DYNAMIC ANALYSIS OF AUTOMOTIVE CHASSIS USING FEA

Kallappa Khannukar[1], Vinayak Kallannavar[2], Dr. B. S. Manjunath[3]

[1] Assistant Professor, Mechanical Engg. Dept, MMEC Belgaum, Karnataka, India
[2] Trainee Engineer, Quest Global Engineering Pvt. Ltd, Karnataka, India
[3] Professor, Mechanical Engg. Dept, KLE's Dr. MSSCET Belgaum, Karnataka, India

---

Abstract – ***Chassis provide the structural integrity to automotive vehicles, where as it is also important to have the stiff and light weight structure. The present study lay the emphasis on dynamic analysis of chassis. The modal analysis is carried out using both analytical and FEA techniques, and the results were found to be in good agreement with each other. Also the chassis structure is optimized by varying the design parameters with the help of acceleration response of the system. It was observed that the acceleration levels were reduced to 20.24% by increasing the thickness of ribs and horizontal channel to twice of their initial dimensions.***

Key Words: *FEA, Hypermesh, Chassis, Stiffness.*

1. Introduction

Chassis is the structural backbone of a passenger vehicle. When a vehicle travels, the chassis experience the dynamic forces due to the road roughness, vibrations of engine, transmission systems etc. Due to the dynamic excitations, chassis will tend to vibrate and can lead to ride discomfort, ride safety and stability problems. It is difficult to mathematically model and solve these mechanical problems by analytical techniques as they include complex nonlinear boundary conditions. This difficulty is overcome by seeking some form of numerical solutions. The finite element method (FEM) is the most appropriate and accurate numerical technique to solve the compound problems.

In past few decades many academicians, researchers have worked to analyze and understand the dynamic nature of the vehicle chassis using numerical, analytical and experimental methods under various loading conditions [1-4]. The sensitivity analysis is carried out by Hirak. P et al. [5] on the truck chassis to optimize the weight. They achieved 17% of weight reduction in the truck chassis by analyzing it at different sections. Dr. R. Rajappan et al. [6] simulated the dynamic characteristics of truck chassis using ANSYS FEA tool. The mode shapes and natural frequencies at mounting locations of the chassis are studied and the modified design is proposed. S. Ganesan et al. [7] carried out modal analysis of bus chassis using different shapes of reinforcement plates to increase its stiffness. The present work addresses the optimization of ladder type chassis under dynamic loading conditions.

2. Finite Element Analysis

The CAD model of the chassis is prepared in Catia V5 modeling environment as shown in Fig -1 and MSC Nastran simulation tool is used for the FEA analysis. For the ease of analysis the ladder type chassis is divided into subpart like side rails, C-channels, horizontal channels, cross members, and brackets as shown in Fig -2. Steel AISI 1015 is used as the chassis material and its material properties are listed Table 1.

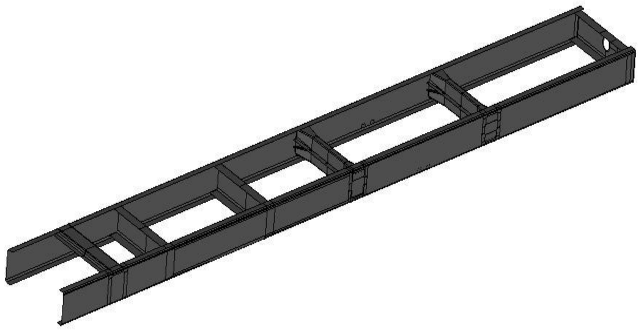

Fig -1: CAD model of the chassis

2.1 Modal Analysis

The modal analysis is the field of measuring and analyzing the dynamic response of structure when excited by an input disturbance. Modal analysis of the system is closely associated with the natural frequency of the system. Stiffness (k) and mass (M) of the system are the deciding factors for the natural frequency and can be given by equation 1,

$$\omega_n = \frac{1}{2\pi}\sqrt{\frac{k_{eq}}{M}} \qquad (1)$$

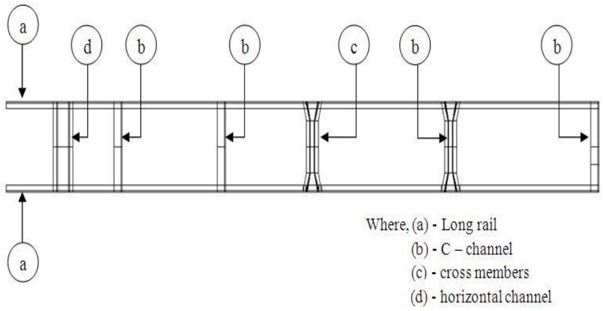

Fig -2: Sectional view of the chassis

Table -1: Material properties of AISI 1015 steel

Property	Value
Young's Modulus (N/m^2)	2.1e5
Density (kg/m^3)	7860
Poisson's Ratio	0.3

As the stiffness of system depends on the arrangement of the components (Series/Parallel), the stiffness of individual components are calculated and listed in Table 2.

Table -2: Name of the Table

Sl. No	Natural frequency	Description
1	5.173275x10^{-04}	Rigid body translation along x – axis
2	3.474783x10^{-04}	Rigid body translation along y – axis
3	2.004669x10^{-04}	Rigid body translation along z – axis
4	1.759311x10^{-04}	Rigid body rotation along x – axis
5	3.893044x10^{-04}	Rigid body rotation along y – axis
6	5.696454x10^{-04}	Rigid body rotation along z – axis
7	7.931197x10^{+00} (1st mode)	Torsional mode along x-axis
8	1.599375x10^{+01} (2nd mode)	Torsional mode along y-axis
9	2.364815x10^{+01} (3rd mode)	Bending mode along x – axis
10	2.456802x10^{+01} (4th mode)	Bending mode along y – axis
11	3.060098x10^{+01} (5th mode)	Torsional mode along z-axis
12	4.038740x10^{+01} (6th mode)	Torsion + Bending mode along z – axis
13	9.04825x10^{+01} (23rd mode)	Torsion + Bending mode along x – axis

From Table 2,
Equivalent stiffness Keq = 136.98x10^6 N/m

Total mass of the chassis = 438 Kg
Substituting these values in equation 1,

$$\omega_n = \frac{1}{2\pi}\sqrt{\frac{136.98\times10^6}{438}} = 89Hz$$

The finite element model was generated using HYPERMESH which includes ~88500 elements, and the free-free boundary condition was adopted in order to obtain the natural frequencies and mode shapes.

There were thirteen natural frequencies are calculated for the normal mode analysis and are tabulated in Table -2. 23rd natural frequency was near to the value calculated by analytical method. Normally, the operating frequency is always related to dynamic forces induced by road roughness, bumps, engine, transmission and many more. Each of these forces has its own excitation frequency. Fig -3 shows the mode shapes obtained for some of the natural frequencies of the chassis.

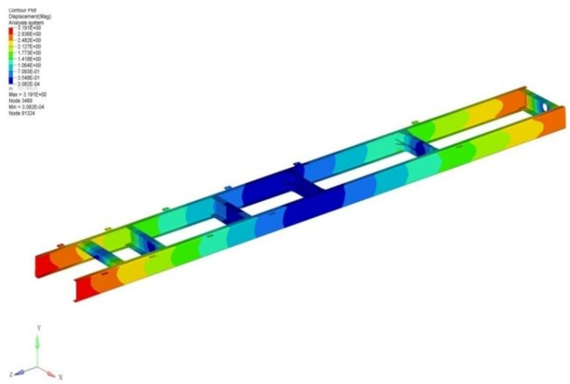

Fig -3.1: Mode shape at 7.94 Hz

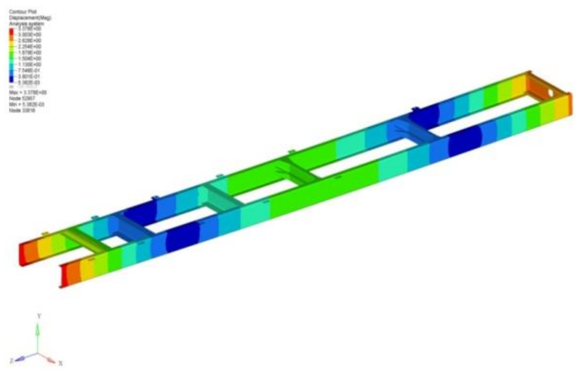

Fig -3.2: Mode shape at 15.99 Hz

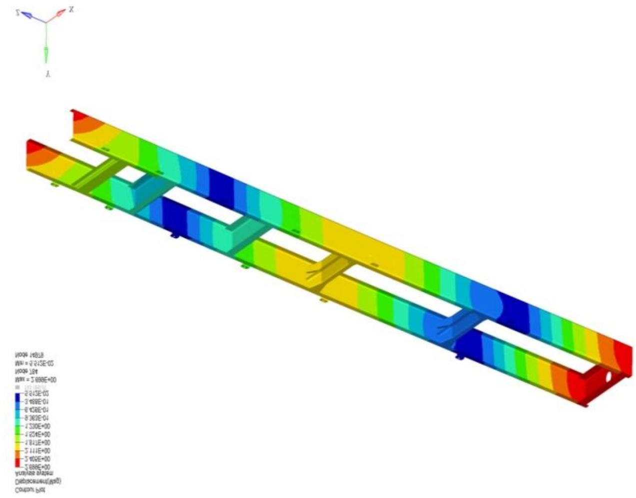

Fig -3.3: Mode shape at 23.65 Hz

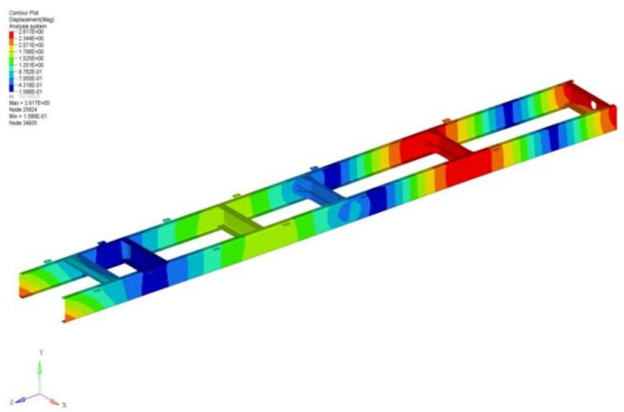

Fig -3.4: Mode shape at 24.57 Hz

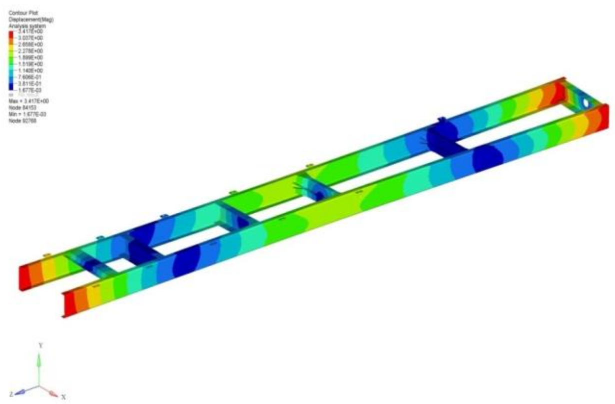

Fig -3.5: Mode shape at 30.61 Hz

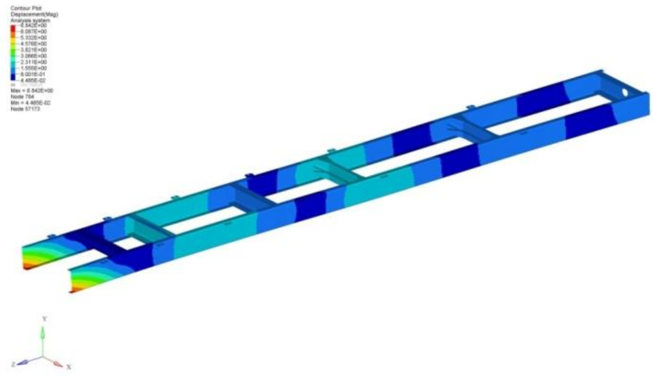

Fig -3.6: Mode shape at 40.39 Hz

From the above results it is observed that natural frequency calculated by analytical method (89 Hz) is almost is close to the 23rd natural frequency i.e., 90.48 Hz. Table 3 shows result comparison of analytical &FEA methods.

Table -3: Comparison between FEA & analytical results

Sl. No	Natural frequency (Hz)		Percentage difference
	Theoretical	FEA	
1	89	90.48	1.67%

2.2. Transfer Path Analysis

Transfer path analysis (TPA) is used to assess the structure borne energy paths between excitation source and receiver location. Transfer path analysis is a systematic method to understand the relation between multiple sources of noise and vibration and their effect on perceived user comfort and health.

2.2.1 Acceleration Response of Chassis

Acceleration response is the acceleration levels of the chassis when excited by external force. From the results it was observed that maximum acceleration was at the horizontal channel. Fig -4 shows the fringe plot of the chassis when excited by unit acceleration and Fig -5 illustrates the acceleration response of the chassis at node 522127 where acceleration is maximum.

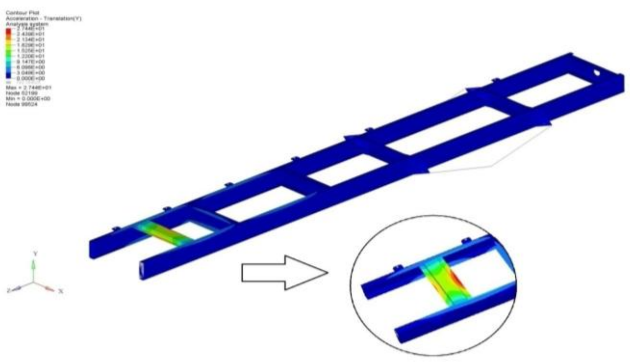

Fig -4: Fringe plot of the chassis

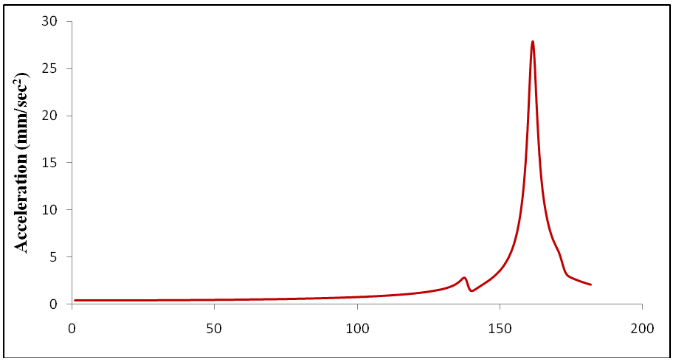

Fig -5: Acceleration response of the chassis at node 52217

The aceeleration Vs frequency plot recorded in Fig -5 shows the maximum acceleration of 27.48 mm/sec^2 at a frequency of 162 Hz. The effort has been made to enhance the attenuation characteristics of the chassis by making small changes in the design parameters.

a. Increasing thickness of horizontal channel

Thickness of the horizontal channel was 5 mm, to improve its vibratory characteristics its thickness was doubled as shown in Fig -6.1. From Fig -6.2 it is evident that the maximum acceleration value for the present load case is 21.22 mm/sec^2.

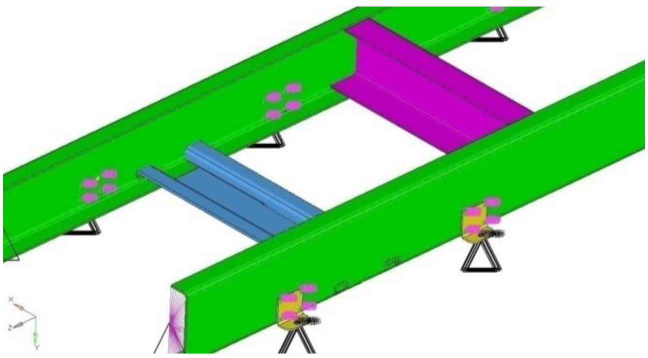

Fig -6.1: Horizontal channel section thickness doubled

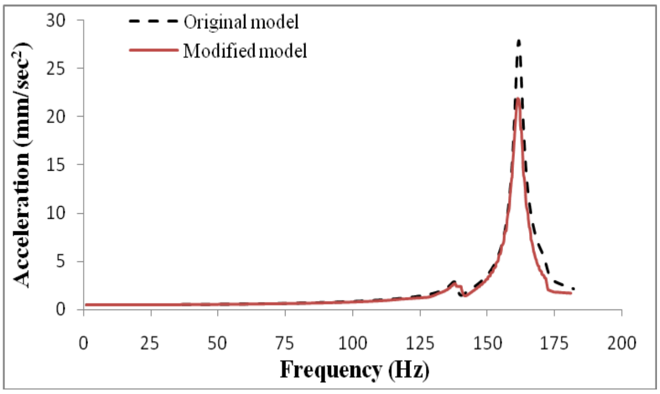

Fig -6.2: Accleration response comparison of modified & original model

b. Addition of ribs along long rail members and horizontal channel only

The ribs of 5 mm thickness were added on the horizontal channel and long rail members (Fig -7.1). Reduction in acceleration level of 12.78% is observed (Fig -7.2).

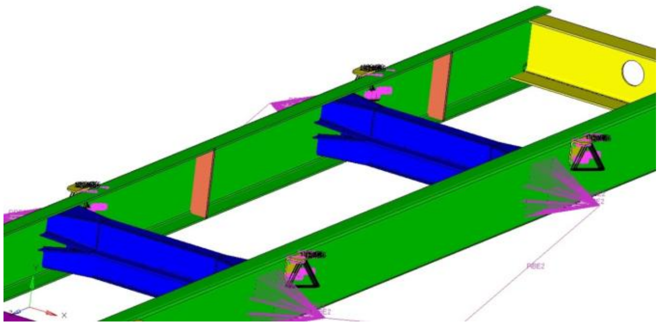

Fig -7.1: Addition of ribs along horizontal channel and long rail members

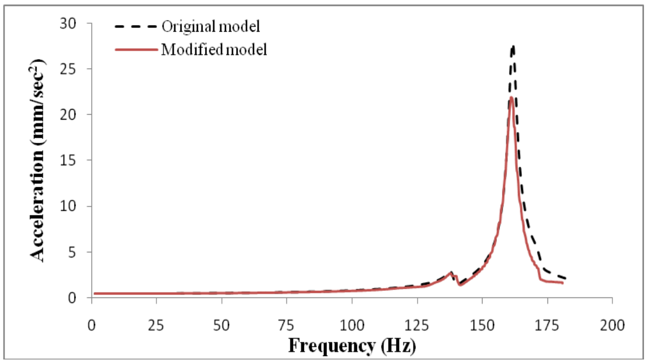

Fig -7.2: Accleration response comparison of modified & original model

c. Addition of ribs along other C-channel sections

Ribs were added along C-channels of the chassis as shown in Fig -8.1. Reduction in acceleration level of 13.14% is observed for the present loading condition (Fig -8.2).

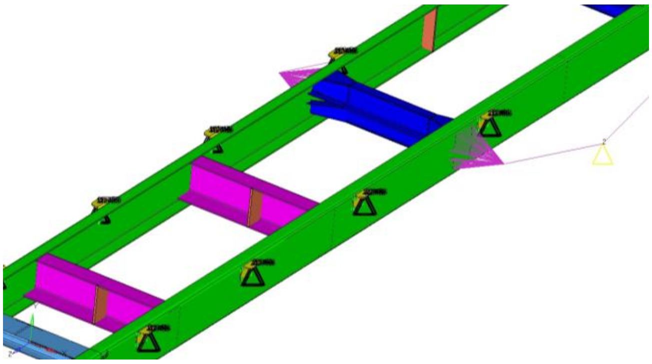

Fig -8.1: Addition of ribs along other C-channel sections

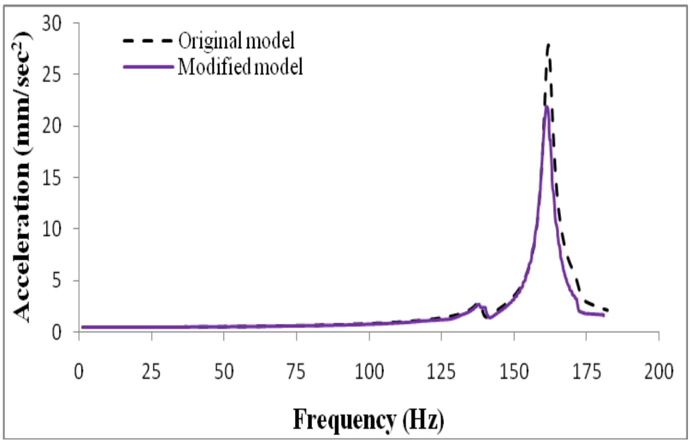

Fig -8.2: Accleration response comparison of modified & original model

d. Increasing the number of ribs to 3 on horizontal channel only

To reduce the acceleration levels three ribs of 5 mm hickness were added to the horizontal channel (Fig -9.1). 12.27% of reduction in acceleration level is evident from Fig -9.2.

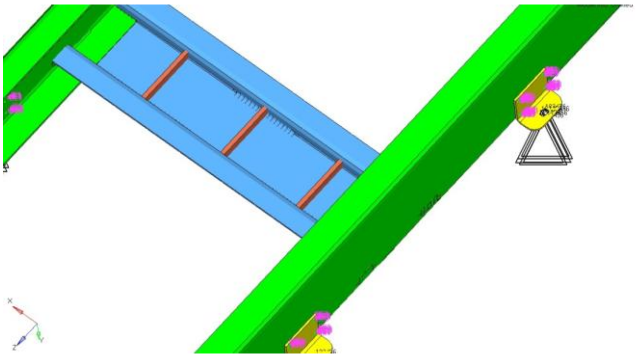

Fig -9.1: Increasing the number of ribs to 3 on horizontal channel only

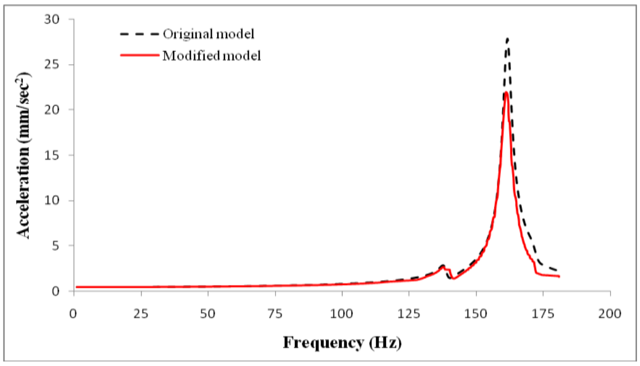

Fig -9.2: Accleration response comparison of modified & original model

e. Increasing thickness of ribs to twice of initial thickness

To study the effect of thickness of ribs on horizontal channel thickness of the rib was doubled. Present load case was solved with increasing the thickness of the rib to twice that of initial thickness (Fig -10.1). Thickness maintained during this load case was 10 mm. Reduction in acceleration level of 7.03% is observed from Fig -10.2.

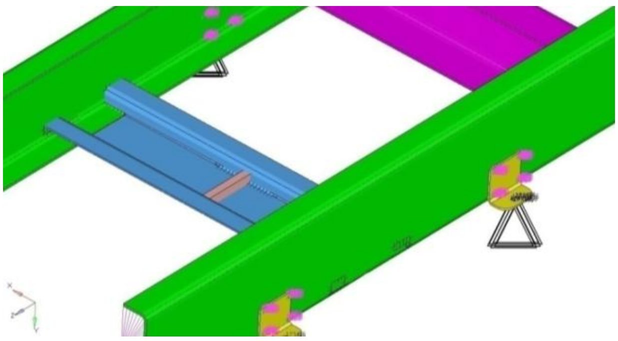

Fig -10.1: Increasing thickness of ribs to twice of initial thickness

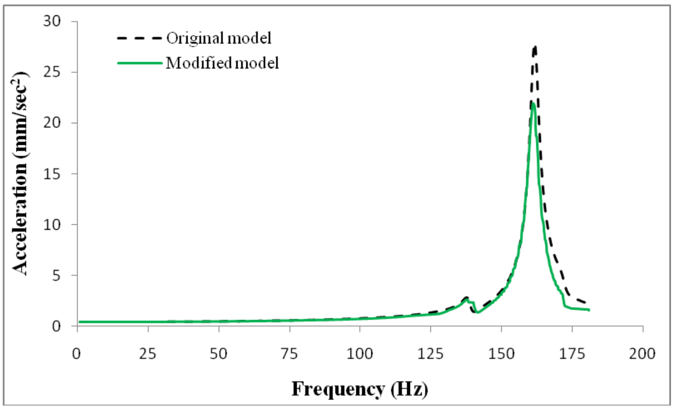

Fig -10.2: Accleration response comparison of modified & original model

f. Increasing thickness of ribs and horizontal channel to twice of that initial section

Present load case determines the combined effect of thickness of rib and horizontal channel on the acceleration levels of the chassis. In the present load acceleration levels of the chassis was determined by increasing the thickness of ribs and horizontal channel. From Fig -11.2 it is apparent that reduction in acceleration level of 20.24% is achieved.

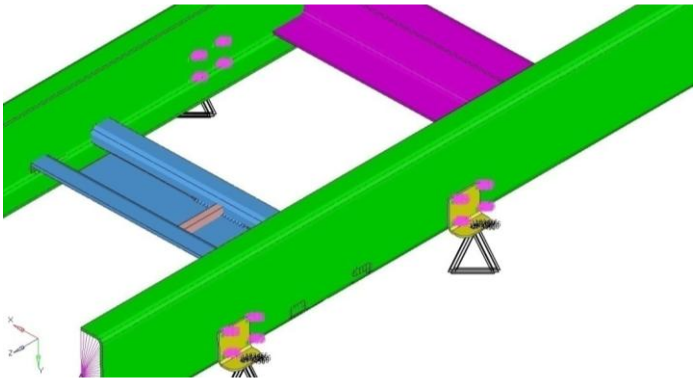

Fig -11.1: Increasing thickness of ribs and horizontal channel to twice of that initial section

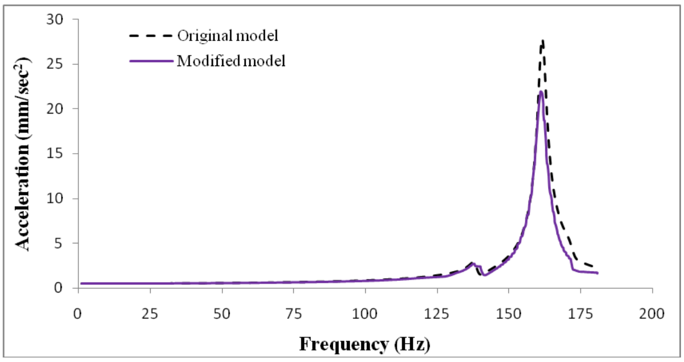

Fig -11.2: Accleration response comparison of modified & original model

3. CONCLUSIONS

In the present study dynamic characteristics of a chassis like natural frequency and mode shape are evaluated by finite element analysis. Values of natural frequency obtained by finite element method were in good agreement with analytical results. Acceleration responses of the chassis excited by road surface were analyzed. From the results it was observed that by increasing thickness of ribs and horizontal channel to twice of that initial section acceleration levels were reduced by 20.24%.

REFERENCES

[1] Pravin A Renuke, *International Journal of Engineering Research and Applications,* Vol. 2, Issue 6, November-December 2012.

[2] Haval Kamal Asker, Thaker Salih Dawood and Arkan Fawzi Said, *ARPN Journal of Engineering and Applied Sciences,* VOL. 7, NO. 6, JUNE 2012.

[3] Thanneru Raghu Krishna Prasad, Goutham Solasa, Nariganani SD Satyadeep, G.Suresh Babu, *International Journal of Engineering and Advanced Technology (IJEAT),* ISSN: 2249 – 8958, Volume-2, Issue-5, June 2013.

[4] Goolla Murali, Subramanyam.B, Dulam Naveen, "Design Improvement of a Truck Chassis based on Thickness", *Altair Technology Conference* India 2013.

[5] Hirak Patel, Khushbu C. Panchal, Chetan S. Jadav, *International Journal of Engineering and Advanced Technology,* ISSN: 2249 – 8958, Volume-2, Issue-4, April 2013.

[6] Dr.R.Rajappan,M.Vivekanandhan, *National Conference on Emerging Trends In Mechanical Engineering*, 2013.

[7] S. Ganesan and K. Panneerselvam, *ARPN Journal of Engineering and Applied Sciences,* VOL. 10, NO. 8, MAY 2015.