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

A Review

Pranit Patil[1], Rohan patil[2], Hitesh patil[3], Tushar saste[4], Prof. S.G.Solanke[5]

[1]Student, Saraswati College of Engineering, India, pranitp976@gmail.com
[2]Student, Saraswati College of Engineering, India, patilrohan869@gmail.com
[3]Student, Saraswati College of Engineering, India, patil.hitesh.hp13@gmail.com
[4]Student, Saraswati College of Engineering, India, tushar.saste@gmail.com
[5]Professor, Saraswati College of Engineering, India, solanki.sachin75@gmail.com

Abstract*: The universal joint consists of two forged-steel yokes or forks joined to the two shafts being coupled and situated at right angles to each other The research work deals with an Investigation on Stress Distribution for Optimization of Yoke in Universal Joint under Variable Torque Condition. The analysis will be carried out under steady state conditions using numerical method by applying the boundary condition. Calculation of stress distribution and element density distribution in Yoke to reduce stress concentration in affected region by iteration shape optimization or weight reduction.

Keywords: universal joint, stress distribution, yoke joint, Stress reduction.

1. Introduction

The universal joint consists of two forged-steel yokes joined to the two shafts being coupled and situated at right angles to each other. Comprised of three main component-two Yokes and cross trunnion a universal joint is linkage used to transmit rotational motion from one shaft to another when the axis are coplanar, but not coinciding. 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 highest stresses are occurred at the crack beginning location of the yoke. A possible surface fault could have started the crack propagation period at the highly stressed point of area. After a crack propagation period, the arm had completely fractured.

The research work deals with an Investigation on Stress Distribution for Optimization of Yoke in Universal Joint under Variable Torque Condition. The analysis will be carried out under steady state conditions using FEM by applying the boundary condition and stresses is visualized in the model by experiment method such as photo elasticity method.

2.1 Literature Survey

Gagandeep Singh [1] Represent The present work is about the optimization of a universal joint using structural steel in order to reduce the mass and the stress values its sensitive part using FEM. Mr. S. K. Dhage[2] In this research the focus is on change in geometry of failure yoke at fracture region without changing materials. Weight optimization simultaneously consider in this scope of the research.

Mr. Anuj A. Muley [3] the results obtained by analytical calculation can be used as relevant but those calculations considered general simplified mathematical relationship used. The further complexity of shape with other dimensions can induce the changing in stress values. AbhishekMandal [4] we can easily conclude that the fork pin experiences the maximum compressive stresses and strains as referenced earlier. The region where the fork and the fork pin makes contact experiences generally higher compressive stress and bending stresses. This leads to failure of the transmission system. B.SWATHI, G. RAMESH [5]

Universal joint in a rigid rod that allows the rod to bend in any direction, and is commonly used in shafts that transmit rotary motion Main objectives are to reduce shear failures by Modification of pin (cross) in existing design of universal coupling.

Sang June Oh and John T Woscek[6] The most important point to make from this low, sayθstudy is that if the joint angle approximately well within –6° and +6°, and if is minimal (near zero), the Cardanβthe angle Joint can be adequate.

Dong-Kyun Min, Min-Eung Kim, [7] Explains or revealed that the manufacturing productivity and mechanical properties an united steering yoke largely depends upon the manufacturing process such as forging. The precision cold forging process for the steering yoke of an automobile has been analyzed by using a rigid–plastic finite-element analysis. The rigid–plastic finite-element method for precision cold forging has been used in order to reduce development time and die cost Avinash C Vasekar. [8] In this paper weight optimization has been carried out with help of hyper mesh Experimental analysis has been carried out which is compared to hyper mesh to get optimal design or exact value of maximum stress. S. P. Chaphalkar, Subhash. N. Khetre Arun M. Meshram[9] We studied the analytical behavior of the yoke under the tensile loading condition and observed that there is stress concentration near to elliptical hole in the yoke, so the yoke will fail in that region. The design for the yoke would be subjected to F.E Analysis to find the effect of loads on the yoke. 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 performance criterion. As the thickness decreases, other dimension would remain constant. A S Bharule [10] 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, drivable joint angle, cyclic load to avoid this problems various method such as a topology optimization method, Weight reduction method, Shape optimization method, manufacturing method S.K.Chandole, M.D.Shende, M.K.Bhavsar[11] In this work „design and finite element analysis of steering yoke in automobiles by hypermesh is carried out The element stress analysis of steering yoke covers the maximum stress and structure is expected to sustain, without fatigue failure. Hence steering yoke is analyzed for the stress produced under torque conditions. Naik Shashank, Giridhar, Sneh, Hetawal, Baskar P [12] Analysis of the Yoke clearly shows that by a small modification in the existing design the strength of the part can be increased significantly. A.V.PATIL, RITESH P. NEVE [13] The use of composite material reduces the weight of joint significantly as composite having lower density.. The reduction in weight gives further advantages in increase in fuel economy of vehicle. In this study analysis is being perform on universal joint .in this joint certain modification are made in the existing geometry and analyzes for the identical loading and boundary condition. Universal joint will be analyzing in the ANASYS and result will be compared. Farzad Vesali, Mohammad Ali Rezvani and Mohammad Kashfi, [14] Focuses on increased the performance and improve the life expectancy of the cardan joint. The used of the intermediate spring and damper increasing the torque arm, increasing the degree of freedom in order to remove the regularity in the impact load, installing rigid ring over cardan joint arm to act as the inner ring. R.O. RITCHIE[15] The mechanisms of fatigue-crack propagation are examined with particular emphasis on the similarities and differences between cyclic crack growth in ductile materials, such as metals, and corresponding behavior in brittle materials, such as intermetallics and ceramics. Heyes AM [16] studied the common failure types in automobiles and revealed that the failures in the transmission system elements cover 25% of all the automobile failures. The crack beginning location of the joint yoke corresponds to highest stress points.

Dhananjay S Kolekar [17] Steering Yoke is subjected to torsion loads while manoeuvring the vehicle on a given terrain. 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. Abhay M. Kalje [18] The steering system and steering column are the one of the most important devices of an automobile. Yoke assembly is always subjected to torsion and shear. In this paper finite element analysis of the component is carried out to find the stress and displacement of the final product. Purvesh Shah [19] Universal joint is defined as a mechanical device that can transmit a power or rotational motion from one shaft to another at fixed and varying angles of intersection of the shaft axis. Here are-design of a Universal coupling which will allow power transmission between two misalign axis is carried out. As compare to existing design it has much better strength to withstand against given load. Dhwani V. Sanghan [20] Main objectives are to reduce shear failures by Modification of pin in existing design of universal coupling. The modeling of proposed design is to be done by ANSYS software & results are compared with existing design.

Ravikant [21] the work described here forms part of an optimization project carrying out the design optimization of universal joint yoke and drive shaft using the ANSYS software. The model of universal joint yoke has been developed in Solid works then imported in ANSYS workbench and the model of drive shaft has been generated in ANSYS itself. In this work finite element analysis of a universal joint yoke and drive shaft has been taken as a case study.

Future Scope:

Following work may for the scope for future work,

1. The variable stresses acting on yoke calculate by analytical method can be validated by FEA for fatigue analysis approach.
2. The analysis of yoke can be further extended for advance engineering materials and cost factor.
3. During investigation the probable cause of yoke failure is improper lubrication, the further work can be done on effective lubrication.
4. The analysis can be further extended for design modification to overcome the failure.

References:

[1] Gagandeep Singh 2017" Finite Element Methods based Structural Analysis of Universal Joint using Structural Steel Material"

[2] Mr. S. K. Dhage 2017 Failure Analysis and Modification of Yoke Assembly of Transmission System in Automobiles

[3] Mr. Anuj A. Muley [2016] failure analysis of yoke joint assembly

[4] AbhishekMandal Utkarsh Sharma Harshit Pant [2016] "Static Structural Analysis of Universal Joint to Study the Various Stresses and Strains Developed in Power Transmission Systems"

[5] B.SWATHI [2015] "DESIGN AND ANALYSIS OF UNIVERSAL COUPLING IN POWER TRANSMISSION DESIGN AND ANALYSIS OF UNIVERSAL COUPLINGIN POWER TRANSMISSION"

[6] Sang June Oh [2015] "ANALYSIS OF RZEPPA AND CARDAN JOINTS IN MONORAIL DRIVE TRAIN SYSTEM"

[7] Dong-Kyun Min, Min-Eung Kim,[2015] "A study on precision cold forging process improvements for the steering yoke of automobiles by the rigid–plastic finite-element method," Journal of Materials Processing Technology 138,339–342.

[8], Avinash C Vasekar [2015] "failure analysis and optimization of universal joint yoke subjected by torsion and shear"

[9] S. P. Chaphalkar [2012] "Analysis and development of Yoke for Medium duty Automobile Vehicle Using Finite Element analysis"

[10] A S Bharule [2014] "AN INVESTIGATION ON STRESS DISTRIBUTION FOR OPTIMIZATION OF YOKE IN UNIVERSAL JOINT UNDER VARIABLE TORQUE CONDITION: A REVIEW"

[11] F.Schmelz,(2011)"Failure investigation and stress analysis of a longitudinal stringer of an automobile chasis,"Engineering Failure Analysis 16, 1696–1702.

[12] Naik Shashank Giridhar, Sneh Hetawal, Baskar P,2013 "Finite Element Analysis of Universal Joint and Propeller Shaft Assembly" International Journal of EngineeringTrendsand Technology(IJETT)–Volume 5 Number 5 - Nov 2013 ISSN: 2231 "[13]A.v.patil, Ritesh p. Neve [2012] "Material Optimization and Weight Reduction of Universal Joint Yoke Using Composite Material"

[14]Farzad Vesali, Mohammad Ali Rezvani and Mohammad Kashfi,(2009) "Dynamics of universal joints, its failures and some propositions for practically improving its performance and life expectancy," Journal of Mechanical Science and Technology 26 2439~2449.

[15] R.O.Ritchie 1999 Mechanisms of fatigue-crack propagation in ductile and

brittle solids International Journal of Fracture 100: 55–83.

[16] Heyes AM, 1998 “Automotive component failures,” Eng Fail Anal; 5(2):129–41,

[17] Dhananjay S Kolekar [2014] “STRUCTURAL ANALYSIS OF UNIVERSAL JOINT USING FINITE ELEMENT METHODOLOGY”

[18] Abhay M. Kalje “DESIGN, DEVELOPMENT AND STRUCTURAL ANALYSIS OF UNIVERSAL JOINT

[19] Purvesh Shah, “Modelling & FEA of Universal Coupling of an Automobile Truck”

[20] Dhwani V. Sanghan, “Modelling & Analysis of Universal Coupling Used In Agro Industry”

[21] Ravikant “Finite Element Analysis and Optimization of Universal Joint Yoke using CAE Tools”