"Failure Analysis and Design Modification of Bracket of Center Bearing of Propeller Shaft of ST Bus" - A Review

Vishal U. Kamble
Mechanical Engineering dept.
Bapurao Deshmukh college of Engineering
Sewagram,Maharashtra,India

U. D. Gulhane
Mechanical Engineering dept.
Bapurao Deshmukh college of Engineering
Sewagram,Maharashtra,India

***Abstract*— The Propeller shaft is one of the most important components in transmitting power from the gearbox to the rear axle. Heavy duty vehicles like buses have their propeller shaft supported by bracket fixed to the chassis. The center bearing bracket plays an important role in overhanging the first half of the propeller shaft with ball bearing to the chassis. It also reduces the length of the propeller shaft between two hook (universal) joints. Center bearing bracket fatigue failure has been continuously a concern which may lead to operational failure of the propeller shaft which ultimately results in transmission failure. Some common causes of failure are manufacturing, design, maintenance, raw material and the operating conditions. Thus the aim of the project is to find out the probable reason of failure of bracket by analyze it with cad software and to provide a satisfactory solution to its failure by changing dimension. This paper research is divided in two parts first to conduct survey amongst the buses, examine, the causes of failure and second is to design and analyze and to recommend best possible alternatives of propeller shaft with the aid of advanced design tools like CAD. Propeller shaft failure is one of the major problems facing for MSRTC buses.**

This paper presents the available literature of failure analysis of a propeller shaft mounting bracket and analyzes the premature failure in Center Bearing Bracket. In the Analysis of the Center Bearing Bracket Creo elements / pro-e software is used for modeling and with the help of Ansys the stress and strain analysis were carried out. Based on the results of the existing Bracket dimensions, the new bracket is designed and found that the stresses are within the allowable range.

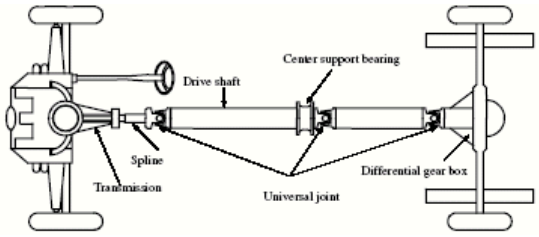
Keywords: Propeller shaft, Center Bearing Bracket, Transmission system, Failure analysis,

INTRODUCTION

In an automobile the power is developed in an I.C. engine after combustion of fuel in combustion chamber. In case of rear wheel drive or four wheel drive vehicle this power is to transmit to the driving wheels. An element propeller shaft is used to transmit the power to the rear axle. The requirement is that the propeller shaft should maintain it at proper position while it is rotating itself and vehicle propel on the road. It can only be possible with the use of center bearing.

It enables the smooth rotation of propeller shaft with negligible friction and less vibration. To document and are carry this center bearing along with the frame of the vehicle an component; known as 'Bracket' is required, which assemble the center bearing & frame together.

There are different forces and vibrations are transmitted to the bracket through the centre bearing .In spite to the bracket is provided with rubber bush to minimize the vibrations bracket fails to sustain the stresses and can be damaged.



Schematic arrangement of Underbody of an Automobile

Sachin Shelke et al[1] in their paper reveal that during power transmission from engine to the driving wheels through propeller shaft various forces act on the shaft and shaft carrying element known as bearings.The result is failure of bearing cup and which is identified after critical analysis of the drive shaft assembly. The work highlighted the methodology adopted for finalizing the solution to this problem by means of the FEA analysis supported by logical reasoning. Various Heat Treatment processes are compared and it was found that Carbonitriding process is the optimum solution which will reduce the failure of bearing cup as well as reduce the overall manufacturing cost.

The steering wheel along with steering system pays a vital role in controlling the direction of the vehicle by operating the steering wheel. The intermediate shaft connects the steering shaft to the steering pinion. These components cannot be arranged on the same axis due to the vehicle design limitations.

They are arranged with the universal joints. The stresses in either direction, while moving the vehicle to the right or to the left, happen to be a source of failure of the mechanical joint. Dhananjay S. Kolekar et al [2] in their work focused on various probable solutions to the problem of universal joint failure. They planned to model the yoke web using Finite Element Techniques. Mathematical evolutions suggested estimating responses to the significant parameters under observations They decided to set certain standards (ie benchmark) in sake of evaluating results towards validation.

Bhirud Pankaj Prakash et al [3] concentrated their study towards substituting composite structures for conventional metallic structures which have many advantages because of higher specific stiffness and strength of composite materials. In the recent days, there is a huge demand for a light weight material such as fiber reinforced polymer composites seems to be a promising solution to this arising demand. These materials have gained attention due to their applications in the field of automotive, aerospace, sports goods, medicines and household appliances. The overall objective of this work is to analyze a composite drive shaft for power transmission.

Substituting composite structures for conventional metallic structures has many advantages because of higher specific stiffness and strength of composite materials. This work deals with the replacement of conventional steel drive shafts with a Kevlar/epoxy or E glass polythene resin composite drive shaft for an automotive application. The intention of work is to minimize the weight of drive shaft. In this paper an attempt has been made to estimate the deflection, stresses, and natural frequencies under subjected loads using FEA (Ansys). Further comparison carried out for both loads using FEA. Further comparison carried out for both optimized and stress intensity factor found for both Steel and composite drive shafts.

Usually the propeller shaft is made up into two to three pieces. There are three universal joints to allow easy and smooth movement of propeller in it. The weight of all assembly add the forces and overall stresses on the propeller increases. By introducing propeller shaft with composite material we can reduce the propeller shaft into single piece and therefore overall weight of the assembly can be decrease.This can be achieved without losing strength of propeller shaft and no increments in overall cost. Amol S. Bhanage et al [4] in their study revealed that the forces and load act on propeller shaft are finally transmitted to the center bearing bracket. By calculating force value on propeller shaft the load on bracket can easily be calculated.

Automotive drive Shaft is a very important component of vehicle. The overall objective of this paper is to design and analyze a composite drive shaft for power transmission. Substituting composite structures for conventional metallic structures has many advantages because of higher specific stiffness and strength of composite materials. This work deals with the replacement of conventional two-piece steel drive shafts with Composite materials. In their work R. P. Kumar Rompicharla et al [5] used Kevlar /Epoxy as composite material the design parameters were optimized with the objective of minimizing the weight of composite 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, and natural frequencies under subjected loads using FEA. Further comparison carried out for both steel and composite materials and weight of the shaft is optimized and stress intensity factor found for both Steel and composite drive shafts.

Majority of the work in this regards is modeling of the components and trying with different materials. An attempt is also made to use composite material. Design optimization on the basis of strength of the material has also done and proved effective in reducing the failure.

In the present work actual testing of material has been done and the strength values are obtained through lab testing. Major thrust will be on modeling the actual load conditions and determining maximum stress values due to overloading and impact due to bad road conditions. These critical values will be used for designing modified bracket.

AIM & OBJECTIVES

1. Study analysis

The study about basic concept of the material for the manufacturing of bearing bracket and basics of design, etc.

2. Design Analysis

The design of existing bracket is prepared and then formulation of model with the help of software likes Pro-e or CATIA.

3. Stress analysis

As various forces acts over the bracket which produces the stresses causing failure will be analyzed by using ANSYS software.

SCOPE OF PROBLEM

When a S.T. bus propels on the road the different road conditions cause the production of stresses which will transmit to the bracket of center bearing. It is found that failure of bracket of center bearing is frequent problem faced by local ST workshop. The cost of replacing the bracket at an interval of two to three month is quite much. After discussing with engineers and supervisors it is reveal that the bracket failure is genuine problem.

RESEARCH METHODOLOGY

1) Conventional Analysis.
2) Computer Aided Analysis.
3) Computer Aided Design for Manufacturing and Optimization.

REFERENCES

[1] Sachin Shelke,Pradnya Kosbe,“Failure analysis of bearing cup”, vol.3 Issue 6,june 2014, International Journal of Innovative Research in Science, Engineering and Technology.
[2] Dhananjay S.Kolekar, Abhay M Kalje “Structural analysis of universal joint using finite element Methodology”vol.2 issue 3,International Journal of Scientific Research and Management Studies (IJSRMS)
[3] Bhirud Pankaj Prakash, Bimlesh Kumar Sinha “Analysis of Drive Shaft”, Volume- 2, Issue- 2, Feb.-2014,International journal of Mechanical and production Engineering.
[4] Amol S. Bhanage A, Vijay B. Patil and Rajat S. Patil “Finite Element Simulation of Automotive Drive Shaft Using Composite Materials”,International Journal of Current Engineering Technology.
[5] R.P. Kumar Rompicharla, Dr. K. Rambabu “Design and Optimization of Drive Shaft with Composite Materials”, Vol.2, Issue.5, Sep-Oct. 2012, International Journal of Modern Engineering Research (IJMER)