RESEARCH ARTICLE OPEN ACCESS

Finite Element Analysis of Fire Truck Chassis for Steel and Carbon Fiber Materials

Salvi Gauri Sanjay[*1], Kulkarni Abhijeet[*2], Gandhi Pratik Pradeep[**], Baskar P. [#]
*M. Tech., Automotive Engineering, SMBS, VIT University, Vellore – 632014, India.
**M. Tech., CAD-CAM, SMBS, VIT University Chennai Campus, Chennai – 600048, India.
#Assistant Professor, School of Mechanical and Building Sciences, VIT University, Vellore – 632014, India.

ABSTRACT
Chassis is the foremost component of an automobile that acts as the frame to support the vehicle body. Hence the frame ought to be very rigid and robust enough to resist shocks vibrations and stresses acting on a moving vehicle. Steel in its numerous forms is commonly used material for producing chassis and overtime alumimium has acquired its use. However, in this study traditional materials are replaced with ultra light weight carbon fiber materials. High strength and low weight of carbon fibers makes it ideal for manufacturing automotive chassis. This paper depicts the modal and static structural analysis of TATA 407 fire truck chassis frame for steel as well as carbon fibers. From the analyzed results, stress, strain and total deformation values were compared for both the materials. Since it is easy to analyze structural systems by finite element method, the chassis is modified using PRO-E and the Finite Element Analysis is performed on ANSYS workbench.
Keywords – ANSYS, FEA, Ladder frame, Modal analysis, Structural analysis

I. INTRODUCTION

Automotive chassis is the most vital component that offers strength and stability to the vehicle when it is subjected to different conditions. Chassis is the supporting frame of any vehicle on which engine body, axles, power train and suspension system is held together. Tie bars are used as fasteners to bind all the automobile components [1]. Ladder chassis is taken into account to be one of the oldest styles of automotive chassis. As the ladder chassis posses superior load carrying capacity, they are employed in most of the SUVs as well as in heavy commercial vehicles. Higher load carrying capacity of the chassis provides good driving dynamics and high ride comfort. Hence, ladder chassis are widely preferred over unibody and backbone frames.

Ladder chassis frame comprises of longitudinal members called as side bars attached with cross members [2]. Ladder frames also consists of brackets to support body and dumb iron to act as bearing for spring shackles. The chassis components are connected by riveted joints, weld joints or bolts. The ladder frame is made upswept at front and rear to accommodate the springing action of suspension system. The frame is narrowed at front to have better steering lock. The various cross sections used in frame construction includes channel, box, hat, double channel and I-section.

Stress analysis is carried out on the chassis to find the critical point having maximum stress [3]. Critical point is the crucial element that causes fatigue failure of the chassis frame. Thus, life span of the fire truck chassis completely depends on the magnitude of stress. In this paper, modal and static structural analysis is carried out on ladder chassis. Modal analysis includes determination of natural frequency and mode shape. The static structural analysis includes identification of maximum stressed area and also the torsional stiffness.

Chassis must be rigid and strong enough to absorb the vibrations produced by engine, suspension and drive line. The most commonly used materials for chassis are steel and aluminium. However carbon fibres are found to be advantageous over these traditional materials as carbon fibres have higher strength and stiffness and are also light in weight. They can also be easily moulded into different shapes. Light weight chassis reduces the fuel consumption of the vehicle thereby rising its fuel efficiency. Thus, ultra light weight carbon fibre chassis are preferred for manufacturing chassis to enhance the vehicle strength and stability.

II. MATERIALS AND METHODS

2.1 Basic Concept of FEA

Finite Element Analysis has now become an integral part of Computer Aided Engineering (CAE) and is being extensively used in the analysis of many tedious real time problems. The field of finite element analysis is matured and depends on rigorous mathematical foundation. Many powerful software tools and packages are available, promoting its widespread use in industries [4].

2.2 Modelling of Existing Chassis Frame

In order to proceed with this study, the dimensions of the chassis were gathered from the Tata 407 fire truck model. The 3D model of the chassis was designed in PRO-E.

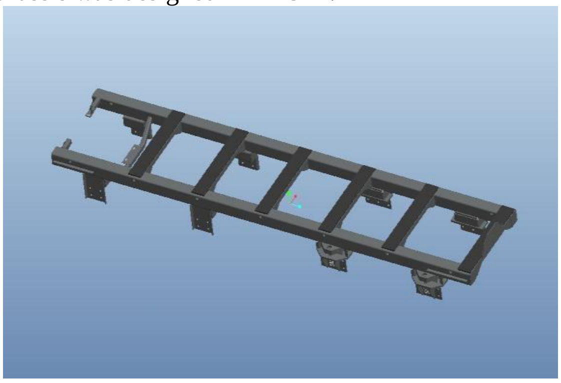

Figure 1: Solid model of fire truck chassis

2.3 Analysis in ANSYS

2.3.1 Engineering properties of materials

The required properties of steel and carbon fibres, to carry out the analysis are mentioned in table I.

Table 1: Properties of the materials

Material Properties	**Steel**	**Carbon Fibre**
Density (kg/m^3)	7850	1570
Young's modulus (MPa)	200	190000
Poisson's ratio	0.3	0.25
Yield stress (MPa)	250	200

2.3.2 Meshing of chassis frame

The chassis frame model is meshed in ANSYS with 152798 number of nodes and 58130 number of tetrahedral elements (Fig.2). In order to achieve better results, finer mesh is given to the region which is likely to have maximum stress. Finite element analysis is carried out on ANSYS workbench to find equivalent stress, maximum elastic strain and total deformation.

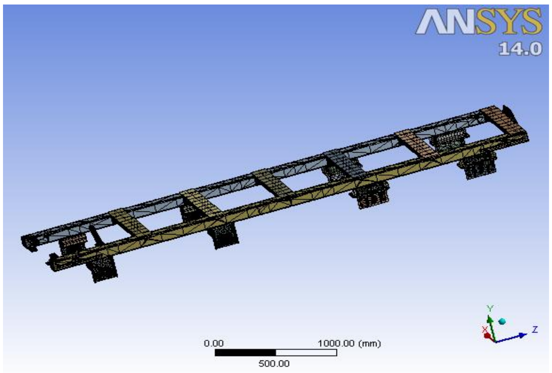

Figure 2: Meshed model

2.3.3 Loading condition for chassis frame

In this analysis, static forces are applied as loads on the chassis model. Maximum loaded weight of truck and body is considered to be 10 tonnes. It is assumed that the load is uniformly distributed over the chassis model. It can be seen from the Figure 3 that fixed supports are given at each brackets. Net force applied on upper side of the chassis frame is 156960 N. The two side bars withstand this net load equally. Thus each side bar carries the load of 78480 N. The type of connection employed in this analysis is of riveted type for joining the cross bars, side bars and brackets [5].

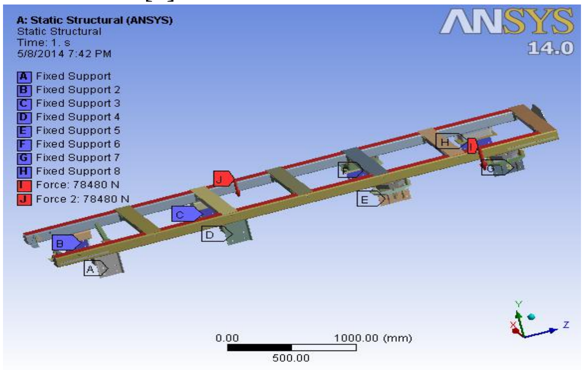

Figure 3: After applying the boundary conditions

2.4. Procedure

To analyse the ladder frame it is required to import into any analysis software package. Here ANSYS workbench has been selected and following steps are carried out for the analysis [6].

i. Creation of the project for the modal and structural analysis respectively.
ii. Selection of material from the engineering data.
iii. Meshing properties are specified to create the mesh.
iv. After generating the mesh, boundary condition such as fixed supports (loading conditions and center of gravity for structural analysis only) are applied.
v. Update the project for the given boundary conditions and results are obtained.

III. RESULTS AND ANALYSIS

3.1 Modal Analysis

For the modal analysis of the frame the boundary conditions are given as fixed support only. It is not required to give the other boundary conditions like loading and acceleration due to gravity. The modal analysis is carried out on structural steel and mode shape is obtained as follows.

3.1.1 Modal analysis of the chassis with material as structural steel

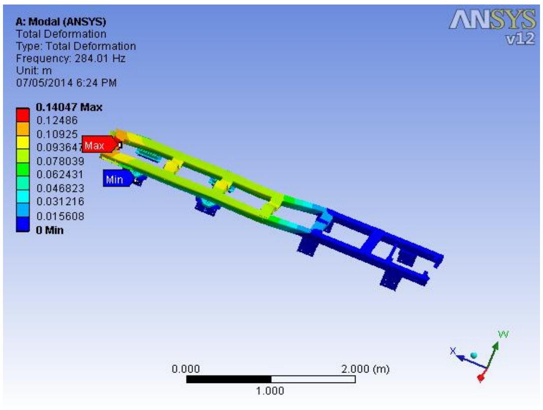

Figure 4: Deformation (First mode)

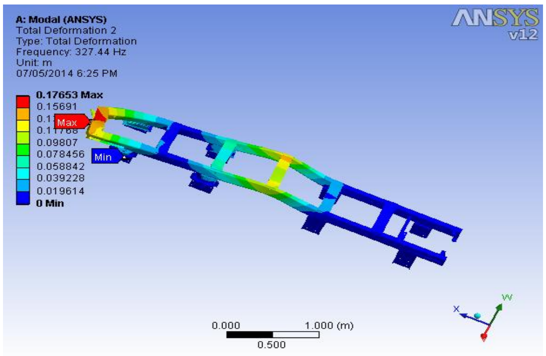

Figure 5: Deformation (Second Mode)

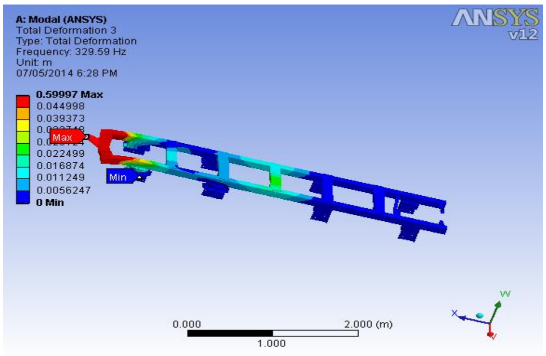

Figure 6: Deformation (Third Mode)

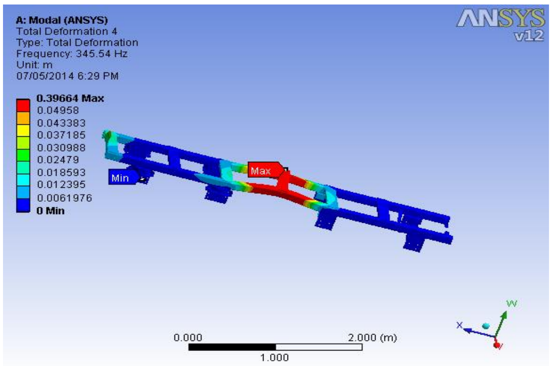

Figure 7: Deformation (Fourth Mode)

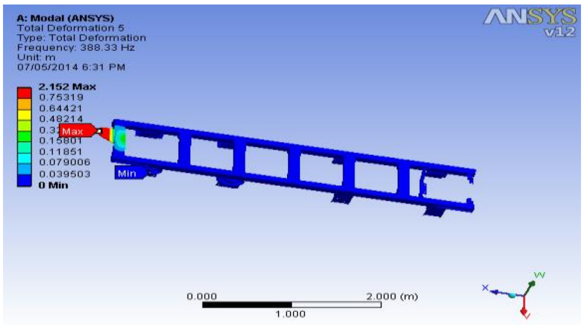

Figure 8: Deformation (Fifth mode

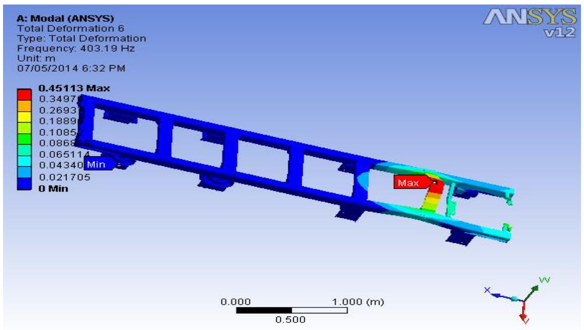

Figure 9: Deformation (Sixth Mode)

Table 2: Frequency and deflection for the chassis considering the steel material

STEEL Material	Frequency (Hz)	Deflection (m)
Deformation 1	284.01	0.14047
Deformation 2	327.44	0.17653
Deformation 3	329.59	0.59997
Deformation 4	345.54	0.39664
Deformation 5	388.33	2.152
Deformation 6	403.19	0.45113

Modes are the displacement pattern of body under consideration. In this study modal analysis was done for 6 different mode shapes. The response of the steel chassis due to six different natural frequencies was analysed. Results can be seen from the above figures.

3.1.2 Modal analysis of the chassis with material as carbon fibre

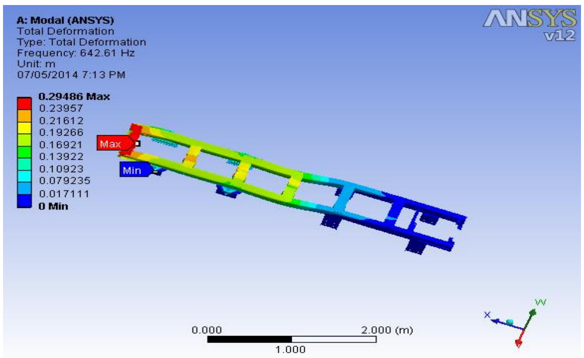

Figure10: Deformation (First Mode)

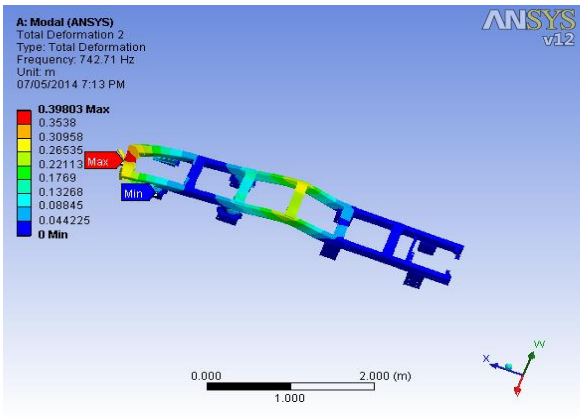

Figure 11: Deformation (Second Mode)

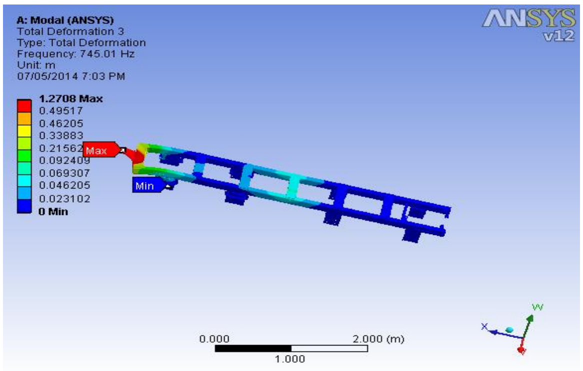

Figure 12: Deformation (Third Mode)

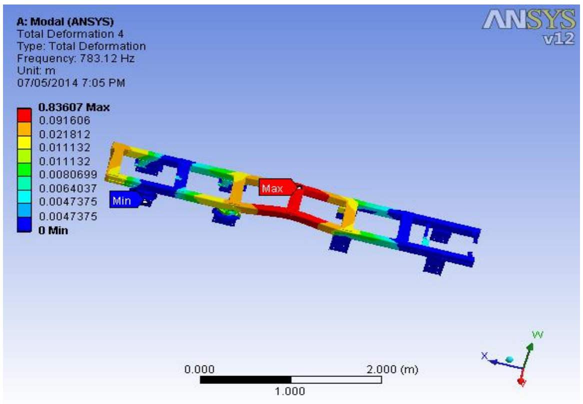

Figure 13: Deformation (Fourth Mode)

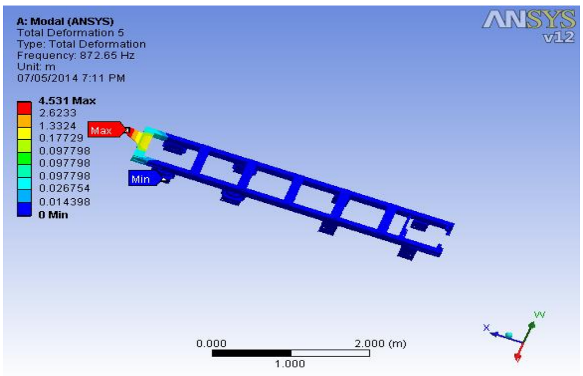

Figure 14: Deformation (Fifth Mode)

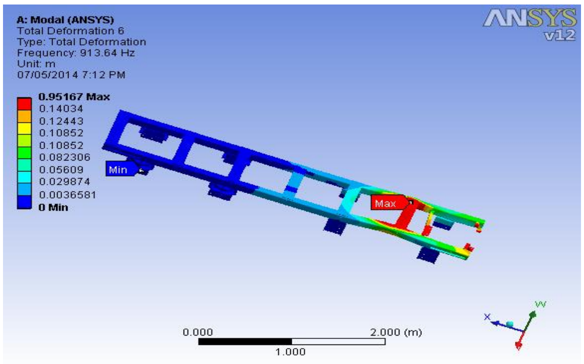

Figure 15: Deformation (Sixth Mode)

Table-3 Frequency and deflection for the chassis considering the Carbon Fibers

CARBON FIBER MATERIAL	FREQUENCY (Hz)	DEFLECTION (m)
Deformation 1	642.61	0.29486
Deformation 2	742.71	0.39803
Deformation 3	745.01	1.2708
Deformation 4	783.12	0.83607
Deformation 5	872.65	4.531
Deformation 6	913.64	0.95167

From the modal analysis results of steel, it can be seen that natural frequencies increases from the first mode to the sixth mode from 284.01 Hz to 403.19 Hz respectively. But, from the modal analysis results for the carbon fibers natural frequencies increases from 642.61 Hz to 913.64 Hz for the first and sixth mode respectively. It can therefore be concluded that carbon fibers excite at comparatively higher natural frequencies than the steel.

3.2 Static structural analysis of the fire truck frame

For structural analysis, fixed support, loading condition (pressure along longitudinal members) and acceleration due to gravity are given as boundary conditions.

3.2.1 Static structural analysis by using Steel

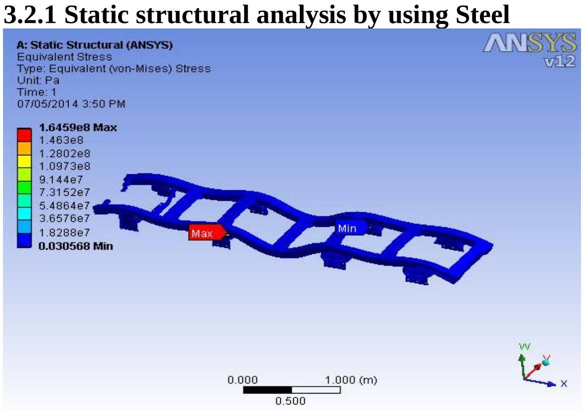

Figure16: Equivalent (Von Mises) Stress for steel

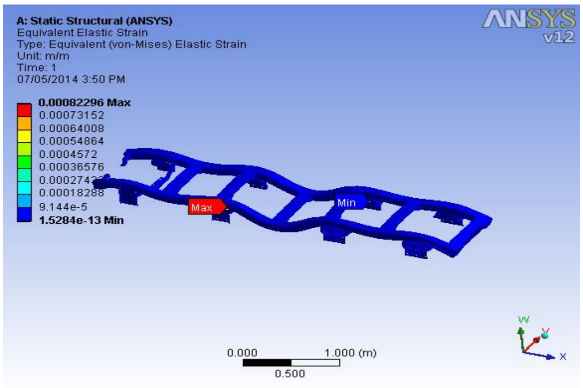

Figure17: Equivalent (Von Mises) Strain for steel

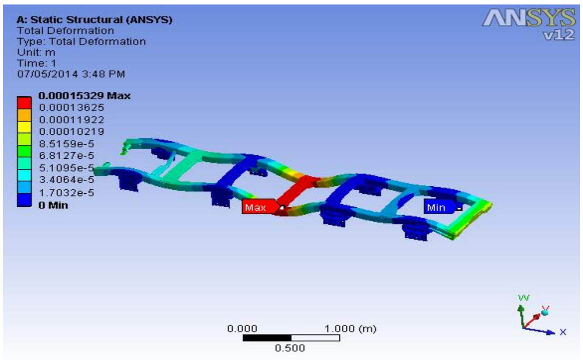

Figure 18: Total Deformation for steel

From the figure 16, it is observed that the maximum stress obtained from the analysis for overall component is 1.6459e8 MPa which is less than the yield strength of material which is 250MPa. Thus the design is safe. From figure 17 and 18, it is observed that the maximum Von Mises strain and maximum total deformation values of the chassis obtained are 0.00082296 and 0.00015329 m respectively, which are very less than the deformation limit of material.

3.2.2 Static structural analysis by using Carbon Fiber

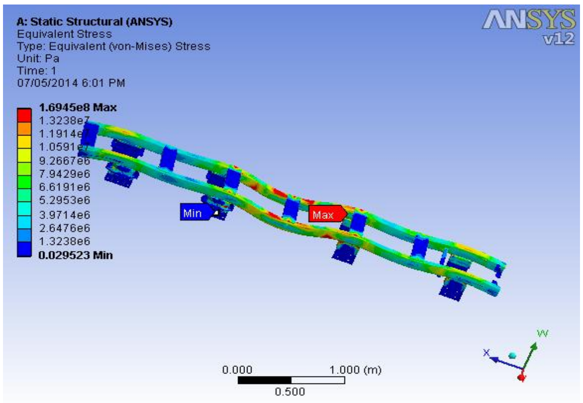

Figure 19: Equivalent (Von Mises) Stress for Carbon Fiber material

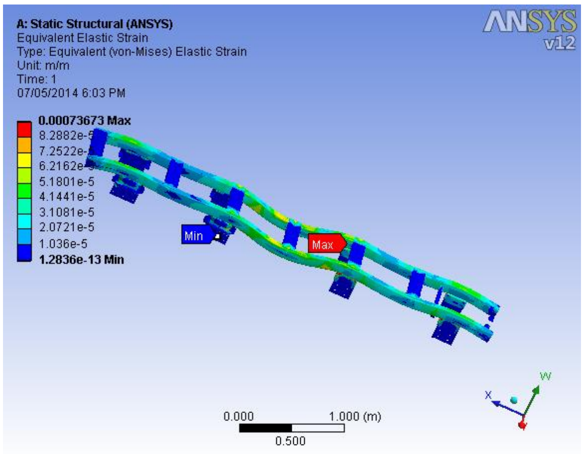

Figure 20: Equivalent (Von Mises) Strain for Carbon Fiber material

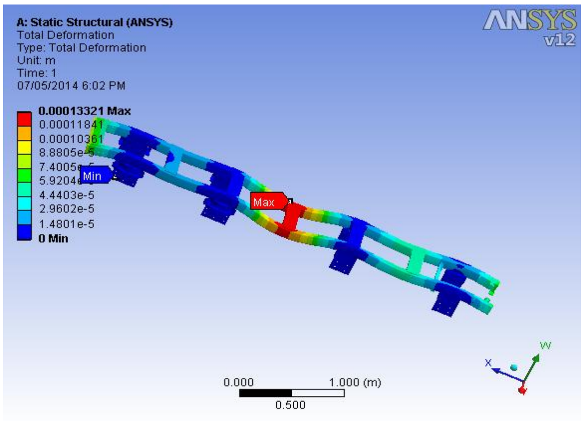

Figure21: Total deformation for Carbon Fiber material

From the figure 19, it is observed that the maximum stress obtained from the analysis for overall component is 1.6945e8 MPa which is less than the yield strength of material which is 250MPa. Thus the design is safe. From figure 20 and 21, it is observed that the maximum equivalent Von Mises strain and maximum total deformation values of the chassis obtained are 0.00073673 and 0.00013321 m respectively, which are very less than the deformation limit of material.

Table 4: Analyzed Static Structural results

Parameter	STEEL	CARBON FIBER
Equivalent Stress	1.6459E+008	1.6945E+008
Equivalent Strain	0.00082296	0.00073673
Total Deformation	0.00015329	0.00013321

IV. CONCLUSION

The modal analysis and static structural analysis on the Tata 407 ladder chassis of fire truck was carried out. From the above results of steel and carbon fibres, it can be seen that von equivalent stress for carbon fibres has increased and the total deformation has reduced. Thus the stress values for carbon fibres are under acceptable limit. So it is ideal to use the carbon fibre as a chassis material for vehicles because of its high strength and low weight. Also for the same load carrying capacity, carbon fibres are preferable instead of steel for the manufacturing of ladder frame because it (mass of steel frame 170.45 Kg and carbon fibres = 54.28 kg) reduces the weight by 60-68% and increase the stiffness of the chassis frame.

REFERENCES

[1] Chandra M. R., Sreenivasulu S., Hussain S. A., *Modeling and Structural Analysis of Heavy Vehicle Chassis Made of Polymeric Composite Material by Three Different Cross Sections, International Journal of Modern Engineering. Research, Volume 2, Issue.4*, July-Aug. 2012, ISSN: 2249-6645.

[2] Juvvi Siva Nagaraju, U. Hari Babu, *Design And Structural Analysis Of Heavy Vehicle Chasis Frame Made Of Composite Material By Varying Reinforcement Angles Of Layers, International Journal of Advanced Engineering Research and Studies, Volume 1, Issue 2*, January March 2012, ISSN: 2249-8974

[3] Tulasiram N., Charyulu T. N., *Design and Analysis of Vehicle Chassis Frame, Indian Streams Research Journal, Volume 2, Issue 7*, August 2012, ISSN: 2230-7850.

[4] Seshu P., *Textbook of Finite Element Analysis* (PHI Learning Pvt. Ltd., May 2009, ISBN: 97881-203-23155).

[5] Patel T. M., Dr. Bhatt M. G. and Patel H. K., *Analysis and Validation of Eicher 11.10 Chassis Frame using ANSYS, International Journal Of Emerging Trends And Technology in Comp. Science (IJETTCS), Volume 2, Issue 2*, March – April 2013, ISSN 2278-6856.

[6] Patel V. V. and R. I. Patel, "*Structural Analysis of a Ladder Chassis Frame*" *World Journal of Science and Tecnology*, 2012, 2(4):05-08, ISSN: 2231–2587.