International Journal of Scientific & Engineering Research, Volume 8, Issue 3, March-2017
ISSN 2229-5518

Modal analysis of Truck Chassis Frame

Shubham Bhise[1], Vaibhav Dabhade[1], Sujit Pagi[1], Apurvi Veldandi[1].
[1]B.E. Student, Dept. of Automobile Engineering,
Saraswati College of Engineering,
Navi Mumbai, India.

Vishnudas Chodankar[2].
[2]Asst. Prof, Dept. of Automobile Engineering,
Saraswati College of Engineering,
Navi Mumbai, India.
vipul_chodankar@rediffmail.com

***Abstract*- A study will be carried out to find the characteristics of mode (and vibrational) responses of heavy vehicle chassis subjected to vibrations .The chassis of the truck is modeled in CATIA V5 software and meshing in a Hyper mesh 12 software for carrying out analysis. Further this meshed model is imported to ANSYS and modal analysis is performed. The vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component during free vibration modal analysis are determined by modal analysis. The natural frequency and mode shapes are important parameters in the modal analysis of any component. Mode shapes are obtained by giving suitable conditions with some excited input (frequency).Hence, using Modal analysis defects can be detected and necessary modifications such as change in material, geometry of stiffener, etc. can be made.**

Keywords—Chassis, Modal analysis, ANSYS.

I. INTRODUCTION

Chassis is a major component in a vehicle system. The study involves modal analysis to determine key characteristics of the truck chassis. The identification of location of high stress area and determination of the torsional stiffness of the chassis is included in static characteristics. Finite element method is used to determine the dynamic characteristics of truck chassis such as the natural frequency and mode shape. The validation of the FE model is done by experimental modal analysis. Modal updating of the truck chassis model was done by adjusting the selective properties. Predicted natural frequency and mode shape were compared with experimental results. To reduce the vibration, improve the strength and to optimize the weight of the truck chassis, the modifications were made to the truck chassis model. A modal analysis determines the vibration characteristics natural viz. frequencies and mode shapes of a structure or a machine component. The natural frequencies and mode shapes are important parameters in the design of a structure. It can also perform a modal analysis on a pre-stressed structure, such as a spinning turbine blade. Modal analysis is the study of the dynamic properties of structures under vibrational excitation. Finite element analysis normal mode solutions can be correlated with the results obtained.

II. LITERATURE REVIEW

[1] The aim of the project is to find the characteristics of mode (and vibrational) responses of heavy vehicle chassis at particular frequency inputs. The chassis dimensions are measured from an automobile workshop and are developed in CATIA V5 R20 media and are imported to ANSYS commercial software to carry out the modal analysis. Determination of the vibration characteristics viz. natural frequencies and mode shapes of a structure or a machine component during free vibration is carried out by modal analysis. The natural frequency and mode shapes are important parameters in the modal analysis of any component. Mode shapes are obtained by giving suitable conditions with some excitations. Block Lanczos method is used in modal analysis. We can detect the defects in the component by analyzing the mode shapes by analyzing mode shapes .Thus the damage can be fixed by changing the natural frequency or other parameters. We come to know of the mode shapes and their changes according to the frequencies by using the 'Modal Analysis' feature of ANSYS. In this way the unwanted vibrations are determined and eliminated by further respective processes. This paper shows only mode shapes and their results, in the next step we will do the analysis of mode shapes.

[2] Ladder type chassis frame is analyzed in this study. The Chassis consists of side members attached with a series of cross members to form the ladder like structure. They were designed for functionality and provided little torsional stiffness. For stress analysis the FEM tool is used. Determination of critical regions in the chassis frame is carried out by FEM with required boundary conditions. Design modification has been done based on the results obtained. To determine the natural frequency and mode shapes of the system the modal analysis of the chassis frame is done. The rigidity of the system was analyzed and their resonance could be avoided.

[3] The vibrational characteristics of the car chassis including the natural frequencies and mode shapes are studied in this paper. Chassis forms the structural backbone of an automobile vehicle. The car chassis is excited by dynamic forces caused by the road roughness, engine, transmission and more. Modal analysis using Finite Element Method (FEM) can be used to determine natural frequencies and mode shapes. The modal analysis has been carried out by the Commercial finite element packaged ANSYS in this paper. The simulation of the model with appropriate accuracy and with considering the effect of bolted and riveted joints has been carried out. The model has been analyzed and first 6 frequencies that play important role in dynamic behavior of the chassis have been explained.

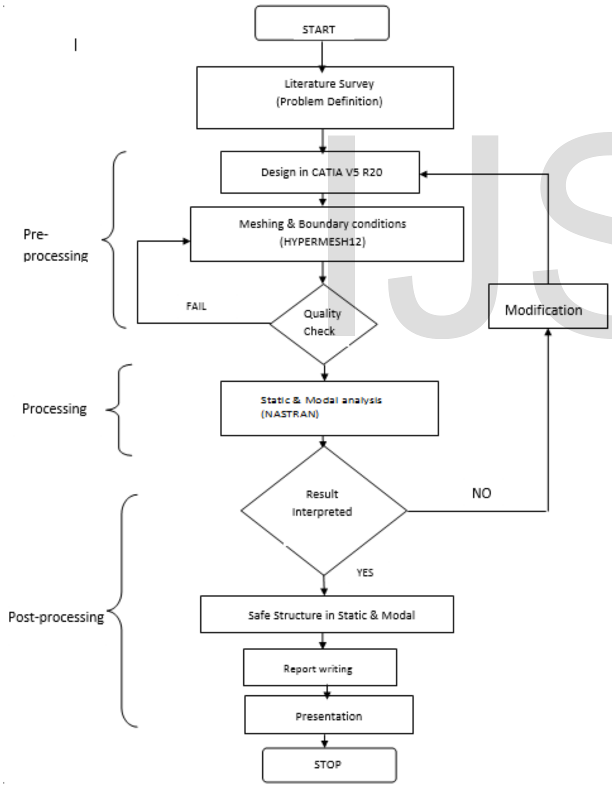
International Journal of Scientific & Engineering Research, Volume 8, Issue 3, March-2017
ISSN 2229-5518

I. Problem Definition and Objectives

It identifies that the work should focus on vibration aspects of chassis design. Therefore, the modelling and analysis of truck chassis and study of the natural frequencies is aimed in the present study. In the present study, we have consider the following model.
The project objectives are as follows:

- Determination of the modal frequency and mode shape of the chassis by using modal analysis.
- To study the dynamic behaviour of the truck chassis by changing the geometry, topology and structural properties of the chassis.
- To improve chassis design by reduction of the weight of chassis.

II. Methodology



III. Design And Meshing

The chassis is designed in order to carry the loads and weights of other structural components. There are three phases in which modelling and analysis is carried out. In first phase, the FE model of the truck chassis frame is built using CATIA V5 R20. The detailed CAD model is shown in Figure 3.1, which shows the dimensions of the truck chassis frame in detail.

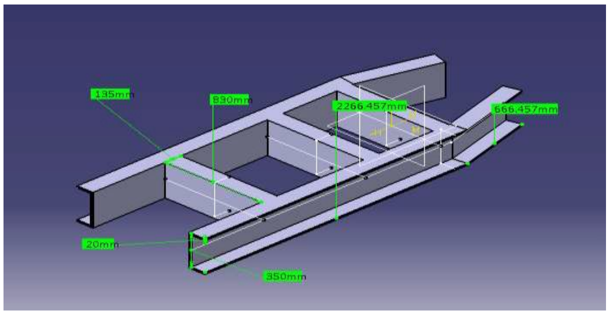


Figure: 3.1 Chassis frame CAD model

The second phase consists of exporting the CAD model to Hypermesh 12. In Hypermesh 12, meshing is done and boundary conditions are applied. During mesh operations, some quality parameters should be maintained viz. aspect ratio, skew, Jacobean, minimum element size, warpage. The model after meshing is shown in Figure 3.2. The total number of elements formed is 17309. The size of the element is maintained 15 mm throughout. The chassis is made of structural steel.

The material properties are:

Material: Steel 1020

Young's modulus: 210 GPa

Mass Density: 7.8300E-09 tonnes/mm^3

Poison's ratio of the material: 0.3

Then in the third phase, the modal analysis is carried out, in which the following assumptions are made:

- Damping should be ignored in the modal analysis.
- All applied loads should be ignored, if any.
- Material is presumed to be homogeneous, isentropic & elastic.
- All the geometrical and material properties of the model are given in SI unit system.
- Block Lanczos method is applied for extraction and also for expansion of the modes.
- Results which include natural frequencies of the model and mode shapes are viewed in ANSYS workbench in post- processing.

Modal analysis is performed after creating the chassis finite element model. This finite element is meshed in free-free state with no constraints. The results are calculated for the initial 12 frequency modes. And the results indicate that road simulations are the most important concern for chassis. In modal analysis, the use of subspace method in ANSYS is done. Since there were no constraints considered, the initial 6 frequency modes are close to zero value (order 10^{-2}).

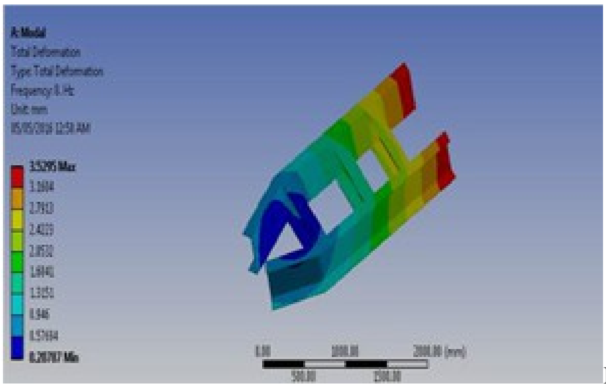


Fig.3.3 Modal Analysis in ANSYS

IV. Results

By using ANSYS software, 12 modes of minimum frequency are obtained by applying Block Lanczos extraction method. It is observed that the natural frequencies of the model are obtained to be high when the stiffness is of high value.

4.1 Mode Shapes of Original Model

Mode shapes can be defined as the shapes of the structure at different natural frequencies. The determination of the mode shapes is done by using Eigen value of vibration equations.

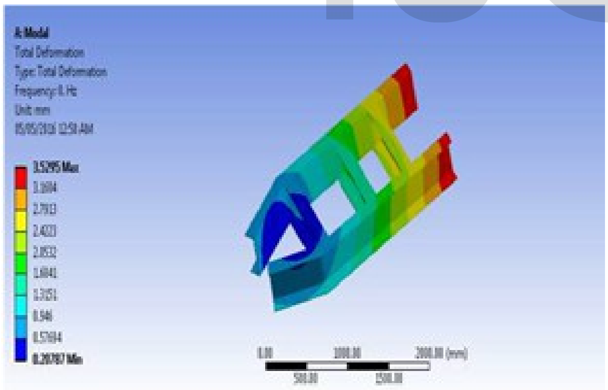


Figure 4.1.1 Mode 1-Natural frequency 0 Hz

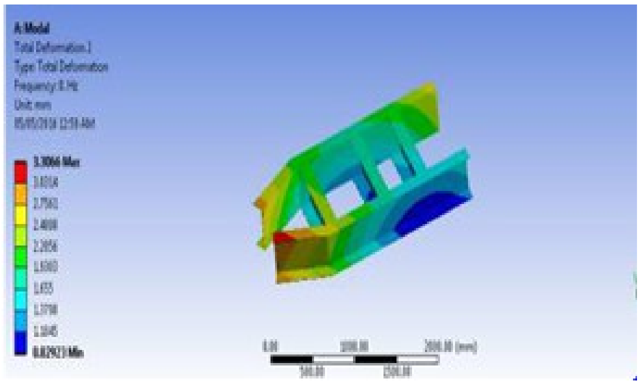


Figure 4.1.2 Mode 2-Natural frequency 0 Hz

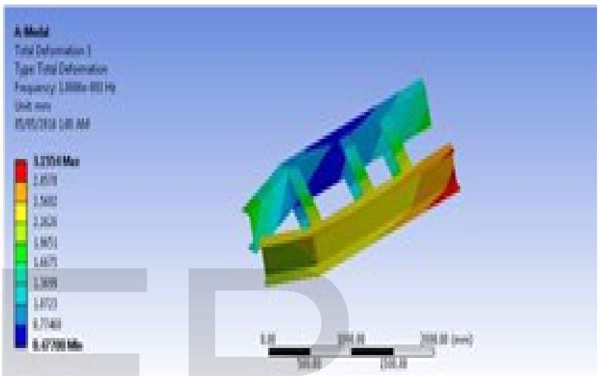


Figure 4.1.3 Mode 3-Natural frequency 0 Hz

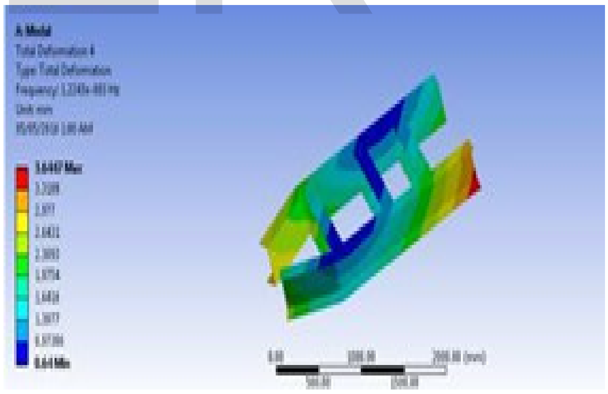


Figure 4.1.4 Mode 4-Natural frequency 0 Hz

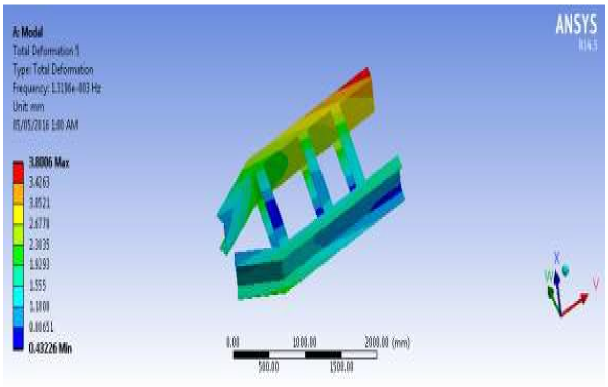


Figure 4.1.5 Mode 5-Natural frequency 0 Hz

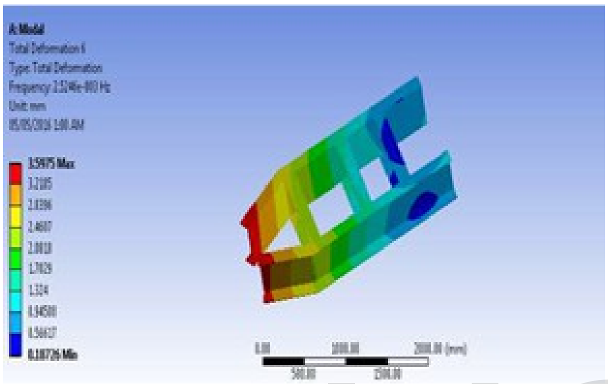


Figure 4.1.6 Mode 6-Natural frequency 0 Hz

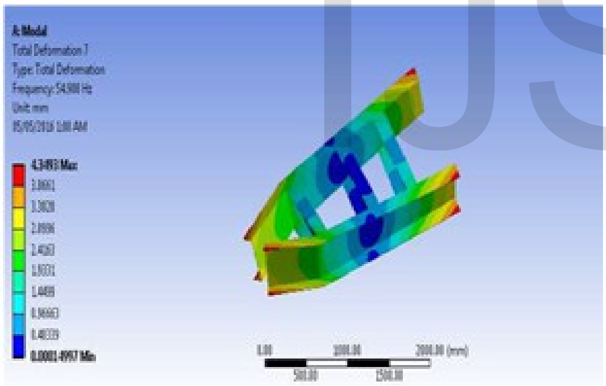


Figure 4.1.7 Mode 7-Natural frequency 54.98 Hz

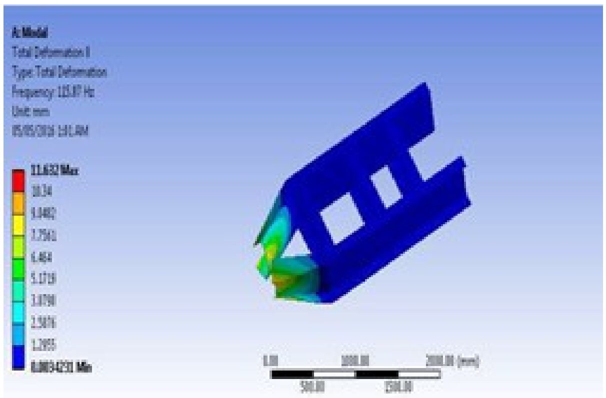


Figure 4.1.8 Mode 8-Natural frequency 115.87 Hz

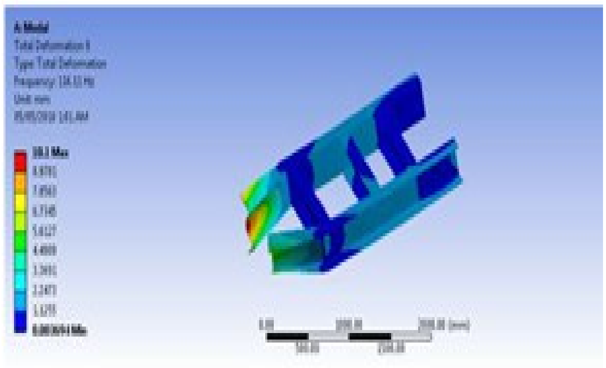


Figure 4.1.9 Mode 9-Natural frequency 116.11 Hz

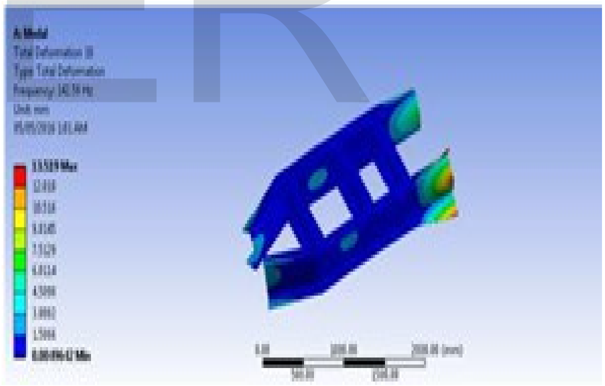


Figure 4.1.10 Mode 10-Natural frequency 142.59 Hz

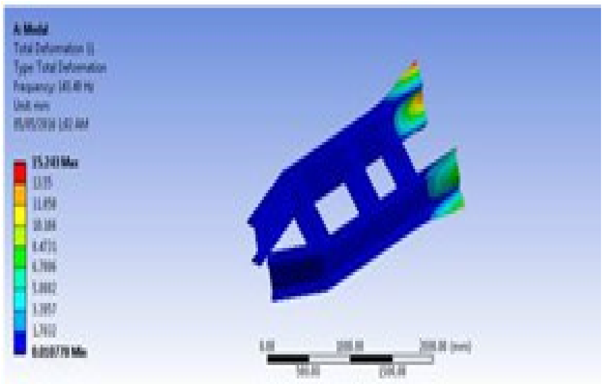


Figure 4.1.11 Mode 11-Natural frequency 143.49 Hz

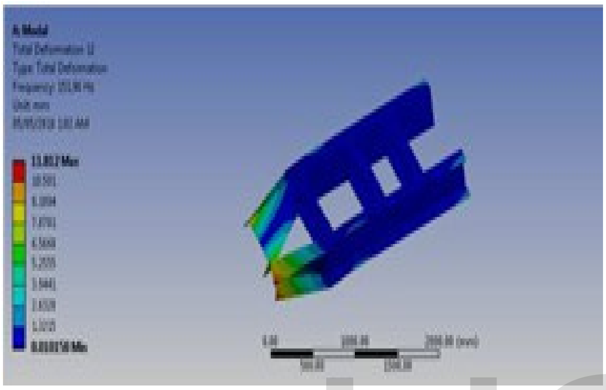


Figure 4.1.12 Mode 12-Natural frequency 151.96 Hz

Table 4.1 Mode numbers & frequencies of the original model

MODE NUMBER	NATURAL FREQUENCY(Hz)
1	0
2	0
3	0
4	0
5	0
6	0
7	41.251
8	114.1
9	121.64
10	127.78
11	142.88
12	151.96

Results' graph:

For the respective model, the frequencies of vibrations and the mode shapes are shown in fig. 4.2.1 to 4.2.12. The graph of the mode shapes versus frequencies is plotted in fig. 4.1.13.

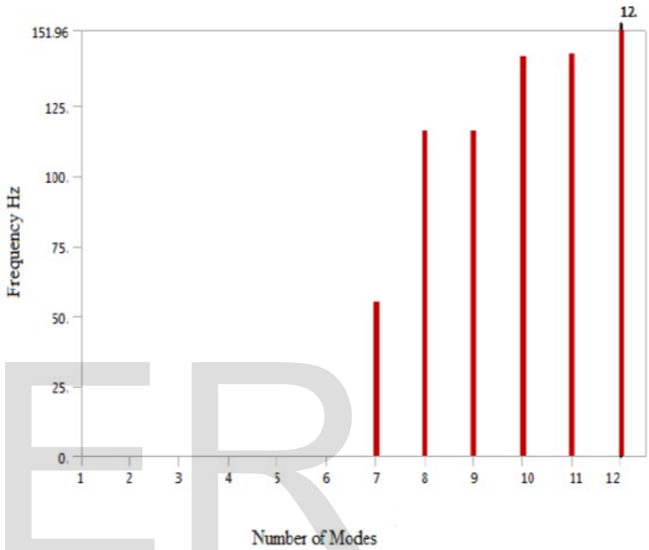


Figure 4.1.13 Mode shapes versus frequency

4.2 Mode Shapes of the Improved Model.

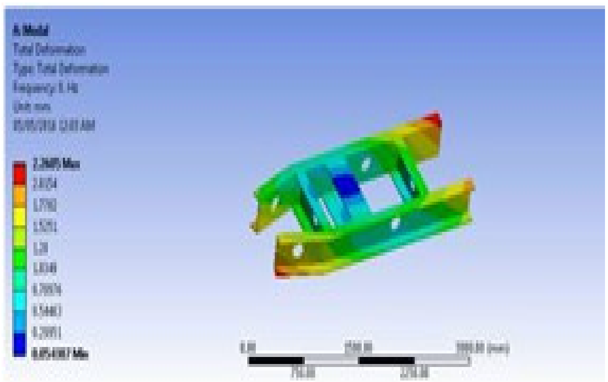


Figure 4.2.1 Mode 1-Natural frequency 0 Hz

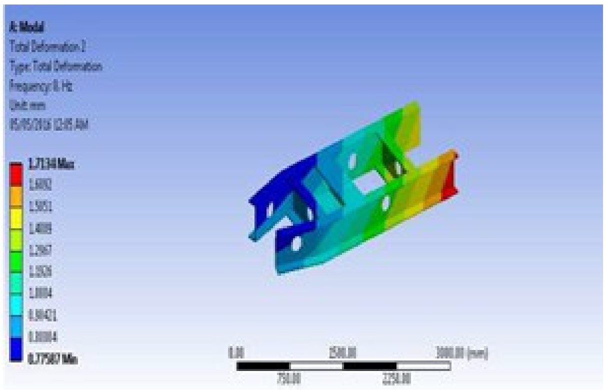


Figure 4.2.2 Mode 2-Natural frequency 0 Hz

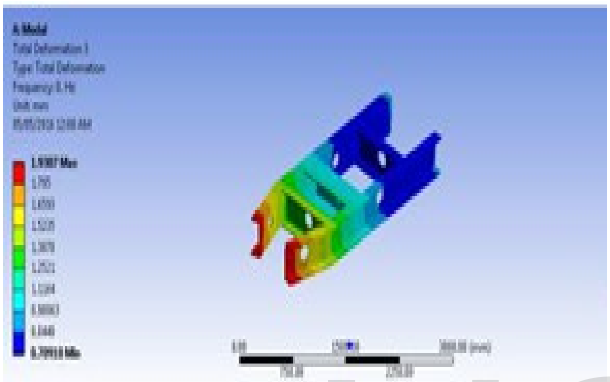


Figure 4.2.3 Mode 3-Natural frequency 0 Hz

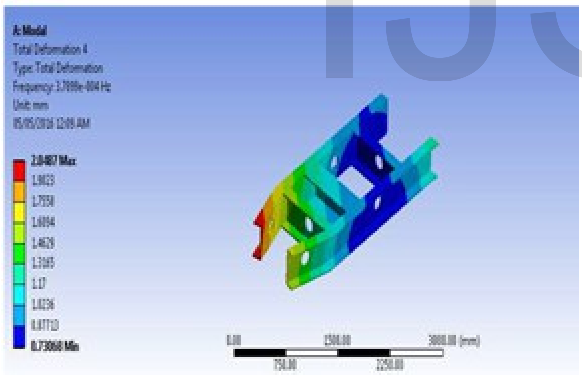


Figure 4.2.4 Mode 4-Natural frequency 0 Hz

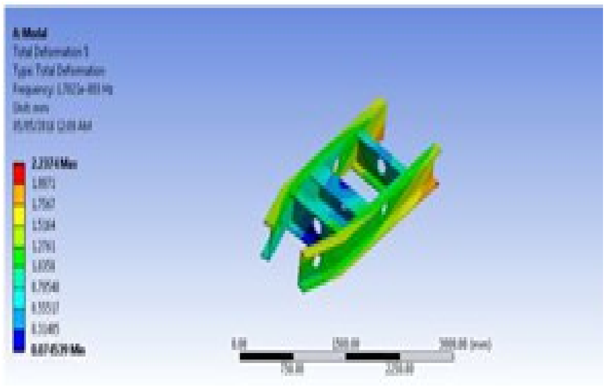


Figure 4.2.5 Mode 5-Natural frequency 0 Hz

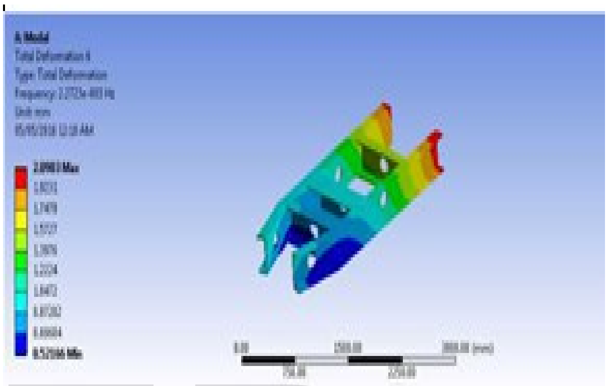


Figure 4.2.6 Mode 6-Natural frequency 0 Hz

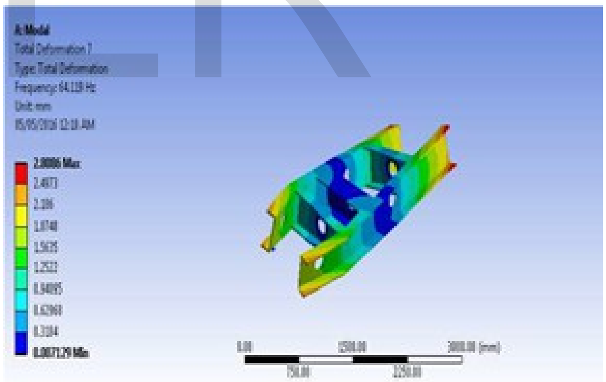


Figure 4.2.7 Mode 7-Natural frequency 64.119 Hz

International Journal of Scientific & Engineering Research, Volume 8, Issue 3, March-2017
ISSN 2229-5518

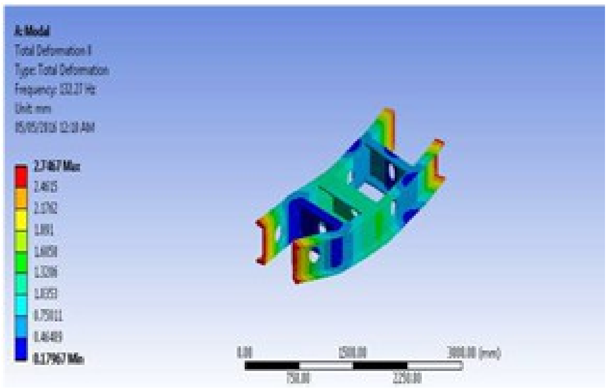


Figure 4.2.8 Mode 8-Natural frequency 132.10 Hz

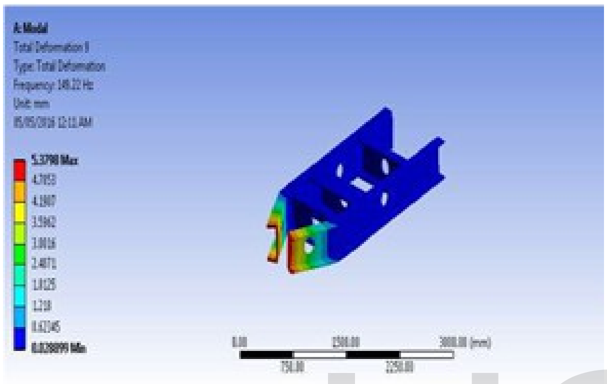


Figure 4.2.9 Mode 9-Natural frequency 149.22 Hz

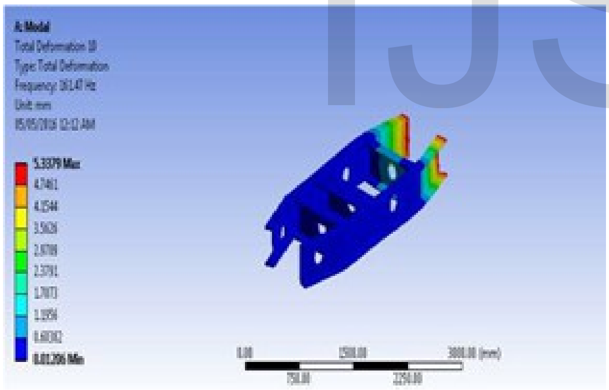


Figure 4.2.10 Mode 10-Natural frequency 161.47 Hz

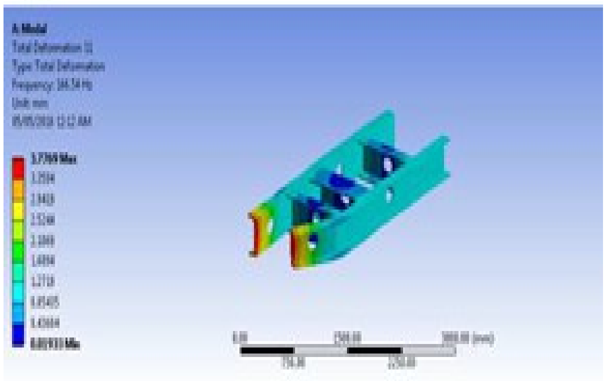


Figure 4.2.11 Mode 11-Natural frequency 166.54 Hz

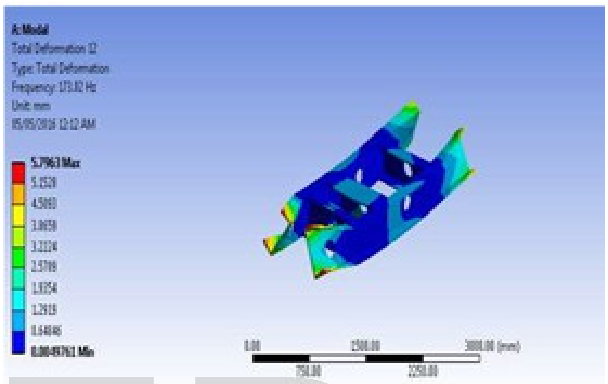


Figure 4.2.12 Mode 12-Natural frequency 173.82 Hz

Table 4.2 Mode Number & frequencies of the improved model

MODE NUMBER	NATURAL FREQUENCY(Hz)
1	0
2	0
3	0
4	0
5	0
6	0
7	64.119
8	132.1
9	149.22
10	161.47
11	166.54
12	173.82

Results' graph:

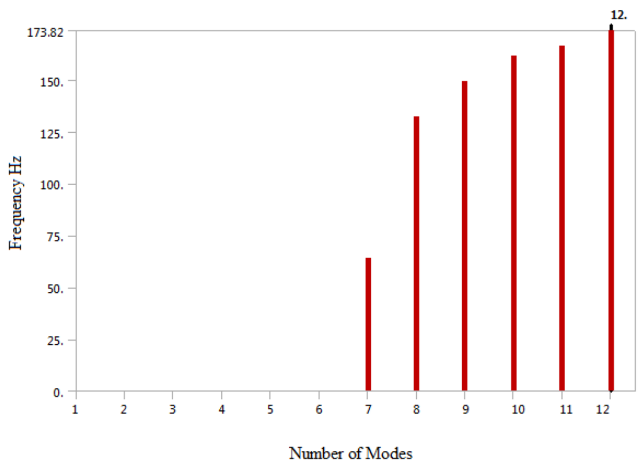


Figure 4.2.13 Mode shapes versus frequency for the improved model

4.3 Comparison of the original and improved model.

In the following table, the natural frequencies for the corresponding mode shapes have been compared for the original and improved model.

Table 4.3 Comparison of the natural frequencies

Number of Modes	Natural Frequency Hz	
	Original model	Improved model
1	0	0
2	0	0
3	0	0
4	0	0
5	0	0
6	0	0
7	41.251	64.119
8	114.1	132.1
9	121.64	149.22
10	127.78	161.47
11	142.88	166.54
12	151.96	173.82

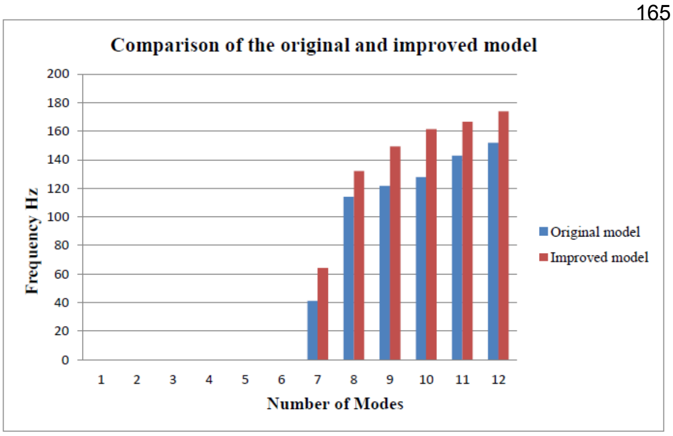


Figure 4.3.1 Mode shapes versus frequency graph for original and improved model

From the table 4.3 and graph fig. 4.3.1, it is observed that the modal frequency of the model is increased from 0 to 173.82 Hz for the mode shape 1 to 12 in the improved model. And hence the modal analysis can be conveniently used in the design phase. It can help the designer to understand and predict the vibration characteristics of the design model in order to construct a design for the better stability and strength of the chassis.

V. Conclusion

- It can be concluded that the initial six modes (closer to zero frequencies) will produced rigid body motion. It indicates that there are no connectivity issues in the FEMs.
- From the observation of 7th and 12th mode for original model, which shows positive modes with the frequencies greater than 40Hz for 7th mode and 12th mode with frequencies under 150Hz, it is concluded that no resonance will be observed if the components that are to be mounted on the frame have their natural frequencies below 40Hz or above 150Hz.
- From the observation of 7th and 12th mode for improved model, which shows positive modes with the frequencies greater than 64Hz for 7th mode and 12th mode with the frequencies under 174Hz, it is concluded that no resonance will be observed if the components that are to be mounted on the frame have their natural frequencies below 64Hz or above 174Hz.
- In general, the fuel tank, fuel filter, electronic air processing unit (EAPU) and their mounting brackets, etc. are mounted on the chassis frame. Therefore, it should be implemented that these components do not produce any vibrations when put into operation keeping in consideration the results of modal analysis.

VI. Future Scope

- The natural frequencies of vibrations that are obtained in modal analysis can be used in the further system analysis of the frame.
- This modal analysis can be extended further for analysis including the pre-stressed loads and the damping characters.
- The modal analysis can be extended further to understand the behaviour of the chassis frame under metaphysics problems.
- The modal analysis can be extended further to understand the modal behaviour of the chassis frame under fracture conditions.

VII. Acknowledgement

We would like to express our sincere gratitude and appreciation to the Department of Automobile Engineering, Saraswati College of Engineering, Kharghar.
We feel a great privilege to have had **Prof. Vishnudas Chodankar,** Department of Automobile Engineering, Saraswati College of Engineering, Kharghar, guiding our project. We are indebted for their extremely valuable guidance and hearted co-operation throughout the process. Their constant encouragement and advice gave us the inspiration for the complication of this dissertation.
We express our sincere thanks to **Mr. T. Z. Quazi,** professor and head of the Department of Automobile Engineering, Saraswati College of Engineering, Kharghar, for the generous fueling over the entire duration of this project.
Finally, our sincere thanks to all the staff members of Department of Automobile Engineering, Saraswati College of Engineering, Kharghar, and all our friends, who have contributed directly or indirectly to make this project successful.

VIII. References

[1.] Goutham Solasa, Nariganani S D Satadeep,T.Raghu Krishna Prasad, G.Suresh Babu *"Modal analysis of chassis"*, International Journal of Engineering and Advanced Technology (IJEAT) ISSN: 2249 – 8958, Volume-2, Issue-4, April 2013.

[2.] Madhu P. and Venugopal T. ,**"***Static analysis, design modification and modal analysis of structural chassis frame"* , India ISSN: 2248-9622, Vol. 4, Issue 5(Version 3), pp.06-10 , May 2014.

[3.] Pravin A Renuke ,*"Dynamic analysis of chassis"* ISSN: 2248-9622 www.ijera.com Vol. 2, Issue 6,, pp.955-959, November- December 2012.

[4.] *"An Introduction to the Finite Element Methods"* J. N. Reddy, Tata McGraw Hill. Edition, 2003.

[5.] ANSYS Workbench @Help system, 14.5.