Design and Analysis of Automobile Frame

Sk. Avinash Kapil[1] | K V Jagadeesh[2] | N Venkatesh[3] | V Manikanta[4] | SkMd Rafi[5]

[1,2,3,4,5]Department of Mechanical Engineering, Godavari Institute of Engineering and Technology (A), Rajahmundry.

Abstract: The automotive chassis serves as a frame work for supporting the body and different parts of the automobile. Chassis or frame is a major component in a vehicle. In chassis different type of failures are occur due to static and dynamic loading condition. In this present work static and dynamic load characteristics are analyzed using FE models from this work. It is found that identifying location of high stress area, analyzing vibration, natural frequency and mode shape by using finite element method. Modal updating of automobile chassis model will be done by adjusting the selective properties such as mass density and Poisson's ratio. Predicted natural frequency and mode shape will be finding by using ansys software. The cad geometry of chassis is generated in solid works and model and static analysis is done in ansys workbench.

Keywords: Chassis, Frame, Vehicle, Dynamic, FE, Modal.

DOI of the Article: https://doi.org/10.46501/GIETME16

Available online at: http://ijmtst.com/vol7si05.html

As per UGC guidelines an electronic bar code is provided to seure your paper

To Cite this Article: Sk. Avinash Kapil; K V Jagadeesh; N Venkatesh; V Manikanta and SkMd Rafi. Design and Analysis of Automobile Frame. *International Journal for Modern Trends in Science and Technology* 2021, *7*, pp. 97-102. https://doi.org/10.46501/GIETME16

Article Info. Received: 12 June 2021; Accepted: 16 July 2021; Published: 20 July 2021

INTRODUCTION:

Chassis is a French term and was initially used to denote the frame parts or Basic Structure of the vehicle. It is the back bone of the vehicle. A vehicle without body is called Chassis. The components of the vehicle like Power plant, Transmission System, Axles, Wheels and Tyres, Suspension, Controlling Systems like Braking, Steering etc., and also electrical system parts are mounted on the Chassis frame. It is the main mounting for all the components including the body. So it is also called as Carrying Unit. The following main components of the Chassis are

1. Frame: it is made up of long two members called side members riveted together with the help of number of cross members.
2. Engine or Power plant: It provides the source of power
3. Clutch: It connects and disconnects the power from the engine fly wheel to the transmission system,
4. Gear box. The frame is the main part of the chassis on which remaining parts of chassis are mounted.

The frame should be extremely rigid and strong so that it can withstand shocks, twists, stresses and vibrations to which it is subjected while vehicle is moving on road. It is also called underbody. The frame is supported on the wheels and tyre assemblies. The frame is narrow in the front for providing short turning radius to front wheels. It widens out at the rear side to provide larger space in the body.

TYPES OF FRAME:

There are three types of frames:

a) Conventional frame

b) Semi integral frame, and

c) Integral frame.

OBJECTIVE

The vehicles are the back bone of a country's economy. In developing countries like India almost most of the transportation works goes by the road. The reason being that the Railways take longer time in transportation than the roadways. Thus the business in India runs on road. Also India is one of the largest consumers of vehicle in the world. There are number of Indian as well as foreign automobile manufacturers who depend on this sector. For transportation and travels the vehicles are preferred even today than any other means.

RELATED WORK

Kenji KARITA, Yoichiro KOHIYAMA, Toshihiko KOBIKI, Kiyoshi OOSHIMA, Mamoru HASHIMOTO (2003) had developed a chassis made by Aluminium. The material selected for the frame is 6061-T6. They used the Variable section extrusion method for making the chassis. It's developed with the help of computer Aided Engineering. Aluminium material gives an advantage of weight reduction. From this study authors found that the Aluminium chassis meets the target of weight reduction, strength and rigidity. Also they concluded that the remaining technical issues will be addressed to enable commercial adoption of the aluminum frame.

Alireza Arab Solghar, ZeinabArsalanloo (2013) studied and analyzed the chassis of Hyundai Cruz Minibus. ABAQUS Software was used for modeling and simulation. Self weight of the chassis is considered for static analysis and Acceleration, Braking and Road Roughness were considered for dynamic analysis. It's observed that the stresses on chassis caused by braking were more compared with acceleration.

THREE DIMENSIONAL CAD MODELS

The Three Dimensional CAD models of the chasis are designed based on the requirements and all design aspects using the CATIA V5 R20. Few important components of the chasis are mentioned below:

- Side bars
- Cross members
- Brackets
- End member
- Rivets.

This model is analyzed in the ANSYS software using three different materials mentioned below:

- Structural steel
- Aluminium alloy and
- Titanium alloy

ANSYS WORKBENCH

Dr. John Swanson founded ANSYS. Inc in 1970 with a vision to commercialize the concept of computer simulated engineering, establishing himself as one of the pioneers of Finite Element Analysis (FEA). ANSYS inc. supports the ongoing development of innovative technology and delivers flexible, enterprise wide engineering systems that enable companies to solve the full range of analysis problem, maximizing their existing investments in software and hardware. ANSYS Inc. continues its role as a technical innovator. It also supports a process-centric approach to design and manufacturing, allowing the users to avoid expensive and time-consuming "built and break" cycles. ANSYS analysis and simulation tools give customers ease-of-use, data compatibility, multi-platform support and coupled field multi-physics capabilities.

PROPOSED METHODOLOGY

DESIGNING OF FRAME USING SOLID WORKS

It is a document that includes the specifications for a part's production. Generally the part drawings are drawn to have a clear idea of the model to be produced. The components that are generated in part module are imported to assembly module and by using 'insert components' command and all these components are mated together to form the required assembly. The different views of assembly and the drawing are as shown below.

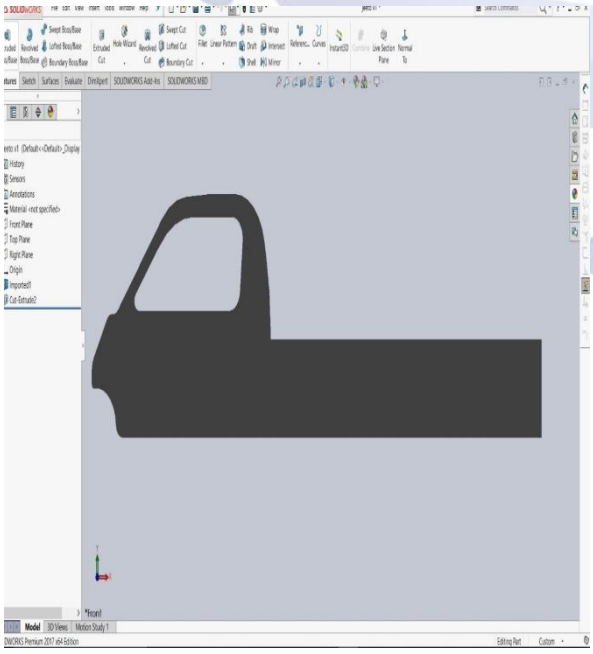

Fig 1: front view of the mahindrajeeto frame

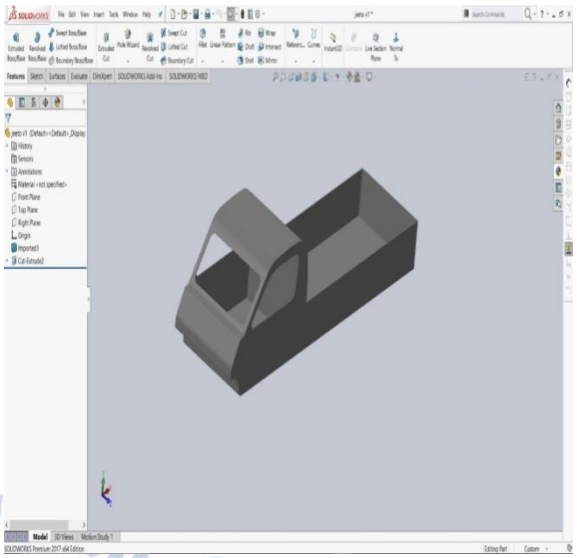

Fig 2 : isometric view of the mahindrajeeto frame

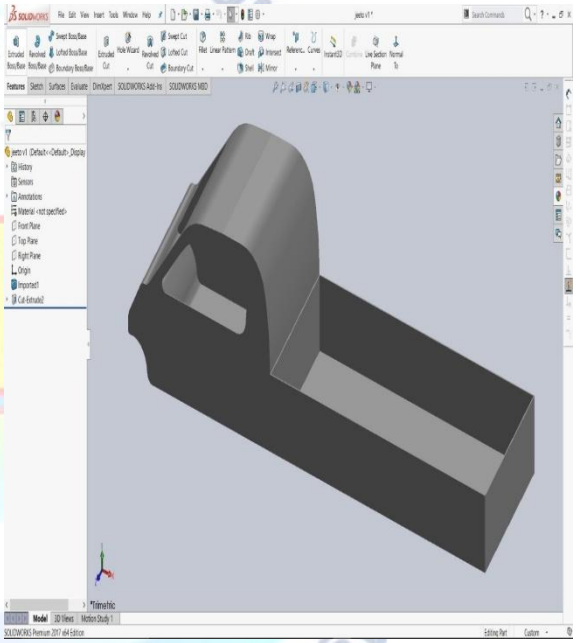

Fig 3: Trimetric view of the Mahindra jeeto frame

SIMULATION RESULTS

Table 1: Analysis of Steel frame

Property	Value	Units
Material name	Structural Steel	
Density	7850	Kg m^3
Thermal expansion	1.2E+05	C^-1
Young`s Modulus	2E+11	Pa
Posisson`s Ratio	0.3	-
Tensile Yield Strength	2.5E+08	Pa
Compressive Yield Strength	2.5E+08	Pa
Tensile Ultimate Strength	4.6E+08	Pa

Table: Analysis of Alluminium frame

Property	Value	Units
Material name	Structural Steel	
Density	2.77e-06	Kg mm^3
Thermal expansion	1.5E+05	C^-1
Young`s Modulus	7100	MPa
Posisson`s Ratio	0.33	-
Tensile Yield Strength	280	MPa
Compressive Yield Strength	280	MPa
Tensile Ultimate Strength	310	MPa

Geometry imported

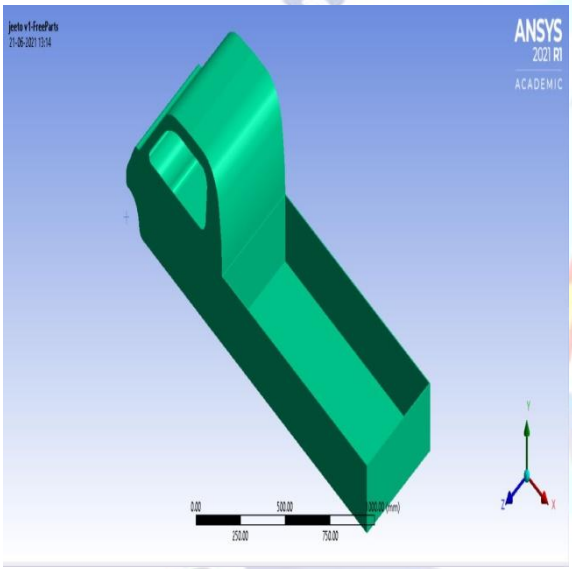

Fig 4: Geometry Imported in Ansys

6.2 Boundary Conditions

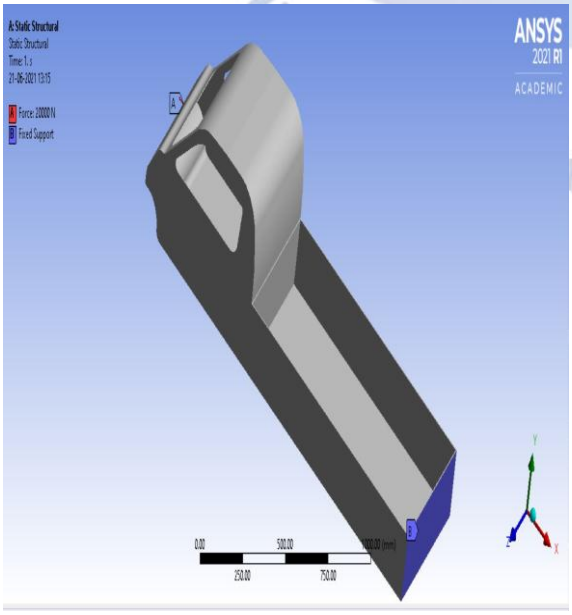

Fig 5: Boundary Conditions Applied

Front and Rear Impact analysis with Structural Steel:

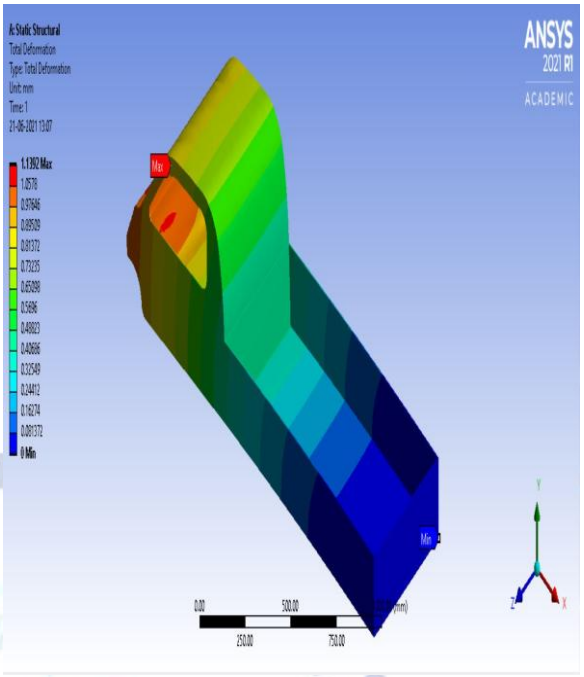

Fig 6: Front Impact Stress

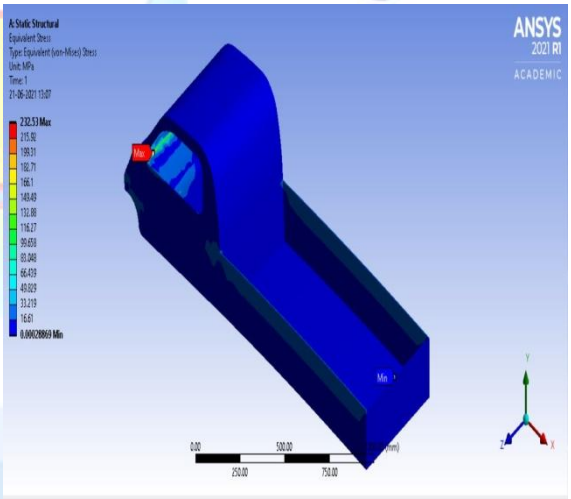

Fig 7: Front Impact Deformation

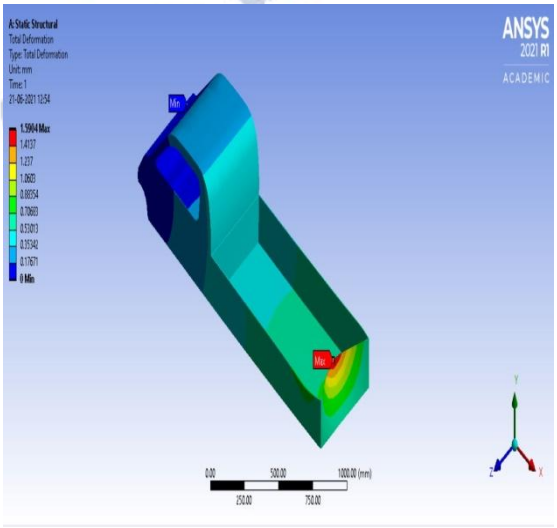

Fig 8: Rear Impact Stress

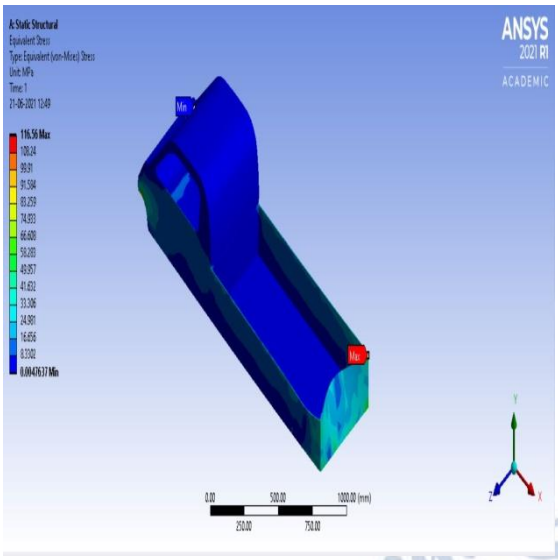

Fig 9: Rear Impact Deformation

Front and Rear Impact analysis with Aluminium Alloy:

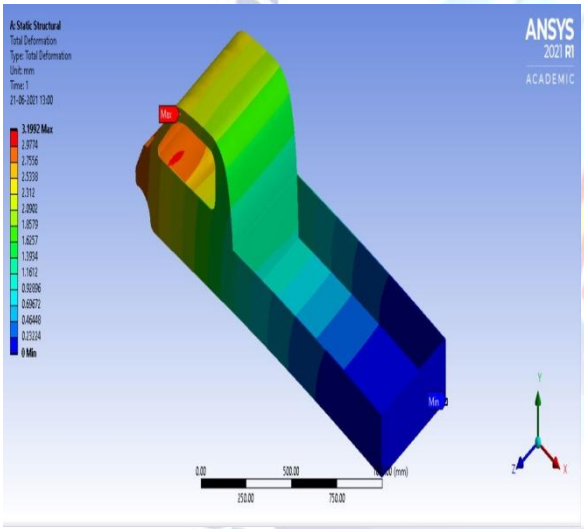

Fig 10: Front Impact Stress

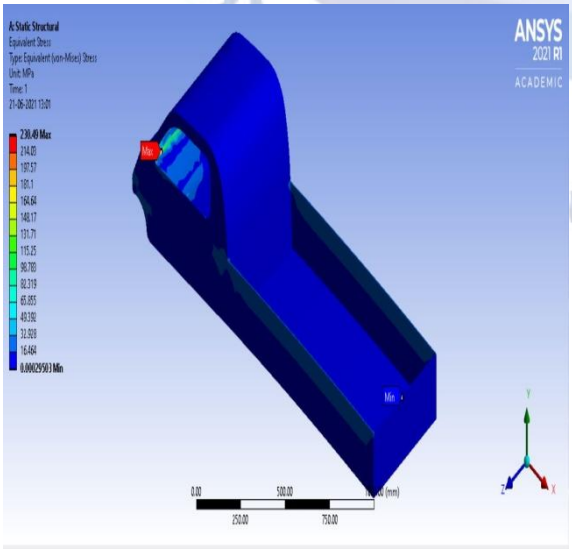

Fig 11: Front Impact Deformation

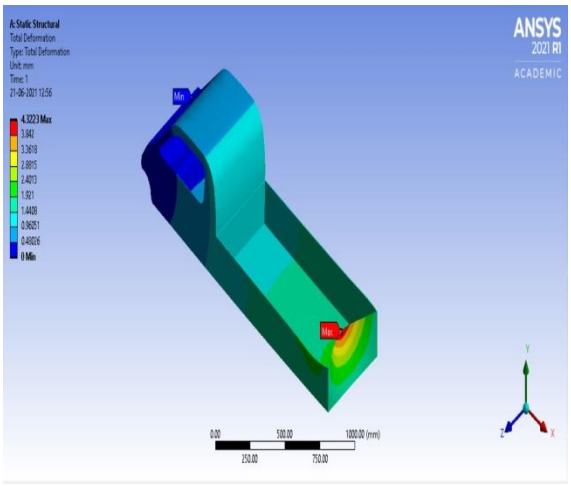

Fig 12: Rear Impact Stress

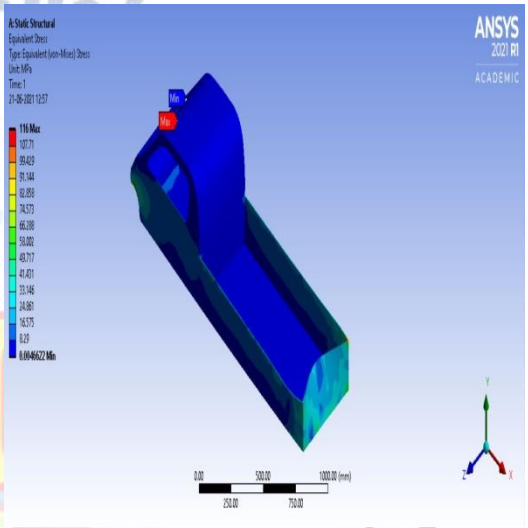

Fig 13: Rear Impact Deformation

CONCLUSION

By performing the structural analysis of mahindrajeeto frame with different materials we conclude that frame with aluminium alloy giving better results as compare with structural steel material. The maximum stress induces in the front impact where Al alloy material is inducing 230.49 whereas structural steel induces 232.53 but due to ductile property in aluminium material deforming 4.3223 mm where as structural steel deforming 1.5904mm.

For further we would like to perform explicit dynamic analysis to study the impact of contact dynamics of frame with different materials. In the explicit dynamic analysis, we would like to perform impact analysis for different speeds to see the stress and deformation in the frame. Where we can analyse the damage of frame for different working conditions.

REFERENCES

1. Abhishek Singh, Vishal Soni and Aditya Singh. 2014. Structural Analysis of Ladder Chassis for Higher Strength: International Journal of Emerging Technology and Advanced Engineering, Vol. 4. Issue. 2.
2. AnandGosavi, Ashish Kumar Shrivastava and Ashish Kumar Sinha. 2014. Structural analysis of six axle trailer frame design and modification for weight reduction: International Journal of Emerging Technology and Advanced Engineering, Vol. 4. Issue. 1.
3. Ashutosh Dubey and VivekDwivedi, 2003. Vehicle Chassis Analysis: Load Cases and Boundary Conditions for Stress Analysis.
4. Balbirsingh, R., Guron, Dr. Bhope, D.V. and Prof. Yenarkar, Y. L. 2013. Finite Element Analysis of Cross Member Bracket of Truck Chassis: IOSR Journal of Engineering, Vol. 3, Issue. 3. PP. 10-16.
5. Deulgaonkar, V. R., Dr. Matani, A. G. and Dr. Kallurkar, S. P. 2012. Advanced Mathematical Analysis of Chassis Integrated Platform Designed for Unconventional loading by using simple technique for static load: International Journal of Engineering and Innovative Technology, Vol. 1. Issue. 3.
6. Dr. Rajappan, R. and Vivekanandhan, M.2013. Static and Modal Analysis of Chassis by Using FEA: The International Journal Of Engineering And Science, Vol. 2. Issue. 2. Pp. 63-73.
7. Dr. Rajappan, R. and Vivekanandhan, M.2013. Static and Model Analysis of Chassis By Using FEA: Proceedings of the National Conference on Emerging Trends In Mechanical Engineering.
8. GoollaMurali, Subramanyam, B. and Dulam Naveen.2013. Design Improvement of a Truck Chassis based on Thickness: Altair Technology Conference. India.
9. Harshad, K., Patel, Prof. Tushar, M. and Patel.2013. Structural Optimization Using FEA-DOE Hybrid Modeling –A Review: International Journal of Emerging Technology and Advanced Engineering, Vol. 3. Issue. 1
10. Haval Kamal Asker, ThakerSalihDawoodl and ArkanFawzi Said. 2012. stress analysis of standard truck chassis during Ramping on block using finite element method: ARPN Journal of Engineering and Applied Sciences, Vol. 7. No. 6.
11. Hemant, B., Patil, Sharad, D., Kachave, Eknath, R. and Deore. 2013. Stress Analysis of Automotive Chassis with Various Thicknesses: IOSR Journal of Mechanical and Civil Engineering, Vol. 6. Issue. 1. PP. 44-49.
12. Hirak Patel, Khushbu, C., Panchal, Chetan S. and Jadav. 2013. Structural Analysis of Truck Chassis Frame and Design Optimization for Weight Reduction: International Journal of Engineering and Advanced Technology, Vol. 2. Issue. 4.
13. Kim, H. S., Hwang, Y. S. and Yoon, H. S. 2010. Dynamic Stress Analysis of a Bus Systems: Commercial Vehicle Engineering & Research Center. Hyundai Motor Company. 772-1, Changduk. Namyang. Whasung. Kyunggi-Do. Korea.
14. Kurdi, O., Abd- Rahman, R. and Tamin, M. N. 2008. Stress Analysis of Heavy Duty Truck Chassis Using Finite Element Method.
15. Monika, S., Agrawal, Razik, Md.2013. Finite Element Analysis of Truck Chassis: International Journal of Engineering Sciences & Research Technology, Agrawal. 2(12).
16. Paul, I. D., Sarange, S. M., Bhole, G. P., and Chaudhari, J. R.2012. Structural Analysis Of Truck Chassis Using Finite Element Method: International J.ofMultidispl.Research and Advcs. in Engg., Vol. 4. No. I. pp. 85-98.
17. Ravi Chandra, M., Sreenivasulu, S. and Syed Altaf Hussain. 2012. Modeling and Structural analysis of heavy vehicle chassis made of polymeric composite material by three different cross sections: International Journal of Modern Engineering Research, Vol. 2. Issue. 4. pp. 2594-2600.
18. RoslanAbd Rahman, Mohd Nasir Tamin and Ojo Kurdi.2008. Stress Analysis Of Heavy Duty Truck Chassis As A Preliminary Data For Its Fatigue Life Prediction Using Fem: JurnalMekanika. No. 26. PP. 76 – 85.
19. SairamKotari, Gopinath, V. 1998. Static and Dynamic Analysis on Tatra Chassis: International Journal of Modern Engineering Research, Vol. 2. Issue. 1. Pp. 086-094.
20. Salvi Gauri Sanjay, Kulkarni Abhijeet, Gandhi Pratik Pradeep and Baskar, P.2014. Finite Element Analysis of Fire Truck Chassis for Steel and Carbon Fiber Materials: Journal of Engineering Research and Applications, Vol. 4. Issue. 7. Version. 2. PP. 69-74.
21. SandipGodse and Prof. Patel, D.A.2013. Static Load Analysis Of Tata Ace Ex Chassis And Stress Optimisation Using Reinforcement Technique: International Journal of Engineering Trends and Technology, Vol. 4. Issue. 7.
22. Tushar, M., Patel, Dr. Bhatt, M. G., and Harshad, K. and Patel. 2013. Analysis and validation of Eicher 11.10 chassis frame using Ansys: International Journal of Emerging Trends & Technology in Computer Science, Vol. 2, Issue. 2