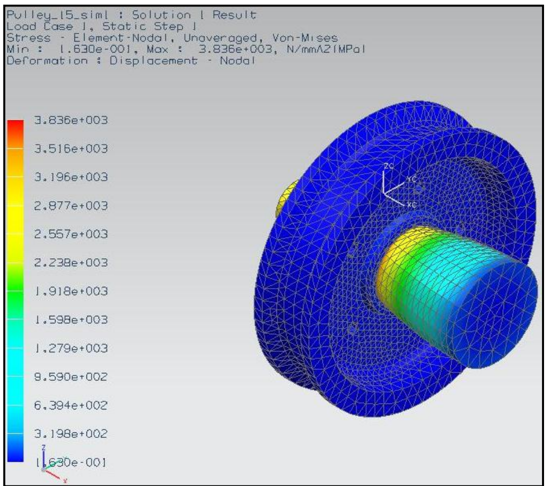


Figure 32 Re-design pulley

By looking at figure 31 and 32 could notice the changes which been done to the new wheel shaft.

The below figures indicated the results of the re-design wheel shaft.



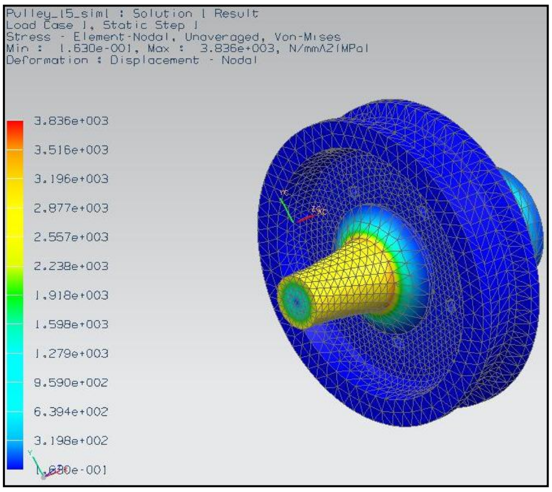


Figure 33 Max stress: 3.836e+003 N/mm^2 Figure 34

The Von-mises (Max stress) is found to be 3.836e+003

$$\textbf{FOS} = \frac{yield\ strenght}{maximum\ stress} = \frac{137895}{383} = 0.360$$

	Original Wheel_Shaft	Re-design Wheel_Shaft
Von-mises	3.71e+003	3.944e+003
SOF	0.372	0.360

Table 9

8. Discussion

Comparing the round bar hand calculation with the simulation results with mesh size 10 could highlight that the results were almost similar when compared to each other. Also, it could be clearly seen in mesh refinement section that by increasing the size of radius the maximum stress on the part will decrease. Therefore, avoiding very sharp edges in the design could decrease stress concentration factor as well as the maximum stress. Furthermore, according to the mesh refinement section graphs reducing the element size in meshing process can increase the accuracy of the result.

Taking the all above reasons into account, then in order to reduce the maximum stress which is on the shaft and improving the factor of safety, the pulley was re-designed for the better performance. The overall sizes of the original pulley were kept same as they were; however, the modification on the component was carried out over small details of the pulley for improving the FOS.
Since the wheel is press-fitted onto the shaft and shaft has a torque of 20KN-m, therefore the maximum stress was applied to the middle section of the shaft bar. For that the pulley was re-designed as following:

The hole on the original shaft has been removed and the mid section of the shaft bar been re-sized by 4 mm. Additionally to that, some fillet been applied to both ends of mid section of the shaft bar. This method increases the stiffness of the component as well as distributing the load over wider areas (See figure 32).

The mid section of the original wheel where the holes were was re-designed and instead replaced with 6 small round holes with 10mm radius. The flat top section of the wheel where the load applies on the wheel has been modified to the bigger area and thinner wall section. By doing that the load can be distributed over the wider area of material - that increases the factor of safety of the pulley (Figure 32).

9. Design suggestion

By comparing the FOS of the original model to re-designed model, it could be noticed that the FOS of re-designed model was decreased by 0.23 percent which makes the model less reliable. In order to improve that, there are some changes which could be applied to the re-designed model

Since all the stress is on the shaft and the wheel is fixed all the time, therefore it would be better to leave the wheel geometry same and to re-design the shaft. In order to improve the FOS of the shaft the following can be suggested:

1. Changing the radius of the shaft to make it thicker
2. Re-design the shaft into 2 sections instead of 3
3. Applying fillet to the edge to avoid the sharp edges which can reduce the max stress

10. Conclusion

Finite element (FEA) method is a numerical analysis technique, which is used by engineers, scientists and mathematicians to achieve solutions as well as integral equations. This method approximately describes a wide range of physical and non-physical problems.

At the beginning of the assignment some hand calculation were done for the round bar with different radius of fillet (10, 15, 20, 30) in tension, torsion and bending. All the results were compared with simulation which was carried out in NX6 simulation software. In the next stage of the progress tension case been chosen to do the mesh refinement to converge to maximum stress which was found in previous section from the hand calculation and graphs.

The process of wheel shaft simulation started after above steps by applying 20KN-m torque to one end of shaft and fixing the other end as well as the wheel. After finding the results from the simulation the wheel shaft was re-designed in order to improve the factor of safety of wheel shaft as well as the maximum stress over the wheel shaft geometry.

Finite Element Analysis

Today FEA simulation is widely used all over the world, engineers use this digital simulation to save time and money before manufacturing any component. However, there is a prototype testing as well which is more close to real life situation. By testing the prototype component, the accuracy of the digital simulations could be proven.

11. References

About.com, (2010). Introduction to heat transfer. [Online] Available from: http://physics.about.com/od/thermodynamics/f/heattransfer.htm [Accessed: 11 June, 2010].

Laboratory for Scientific Visual Analysis, (1997). Introduction to Finite Element Analysis. [Online] Available from: http://www.sv.vt.edu/classes/MSE2094_NoteBook/97ClassProj/num/widas/history.html [Accessed: 11 June, 2010].

Salford University, (2010). Finite Element Method. [Online] Available from: http://www.acoustics.salford.ac.uk/student_area/bsc3/computer_simulation/Fem.pdf [Accessed: 10 June, 2010].

Technische Universitat Munchen, (2010). Finite Element Modeling: Introduction. [Online] Available from: http://www.st.bv.tum.de/content/teaching/fem1/ifem.ch07.pdf [Accessed: 11 June, 2010].

Urry, S.A., Turner, P.J., (1986). Solving Problems in Solid Mechanics. Volume 1. England: Longman Scientific & Technical.

Urry, S.A., Turner, P.J., (1986). Solving Problems in Solid Mechanics. Volume 2. England: Longman Scientific & Technical.