VOL. 11, NO. 9, MAY 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

FINITE ELEMENT ANALYSIS OF A SHAFT SUBJECTED TO A LOAD

K. Sathishkumar and N. Ugesh
Department of Mechanical Engineeering, United Institute of Technology, Coimbatore, India
E-Mail: sathishkuppuraj@gmail.com

ABSTRACT

The project is mainly concentrated about the analysis of a shaft with the help of a ANSYS software under workbench. In this the shaft is taken from the head stock of the lathe machine. In this analysis the shaft is connected with bearing and gear. This is the major important component to be taken into account while designing. The objective is to build a model and assemble the part files and to analyze the various stress and deformation. The part files and assembly are done by using CREO software and the analyzing are done by using a ANSYS software. The static analysis is used to analyze the stress and deformation of the shaft when it is subjected to a particular load and the modal analyze is executed to govern the vibration features (mode shapes and natural frequencies) of shaft. The results obtained by the stress analysis is found to be good agreement and modal analysis i.e., vibration characteristic like frequency and mode shapes are presented are within the limit

Keywords: shaft, bearing, gear and pinion, static stress analysis, FEA, CREO, modal analysis, natural frequency, mode shapes.

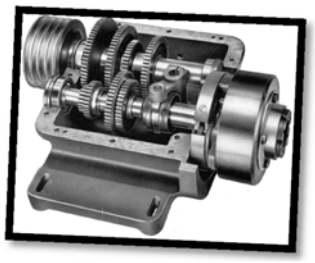
1. INTRODUCTION

The shaft is one of the major components in the power transmitting system. In head stock shaft plays a major role in the transmitting power to the spindle, The shaft assembly has a ball bearing and gear parts in it so it leads to some of a stresses induced with itself it produces some amount of deviation of a shaft.

Due to the stresses induced in the gears and shaft it leads to failure or reduce of a life of the shaft

1.1 Objective of the work

- Build a model of a shaft assembly.
- To carry a static analysis using ANSYS WORKBENCH for analyzing the load effect on the shaft. FEM enables to find critical location and quantitative analysis of the stress distribution and deformed shapes under the loads
- To carry out their modal analysis i.e., natural frequencies and the mode shapes.



2. MODELING AND FINITE ELEMENT ANALYSIS OF A SHAFT

The 3-D solid model of the shaft and its component was built using CREO software. A sys workbench so used for pre-processing, solving and post processing.

Material property of the shaft assembly is coincided as structural steel for shaft were selected from ANSYS metal library. Boundary conditions are applied to the shaft and the finite element model is used to calculate the stress, deformation, and also the frequency and mode shapes in the shat assembly using ANSYS software.

2.1 Parts of a model

The various parts in the assembly are,

i. Shaft
ii. Ball bearing
iii. Gear
iv. Pinion

Shaft

A shaft is a part that is used to transmit a power from one drive to other drive. In machining tools, a spindle is a rotating axis of the machine, which often has a shaft at its heart. In our the shaft is made up of a steel material with 50mm diameter and a length of a 750mm

Ball bearing

A ball bearing is a type of rolling-element bearing that uses balls to maintain the separation between the bearing races. The purpose of a ball bearing is to reduce rotational friction and support radial and axial load. As one of the bearing races rotates it causes the ball to rotate as well. Because the balls are rolling they have a much lower coefficient of friction that if two flat surfaces were sliding against each other. In this the project the bearing diameter is 80 diameter for outer and 50 for inner diameter for this I taken the deep grove ball bearing of SKF6010

Gear and Pinion

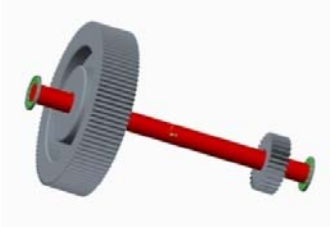
A Gear or cog wheel is a rotating machine part having cut teeth, or cogs, which mesh with another part to transmit torque, in most of the cases with teeth on the one gear being of identical shape, and often and also with the shape of the other gear. The most common situation is for a gear to mesh with another gear; however a gear can also

mesh with a linear toothed part, called a rack, there by producing transudation instead of rotation

A Pinion is a gear with a small number of teeth, especially one engaging with a rack or large gear. Although the larger gear is rotating less quickly, its torque is proportional greater. One subtlety of this particular arrangement is that the linear speed at the pitch diameter is the same on the both gears. In this the outer diameter of the gear is of diameter 500 mm and the pinion has a outer diameter of a 75 mm.

A. Shaft assembly that is designed in creo

The Figure 2.5 and 2.6 shows the pictorial representation of the shaft assembly. The following model is designed in the creo.



The CREO model is imported into the respective file format of the FEM design software ANSYS R14.5

B. FEM Modal

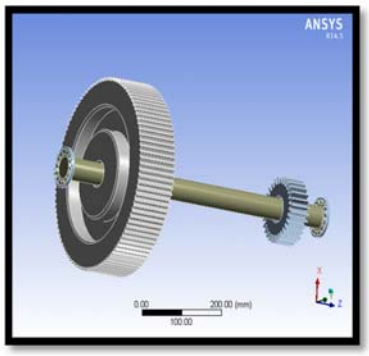


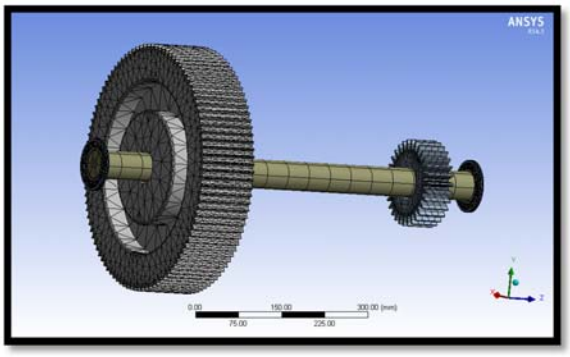
Figure-1. 3-D Model.

Units

Unit system	**Metric** (mm, t, N, s, mV, mA) **Degrees** rad/s Celsius
Angle	Degrees
Rotational velocity	rad/s
Temperature	Celsius

FEA Modelling helps in efficient managing of deformation, von misses stress and shear stress and also in finding the natural frequencies and mode shapes in any mechanical component and system.

The figure shows the discrete modelling of a shaft assembly.



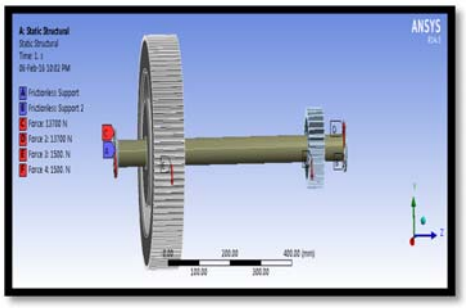
The above figure gives the meshing of a model.

Meshing details

Nodes	61034
Elements	26457

3. BOUNDARY CONDITION

The shaft has a bearing on the left and right end on the shaft due to the bearing on the both ends the bearing executes the static load C_0 = 1370 kg on the both sides. From the left hand side a gear is connected at a distance of 150 mm and a small pinion is placed from the right hand side of the shaft at a distance of 100 mm. The two bearing has excited a force on 13700 N on Y direction and the other force is excited on the opposite direction the Y axis. The two forces are acting on the gear and the pinion on the shaft. The frictional support is placed on the either side of the bearing. The condition which is applied in the ansys is shown below.



4. RESULTS AND DISCUSSION

The project is divided into two domains:

a) Static analysis

b) Modal analysis

4.1 Static stress analysis

It is used to define a displacement, stresses, deformation etc.., due to the influence of static loading condition. It estimates the properties of a steady loading

condition on a component, but over loading the inertia and damping effects, such as the one affected due to time varying load.

The some of the result that is taken from the ANSYS Software when it is subjected to 2 bearing loads and 1 gear and 1 pinion loads,

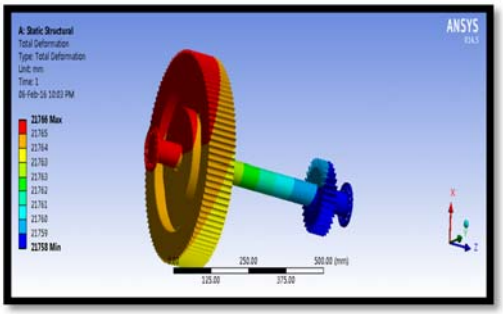


Figure-2. Total deformation on a shaft.

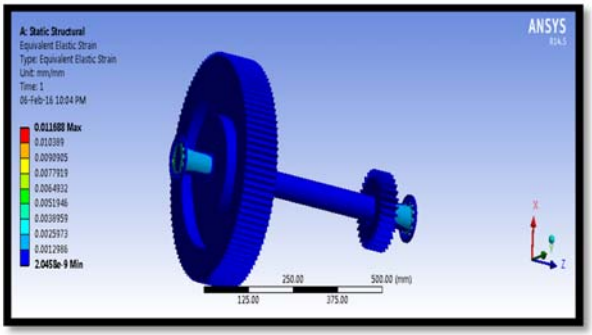


Figure-3. Equivalent elastic strain on the shaft

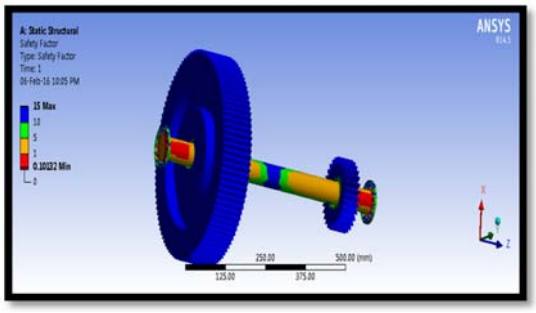


Figure-4. Factor of safety on the shaft.

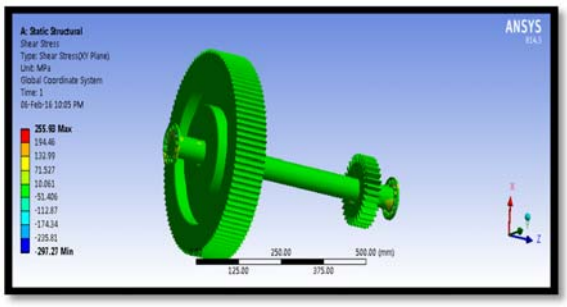


Figure-5. Shear stress on the shaft.

Description	**Results** (mm)
Total Deformation	21758-21766 mm
Equivalent Elastic strain	0.011688 mm
Factor of Safety	0.10132 – 15 mm
Shear stress	255.93 mm

These results were captured in ANSYS work bench.

The total deformation is shown in Figure-2, the equivalent elastic strain is shown in Figure-3, Factor of safety is shown in Figure-4 and the shear stress is shown in Figure-5.

4.2 Modal analysis

It is used for determining the mode shapes i.e.., vibration characteristics and the natural frequencies of a machine structure or component while it is being designed.

The modes are used as easy and an effective way of describing the resonant vibration and majority of the structures can be made to resonant i.e.., under the proper condition, a structure can be made to vibrate with sustained, excessive and oscillatory motion. When the elastic and inertia properties of the material interacts then the resonant vibration occurs which is the major vibration related problems in the machine components or structures. The resonance is the fundamental foe better understanding many of the structural vibration problems hence it necessary to identify and qualify. This can be accomplished by defining the structures model parameter.

Performing the model analysis

1) Build the model.

2) Obtain the solution.

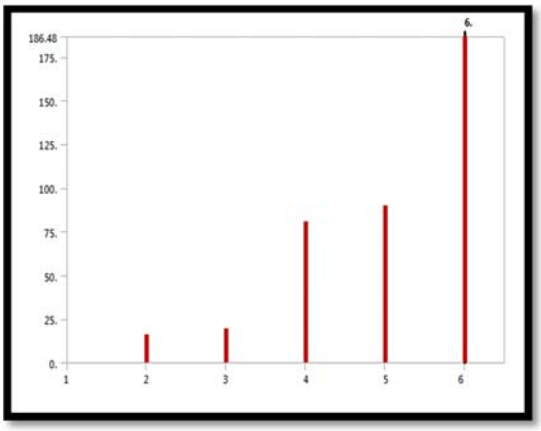
3) Developing of modes.

4) Evaluate of results.

The first six natural frequencies and mode shapes of the model are listed in the table no.1 at the defined boundary condition

Table-1.

Mode	**Frequency** [Hz]
1.	0.19027
2.	16.012
3.	19.687
4.	80.801
5.	89.917
6.	186.48



The flow chart indicates the frequency at each different calculated mode

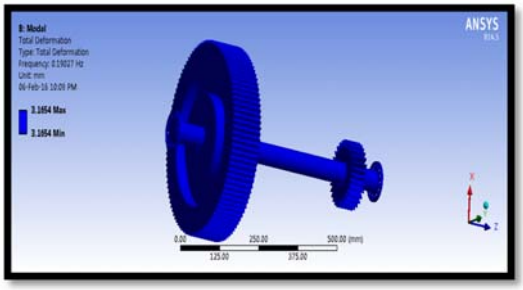


Figure-6. Total deformation of the shaft at 0.19027HZ at node 1.

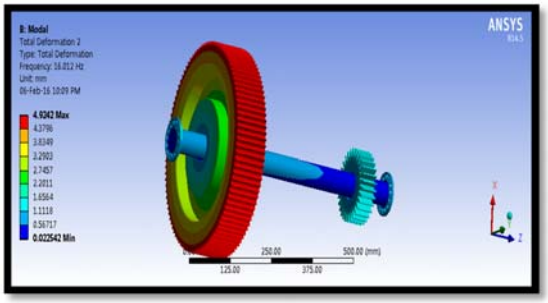


Figure-7. Total deformation of a shaft at 16.012 HZ at node 2.

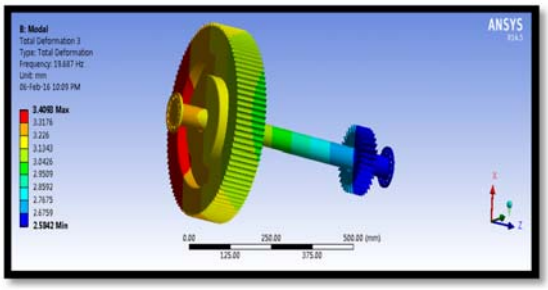


Figure-8. Total deformation of a shaft at 19.687 HZ at node 3.

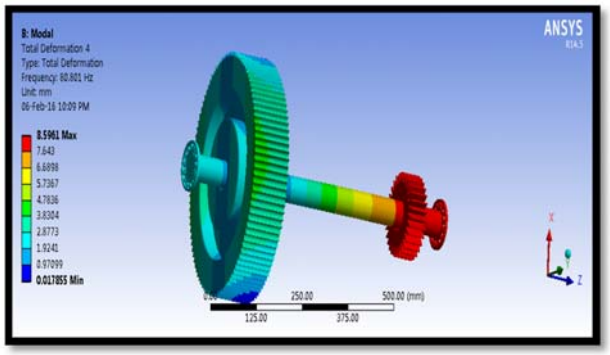


Figure-9. Total deformation of the shaft at 80.801HZ at node 4.

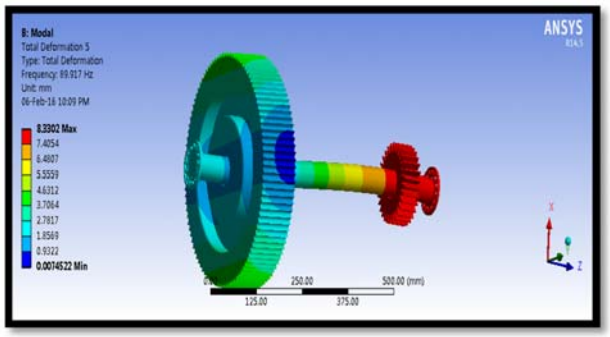


Figure-10. Total deflection of a shaft at 89.917HZ at node 5.

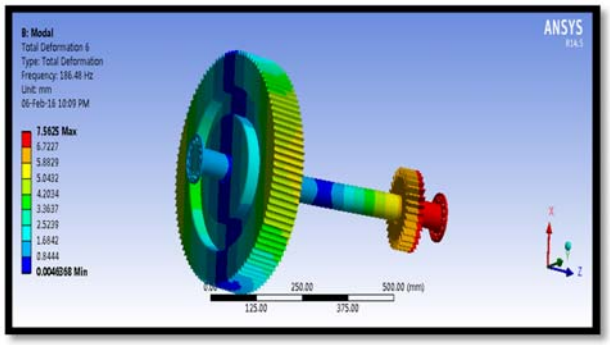


Figure-11. Total deformation of a shaft at 186.48HZ at node 6.

Mode	**Frequency** (HZ)	**Deflection** (mm)
1	0.19027	3.1654
2	16.012	4.9242
3	19.687	3.4093
4	80.801	8.5961
5	89.917	8.3302
6	186.48	7.5625

The above table will clearly shows the deformation of the shaft at each frequency and each node.

5. CONCLUSION

From the study the following conclusions are drawn:

- The shaft and bearing assembly which is used for head stock of the lathe machine is designed and assembled using CREO 2.0 and the analysis are done using ANSYS WORK BENCH V-14.5 software.
- The deformation which is produced in the shaft is mainly due to the bearing since the deformations are too small so the shaft can with stand at long time. We can control the deformation by changing the bearing type.
- According to the static analysis the the load of the bearing will only affect the shaft life and it leads to the deformation.
- The model analysis i.e.., vibration characteristics results revealed that the natural frequency of the setup increases with the increase of number of modes selected the setup tends to fluctuate vigorously with the increase in frequencies.
- As the frequency of vibration tends to increase the deformation of a shaft tends to increase proportionally.

It can be finally concluded that the outcomes through the analysis is found to be in a good agreement and are within the safe limit.

REFERENCES

Modal analysis of a distributed parameter rotating shaft. Vol. 122, Issue 1, 8 April 1988, Pages 119-130.

Hydrodynamic lubrication analysis of journal bearing considering misalignment caused by shaft deformation. Vol. 37, Issue 10, October 2004, pp. 841-848.

Krishnaraj, C., Mohanasundram, K. M. 2012. Design and Implementation Study of Knowledge Based Foundry Total Failure Mode Effects Analysis Technique, European Journal of Scientific Research. 71(2): 298-311.

Modal analysis of drive shaft using FEA, International Journal of Engineering and Management Research, Vol.-3, Issue-1, February 2013.

Design of machine Elements by R.S. Khurmi.

Design of machine Elements by Dr. Vijayaragavan, Dr. Vishnupriyan lakshimi publication.