INTERNATIONAL RESEARCH JOURNAL OF ENGINEERING & APPLIED SCIENCES
ISSN: 2322-0821(O) VOLUME 7 ISSUE 4
ISSN: 2394-9910(P) Oct 2019 - Dec 2019
www.irjeas.org

# DESIGN AND STUDY OF AUTOMOTIVE CHASSIS BRACKET USING STRUCTURAL FINITE ELEMENT ANALYSIS

**Mohammad Faraz Rayeen[1], Arun Singh Patel[2]**

*[1] M.Tech Scholar, Department of Mechanical Engineering, NRI Institute of Information Science and Technology, Bhopal*
*[2]Asst. Professor and Head, Department of Mechanical Engineering, NRI Institute of Information Science and Technology, Bhopal*

***Abstract- Producing better quality, reducing the manufacturing cost and delivering the product quickly, have now become the major targets for manufacturing industry. FE simulation can play an important role in the integration of design and manufacture during the development of automobile parts. FEA model has been generated for different chassis parts with the specified quality criteria and analyzed for optimized results. Static analysis is carried out on bracket which is a integral part of the chassis.***

***Keywords – Automotive CATIA, Chasis, Design, FEA***

## I. INTRODUCTION

In an automotive vehicle, the engine rests on brackets which are connected to the main-frame or the skeleton of the car. Hence, during its operation, the undesired vibrations generated by the engine and road roughness can get directly transmitted to the frame through the brackets. This may cause discomfort to the passenger(s) or might even damage the chassis. When the operating frequency or disturbance approaches the natural frequency of a body, the amplitude of Vibrations gets magnified. This phenomenon is called as resonance. This magnification is most severe in low frequency ranges up to 50Hz. Also, at high operating frequencies noise becomes a serious concern. Hence, damping of these engine vibrations becomes an important function of the mount brackets. The Noise & Vibration Harness analysis (or NVH) is one the most important considerations in automotive designing today. If the brackets have their resonance frequencies close to the operating engine frequencies, then the large amplitude of vibration may cause its fatigue failure or breakage, thus reducing its estimated or desired life. Vibration damping can be either provided by using separate dampers (anti-vibration mounts) or by suitably deciding the material and dimensions of the brackets. Moreover, the brackets also undergo deflection under static and dynamic loads. This deflection should be under permissible limits. [1]
Automotive organizations are showing major interest in weight reduction of various components. Weight can be reduced by the introduction of new materials having better properties, better manufacturing processes. The specific strain energy capacity is greater in materials with lower modulus and density. As the composite materials have more elastic strain energy storage capacity and high strength-to-weight ratio as compared to steel. The weight of the bracket is reduced by introducing composite material in bumper without affecting load carrying capacity and stiffness. The FE analysis has been done for bracket deflection & stresses. Each bracket is attached to the longitudinal member of the chassis. The other cross member components on which the engine is mounted, is attached to flange of the bracket with six rivets. The height of the flange is 82 mm from the fixed central portion of the bracket plate which is fixed with the chassis. The thickness of the bracket plate is 6 mm. The overall dead load of about 0.5 ton of engine acts on the two crosses members which are mounted on the truck chassis frame. This load is then distributed by the cross member over each bracket [6].

A conventional process of developing a new sheet-metal product may include design of the part geometry, the binder face and addendum, the design of the tools, tool manufacturing/try-out, and manufacturing of the parts. The traditional development, in general, may be seen as a consecutive process. Contrarily, a concurrent design and manufacture process may include an integrated, parallel design of the product, the process and the tools, and tool try-out, with emphasis on teamwork [2,3]. The process could enable significant reduction of product cost and development lead-time.

The bumper bracket of a bumper having extension over the width of the vehicle body, on a back side of the bumper bracket has two deformation members mounted at a distance from one another facing the vehicle body. The deformation members offer more resistant to deformation than bumper bracket so that they deform only after bumper bracket is deformed.

The glass-fiber-reinforced plastic is used for the deformation members in the form of oval rings, the deformation member lengthwise central axis extending vertically, having one wall abutting the side member of the vehicle body and the opposite wall abutting the back of bumper bracket and being mounted releasably by bolts. The deformation member offers a unique property of not breaking when overloaded but of absorbing energy by de-lamination of individual fiber layer. The motive behind this property is that the damage caused due to the impact speeds above the design speed for the bumper can be shielded. The glass-fiber-reinforced plastic is used for manufacturing bumper bracket designed as a box girder with a closed hollow cross section which provides transverse reinforcement to absorb torsion force and transverse shear by endless fiber and fabrics. The foam can also be filled in the bumper bracket. The deformation member will be only deformed in case of minor collision, so that repair can be limited to replacing the deformation member only.

Austenitic stainless steels such as 310 and 304 are attractive as structural materials for load frame materials in automobile and transportation industries because of their high strength, toughness, formability and corrosion resistance which make automobiles lighter and corrosion resistant. Additionally, they are environmentally friendly material that is easy to recycle. However due to their high cost (because of high Ni content, 8–12 wt.%), these steels have been limited to applications where they would otherwise be an ideal choice, so attempts are being made to develop cheap stainless steels while still maintaining relatively high corrosion resistance. For such a reason, high Mn–N austenitic stainless steels replaced expensive Ni with low cost Mn and are actively being developed by many special steel companies like AK steel, Carpenter, Allegheny, POSCO. These steels contain 6–11 wt.% Mn (c-stabilizer) which is 7 to 8 times cheaper than Ni at an equivalent weight.

## II. LITERATURE SURVEY

A review of the literature related to the design and analysis with a focus on vibration analysis of these mounting brackets is presented here. **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.

**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.

**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.

## III. EXPERIMENTAL METHOD

Concurrent engineering (CE) for developing new sheet-metal parts suggests that the design of the part, the process and the tools, tool manufacturing and try-out, and the forming of the parts and inspection, may be carried out concurrently. With the help of FE simulation, amendment of the preliminary design of the product after the tool try-out can, almost, be avoided. More reliable forming-parameters, which would result in a product with the required quality, can be determined in an efficient way.

Part Design and process design

Part design and process design are two separated processes. The part design may be delivered by design engineers who may not have sufficient knowledge of forming processes. At the same time, a design that is functionally "optimal" may, however, require complex or expensive processes to produce it. The link of part design and process design is of particular significance for product development in metal forming, since a part-form (or component-form) and the quality of the products is largely prescribed by the forming process to be used. Properly selecting a process that is economically feasible and designing a feature that is practically achievable is an aim of implementing "concurrent design and manufacture" philosophy in sheet-metal product development [4].

**Static Structural Analysis**

A static structural analysis is the analysis displacements, stresses, strains and forces on structure or a component due to load application. The structures response and loads are assumed to vary slowly with respect to time. There are various types of loading that can be applied in this analysis which are externally applied forces and pressures, and temperatures. A static structural analysis can either be linear or non linear.

**Modal Analysis**

Modal analysis determines the vibration characteristics of a structure or a particular component in the form of natural frequencies and mode shapes. From this analysis we can do more detailed dynamic analysis such as transient dynamic analysis, harmonic analysis or spectrum analysis. The natural frequencies and mode shapes are important in the design of a structure for dynamic loading conditions. In this analysis, only linear behavior is valid. Damping is not considered and applied loads are ignored in modal analysis. A static structural analysis is required first for performing pre stressed modal analysis.

The analysis of many engineering and automotive components is done with the help of finite element techniques. These analysis are very helpful for approving or making some design changes during the post processing stage. The design changes mainly depend on the product life cycle and helps design engineers or the analysts to finales dimensions and material of the components. This analysis of an engine mounting bracket is done with the help of FEA software [5].

In routine run and loading, the cross member fails, as the chassis tends to flab in the central portion. This problem is solved by putting extra clamps over the chassis frame externally, before bringing the new vehicle on the road. Dampers provided by the truck manufacturer are removed off once the truck is in the field, for ease in the regular repair work, and because of increase in leaves in leaf spring set for overloading. The problem is compounded in the absence of the dampers in the suspension. Effective damping means less wear and tear of tyres , suspension components, and considerably lesser stresses in the chassis components. The damper provides additional cushioning which is very important. The damper support to absorb the extra load or impact is needed.

The front cross member under the engine rear is the most stressed and failure prone. It facilitates in proper handling, performance, effectiveness of the suspension and keeping the mountings and body panels etc. in alignment. It has to be strong enough not to yield due to the loads. It has to also resist torsional and bending loads [6].

This analysis determines the response to externally applied transient vibrations. Modal analysis gives us the form or mode shapes corresponding to the natural frequencies without considering any applied forces. The mode shapes obtained are important because they show us the direction of deflection and free amplitude of vibration for each natural frequency.

Since the brackets are pre-stressed the modal analysis needs to be done under this condition. Maximum amplitude of deflection under free vibration was checked for each design at each of the resonant frequencies lying in the operating frequency-range.

**Static Structural Analysis**

*Equivalent Stress* (*Von Mises Stress*)

While the Equivalent Stress at a point does not uniquely define the state of stress at that point, it provides adequate information to assess the safety of the design for many ductile materials. Unlike stress components, the Equivalent Stress has no direction. It is fully defined by magnitude with stress units. To calculate the factors of safety at different points, the Von Mises Yield Criterion is used, which states that a material starts to yield at a point when the Equivalent Stress reaches the yield strength of the material.

Equivalent stress is related to the principal stresses by the equation:

$$(S1-S2)^2 + (S2-S3)^2 + (S3-S1)^2 = 2Se^2 \qquad (1)$$

Equivalent stress is often used in design work because it allows any arbitrary three-dimensional stress state to be represented as a single positive stress value. Equivalent stress is part of the maximum equivalent stress failure theory used to predict yielding in a ductile material.

*Total Deformation*

Physical Deformations can be calculated on an inside a part or an assembly. Fixed supports prevent Deformation; locations without a fixed support usually experience deformation relative to the original location. Deformation is calculated relative to the part or assembly in world coordinate system.

$$U2 = (Ux^2 + Uy^2 + Uz^2) \qquad (2)$$

Ux, Uy and Uz are the three components of Deformation [7].

**Constraints**

Deciding the boundary conditions is very important in Finite Element Analysis. Here, the welded area is directly constrained or connected to rbe2 elements. Bolts are connected to the brackets using rbe2 rigid elements.

**Fatigue Analysis**

Since the brackets are subject to continuously varying loads, their fatigue life needs to be calculated. First the approximate average mean and amplitude stresses are calculated. Then from the calculated endurance limit, the no. of life cycles is determined. But due to the presence of weld joint on the chassis, the fatigue life reduces. Hence, the type and dimensions of the weld is suitably decided. Also to increase the weld area the holding bracket is given a curve according to the profile of the chassis member.

**Acceptance Criteria**

The maximum von-mises stress under worse loading condition obtained from the FEA results should be safely

International Research Journal of Engineering & Applied Sciences, IRJEAS
www.irjeas.org, ISSN(O): 2322-0821, ISSN(P): 2394-9910, Volume 7 Issue 4, Oct 2019-Dec 2019, Page 01-06

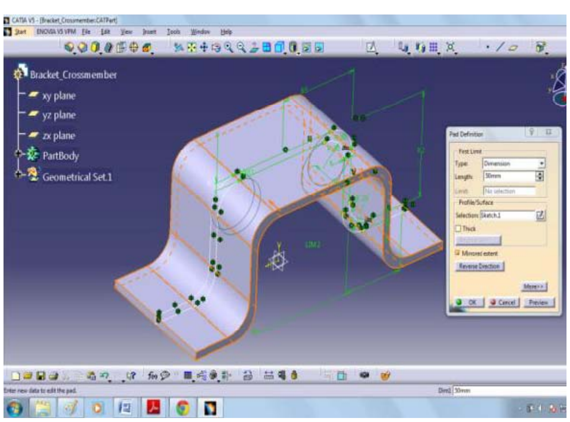
below the yield strength of the material. Because yielding is considered to be failure of the design, even if there might be no breakage. The maximum deflection is also noted down to compare the preliminary and final design [1].

## IV. METHODOLOGY

**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.

**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.

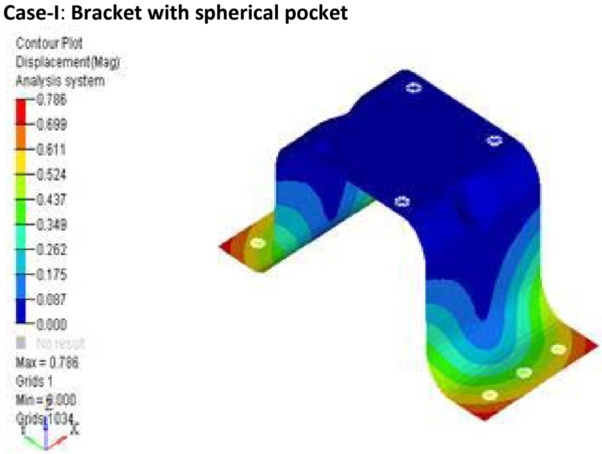


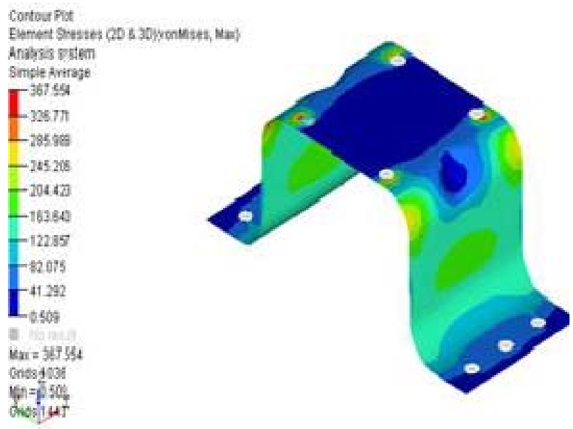

Model Cross member Bracket

**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, Density7.9X10-9Kg/mm3 , 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 .

**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.

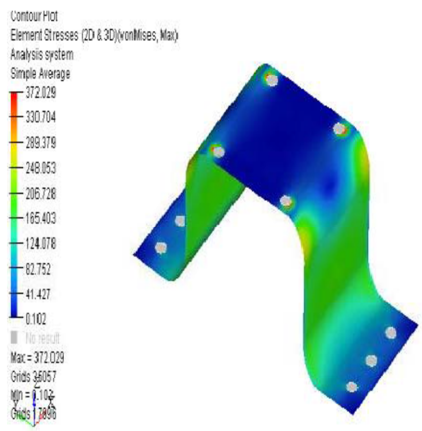
**Case-I**: **Bracket with spherical pocket**

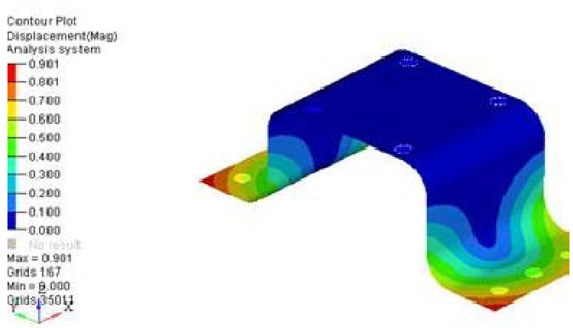





Displacement and Stress in bracket with spherical pocket

**Case-II: Bracket without spherical pocket**




Displacement and Stress in bracket without spherical pocket

## V. RESULTS AND DISCUSSION

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.

## CONCLUSION

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.

## REFERENCES

[1] Jasvir Singh Dhillon, Priyanka Rao & Vikram Sawant, Design of engine mount bracket for a FSAE car using Finite Element Analysis, Int. Journal of Engineering Research and Applications, ISSN : 2248-9622, Vol. 4, Issue 9( Version 6), September 2014, pp.74-81.

[2] T. Nakagawa, Recent manufacturing technologies for auto-body panel forming tools, J. Mater. Proc. Technol. 46 (1994) 277–290.

[3] D.C. Bayliss, et al., Concurrent engineering philosophy implemented using computer optimized design, Proc. Inst. Mech. Eng. Part B 209 (1995) 193–199.

[4] Y Liu, FE simulation for concurrent design and manufacturer of automotive sheet-metal parts,

Journal of Materials Processing Technology 150 (2004) 145–150.

[5] Sahil Naghate, Modal Analysis of Engine Mounting Bracket Using FEA, Vol. 2, Issue4, July-August 2012, pp.1973-197.

[6] Balbirsingh R. Guron, Finite Element Analysis of Cross Member Bracket of Truck Chassis, IOSR Journal of Engineering, Vol. 3, Issue 3 (Mar. 2013), pp 10-16.

[7] Rajesh Purohit, Design and Finite Element Analysis of an Automotive Clutch Assembly, Procedia, Materials Science 6 (2014 ) 490 – 502.

[8] Pavan B. Chaudhari & 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.

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

[11] 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.