Design Optimization and Analysis of Propeller Shaft

R. Harimaheswaran
Assistant Professor, Department of Mechanical Engineering, Shivani College of Engineering &Technology, Tiruchirappalli, Tamilnadu, India.

N.Jeyakumar
Assistant Professor, Department of Mechanical Engineering, Shivani College of Engineering &Technology,Tiruchirappalli, Tamilnadu, India.

R.Satheeshkumar
Assistant Professor, Department of Mechanical Engineering, Shivani College of Engineering &Technology,Tiruchirappalli, Tamilnadu, India.

P.Viswabharathy
Assistant Professor, Department of Mechanical Engineering, Shivani College of Engineering &Technology,Tiruchirappalli, Tamilnadu, India.

Abstract – Automotive drive Shaft is a very important component of vehicle. The overall objective of this paper is to design and analyse a drive shaft for power transmission. Substituting low alloy steel for conventional metallic structures has many advantages because of higher specific stiffness and strength of alloy materials. This work deals with the replacement of conventional drive shafts with a alloy material's. In this work AISI 8750 is used as low alloy material. The design parameters were optimized with the objective of minimizing the weight of c drive shaft. The design optimization also showed significant potential improvement in the performance of drive shaft. In this present work an attempt has been to estimate the deflection, stresses, natural frequencies under subjected loads using FEA. Further comparison carried out for both steel and alloy materials and weight of the shaft is optimized and stress intensity factor found for both Steel and drive shafts.

Index Terms – Drive Shaft, AISI 8750.

1. INTRODUCTION

1.1. PROPELLER SHAFT

The term Drive shaft is used to refer to a shaft, which is used for the transfer of motion from one point to another. Whereas the shaft, which propel are referred to as the propeller shaft. Propellers are usually associated with ships and planes as they are propelled in water or air using a propeller shaft because apart from transmitting the rotary motion from the front end to the rear end of the vehicle forward. The shaft is the primary connection between the front and the rear end which performs both the motion and propelling the front end. Thus, the terms Drive Shaft and Propeller Shaft are used interchangeably.

The propeller shaft is a longitudinal drive shaft used in vehicles where the engine is situated at the opposite end of the vehicle to the driven wheels. A propeller shaft is an assembly of one or more tubular shaft connected by universal, constant velocity or flexible joints. The number of tubular pieces and the joints depends on the distance between the gearbox and the axle. On some four wheel drive vehicles one propeller shaft is used to power the rear wheels as with rear wheel drive and a second propeller shaft is used to power the front wheels. In this case the second propeller shaft is replaced between a transfer gear box and the front axle. Hence, it can be observed that a drive shaft is one of the most important components, which is responsible for the actual movement of the vehicle once the motion is produced in the engine. The designing of such a critical component is usually stringent, as any fracture in this part could lead to as catastrophic failure of the vehicle when it is in motion.

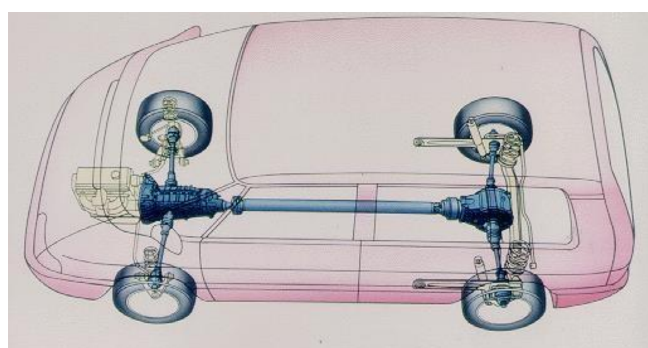


Fig 1.1 :Propeller shaft

1.2. PURPOSE OF THE PROPELLER SHAFT

- It must transmit torque from the transmission to the differential gear box

- The drive shaft must also be capable of rotating at the very fast speed required by the vehicle.
- The drives shaft must also operate through constantly changing the angles between the transmission, the differential and the axels.
- The length of the drive shaft must also be capable of changing while transmitting torque.
- The drive shaft should provide a smooth, uninterrupted flow of power to the axles.

1.3.FUNCTIONS OF THE DRIVE SHAFT

- First, it must transmit torque from the transmission to the differential gear box.
- During the operation, it is necessary to transmit maximum low-gear torque developed by the engine.
- The drive shaft must also be capable of rotation at the very fast speeds required by the vehicle.
- The drive shaft must also operate through constantly changing angles between the transmission, the differential and the axles. As the rear wheels roll over bumps in the road, the differential and the axle move up and down. This movement changes the angle between the transmission and the differential.

1.4. DEMERITS OF A CONVENTIONAL DRIVE SHAFT

- They have less specific modulus and strength
- Increased weight
- Conventional steel drive shafts are usually manufactured in two pieces to increase the fundamental bending natural frequency because the bending natural frequency of a shaft is inversely proportional to the square of beam length and proportional to the square root of specific modulus. Therefore the steel drive shaft is made in two sections connected by a support structure, bearings and U-joints and hence overall weight of assembly will be more.
- Its corrosion resistance is less as compared with composite materials.
- Steel drive shafts have less damping capacity.

2. PROJECT OBJECTIVE& METHODLOGY

2.1. OBJECTIVE

The power transmission system is the system which causes movement of vehicles by transferring the torque produced by the engines to the wheels after some modifications. The transfer and modification system of vehicles is called as power transmission system. The power transmission system of vehicles consists of several components which encounter unfortunate failures. These failures may be attributed to material faults, material processing faults, manufacturing and design faults, etc.Maintenance faults and user originated faults may also responsible. Propeller shaft and the universal joints form the important links that help in transmitting power from the engine to the wheels. In this study, analysis is being performed on the universal joint yoke and the propeller shaft. In the universal joint yoke, certain modifications are made in the existing geometry and 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.

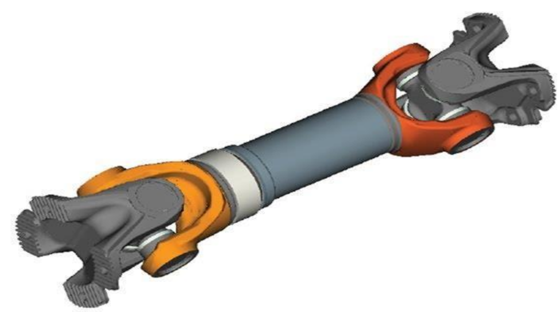


Fig2.1: solid model of propeller shaft

2.2. PROJECT METHOLOGY

The model (Universal joint yoke) was analysed in ANSYS considering the component to be made up of AISI 4063, which is a material in the low alloy steel group. The typical mechanical properties are1338 Mpa as ultimate tensile strength and 1476Mpa as the yield strength. A force applied at the spider mounting location is a torsional moment of 200Nm.In addition the given torsional moment, a 500rmp speed was also considered. The material as well as the loading conditions of torsional moment and the rotational speed is kept the same. The model was analysed in ANSYS12. However there were some changes made in the geometry of the existing model.

3. EXPERIMENTAL DETAILS

3.1.Introduction To CATIA

Computer aided three dimensional interactive applications as high end CAD/CAE/CAM tool used worldwide. Catia v5 is developed by Dassault Systems. France is a completely re-engineered next generation family of CAD/CAM/CAE software solutions for product lifecycle management. Through its exceptionally easy to use state of the art user interface CATIA V5 delivers innovative technologies for maximum productivity and creativity from concept to the final product.

CATIA V reduces the learning curve as it allows the flexibility of using feature based and parametric designs. CATIA V5 provides three basic platforms – P1, P2 and P3. P1 is for small and medium sized process oriented companies which wish to grow towards the large scale digitized product definition. P2 is for the advanced design engineering companies that require product, process and resources modeling. P3 is for the high-end design application and is basically for automotive and aerospace industry where high equality surfacing or Class-A surfacing is used for designing. The subject of interpretability offered by CATIA V5 includes receiving legacy data from the other CAD systems and even between its own product data management modules. The real benefit is that the links remain associative. As a result any changes made to this external data are notified and the model can be updated quickly. CATIA V5 serves the basic tasks by providing different workbenches. A workbench is defined as a specific environment consisting of a set of tools which allows the user to perform specific design tasks in a particular area.

3.2.Introduction To Finite Element Analsys

The finite element analysis (finite element method) is a numerical technique for finding approximate solutions of partial differential equations as well as of integral equations. The solution approach is based on either eliminating the differential equation completely (steady state problems) or rendering the partial differential equation into an approximating system of ordinary differential equations, which are then numerically integrated using standard techniques such as Euler's method, Runge- Kutta method etc. In the finite element method, a structure is broken down into many small simple blocks or elements. The behavior of an individual element can be described with a relatively simple set of equations. Just as the set of elements would be joined together to build the whole structure, the equations describing the behaviors of the individual elements are joined into an extremely large set of equations that describe the behavior of the whole structure.

3.2.1GENERAL PROCEDURE OF FEA

The following steps summarize the general procedure for finite element analysis.

STEP 1 - The continuum is a physical body, structure or solid being analyzed. Discretization may be simply described as process by which the given body is subdivided into equivalent system of finite elements.

STEP 2 - The selection of displacement or temperature models or shape functions representing approximately the actual distribution of the displacement or temperature.

The three factors which influence the selection of shape functions are a. The type and degree of displacement model b. Displacement magnitudes c. The requirements to be satisfied which ensuring correct solution.

STEP 3 - The derivation of the stiffness matrix which consists of the coefficients of the equilibrium equations derived from the geometric and material properties of the element. The stiffness relates the displacement at nodal points to applied forces at nodal points.

STEP 4 - Assembly of the algebraic equations for the overall discredited continuum includes the assembly of overall stiffness matrix for the entire body from individual element stiffness matrices and the overall global load vector from the elemental load vectors.

STEP 5 - The algebraic equations assembled in step 4 are solved for unknown displacements by imposing the boundary conditions. In linear equilibrium

problems, this is a relatively straightforward application of matrix algebra techniques.

STEP 6 - In this step, the element strains and stresses are computed from the nodal displacements that are already calculated from step 5.

3.2.2.Advantages and Limitations Of FEA

Planning the analysis is arguably the most important part of any analysis, as it helps to ensure the success of the simulation. Oddly enough, it is usually the one analysis leave out. The purpose of an FEA is to model the behavior of a structure under a system of loads. In order to do so, all influencing factors must be considered and determined whether their effects are considerable or negligible on the much dependent on the level of planning that has been carried out. FEA is an approximate way of simulation the system behavior. But the results can be quite close to actual testing values. FEA can never replace actual physical testing all the times. This is due to fact, the information required for FEA simulations like material properties emanates from physical testing. FEA results by themselves can never be taken as complete solution. Usually at least one prototype testing is necessary before the design guided/validated through FEA can be certified. But when effectively used FEA can predict the results/behavior quite close to reality and can reduce the design lead times as well as number of prototypes to be tested. Also there are some situations like gears in contact, which cannot be simulated exactly using FEA techniques. Under such conditions some work around such as simulating the worst conditions that can happen can be followed. Especially in situations like studying the behavior of a component by changing material, FEA can be highly handy as it is amounts to changing few numbers and re-running the analysis to know the component/system behavior.

3.2.3.APPLICATIONS OF FEA

- Structural engineering (analysis of frames, trusses, bridges etc).

- Aircraft engineering (analysis of aero plane wings, different parts of missiles and rockets).
- Heat engineering (analysis on temperature distribution, heat flux etc).
- Hydraulic and hydrodynamic engineering (analysis of viscous flow, potential and boundary layer flows).

3.2.4.POPULAR FEA SOFTWARES

There are varieties of commercial FEA software available over the market. No single software is supposed to have all the capabilities that can meet the complete simulation requirements of a design. Hence based upon the requirements, some of the firms use one or more FEA software. While some other firms develop their own customized versions of software.

Some of the popular commercially available FEA software are as follows.

- Adina
- Abaqus
- Ansys
- MSC/Nastran
- Cosmos
- NISA
- Marc
- Ls-Dyna
- MSC/Dytran
- Star-CD

3.3.Introduction to ANSYS

ANSYS is a general-purpose finite element-modeling package for numerically solving a wide variety of mechanical problems. These problems include: static/dynamic structural analysis (both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electro-magnetic problems. It enables engineers to perform the following tasks - build computer models or transfer cad models of structures, products, components or system, apply operating loads or other design performance conditions, study physical responses such as stress levels, temperature distributions or electromagnetic fields, optimize a design early in the development process to reduce production costs, carryout prototype testing in environment where it otherwise would be undesirable or impossible.

3.3.1.HISTORICAL DEVELOPMENT

Development of the finite element method closely parallels the timetable of the Development of the digital computer. Prior to the advent of the digital computer, work during the 1940's involved the approximation of continuous solids as a collection of line elements (bars and beams). However, due to the lack of computation tools, the number of line elements had to be kept to a minimum. The first appearance of two-dimensional elements appeared in a paper published in 1956 by The IJES Page 166 Turner, Clough, Martin, and Top . However, Clough did not use the term finite element until 1960 in a paper. The 1960's were an era in which most large corporations began installing mainframe computers. However, most finite element analysis work was done as a research exercise, rather than being part of the normal product design cycle. During the 1970's, several large general purpose finite element programs running on mainframe computers began to appear. However, due to the dependence on large computing facilities, finite element Analysis was generally used by only large corporations. Computer graphic displays were not prevalent until the late 1970's. This forced the pre-and postprocessing steps to rely on hardcopy graphical displays produced on plotters. This greatly increased the time required to perform the steps required in pre- and post-processing phases. During the 1980's, many finite element software packages were running on minicomputers along with highly interactive graphically oriented pre-and post-processors. The late 1980's and 1990's found many of these finite element packages being moved onto personal computers. However, even today, some finite element analysis is still done on large scale computers for problems which involve very large models, such as fluid flow computations, casting solidification and some non-linear Structural analysis.

3.3.2ADVANTAGES OF ANSYS

ANSYS provides a cost-effective way to explore the performance of products or processes in a virtual environment. This type of product development is termed virtual prototyping. With virtual prototyping techniques, users can iterate various scenarios to optimize the product life before the manufacturing is started. This enables a reduction in the level of risk, and in the cost of ineffective designs. The multifaceted nature of ANSYS also provides a means to ensure that users are able to see the effect of design on the whole behaviour of the product, be it electromagnetic, thermal, mechanicals etc .

3.4 DESIGN REQUIRMENT

3.4.1 Design data:

S.No	Parameter of Shaft	symbol	Values
1	Outer radius	Ro	0.036m
2	Inner radius	Ri	0.011m
3	Length	L	1.5m
4	Torque	T	151N

Table3.1: Design data's

3.4.2.Material properties:

Material property of shaft(conventional(steel s45))

S.No	Physical properties & mechanical	SYMBOL	Shaft
1	Density	ρ	7750kg/m3
2	Modulus of elasticity	E	201-209mpa
3	Tensile strength, ultimate	$Y_{ultimate}$	1338mpa
4	Tensile strength, yield	Y_{yield}	1476mpa

Table3.2: Material property of shaft(convention)

Material property of shaft(AISI4063)

S.N o	Physical properties & mechanical	symbo l	Shaft material	Shaft Material (modified)
1	Density	ρ	7750kg/m3	8280kg/m3
2	Modulus of elasticity	E	201-209mpa	195-210Gpa
3	Tensile strength, ultimate	$Y_{ultimate}$	1855mpa	1655-1862mpa
4	Tensile strength, yield	Yy_{ield}	1593mpa	1379-1551mpa

Table3.3: Material property of shaft (modified)

Material property of universal joint (low alloy steel AISI4053)

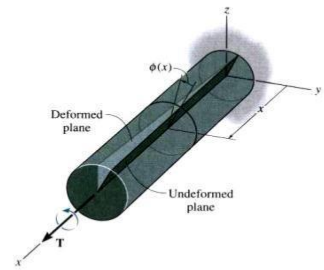
S.N o	Physical properties & mechanical Properties	symbol	Universal joint+-- Material	Universal joint Material
1	Density	ρ	7750kg/m3	7850kg/m3
2	Modulus of elasticity	E	201-209mpa	200Gpa
3	Tensile strength, ultimate	$Y_{ultimate}$	1724mpa	770-1525Mpa
4	Tensile strength, yield	Y_{yield}	1538mpa	442-1149Mpa

Table3.4.Material properties

4. ANALYSIS OF PROPELLER SHAFT

4.1.Theoretical analysis

The drive shaft for simplicity has been first idealized as a hollow cylindrical shaft which is fixed at one end and on other end which a torque of 151Nm is applied as represented below



The angle of twist φ(x) increases as x increases.

4.1.Shaft with torsional load

For the hollow shaft, Let

Ro = 0.036m ; Ri = 0.011m ; l = 1.5 m ;

E= 207e9 ; Torque = 151Nm

Where Ro-Outer Radius of shaft Ri- Inner Radius of shaft L= Length of the shaft

E= Young's modulus of steel (SM45C) T=Applied torque,

Then: Deflection = $Ymax = \frac{ML^2}{2EI}$

$$= 151\ X\ \frac{151\ X\ 1.5^2}{207E9\ X\ 1.178E6}$$

$$= 0.00069 = 0.69\text{mm}$$

Maximum deflection = $T * \frac{do}{2I} = 151 * \frac{0.036}{\frac{\Pi}{2}} * \frac{1}{[(0.036^4 - 0.011^4)]}$

$$= 66.50\ \text{Mpa}$$

Maximum shear stress = $\frac{T*RO}{j} = 20.78mpa$

4.2.PROPELLER SHAFT ANALYSIS using ANSYS 12.0

The propeller shaft was designed taking into consideration the torque to be transmitted by it. The Material utilized for the shaft as per the reference paper is AISI 4053(Material- 2) which is a alloy steel (oil quenched and tempered) with Boron as an alloying element. The tensile strength and the yield strength values are 1724Mpa and 1538Mpa respectively.

Considering the given loading conditions, the max stress value in the component when calculated analytically was found to be 24.14 Mpa using Von mises stress theory.The propeller shaft model when analysed in ANSYS using identical loading conditions as incase of the yoke, the maximum stress encountered(at the surface) using AISI4053 is 24.68Mpa which is approximately equal to the value calculated analytically. As far as the material strength is concerned, the maximum allowable stress in the component is found to be 428 Mpa.This allowable stress value is obtained as per the standard fatigue limit estimation process from the tensile stress (half of tensile strength with size correction factor). Thus we can conclude that the shaft of material-2 can perform satisfactorily under the given loading condiitions.

However, the above mentioned class of steel has a density of 7750Kg/m3. For the design dimensions utilized in the analysis, the mass of the shaft obtained is about 3.45 Kg. Hence, emphasis is given on reducing the weight of the propeller shaft,without affecting the dimensions of the component. Hence, we are able to significantly reduce the weight of the propeller shaft without making any changes in the original dimensions.

Elements used for Analysis and its characteristics

S.No	Generic element type name	Ansys Name	Description
1	20 Node Quadratic Hexahedron	Solid 95	20 node structural solid
2	20 Node Quadratic Hexahedron	Solid 186	20 Node structural solid
3	Quadratic Quadrilateral Contact	Conta 174	3D 8 Node surface to surface contact

Table4.2 element used for analysis and its characteristics

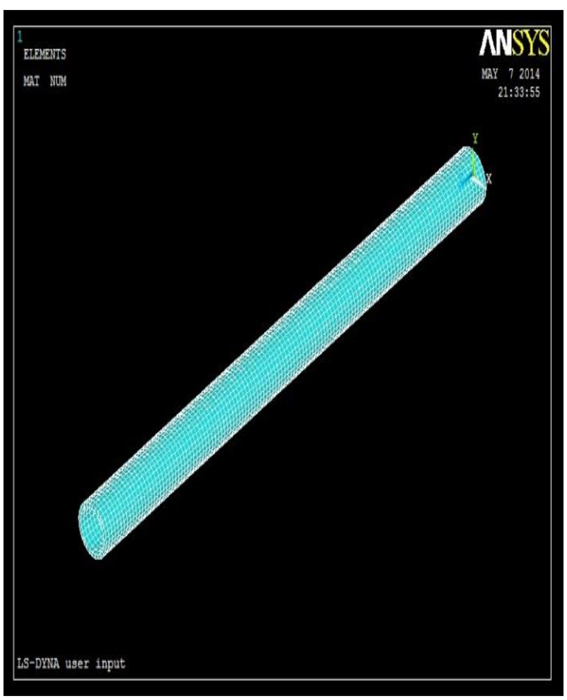


Fig 4.2 Mesh view of circular propeller shaft

NORMAL PROPELLER SHAFT ANALYSIS(STEEL S45)

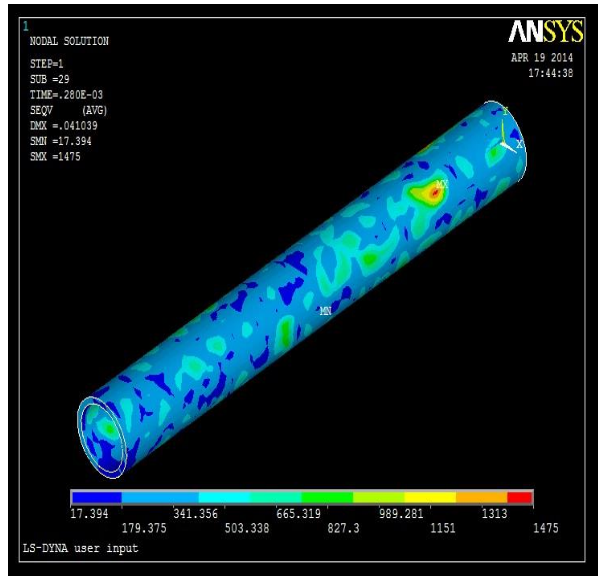


Fig 4.3 conventional propeller shaft

Modified propeller shaft (AISI4063)

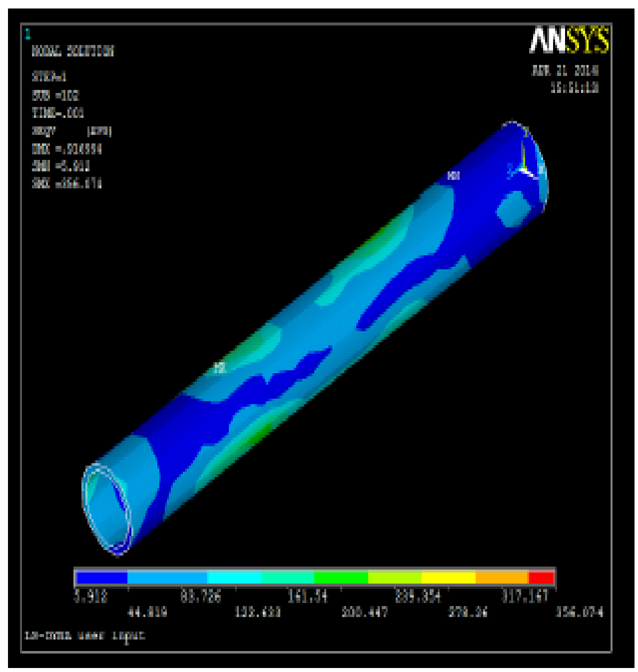
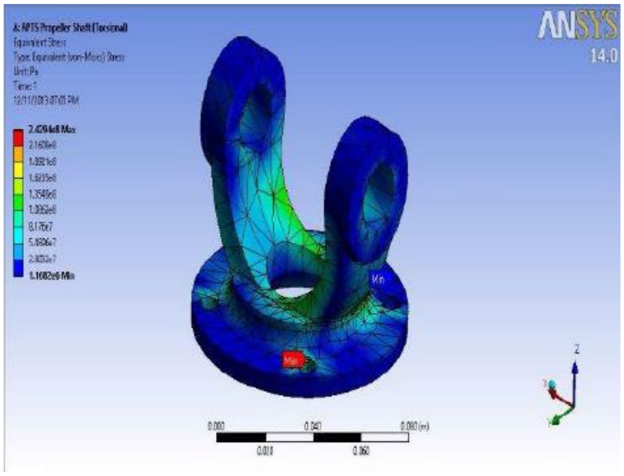


Fig 4.4.modified propeller shaft

4.3.UNIVERSAL JOINT YOKE ANALYSIS

Due to the introduction of the chamfer in the geometry, the weight of the component reduces, but we have also introduced a fillet, resulting in material addition. But, the resultant effect is reduction in the overall material content of the component.



The ANSYS analysis shows that the maximum stress value obtained is 242.94 Mpa, which is within the safety limits as prescribed in the paper. Moreover; this value is also less than the maximum stress value in the original component. Thus we can conclude that the new modelled component is well within the safe limit. Figure below shows the analysis of the universal joint yoke in ANSYS. Analysis was carried out for Identical loading conditions as in the reference paper.In design there are two main factors considered, the first being the maximum stress encountered in the component and the other being the area affected by this maximum stress. Due to introduction of the fillet the maximum stress encountered is reduced.

The point of maximum stress also shifts and the region affected by the maximum stress is significantly reduced. The maximum stress affected zone reduces to a point as shown in the fig 6.The stresses in the component even out leading to reasonably uniform stress distribution. Thus the modified geometry results in a component having lesser weight and at the same time is able to respond to the applied loading conditions.

5. ADVANTAGE OF MODIFICATION

5.1. Advantages

1. They have high specific modulus and strength.

2. Reduced weight.

3. Due to the weight reduction, fuel consumption will be reduced.

4. They have high damping capacity hence they produce less vibration and noise.

5. They have good corrosion resistance.

6. Greater torque capacity than steelS45 or aluminum shaft.

7. Longer fatigue life than steel or aluminum shaft.

6. RESULT

The result of analysis of propeller shaft

Analysis Type	**AISI 4063**	**Steel s45 conventional**	**AISI 4053 (universal joint)**
Total deformation ($x10^{-2}$mm)	0.27036	1.1981	0.565
Maximum principle elastic strain	6.117	1.1981	8.116
Maximum principle	1.4616	8.3599	2.665
Equivalent stress(Mpa)	1.3606	8.5473	5.6473

Table 6.1 Result of analysis

7. CONCLUSION

- The replacement of conventional drive shaft results in reduction in weight of automobile.
- The finite element analysis is used in this work to predict the deformation of shaft.
- The usage of low alloy steel materials has resulted in considerable amount of weight saving in the range of 81% to 72% when compared to conventional steel drive shaft.
- The present work was aimed at reducing the fuel consumption of the automobiles in particular or any machine, which employs drive shaft, in general. This was achieved by reducing the weight of the drive shaft with the use of low alloy steel materials.
- Natural frequency using Bernoulli – Euler and Timoshenko beam theories was compared. The frequency calculated by Bernoulli – Eule
- Is high because it neglects the effect of rotary inertia & transverse shear.

REFERENCES

[1] Advance in Polymer Composites: Macro- and Micro composites-State of the Art, New Challenges, and Opportunities. by Josmin p.jose, Sant Kumar malhotra, Sabu Thomas, Kuruvilla Joseph,Koichi Goda, and Meyyarappallil Sadasivan Sreekala.
[2] Polymer Matrix Composite in Driveline Applications-Dr Andrew Pollard,
[3] Static Torsion Capacity of a Hybrid Aluminum Glass Fiber Composite Hollow Shaft-S.A.Mutasher, B.B.Sahari and A.M.S. Hamouda, S.M.Sapuan.
[4] RAMURTHY, V.(1998), Computer-aided mechanical design and analysis,3rd Edition, Mc Graw-Hill, New York.