FAILURE ANALYSIS AND OPTIMIZATION OF UNIVERSAL JOINT YOKE SUBJECTED BY TORSION AND SHEAR

Avinash C Vasekar, Ranjitsinha R. Gidde,

1Research scholar, Department of Mechanical Engineering, SVERI's College of Engineering, Gopalpur ,Pandharpur

2Department of Mechanical Engineering, SVERI's College of Engineering, Gopalpur, Pandharpur

-- ----------------------***---

ABSTRACT

Automobile transmission system consist several components which sometimes suffer from different stresses (failures). Yoke assembly are one of the most important part in propeller shaft. This Yoke assembly are always subjected by Torsion and Shear. Yoke assembly are rotating part & sometime suffer from fatigue by application of variable torque. In this study, failure analysis and weight optimization of a universal joint yoke of an automobile power transmission system are carried out. The universal joint consists of two forged-steel yokes flange & tube Yoke cross trunnion hold two Yoke together at right angles to each other.

Key words: Yoke joint, Power transmission system, Analytical modelling, Optimization Torsion & Shear.

I.INTRODUCTION

Power transmission system of vehicles consist several components which sometimes encounter unfortunate failures. Some common reasons for the failures may be manufacturing and design faults, maintenance faults, raw material faults, material processing faults as well as the user originated faults. Spectroscopic analyses, metallographic analyses and hardness tests are done. For the determination of stress conditions at the failed section, stress analysis is also carried out by the finite element method. The common failure types in automobiles are the failures in the transmission system elements which cover 25% of all the automobile failures.

In day-to-day life every aspect is influenced by the work of engineer. The equipments we use, the food we eat, and the vehicles we travel in and many more all are developed with the assistance of design engineering. Traditional design has been done by simple calculation. But with increase in product performance and reliability it is difficult to follow the traditional iterative design procedures. As product performance becomes more important and as designs becomes more complex the simple method have becomes inadequate. To understand the growth and its implication for design, it is necessary to look at how design solutions are implemented. To satisfy the market needs it is necessary to provide a computational capacity along with the creativity of the human being. By adding computer technology to the armory of the designer, the best qualities of the designer can be linked with the best qualities of the computer. Most engineering designs are too complex for traditional approach. For example a structure may have spatially dependent material properties if different materials are used; the geometry may be irregular in some sense or the boundary condition may be complex. In all these examples no solution functions exist and so solutions can be achieved only by resorting to an approximate numerical method. A widely used numerical method for solving structural problems in both industry and academia is "FINITE ELEMENT METHOD [1] studied power transmission system of vehicle consist several components which sometimes encounter unfortunate failures. The fracture analysis of a universal joint yoke and a drive shaft of an automobile power transmission system were carried out. Spectroscopic analyses, metallographic analyses and hardness measurements were carried out for each part. For the determination of stress conditions at the

failed section, stress analysis was also carried out by the finite element method. An experimental and finite-element analysis of universal coupling was carried out with help of ANSYS for different torque condition. [2] studied some common reason for failure of drive shaft and universal joint yoke of automobile power transmission system such that failure may be manufacturing and design faults, material processing fault and user originated faults. The failure analysis was carried out for a universal joint yoke and a drive shaft of an automobile power transmission system. The investigation was carried out for spectroscopic and metallographic analyses and hardness measurement for determination of stress condition at the failure section. investigated about effect of an optimal magnetic yoke configuration maximizing the magnitude flux density and improving flux uniformly in the strips, which will increase the transducer signal output. Topology optimization method was employed to find optimal yoke configuration in this investigation. The series of experiments was performed on an aluminum plate, that experiments were conducted using a grating with and without the designated yoke, when the yoke was used; the signal outputs increased up to 60%. [4] investigated about failure reason of yoke connectors of over head power lines these components were produced by bending steel plate and their subsequent hot dip galvanizing. The performed investigation included metallographic analyses, semi-quantitive chemical microanalyses, micro-hardness, measurements and numerical simulation also observed that failure occured due to several degradation process related to liquid metal assisted cracking which occurred during bending. [5] carried out model analysis of drive shaft using FEA, the inherent frequencies and vibration mode shapes with their respective deformation.In this work finite element analysis of a drive shaft had been taken as a case study. The maximum stress point and dangerous areas were found by the deformation analysis

of drive shaft. The relationship between the frequency and the vibration modal was explained by the modal analysis of drive shaft.The deformation and stress contours were plotted and patterns were studied. The results were compared and verified with available existing results. The optimization of drive shaft also achieves the reduction in the weight of the assembly of universal joint and thus reduction in cost. [6]carried out metallurgical evaluation of aluminum alloy yoke from an aerial platform device and performed chemical and mechanic testing of yoke tube. Studied on design issues associated with fatigue cracking, photo documentation, metallurgical replication etc. and chemical analysis, tensile testing and scanning electron microscopy was done. A metallurgical inspection and evaluation was performed on an aerial platform yoke from a fire-fighting vehicle chassis that had been removed from service

II. SIMULATION

1. Finite element analysis of Universal joint yoke

In ANSYS it's very difficult to model the part with parametric modeling as compared with the available modeling software such as CATIA and Pro-E. To create a 3D model of valve body with all intricate geometric details CATIA software is used. The created 3D model of yoke is as shown in fig 2.1.
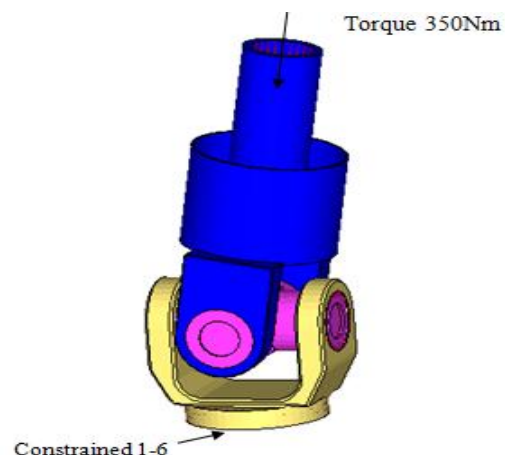

Figure 2.1 – 3D model of yoke

While creating 3D model care has been taken to model it with parametric expression, so as the dimensions changes it will reduce the repetitive time required for modeling. Small steps and champers are eliminated while modeling. The created 3D model is saved in part.igs file format, as this file format is suitable during importing this model for meshing in Hypermesh software. Meshing of the 3D yoke model

In simple term meshing means connecting elements with each other. Elements are the building blocks of the finite element analysis. Meshing is carried out by using Hypermesh software as Hypermesh is dedicated software largely used for meshing.

Meshing is an important step in FEA analysis. The meshing is adequately done to obtain the accurate results while computations. Model is meshed by using SOLID 45 element and with 7 element size. Total **84213** elements and **116154** nodes were created after meshing.

When more the number of elements taken, better accuracy is obtained but simultaneously the computational time increases tremendously.

No of elements	84,213
No of Nodes	1,16,154
Element	C3D10 second order tetra element
Pre Processing	Hypremesh 12
Solver	Abaqus
Post Processing	Hyper view
Analysis	Implicit analysis
Material	M3A452_A23_MS
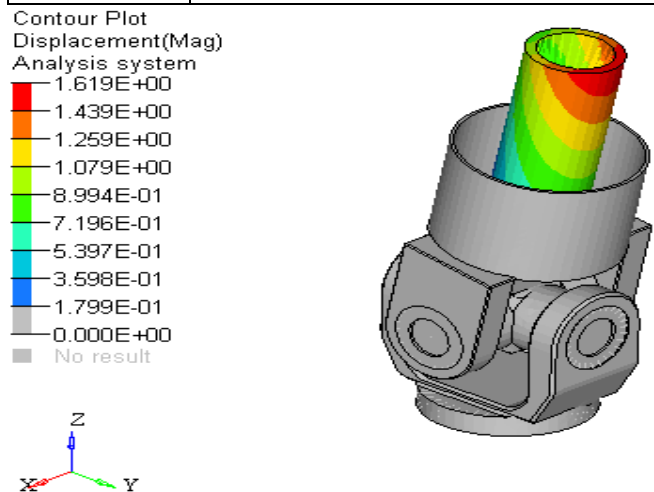
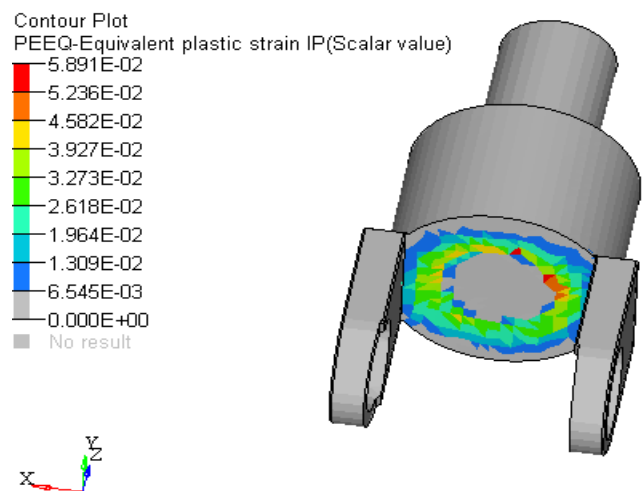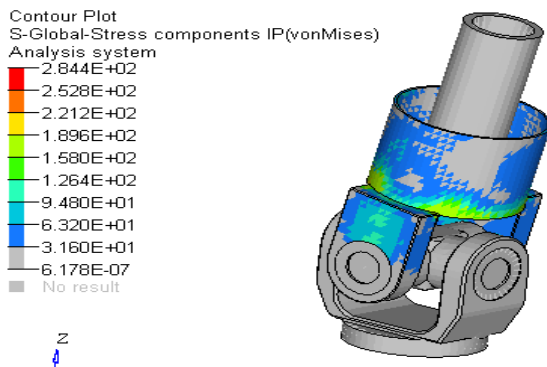**Fig.2.3 Displacement contour**

Figure 2.2 - Meshed model

Fig.2.4 Stress contour

Max Stress is 284 MPa
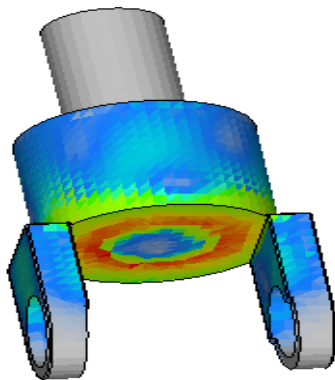
Figure 2.5 Stress Contour

Figure 2.6 Stress Contour

- Transmission Yoke is analysed under 350Nm
- Torque load. Max stress observed 284 MPa (yield 170 MPa).
- Stresses observed in Yoke are more than yield limit.

2. Further scope of work-

Weight optimization Strategy Adopted – Topology Optimization

The aim of topology optimization of transmission Yoke is to find the best use of material for a body to given constraints (deflection, volume or mass reduction). Generally, topology optimization is carried out before shape or size optimizations. Preliminary stage of any static analysis is to decide the loading and boundary conditions; the experimental testing was carried out to determine the stresses acting on yoke assembly for modified part. Topology optimization is, in addition, concerned with the number of connected components/boundaries belonging to the domain. Such methods are needed since typically shape optimization methods work in a subset of allowable shapes which have fixed topological properties, such as having a fixed number of holes in yoke. Topological optimization techniques can then help work around the limitations of pure shape optimization.
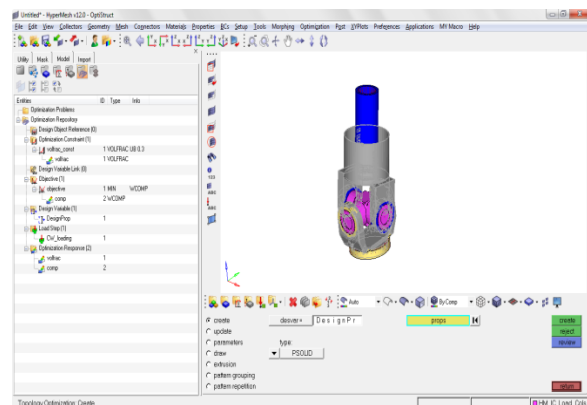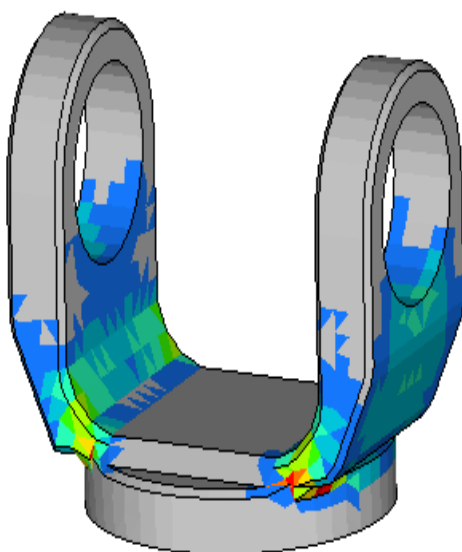
Objective – To reduce stress levels below yield limit.

Response – volume fraction and stresses
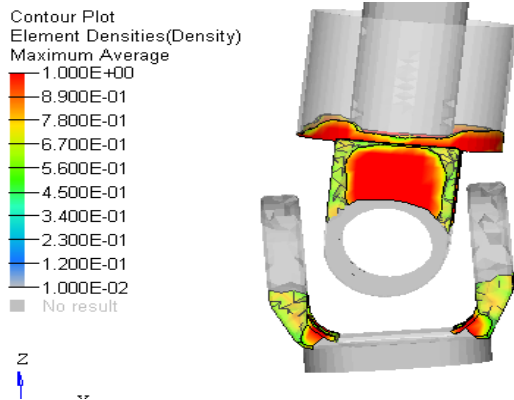
Constraint – Stress limit,30% volume of current volume.

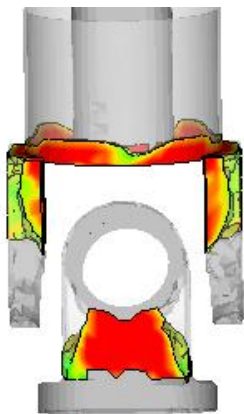**Figure 2.7 Element density distribution**

Figure 2.8 Element density distribution

Gray color region describes dead zone or elements from same region are not contribute to external load. There is a scope of removing material from same region.
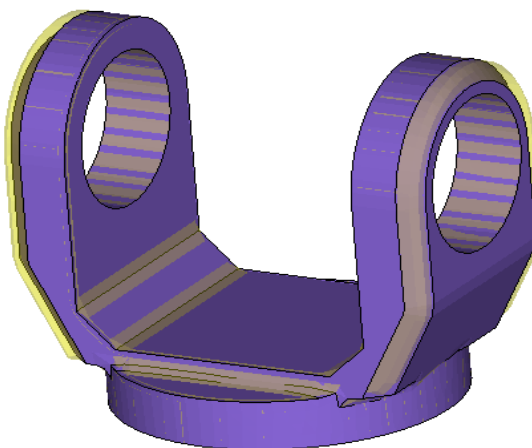

Figure 2.9 Modified geometry

Modified geometry imposed over baseline geometry. Fillets are provided in order to reduce stress concentration.
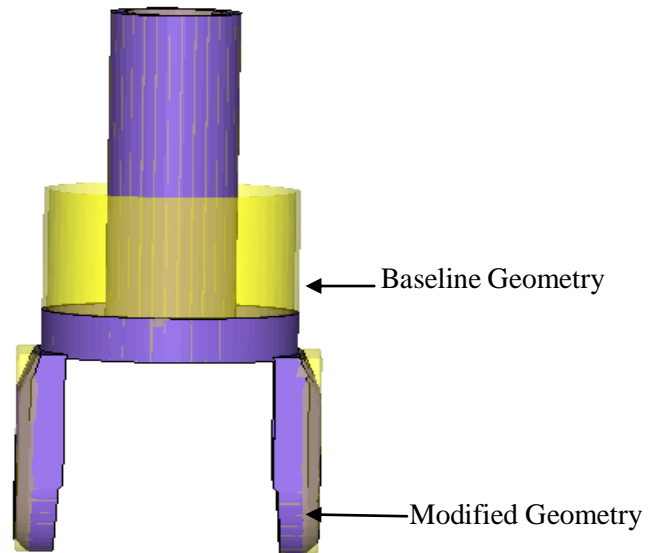

Figure 2.10 Modified geometry

Modified geometry imposed over baseline geometry. Geometry modified in order to reduce stress level with reducing stress levels. Stiffness increase in critical zone.
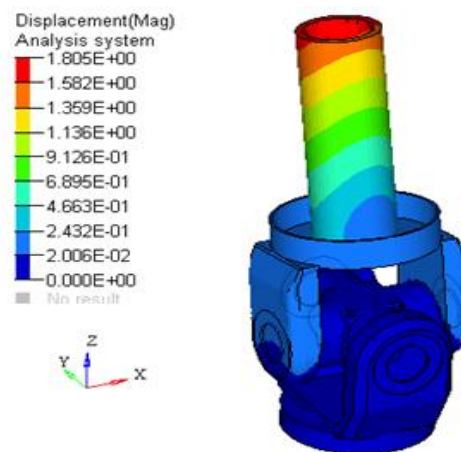

Figure 2.11 Modified geometry- element distribution
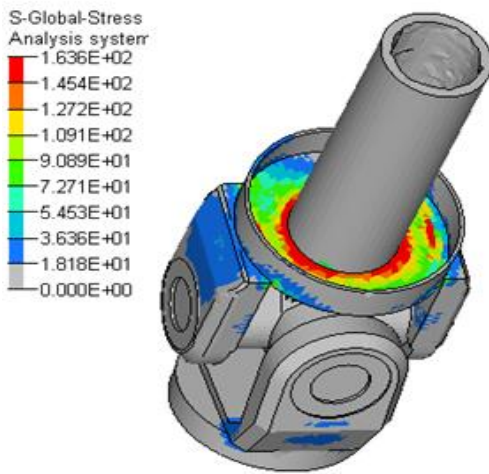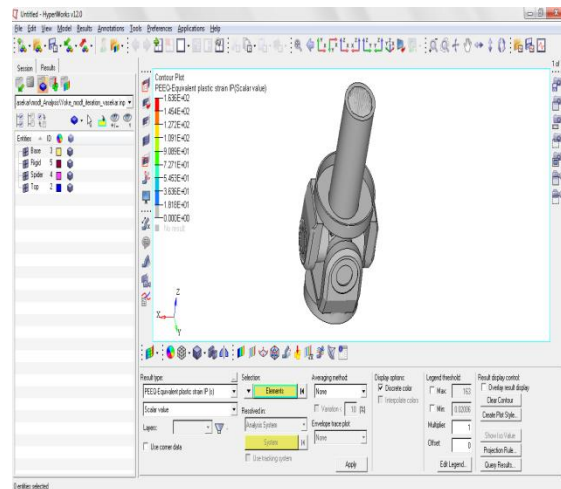
Max deflection 1.8 mm

Figure 2.12 Modified geometry– stress contour

Max stress observed 164 MPa (yield 170 MPa) No Plasticity

3.Experimntal Analysis

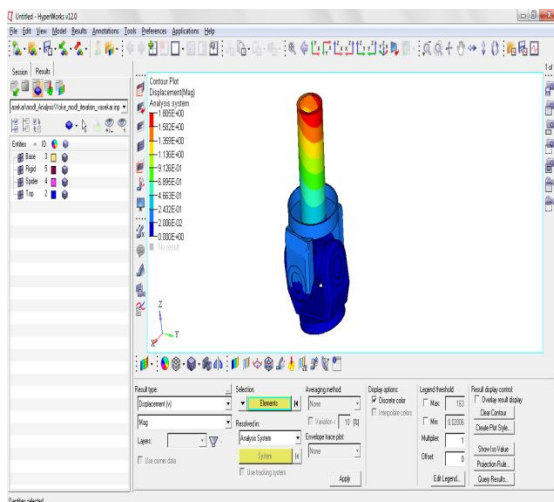Experimental analysis has been carried out as shown in the figure

Fig. 2.13 Experimental set up

Experimental set up   as shown in figure in this experimentation the torque testing machine used to apply torque, from this it observed that the value of maximum stress is 169 Mpa.

Parameter	Stress determined by FEA	Stress determined by Experimentally	Parentage variation in results
Max. stress(Mpa)	164	169	2.96

III.RESULT AND DISCUSSION

In this paper weight optimization has been carried out with help of hyper mesh. Firstly the weight of universal joint was 1.11 kg on which analysis carried out from 169MPa value of maximum stress is obtained. After modification or weight optimization the weight of joint get reduced to 0.967kg that 12.8% reduction in weight .the optimization is carried out in hyper mesh in that the value of maximum stress is 169 Mpa. Experimental analysis has been carried out which is compared hyper mesh to get optimal design or exact value of maximum stress

Table no. 3.1 Weight of universal joint without reducing weight Vs Weight of universal joint after optimization weight

Weight of universal joint without reducing weight	Weight of universal joint after optimization weight
1110gm	967gm

As shown in above table the value optimized. From this optimization the weight reduction is 12.8 %

Table no. 3.2 stress value without optimization Vs stress value with optimization

Stress without optimization(N/mm^2)	Stress with optimization(N/mm^2)
170	164

From the analysis the resultant stress value obtained

IV.CONCLUSION

➢ No plasticity observed after modification.

➢ Weight optimization study carried out in Optistruct.

➢ This study helps to reduce weight of yoke by 12.8 % over original component.

➢ Weight of base geometry is 1.11 kg while modified geometry is 967 gm

V. REFERENCES

1. Siraj Mohammad Ali Sheikh, "Analysis Of Universal Coupling Under Different Torque Condition", International Journal of Engineering Science & Advanced technology,May-June 2012, Volume-2, Issue-3, 690 – 694.

2. H Bayrakceken, S. Tasgetiren, I. Yavuz, "Two cases of failure in the power transmission systemon vehicles: A universal joint yoke and a drive shaft", Engineering Failure Analysis14 (2007)pp.716-724.

3. IkKyu Kim,Woochul Kim, Yoon Young Kim, "Magnetostrictive grating with an optimal yoke for generatinghigh-output frequency-tuned SH waves in a plate" , Sensors and Actuators A137(2007)pp.141-146.

4. Miroslav Dzupon, LadislavFalat, Jan Slota, PavolHvizdos, "Failure analysis of overhead power line yoke connector", Engineering Failure Analysis33(2013)pp.66-74.

5. Ravikant, GopalKrishan, MukeshDidwania"Modal analysis of drive shaft using FEA",International journal of engineering and Management Research, Vol-3, Issue-1, Feb 2013 pp.4-7.

6. Timothy R. Smith, "An evaluation of fatigue cracking in an aerial platform yoke from a five-fighting vehicle", Engineering Failure Analysis 9,March-2003,Page 303-312.