Report on Failure Analysis of Universal Joint of A Transmission Drive Shaft

Mr.S.K.Dhage[1] Prof. Hredeya Mishra[2] Prof.P.G. Tathe[3]
[3]HOD
[1,2,3]Department of Mechanical Engineering
[1,2]Jaihind College of Engineering, Kuran (Pune University) [3]SCSCOE, Rahuri Factory (Pune University)

Abstract— 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 Yoke joint, Cardan joint or Hooke joint. 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. It Is generally, but not necessarily, constant during operation. Failed Yoke joint is the most common vehicles driveline problem. In this study, failure analysis of a universal joint yoke of an automobile power transmission would be carried out. For the determination of stress conditions at the failed section, stress analyses can also carried out by the finite element method.

Key words: Yoke joint, failure analysis, FEA, Torsion & Shear

I. INTRODUCTION

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. A spider hinges these two yokes together. Since the arms of the spider are at right angles, there will be four extreme positions during each revolution when the entire angular movement is being taken by only one half of the joint. This means that the spider arm rocks backwards and forwards between these extremes and the output-shaft speed will therefore increase or decrease twice in one revolution.

Fig. 1: Typical Yoke-joint Assembly

During operation yoke should undergo for an infinite life with different loading. However, the highest stresses occur at the crack beginning location of the yoke. A possible surface fault leads to crack propagation at the highly stressed point. After a crack propagation period, the component undergoes fracture. A finite element stress analysis need to be carried out at the failure region to determine the stress distribution and possible design improvement. An FEA based software like Nastran or ANSYS or any suitable software is utilized for the solving the given problem. An attempt to evolve an improved design resisting the failure and in turn enhancing the life would be the objective for this work. Figure show 3D geometry model of Yoke assembly.

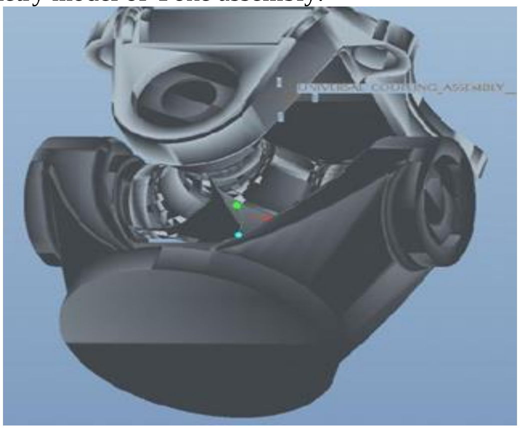

Fig. 2: 3D Geometry Yoke-joint Assembly

II. PROBLEM STATEMENT

Yoke assembly are one of the most important components in drive shaft. It generally subjected to torsion Stress and shear stress. Thus, these rotating components are susceptible to fatigue by the nature of their operation. Failed U-joint is the most common vehicles driveline problem. U-joint yoke failures are commonly occurred because of excessive torque load, shock loads, improper application, and fatigue. To offer Engineering solution to the component named `Yoke' while addressing functionality of the transmission drive shaft during the service life of the component. It is very important to know the accurate prediction for the drive shaft to fail. (2)

III. YOKE ASSEMBLY FAILURE DATA

A. Brinelling

Brinelling is when needle marks appear on the surface of the U-joint cross, which is usually caused by excessive torque, driveline angle. It can also be caused by a seized slip yoke or by a sprung or bent yoke.

Fig. 3: Universal Joint Brinelling

B. Spalling

Spalling looks like the bearing surface of the U-joint has been "scraped"away. Spalling is usually caused by water or dirt contamination.

Fig. 4: Universal Joint Spalling

C. Burned U-joint cross Trunnions

Improper lube procedures, where recommended purging is not accomplished, can cause one or more bearings to be starved for grease. Always make sure new, fresh grease is evident at all four Yoke joint seals.

Fig. 5: Burned U-Joint Trunnion

D. U-joint Fractures

U-Joint fractures are usually not only caused by a Torsion & Shear load, but also be caused by an improper application of torque load. Calculate the torque transmitted by the engine/trans combination

Fig. 6: Universal Joint High Torque Fracture

E. Bent or Deflected End Fitting

Bent yokes will put abnormal loading on the U-joint bearings and lead to premature failure. A yoke can be bent by a Torsion load or by over torque the yoke.

Fig. 7: Weld Yoke Failure (5)

F. U-joint Cross, Broken at a Bearing Surface

U-joints seldom break off at the bearing surface. It takes a very large Torsion load to cause this type of failure. It is also very difficult to inspect for this type of failure because they, many times, start as a small crack and progress into a complete failure some time further "down the road".

G. Prime Mode of Failure

Yoke is subjected to Torsion load as one of the important failure mode. Spider hinge and inner faces of Yoke subjected to shear load. Yoke assembly undergoes fatigues loading. Risk priority number of Torsion, Shear & Fatigue is higher than critical value. An attempt to evolve an improved design resisting the failure focus on Torsion, Shear & Fatigue.

IV. Generic Specifications for a Typical Yoke

- Material used:- Rolled/ Forged Steel - Carbon Steel C1021 or suitable
- Heat Treatment - Annealed. Typical hardness of the material-HRC 35-48
- Application for case study – Automotive (Transmission Drive Shaft)

V. Need for Analysis

Besides other type of loading, Yoke is subjected to torsion load as one of the important failure modes. A finite element stress analysis need to be carried out at the failure region to determine the stress distribution and possible design improvement. Since suitable software like UG or CATIA V5 is normally utilized for creating the geometry of the component (3D model), this could be further extended over the scope of CAE during the natural phase of design progression. The design verification can be achieved without elaborate need for prototypes at each phase saving time and effort. A final prototype for the final design review can be employed for verifying the analytical results.

VI. Objectives

An attempt to evolve an improved design resisting the failure and in turn enhancing the life would be the objective for this dissertation work. The key objectives for this work:

1) Identify and study using software tools (for simulation/ analysis), the nature and characteristics of stresses acting on the component
2) Evaluate the influence of the loads/ mass/ geometry/ boundary conditions over the nature and extend of stresses
3) Review the existing design and consider improvement for negating the harmful influences of undue stresses (Torsion or Shear)
4) Carry out physical experimentation to validate the model

VII. Methodology

A. *Analytical Method*

After the construction of the geometry (3D model) and preprocessing (meshing), a static stress analysis is planned by using the mechanical properties of the material (Elasticity modulus = 205 GPa, Poisson's ratio = 0.29 of the typical Carbon steel material variant) as input data for preparing the model for analysis. The solid model followed by finite element mesh followed by static analysis for assessing the distribution of stress values should offer good inputs, in turn, to review the design in the light of these results. The analytical/ computational approach offers results through simulation/ analyses for the case study predefined for the solver. The technique would deploy any of the following software tools: Patran/ HyperMesh Nastran, ANSYS, Abaqus, Radioss or any compatible CAE software in the `Structural' domain.

B. *Experimental Method*

Upon creating a physical prototype identical in geometry and mechanical properties to the intended component during production, the same is set-up for testing under identical service conditions for the component on field. A comparison of the results obtained through physical experimentation and the analytical (using simulation/ software) could offer a basis for validation. To simulate the working conditions, the force considered to be applied at the spider mounting location as a torsional moment could be about 500Nm and above (based on the application and the size of the vehicle). However the value takes a minimum and a maximum limit depending on the driving conditions and the auxiliary mechanisms to assist the maneuverability of the vehicle.

VIII. Steps for Proposed Work

1) Creation of Geometry for Yoke.
2) Importing the geometry for meshing.
3) Assigning the nature of loads and the values for loading
4) Solving for the meshed model to identify stressed areas.
5) Viewing the results.
6) Modifying the geometry/ mass/ boundary conditions
7) Solving the meshed model again (iterations)
8) Comparison / Interpretation of the results
9) Recommendations.

IX. Scope of the Project

This project will focus on performing failure analysis of Yoke assembly of transmission drive shaft using Hyper mesh or PATRAN-NASTRAN software. The model must be constructed using CATIA or SOLIDWORK software before it can be imported into Hyper mesh or PATRAN-NASTRAN to be analyzed. Spectroscopic analysis need to be done to the Yoke joint using FOUNDRY-MASTER UV knows the type of material used. The physical and mechanical properties that will use in stress analysis were taken from the material website database. In this study, fracture analysis of a universal joint yoke in a drive shaft of an automobile power transmission would be carried out. An attempt to evolve an improved design resisting the failure and in turn enhancing the life of Yoke assembly.

References

[1] Siraj Mohammad Ali Sheikh*2012"Analysis of universal coupling under different torque condition"
[2] Scott Randall Humme, Constantin Chassapis , 1998"Configuration design and optimization of universal joints With manufacturing tolerances"
[3] Bayrakceken.Tasgetiren, I.Yavuz 2006,"Two cases of failure in the power transmission system on vehicles: A universal joint yoke and a drive shaft"
[4] S.KINME. T.KAMIKAWA. A.NISHINO "Development of stamped Yoke for high rigidity intermediate shaft"
[5] M. Godec, D j. Mandrino, M. Jenko "Investigation of the fracture of a car's drive shaft"
[6] Monolithic Insulated joint brochure " SYPRIS TECHNOLOGIES".

