Structural Strength Analysis and Validation of Steering Yoke Assembly by Optimization

Kulkarni Sagar Y[1] Prof. V. R. Bajaj[2] Swapnil S.Kulkarni[3]
[3]Associate Professor
[1,2]Department of Mechanical Engineering
[1,2]Alard College of engineering & management, Pune, India [3]Advent Tool Tech India Pvt. Ltd., Pune

Abstract— The force offered by the driver of an automotive in navigating the vehicle along the road is transmitted through the steering wheel through the steering system up to the front wheels. Manual or hydraulic mode is utilized to turn the wheels. This dissertation work aims to focus on the 'universal joint' also called the 'yoke' .The component have two halves that pair to effect a flexible axial joint while accommodating misalignment. Forces are transmitted through the 'spider' or the 'web' within the two halves. This work attempts to evaluate the strength of the joint to avoid failure. Besides, the scope for mass reduction in realizing a lighter component shall also be investigated using the methodology for 'Topology Optimization' while engaging `Altair Hyperworks - OptiStruct' in dealing with problem. Mathematical treatment is offered to understand the extent of forces to be borne by the joint during its operation. A tentative target of about 2% mass reduction is set while pursuing this work. The result determined by the computational methodology shall be validated through physical experimentation.

Key words: Yoke, universal joint, mass reduction, Hyper Works, Opti Struct, torque, steering system

I. INTRODUCTION

The steering system is often designed late in the build process. In a lot of cases it's best to mock up the steering when the engine and exhaust components are installed. Positioning the column, steering shafts, and `u-joints early on with respect to the engine and steering box will ensure that you select the correctparts. Steep angles and limited space may require a little trial and error to achieve the correct geometry, but with a little ingenuity and careful measuring, it can be accomplished.

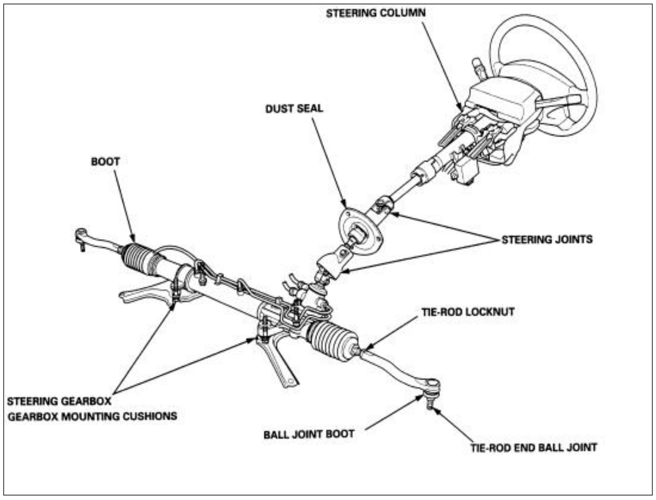

Fig. 1: A typical rack-and-pinion steering layout, showing how the rack acts directly on the road-wheel steering arm.

The steering system converts the rotation of the steering wheel into a swivelling movement of the road wheels in such a way that the steering-wheel rim turns a long way to move the road wheels a short way. The steering effort passes to the wheels through a system of pivoted joints. These are designed to allow the wheels to move up and down with the suspension without changing the steering angle.

U-Joint: Universal joints, or U-joints, transmit rotary motion at an angle, allowing you to route the shafts around obstacles. As the angle increases, the strength of the joint decreases proportionately. U-joint strength is rated at a maximum angle by the manufacturer. Build your system with the least amount of angle that allows adequate clearance, and never exceed the angle at which the joint is rated, typically 30 to 35 degrees; if a greater angle is necessary, use more U-joints. Flaming River recommends 15 degrees as the optimum angle.

II. LITERATURE REVIEW

In the interest of optimisation technique lot of research is completed and some are in process. Out of that many articles are focused on Topology Optimization and shape optimisation. The main aim of this researcher is to reduce maximum weight of component as well as reduce the cost of the components. Generally finite element analysis technique are used to finding out stress and corresponding strain induced in components, from this reference non affecting area are identify and researcher remove the material to optimize shape[3][4][5]

sometime optimization is done by changing material property [8]

Before starting optimization, entire steering system or similar component used is in different system are studied [1][2][6]

The universal joint is prove to failure on account of the torsion loads or torque that it has to sustain during its operation. The application of manual or hydraulic force activated by the driver or the reaction experienced by the joint by virtue of road conditions poses a risk of failure of its structural elements. The joint made of 'steel' or its special alloys capable of bearing such loads. A suitable variety of 'steel' needs to be chosen while considering the options available in making a choice. The same needs to be evaluated using suitable popular technique or methodology like the Finite Element Analysis i.e. F.E.A. Also the geometry of the joint. the on struction/shape of the two halves, the spider or web acting as coupler for the halves and the provision of suitable mating parts or features for respective elements in the steering system all force towards reinforcing strength in this joint. The design of the joint needs to be feasible for manufacturing which considering the available methods of producing the part, The mating should be effected in a manner so as not to risk any slippage or sudden disengagement during the operation. The assembly or disassembly of the components within the universal joint should be easy for manual or automatic assembly techniques followed in the industry. The strength of the mating parts may not be compromised, though in recommending a feasible design. The weight of the entire vehicle is desired to be as low as possible in realizing a fuel efficient vehicle which is using the steering system. The components in the steering system including the yoke be evaluated for identifying scope of mass or weight reduction. This work should focus on mass reduction of the yoke to the tune of 2% while ensuring adequate structural strength. The same could be attempted using topology optimization as a technique or methodology for realizing this objective.

III. Methodology

In this thesis work, we are using 3 methodologies as below:

A. Mathematical Methodology:

Based on vehicle design specification i,e front axle weight (FAW),Rear Axle weight (RAW), Wheel Base, Ground Clearance, Kerb Weight and Gross vehicle weight torque required steering gear input shaft is calculated numerical. This required torque is used for calculation. Torque applied in the yoke can be calculated with the help of standard empirical formulae. Distortion of the component can be calculated by considering the simplified geometry.

Steering Wheel Torque Calculations:

$\text{Operating load} = 30\ \text{Kg} = 30 \times 9.81\ \text{N} = 294.3\text{N}$

For calculation purpose we take 300N

$\text{Steering wheel radius} = 200\ \text{mm}$

$\text{Torque} = 300 \times 200 = 60000\ \text{N} - \text{m} = 60\ \text{Nm}$

Yoke analysed under torque of 60 Nm (Operating torque)

B. Computational Methodology:

In this method, FEA tools are used to compute the stress and displacement of the component. CAD model will be pursed from design department in .step or .iges format. Basic methodology for structural analysis is as follows:

With the advancement with the finite element analysis (FEA) modelling. Model based design of mechanical structures is replacing the traditional trial and error approach. Here the finite element analysis of steering yoke is done in Hyperworks software there are three steps

1) Pre-processing
2) Processing
3) Post-processing

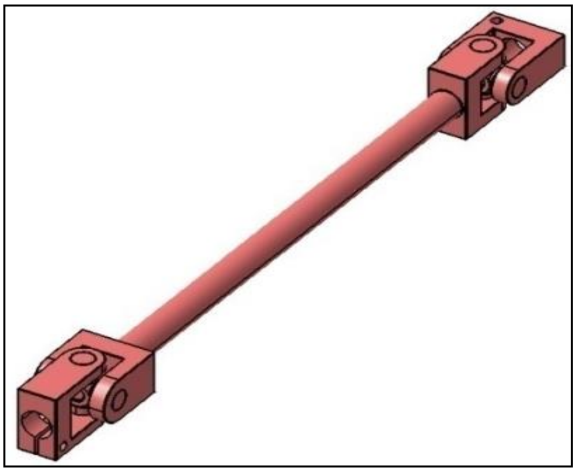

Fig. 2: Steering yoke 3D model

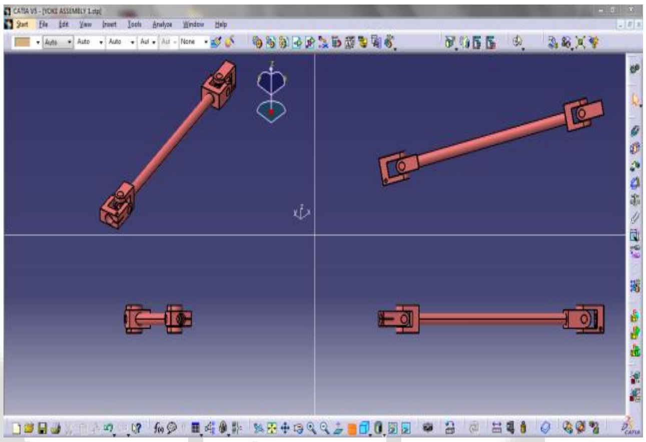

Fig. 3: Multiple views of steering yoke in CATIA V5 interface

1) Pre-Processing

After modelling the component and importing to hypermesh window meshing is carried out the tetrahedral meshing approach is employed for the meshing of the solid region geometry. Tetrahedral meshing produces high quality meshing for boundary representation of solid structural model. Since the tetrahedral is found to be the best meshing technique Divide the whole model into several parts and mesh one by one in different mesh densities.

Fig. 4: Steering Yoke 3D Mesh model

2) Meshing Details

No of elements = 119339

No of Nodes = 72136

Pre Processing = Hypremesh 11
Solver = Optistruct
Post Processing = Hyper view
Analysis = Linear Static
Material: Generic Steel

3) Material Properties

Material properties assigned to components are as follows:

Material :AISI 303
Young's Modulus : 2.8 E05 N/mm2
Poisson Ratio : 0.29
Density :7.8 x10^{-9} Ton/mm3

4) Processing

After pre-processing, Loads & boundary conditions are applied as shown in figure no.5

Boundary conditions - 60 Nm torque applied at top of yoke and Constraint (1-6) at lower side side of yoke.

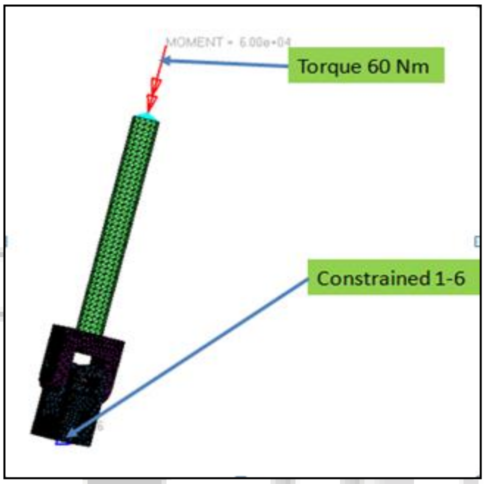

Fig. 5: Boundary Condition

5) Angular Deformation

By analytical method angular deformation is calculated by following formula,

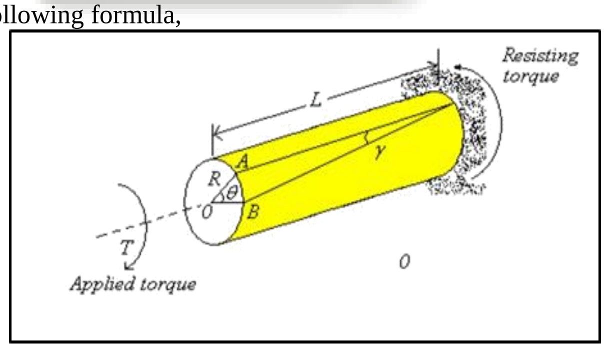

Fig. 6: General representation of torque applied on shaft.

$$\frac{T}{Ip} = \frac{Fs}{R} = \frac{C\theta}{l}$$

where,

T= Torque in N-m
Fs= Shear Stress in N/mm^2
R = Radius in mm
Ѳ = Angular Deflection in Radian.
Ip = Poplar movement of inertia in mm^4
C = Modulus of Rigidity in N/mm^2
L =Length in mm

Angular Deflection :	
Applied Torque (N-m)	60
Modulus of Rigidity (N-m)	80
Diameter (mm)	19
Length on (mm)	170

Table 1:

by putting valve in above equation, we get

Ѳ = 9.96x 10^{-3} rad

so, Ѳ =0.571°

6) Post-Processing

After successfully completed the process, the results can be viewed in hyper view. To view the results "Hyper view" panel next to optistruct is selected. In the hyper view the results are displayed. The two important results are noted and they are: 1.Vonmises elemental stress 2.Displacement. Firstly the component was designed for that vonmises elemental stress and displacement results are obtained from Hyperworks.

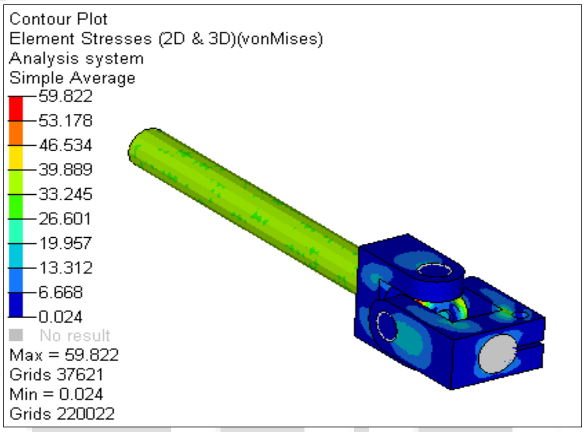

Fig. 7: Stress Contour

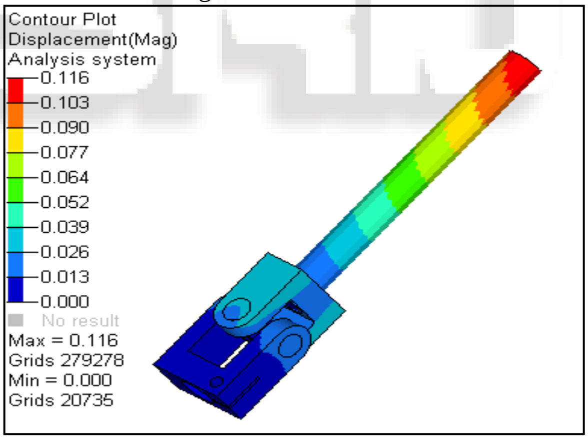

Fig. 8: Displacement Contour

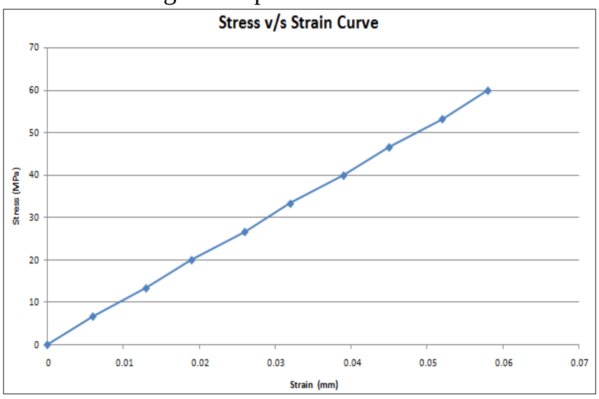

Fig. 9: Stress v/s Strain Diagram

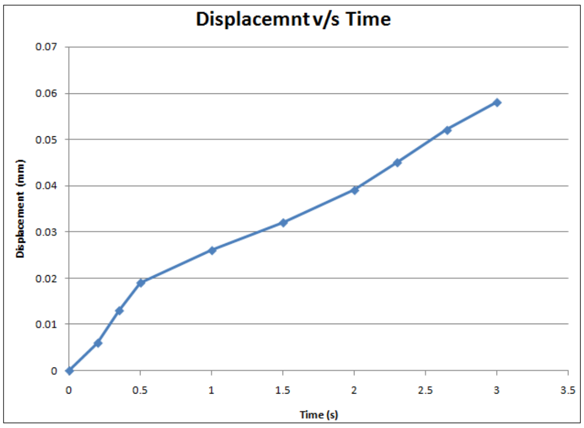

Fig. 10: Displacement v/s time Diagram

Maximum Displacement = 0.116 mm

Maximum Von-Mises stress = 119.64 MPa

7) Optimization of Existing Steering Yoke Assembly

Hypremesh optistruct software are an industry proven, modern structural analysis solver for linear and nonlinear problems under static and dynamic loadings, In optistruct software we need to defined designed & non designed area so that material can remove material from design area where no stress or displacement is observed as shown in below figure no

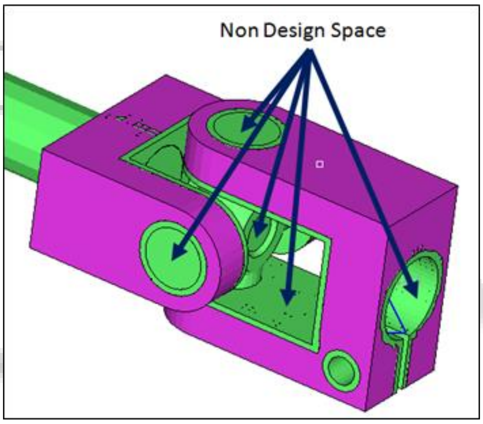

Fig. 11: Non Design Area define

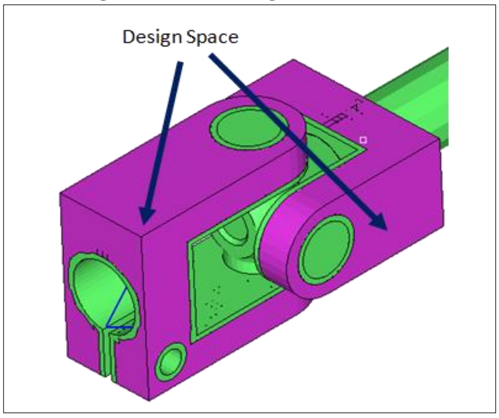

Fig. 12: Non Design Area define

Boundary conditions - 60 Nm torque applied at top of yoke and Constraint (1-6) at lower side side of yoke.

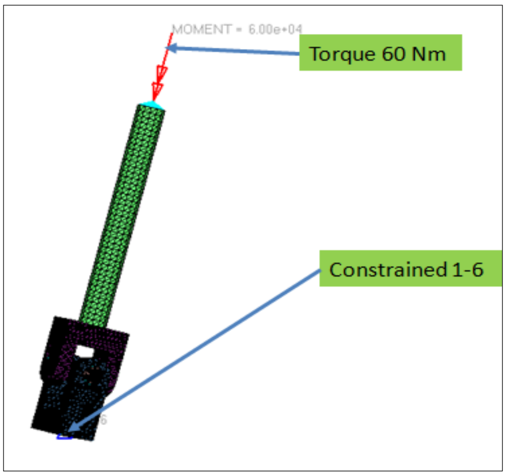

Fig. 13: Optimize Prepared Optimization

8) Meshing details for Modified Steering Yoke Assembly

No of elements = 64307

No of Nodes = 36202

Pre Processing = Hypremesh 11

Solver = Optistruct

Post Processing = Hyper view

Analysis = Linear Static

Material : Generic Steel

9) Material Properties

Material properties assigned to components are as follows:

Material :AISI 303

Young's Modulus : 2.8 E05 N/mm2

Poisson Ratio : 0.29

Density :7.8 x10^{-9} Ton/mm3

10) Post-Processing Modified Steering Yoke

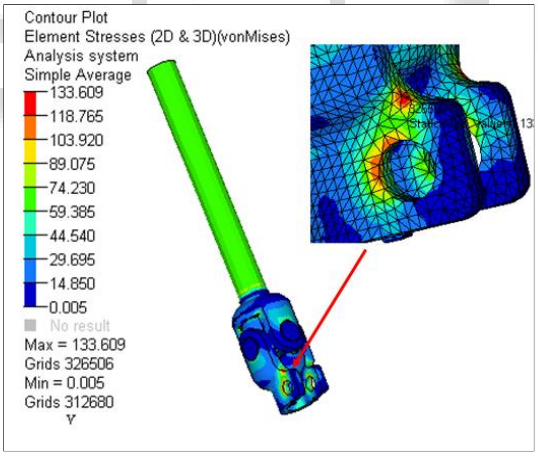

Fig. 14: Stress Contour (Modified steering Yoke)

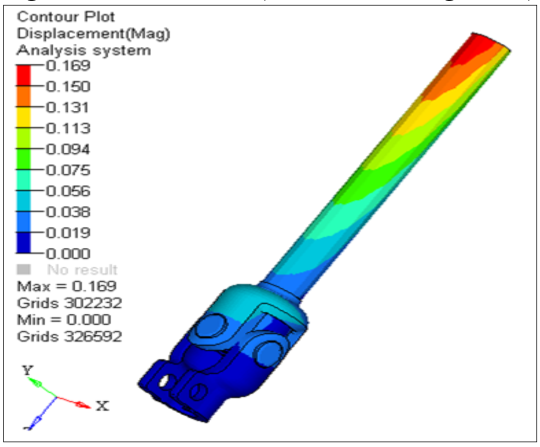

Fig. 15: Displacement Contour (Modified steering Yoke)

11) Angular Deformation of Steering Shaft

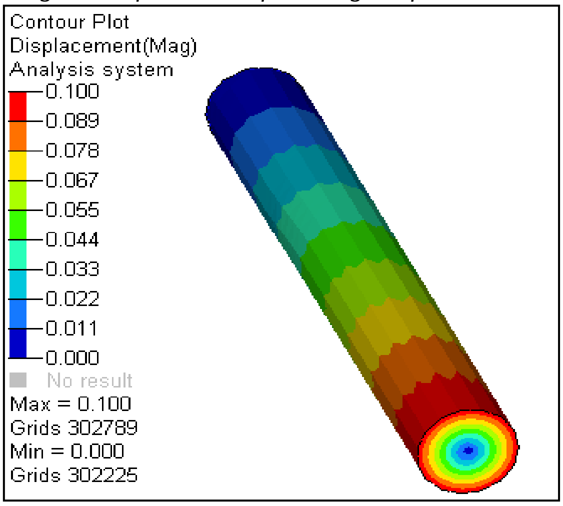

Fig. 16: Displacement Contour

Max Displacement = 0.100mm
Radius of the shaft = 9.5mm
Angular Deformation (ϴ)
tan ϴ = 0.1/9.5 so, ϴ = **0.603**°

IV. EXPERIMENTATION SET UP

Fig. 17: Typical Test setup for Physical Experimentation

The objective of current work is to determine the angular deformation acting on the steering yoke by using Torsional Testing Machine. Design of steering yoke assembly by theoretical approach is very critical job because of its complicated structure and subjected to combination of loading. The angular distortion in (θ deg.) in the shaft having same diameter is evaluated using torque testing machine. The same can be evaluated and compared using mathematical calculation and computational method.

V. RESULTS & DISCUSSION

From results obtained through CAE analysis for Existing Steering yoke assembly & optimized steering yoke assembly are shown which helps to understand comparison of behaviour of component before & after optimization. The comparison is as shown in below fig.15

From stress and displacement plot, it is observed that max displacement observed in the steering yoke assembly is 0.116 mm and stress is 119.6 MPa where stress are below yield limit of material (AISI303=415MPa)

Variant Name	Displacement (mm)	Von Mises Stress (MPa)	Mass (Kg)	Percentage of Mass reduction (Baseline – Existing)
Existing Steering Yoke Assembly	0.116	119.64	1.79	0%
Modified Steering Yoke Assembly	0.169	133.60	1.323	26.08%

Fig. 18: Result table

In optimized component, displacement found 0.169 mm & stress is 133.60 MPa. Result also shows 26.0% (0.437 Kg) mass reduction. The results obtained in optimized component is acceptable as the results are within limits of material properties

VI. CONCLUSION

From results, it is conclude that Topology optimization method is helpful to reduce the weight of component. Reduction of weight by 26.0% keeping maximum stresses & displacement within limit. The stresses observed in optimized component is within limit so it shows that change in component weight can reduce cost of component and overall steering system weight.

REFERENCES

[1] Stefan Marks, Robert Wellington," Experimental Study of Steer-by-Wire Ratios and Response Curves in a Simulated High Speed Vehicle" Fourteenth Australasian User Interface Conference (AUIC2013), Adelaide, Australia
[2] A.A. Adekunle S. B. Adejuyigbeb O.T. Arulogunc," Development of CAD Software for Shaft under various Loading Conditions" International Conference on Modeling, Optimization and Computing(ICMOC 2012)
[3] T. G. Rao, S.K. Saha, I.N. Kar , "Sensor-Actuator Based Smart Yoke fora Rack and Pinion Steering System" Fourteenth Australasian User Interface Conference (AUIC2013), Adelaide, Australia
[4] Nitin S Duryodhan and R R Gawande," Modelling of Intermediate steering shaft of Fiest car and Its static Structural Analysis" International Journal of computer application (0971-8887)
[5] Naik Shashank Giridhar, Sneh Hetawal, Baskar P," Finite Element Analysis of Universal Joint andPropeller Shaft Assembly" International Journal of Engineering Trends and Technology (IJETT) – Volume 5 Number 5 - Nov 2013
[6] P. Jayanaidu,M Hibbatullah, and Prof. P Baskar "Analysis of a Driver shaft for Automobile Applications"IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 10, Issue 2 (Nov. - Dec. 2013), PP 43-46

[7] Anup A. Bijagare , P.G. Mehar , V.N. Mujbaile," Desig Optimization & Analysis of Drive Shaft" VSRD-MAP,Vol. 2 (6), 2012, 210-215
[8] G. Pantazopoulos, A. Sampani and E. Tsagaridis,Torsional failure of a knuckle joint of a universal steel, Engineering Failure Analysis 14 (2007) 73–84.
[9] M. Godec,Dj. Mandrino and M. Jenko, Investigation of the fracture of a car's drive shaft, Engineering Failure Analysis 16 (2009) 1252–1261.
[10] Scott Randall Hummel and Constantine Chassapis, Configuration design and optimization of universal joints with manufacturing tolerances, Mechanism and Machine Theory 35 (2000) 463-476.
[11] K. Ito and T. Fujishiro, Stability Analysis of Automatic Lateral Motion Controlled Vehicle with Four Wheel Steering System.
[12] Yu Jianguo and Ba Xingqiang, The Simulation Analysis of All Terrain Vehicle Steering System based on FVP ,IEEE, Vehicle Power and Propulsion Conference (VPPC), September 3-5, 2008, Harbin, China.
[13] R.B.Gupta, Automobile engineering, 12th edition-2009, Page No.538-539, 595-606.
[14] Dr. Kirpal Singh, “Automobile Engineering”, volume 2, Standard Publishers Distributors, Delhi, 12th edition-2009, Page No.127-135, 198-217.