iJRASET
International Journal For Research in Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 12 **Issue:** VIII **Month of publication:** August 2024

DOI: https://doi.org/10.22214/ijraset.2024.64066

www.ijraset.com

Call: 08813907089 | E-mail ID: ijraset@gmail.com

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 12 Issue VIII Aug 2024- Available at www.ijraset.com

Photo Stress & Finite Element Analysis of Slip Yoke of Ashok Leyland Dost

Mr. Ajit Sanjaykumar Naik[1], Dr. P. J. Patil[2]
Mehanical Engineering, Shivaji University Kolhapur

Abstract: ***We studied the simulation, experimental behaviour of slip yoke under loading condition. Slip yoke used for transmitting torque and rotation, is one of the most important components in motor vehicles. It is required to connect the other components of the drivetrain such as transmission and differential to each other. Basically, a slip yoke comprises of yoke parts, splined parts. Each component on slip yoke is exposed to torsion while they perform their functions such as angular and axial movement. Therefore, each component should be strong enough against to torsion in the design and analysis processes. In this study, experimental and simulation methods were used to design the slip yoke, investigated, and compared with each other. In the simulation studies, finite element analysis (FEA) was implemented by using three different loading conditions. photo elastic stress analysis (experimental method) is carried out. The comparative analysis of both methods is done & failure cause of slip yoke is found out.***

Keywords: Simulation analysis, Experimental analysis, Photo elastic analysis etc.

I. INTRODUCTION

We studied the simulation, experimental behaviour of slip yoke under loading condition. Slip yoke used for transmitting torque and rotation, is one of the most important components in motor vehicles. It is required to connect the other components of the drivetrain such as transmission and differential to each other. Basically, a slip yoke comprises of yoke parts, splined parts. Each component on slip yoke is exposed to torsion while they perform their functions such as angular and axial movement. Therefore, each component should be strong enough against to torsion in the design and analysis processes. In this study, experimental and simulation methods were used to design the slip yoke, investigated, and compared with each other. In the simulation studies, finite element analysis (FEA) was implemented by using three different loading conditions. photo elastic stress analysis (experimental method) are carried out. The comparative analysis of both methods is done & failure cause of slip yoke is found out

II. PROBLEM STATEMENT

In operation slip yoke is subjected to both bending load as well as torque. The slip yoke of Ashok Leyland Dost is in operation failed frequently. The fig. shows failure of yoke joint. The theoretical analysis provides magnitude of stress but to observe stress pattern experimental and Finite Element Analysis is necessary.

III. OBJECTIVES

1) To carry out dynamic state analysis of slip yoke for bending stress using Finite element analysis software
2) To carry out dynamic state analysis of slip yoke for bending stress using photo stress technique.
3) To compare results of experimental and finite element analysis method.

IV. PROPOSED WORK:

A. Methodology

It is proposed to carry out experimental and Finite Element Analysis of slip yoke in Actual working Condition Phase I- Literature survey. In this phase literature survey of photo stress analysis technique will be carried out by referring journal like ASME journal, Springer link, European patents, US patents, etc.

Phase II – In this phase Finite Element analysis And Experimental Analysis of slip yoke In Actual Condition is carried out. A) Finite element analysis:

1) To carry out assembly of slip yoke joints models.
2) To import models in analysis software. To carry out dynamic analysis using FEA software

B. Experimental Work

It is decided to carry out dynamic stress analysis by using Photo stress technique as follows:

1) To select proper coating material.
2) To apply coating on slip yoke at critical section.
3) To place slip yoke on to the setup
4) To find out different stresses at critical section.

C. Simulation Analysis

Finite Element Analysis Finite Element Analysis, CAD model of Yoke Joint is created and imported in ANSYS software for Stress Analysis. Constraints are applied at the one side of shaft and the force is applied by giving this condition, deformation and stresses are evaluated and presented in next chapter. • Data are collected for the Yoke Joint used in Bus from S.T Workshop and various dimensions are taken by measurement. • The various parameters such as torque, stresses are obtained. • The Existing yoke joint is modelled using Creo parametric-3.0 software and the same joint is analysed using ANSYS 14.0 software.

D. Modelling

The slip yoke geometry was obtained through precise measurements using callipers and/or by importing CAD models from the manufacturer. The slip yoke was modelled in [insert CAD software name] using the acquired geometry. Features such as splines, bearing surfaces, and keyways were accurately incorporated into the model.

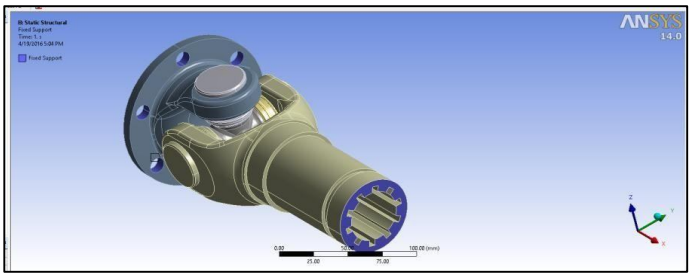


Figure 3.1-3D Model of slip yoke

E. Meshing

In Geometry Preparation Prepare the geometry for meshing. This might involve simplifying complex features, removing unnecessary details, and ensuring the model is watertight. In Mesh Type Choose an appropriate mesh type based on your analysis requirements. For structural analysis, a tetrahedral or hexahedral mesh might be suitable. In Mesh Settings Define mesh sizing controls to ensure that critical areas of the slip yoke are adequately resolved. You may need finer mesh near areas of high stress or strain. In Mesh Generation It Generate the mesh using ANSYS meshing tools. Ensure that the mesh quality is acceptable, with sufficient element quality and aspect ratio. After generating the mesh, perform checks to ensure its quality. Look for elements with poor quality, skewed elements, or excessive distortion. In Post processing: Once the mesh is generated, you can proceed to perform your analysis. Apply appropriate boundary conditions and loads based on your simulation requirements. In Analysis Run the analysis using ANSYS solver. Monitor convergence and ensure that the results are reasonable. After the analysis is complete, post-process the results to extract relevant information such as stresses, displacements, and other quantities of interest.

Number of elements = 27997 Number of nodes = 28647 Element size = 2.5

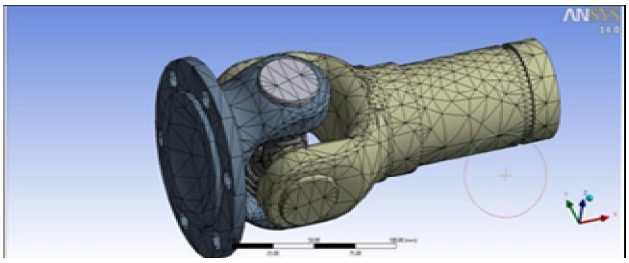


Figure 3.2- Meshing of Yoke Join

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 12 Issue VIII Aug 2024- Available at www.ijraset.com

V. RESULTS ANALYSIS

A. FEA Results Analysis

Stress distribution in slip yoke Is as shown in figure.

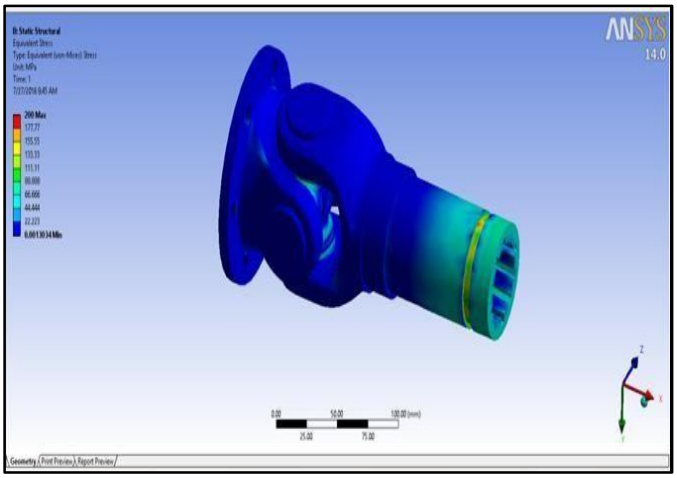


Fig. Stress distribution at first condition

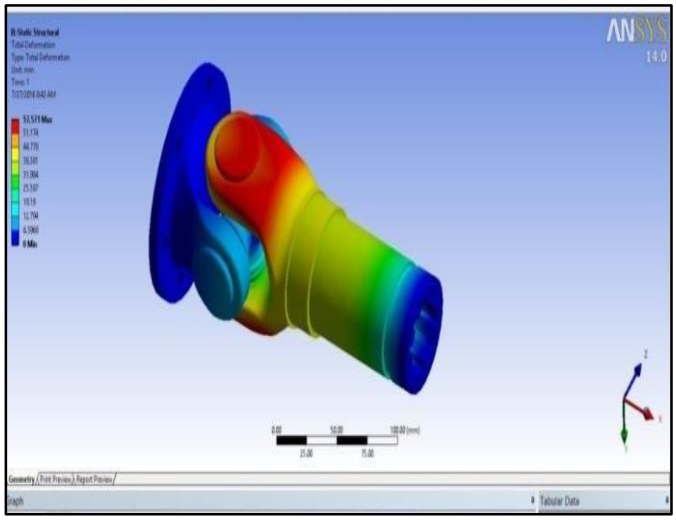


Fig. Stress distribution at second condition from analysis the following stress values are simulated.

Stress Values by ANSYS

Sr. No.	Speed of Driven Slip Yoke	Stress (N/mm^2)
1	1000	237.56
2	2000	298.21
3	3000	373.36

Thus, the slip yoke is meshed and analyzed by using ANSYS software and the results are obtained. Here the results obtained i.e. Stresses observed in yoke are greater than yield strength. There is plastic straining and permanent set. Therefore, the slip yoke is damaged.

B. Experimental Results Analysis

Sr No.	Speed in RPM	n	Load (kg)	Stress (σa)
1	1000	0.80	1	235.56
2	1500	0.74	1	257.23
3	2000	0.70	1.5	295.21
4	2500	0.66	1.5	352.12
5	3000	0.67	2	381.36

The technique of photoelasticity can be effectively explored to experimentally measure the internal stresses in indeterminate structures. The stresses generated in experimental analysis is varies from 235.56 N/m^2 to 381.36 N/m^2 which are very high compared to yield strength. In simulation analysis stress generated is 237.56 N/mm^2 to 373.36 N/mm^2.Due to this reason of practical stresses generated is high so slip yoke continuously undergoes failure.

VI. CONCLUSION

In this study, for the analysis of slip yoke, finite element analysis and photo elastic stress analysis (experimental method) are carried out.

1) The result of 373.36 N/mm^2 from the finite element analysis at highest loading condition. When considering the experimental studies, the results from finite element analysis and the results from experimental method, are similar to each other with a deviation approximately equal to 7 N/mm^2 . These results reveal that,

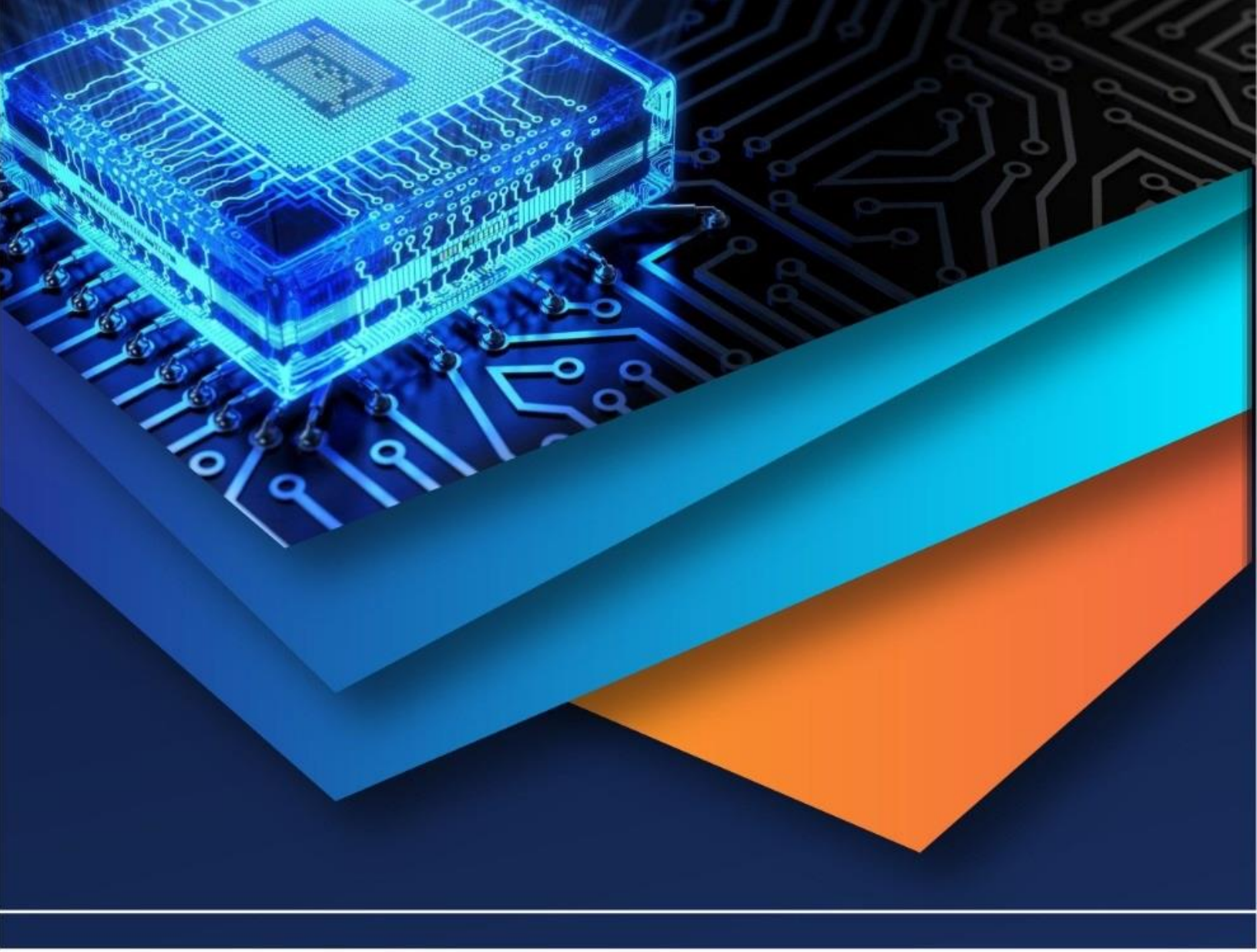
2) Stress values of critical section obtained from FEA, supports the results from experimental method. Because, the results from the studies where the both loading types are used are as close to each other.

3) Maximum stress on the critical area of the slip yoke is determined higher than the yield strength of the material. Stresses observed in yoke are greater than yield strength. There is plastic straining and permanent set. Therefore, the slip yoke is damaged.

4) Maximum stress observed in the slip yoke is 373.36 N/mm2 by finite element method & from photoblastic stress analysis (experimental method) stress generated is 381.36 N/mm^2 (Material Yield stress 350 N/mm^2). Due to this reason of practical stresses generated is high compared to yield strength of slip yoke so continuously undergoes failure after periodic time.

REFERENCES

[1] Scott Randall Humme, ConstantinChassapis 2000“Configuration design and optimization of universal joints With manufacturing tolerances”Mechanism and Machine Theory 35 463±476
[2] Siraj Mohammad Ali Sheikh 2012“Analysis of universal coupling under different torque condition”International journal of engineering science & advanced technology Volume-2, Issue-3, 690 – 694 .
[3] FarzadVesali, Mohammad Ali Rezvani.2012“Dynamics of universal joints, its failures and some propositions for practicallyimproving its performance and life expectancy”Journal of Mechanical Science and Technology 26 (8) 2439~2449
[4] Nick Cristallo, T.Kamikawa, A.Nishino 2004 “Development of stamped Yoke for high rigidity intermediate shaft” Koyo engineering journal English research edition no 165
[5] Shinde Dinesh S, SolankiPradeep M. 2013“Wear analysis of u- joint needle bearing”International Journal of Science, EnvironmentandTechnology, Vol. 2, No 2, 162 - 167
[6] MohdZulhilmi. 2010 “Failure of drive shaft using finite element analysis”Universiti Malaysia Pahang
[7] Kumarel, Magalhães, Bicalho.2009 “Failure investigation and stress analysis of a longitudinal stringer of an automobile chassis”Engineering Failure Analysis 9 303–312
[8] Mirasiav, Jenko.2010“Analysis of the drive shaft fracture of the bucket wheel excavator”Engineering Failure Analysis 33 66–74
[9] M. Godec, D j. Mandrino, M. Jenko2009 “Investigation of the fracture of a car’s drive shaft”Engineering Failure Analysis 16 1252–1261
[10] Bayrakceken,Tasgetiren,I.Yavuz 2007“Two cases of failure in the power transmission system on vehicles: A universal joint yoke and a drive shaft”Engineering Failure Analysis 14 716–724



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY