Scientific Journal of Impact Factor (SJIF): 5.71

e-ISSN (O): 2348-4470
p-ISSN (P): 2348-6406

International Journal of Advance Engineering and Research Development

Volume 5, Issue 04, April -2018

STRESS ANALYSIS OF SHAFT OF PTO DRIVE USING FINITE ELEMENT ANALYSIS

Prof. M. K. Khunti[1]

[1]*Mechanical Department, Government Engg. College, Rajkot,*

Abstract —*In this paper analysis has been carried out to investigate stresses and deformation for the manufacturing the drive shaft of PTO shaft of Field marshal mini tractor. The shaft is connected with gear and pinion. This is the major important component to be taken into account while designing. The stress concentration and deformation in the shaft in the different regions were analyzed. In this paper, the part files and assembly are done by using CREO software and the analyzing are done by using a ANSYS 15.0 workbench. 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 which helpful for pre manufacturing analysis.*

Keywords *- Shaft, Gear and Pinion, stresses, ANSYS.*

I. INTRODUCTION

A shaft is a part that is used in vehicle to transmit a power from an engine to other drive. 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 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 rotation of one shaft about its own axis results in the rotation of the other shaft about its axis. . 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. Finite element analysis is a computer based analysis technique for calculating the strength and behavior of structures. In the FEM the structure is represented as finite elements. These elements are joined at particular points which are called as nodes. The FEA is used to calculate the deflection, stresses, strains temperature, buckling behavior of the member. In our project FEA is carried out by using the ANSYS 15.0. A static analysis can however include steady inertia loads such as gravity, spinning and time varying loads. In static analysis loading and response conditions are assumed, that is the loads and the structure responses are assumed to vary slowly with respect to time. The drive shaft was generated in CREO and imported in ANSYS workbench. The main objective of this work is to perform the Finite Element Analysis of drive shaft using ANSYS, so as pre 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 presented.

1.1 Objective of the work

- Build a model of a shaft assembly in CREO.
- 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

1.2 Modeling and Finite Element Analysis of a shaft

The 3-D solid model of the shaft and its component was built using CREO software. ANSYS workbench so used for pre-processing, solving and post processing. Material property of the shaft assembly is coincided a 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 in the that assembly using ANSYS software.

The various parts in the assembly are,

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

II. LITERATURE REVIEW

K. Sathishkumar and N. Ugesh et al (2016) [1] 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.

Ravikant, Gopal Krishan, Mukesh Didwania et al (2013) [2] The model of drive shaft has been generated in Solidworks and then impored in ANSYS workbench. In this work finite element analysis of a drive shaft has been taken as a case study. In the present work the modal analysis of a drive shaft has been carried out the inherent frequencies and vibration mode shapes with their respective deformation. The maximum stress point and dangerous areas are found by the deformation analysis of drive shaft. The relationship between the frequency and the vibration modal is explained by the modal analysis of drive shaft.

Sumit P.Raut, Laukik P.Raut et al (2014) [3] 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.

Rakesh A Oza et al (2014) [4] designed the shaft analytically and analysis work on the ANSIS 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) [5], studied deal with optimization of drive shaft using the ANSIS. 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) [6], 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 ANSIS. 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) [7], 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.

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.

Power of Gear motor = 16.5. HP =12.10 KW

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

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

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

Table 1. Material Properties of 20MnCr5

Properties	20MnCr5
Tensile Ultimate	1000MPa
Yield Stress	680MPa
Young's Modulus	2.1e5
Poisson Ratio	0.3
Density	7850 Kg/m^3

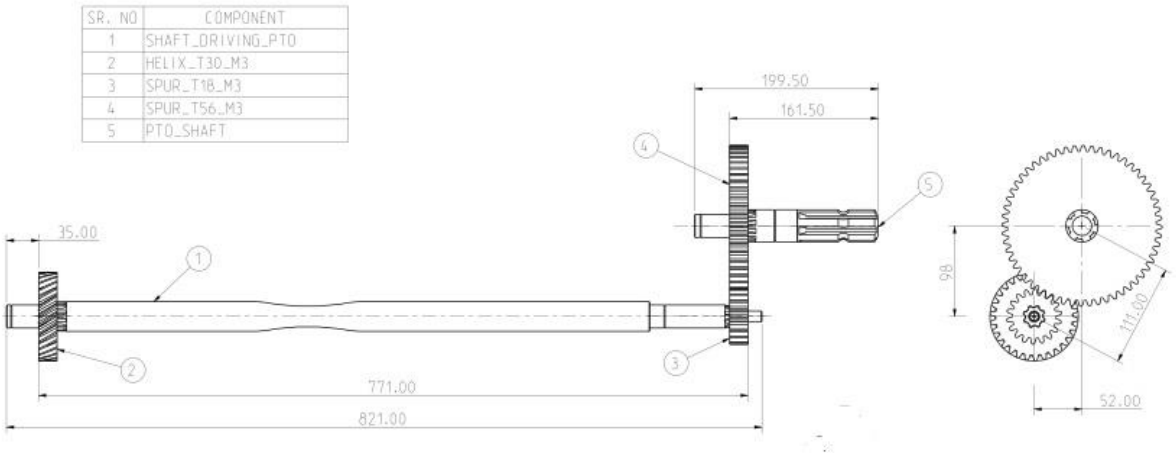


Fig. 1 PTO Drive assembly

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 $\tau = 17.17$ MPa.

III. 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.

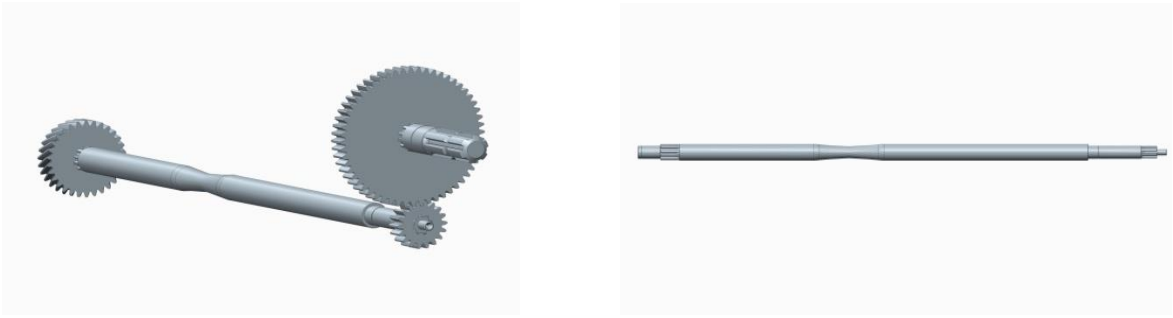
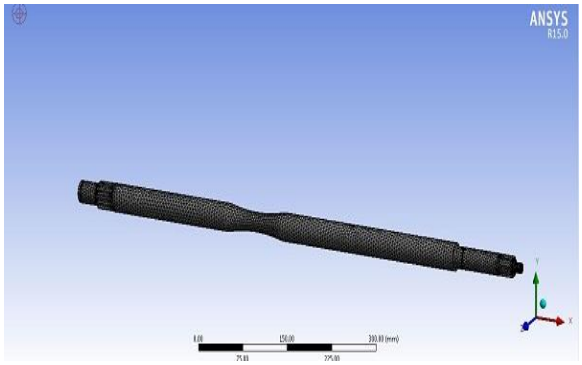
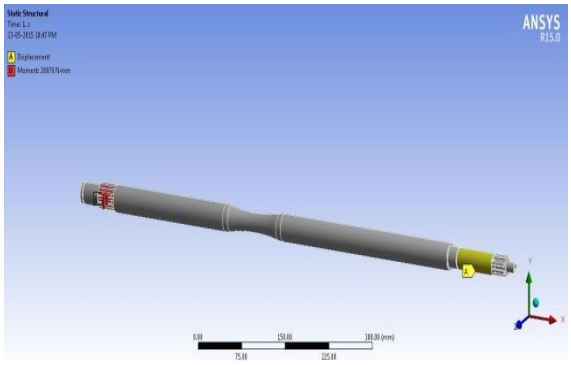


Fig 2. CREO Model - Assembly of PTO Drive and shaft

3.1 Meshing and Boundary Condition

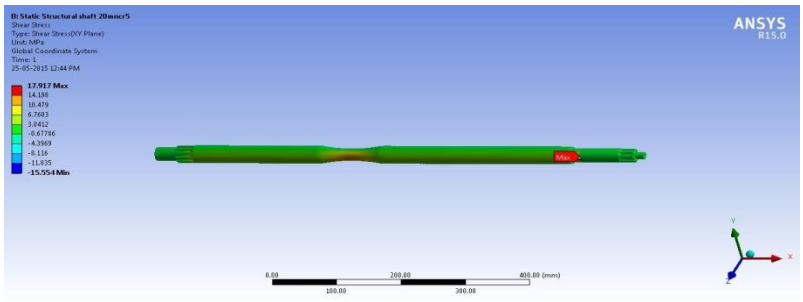




Nodes	115412
Elements	73642

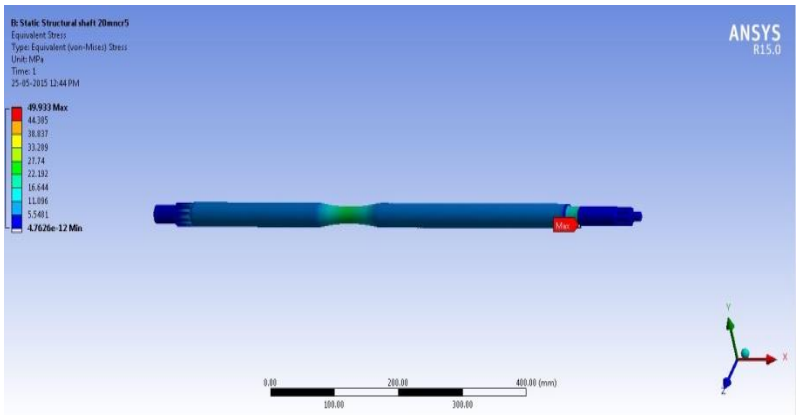
IV. FINITE ELEMENT ANALYSIS RESULT

- Maximum Shear Stress



Maximum shear stress is 17.917 MPa.

- Maximum Von-Mises Stress



Maximum Von mises stress is 49.933 MPa.

- Total Deformation

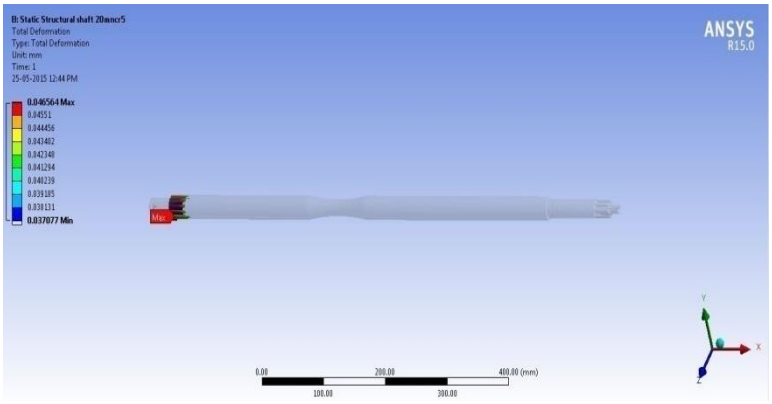


Table 2 Result comparision

Material	Shear Stress Analytical Results (MPa)	Shear Stress ANSYS Results (MPa)
20MnCr5	17.17	17.917

IV. RESULTS AND DISCUSSIONS

FEA helps the designers to predict the Design parameters and necessary changes in the Design modification of the parameters before manufacturing the product which leads to ultimate reduction in cost and time. Initially we don't know the displacement and other quantities like strains, stresses which are then known from 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 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 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.

V. 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 deformation areas are found by the deformation analysis of drive shaft. The results provide a theoretical basis to optimize the design and fatigue life calculation. 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 load condition.

REFERENCES

[1] K. Sathishkumar and N. Ugesh, "ARPN Journal of Engineering and Applied Sciences", VOL. 11, NO. 9, MAY 2016, ISSN 1819-6608, PP 5996-6000

[2] 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

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

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

[5] 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

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

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

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

[9] Johny James .S, Amarnath.P, Santhosham.M, A.Haiter Lenin, International Journal of Applied Engineering Research, ISSN 0973-4562 Vol. 10 No.57 (2015)

[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

