Design and Analysis of Yoke Joint Assembly

Mr. Anuj A. Muley
[1]PG Scholar, Mechanical Engineering,
Bapurao Deshmukh College of Engineering,
Sevagram,Wardha , Maharashtra ,
India.

Dr. M. J. Sheikh
[2]Professor, Mechanical Engineering,
Bapurao Deshmukh College of Engineering,
Sevagram, Wardha , Maharashtra ,
India.

***Abstract*- A Universal Yoke 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. Since they subjected to large amount of variable stresses induced in Universal Yoke Joint and effect on different components of it. Numerical and Finite Element Analysis method are used. For Analysis TATA MIDI-712 model has been taken into consideration. It is observed that the failure of Yoke Joint has been occurred due to the various stresses induced which are taken into consideration for analysis of failure part (Yoke Joint). For Finite Element Analysis, CAD Model of Universal Yoke Joint is prepared using CREO PARAMETRIC and this model is imported in ANSYS where stresses analysis by Finite Element Analysis.**

Key word(s): Yoke Joint, Critical Stress, Numerical Methods, FEM.

I. INTRODUCTION

A universal joint is a positive, mechanical connection between rotating shafts, which are not parallel, but intersecting. They are used to transmit motion, power or both. The simplest and most common type is called the Yoke 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.The flexibility is achieved by constructing the joint with two U-shaped yokes which is joined by a cross shaped hub. One of the yoke is attached to the end of each portion of the split shaft and joined with the cross hub, with the U-sections oriented at 90 degree to each other. It is commonly known for its use on automobiles and Bus, trucks etc.

Figure-1: Universal Yoke Joint and Transmission Drive Shaft (Midi-712)

The power transmission system of vehicles consists of several components which encounter unfortunate failures. These failures may be attributed to material faults, manufacturing and design faults, etc.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 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-2: Universal Yoke Joint Assembly(Midi712)

II. LITERATURE REVIEW:

H. Bayrakceken, et.al. (2005) presented spectroscopic analyses, metallographic analyses and hardness measurements are carried out for universal joint yoke and a drive shaft of an automobile power transmission system. For the determination of stress conditions at the failed section, stress analyses are also carried out by the finite element method.

S. Kinme, et.al. (2004) analysed was effectively utilized for repeatedly reviewing the design. As a result, high rigidity and low cost stamped yoke was developed.

FarzadVesali, et.al. (2012) analyzed the loading behavior and the surface conditions of the defected bearings. By comparison with the known fatigue theories attempts are made in order to dig into the causes for the failures.

NaikShashank, et.al. (2013) analysed 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 analysed in ANSYS and the results are compared.

S. G. Solanke and A. S. Bharule (2014) investigated Stress Distribution for Optimization of Yoke in Universal Joint under Variable Torque Condition. Yoke is analyzed under torque load from steering rod observing hot spot Stress concentration region, hot area potentially carrying load. Scope to optimize part by eliminating density in order to maximize area of hot region with reducing in dead region.

S. K. Chandole, et.al. (2014) analysed of the component is carried out to find the stress and Displacement of the final product. Using FEA analysis identified the nature and characteristics of stresses acting on the yoke and also evaluates the influence of the loads/ mass/ geometry/ boundary conditions over the yoke.

Kamal Kashyap and D. G. Mahto, (2014) Presented 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.

Swapnil S. Kulkarni, et.al. (2014) analysed 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 turn-on hold two Yoke together at right angles to each other.

Ritesh P. Neve (2015) analysed is being performed on universal joint certain modifications are made in the existing geometry and analyses for the identical loading and boundary condition.

III. IDENTIFICATIONOF PROBLEMS:

By the discussion with the workers and executives of ST workshop, it is found that the yoke joint failure is a serious issue in ST Bus. It is found that the Yoke Joint fails mainly due to

I) Slip Yoke Fracture And

II) Flange Yoke Fracture

Hence it is decided to perform failure analysis to check the dimension of Yoke Joint with existing material in order to identify the other causes of failure.

It is planned to redesign the Yoke Joint if required. It is planned to check the strength of Yoke Joint by changing the material. It is decided to validate the design and analysis by CAD and FEM.

TABLE-1: Yoke Joint Failure Part	
	Slip Yoke Fractures
	Flange Yoke Fractures

IV. SCOPEOF PROBLEMS:

When a ST-BUS moves on the road, the stresses will be developed in the Transmission Drive Assembly due to road conditions, proper servicing, maintenance and lubrication which causes failure of YOKE JOINT, which is a frequent problem faced by ST-BUS. Due to the failure of YOKE JOINT the BUS will stops which are a great inconvenience to Passengers, ST-BUS Employee and Worker.

Hence YOKE JOINT failure is genuine problem. This study helps to highlight the causes of YOKE JOINT failure and suggest the design modification to reduce failure of YOKE JOINT, this study will help to minimize the inconvenience to Passengers, ST-BUS Employee and Worker.

V. ANALYTICAL CALCULATION OF YOKEJOINT:

<u>Strength at weakest section X-X shown in Fig-3</u>

Analytical and numerical calculation of stresses for joint yoke that was in explanation was carried out with real dimensions. After that, the parameters are varying and the stress distribution was calculated by numerical method in order to identify the best design solution with the lowest level of stresses.

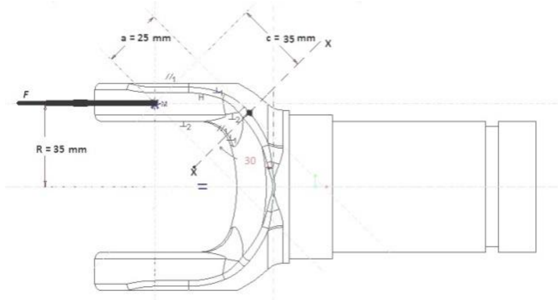

Figure-3: Scheme of Slip Yoke Joint (midi 712)

For yoke joint that weakest section X-X condition was carried out with real dimension.

After that, the yoke joint parameters are varying and the stress distribution was calculated by numerical method to identify the new design solution with the lowest level of stresses.

From fig-1.4 shows some parameters to calculating for stress.
(a) = 35 mm
(b) = 20 mm (Width of Slip Yoke Joint)
(c) = 65 mm
Dimension of Slip Yoke Joint from Figure-3:-
Diameter of yoke slip (D) = 70 mm

Radius of yoke slip (R) = 35 mm
Hub diameter (h) = 55 mm

The materials properties of the steel are C-50 Annealed (SAE1050) are given in TABLE-2:

Mechanical properties	Symbol	Steel Units
Young's Modulus	E	202 GPa
Shear Modulus	G	78 GPa
Poisson's Ratio	V	0.3
Density	ϱ	7600
Yield Strength	Syt	365 MPa
Shear Strength	Sys	246 MPa

Now,

(бt) = Allowable Tensile Stress
(бτ) = Allowable Shear Stress
(бc) = Allowable Crushing Stress
(FOS) = Factor of Safety = 4

1) **(бt)** = $\frac{Syt}{FOS} = \frac{365}{4}$ = 91.25 MPa
2) **(бτ)** = $\frac{Sys}{FOS} = \frac{246}{4}$ = 61.5 Mpa
3) **(бc)** = 2× б**τ** = 2 × 61.5 = 123 Mpa

The basic data for calculation are as follows,
Input power (Pi) = 15 kw
Input number of rotation (N) = 2000 rpm
Angle of path (α) = 30 degree

<u>To Calculate Maximum Torque</u>:-

Tmax $= \frac{p\times60}{2\pi\ N} = \frac{15\times10^3\times60}{2\pi\times2000}$ **= 71.61 N-m**

The calculation of maximal load force at yoke joint of (F)

T = R F sin∝
71.61 =0.035× F × sin (30)
F = 4.092 KN

Now,

1) Resisting Bending Torque (Wb)

Wb $= \frac{bh^2}{10} = \frac{0.020\times0.055^2}{10} = 6.05\times10^{-6}m^3$

2) Resistant Shear Torque (Ws)

Ws $= \frac{hb^2}{5} = \frac{1}{5}\times0.055\times0.020^2 = 4.4\times10^{-6}m^3$

3) The Bending Stress (Бmax)

бmax $= \frac{F.\ c}{Wb} = \frac{4.092\times10^3\times0.065}{6.05\times10^{-6}}$ = 43.96 MPa

4) The Shear Stress (τmax)

$\tau max = \frac{F.\ a}{Ws} = \frac{4.092\times10^3\times0.035}{4.4\times10^{-6}}$ = 32.55 MPa

5) The equivalent stress (бe)

Бe= $\sqrt{б^2+3\times\tau^2} = \sqrt{46.93^2+3\times32.55^2}$
бe = 73.35 MPa

The stress is greater than the allowable stress which is (бτ) = 61.5 MPa.
i.e. Yoke Joint may as failure by use of material SAE1050.

VI. CONCLUSION:

The analysis of YOKE JOINT has been carried out considering various aspects like torsional, bending and impact loading. It is required to redesign the joint. It is decided to change the material and dimensions to withstand the required load. Further it is planned to validate the design using CAD and FEM.

VII. REFERENCES:

1. H. Bayrakceken, S. Tasgetiren, I. Yavuz, “Two cases of failure in the power transmission system on vehicles: A universal joint yoke and a drive shaft.” Afyon 03200, Turkey 16 November (2005).
2. 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 165 (2004).
3. FarzadVesali, Mohammad Ali Rezvani and Mohammad Kashfi, “Dynamics of universal joints, its failures and some propositions for practically improving its performance and life expectancy.” August (2012), Volume 26, Issue 8, pp 2439-2449.
4. NaikShashankGiridhar, SnehHetawal and P. Baskar, “ Finite Element Analysis of Universal Joint and Propeller Shaft Assembly.”, International Journal of Engineering Trends and Technology (IJETT) - Nov(2013) – Volume Number 5.
5. 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.
6. S. K. Chandole, M. D. Shende, M.K. Bhavsar “ Structural Analysis Of Steering Yoke Of An Automobile For Withstanding Torsion/ Shear Loads”, International Journal of Research in Engineering and Technology (IJRET) : Volume: 03 Issue: 03- Mar-(2014).
7. Kamal Kashyap and 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).
8. Swapnil S. Kulkarni, P.G. Tathe and Prof. D.S. Bajaj, “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).
9. Ritesh P. Neve, “Material Optimization and Weight Reduction of Universal Joint Yoke Using Composite Material”, IOSR Journal of Mechanical (IOSR-JMCE)ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 12, May - Jun. (2015).