International Journal of Advances in Engineering and Management (IJAEM)
Volume 2, Issue 4, pp: 81-86 www.ijaem.net ISSN: 2395-5252

Investigation and optimization of mechanical strength for cabin mounting bracket in commercial vehicle using ANSYS.

Santosh. P. Yadav[1], Prof. Dr. A. D. Desai[2], Prof. A. B. Bhane[3]
[1]PG Student. [2]Professor, [3]Asst. Professor of Department of Mechanical Engineering,
Shree Ramchandra College of Engineering, Lonikand, Maharashtra,India
Savitribai Phule Pune University, Pune
Corresponding author: Santosh. P. Yadav

Date of Submission: 30-07-2020 Date of Acceptance: 09-08-2020

ABSTRACT: Increasing competition and innovation in automobile sector tends to switch the prevailing products or replace old products by new and advanced material products. A mounting system of vehicle is also an area where these innovations are carried out regularly. Mounting brackets are widely used as suspension system & cabin mounting in automotive vehicles. In this project, mounting bracket design optimization will be performed by changing from conventional steel to composite material. They're widely utilized in the low weight applications and also as an alternate for metals to scale back the fabric cost. FEA analysis in Ansys will be carrying out by using ACP toll where actual composites layer modeling will be used to create fiber ply. Conventional steel, glass fibber material will be studied and among best will be recommended for future products. To validate FEA results, Compression testing of mounting bracket will be done on UTM machine. Static Analysis mounting bracket and will be compared with FEA results for validation. The developed program for mounting brackets may be useful for fast and reliable optimization of various parameters of mounting brackets in automotive vehicles. Consequently, there will be tremendous saving in material in mounting brackets manufacturing industries as a result of optimization.

Keywords: Mounting bracket,suspension system, composite material, optimization, conventional steel, glass fibber&UTM machine, etc.

I. INTRODUCTION

The cabin or cab of a truck is an indoor space during a truck where the driving force is seated. Modern long-haul trucks′ cabs usually feature air con, heater, an honest audio system, and ergonomic seats (often air-suspended). Automobile Body is the super-structure of the vehicle. Body is bolted to the chassis. 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.In Commercial Vehicle cab mounting system is supported to driver from road generated vibrations. The vehical cab mounting bracket is shown in Fig.1.

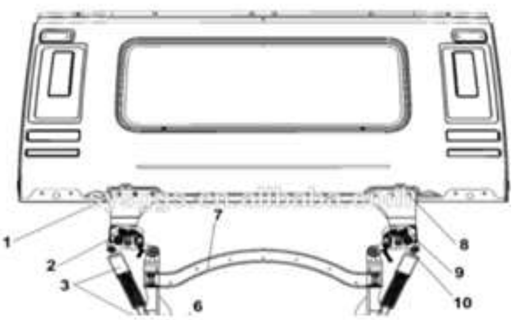


Fig.1: Mounting Bracket of Commercial Vehicle.

II. LITERATURE SURVEY

As our primary objective is to study the use of alternative material for mounting bracket and hence a literature survey is carried out to understand current trends in this field. Many papers related to static analysis in FEA & validate FEA results with analytical calculations or experimental testing is reviewed. Before proceeding to actual work it was essential to understand what are different types methods possible to carry out with existing software's, when to use those and what are the overall capabilities of such software's. As the software available for static analysis is ANSYS which we were not able to use at the beginning, the most basic and immediate need was to learn the software and be able to setup at least simple static analysis studies with it. Thus the literature survey included help files of these software's as well which turned out to be very fruitful. Almost entire software learning was done using the help files and tutorials. Few papers were found in which similar studies were carried out using ANSYS. In this project, will discuss the outcomes and the setup of such problems in software and hence provided the evidence that such works can be carried out. Surveying help files of software's also helped in understanding the working of software's and effect of many options available in the software on results obtained.

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. 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 a maximum stress as a constraint, optimization is performed.It can be said that FPM is very useful tool to calculate life of component using available load history.

PROBLEM STATEMENT

- Design of mounting bracket to support LCV Truck Cabin weight.
- Optimization of mounting bracket by changing material to Composite.
- Validation of FEA with results.

OBJECTIVES

1. Composite layer modeling in ANSYS ACP Tool.
2. Static Analysis to find out structural strength of mounting bracket.
3. Optimization of mounting bracket by changing material for cost benefit.

III. METHODOLOGY

In this study, modeling and analysis of mounting bracket will be performed by using Finite Element Method. The commercial finite element package ANSYS version 16 is use for the solution of the problem. The modeling of bracket will be done using Ansys Design Modeller. The simulation part will be carried out using the Analysis software, ANSYS. ANSYS ACP will be used for composite layer modeling for bracket.the Boundary condition constrain and load applied on bracket is analyzed. Load vs deflection curve will be generated through FEA and that will be validated by performing compression test of bracket on Universal Testing Machine. The bracket will be optimizing by changing material to composite. Composite material Glass fiber (GFRP) will be study for weight reduction and cost saving.

The methodology used for doing the analysis is as follows:

1. Develop a 3D model using SOLID WORKS software
2. The 3D model is converted into .stp and imported into ANSYS.
3. Performed Modal, Static and Fatigue analysis in ANSYS.
4. Plot deflections, stresses and life for all material used in FEA.
5. Proposed best material which has less mass ,weight and higher factor of safety.
6. Results with FEA for validation.

IV. DESIGN OF ENGINE MOUNT BRACKET

The vehicle mounting bracket mainly support to engine of the vehicle and it takes 30% load of the total weight. To manufacture the front engine mounting bracket, CASTING manufacturing process is employed. Fig.3 shows 2D drafting of front engine mounting bracket and developed 3D design model of the same.

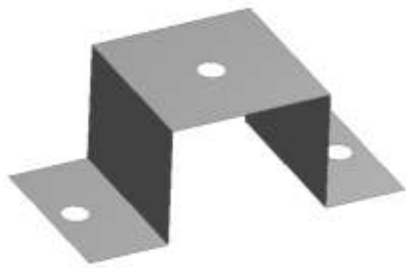


Fig.3: 3D model of engine mount bracket.

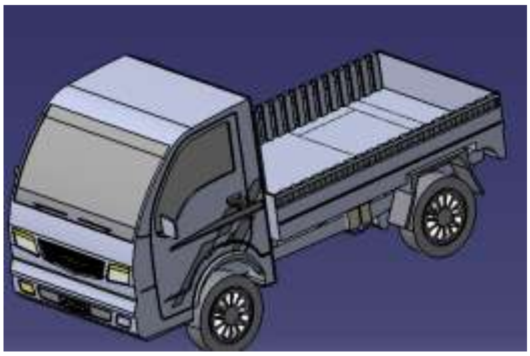


Fig.4: Tata ACE CAD Model

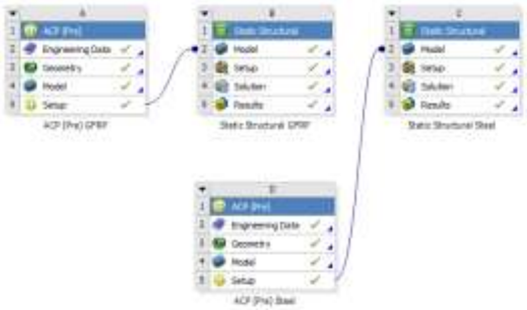
International Journal of Advances in Engineering and Management (IJAEM)
Volume 2, Issue 4, pp: 81-86 www.ijaem.net ISSN: 2395-5252

V. STATIC ANALYSIS OF VEHICLE MOUNTING BRACKET

Ansys procedure

1) Pre-Processing
- Geometry Modeling
- Meshing
- Material and Contact Definition
- Loading and boundary condition

2) Solution &
3) Post-Processing
- Deformation
- Stresses



Material Properties: The following are the material properties of the given mounting bracket.

Structural Steel Material Properties

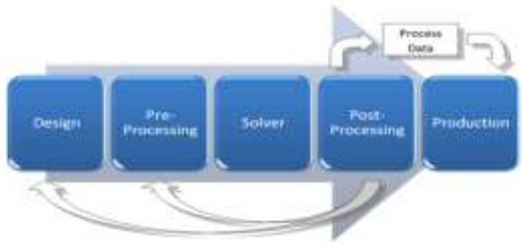
Material	Young's Modulus (GPa)	Poisons Ratio	Density (Kg/m3)
Steel	200	0.3	7850

	Material Strength (MPA)		
Material	Yield Strength	Ultimate Tensile Strength	Ultimate Compressive Stress
Steel	850	1150	1150

GFRP (Glass Fiber) Material Properties

Material properties	Value
Tensile modulus along X-direction (Ex), MPa	34000
Tensile modulus along Y-direction (Ey), MPa	6530
Tensile modulus along Z-direction (Ez), MPa	6530
Tensile strength of the material, MPa	900
Compressive strength of the material, MPa	450
Shear modulus along XY-direction (Gxy), MPa	2433
Shear modulus along YZ-direction (Gyz), MPa	1698
Shear modulus along ZX-direction (Gzx), MPa	2433
Poisson ratio along XY-direction (NUxy)	0.217
Poisson ratio along YZ-direction (NUyz)	0.366

ACP (ANSYS Composite Prep-Post)
Principle:
ANSYS Composite Pre-Post (ACP) is an add-in to ANSYS Workbench and is integrated with the standard analysis features. The entire work flow for composite material model can be completed for design .



The geometry modeling of a composite structure is basis form static analysis. Based on FE mesh, the boundary conditions and composite definitions are loads applied to the model structure. the after pre-processing stage is completed solution. the post-processing is used to findout the performance of the design . The insufficient design or composite material failure.the geometry has to be modifiey and findout is repeated.

ACP has a pre- processing and post-processing mode. the pre-processing mode, all composite material definitions can be created and mapped to the geometry modeling (FE mesh). This composite material can be definitions are transferred to FE model and solve.the post-processing mode, after a complete solution to be import of the result files for post-processing results (failure, safety, strains and stresses) can be evaluated and visualized.

Composite later of 0.25mm was considered here fiber creation and total 25 layers were modelled to create thickness of 10mm mounting bracket.

International Journal of Advances in Engineering and Management (IJAEM)
Volume 2, Issue 4, pp: 81-86 www.ijaem.net ISSN: 2395-5252

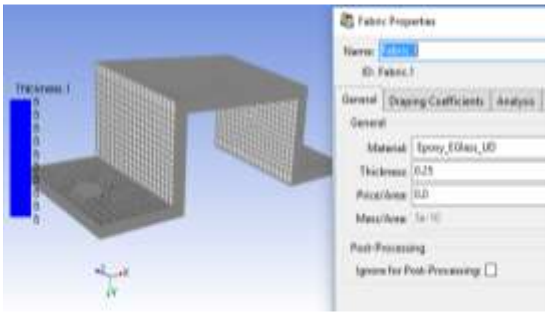


Fig.5: Composite layer thickness

All composite layers modelled with fiber ply angle of Zero.

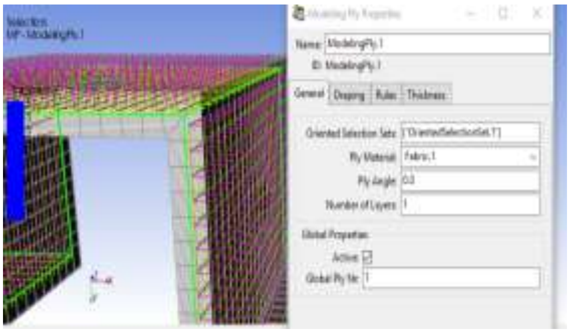


Fig.6: Composite layer Ply Angle

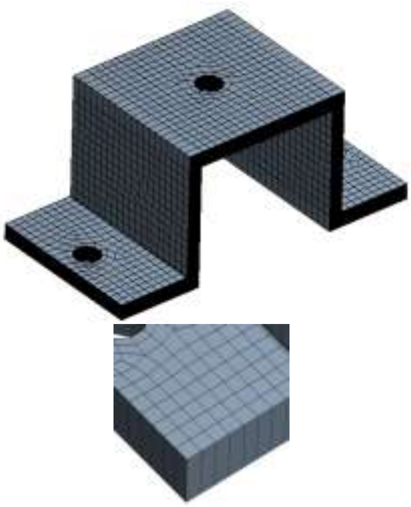


Fig.7: FE Modeling details; Nodes: 30282, Elements:26860

Loads and Boundary Conditions: Bottom two holes are fixed and top one hole where force as per cabin weight is applied.

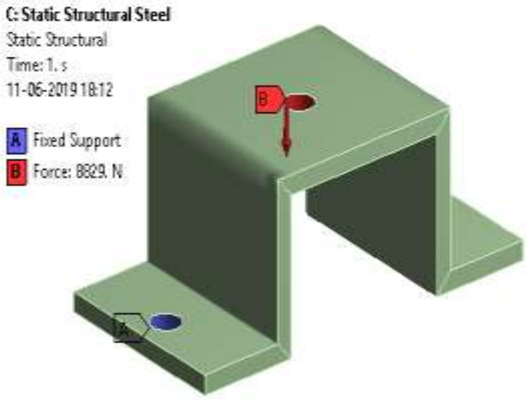


Fig.8: Loads and bc's

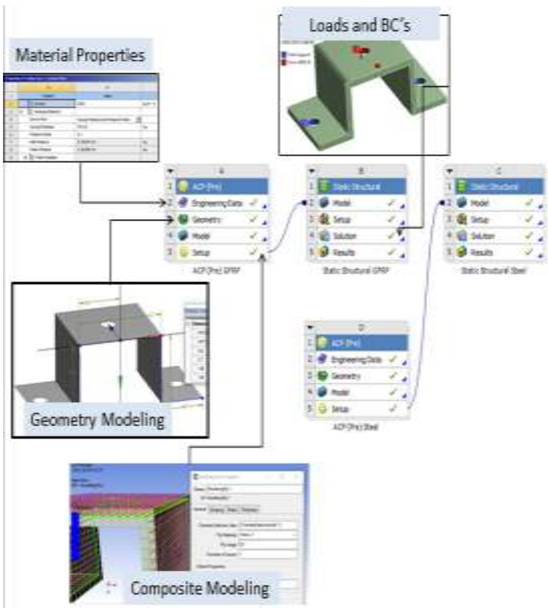


Fig.9: Schematic flow of whole Analysis

FEA Results:

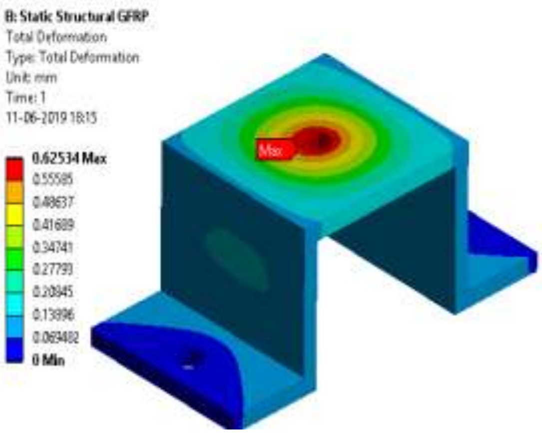


Fig.10: Deformation Plot of GFRP

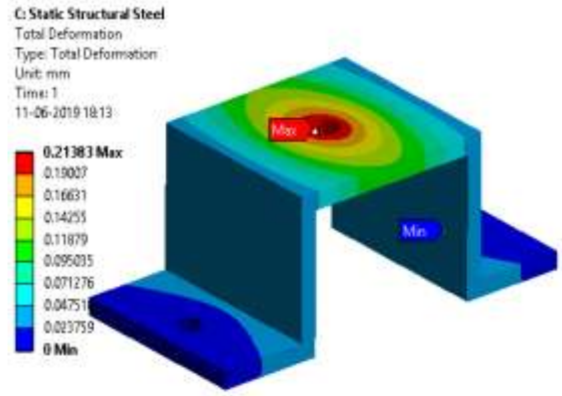


Fig.11: Deformation Plot of Steel

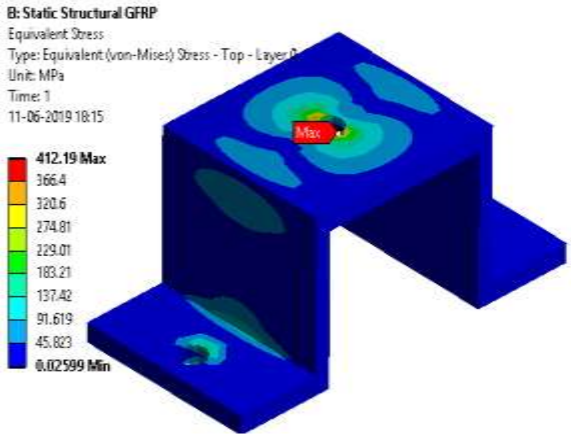


Fig.12: Stress Plot of GFRP

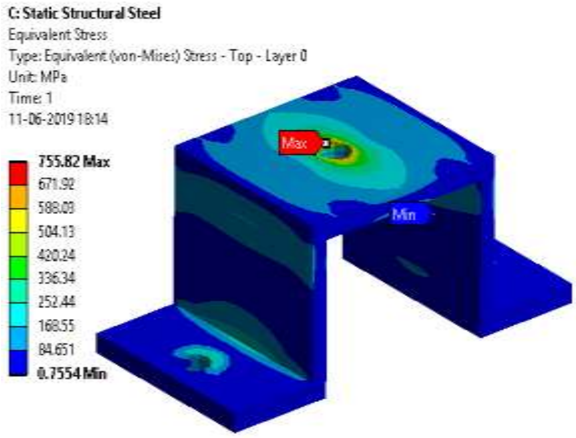


Fig.13: Stress Plot of Steel

Weight/Mass results:

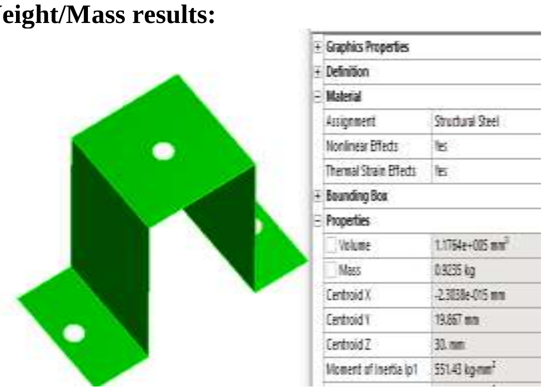


Fig.14: Steel bracket mass: 923gram

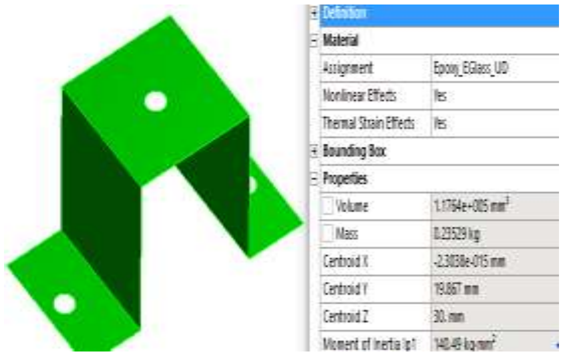


Fig.15: GFRP bracket mass: 235gram

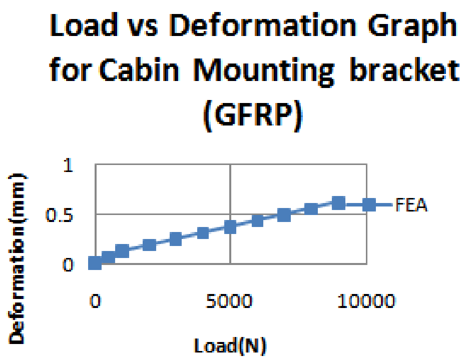
Results Summary Tables:

Bracket FEA results		
Material	Total Deformation(mm)	von Mises Stress (MPa)
Steel	0.214	755.820
GFRP	0.625	412.190
% Change	-192%	45%

Material	Mass(Kg)
Steel	0.923
GFRP	0.235
% Change	75%

Compression loading Results to compared with Experimental.

FEA	
Load(N)	Deformation(mm)
0	0
500	0.063
1000	0.125
2000	0.188
3000	0.250
4000	0.313
5000	0.375
6000	0.438
7000	0.500
8000	0.563
9000	0.625



VI. CONCLUSION

1. Composite material GFRP by using ANSYS ACP is studied.
2. Bracket max. Stress with steel material is 755MPa yield strength material of 850MPa and having FOS of 1.12.
3. Bracket max. Stress with GFRP composite material is 412MPa UTS material of 900MPa and having FOS of 2.18.
4. Stresses of all material are within material limit and hence meeting static analysis acceptance limits.
5. For same load (8829N) Glass fiber gives higher deformation of 0.625mm, where as compared to steel of 0.214 mm. As GFRP is more flexible and that will help to reduce vibration level by absorbing road vibration.
6. 75% reduction is observed for GFRP material vehicle mounting bracket compared to conventional steel material mounting bracket.
7. Thus GFRP material is high stiffness and material weight ratio compared to the convectional steel material . the weight of the GFRP is very low compared to the convectional steel mounting bracket it is very high performance to use GFRP mounting bracket instead of steel mounting bracket to reduce the overall weight of the vehicle when used in automotive industries

REFERENCES

[1]. P. Meghana, Y. Vijayakumar, Dr P. Ravinder Reddy, P.Seema Rani, Analysis of Cabin Mounting Bracket of Truck Using ANSYS, ISSN (Online): 2319 – 6734, ISSN (Print): 2319 – 6726.

[2]. Semiha Türkay, Hüseyin Akçay, studies the multi-objective control design for a truck cabin,Preprints of the 19th World Congress The International Federation of Automatic Control Cape Town, South Africa. August 24-29, 2014

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

[4]. 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

[5]. 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.

[6]. Willem-Jan Evers, Igo Besselink, Arjan Teerhuis, Albert van der Knaap and Henk Nijmeijer, controlling active cabin suspensions in commercial vehicles, Eindhoven University of Technology, Eindhoven, this research project is supported by TNO Automotive within the framework of the Competence Centre for Automotive Research (CCAR).

[7]. Greg Schade, Stacey Hamill, Vehicle Ride Analysis of a Tractor-Trailer, 2000 International ADAMS User Conference.

[8]. Dattatray B. Gavade, Prof. Satej S.Kelkar, optimization & analysis of structural behavior of front suspension rear shackle bracket, Gavade et al, International Journal of Advanced Engineering Research and Studies E-ISSN2249–8974.

International Journal of Advances in Engineering and Management
ISSN: 2395-5252

IJAEM

Volume: 02 Issue: 01 DOI: 10.35629/5252

www.ijaem.net
Email id: ijaem.paper@gmail.com