International Journal of Technical Innovation in Modern Engineering & Science (IJTIMES)
Impact Factor: 3.45 (SJIF-2015), e-ISSN: 2455-2585
Volume 4, Issue 5, May-2018

Finite Element Analysis of a Shaft of PTO drive subjected to a load

Prof. M. K. Khunti[1]
[1]*Mechanical Department, Government Eng., College, Rajkot, mkkhunti@yahoo.co.in*

***Abstract* — The work is mainly concentrated about the analysis of a shaft with the help of a software ANSYS 15.0 workbench. In this the shaft is taken from the PTO shaft gear drive of Field Marshal make mini tractor. In this analysis the shaft is connected with gear and pinion. 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. The results obtained by the stress analysis is found to be good agreement with analytical stress value and modal analysis i.e., stresses and deformation presented are within the limit**

Keywords- Shaft, Gear and Pinion, stress, FEA, CREO.

I. INTRODUCTION

The shaft is one of the major components in the power transmitting system. In PTO drive assembly shaft plays a major role in the transmitting power from engine to PTO shaft. The shaft assembly has a gear and pinion in it so it leads to some of a stresses induced with itself it produces some amount of deviation of a shaft and deformation. Due to the stresses induced in the gears and shaft it leads to failure or reduce of a life of the shaft. As this work is based on finite element analysis, so it is required that a component on which analysis is to be done should have practical application. The component chosen for this purpose is a drive shaft which finds widespread applications in all vehicles. The drive shaft was generated in CREO and imported in ANSYS workbench. The rotation of one shaft about its own axis results in the rotation of the other shaft about its axis. The main objective of this work is to perform the Finite Element Analysis of drive shaft using ANSYS, so as to determine the total deformation and stress distribution in the shaft. The results are compared and verified with available existing results. There is a vast amount of literature related to Finite Element Analysis is present. The literature review presented here considers the major development in implementation of FEA.

II. LITERATURE REVIEW

Rakesh A Oza et al (2014) [1] designed the shaft analytically and analysis work on the ANSYS for comparing the difference such as bending stress, shear stress and deflection of the sharp for existence condition as well as the new design which one developed for this condition. By comparing the both results, according to the maximum stress theory is higher than the principal stress theory. The weight reduction is checked by results and E-Glass/Epoxy is most beneficial then EN-8 and HM Carbon/Epoxy.

P. Jayanaidu et al (2013) [2], studied deal with optimization of drive shaft using the ANSYS. Substitution of Titanium drive shafts over the conventional steel material for drive shaft raises increasing the advantages of design due to its high specific stiffness, stress and law weight. The results obtained from this model is an useful approximation to help in the easier stages of the development, saving development time and helping in the decision making process to optimize design before going into a detailed finite element analysis. The replacement of conventional drive shaft results in reduction in weight of automobile.

D.K. Padhal et al (2013) [3], worked on the frequent failure analysis of output shafts of gear motor used for cold-rolling mill to drive the Pay-off Four-HI. By considering stresses analysis is also carried out with analytical methods and comparing these results with the software results get by ANSYS. After getting all different parameter, redesigned of shaft is done and again analyzing shaft using ANSIS for torsional rigid shaft. From the conclusion, the stress find out by analytical method and by using software both are nearly send and above the stresses of the standard specimen.

Promod J. Bachche et al (2013) [4], observed analysis of shaft behavior in static and dynamic condition for deflection and stress. Results obtained by graphical integration method and finite element analysis are satisfactory and are in working limit for current set of input condition. So we can conclude whatever design made for is safe and shaft will not fail for current working condition.

Goksenli, I.B. Eryurek et al (2009) [5], described the Failure analysis of an elevator drive shaft. In this study failure analysis of an elevator drive shaft is analyzed. By stress analysis, minimum and maximum normal shear stress values occurring at the fracture surface during operation is investigated. Due to too long length of shaft only keyway and fracture region is modelled and analysed. At first forces and torque acting on the shaft are determined. To examine stress distribution at the keyway and fracture surface, finite element method was applied. After visual investigation of the fracture surface it is concluded that fracture occurred due to torsional-bending fatigue. Fatigue crack initiated at the keyway edge. Considering elevator and driving systems, forces and torques acting on the shaft are determined; stresses occurring at the failure surface are calculated. Stress analysis is also carried out by using finite element method (FEM) and the results are compared with the calculated values. Endurance limit and fatigue safety factor is calculated, fatigue cycle analysis of the shaft is estimated. Reason for failure is investigated and concluded that fracture occurred due to faulty design or manufacturing of the keyway (low radius of curvature at keyway corner, causing high notch effect).

Sumit P.Raut, Laukik P.Raut et al (2014) [6] reviewed the various methodology used for failure analysis of the shaft used in different application by various authors in this paper. This paper presents the comparison of the different methodology used, their application and limitation by various authors. The objective of present work is to study the various methodologies used for the shaft failure analysis and to choose best methodology suitable for the failure analysis of shaft used in gear box which is mounted on the overhead crane to prevent repetitive failure. Shaft failure leads to heavy loss due to stoppage and repairing cost associate with the breakdown.

III. MODELING OF SHAFT

The PTO drive used for power or torque transmission. The model of PTO drive shaft is design using Creo Software.

Creo is software which is used for creation and modifications of the objects. In Creo design and modeling feature is available. Design means the process of creating a new object or modifying the existing one. Drafting means the representation or idea of the object. Modeling means create and converting 2D to 3D. By using Creo software, create the model of the PTO drive.

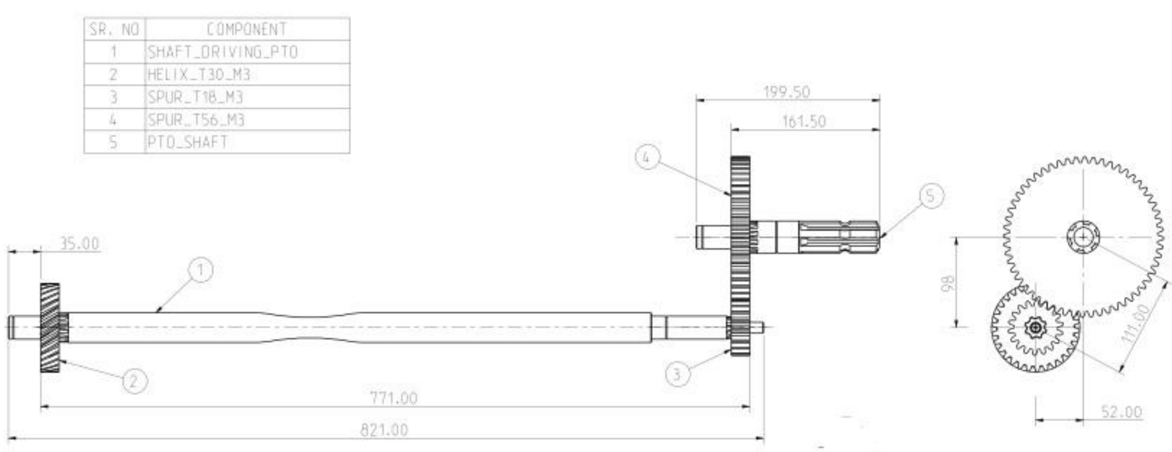


Fig. 1 PTO Drive Assembly

Power of Gear motor = 16.5. HP =12.10 KW

Therefore Power of shaft P= $\frac{2\pi NT}{60}$ $\quad \therefore T = \frac{60P}{2\pi N}$

$\therefore T = \frac{60*12.10*\ 10^3}{2\pi*1686}$ = 68.53N

rpm of the gear ng = 540 rpm of pinion np = 1686

TABLE 1
MATERIAL PROPERTY OF EN 8

Properties	EN 8
Tensile Ultimate	650 MPa
Yield Stress	465 MPa
Young's Modulus	2e5
Poisson Ratio	0.3
Density	7870 Kg/m^3

Analytically check the failure of shaft under twisting moment for material EN8 by using equation, $T_{max} = \pi/16 \times \tau \times d^3$ maximum stress induced is τ = 17.189 MPa.

IV. CREO Model

To determine maximum stress during the transmission of torque of 68.53Nm by EN8 using finite element analysis we sketched and modeled drive shaft in the CREO.

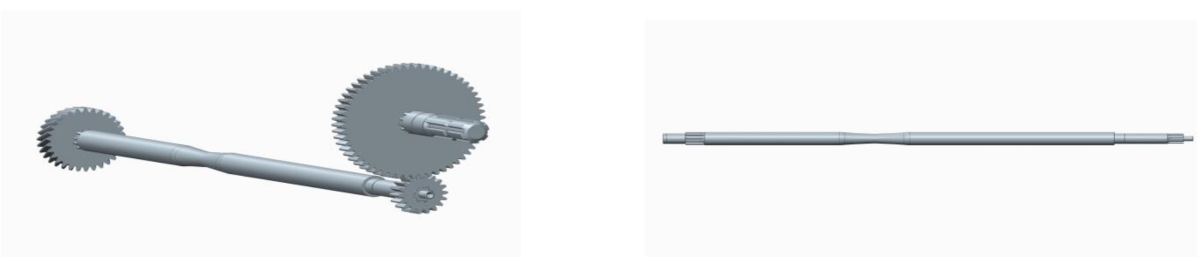
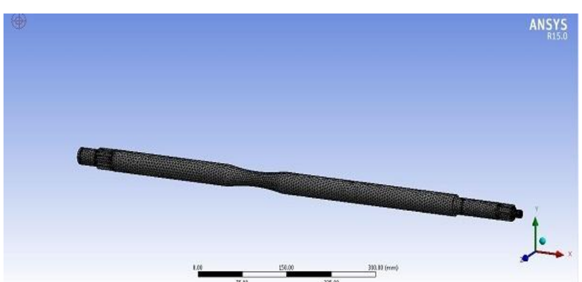
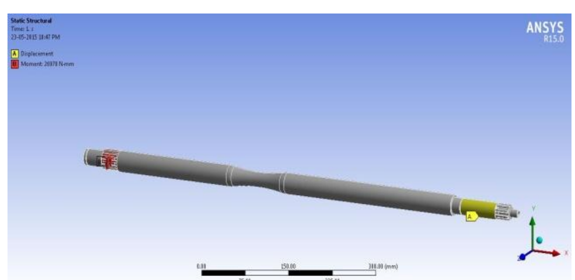


Fig 2. CREO Model - Assembly of PTO Drive

4.1 Meshing and Boundary Condition

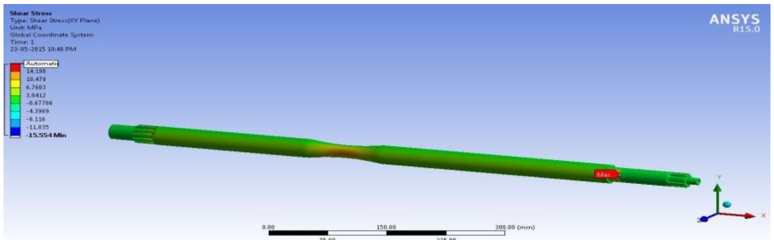




Nodes	115412
Elements	73642

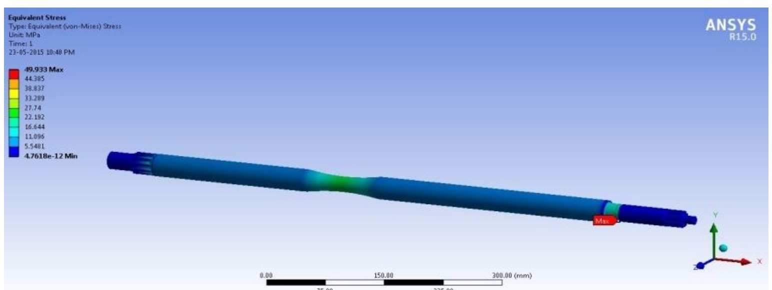
V. FINITE ELEMENT ANALYSIS RESULT

- Maximum Shear Stress



Maximum shear stress is 17.17 MPa.

- Maximum Von-Mises Stress



Maximum Von mises stress is 49.933 MPa.

- Total Deformation

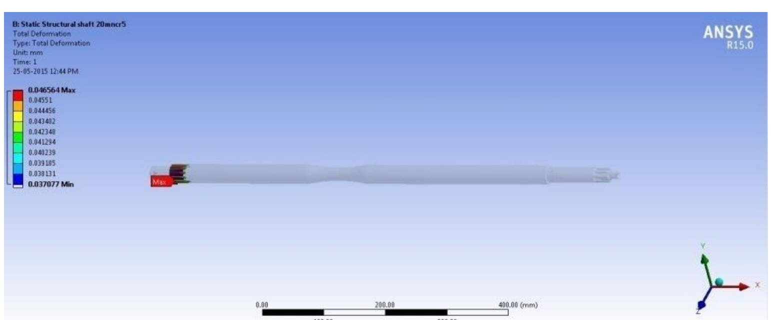


TABLE 2

RESULT COMPARISION

Material	Shear Stress Analytical Results (MPa)	Shear Stress ANSYS Results (MPa)
EN 8	17.189	17.17

VI. RESULTS AND DISCUSSIONS

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 loads. FEA Modeling 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 modeling of a shaft assembly.The designed model of PTO drive shaft is applied on to FEA software ANSYS 15. Analysis results were compared and confirmed by the theoretical calculated data. According to those results we can draw the conclusion; design is safe and it was found out that the numerically obtained values of stress distribution were in good agreement with the theoretical results.

VII. CONCLUSION

In this paper, the analysis of the PTO drive shaft and the use of software Ansys Workbench in the analysis of the same were shown. The maximum stress point and failure areas are found by the deformation analysis of drive shaft. After the analysis, from the table with the results, it can be concluded that the outcomes through the analysis is found to be in a good agreement and are within the safe limit. So we can conclude whatever design made for is safe and shaft will not fail for current working condition. The results provide a theoretical basis to optimize the design and fatigue life calculation

REFERENCES

[1] Mr. Rakesh A Oza, S P Patel ,"International Journal of Engineering Research and Applications (IJERA)" ,Vol.1, Issue 2, pp.055-058

[2] P. Jayanaidu, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 10, Issue 2 (Nov. - Dec. 2013), PP 43-46

[3] D. K. Padhal, Research Inventy: International Journal Of Engineering And SciencVol.3, Issue 4 (July 2013), PP 17-24

[4] Prom J. Bachche, et al., IOSR Journal of mechanical and civil engg. Volume-7, (July August 2013)

[5] Goksenli, I.B. Eryurek ,Article *in* Engineering Failure Analysis 16(4):1011-1019 · June 2009

[6] Sumit P.Raut, Laukik P.Raut," International Journal of Engineering Research and General Science Volume 2, Issue 2, Feb-Mar 2014"

[7] Ravikant, Gopal Krishan, Mukesh Didwania," International Journal of Engineering and Management Research", Vol.-3, Issue-1, February 2013,ISSN No.: 2250-0758,Pages: 4-7

[8] Design of machine elements, V.B. Bhandari, The TATA Mcgraw-Hill

[9] The Textbook of Machine Design, R. S. Khurmi & J K Gupta, S. Chand Publication