Finite Element Analysis of Universal Joint

[1]Swati Mahajan, [2]Prof. S. D. Khamankar
[1]PG scholer [2]Associate Professor
[1]Dept. of Mech. Engg.
[1]RCERT Chandrapur, Maharashtra, India

***Abstract*—Universal joint is important component in transmission system from safety point of view. Since they are subjected to large amount of variable stresses induced in universal joint and effect on different components of it, numerical and FE analysis methods are used. For analysis TATA 1210 model has been taken into consideration. It is observed that the failure of yoke has been occurred due to the various stresses induced which are taken into consideration for analysis of failure part (yoke). Sotherberg theory is used for numerical analysis of universal joint. For FE analysis, CAD model of universal joint is prepared using Pro-E and this model is imported in ANSYS where stress analysis is done by FEM. Finite element analysis have been performed by various parameters. From the output of this analysis it is observed that results obtained are in close agreement with each other and max stress concentration occurs at shaft and outer surface of yoke.**

***Index Terms* - ANSYS, FE analysis, Sotherberg theory, universal joint. Etc**

I. Introduction

An automobile engine produces power which is conveyed to the wheels so as to enable the vehicle to run. Universal joint and propeller shafts play an important part to transfer power from the engine to the wheels. The purpose of Universal joint is to transmit power (torque) even at varied angles of the transmission system (propeller shaft). A single Universal joint is shown in figure-1 is a driving yoke on one side which is connected to the main shaft of the gear box and the driven yoke is connected to the propeller shaft. These two yoke are connected by means of a grossed spider when the driving shaft rotates, the driven shaft also rotates at the same time the universal joint permits angular motion this propeller shaft can rotate at any angle. Thus power is transmitted from the gear box to the propeller shaft at any particular angle.

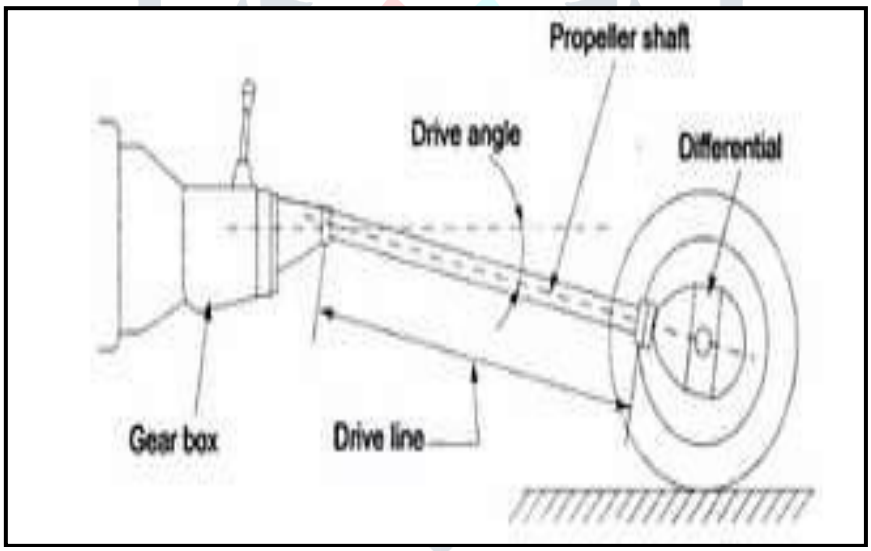


Figure 1 Transmission system

II. INTRODUCTION TO PROBLEM, SCOPE AND METHODOLOGY

Drive shafts are one of the most important components in vehicles. Thus, these rotating components are susceptible to fatigue by the nature of their operation. Common sign of driveshaft failure is vibration or shudder during operation. Driveshaft mainly involves in steering operation of vehicle. Drivers will lose control of their vehicle if the drive shafts broke during high speed cornering. Because of this human life can be in great danger if we don't know when, where and how the drive shaft will fail. It is very important to know the accurate prediction for the drive shaft to fail.

For this purpose to check the stress induce in yoke, different methods are been carried out that is theoretical and F.E. analysis by using various parameters. Failure analysis is the process of collecting and analyzing data to determine the cause of a failure and how to prevent it from recurring. Failure analysis and prevention are important functions to all of the engineering disciplines. A component or product fails in service or if failure occurs in manufacturing or during production processing. In any case, one must determine the cause of failure to prevent future occurrence, and/or to improve the performance of the device, component or structure. It is possible for fracture to be a result of multiple failure mechanisms or root causes. A failure analysis can provide the information to identify the appropriate root cause of the failure.

Figure 2 Photograph of universal joint TATA1210

(Yoke) the problem for this analysis was taken under consideration from the given figure 2 the universal joint (yoke) was considered the component made up to SAE 1137, which is a material in the low alloy steel group of density 8.03 g/cc the model was analyzed in ANSYS 14.0 considering the mechanical properties as ultimate tensile strength 675 MPa and yield strength as 344.7 MPa. The different therotical stresses are calculated for various components by applying mean torque to driver shaft of yoke 2865.98 N-m and the results found for maximum shear stress were 193.97 MPa. For this study the model was replicated for the same torsional moment and the design, results were been tabulated for theoretical and FE analysis. For the further analysis the shaft of yoke was considered for different conditions for shaft 100% hallow, then 75 % hollow and 25% solid, then 50% hollow and 50% solid, by increasing 1mm thickness of yoke and applying fillet of 4mm at base of yoke. Results of mean shear stress and deformation were tabulated.

III. ANALYTICAL ESTIMATION OF STRESSES IN YOKE

Since many of the machine parts such as axle's shafts, crankshaft, connecting rods, etc are subjected to variable as alternating loads also known as fluctuating as fatigue loads. If has been found experimentally that when a material is subjected to repeated stresses, if fails at stress below the yield point stresses such type of failure of a material is known as fatigue. The failure is caused by means of a progressive crack formation which are casually fine and of microscope size. The failure may occur even without any prior indication. The fatigue of material is affected by the size of component, relative magnitude of static and fluctuating loads and the number of load reversal. Soderberg method is used for ductile material for a machine component subjected to reversed shear loading.

Equivalent shear stress

$$\tau_{es} = \tau_m + \frac{\tau v \times \tau y \times Kfs}{\tau e \times Ksur \times Ksz}$$

Working/design stress, τ

$$= \frac{16}{\pi d^3} \left\{ T_m + \left(\frac{\tau y}{\tau e}\right) \times kfs \times Tv \right\}$$

Where,

τ_{es} - Equivalent shear stress
τ_m - Mean shear stress
τ_v - Variable shear stress
τ_y - yield strength in shear
τ - Working or design shear stress
T_m - Mean torque
Ksur - Surface finish factor
Ksz - Size factor
Kfs - Fatigue stress concentration factor
d - Diameter of shaft

By considering different torque conditions the comparisons between both theoretical and FE results are available [ ]

IV. Preparation of CAD Model of Universal Joint

In this work, stress analysis of universal joint is considered by using FEM. For this purpose it is essential to validate the finite element analysis of universal joint by analytical procedure. Thus, analysis has done on to determine the shear stress and deflection which is validated with analytical equations. FE analysis carried out using model TATA 1210 of same geometrical dimensions as used in theoretical and applied same torque.

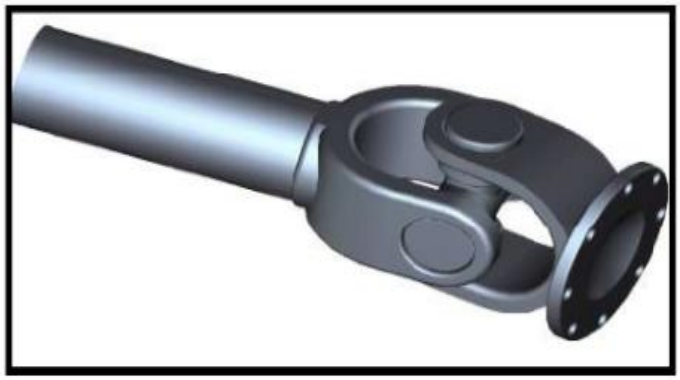


Figure 3 CAD model of Universal joint

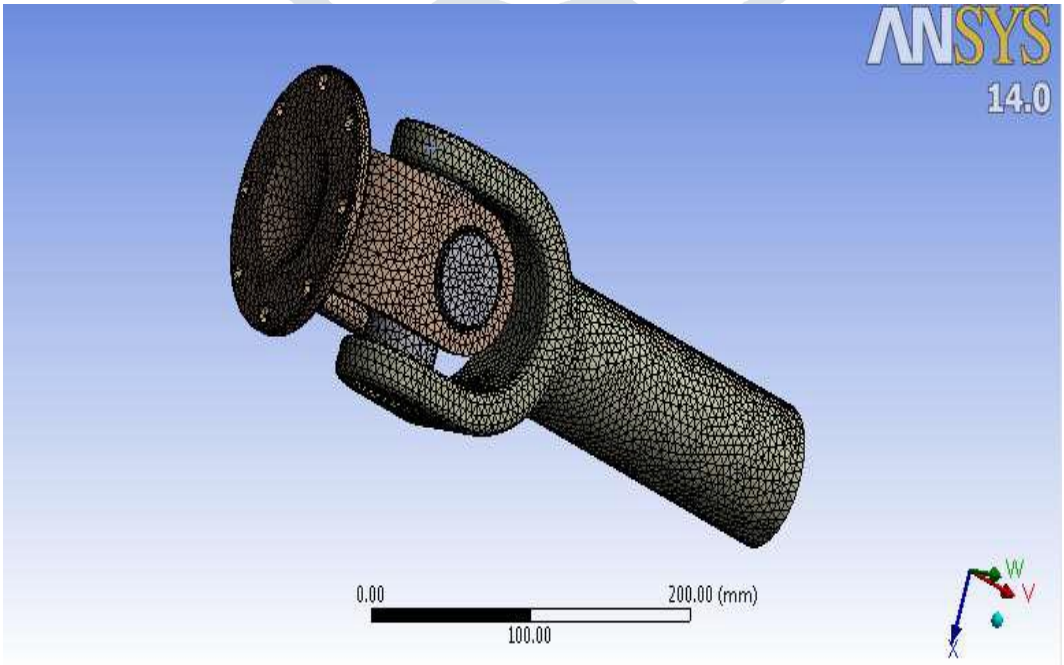
V. Stress Analysis using FEM

The solid CAD model in .igs format is imported to ANSYS for FEA.A structural 10 node Tetrahedral Solid 187 element is selected for creating FE model of Universal joint. Material properties as shown in table are assigned.

Table 1 Mechanical Properties of Material SAE1137

Sr. no.	Symbols	Parameters	Values
1	E	Young's modulus	190 GPa
2	μ	Poisson's ratio	0.27

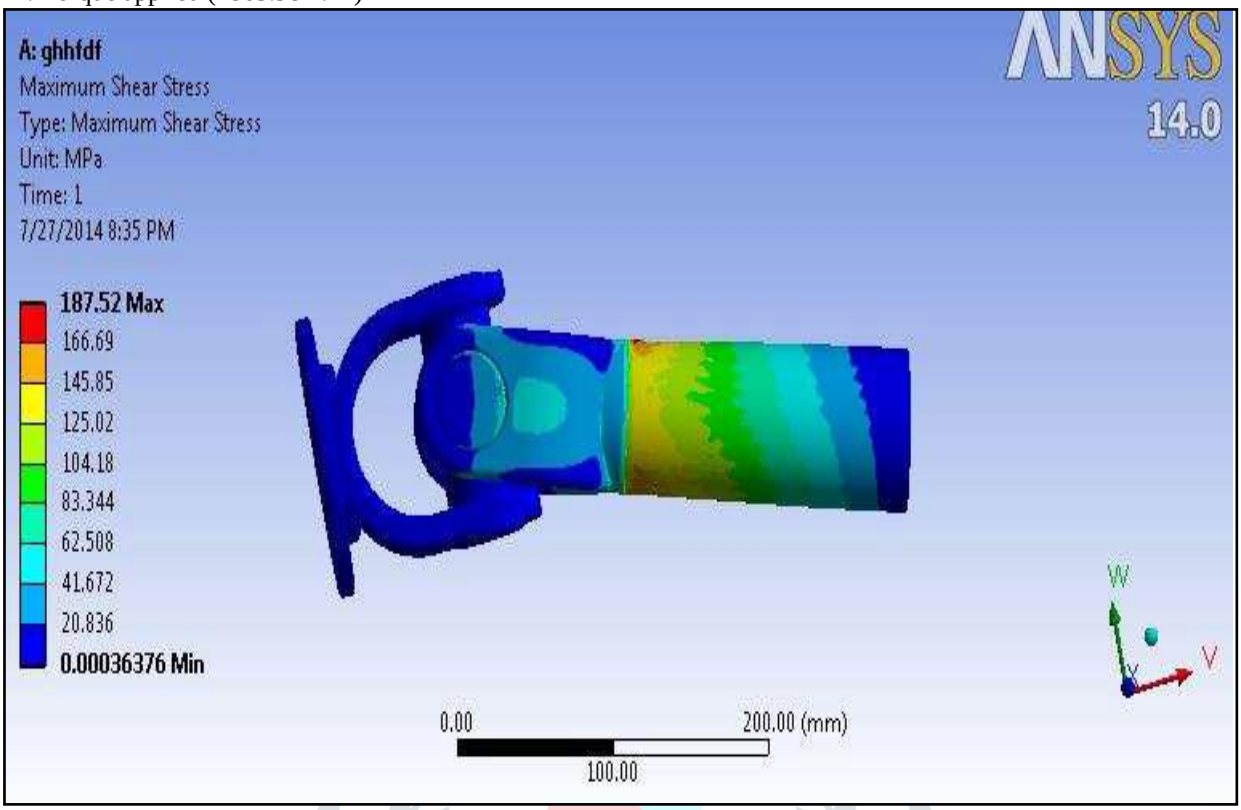
And model is meshed using free meshing and smart size option. Slider for smart size is set to 5. The meshed FE model created is shown in fig.



Meshing view of universal joint

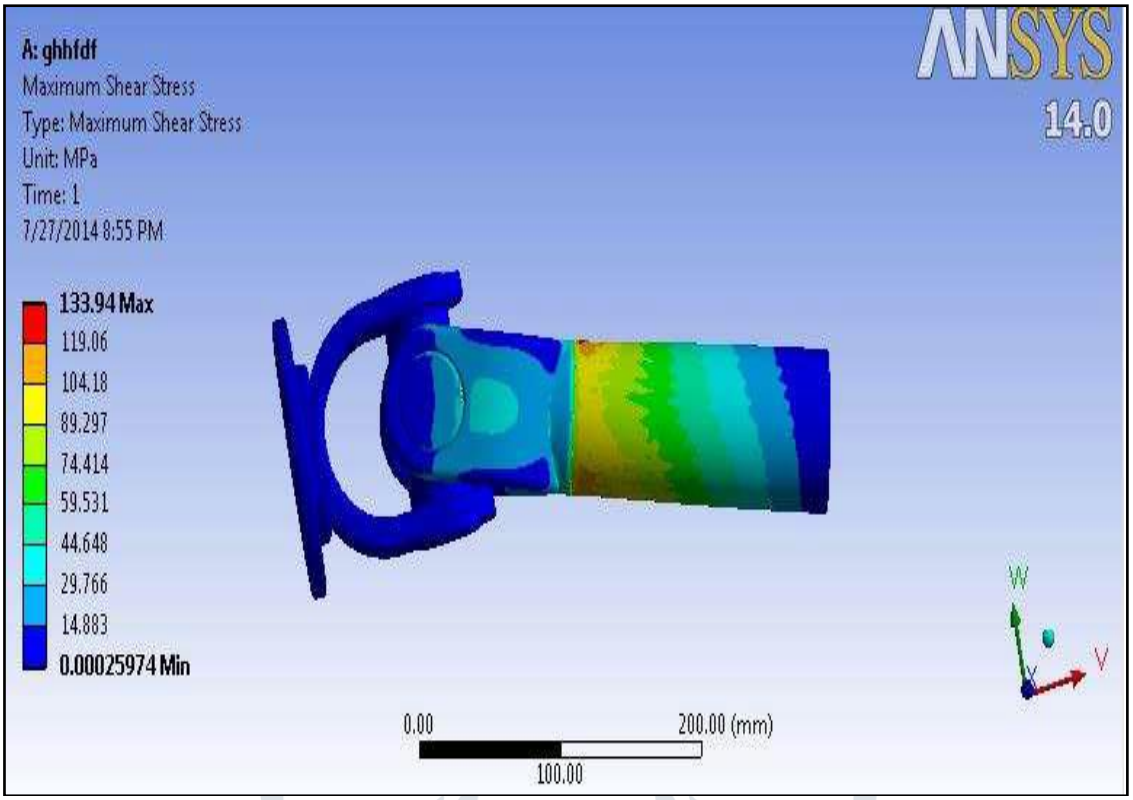
VI. FE RESULT FOR DIFFERENT TORQUE CONDITIONS

A. Torque applied (2865.98N. m)



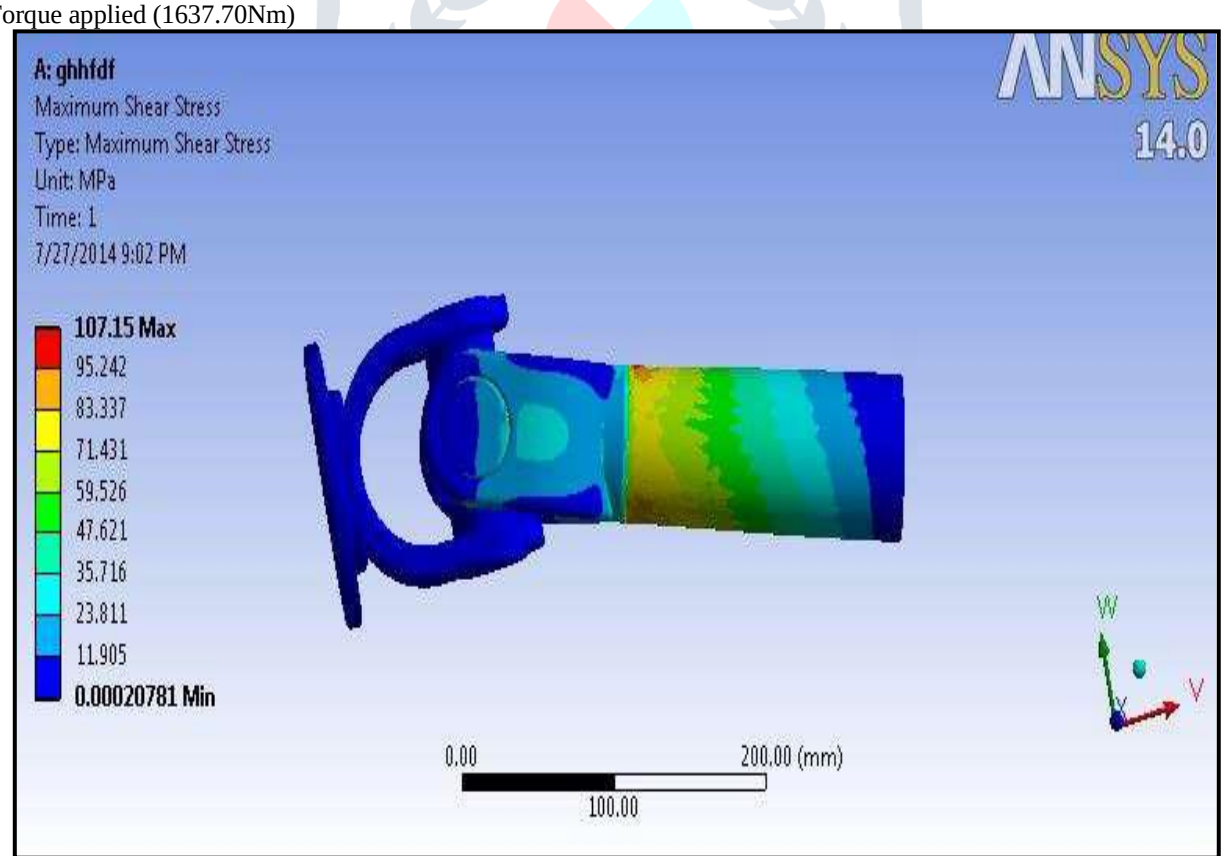
FE result for Max shear stress

B. Torque applied (2047.13Nm)



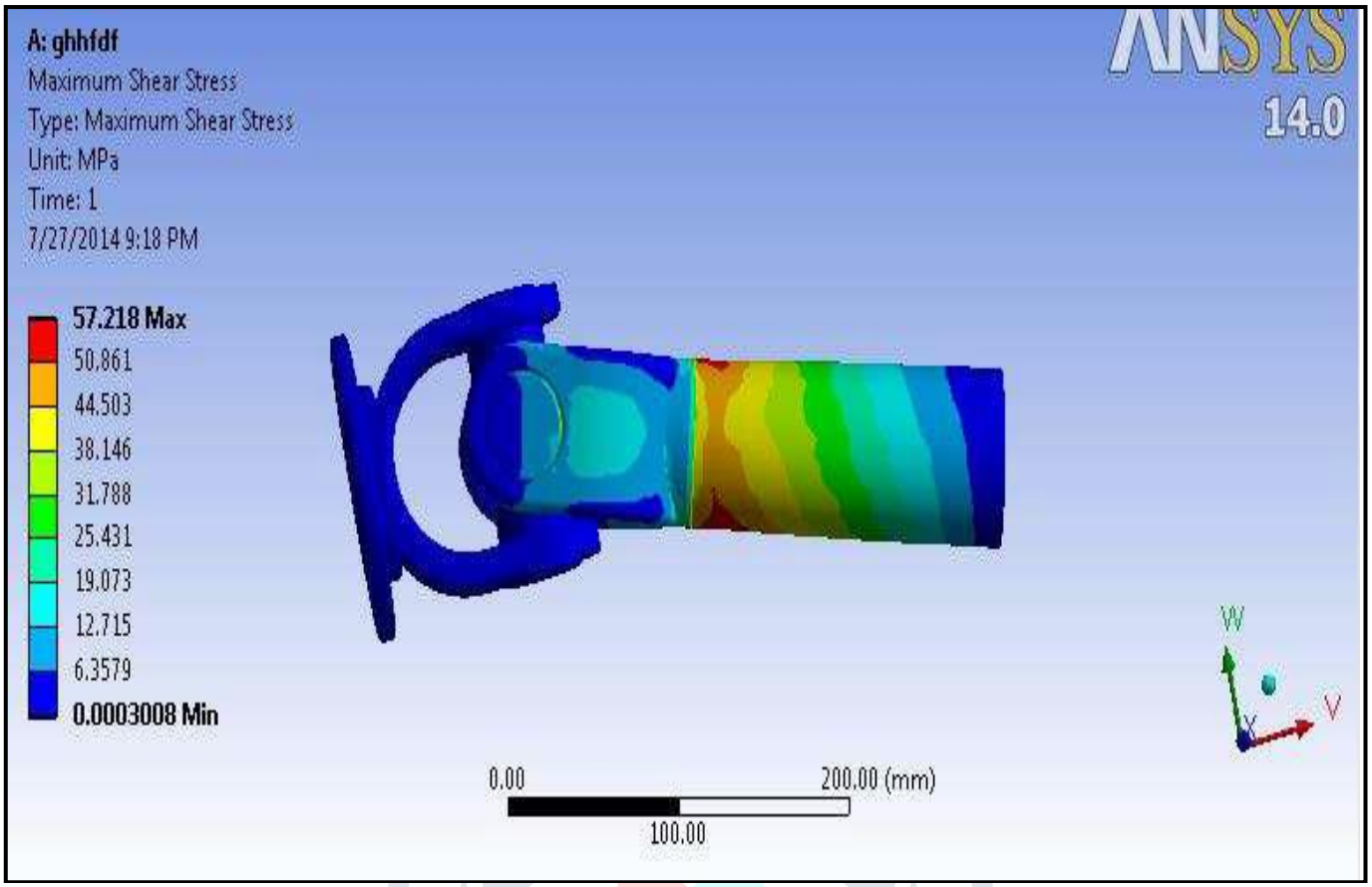
FE result for Max shear stress

C. Torque applied (1637.70Nm)



FE result for Max shear stress

D. Torque applied (1023.57Nm)



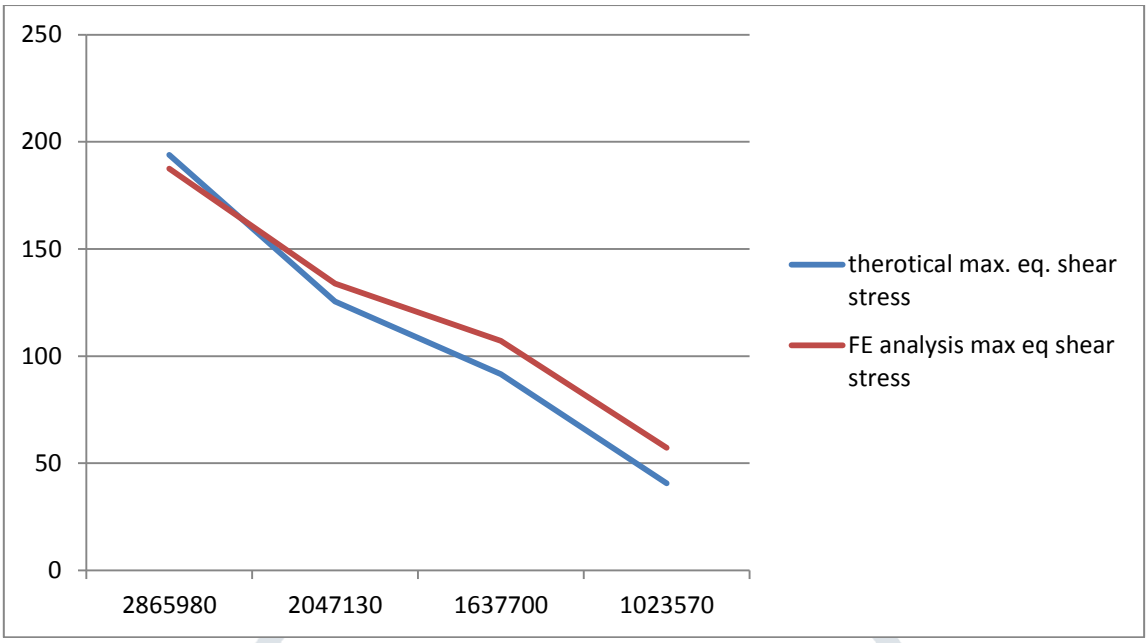
FE result for Max shear stress

VII. RESULT, DISCUSSION AND CONCLUSIONS

The results of stress analysis evaluated from analytical and FE for Universal joint are presented below.

Table 4.4 Comparison table of theoretical

Sr. No.	Mean torque Tm in N-mm	Theoretical max. equivalent shear tress τ_{es} (max) in N/mm^2	FE analysis max. equivalent shear tress τ_{es} (max) in N/mm^2
1	2865.98×10^3	193.97	187.52
2	2047.13×10^3	125.45	133.94
3	1637.70×10^3	91.56	107.15
4	1023.57×10^3	40.70	57.22



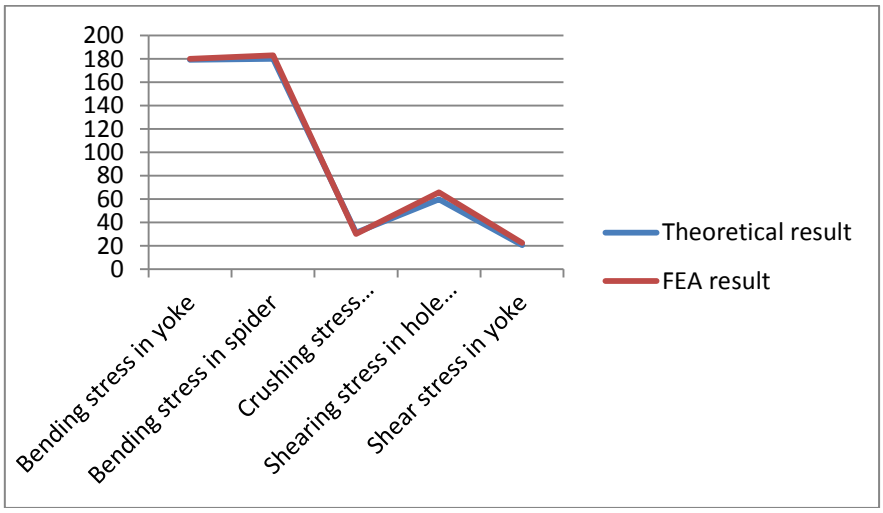
Graph 1 FE analysis results for max shear stress for mean torque

The theoretical analysis and FE analysis of actual model of universal joint reveal that the maximum torsional shear stress induced in the yoke of universal joint is 193.97 MPa and 187.52 MPa respectively which is less than yield strength in shear of the material used for model means the design is found to be safe.

Table .4.4 Comparison table of theoretical

Sr no	Stresses in component	Theoretical Result(N/mm^2)	FEA result(N/mm^2)
1	Bending stress in yoke	178.97	160-200
2	Shear stresses in yoke	20.47	20-41
3	crushing stresses between pin and hole of yoke	31.15	20-41
4	shearing stress in hole of yoke	59.70	42-81
5	bending stress in spider in cross	179.95	153.27-183.92

A. Mean FE analysis results for stresses in components



The stresses in bending stress in yoke, shear stresses in yoke, crushing stresses between pin and hole of yoke, shearing stress in hole of yoke and bending stress in spider in cross are determine analytically and by FEA.

VIII. SCOPE FOR FUTURE WORK

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] Boris Rakić1, Lozica Ivanović, Danica Josifović, Blaža Stojanović, Andreja Ilić, "The Influence of Variation In Position of Output Shaft To Load on The Cardan Joint Cross Shaft." Faculty of engineering, Sestre Janic 6, 34000 Kragujevac, Serbia. UDC: 621.825, Volume 39, Number 1,2013, pg. no. 53- 64.

[2] B. D. Shiwalkar "Design Data for Machine Elements. Dattatraya Publications Nagpur, 2011 Edition.

[3] DESIGN DATA, Data Book Of Engineers, PSG College of Technology, Faculty of Mechanical Engineering, Coimbatore.

[4] Kamal Kash yap, D. G. Mahto, "Anal ysis of Hooks Joint Using Ans ys b y VonMises Method" International Journal of Engineering Science and Technology (IJTES), ISSN: 2249 – 8958, Volume-3, Issue-3, February 2014,pg.no.304-305.

[5] Iqbal, J. and Qatu, M. "Vibration Analysis of A Three-Piece Automotive Shaft."

[6] John R. Dharte, "Universal Joint Grease Development."American Axle and Manufacturing, SAE Technical Paper 2002-01 0299, 2002, doi: 10.4271/2002-01-0299.

[7] Majid Yaghoubi 1*, Seyed Saeid Mohtasebi 1, Ali Jafary1 " Static And Dynamic Simulation Of A New High Deflection Constant Velocity U-Joint (Persian Joint)" Faculty of Engineering and Technology, University of Tehran, Iran.pg.no. 41 44.

[8] Naik Shashank Giridhar, Sneh Hetawal, Baskar P. "Finite Element Analysis Of Universal Joint And Propeller Shaft Assembly."International Journal of Engineering Trends and Technology (IJETT)-Volume 5 5-NOV 2013, pg. no. 226 229.

[9] Nick Cristello, and Il Yong Kim "Design Optimization Of An Automotive Universal Joint Considering Manufacturing Cost."Proceeding of the 17th IASTED International Conference, May 24-26, 2006, Montreal, QC, Canada. Pg. no. 499-504. 67

[10] Siraj MohammadAli Sheikh1 "Analysis of Universal Coupling Under Different Torque Condition" International Journal of Engineering Science and Advanced Technology (IJESAT), Volume-2, Issue-3, 690-694.

[11] R. S. Kumar, J.K.Gupta, "A Text Book of Machine Design" S. Chand Publication, first edition, pg. no. 181-223.

[12] Vishal Rathi, N. K. Mandavgade "Fem Analysis Of Universal Joint Of Tata 407"