© 2017 IJNRD | Volume 2, Issue 7 July 2017 | ISSN: 2456-4184

# DESIGN AND WEIGHT OPTIMIZATION OF CABIN MOUNTING BRACKET FOR HCV

**Mr. Rajkumar Ghadge[1], Mr. Pankaj Desle[2]**
[1] Department of Mechanical Engineering, RMDSSOE, Pune, Maharashtra, India, 411058
[2] Assistant Professor, Department of Mechanical Engineering, RMDSSOE, Pune, Maharashtra, India, 411058

***ABSTRACT: Automobile sector is one of the largest branch of mechanical engineering industry. It consumes a lot of fuel while transporting goods and people from one place to other by road. Reducing automobile weight for better economy is the challenge industry faces right now. Project work is focused on design and weight optimization of HCV truck's front Cab mounting bracket. Study will be focused on finding alternative design or material for cab mounting bracket of the truck.***

***Keywords –, Optimization, Bracket, ANSYS***

## I. INTRODUCTION

An Automobile is a self-propelled vehicle which is used for transportation of goods and passengers. The motor vehicles, both passengers' car and trucks are generally considered to be made up of two major assemblies: Body and Chassis. Chassis is a frame or main structure of a vehicle. The chassis contains all the major units necessary to propel the vehicle. Body is the super-structure of the vehicle. Body is bolted to the chassis. The chassis with the body make the complete vehicle. The truck consists of various assemblies performing their functions smoothly. Although there are many important parts, the cabin is a place where the driver and co-driver are seated. Their weight will be mainly on the floor where it should withstand many loads coming from different ways in different directions. This makes the driver seated without any vibrations and distractions. A flat sheet of thin material such as the floor panel is very flexible for out-of-plane loads. The aim of the floor is to carry the local applied loads from their point of application to the major structural components of the vehicle, such as the side frames. Floors are subject to loads normal to their plane. Under such circumstances they do not act as simple structural surfaces. The floor stiffened against out-of the plane loads by added beams arranged into a planar framework. The advantages of tilting cabin than rigid cabin is ease of servicing, less weight, easy for design modifications and provide fewer vibrations.

Growing competition of automotive market has made it more and more important to reduce time and cost of the product development process. One of the most costly phases in the vehicle development process is the field test and high expenses for this phase can sorted to the number of prototypes used and time/efforts needed for its execution. Also, multiple iterations during designing, building and prototype testing are not affordable against the time and cost constraints for developing a product. Today, analytical tools in the form of computer simulation have been developed to such a level that they reliably predict performance. Hard prototypes cannot be created in early design phase, but, today with the use of CAE virtual models can be created to accurately represent physical models and to take right decisions at the right time.

In Heavy Commercial Vehicle (HCV) cab mounting system is utilized to isolate driver from road generated vibrations. The vehicle cab is typically mounted on the chassis with the help of four supports. At the rear end it is mounted on Cab cross member center channel. Isolators are used to mount the cab on this center channel. The center channel is attached to the frame through cross member end bracket. The front side of the cab is mounted on the frame using an assembly of two brackets namely cab side mounting bracket and frame side cab mount. The cab side mounting bracket is joined to frame side cab mount using a bushing to provide vibration isolation. The frame side cab mount is bolted with the frame rail. The front cab mount also gives facility to tilt the cab for inspection/maintenance of the under-cab systems. The cab mounting system with its main components is shown in Figure 1.

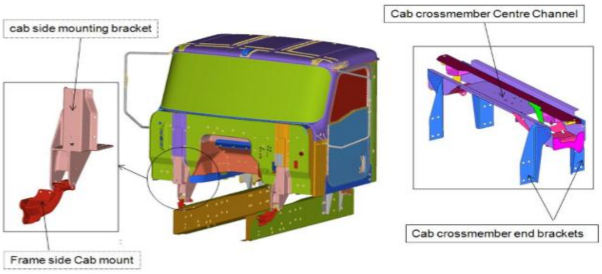


Fig. 1.Schematic Diagram of Cab Mounting System

## II. LITERATURE REVIEW

In the research paper "DURABILITY ANALYSIS OF HCV CHASSIS USING FPM APPROACH" published by Shaileshkadre, Shreyasshingavi and Manojpurohit focused on the durability analysis and FPM. In this research durability analysis was performed for historical loads such as inertial, racking and twists loads. Also, for component level analysis, full frame model was used to retain accuracy related to boundary conditions. In the analysis the forces obtained through road tests were effectively used to come out with durability load cases for the component level analysis. Structural analysis was performed followed by detailed strain based fatigue calculations with the help of HyperWorks Fatigue Process Manager (FPM), which uses strain - life method. It was observed that this approach produces realistic results with considerable cost savings on pre and post processing efforts as well as through reduced solution time. The simplified approach was used on mounting system of a truck assembly. All the CAE results were validated against the test data. Good test correlation was observed between test and FEA results. The results obtained in this methodology were also compared with the other commercially available FE solvers.

In paper "Study Concerning the Optimization of the Mounting System of the Truck Cab" published by Cornelia Stan, Daniel Iozsa and RazvanOprea present analytical study concern the optimization of mounting system of truck cab using finite element model. This model is useful to studies on the vibration transmissibility among different elements of the structure, in order to improve the comfort in the truck cab. analytical study is validate with the experimental study and conclude that optimization study of the component mounting systems, in order to find an optimal solution for, the elastic and damping characteristics can be easily changed in the model. The model is also useful in the body vibration behavior analyze during various working regimes as: acceleration, braking, passing over obstacle and turning on different types of road.

"Optimization Of Cabin Mounting" present by Richard Ambróz. This article describes the composition of the FEM model of anti-vibration mounting of tractor cabin. The task was find an suitable mathematical model which could really describe behaviour of anti-vibration mountings in static tests. The next target it was suggest a change of structural of ant vibration mounting and its effect on vibrations. The primary preliminary data are shape of anti-vibration mounting, its loading from cabin and material from which is made. The main part of my work was to form the model of anti-vibrations moan and find out the right setting. The mathematic model of rubber was Mooney-Rivlin model with two of parameters, which were got from the table available on internet. But this model is not right one for very real behaving of the rubber in high deformations. It would be better to use the model with five parameters. For finding these parameters is necessary to make the set of material measuring which is this anti-vibrations model made from or get the results of proving from the producing company. The next work will be form the dynamic model of anti-vibrations moan whereby the transmission of vibrations into the cabin would be possible to count. The static model of the anti-vibration moan will be used by counting the impact test of the cabin.

Paras Jain presents “Design and Analysis of a Tractor-Trailer Cabin Suspension”. In this paper the work done to overcome the ride problem of a Tractor-trailer vehicle. Ride of any vehicle can be improved by maintaining the low frequency of its suspension. The typical target frequencies for a car are 1-1.5 Hz whereas for a truck, it is 2-2.5 Hz. In heavy commercial vehicles, loading capacity is an USP, which do not allow softening of the suspension below a limit. A unique four point suspension has been designed to achieve the low ride frequencies of the cabin to improve the ride comfort. This paper describes some of the insight and knowledge gained from the effort to lead cabin suspension design for a heavy commercial vehicle. Initial ride evaluation and driver’s feedback has indicated the universal desire for great ride comfort. A significant improvement in ride can be achieved through the application of low frequency system at cabin mounting areas. The discussed methodology is very effective for improving the ride comfort of any commercial vehicle. This methodology can also be used for other passenger vehicles. Numerical simulation methodology discussed here can be used for predicting the dynamic behavior of the any vehicle in different conditions. ISO-2631 guidelines along with subjective evaluation give fair representation of ride in quality and quantitative manner.

“Design and Analysis of Tiltable Truck Cabin Floor” by D Murali Krishna. In this paper author focused on thetruck cabin to withstand loads obtaining from the roads, assemblies and the loads carrying. The floor should be designed to ensure fatigue life and should not fail in service to the instantaneous over-load. It should not pass the vibrations from the bulkheads and engine assembly. To meet the requirements, the floor is designed and analysis is carried out to investigate the behavior of the floor to the applied loads. By using CATIA and ANSYS software, the floor is designed and analysis is performed respectively and concludes that parameters considered for the floor are allowed to be used for the successful operation of the floor. Hence, the floor will operate safely and with ensured life. Hence the design is safe based on strength. From the Modal Analysis the fundamental frequency is obtained as 6.288Hz and Resonance condition exists. Resonance can be decreased by adding stiffness members like cross members etc. From Transient analysis, the maximum displacement is 0.3884mm and the velocity is 2.2839mm/sec. Transient displacements are quite high in the front portion.

## III. THEORETICAL ANALYSIS

**The following steps need to be followed for finding out the actual thickness needed for the bracket:**

1. <u>Calculating maximum bending moment</u>: In order to find out the section where the maximum bending stress occurs, we need to draw bending moment diagram. Let’s consider our case as a cantilever beam (though it is not a proper cantilever beam but for the initial analysis we are considering this as a cantilever beam). From the bending moment diagram we can easily point out the location of maximum bending moment and the value of the maximum bending moment is,

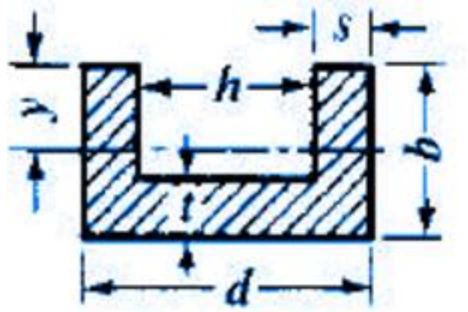


Fig: Cross section C shaped for Bracket

$$M = F \times s$$

2. <u>Calculating Moment of Inertia:</u>

$$I = \frac{b \times t^3}{12}$$

3. <u>Calculating perpendicular distance from the neutral axis :-</u>

$$y = \frac{t}{2}$$

4. <u>Calculating bending stress</u>: The classic formula for determining the bending stress in a member is:

$$\frac{\sigma}{y} = \frac{M}{I}$$

Where,
$\sigma$ - bending stress
$M$ - Moment at the neutral axis
$y$ - Perpendicular distance to the neutral axis
$I$ - area moment of inertia
$b$ - Width of the section being analyzed
$d$ - Depth of the section being analyzed

Let’s consider other notations as per image above and we have assumed that web thickness is constant for the section which means s = t in the equation.

$$\sigma = \frac{M \times y}{I}$$

The force acting on the mounting bracket is on the sideways, in case the vehicle meets accident. The force acting on the mounting bracket is assumed to be 3 times the load of the cabin, i.e. 1 ton. Therefore the load acting on the bracket is given as,

$$F = 750 \times 9.81\ N$$
$$F = 7357.5\ N$$

For steel SS304 which is one of the common steels used in automobile structural components and Channels tensile strength is 215 Mpa (Reference Matweb.com)
Assuming the Factor of Safety as 2,
The allowable yield stress is given as,

$$\sigma = \frac{215}{2} = 107.5\ MPa$$

The allowable shear stress is given as,

$$\tau = \frac{\sigma}{2} = \frac{71.66}{2} = 35.83\ MPa$$

5. <u>Calculating Shear Stress</u>: The shear stress experienced by the mounting bracket is calculated by the formula,

$$\tau = \frac{F}{A}\ MPa$$

6. **Calculating the Crushing/Compressive Stress:** The crushing stress is calculated by the following given formula,

$$\sigma c = \frac{F}{b \times d}\ MPa$$

Using the above formulae, the DOE sheet as shown in the below table,

Table: Design of Experiments for Steel Design

| Force | t | B | D | M | I | sigma | crushing | Shear Stress | weight | total stress |
|---|---|---|---|---|---|---|---|---|---|---|
| 7357.5 | 8 | 50 | 50 | 73575 | 2133 | 137.95 | 2.94 | 34.06 | 0.42 | 140.90 |
| 7357.5 | 8 | 60 | 60 | 73575 | 2560 | 114.96 | 2.04 | 34.06 | 0.51 | 117.00 |
| 7357.5 | 8 | 70 | 70 | 73575 | 2987 | 98.54 | 1.50 | 34.06 | 0.61 | 100.04 |
| 7357.5 | 9 | 50 | 60 | 73575 | 3038 | 109.00 | 2.45 | 34.06 | 0.50 | 111.45 |
| 7357.5 | 9 | 60 | 70 | 73575 | 3645 | 90.83 | 1.75 | 34.06 | 0.61 | 92.59 |
| 7357.5 | 9 | 70 | 50 | 73575 | 4253 | 77.86 | 2.10 | 34.06 | 0.61 | 79.96 |
| 7357.5 | 10 | 50 | 70 | 73575 | 4167 | 88.29 | 2.10 | 34.06 | 0.59 | 90.39 |
| 7357.5 | 10 | 60 | 50 | 73575 | 5000 | 73.58 | 2.45 | 34.06 | 0.59 | 76.03 |
| 7357.5 | 10 | 70 | 60 | 73575 | 5833 | 63.06 | 1.75 | 34.06 | 0.71 | 64.82 |

From the above DOE sheet, we will decide the best combination of the mounting bracket length, width, height and thickness. Then the model will be created and the analysis on the steel mounting bracket will be done. From above table it is quite evident that $8^{th}$ calculation row makes complete sense with stresses all within acceptable limit and weight of the bracket channel minimal of the all. So we will go for the dimensions from the $8^{th}$ row as the final dimensions for the steel column. Similar study can be performed on the GFRP materials.

But now the next step is to design the cabin mounting bracket which will be made of Glass Fiber (GFRP).

With reference to paper Tensile strength and fracture of glass fiber-reinforced plastic (GFRP) plate with an eccentrically located circular hole Xue Feng Yao a,∗, M.H. Kolstein b, F.S.K. Bijlaard b, Wei Xu a, ManQiong Xu we will take tensile strength of the GFRP to be 240 MPa. We will consider higher factor of safety for the GFRP components as GFRP component will fail at 240 MPa directly unlike steel would just cross its yield limit at its tensile stress limit which was considered for the design in the steel calculations. Considering factor of safety of highest order which is 5, we will consider thicknesses of the GFRP plates to be 7,8 and 9 mm in the design variables table and best design will be selected.

Assuming the Factor of Safety as 4,
The allowable yield stress is given as,

$$\sigma = \frac{240}{4} = 60\ MPa$$

Here is the design table for the same

Table 3.3: Design of Experiment for Composite Design

| Force | t | B | D | M | I | sigma | crushing | Shear Stress | weight | total stress |
|---|---|---|---|---|---|---|---|---|---|---|
| 7357.5 | 12 | 50 | 50 | 73575 | 7200 | 61.31 | 2.94 | 34.06 | 0.19 | 64.26 |
| 7357.5 | 12 | 60 | 60 | 73575 | 8640 | 51.09 | 2.04 | 34.06 | 0.23 | 53.14 |
| 7357.5 | 12 | 70 | 70 | 73575 | 10080 | 43.79 | 1.50 | 34.06 | 0.28 | 45.30 |
| 7357.5 | 13 | 50 | 60 | 73575 | 9154 | 52.24 | 2.45 | 34.06 | 0.22 | 54.70 |
| 7357.5 | 13 | 60 | 70 | 73575 | 10985 | 43.54 | 1.75 | 34.06 | 0.27 | 45.29 |
| 7357.5 | 13 | 70 | 50 | 73575 | 12816 | 37.32 | 2.10 | 34.06 | 0.27 | 39.42 |
| 7357.5 | 14 | 50 | 70 | 73575 | 11433 | 45.05 | 2.10 | 34.06 | 0.25 | 47.15 |
| 7357.5 | 14 | 60 | 50 | 73575 | 13720 | 37.54 | 2.45 | 34.06 | 0.25 | 39.99 |
| 7357.5 | 14 | 70 | 60 | 73575 | 16007 | 32.18 | 1.75 | 34.06 | 0.30 | 33.93 |

So for steel we select eighth and GFRP we will select fourth row of the design. As those options present the design iteration with least weight of the bracket and minimum stress for that weight combination, also stresses are within the acceptable limit. Theoretically weight reduction in the design can be calculated by the data from table as design weight of the C channel of steel is 590 grams while same design application using GFRP shows weight of 220 grams. This is 62.7 % of weight reduction.

## IV. FINITE ELEMENT ANALYSIS

The finite element method is a numerical technique for gaining an approximate answer to the problem by representing the object by an assembly of rods, plates, blocks, bricks. Finite element analysis is a method of solving, usually approximately, certain problems in engineering and science. It is used mainly for problems for which no exact solution, expressible in some mathematical form, is available. As such, it is a numerical rather than an analytical method. Methods of this type are needed because analytical methods cannot cope with the real, complicated problems that are met with in engineering. For example, engineering strength of materials or the mathematical theory of elasticity can be used to calculate analytically the stresses and strains in a bent beam, but neither will be very successful in finding out what is happening in part of a car suspension system during cornering. One of the first applications of FEA was, indeed, to find the stresses and strains in engineering components under load. FEA, when applied to any realistic model of an engineering component, requires an enormous amount of computation and the development of the method has depended on the availability of suitable digital computers for it to run on. The method is now applied to problems involving a wide range of phenomena, including vibrations, heat conduction, fluid mechanics and electrostatics, and a wide range of material properties, such as linear-elastic (Hookean) behavior and behavior involving deviation from Hooke's law (for example, plasticity or rubber-elasticity).

Model of the mounting bracket for engine is created using modelling software CATIA as shown in the image below.

**Results of Static Analysis on Bracket**

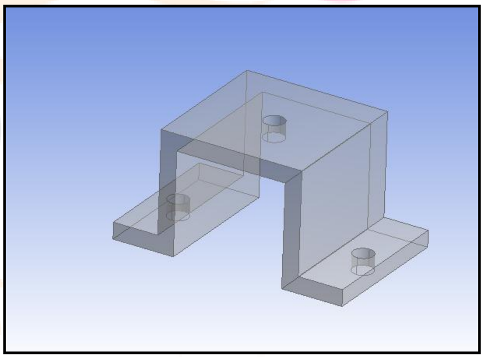

Fig. 8.CAD model of cabin mounting bracket

Mesh sensitivity study is performed on the results of the analysis, different mesh sizes are used and analysis results for the maximum stress at the bracket is observed. Mid surface of the bracket is extracted and shell 181 is used for analysis.

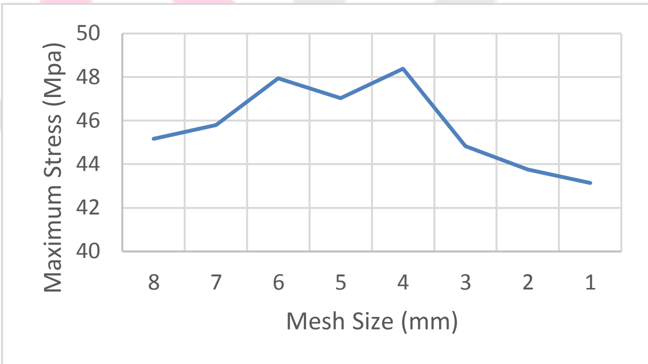


Fig.9.Graph of Maximum von Mises Stress vs mesh size used.

From mesh sensitivity study results 1 mm mesh size is finalized and meshing with 1 mm mesh size is shown in the figure below.

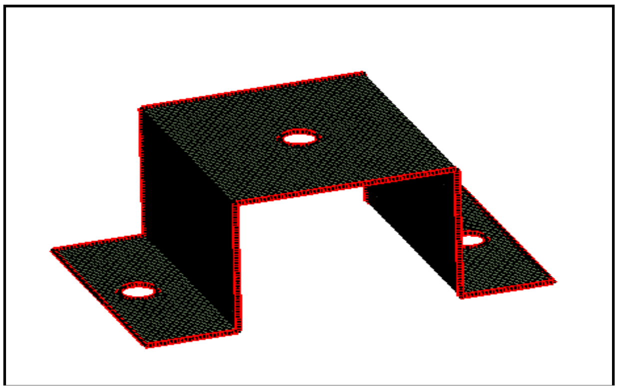

Fig.10. Meshing of baseline bracket steel

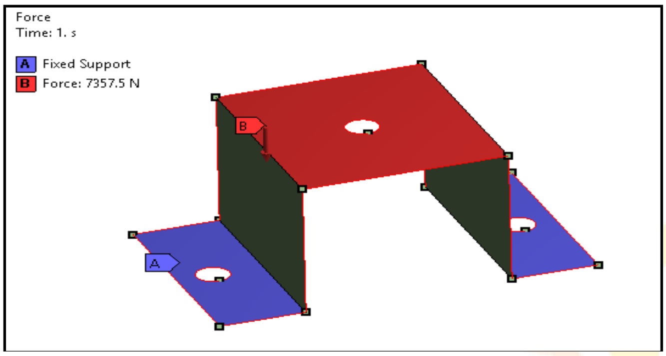


Fig.11. Fixed boundary condition applied on the bottom side of bracket and loading

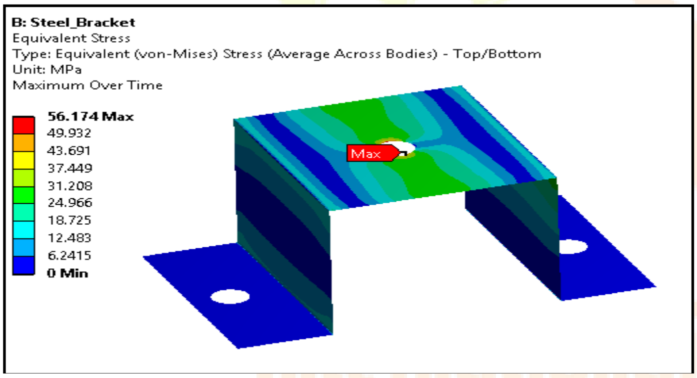


Fig.12.Equivalent Stress plot for Steel Bracket

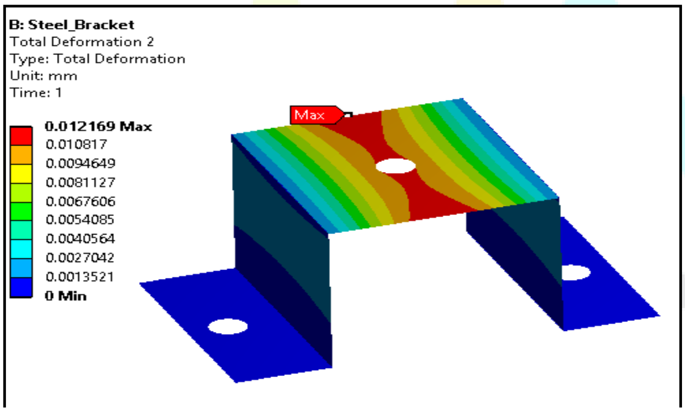


Fig.13.Total deformation plot for the bracket

It can be seen from the above results of the baseline module that maximum stress observed in the bracket is within acceptable limit so design is safe.

We also have designed the bracket for the same application using the GFRP composite for the weight optiimization. Finite Element Analysis on the same is performed same as on steel and results for that are shown below. Mesh size of1 mm is chosen as the preffered mesh size to keep similar analysis settings in both steel and composite bracket analysis.Shell 181 elements are used to mesh the model of composite bracket so that as a function of thickness composite material properties can be assigned to it. ACP ( Ansys Composite Processing) tool is used for applying composite properties to the bracket.

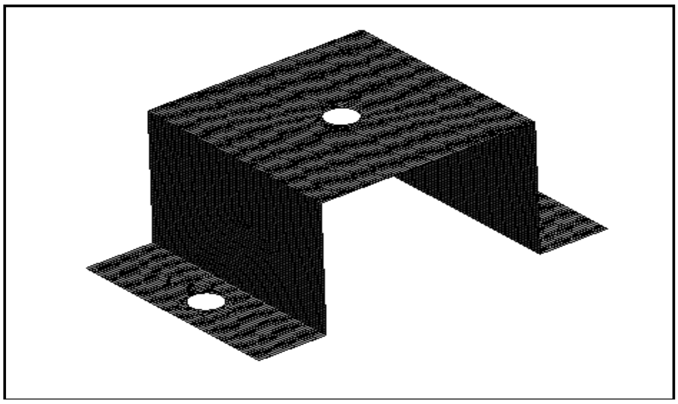

Fig.15.Meshing of baseline bracket steel

Two stack ups of different thicknesses applied to the elements in Composite pre-processing module of ansys. 60 % Epoxy and 40 % Glass fiber cloth are used for modelling the 1 mm thick composite bracket. Below are the two stack ups used on all the elements uniformly in the direction of thickness.

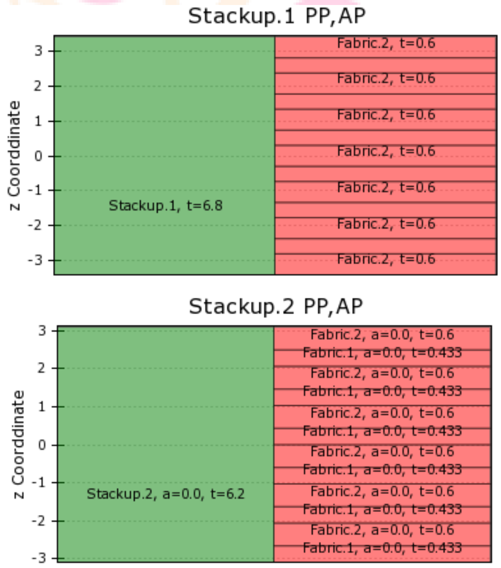


Fig.16.stack up plies sequences for stack up 1 and 2

Fabric 2 of thickness 0.6 mm is layer of epoxy resin and Fabric 1 of 0.433 mm is the layer of glass fiber woven cloth. 13 layers of epoxy resin and 12 layers of glass fiber cloth are used.

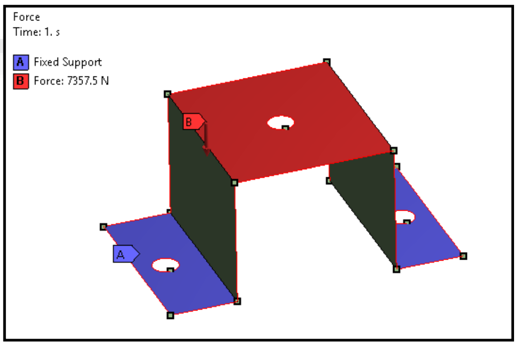


Fig.17.Boundary conditions on the composite bracket

Similar boundary conditions are the steel bracket FEA are applied to the GFRP bracket and analysis is performed to find out maximum deformations and von-mises stresses in the bracket due to loading.

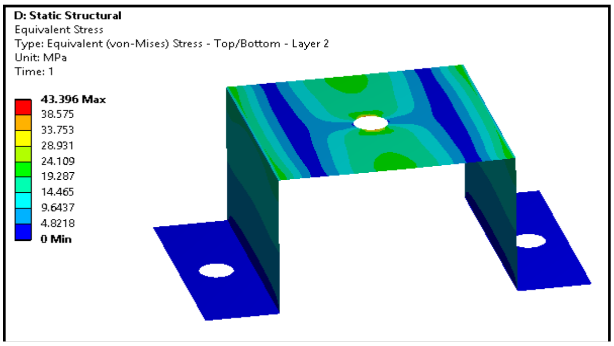

Fig.18.Maximum Equivalent Stress plot at outermost Glass fiber layer

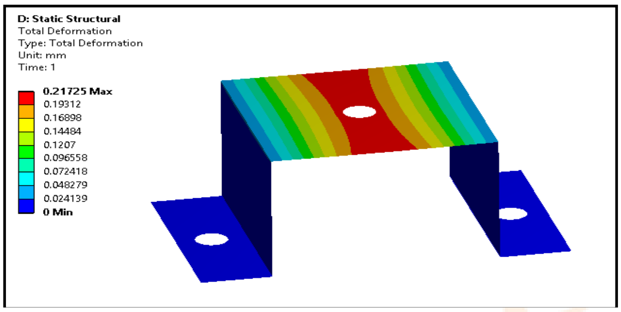

Fig.19.Total deformation plot for the bracket

It can be seen from the above results of the baseline module that maximum stress observed in the bracket is within acceptable limit and it is close to the value of maximum stress in analysis calculations.

## V. EXPERIMENTAL VERIFICATION

For experimental verification the mild steel bracket is manufactured as per dimension by using welding machine and then it's tested on UTM machine for compression loading.

For GFRP bracket is manufactured by design a mould for it. Conventional layup method of composite material is used to pour the material into the mould. After curing time of 24hours the part is removed from mould and drilled as per specification. Then compression loading has done for destructive test.


Fig.20.Compression loading of a steel bracket


Fig.21.Compression loading of a GFRP bracket

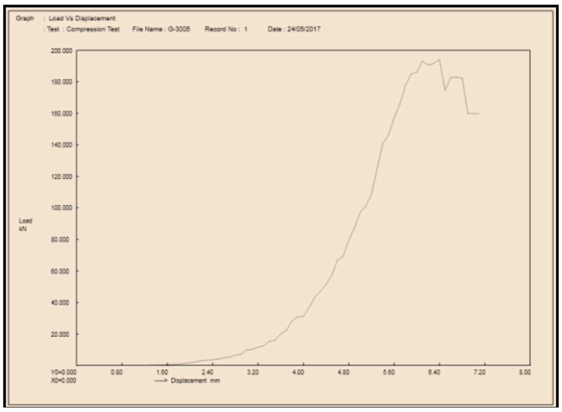
Fig.21.Load vs. Displacement plot for GFRP bracket

Loading taken by the GFRP bracket before breaking was around 175 KN which is more than the requirement of the loading which was much less.

## VI. RESULT AND DISCUSSION

Table.3. Result Summary FEA and calculations

| Material of Design | Steel | GFRP |
|---|---|---|
| von Mises Stress FEA (MPa) | 56.17 | 43.4 |
| Deformation (mm) | 0.012 | 0.217 |
| Mass of the component (grams) | 590 | 220 |
| % weight reduction | 0 | 63% |

Table above shows that weight of the component is reduced by 63% once we go for GFRP design option.

GFRP shows less stress and more deformation according to FEA results.

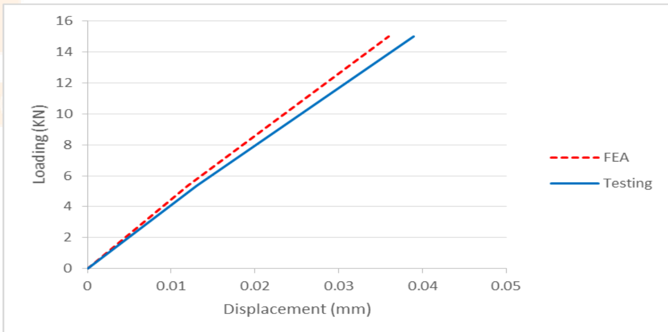

Fig. 22: Load vs. Displacement plot for Steel bracket

For steel bracket loading for the 15 ton loading on both cases shows no failure. In FEA maximum deformation observed at approximately 5 KN was 0.012 mm in actual testing 0.013 mm is observed which is well within the acceptable criteria.

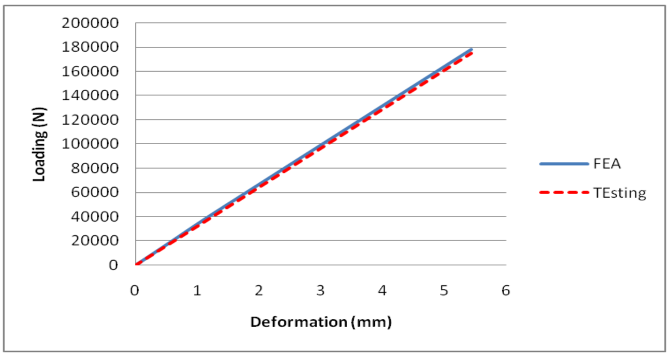

Fig.23.Load vs. Displacement plot for GFRP bracket

Comparison of static testing FEA and actual validation on UTM shows that maximum load taken by the component before breaking in UTM test is 175000 N and according to FEA it is to be 178000 N. Deformations in both the testing are comparable. Results of the above graph show us that FEA is validated by the testing and is in agreement with the results of testing with error of 1.71 %.

## VII. CONCLUSION

Design and analysis of the cabin mounting bracket for steel and GFRP is completed successfully. Total of 63 % weight reduced by designing the cabin mounting bracket using GFRP instead of conventional material. It is clear from the test results that both conventional and GFRP bracket are more than safe in the working load and breaking load of the GFRP material bracket is 175 KN. Weight of the conventional bracket is 590 grams which is reduced to 220 grams in the GFRP design cab mounting bracket.

## VIII. REFERENCES


**[1]** Shailesh Kadre, Shreyas Shingavi, Manoj Purohit, “Durability Analysis of HCV Chassis Using Fpm Approach.”HTC 2011, pp 1-9.

**[2]** Cornelia Stan, Daniel Iozsa, Razvan Oprea “Study Concerning the Optimization of the Mounting System of the Truck Cab”, ISBN: 978-960-474-383-4, pp 272-276.

**[3]** Promit Choudhury, Chinmaya Hemanth Krishna “Mathematical Modelling and Shape Optimisation of Front Damper Mount of Ashok Leyland 1612 Truck Using 3d Finite Element Method”. The International Journal Of Engineering And Science (IJES), Volume 3, Issue 4, April 2014, PP 70-76.

**[4]** Richard Ambroz, Ing. David Kollhammer, Ing. David Palousek. “OPTIMALIZATION OF CABIN MOUNTING” Conference of Diploma Thesis 2007 Institute of Machine design, Institute of Solid Mechanics, Mechatronics and Biomechanics, Faculty of Mechanical Engineering, Brno University of Technology June 5-6, 2007, Brno, Czech Republic

**[5]** Paras Jain, “Design and Analysis of a Tractor-Trailer Cabin Suspension”, SAE Paper No. 2007-26-047. PP 549-557.

**[6]** D. Murali Krishna, Dr. S. Madhu, “Design and Analysis of Tiltable Truck Cabin Floor”, International Journal of Research Studies in Science, Engineering and Technology Volume 2, Issue 7, July 2014, PP 69-73.

**[7]** Teo Han Fui, Roslan Abd. Rahman, “Statics And Dynamics Structural Analysis Of A 4.5 Ton Truck Chassis”, Jurnal Mekanikal December 2007, No. 24, PP 56–67.

**[8]** Xue Feng Yao, M.H. Kolstein b, F.S.K. Bijlaard b, Wei Xu a, ManQiong Xu, “Tensile strength and fracture of glass fiber-reinforced plastic (GFRP) plate with an eccentrically located circular hole”, Polymer Testing 22 (2003) PP 955–963.