ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Design and Optimization of a Center Bearing Bracket Mount of a Propeller Shaft in BS-II Buses

Mr.Amara Raja Goud
PG Student,
Department of Mechanical Engineering,
Malla Reddy College of Engineering,
Secunderabad.

Ms.T.Nina
Assistant Professor,
Department of Mechanical Engineering,
Malla Reddy College of Engineering,
Secunderabad.

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 joints. Center Bearing Bracket fatigue failure has been continuously a concern which may lead to operational failure of the propeller shaft which ul timately results in transmission failure. Some common causes of failure are manufacturing, design, maintenance, raw material and the operating conditions. 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, Creo Elements, FEM, Ansys.

1. Introduction:

The Bracket is a supporting element for overhanging object. Automotive brackets come in all shapes and sizes for all types of uses.

A Propeller shaft is a rotating assembly used to transmit torque from the transmission gear box to the axle differential gear box. The Propeller shaft must operate through constantly changing the relative angles between the transmission and axle. It must also be capable of changing length while transmitting torque. This is accomplished through universal joint, which permit the propeller shaft to operate at different angles, and slip joints, which permit contraction and expansion of the axle to take place. Center or carrier bearings are designed to support long propeller shafts and to smooth out the delivery of power to the axle. The center bearing used on vehicles utilizes ball bearings with the bearing housing enclosed in rubber, further the centre joint bracket holds the center bearing in position to the chassis with bolt and nut assembly. Fig 1 shows a propeller shaft with two universal joints at the ends and a center Bearing Bracket assembly in the middle connecting two propeller shafts.

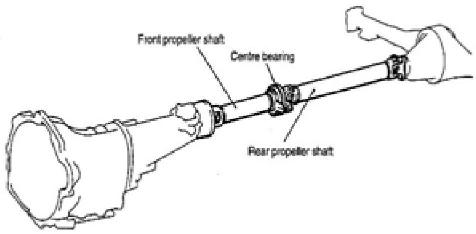

Fig.1 Propeller Shaft with Universal Hooke Joint and center bearing Assembly

1.1. Description of Center Bearing Bracket and its Assembly:

The Center Bearing Bracket is fixed to the chassis to support the ball bearing of propeller shaft. To reduce the shock loads on to the center bracket due to abused operating conditions the rubber bed is provided

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

between the ball bearing and the center bracket. As the rubber bed is highly elastic in nature whenever abused load comes on to the bracket it deforms them, again comes to its original state so that the rubber bed minimizes the load on the ball bearing and bracket. The severity of abused loads are not cushioned by the rubber bed completely there by the over stresses are occurring on the center joint bracket. The exploded view of the Center Bearing Bracket assembly is shown in fig 3.

Fig.2 Center Bearing Bracket Assembly

Fig.3 Exploded view of Center Bearing Bracket Assembly

2. Literature Review:

Failure Analysis is the process of collecting and analyzing data to determine the cause of 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 one must determine the cause of failure or prevent further 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. (Anupam Singhal and R.K. Mandloi, 2013) (Mc Evily and A.J, 2002)

The common causes of service failure are

1. Misuse or Abuse
2. Road condition
3. Environmental condition
4. Improper maintenance
5. Improper material
6. Poor storage condition (Halderman, et al, 2000)

The magnitude of the production volumes has traditionally placed severe requirements on the robustness of the processes used in manufacturing. The strong emphasis on the cost has demanded the component manufacturers to improve the performance of their materials and to find the methods to deliver these materials at reduced cost. The ride and noise characteristics of a vehicle are significantly affected by vibration transferred to the body through the mounting points from the engine and suspension. The automobile engine-chassis-body system may undergo undesirable vibrations due to disturbances from the road and the engine (Sahil Naghate and Sandeep Patil, 2012). In order to control the idle shake and the road-induced vibrations, the Center Bearing Bracket should be stiff and highly damped. Failure assessment of Center Bearing Bracket and consideration of the design can lead to optimal material usage without compromising safety.

3. Dimensions of the Existing Component:

Fig 4 shows the dimensions of the existing bracket that is used in the BS-II buses.

ISSN No: 2348-4845
International Journal & Magazine of Engineering,
Technology, Management and Research
A Peer Reviewed Open Access International Journal

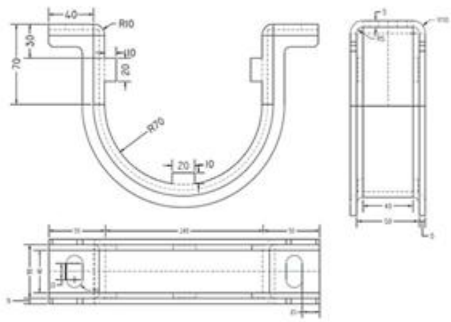

Fig.4 Figure Showing the Front View, Side View & Top View of the Existing Bracket

4. Methodology:

In this paper Static Structural Analysis of the Center Bearing Bracket of propeller shaft is carried out.The following are the steps involved in the Analysis of the Center Bearing Bracket.

1. Geometrical dimensions of the Center Bearing Bracket, of Bharat Stage-II TATA Bus Model are taken.
2. Geometrical solid model is prepared using the Creo Elements / Pro-e Software.
3.The solid model is imported into the Ansys 14 Analysis Software.
4. The material properties of the Bracket are specified and the material is modeled as Linear Isotropic Elastic material.
5. Bracket solid model is meshed using the pre tetrahedral mesh.
6. Pressure load is applied on the Bracket inner surface, in the vertical downward direction and the bracket top end surfaces of both sides are fixed in all directions.
7. The Static Structural Analysis is carried out using the Ansys Software.
8. The Equivalent von misses stress and total deformation results are observed using the post processor.
9. The material used for the Center Bearing Bracket is Fe 540 alloy. The properties of the material are listed below

Chemical composition by weight, % =0.20C, 1.60Mn, 0.040P, 0.040S, 0.45Si, 0.39CE Modulus of Elasticity, E = 200 GPa Mass Density, ρ = 7850 kg/m3
Yield Strength = 300 MPa
Tensile Strength = 410 MPa
10. The center bearing bracket solid model, meshed model and stress-strain results are shown below in figures 5 to 8.

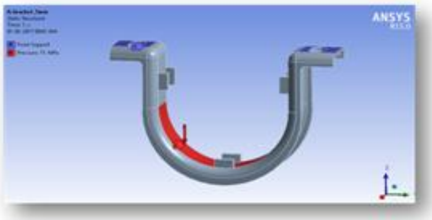

Fig.5 loading conditions with top ends fixed

Fig.6 Meshed model of the bracket

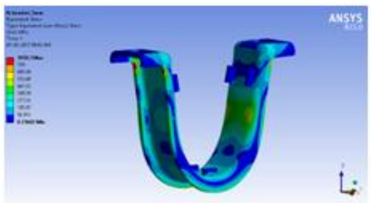

Fig.7 Von-misses stress plot

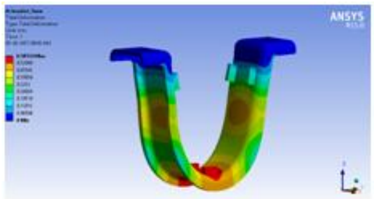

Fig.8 Deformation of Bracket

The failure regions of the existing Bracket are shown below.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Fig.9 crack at the fillet region in working condition

Fig.10 Figure showing some failed regions

In the figures (9&10) shown above the center bearing bracket failed at the junction of bracket flat end and semicircular end. It means that the stress concentration is more at the fillet radius. Similarly in the finite element analysis, the maximum stress is observed at the fillet radius as shown in fig (7). The stress observed at the fillet radius is 830.49 MPa which is very high. The load applied on the Center Bearing Bracket is estimated using the fatigue strength of the bracket by assuming the endurance limit as 0.5. The estimated load calculations are presented below.

5. Design Load Conditions:

Load = Endurance Strength of the material × Cross sectional Area of the Bracket

= 135 × 329.45 × 2

= 88951.5 N

Bearing Bracket Area = Diameter of the Bracket × Width of the Bracket

= 140 × 60

= 8400 mm2

Pressure on the bracket in the downward direction:

Pressure =Load /Area

= 88951.5 / 8400

= 10.58 MPa

= 11 MPa (approx)

The estimated Pressure of 11MPa is applied on the bracket in the vertical downward direction because, the numbers of Center Bearing Bracket failures were observed in the APSRTC, chittoor bus depot INDIA. It means that excessive loads are coming on to the bracket due to abused operating conditions. Normally due to the abused operating conditions certain amount of run-out takes place in the propeller shaft. The intensity of run-out increases gradually because of abuse operating conditions (driving condition & road condition) due to this stress on the bracket is applied cylindrically during the operation continuously. To overcome this bracket failure, the bracket is redesigned and finite element analysis is carried out for 6mm, 7mm, 8mm, and 9mm thicknesses. Figures 11 to 14 show the Center Bearing Bracket cross section for different thickness.

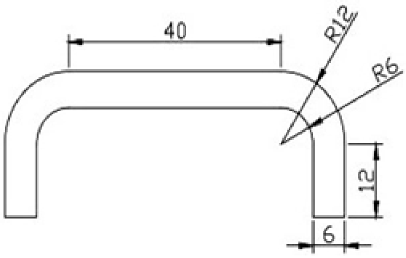

Fig.11: 6 mm thick cross-section

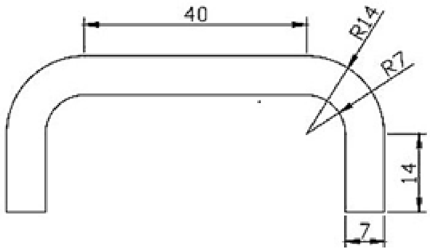

Fig.12: 7 mm thick cross-section

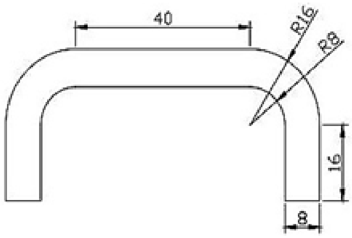

Fig.13: 8 mm thick cross-section

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

The stress-strain results for thickness ranging from 6 mm to 8 mm, with increase in steps of 1mm thickness are shown in figure 15 to 22.

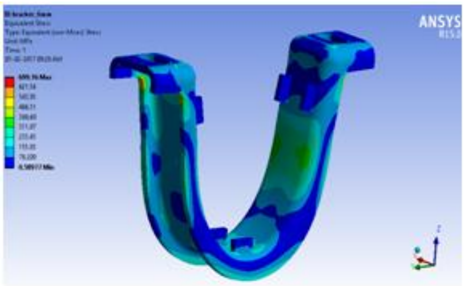

Fig.15 Stress distribution for 6 mm

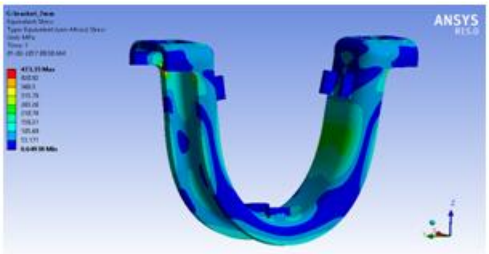

Fig.16 Stress distribution for 7 mm

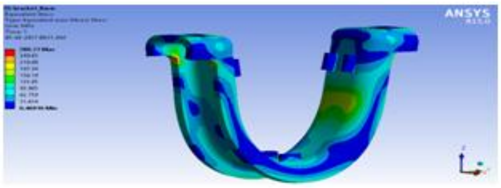

Fig.17 Stress distribution for 8 mm

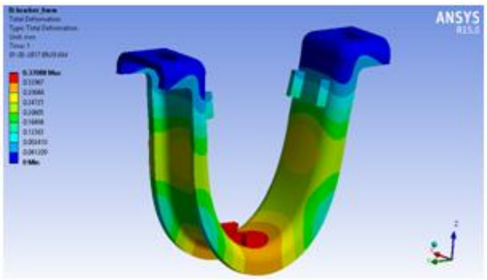

Fig.19 Total deformation for 6 mm

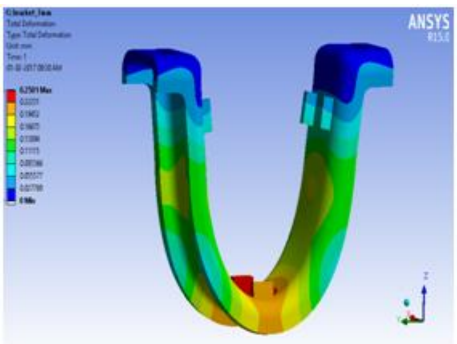

Fig.20 Total deformation for 7 mm

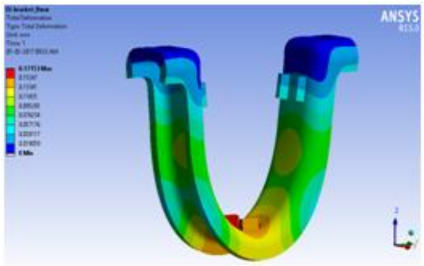

Fig.21 Total deformation for 8 mm

The results from the table show that the maximum stress at the fillet is less than the allowable stress i.e. 280 MPa for the bracket thickness of 8mm. The deformation is as less as 0.15 mm for 8mm thickness Bracket, as compared with the 5mm thickness bracket (existing Bracket). So, the Bracket would be safe if the design is changed to 8mm thickness. Further to reduce the stress on the Bracket, the dynamic analysis is to be carried out for various abused operating conditions and necessary modifications are to be done in the propeller shaft transmission system to minimize the stress on the Center Bearing Bracket.

Table 1: Table showing stress-strain values for different thicknesses

Thick	5mm	6mm	7mm	8mm
Mass	1.48 Kg	1.94 Kg	2.46 Kg	2.92 Kg
Total deformation	0.5855mm	0.3708mm	0.2500mm	0.1715mm
Equivalent stresses	1059 Mpa	699.9Mpa	473.77Mpa	280.68Mpa

6. Results & Discussion:

To the existing Bracket design the pressure of 11 MPa calculated, based on endurance strength is applied on the Bearing Bracket and the stresses corresponding to the pressure are noted. The Equivalent (Von-misses) stress from the analysis is obtained as 830.49 MPa which is beyond the yield stress of the material (Fe410) i.e. 240 MPa. The existing design is not safe as the failure occurs at the fillet region as shown in fig (7). Then the design is modified by altering the dimension of the cross section of the bracket to 6mm.The same pressure of 11MPa is applied on the bracket, even at this design condition the bracket is

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

getting failed at the fillet region with Equivalent (Von-misses) stress as 571.99 MPa as to the desired stress of 300 MPa. By changing the dimensions of the cross section of the bracket to 6mm, 7mm, applying the pressure of 11MPa the analysis is carried out where in each case the stress obtained are tabulated above in table 1 and the analysis results are shown in figures (15-22). In the cases of 6mm & 7mm thick as the Equivalent stresses are 699.77MPa and 473.68 MPa the Bracket is failed at the fillet region. At 8mm thickness the Bracket is with standing a load of 11 MPa with the Equivalent (Von- misses) stress of 280.4 MPa which is less than the yield stress of the material of 240 MPa stating that the design is safe. So, if the Bearing Bracket is designed with the cross section being 8mm thick then the Bracket will with stand the pressure that is exerted by the working condition, road condition of the Propeller shaft Bearing Bracket assembly.

Conclusion:

In the Bharat Stage-II Tata Busses, the propeller shaft Center Bearing Bracket failures are observed. The Center Bearing Bracket failure is observed mainly at the fillet region. Based on the endurance strength of the existing bracket the maximum stress of 1059.49 MPa is observed in the Finite Element Analysis of existing bracket. As the allowable maximum stress is 300 MPa the Finite Element Analysis is carried out for different thicknesses i.e. 6mm, 7mm, 8mm . The results show that at 8mm thickness the bracket design is safe because the maximum stress is 280.41 MPa which is less than allowable stress of 300 MPa. The Analysis clearly shows that the stresses applied on the bracket are high mainly due to abused operating conditions. To reduce the stress on the Bracket the dynamic analysis of the propeller shaft transmission system is to be done, to determine the cause of operating conditions which are exerting very high loads on the Bracket.

References:

1. Anupam Singhal, R.K. Mandloi, (2013), Failure Analysis of Automotive FWD Flexible Drive Shaft – a review, International Journal of Engineering research and applications, Vol.3, Issue 1, pp.577-580.

2. McEvily, A.J, (2002), Metal Failures: Mechanisms, Analysis, Prevention, wiley, New York, pp.303-307.

3. Halderman, James D, Mitchell, Chase d. (jr), (1996-2000) Automotive Chassis System, IInd Edition, Prentice-Hall Inc. pp.280-289.

4. Sahil Naghate, Sandeep Patil, (2012), Modal Analysis of Engine Mounting Bracket Using FEA, International Journal of engineering Research and Applications, Vol. 2, Issue4, pp.1973-1979.

5. Senthilnathan Subbiah, O.P.Singh, Srikanth K. Mohan, Arockia P. Jeyaraj, (2011), Effect of muffler mounting bracket designs on durability, Journal of Engineering Failure Analysis,18, pp. 1094-1107.

6. Bayrakceken, H, (2006), Failure analysis of an automobile differential pinion shaft, Engineering Failure Analysis 13 pp.1422-1428.

7. Balbirsingh R. Guron, Dr. D.V. Bhope, Prof. Y. L. Yenarkar, (2013), Finite Element Analysis of Cross Member Bracket of Truck Chassis, International organisation of scientific Research Journal of Engineering, Vol.3, Issue 3, pp 10-16.

8. Teo Han Fui, Roslan Abd. Rahman, December (2007), Statics and Dynamics Structural Analysis Of a 4.5 Ton Truck Chassis, jurnal Mekanikal, No.24, 56-67.