IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 15, Issue 1 Ver. I (Jan. - Feb. 2018), PP 01-06
www.iosrjournals.org

Optimum Design of Automobile Chassis Bracket Based on Topography Optimization

Polusani.Sampath Rao [1]

[1]*Professor Department of Mechanical Engineering, Rural Engineering College, Nizamabad (Telangana), India.*

Abstract: *A Bracket is a component used to support load or any attachments to structural element for heavy vehicle. It 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 causes 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 behavior of the truck cross member bracket, failure analysis and reduction of weight 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 by using standard FE tools. In this investigation FEA model has been generated for truck cross member with the specified quality criteria and analyzed for the optimized results. The results show the reduction in stress values and leads to safe design. The final geometry of bracket weight is reduced to 61%.*

***Key Words*: *Bracket, High Carbon Steel , Carbon fiber Composite Material, Hypermesh, FE model, displacement and stress*.**

Date of Submission: 12 -11-2017 Date of acceptance: 08-01-2018

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 *et al* [1]** 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. **Pavan B. Chaudhari *et al.* [3],** 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. **Senthilnathan Subbiah et al. [4]** have developed durability tests on vehicles in the end-user environment to reduce failures and warranty costs in the end-user hands. 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. **Jeong Woo Chang *et al.* [5],** 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. **S. Irving et al.[6]** has investigated the fatigue performance of two different bracket connections for use in high-speed ocean craft. Constant amplitude, cyclic tests revealed that weld quality within the curved or nested insert has a profound effect upon the fatigue behavior. **Doo-Ho Lee *et al.* [8]** 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 objectives of the present work are the determine the static and dynamic mode shape of the truck cross member bracket by using static analysis testing, 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.2

Fig.1: structural frame with Cross member Bracket

Fig.2: Chassis Bracket

II. Research Methodology

2.1 Identification of Problem

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. For this work, the truck cross member is categorized under the ladder frame type chassis. Figure 1 shows a typical ladder frame chassis for commercial vehicle. A ladder frame can be considered structurally as grillages. The cross members function as a resistance to the shear forces and bending loads while the cross members give torsion rigidity to the frame. Most of the light commercial vehicle chassis have sturdy and box section steel frames, which provide this vertical and lateral strength and resistance to failure stresses at cross member bracket location. In this work Finite Element and other analysis were used to determine the characteristics of the truck chassis.

2.2 Modeling of cross member bracket

Modeling is done by commercially available packages Dassult system's CATIA software. 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) and 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 fig3

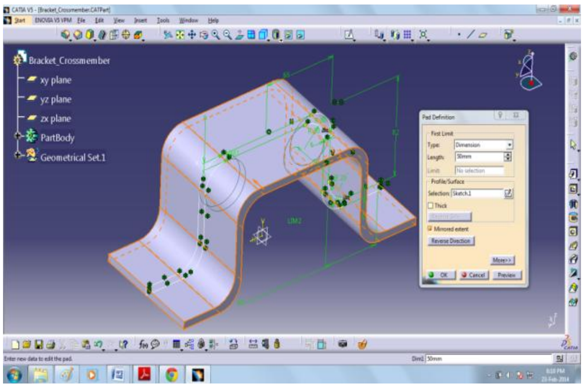

Fig.3: Model Cross member Bracket

2.3 Materials and Load analysis

The cross member bracket of automobile subjected to different loads in service, the selection of materials is important criteria, in this analysis chosen the material from material library that is High Carbon Steel with properties of Young's Modulus 197000Mpa, Density7.9X10^{-9}Kg/mm^3 , Poisson's Ratio of 0.29, Yield strength of 640MPa and composite material as Carbon fiber Composite Material. As the cross member is used as a support member, the bracket is loaded only vertically along its width. 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 9000N exerts on each bracket when the truck moves with the noted speed of 60 km / hr .

2.4 Analysis of 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, 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**

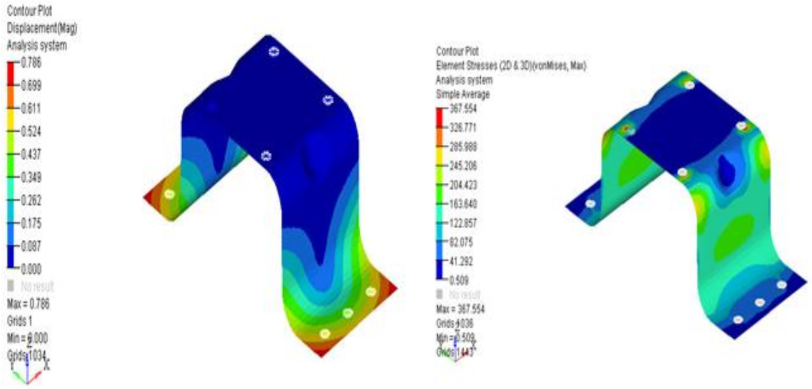

Fig.4: Displacement and Stress in bracket with spherical pocket

Case-II: Bracket without spherical pocket

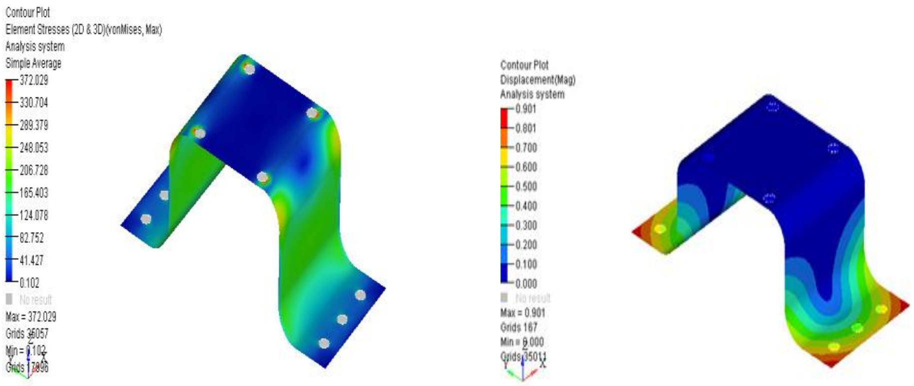

Fig.5: Displacement and Stress in bracket without spherical pocket

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

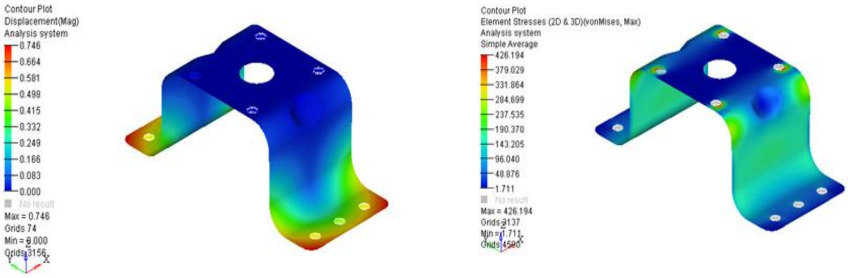

Fig.6: Displacement and Stress in bracket with 30mm diameter hole at base

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

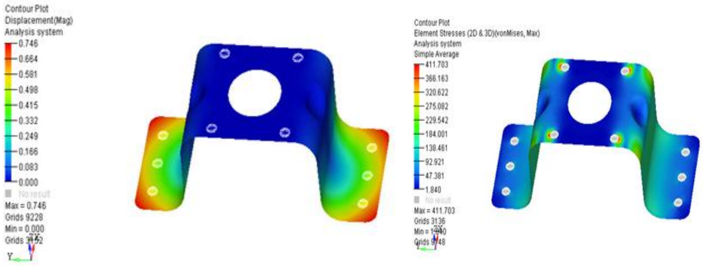

Fig.7(a): Displacement and Stress in bracket with 50 mm diameter hole at base

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

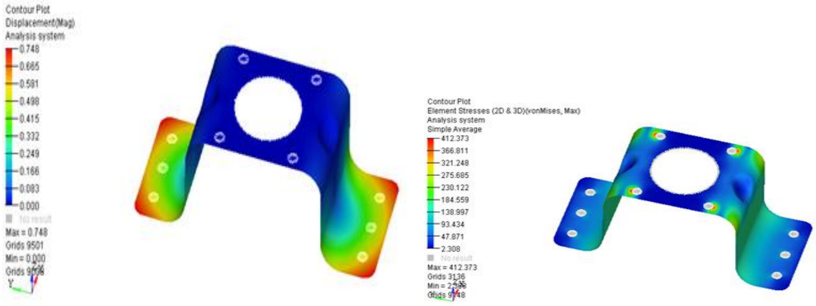

Fig.7(b): Displacement and Stress in bracket with 64 mm diameter hole at base

Case-VI: Bracket with rectangular bracket at base

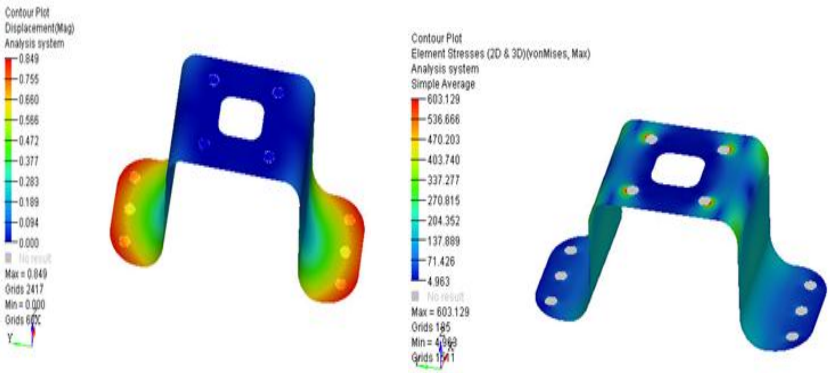

Fig.8: Displacement and Stress in bracket with rectangular pocket at base

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

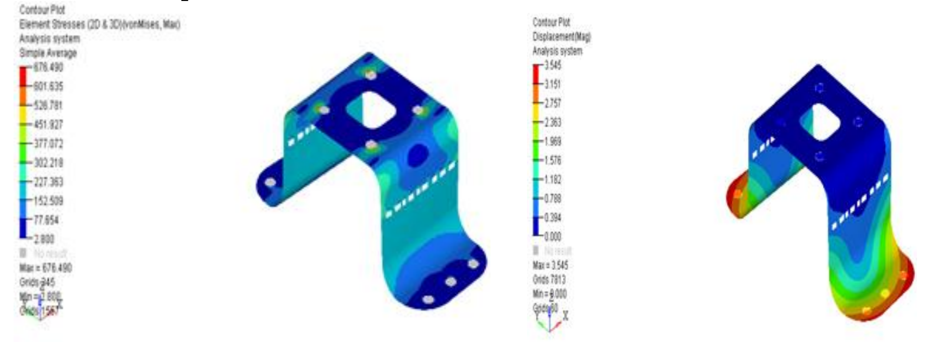

Fig.9: Stress and displacements in 6mm thickness composite material bracket

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

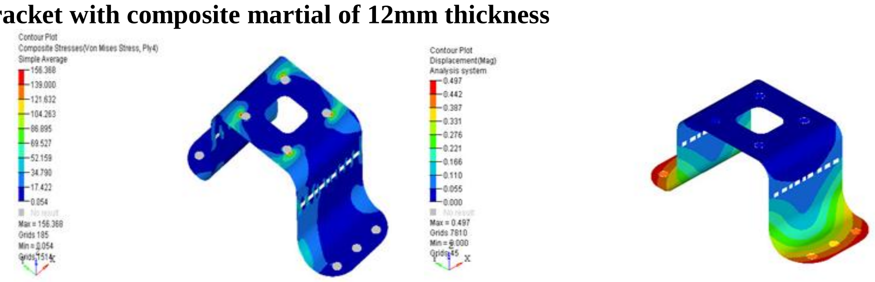

Fig.10 Stress and displacements 12mm thickness bracket with composite material

III. Results And Discussion

The presented work is Optimum Design of cross member bracket for automotive applications based on the topography optimization. In this investigation the design and 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. 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.

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

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

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 12 mm 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 53%.
- Weight is reduced by 61%.

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

iii. Provision of chamfers at outer corners of the bracket reduced the displacements and weight considerably.

iv. From presented work based on comparison of different materials bracket with composite martial of 6 mm thickness is better design.

References

[1]. **G. Phani Sowjanya, P. Divakara Rao and Dr. C.Udaya Kiran, "**Finite Element Analysis of Vibration Fixture Made of Aluminum and Magnesium Alloys**",** International Journal of Latest Trends in Engineering and Technology (IJLTET),Vol. 2,**(**2012), pp 84 – 89 ,

[2]. **Nouby M. Ghazaly, Sufyan Mohammed andAli M. Abd-El-Tawwab, "**International Journal of Engineering Research and Applications (IJERA)", Vol. 2,**(**2012), pp241 – 250.

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

[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]. **S. Irving , F.Ferguson-Smith, X.Z. Hu, Y. Liu,** "Comparative fatigue assessment of soft toe and nested bracket welded aluminium structures", Engineering Failure Analysis 12 (2005) 679–690.

[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 2002,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 Rainer Rauch** "A comparison of all-Hexahedra and all Tetrahedral Finite Element Meshes for elastic analysis", Proceedings 4th of international conference, 179-181. **(1995)**,

Polusani.Sampath Rao "Optimum Design of Automobile Chassis Bracket Based on Topography Optimization." IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) , vol. 15, no. 1, 2018, pp. 01-06