International Journal of Mechanical Engineering (IJME)
ISSN(P): 2319-2240 ; ISSN(E): 2319-2259
Vol. 5, Issue 1, Dec – Jan 2016, 101-108

STATIC ANALYSIS, MODAL ANALYSIS AND DESIGN MODIFICATION IN CHASSIS FRAME TO OPTIMIZE WEIGHT BY USING COMPOSITE MATERIAL

ARCHIT TOMAR[1] & DHEER SINGH[2]

[1]School of Civil and Mechanical Engineering, Galgotias University, Greater Noida, India

[2]Assistant Professor, School of Civil and Mechanical Engineering, Galgotias University, Greater Noida, India

ABSTRACT

In the case of vehicle the term frame means the part of automobile that holds all the important components all these components constitute together to from chassis. The chassis frame have to be robust enough to resist various forces due to undulation in surface of road or any other reason. Forces act on chassis frame like shock, twist vibration and also due to heavy weight of chassis fame add extra stress. Along with strength the most important in frame designing is to have sufficient bending stiffness. Natural frequency and also played most important role in chassis frame the excitation frequency and chassis frame frequency never match otherwise it create resonance and damage will incur in chassis frame. Now a day’s lightweight material gained popularity worldwide due their high strength and less weight.

This paper present the static structural analysis is done using FEA method, modal analysis of a chassis frame is done do determine natural frequency and corresponding vibrational mode shapes, and also design modification done to optimize weight of chassis frame to perform this work the chassis frame designed in CATIAV5R19 and analysis is done in ANSYS14.5 .Material used for chassis frame is steel 52 and carbon epoxy composite material.

KEYWORDS: Chassis Frame, Composite Material, Design Modification, Finite Element Model, Modal Analysis, Static Analysis, Weight Optimization

INTRODUCTION

Chassis frame plays an most important role in every automobile it act as a skeleton for vehicles almost all the important parts are attached with chassis frame like engine, suspension system, steering system, tires, driveline and also hold almost all sprung and unsprung weight and chassis frame also heavy in weight so chassis frame must be strong enough to withstand shock, twist, vibration and other stresses. Chassis frame consist of side members along with supported cross member‘s, most important role of frame is to give strength and stability to automobile in different conditions. Automotive chassis s frame helps to keep up an automobile rigid, stiff and unbending. Automobile frames basically manufactured from aluminum and steel which make it heavy in weight and heavy weight directly effects the efficiency of vehicle so now a days the light weight material have gained a world wide popularity like composite materials – carbon fiber , epoxy glass etc. The composite materials have specialty that they are lighter in weight and many time in strength as compare to conventional steel used in chassis frame.

Finite element analysis is the one of the mostly used tool for stress analysis. Finite element analysis with required boundary conditions is used to determine the critical regions in chassis frame. Static analysis has been done to determine the maximum stress region.

Modal analysis is done to determine the natural frequency and different mode shapes on different frequency modal analysis helps to study the vibrational characteristics of a body which helps in avoiding the resonance. the outcome of modal analysis can be used as a reference value to other analysis like random analysis, harmonic analysis etc. the main characteristics of each mode of structure can be figured out through the modal analysis and actual vibration's can be anticipated.

This paper also present weight optimization by doing design modification in chassis frame all the parameter have been take into care while modifying the chassis frame parameters like mounting position, bolt position, rivet positions, so that design modification does not alter the position of these parts. Weight reduction helps in improving the efficiency and also less emissions.

CHASSIS FRAME SPECIFICATION

Model- EICHER 11.10

Side bar cross section= 210mm×76mm×6mm

Number of side bar= 2

Number of cross bar= 6

Channel = C- channel

Rear overhang= 1620mm

Front overhang= 935mm

Wheel base= 3800mm

Capacity of truck =8 ton

Weight of body and engine= 2 ton

MATERIAL USED

Table 1: Material Properties

SL.No.	Properties	Units	Steel-52(st52)	Carbon Epoxy
1.	Density	Kg/m^3	7850	1500
2.	Young's Modulus	N/mm^2	2×10^{11}	1.55×10^{11}
3.	Poisson Ratio	-	0.3	0.38

BASIC CALCULATIONTO FIND OUT THE LOAD ACTING ON FRAME

Truck Capacity= 8ton= 8000kg = 78480N

Truck Capacity with over loading considering 1.25% extra load = 1.25%×78480 = 98100N

Engine and body weight= 2 ton= 2000Kg = 19620N

Total load acting on chassis frame= 98100+19620 = 117720N

Load Acting on single frame = Total load/2= 58860N/Side Bar

3-D MODEL OF CHASSIS FRAME

CATIA V5R19 Designing software is used to design the chassis frame 3-D model by using specified dimensions. Then this chassis frame model imported In ANSYS 14.5 for further analysis procedure. The 3-D model is shown below

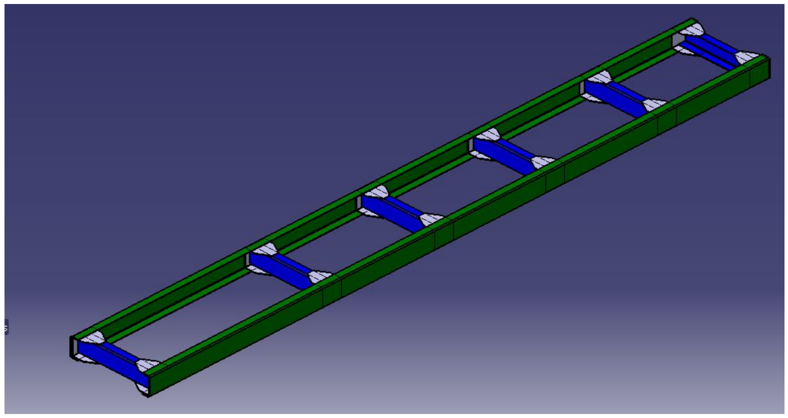

Figure 1: Chassis Frame CAD Model

MESHING IN ANSYS

The method used for meshing is tetrahedrons and finer meshing is applied at certain portion. The number of Nodes is 171869 and number of elements is 83060. The distance between the elements is trying as minimum as possible so that result outcome will be as precise and possible.

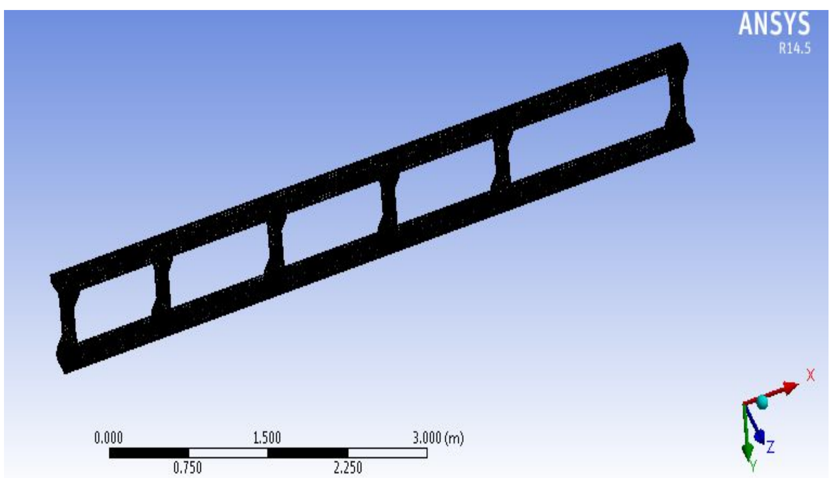

Figure 2: Meshing of Chassis Frame in ANSYS

BOUNDARY AND LOADING CONDITIONS

For this chassis frame the loading is assumed to be uniformly distributed load which is obtained from total load divided by total length of frame .Static load is acting on the chassis frame which is due to truck weight and truck capacity weight. The total load acting on upper area of chassis frame is 117720N. Gravity also considered as a loading factor and is considered during this analysis. There are total of seven boundary condition, front two and last two cross member are fixed and forces acting on both side member and one for gravity.

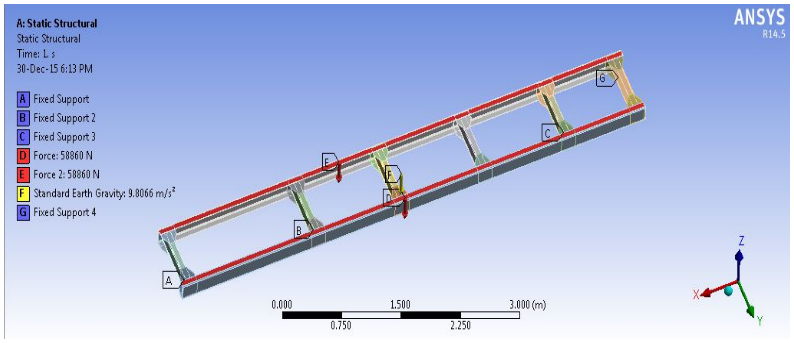

Figure 3: Boundary Condition of Chassis Frame in ANSYS

STATIC STRUCTURAL ANALYSIS USING CONVENTIONAL AND COMPOSITE MATERIAL

Static analysis shows the portion of chassis of maximum and minimum von mess stress the figures below shows the von-misses stress occur in conventional and composite material in chassis frame:

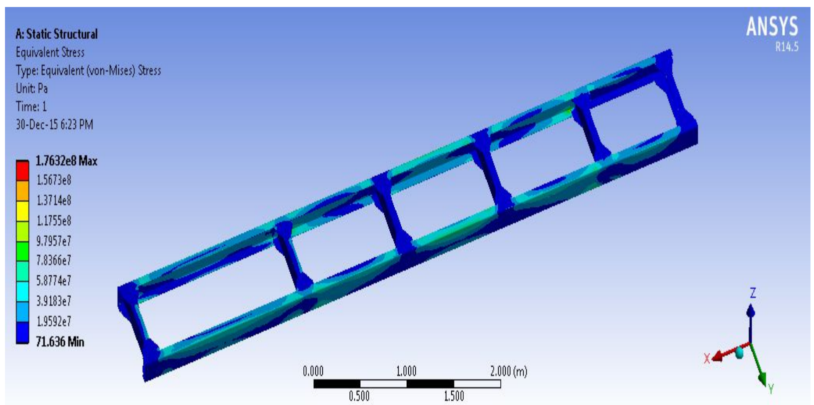

Figure 4: Von-Misses Stress in Chassis Frame Material Steel-52

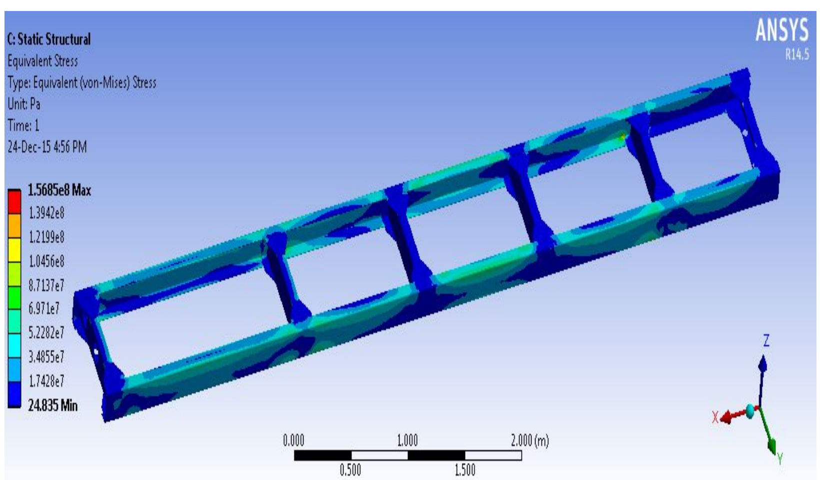

Figure 5: Von-Misses Stress in Chassis Frame Material Carbon/Epoxy

The von- misses generate in conventional material i.e. steel 52 is 176.32 MPa and von misses generate in carbon/epoxy composite material156.85 MPa

MODAL ANALYSIS OF CHASSIS FRAMES USING COMPOSITE AND CONVENTIONAL MATERIAL

The modal analysis is one of the most basic analysis the is done to judge the dynamic character .Modal analysis also help in determining the natural frequency of a chassis frame and also a mode shapes at particular frequency.

Modal analysis is most important for the purpose of avoiding the resonance condition. In this paper modal analysis is carried out by using ANSYS WORKBENCH the 6th order natural frequency of a chassis frame is carried out of both conventional and composite material.

Table 2: Natural Frequency

Mode	Steel – 52 Natural Frequency(Hz)	Carbon /Epoxy Natural Frequency(Hz)
1	42.31	85.311
2	83.698	168.76
3	88.69	178.29
4	116.37	227.17
5	112.73	234.87
6	131.31	264.84

Mode Shapes

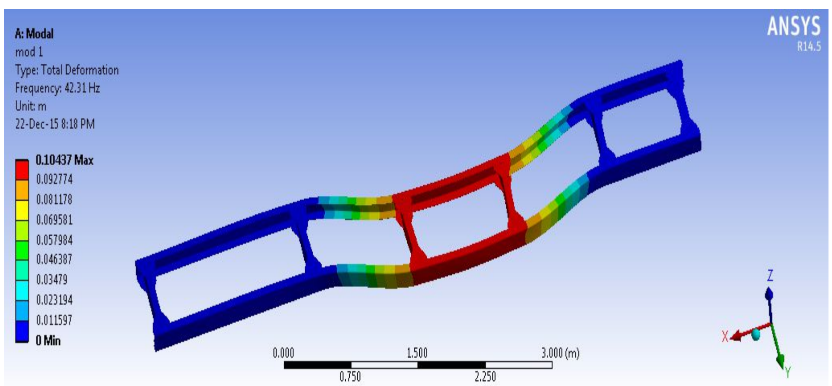

Figure 6: Mode Shape 1 Material Steel-52

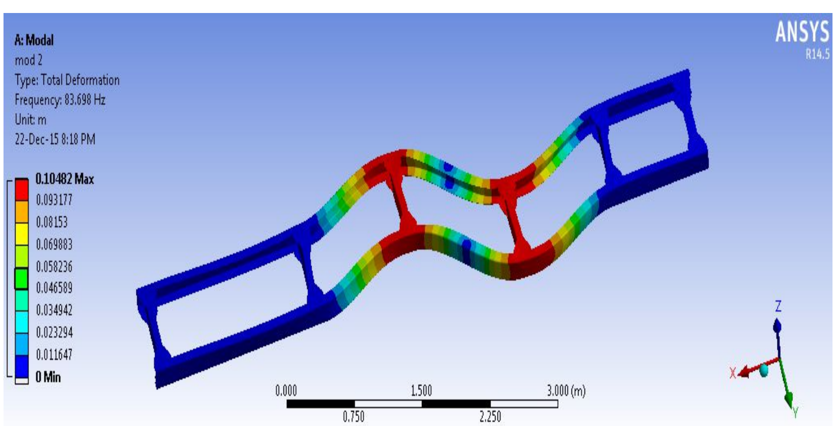

Figure 7: Mode Shape 2 Material Steel-52

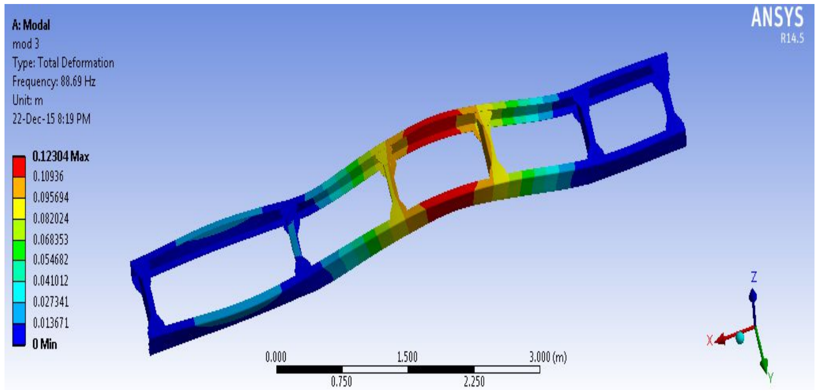

Figure 8: Mode Shape 3 Material Steel-52

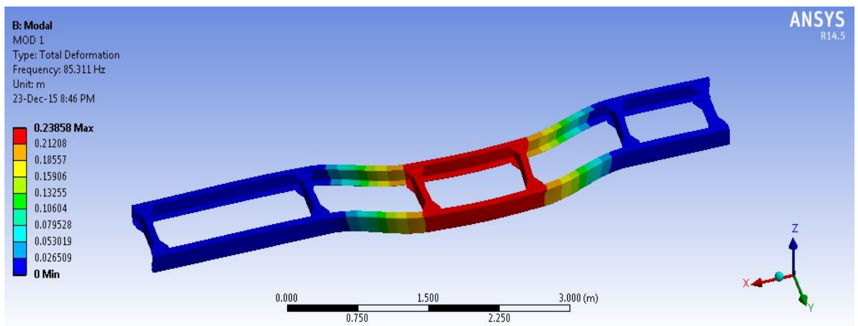

Figure 9: Mode Shape 1 Material Carbon/Epoxy

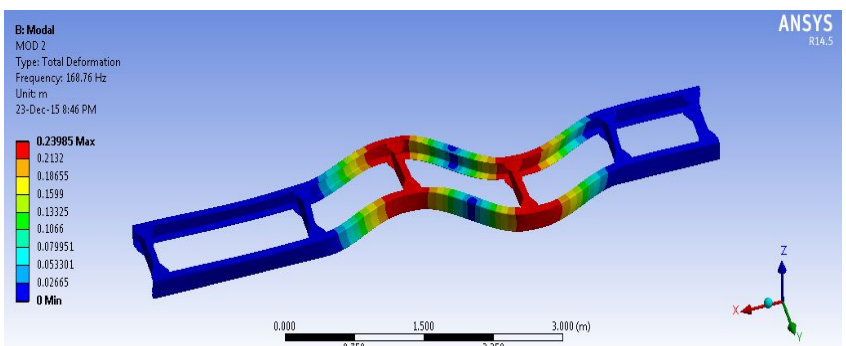

Figure 10: Mode Shape 2 Material Carbon/Epoxy

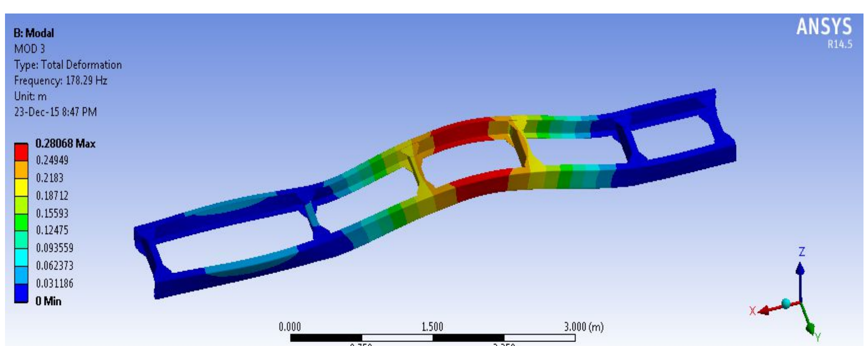

Figure 11: Mode Shape 3 Material Carbon/Epoxy

DESIGN MODIFICATION AND WEIGHT OPTIMIZATION OF CHASSIS FRAME

CAD Model of Modified Chassis Frame

Chassis frame is been modified to optimize the weight and strength of chassis frame remains the same.

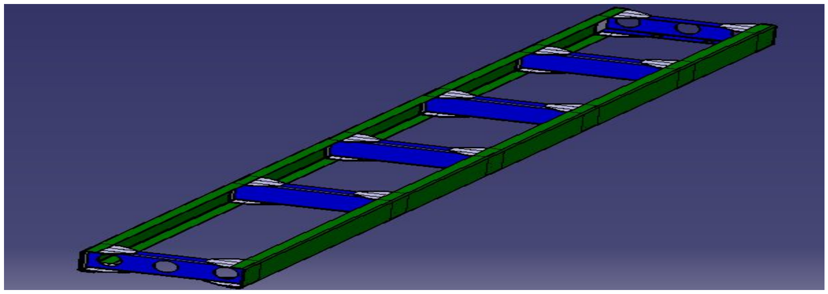

Figure 12: Modified Chassis Frame

A small hole's is made and rear and front cross member as clearly visible in CAD model .Number of circular profile that have been cut out is three in front and three in rear cross member a careful observation have been done while modifying the chassis frame so that other stuffs remains undistorted in this weight optimization the material used for chassis is carbon/epoxy composite material

WEIGHT OPTIMIZATION TABLE

Table 3: Weight Optimizing

Chassis Design	Material	Weight
conventional	Steel- 52	334.31 kg
conventional	Carbon/epoxy	63.910
modified	carbon /epoxy	63.208

As it is clear from above table that it is possible to decrease weight by using composite material and after modified the design there is further decrease in weight .By weight reduction in chassis frame it helps in increasing the efficiency of an auto vehicle.

CONCLUSIONS

This paper has looked into a static and dynamic analysis of a chassis frame and comparing it with the composite material and von- miss's stress induced in chassis frame of composite material is less than steel-52 chassis frame and weight of chassis frame also reduced which help in increasing efficiency of auto vehicle. Modal analysis also helps us to look into natural frequency of a chassis frame of both the material which is very important to study the vibrational characteristic also take out natural frequency which help us to avoid resonance. Design modification is done in chassis frame to optimize the weight and there is a net reduction in chassis frame weight due to design modification.

REFERENCES

1. Vijaykumar V. Patel, and R. I. Patel, "Structural analysis of a ladder chassis frame", World Journal of Science and Technology 2012, 2(4):05-08 ,ISSN: 2231 – 2587, April 21, 2012.
2. Monika S.Agrawal, "Finite Element Analysis of Truck Chassis Frame", International Research Journal of

Engineering and Technology (IRJET, Volume: 02 Issue: 03 | June-2015.

3. Abhishek Singh, Vishal Soni, Aditya Singh, “Structural Analysis of Ladder Chassis for Higher Strength”, International Journal of Emerging Technology and Advanced Engineering, Volume 4, Issue 2, February 2014.
4. R. L. PATEL, K.R.Gawande, D.B.Morabiya, “Design and analysis of chassis frame of TATA 2516TC”, International journal For Research in Applied Science and Engineering Technology (IJRASET), Vol. 2 Issue III, March 2014.
5. Manpreet Singh Bajwa, Yatin Raturi, Amit Joshi, “Static Load Analysis Of Tata Super Ace Chassis And Its Verification Using Solid Mechanics”, International Journal of Mechanical and Production Engineering, Volume-1, Issue- 2, Aug-2013
6. Parkhe Ravindra,*and Sanjay Belkar, “Performance Analysis of Carbon Fiber with Epoxy Resin Based Composite Leaf Spring”, International Journal of Current Engineering and Technology, Vol.4, No.2 (April 2014).
7. Archit Tomar & Rahul, “A Review On Hybrid Vehicles, Emissions Comparison With Conventional Vehicle And Policies Adoption For Promotion”, IMPACT: International Journal of Research in Engineering & Technology, Vol. 3, Issue 12, Dec 2015, 11-20
8. Dr.R.Rajappan, M.Vivekanandhan, “Static and Modal Analysis of Chassis by Using Fea”, The International Journal Of Engineering And Science (Ijes), Volume- 2, Issue- 2, Pages-63-73, 2013.
9. Yogendra S. Rajput1, Vikas Sharma1, Shivam Sharma1,Gaurav Saxena, “A Vibration Analysis of Vehicle Frame”, IJERA, Vol. 3, Issue 2, April 2013, pp.348-350.
10. Salvi Gauri Sanjay, Kulkarni Abhijeet, Gandhi Pratik Pradeep, Baskar P.’ “ Finite Element Analysis of Fire Truck Chassis for Steel and Carbon Fiber Materials” , Int. Journal of Engineering Research and Applications, Vol. 4, Issue 7(Version 2), July 2014, pp.69-74.
11. Andure M.W, Jirapure S.C., Dhamande L.P., PhD. “Advance AutomobileMaterial for Light Weight Future –A Review”, International Conference on Benchmarks in Engineering Science and Technology ICBEST 2012.
12. Pravin A Renuke, “ Dynamic Analysis Of A Car Chassis”, International Journal of Engineering Research and Applications (IJERA), Vol. 2, Issue 6, November- December 2012, pp.955-959.
13. Indu Gadagottu, and M V Mallikarjun, “Structural Analysis Of Heavy Vehicle Chassis Using Honey Comb Structure”, IJMERR, Vol. 4, No. 1, January 2015.
14. Shaik khajamoinuddin, B.Balaji, “Modal and Static Analysis of a Standard All-Terrain Vehicle Chassis”, IJEDR, Volume 2, Issue 1.