Finite Element Analysis and Weight Reduction of Universal Joint using CAE Tools

Sunil Chaudhry[1], PG Student
Anit Bansal[2], Assistant Professor
[1,2]Department of Mechanical Engineering,
JCDMCOE, Sirsa, India

Gopal Krishan[3],
Assitant Professor
[3]Department of Mechanical Engineering,
AMITY University, Gurgaon, India

***Abstract*—the present work describes the stress distribution of the universal joint using finite element analysis. The finite element analysis is performed using computer aided engineering (CAE) software. The main objective of this dissertation is to investigate and analyze the stress distribution of universal yoke at the real engine condition during power transmit and reduction of weight by modifying the dimensions. The yoke is implemented in the Automobile named Tata 407. Despite all the stress in the yoke is not damaged due to high tensile strength but it may fail under fatigue loading. Thus, it is important to determine the critical area of concentrated stress for appropriate modification. The model is designed in CATIA software and the finite element analysis performed using ANSYS Workbench. The main objective of the work is to reduce the weight and cost of the component.**

Keywords— Universal Joint, FEA, CATIA V5, ANSYS

I. INTRODUCTION

A universal joint is a positive, mechanical connection between rotating shafts, which are usually not parallel, but intersecting. They are used to transmit motion, power or both. The simplest and most common type is called the Cardan Joint or Hooke Joint. It is shown in Figure 1. It consists of two yokes, one on each shaft, connected by a cross-shaped intermediate member called the Spider. The angle between the two shafts is called the operating angle. Good design practice calls for low operating angles, often less than 25°, depending on the application.

In 1545, Italian mathematician Girolamo Cardano theorized that the principle of gimbals could be used to transmit rotary motion through an angled connection. Some credit Cardano with the invention of the universal joint, but it wasn't until the next century that an actual universal joint was produced. English scientist Robert Hooke (1635-1703) was the first to put the Universal joint to work.

Three types of Universal Joints are commonly used:

1. Cross or Spider Joint (variable velocity joint).
2. Ball and Trunnion Joint (variable velocity joint).
3. Constant velocity joints.

The power transmission system of vehicles consists of several components which encounter unfortunate failures [3]. These failures may be attributed to material faults, material processing faults, manufacturing and design faults, etc. Part modeling and analysis is conducted using the Finite Element Analysis package ANSYS and optimization is implemented using MATLAB [2]. The power transmission mechanism consists of one drive body and one driven body, six guide arms, and three connecting arms. The intersecting angle between the input body and the output body that are coupled to input and output shaft can be varied up to 100 degrees while the velocity ratio between the two shafts remains constant [4]. Circumferential stress is applied at the yoke slot and also on the hub and simulated separately [5]. The effect of temperature rise due to friction at the yoke slot, thermal load is gradually increased at the slot.

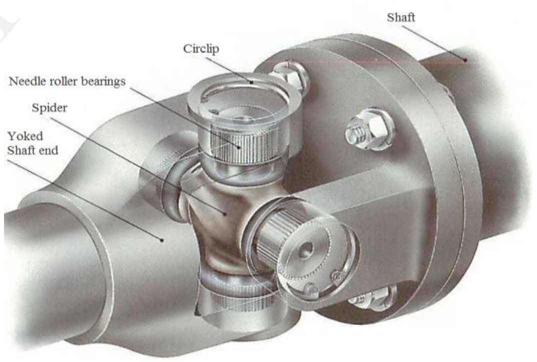

Fig. 1 Universal Joint

II. METHODOLOGY

The problem for the analysis of universal joint was taken from the paper [1]. The universal joint is designed to work for the speed of 2500 rpm and for the Moment of 38 Nm. The vehicle traveled about to be 2 X 10^5 Km which mean more than a 10^8 cycle of loading. For this study, the model was replicated from the above mentioned paper.

The loading conditions of torsional moment and the rotational speed is kept the same. The model of Universal joint was analysed in ANSYS considering the component to be made up of structural steel, which is a material in the low alloy steel group. Structural steel has Tensile Yield strength of 250 MPa and Tensile Ultimate strength of 460 MPa. The CAD model of universal joint was designed in CATIA software and analysed in ANSYS workbench. However, there were some changes made in the geometry of the existing model.

III. MODIFICATIONS

The first step is to design a CAD model of universal joint in CATIA V5. In this work, the thickness of prongs is reduced by 1mm and fillet of 2mm is provided at the sharp edge of the yoke from the existing drawing. The fig. 2 and fig. 3 shows the 2-D drawing of upper yoke and lower yoke respectively after modification. CAD model of each part of universal joint such as upper yoke, lower yoke and spider is generated using CATIA V5 software. Then these parts are assembled in CATIA V5 software. The fig. 4 shows the assembled model of universal joint in CATIA software.

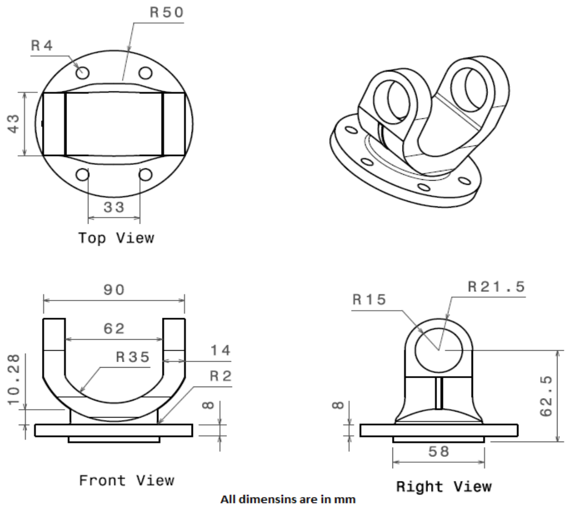

Fig. 2 Drawing of Upper Yoke

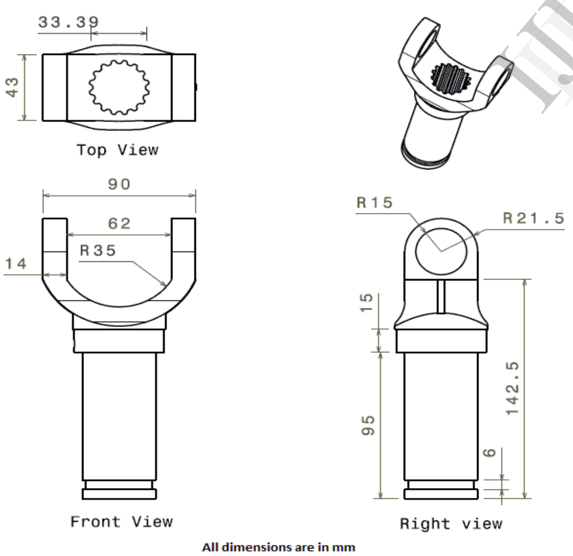

Fig. 3 Drawing of Lower Yoke

After generating the CAD models and assembly in CATIA V5, it is saved in igs format and this igs file is imported in Solid Works software. Then this imported file is saved in the parasolid (x_t) format to avoid the data loss. And finally the model imported in ANSYS Workbench.

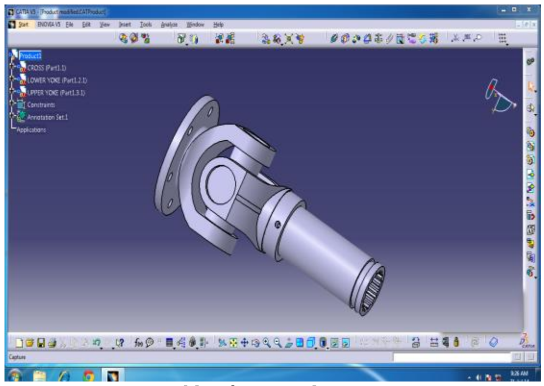

Fig. 4 Assembly of universal joint in CATIA

IV. UNIVERSAL JOINT ANALYSIS

After completion of the Finite Element Model, boundary condition and loads are applied. User can define constraints and loads in various ways. This helps the user to keep track of load cases. The boundary condition is the collection of different forces, supports, constraints and any other condition required for complete analysis.

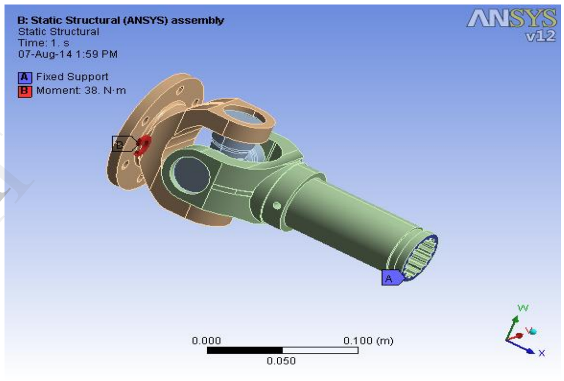

Fig. 5 Applied constraints on the Universal joint

Applying boundary condition is one of the most typical processes of analysis. A special care is required while assigning loads and constraints to the elements.

As per the loading condition, apply the moment of 38 Nm about Z-axis to Upper Yoke. Keep the Lower yoke in fixed condition. Fig. 5 shows the constraints applied on the universal joint in ANSYS workbench. The FEA results of the universal joint have been analyzed and compared with the available results for validation. The color shown in the universal joint represents the stresses and deformation present in the element. Red color shows the maximum stress & maximum deformation, whereas, blue color shows the minimum stress and minimum deformation in the figure. Fig. 6 shows the total deformation in the universal joint (assembly) for a torsion moment of 38 N-m. The maximum deformation is found to be 2.1906e -5m.

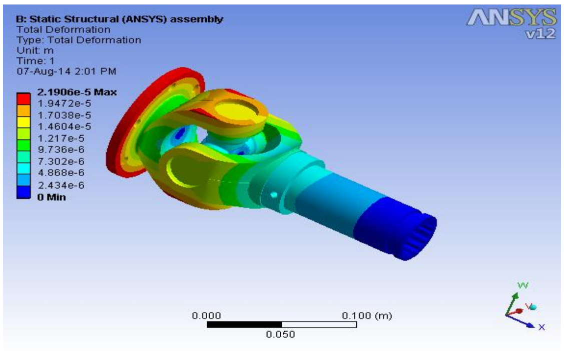

Fig. 6 Total Deformation in the universal joint

V. JOINT YOKE ANALYSIS

The joint yoke is analysed for the maximum principal stress, maximum shear stress, total deformation and maximum principal strain for a torsion moment of 38 Nm. Fig. 7 shows the maximum principal stress in the joint yoke for the torsional moment of 38 Nm.

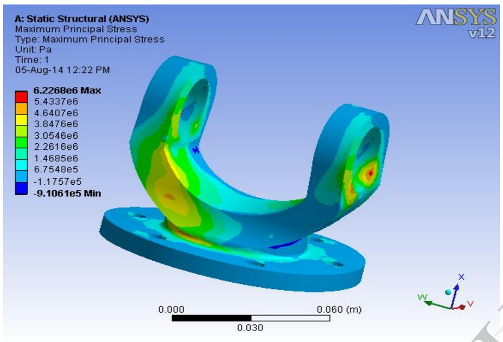

Fig. 7 Maximum principal stress

The analysis results of joint yoke after optimization of the existing model are shown in table below.

Table: FEA results of joint yoke

Sr. No.	Parameters	Existing Results	FEA Results
1.	Maximum Principal Stress	8.89 e6 Pa	6.22 e6 Pa
2.	Maximum shear stress	5.40 e6 Pa	5.87 e6 Pa
3.	Total deformation	1.42 e-5 m	0.35 e-5 m
4.	Maximum principal strain	6.66 e-5	3.39 e-5

The ANSYS analysis shows that the maximum principle stress value obtained is 6.226 e6 Pa, which is within the safety limits as prescribed in the paper. Moreover, this value is also less than the maximum stress value in the original component. Thus we can conclude that the new modeled component is well within the safe limit. For the design dimensions utilized in the analysis, the mass of the joint yoke obtained is about 1.265 Kg. Hence, weight of the joint yoke is reduced about 9.64%.

VI. CONCLUSION

Finite Element Analysis of the universal joint has been done using ANSYS Workbench. From the results obtained from FE Analysis, many discussions have been made. The results obtained are well in agreement with the available existing results. The model presented here, is well safe and under permissible limit of stresses.

1. On the basis of the current work, it is concluded that the design parameters of the yoke with modification give sufficient improvement in the existing results.
2. The weight of the yoke is also reduced by 9.64 %, thereby reducing the cost of the material.
3. The stress is found maximum near the sharp edges.

REFERENCES

[1] Vishal Rathi and Nitin K. Mandavgade (2009) "Fem analysis of universal joint of Tata 407" Second International Conference on Emerging Trends in Engineering and Technology, ICETET-09, ISBN: 978-0-7695-3884-6, PP. 98-103.

[2] Nick Cristello, and Yong Kim (2006) "Design optimization of an automotive universal joint Considering manufacturing cost", 17th IASTED International Conference Modeling and Simulation, ISSN: 530-091 , PP. 499– 504.

[3] H. Bayrakceken, S. Tasgetiren, I. Yavuz (2007) "Two cases of failure in the power transmission system on vehicles: A universal joint yoke and a drive shaft", Engineering Failure Analysis, Vol. 14, ISSN: 1350-6307, PP. 716- 724.

[4] Majid Yaghoubi, Ali Jafary, Seyed Saeid Mohtasebi (2010) "Design, simulation and evaluation of a new universal joint with intersecting angle up to 100 degrees for farm machineries", AJAE 1(4), ISSN:1836-9448, PP: 149-152.

[5] Muhammad Ziaur Rahman, Debasish Adhikary & Tazmin Rashid Mumu (2013) "Failure Analysis of a Universal Coupling using Finite Element Method", Global Journal of Researches in Engineering Mechanical and Mechanics Engineering. Volume 13 Issue 9, ISSN: 2249-4596, PP. 11-15.