International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 10, October- 2014

Comparative Analysis of Heavy Vehicle Cross Member Bracket with Weight Reduction

Manda Vishala
PG Student
Dept. of Mech. Engineering,
Rural Engineering College,
Nizamabad T.S.India.

Advala. Sathish
PG Student
Dept. of Mech. Engineering
Chaitanya Bharathi.Inst.of Tech.
Hyderabad T.S.India

Polusani. Sampath Rao
Professor
Dept. of Mech. Engineering
Rural Engineering College
Nizamabad T.S.India

***Abstract*: A Bracket is a component used to support load or any attachments to structural element. It is fastened to the structural element by means of bolted joint or rivet joint. The bracket is subjected to different types of loading while the vehicle is in motion. One of the main common loading is static load which is assumed to be constant throughout the operation. This static load is main cause for the failure of bracket. To reduce the failures stress concentration needs to be reduced at vicinity of the joint hole of the bracket, as the hole is highly stressed region. The objective of the research work is study the static and dynamic behavior of the truck cross member bracket by changing the geometrical features and structural properties. The failure analysis of the cross member bracket is effected by design and analysis approach. The failure analysis has been carried out using standard FE tools. In this scenario FEA model has been generated for truck cross member with the specified quality criteria and analyzed for the optimized results. The FE model is provided with elemental properties, loads and boundary conditions.The stresses and displacements are noted. Modifications have been made to current bracket & these modifications have resulted in reduction in stress values leading to safe design. The final geometry of bracket weight is reduced to 62%.**

Key Words: Bracket, Hypermesh, FE model, displacement and stress.

I.INTRODUCTION

One critical failure is that of a cross member bracket in the heavy truck chassis. The chassis in the truck is the structural mainframe. The chassis frame configuration in a truck till today is designed on conventional lines. It has two main longitudinal members throughout the length of the truck and has cross members supporting throughout the length of two longitudinal members and are riveted in place as shown in fig.1. The cross section is tubular, channel or double channel type. For cars and lighter vehicles other types are available. The two longitudinal sections of the chassis are robust enough to carry the main play load. Based on the research and development planning, it is expected to achieve a better performance of truck cross member bracket. The cross member bracket plays a very important role in the automotive chassis system. The unbalanced forces produced from the dynamic conditions and which causes for the structure vibrations.

A review of the literature related to the design and analysis with a focus on vibration analysis of these mounting brackets is presented here. **G. Phani Sowjanya**[1] ***et al*** performed a study on vibration parameters to test the avionic equipment. For this suitable design of vibration fixture and analyzed by using finite element analysis and shows various sizes and shapes were suggested. Senthilnathan Subbiah[2] et al examine the failure analysis of muffler mounting brackets of three-wheeler vehicles. Results show high magnitude of stresses and strain energy at the weld location. Analysis of the design suggests that bracket was acting as a cantilever beam with one-plane welding mounted on the engine cradle. Modified design, though eliminated the above failure, shifted the failure mode to the bush-bracket region. Pavan B. Chaudhari[3] *et al.*, optimized the natural frequency of engine mount bracket by using three different lightweight materials by using finite element analysis. Selected materials were Aluminium (Al), Magnesium (Mg) and Cast Iron (CI) and this investigation suggested that Mg and Al both were preferred material for engine mounting brackets. Sahil Naghate[4] *et al.* used finite element analysis tool to analyze the engine mounting bracket using FEA Package. Materials selected were aluminum alloy and magnesium alloy. The results obtained from the static and dynamic analysis have shown that the magnesium is better than aluminum. Jeong Woo Chang[5] *et al.*, optimize the topology at the concept design stage where structural analysis methodology of compressor bracket was verified on the static and dynamic loading condition. It was analyzed that a new bracket would not fail during a vibration testing and these results were verified with a fabricated real sample under the durability condition. Doo-Ho Lee [8]*et al.*, bracket was modeled by solid elements and the compressor was represented by rigid masses. For simulation of the dynamics stresses in the durability test, the lumped mass method was used. Optimal shapes of the bracket were obtained by using MSC/NASTRAN. The verification tests were conducted on the workbench and in a vehicle. The optimized bracket verification tests were fulfilled. Test results showed that the developed optimization procedure of the bracket was valid in the complex real world.

The objective of the work is determining the shape of the truck cross member bracket based on static and dynamic analysis by using finite element method. Also study the static and dynamic behavior of the truck cross member bracket by changing the geometrical features and structural properties and develop a new cross member bracket using topography optimization. For fulfill the objectives of the work, modeling and analysis by using FEM has been carried out for the cross member as shown in fig.1

Fig.1: Cross member Bracket

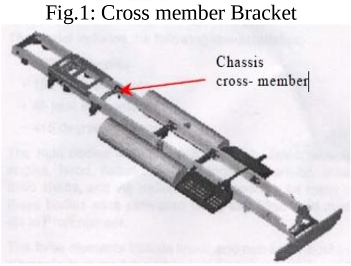

Fig.2: Parallel ladder type frame

II. DESIGN AND ANALYSIS METHODOLOGY

2.1 Specifications and Material data

The truck cross member bracket forms the backbone of the truck and its chief function is to safely carry the maximum load wherever the operation demands. Basically, it must absorb static engine weight, propeller shaft, centrifugal force and gyroscopic couple and absorb shock loads over twisting, pounding and uneven roadbeds when the vehicle moving along the road. Figure 2 shows a typical ladder frame chassis with cross member bracket for commercial vehicle. The bracket takes the load only in the direction of its width and in the outer end portion, with the rivets holding the central face fixed in place attached to the longitudinal member of the truck chassis. The loads acting on the bracket due to Propeller Shaft, Centrifugal Force, Gyroscopic Couple Static Engine Weight and all together around 9000 N exerts on each bracket when the truck moves with the noted speed of 60 km / hr. To resist these forces, the selection of materials is important criteria, which chosen from material library with following specifications.

Height = 82 mm, radius of fillet = 20 mm, thickness of barcket = 20 mm, width = 230 mm 10circular holes diameters = 10 mm, centre hole with varying crossection and diameter.

Properties of High Carbon Steel: Young's Modulus 197000Mpa, Poisson's Ratio 0.29, Density $7.9X10^{-9}$ Kg/mm^3, Yield strength 640Mpa

2.2 Modeling of cross member bracket

Modeling is done by commercially available packages Dassult system's CATIA software and the fig3 shows 2-d drawing. The FEA model consists of shell elements and the mesh is generated with the specified quality criteria. Further the FE model is provided with elemental properties, loads and boundary conditions. To analyze, finite element method (FEM) is used. Commercially available packages Dassault System's CATIA software is used for 3-d modeling, Preprocessing is done with Altair Hypermesh, Solution is obtained using Altair Optistruct and post processing is carried out using Altair Hyperview the steps are followed for modeling of the cross member bracket and obtain the model as shown in fig4.

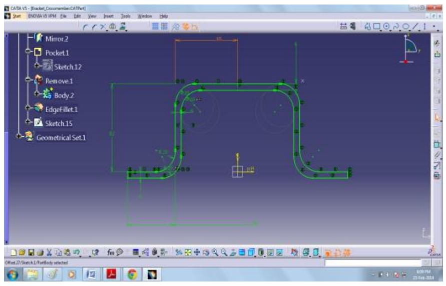

Fig. 3: Cross sectional 2-d drawing

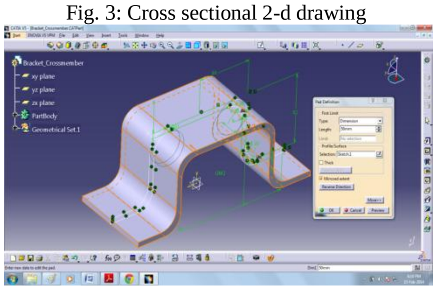

Fig.4: Model Cross member Bracket

2.3 Failure analysis of existing bracket

The presented work is design and analysis of cross member bracket in that view first study the failure behavior of a existing bracket, the actual failure location and stress failure analysis as

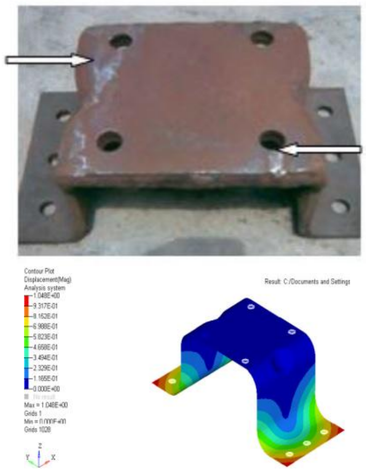

Fig5: Failure locations in original bracket and, Displacement in X-direction

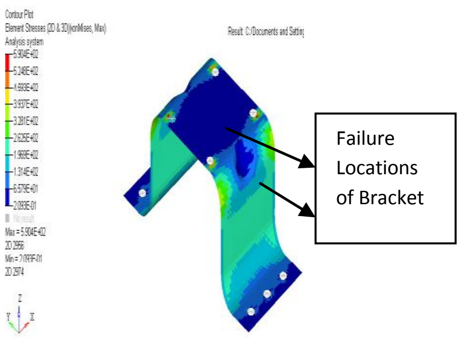

Fig.6: Stresses in the existing bracket and failure locations shown in fig.5 and 6. The maximum value of stress for the existing bracket is 590Mpa and it is generating at diagonal corners as that of actual test piece and these stress exceeding yield strength of material. Therefore component model is validated.

2.4 Analysis of redesign cross member bracket

Altair Hypermesh is a high-performance finite element pre-processor to prepare even the largest models, starting from import of CAD geometry to exporting an analysis run for various disciplines. Importing the CAD model in the Preprocessing Software and meshing is done for the model and property and material is assigned to the model and Static analysis for cross member bracket and dynamic analysis is carried out for cross member bracket. The analysis includes Stress Analysis of Reference cross member bracket, Dynamic Analysis of Reference cross member bracket, Topography Optimization analysis and Modified cross member bracket static and dynamic analysis. In this work analysis of bracket is made on basis of topography optimization with respect to the variation of geometrical features in different cases which as follows.

Case-I: Bracket with spherical pocket

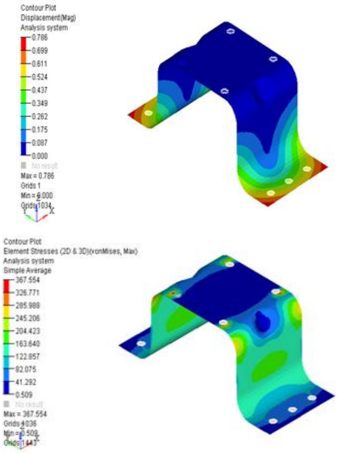

Fig.7 Displacement and Stress in bracket with spherical pocket

Case-II: Bracket without spherical pocket

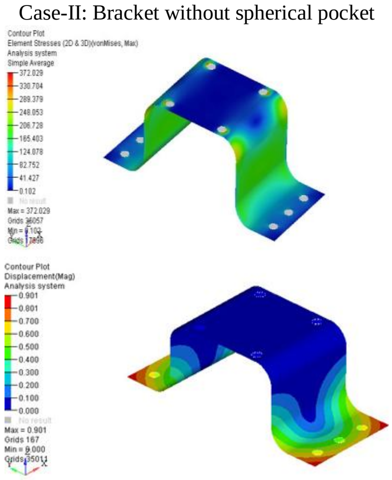

Fig.5 Displacement and Stress in bracket without spherical pocket

Case-III: Bracket with 30 mm diameter hole at base

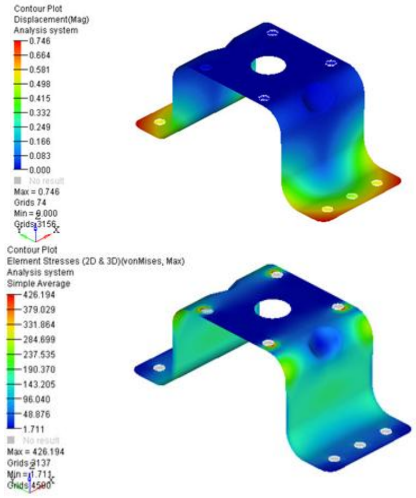

Fig.6: Displacement and Stress in bracket with 30mm diameter hole at base

Case-IV: Bracket with 50mm diameter hole at the base

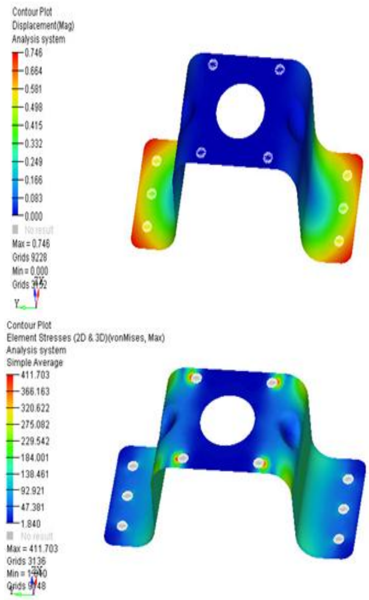

Fig.7: Displacement and Stress in bracket with 50mm diameter hole at base

Case-V: Bracket with 64mm diameter hole at base

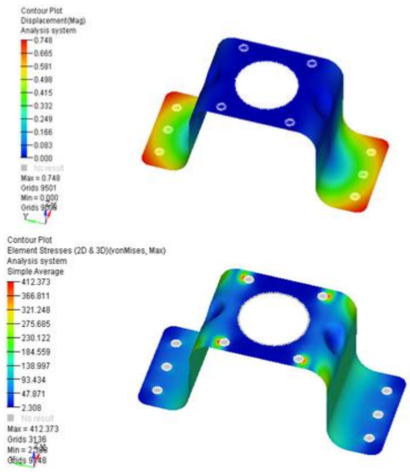

Fig.8: Displacement and Stress in bracket with 64 mm diameter hole at base

Case-VI: Bracket with rectangular bracket at base

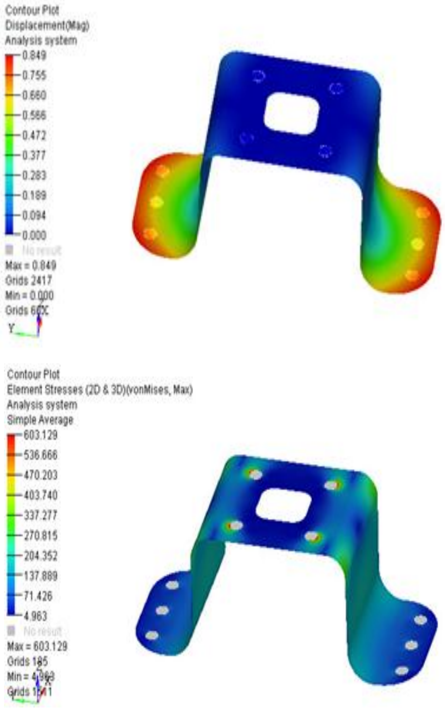

Fig.9: Displacement and Stress in bracket with rectangular pocket at base

Case-VII: Bracket with composite martial of 6mm thickness

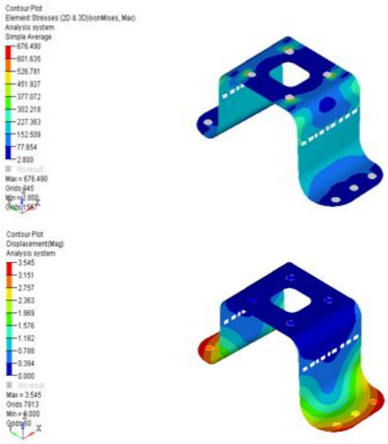

Fig.10: Stress and displacements in 6mm thickness composite material bracket

Case-VIII: Bracket with composite martial of 12mm thickness

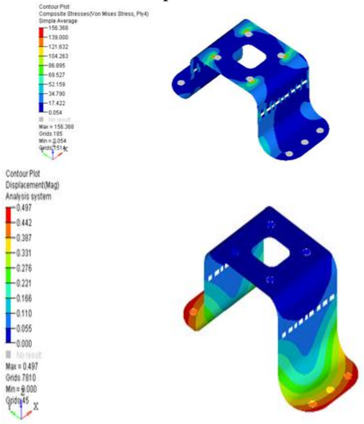

Fig.11 : Stress and displacements 12mm thickness bracket with composite material

III. RESULTS AND DISCUSSION

The presented work is design and analysis of cross member bracket for automotive applications based on the topography optimization. In this investigation the analysis has been carried out for cross member bracket with respect to variation of geometrical features for different models and final results are presented in table.1.The results shows comparison of displacement, stress and weight for different geometrical features.

The analysis of cross member bracket of model from case-I to case-VI for high carbon steel material and case-VII and VIII for composite material. The weight of the bracket is reduced by variation of geometrical features that is different sizes and shapes of hole at base of the bracket. From case-I and case-II it is clear that bracket with spherical pocket is showing less displacement and stress values than bracket without spherical pocket. So it is better to select bracket with spherical pocket. In the both cases it is clear that the reduction of weight is negligible. From case-III to case-V it is clearly shows that the reduction in bracket weight but stress and displacement is uneven, stress is maximum for hole diameter of 30 mm and displacement is maximum for 50 mm diameter hole at base of the bracket.The table1 shows the variation of stress, displacement and weight for different geometrical features.

Table 1: Results Comparison of cross member bracket analysis for different models

Case No.	Model Name	Displacement(mm)	Working Stress(Mpa)	Allowable stress (Mpa)	Weight (Kg)
1	Bracket with spherical pocket	0.786	367.554	420	1.671
2	Bracket without spherical pocket	0.901	372.029	420	1.670
3	Bracket with 30 mm diameter hole at base	0.746	426.192	420	1.632
4	Bracket with 50 mm diameter hole at base	0.849	411.701	420	1.572
5	Bracket with 64 mm diameter hole at base	0.748	412.373	420	1.513
6	Bracket with 30mmX30mm Rectangular hole at base	0.849	603.129	420	1.592
7	Bracket with composite martial of 6 mm thickness	3.545	676.397	420	0.3173
8	Bracket with composite martial of 12mm thickness	0.497	156.733	420	0.6345

From the table 1 it is noted that bracket with composite material of 6 mm thickness showing less weight among all, bracket with composite material of 12 mm thickness showing less value of maximum displacements and stress among all. For same material comparison this analysis shows Bracket with 30 mm X 30 mm Rectangular hole at base is better design. Based on comparison of different materials bracket with composite martial of 6 mm thickness is better design.

IV. CONCLUSIONS

The presented research work is study the static and dynamic behavior of the truck cross member bracket by changing the geometrical features and structural properties. In this scenario FEA model has been generated for truck cross member with the specified quality criteria and analyzed for the optimized results. Static analysis are carried out iteratively for various cases of bracket and the following conclusions are enumerated

i. Carbon epoxy composite material of 12mm thickness is preferred than High Carbon steel of 6 mm thickness for fabricating bracket due to
   - Maximum displacement is reduced by 41%.
   - Maximum stress is reduced by 57%.
   - Weight is reduced by 62%.

ii. Among the high carbon steel Bracket fabricated, having 30mm, 50mm and 64mm hole at the base, it is observed that the last one gave optimum results.

iii. Redesign and analysis of the bracket reduced the displacements and weight considerably.

iv. Based on the design and analysis observe that good agreement between working and allowable stress

REFERENCES

[1] Mantere, Adams, "Finite element analysis based design of cross member beam for heavy truck". MSc programmes in Engineering , Lulea university, ISSN, pages 1402-1667, 2003.

[2] Senthilnathan Subbiah, O.P. Singh,"Effect of muffler mounting bracket designs on durability", Engineering Failure Analysis 18 (2011) 1094–1107.

[3] Pavan B. Chaudhari and Dr. D. R. Panchagade "Comparison of Magnesium, Aluminium and Cast Iron to obtain Optimum Frequency for Engine Bracket using Finite Element Analysis", IJERA Vol. 2, (2012), pp1016-1020.

[4] Sahil Naghate and Sandeep Patil (2012), "Modal Analysis of Engine Mounting Bracket Using FEA", International Journal of Engineering Research and Applications (IJERA), Vol. 2, 1973-1979.

[5] Jeong Woo Chang and Young Shin Lee "Topology optimization of compressor bracket", Journal of Mechanical Science and Technology, Vol. 22, (2008), pp1668-1676.

[6] N.K.Ingole, Dr. D.V. Bhope, " Stress Analysis of Tractor Trailer Chassis for Self Weight Reduction ", IJEST, Vol. 3, issue 9, Sept.2011, pp7218-7225.

[7] Cicek Karao lu, N. Sefa Kuralay "Stress Analysis of Truck Chassis with Riveted Joints Using FEM", Elements in Analysis and Design, Volume 38, Issue 12,oc2002,pp1115 – 1130.

[8] Doo-Ho Lee, Jeong-Woo Chang and Chan-Mook Kim "Optimal Shape Design of an Air-Conditioner Compressor Mounting Bracket in a Passenger Car", Vol 01, (2003), 1667–1672. ,

[9] Erke Wang, Thomas Nelson and RainerRauch "A comparison of all-Hexahedra and all Tetrahedral Finite Element Meshes for elastic analysis", Proceedings 4th of int. conference, 179-181. (1995)