International Journal of Scientific Research and Management Studies (IJSRMS)
ISSN: 23493771 **Volume 2 Issue 3, pg: 136-142**

STRUCTURAL ANALYSIS OF UNIVERSAL JOINT USING FINITE ELEMENT METHODOLOGY

[1]Dhananjay S Kolekar, [2]Abhay M. Kalje, [3]Swapnil S Kulkarni
[1]ME (Mech-Design), [2]Associate Professor & Head of Dept
N.B. Navale Sinhagad College of Engineering, Solapur, Solapur University, India
[3]Associate, Advent Tool Tech, Bhosari, Pune, India

ABSTRACT

Steering Yoke is subjected to torsion loads while manoeuvring the vehicle on a given terrain. The loads may vary depending on the nature of terrain and the speed of the vehicle. While a generalized case shall be taken up for study while pursuing dissertation work on this topic, the findings are expected to throw light on the causes, location and extent of stresses on the parts in the sub-assembly. Typically, the yokes and the web shall be modelled using Finite Element Techniques. Mathematical treatment shall be offered to estimate responses to the significant parameters under observation. The benchmark variant shall be put to test for experimentation for evaluating results towards validation. Design alternatives could deliberate on variants for material and/or geometry. Improvement is sought in stress distribution without radial change in the existing design. The new alternative should be feasible for procurement, development or production upon approval of the design proposed towards conclusion of this study.

KEYWORDS: *Universal joint, steering yoke, FEA*

I. PROBLEM DEFINITION

Manoeuvring the vehicle while driving implies moving the steering wheel and in turns all the components in the sub-assembly. The stresses in either direction, while moving the vehicle to the right or to the left, happen to be a source of failure of the mechanical joint. The two halves of the yoke, the web connecting the two halves or the shaft in the linkages are prone to failure. In such event, the driver could lose control leading to an accident. The steering yoke being a component posing threat to the 'safety' of the vehicle and its occupants, the design of the same needs to be reviewed for encoring structural integrity. An event of failure also leads to secondary and tertiary effects including additional cost for repairs and loss of goodwill among customers.
The design review could look into aspects dealing with the material properties and/or the geometry of the part/s. For this work no radical change is sought in design and the existing design shall be reviewed for feasible alternatives calling for minimal changes in the development or production further.

II. INTRODUCTION

The purpose of a steering system is to control the direction of the vehicle by operating the steering wheel of the steering system. Movement of steering wheel by the driver should cause an accurate response of the road wheels. The intermediate shaft connects the steering shaft to the steering pinion. These components cannot be arranged on the same axis due to the vehicle design limitations. They are arranged with the universal joints. The stresses in either direction, while moving the vehicle to the right or to the left, happen to be a source of failure of the mechanical joint. The two halves of the yoke, the web connecting the two halves or the shaft in the linkages are prone to failure. In such event, the driver could lose control leading to an accident.

A universal joint also known as universal coupling, U joint, Cardan joint, Hardy-Spicer joint, or Hooke's joint is a joint or coupling used to connect rotating shafts that are coplanar, but not coinciding. A universal joint is a positive, mechanical connection used to transmit motion, power or both. Each universal joint assembly consists of three major components: two yokes (flange and weld) and a cross trunnion. An automotive flange yoke has a machined flat face which may be affixed through a bolted connection to the rear differential of a vehicle. A weld yoke incorporates a machined step, and is inserted into the end of the driveshaft and welded in place. The cross trunnion is used to deliver rotation from one yoke to another using four needle pin bearings.

Figure1: Typical Universal Joint

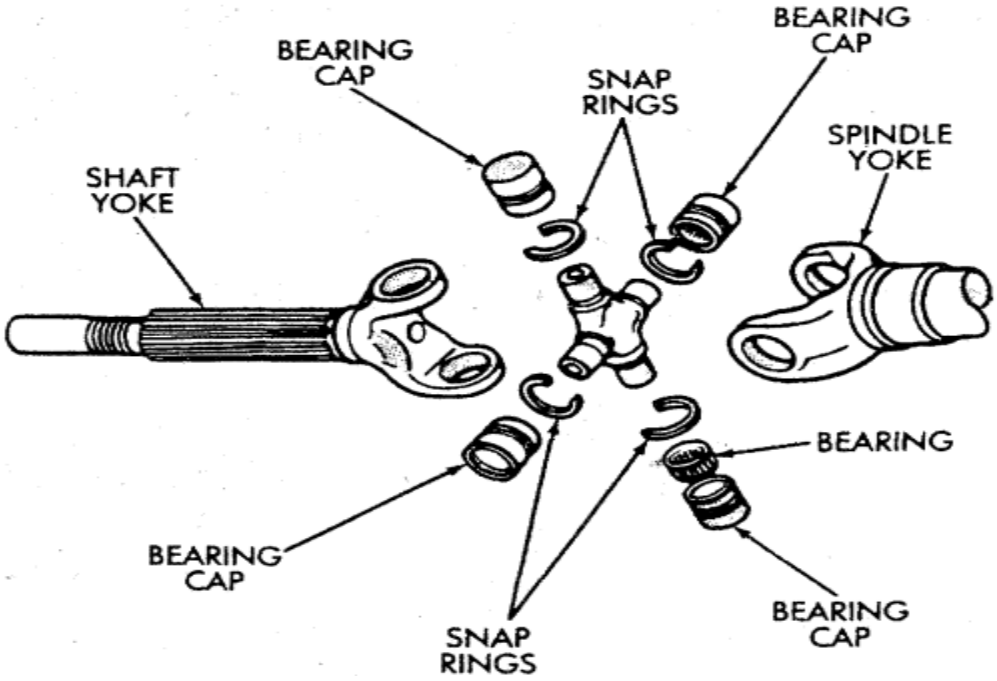


Figure 2: Parts of a Typical Universal Joint

The proposed method utilizes software in the FEA domain for analyzing the effects of the variation in the values of the design parameters influencing the modal behaviour. Also the computational approach will give the results more close to practical values through simulation. The FEM method is used to analyze the stress state of an elastic body with complicated geometry. Also the contact and bending stresses should be calculated by using ANSYS. In this thesis the analysis of characteristics of in volume spur gears in gearbox is intended for the study by using the FEM.

Benefits of using CAE software:

- Design decisions can be made based on their impact on performance.

- Designs can be evaluated and refined using computer simulations rather than physical prototype testing, saving money and time.
- CAE can provide performance insights earlier in the development process, when design changes are less expensive to make.
- CAE helps engineering teams manage risk and understand the performance implications of their designs. Integrated CAE data and process management extends the ability to effectively leverage performance insights and improve designs to a broader community.
- Warranty exposure is reduced by identifying and eliminating potential problems. When properly integrated into product and manufacturing development, CAE can enable earlier problem resolution, which can dramatically reduce the costs associated with the product lifecycle.

III. LITERATURE REVIEW

1.1. S.G.Solanke and A.S.Bharule[1], the purpose of this paper is an Investigation on Stress Distribution for Optimization of Yoke in Universal Joint under Variable Torque Condition. Yoke is analysed under torque load from steering rod observing hot spot location/Stress concentration region, hot area potentially carrying load, scope to optimize/improve part by eliminating/adjusting density in order to maximize area of hot region with reducing in dead region.

1.2 Anup A. Bijagare, P.G. Mehar and V.N. Mujbaile[2], in this study the high strength carbon/epoxy composite drive shaft has been design to replace conventional steel drive shaft of an automobile. A one piece composite drive shaft for rear wheel drive automobiles has been design optimally by using Genetic Algorithm (GA) for high strength carbon/epoxy composites with the objective of minimization of weight of shaft, & analyzed using ANSYS for better torque transmission capacity and bending vibration characteristics.

1.3 S.K.Chandole, M.D.Shende, M.K.Bhavsar[3],in this paper finite element analysis of the component is carried out to find the stress and displacement of the final product. For modeling of the component, CATIA V5 R17 software is used. Preprocessing work like meshing and analysis work is carried out in HYPERWORKS software. Using FEA analysis, we can identify the nature and characteristics of stresses acting on the yoke and also evaluate the influence of the loads/ mass/ geometry/ boundary conditions over the yoke.

1.4.Farzad Vesali, Mohammad Ali Rezvani* and Mohammad Kashfi[4], It is the purpose of this research to study the dynamics of the universal joints and to propose some practical methods for improving their performance. The task is performed by initially deriving the motion equations associated to the universal joints. That is followed by elaborating on the oscillatory behavior in the rotational speed and the torque that transmits through the intermediary shaft. The forces in the joint bearings are calculated by using an analytical method that is also supported by the numerical modeling. Such models are also used in order to calculate the rhythm and the amount of the excess loads on the joint.

1.5 Naik Shashank Giridhar, Sneh Hetawal and Baskar P [5], in this study, analysis is being performed on the universal joint yoke and the propeller shaft. In the universal joint yoke, certain modifications are made in the existing geometry and analyzed for the identical loading and boundary conditions as in the reference paper from which the problem has been taken. In case of propeller shaft a comparative study has been made between two shafts differing in their material, keeping in view the possible weight reduction that can be obtained without affecting the functionality of the shaft. Both the components are analyzed in ANSYS and the results are compared.

1.6 Sunil Chaudhry, Anit Bansal, Gopal Krishan[6], 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. 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.

1.7 P.G. Tathe, Prof. D.S. Bajaj, Swapnil S. Kulkarni[7],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.

1.8 S.K. Chandole, M.K. Bhavsar, S.S. Sarode, G.R. Jadhav[8], In this paper structural optimization of the steering yoke is carried out. For modeling of the component, CATIA V5 R17 software is used. In the early stage of concept generation, topology optimization should be used to develop an efficient structure from the beginning. At this level an automatized variation of optimization parameters was proven useful to and the best feasible design possible. In the later stage, shape and size optimization should be used to fine-tune the structure realized from the topology optimization and carried out physical experimentation to validate the model.

1.9 Kamal Kashyap, D.G.Mahto[9], this paper is to present a finite element analysis predicting the behavior of hooks joint under different loads on different parts. The software package ANSYS is used to model the joint. This paper may help to improve the quality of hooks joint.

1.10 H.Bayrakceken, S. Tasgetiren, I. Yavuz[10], in this study, fracture analysis of a universal joint yoke and a drive shaft of an automobile power transmission system are carried out. Spectroscopic analyses, metallographic analyses and hardness measurements are carried out for each part. For the determination of stress conditions at the failed section, stress analyses are also carried out by the finite element method

1.11 S.Kinme, T. Kamikawa, A.Nishino, K. Ikeda, S. Inoue[11], in this development, experimental result, FEM analysis was effectively utilized for repeatedly reviewing the design. As aresult, high rigidity and low cost stamped yoke was developed

IV. SCOPE OF THE WORK

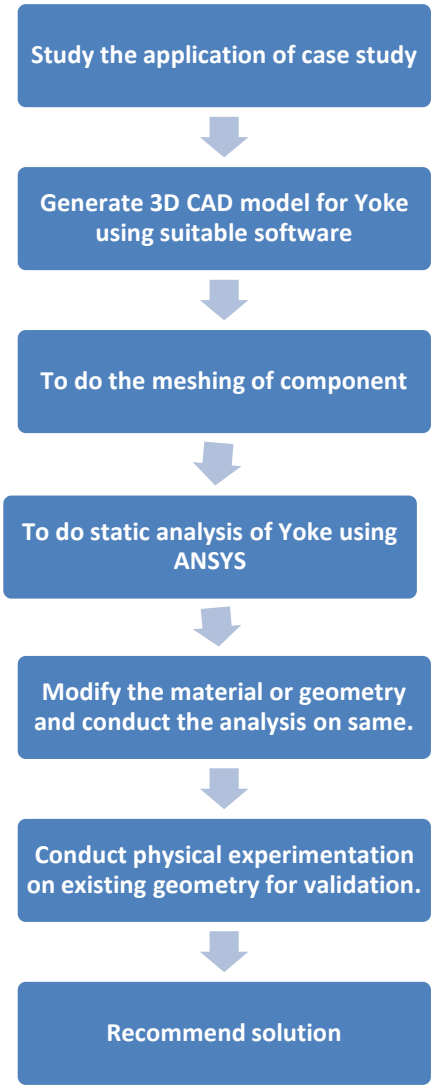
This work consists to develop a computational model to evaluate the design for stresses leading to failure in the yoke. Alternatives shall be proposed for enhancing structural integrity or introduction of new material or geometry for simplification in design or process. The process of validation for this method is sought through the physical Experimentation. The results of Experimental work shall be compared for results with the numerical methodology and vice-versa. The concurrence of the results shall offer validation for this thesis work.

V. OBJECTIVES OF THE STUDY

The following are the objectives of the study

- Study the application of case in consideration.
- Secure 3D Modelling followed by pre-processing (meshing).
- Using Finite Element Methodology, conduct linear static structural analysis over the meshed model.
- Determine design alternatives and analyze for iterations.
- Comparison of analytical results of any one variant with Experimental results for validation.

Proposed Flow Chart and Methodology



Test Setup for Physical Experimentation

Figure 3: Typical Test setup for Physical Experimentation

Figure shows the typical test setup for determining the stresses in the yoke. The gradually increasing torque will be applied and corresponding stress is recorded by using strain gauge.

VI. VALIDATION

The results for this work shall be validated through alternative methodology for Experimentation. An error margin of about 20% could be accommodated in such cases.

VII. FUTURE WORK

The future work consists to develop a computational model. This work consists of developing a computational process to predict the modes of failure in the yoke. 3D model will be secured for processing FE model, Finite element method (FEM) is important numerical technique used in engineering analyses. Usually elements are sub-divided uniformly in FEM to obtain a fair solution with required accuracy. The suitable CAE interface will be deployed for determining analytical results. The process of validation for this method is sought through the physical Experimentation. The results of Experimental work for the existing case study shall be compared for results with the numerical methodology. The concurrence of the results shall offer validation for this thesis work.

VIII. CONCLUSION

The main objective of this work is to take an account of previous work carried out on a Universal joint. From the review, it can be noted that failure of component is occur due manufacturing and design fault, raw material faults, maintains faults, material processing faults, to avoid this problems various techniques such as either, modify material or geometry and conduct analysis on the same and finally implemented in the solution or used to find out the best design of Joint with considering the all the factor such as weight, cost, Fatigue life, stress distribution, stiffness, etc. The investigation of stress in the joint.

REFERENCES

[1] S.G.Solanke and A.S.Bharule, *"An Investigation On Stress Distribution For Optimization Of Yoke In Universal Joint Under Variable Torque Condition"* International Journal of Mechanical Engineering and Robotics Research Vol. 3, No. 2, April 2014 ISSN 2278 – 0149

[2] Anup A. Bijagare, P.G. Mehar and V.N. Mujbaile, *"Design Optimization & Analysis of Drive Shaft"***,**VSRD International Journal of Mechanical, Auto. & Prod. Engg. Vol. 2 (6), 2012.

[3] S.K.Chandole, M.D.Shende, M.K. Bhavsar*" Structural Analysis Of Steering Yoke Of An Automobile For Withstanding Torsion/ Shear Loads",*IJRET: International Journal of Research in Engineering and TechnologyVolume: 03 Issue: 03 | Mar-2014

[4] Farzad Vesali, Mohammad Ali Rezvani* and Mohammad Kashfi, *"Dynamics of universal joints, its failures and some propositions for practically improving its performance and life expectancy"* Journal of Mechanical Science and Technology 26 (8) (2012) 2439~2449

[5] Naik Shashank Giridhar, Sneh Hetawal and Baskar P.,*"Finite Element Analysis of Universal Joint and Propeller Shaft Assembly"*, International Journal of Engineering Trends and Technology (IJETT) - Nov 2013 – Volume Number 5

[6] Sunil Chaudhry, Anit Bansal, Gopal Krishan *"Finite Element Analysis and Weight Reduction of Universal Joint using CAE Tools"* International Journal of Engineering Research & Technology (IJERT) ISSN: 2278-0181 Vol. 3 Issue 10, October- 2014

[7] P.G. Tathe, Prof. D.S. Bajaj and Swapnil S. Kulkarni, "*Failure Analysis And Optimization In Yoke Assembly Subjected By Torsion And Shear*", International Journal of Advanced Engineering Research and Studies E-ISSN2249–8974. July-Sept,2014

[8] S.K. Chandole, M.K. Bhavsar, S.S. Sarode, G.R. Jadhav *"Design Evaluation And Optimization Of Steering Yoke Of An Automobile"* International Journal of Research in Engineering and Technology | pISSN: 2321-7308 Volume: 03 Issue: 11 | Nov-2014,

[9] Kamal Kashyap, D.G.Mahto *"Analysis of Hooks Joint Using Ansys by Von-Mises Method"* International Journal of Engineering and Advanced Technology (IJEAT)ISSN: 2249 – 8958, Volume-3, Issue-3, February 2014

[10] *T.Rangaswamy, et. al "Optimal Design and Analysis of Automotive Composite Drive Shaft", International Symposium of Research Students on Materials Science and Engineering December 2002-04 Chennai India*

[11] H. Bayrakceken, S. TasgetirenI. And Yavuz, *"Two cases of failure in the power transmission systemon vehicles: A universal joint yoke and a drive shaft"*, EngineeringFailure Analysis 14 (2007) 716–724.
[12] S.Kinme, T.Kamikawa, A.Nishino, K. Ikeda and S. Inoue, "*Development of Stamped Yoke for High Rigidity Intermediate Shaft"* Koyo Engineering Journal English Edition 165E (2004)

AUTHOR' S BIOGRAPHY

Dhananjay. S. Kolekar ME (Mech -Design) Passed BE (Mech) from MGM'S college of engineering, S R T Marathwada University, Nanded & pursuing ME (Mech-Design) in N. B. Navale Sinhagad College of Engineering, Solapur, Solapur University.

Abhay M. Kalje. Associate Professor & Head of Dept. ME (Mech-Prod), Experience of 29 years teaching & 02 years industrial Engg Passed B E (Mech) from WCE Sangli, M.E.(Mech) from Shivaji University, Kolhapur. Renowned teacher for the subjects like Engg Graphics, Machine Drawing, Automobile Engg. He is currently a Senior Lecturer & Head of Dept. in department of mechanical engineering, N. B. Navale Sinhgad college of Engineering, Solapur. Areas of interest are Industrial Engg., Energy Engg., Automobile

Swapnil S Kulkarni Associate, Advent Tool Tech India Pvt. Ltd., Pune. The Company offers Engineering Services and Manufacturing Solutions to Automotive OEM's and Tier I and Tier II Companies. He is a Graduate in Industrial Engineering with PG in Operations Management. With around 20 years of working experience in the domain of R&D, Product Design and Tool Engineering, he has executed projects in the Automotive, Medical and Lighting Industry. His area of interest is Research and Development in the Engineering Industry as well as the emerging sector of Renewable Energy.