IJETST- Vol.||01||Issue||10||Pages 1662-1667||December||ISSN 2348-9480 2014

Open access Journal International Journal of Emerging Trends in Science and Technology

Optimization of Truck Chassis for Frequency Response

Authors
Dr A. Gopi Chand[1], S.V. Prasad[2], D. Bhanu Prakash[3]
[1]Professor and HOD, Department of Mechanical Engineering, Swarnandhra College of Engineering and Technology, NARSAPUR, INDIA
[2]P.G student, Department of Mechanical Engineering, Swarnandhra College of Engineering and Technology, NARSAPUR, INDIA
[3]Associate Professor, Department of Mechanical Engineering, Swarnandhra College of Engineering and Technology, NARSAPUR, INDIA

ABSTRACT
FE analysis provides an optimal and quick way to optimize the structural design of any vehicle. When considering the structural design of a vehicle, chassis design plays an important role for all the parts are attached to the chassis only. The better the design of the chassis better is the vehicle. Most of the current heavy vehicles are using C Sections for their chassis design. The current work focuses on using I-section in place of C Section. For this an 8X8 heavy truck chassis is taken and a modified with I sections. Different analyses, namely static, modal and harmonic analyses are performed on both the chasses. It is found that the I-section chassis performs better than C section chassis. Results are discussed in the paper.
Keywords: *Vehicle dynamics, FEA, Heavy truck chassis, Static Analysis, Modal Analysis, Harmonic Analysis, C-Section, I-Section*

1. INTRODUCTION

For any vehicle structure, Chassis is the main part. All the parts like engine, transmission, cabins etc are all attached to the chassis. The chassis is subjected to different kinds of loading. Static loading is due to the Curb and payload while dynamic loading is due to the excitation by engine vibrations and road undulations. Thus it is important to optimize the response of the chassis for frequency response. Many researchers have worked in this area.

Denish, et al [1] performed static analysis on the frame of a roll cage cart to study its response to static loading. Murali, et al [2] used Harmonic analysis to investigate effect of thickness on the performance of the vehicle frame under different loadings. Venugopala Rao and Prassana Priya [3] modeled a three wheeler auto chassis using Beam, mass and combination elements. Ashif Iqubal, et al [4] analytically predicted the stresses in a trailer frame longitudinal members. Dr S.B.Rane, et al[5] optimized the weight of the fork lift truck chassis by taking the thickness of the members as design variable while stress as a constraint. FE Techniques used for this purpose. Asker, et al [6] performed dynamic analysis to study the effects of the dynamic loading that a mini dump truck chassis is subjected to when it traverses over a ramp. William Davis, et al [7] discussed the procedure for chassis design of Formula SAE race care. Static analysis with given loads coming in various situations are performed on the Chassis for understanding its response. Ojo Kurdi, et al [8] investigated the effect of cyclical loading due to road undulations on the truck chassis using FE analysis. The cyclical loads that the chassis is subjected to due to road undulations as well as due to engine are measured experimentally. The chassis model’s performances during full frontal, offset, and corner impacts are investigated by Yucheng Liu [9] using LS-DYANA. Musa [10] performed static analysis to study the effect of bending as well as torsional loads on the truck chassis using FEA and validated them experimentally. Biradar, et al[11] optimized a vehicle chassis based on natural frequencies and deflection under static load. For optimizing, the input values are calculated using FE techniques while their in house software GENESIS used for optimization of structure. Yuchent Liu & M L Day[12] simplified the model of the vehicle frame by modelling it with beam elements. On this model, FE analyses are performed to study static and dynamic characteristics. Ashutosh[13] used shell elements for the longitudinal members & cross members of the chassis. Beam elements have been used to simulate various attachments over the chassis, like fuel tank mountings, engine mountings, etc. Spring elements have been used for suspension & wheel stiffness of the vehicle Light weight SUV frame development stages and the use of FE techniques for design of frame and tooling for the same are discussed by Altair Engineering[14]. Fischer, et al[15] detailed the procedure of performing elastic MB analysis taking care of vibration effects. This procedure is used to optimize the vehicle frame structure.

Based on the above work, FEA is chosen for the investigation in this project. In the current work, the use of I-channel in place of C-channel that is being currently used in a Heavy truck shown in figure 1 is investigated. The truck specifications are given in [16]. Two different chassis designs are prepared and FE analyses are executed on four different configurations. Table 1 & 2 give the list of materials used and the configurations defined for analyses respectively

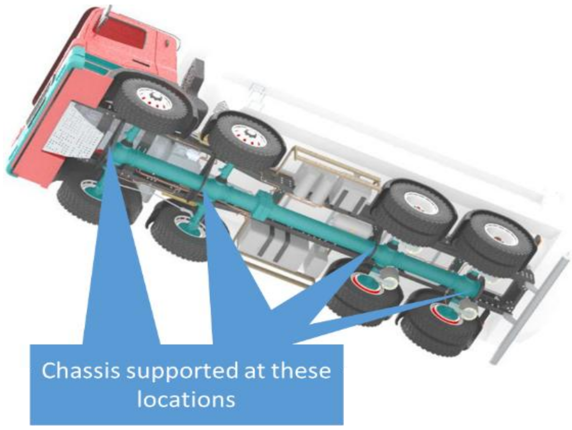


Figure 1: Bottom view of a Heavy Truck showing supports

Table 1: Material Properties used

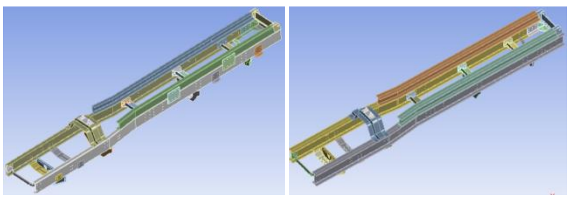
	Structural Steel	E-Glass Epoxy
Young's Modulus (MPa)	2e5	5e8
Poisson's Ratio	0.3	0.33
Density(kg/mm^3)	7.85e-6	2e-7

Table 2: Various Configurations on which FEA is carried out

	Chassis Design	Material
Configuration 1	Existing Design	Structural Steel
Configuration 2	Existing Design	E-Glass Epoxy
Configuration 3	Modified Design	Structural Steel
Configuration 4	Modified Design	E-Glass Epoxy

2. METHODOLOGY

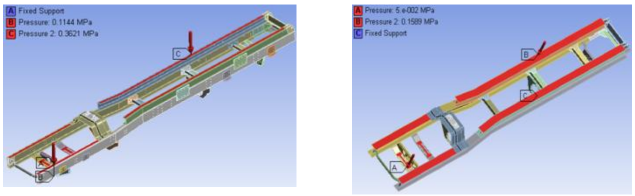
The Chassis existing design and suggested modified design are shown in figure 2 (a) & (b) respectively. The main difference between the two is the use of I channel for the frame. Also in the modified design, engine support plates are modified for strength.



(a) Existing Design (b) Modified Design

Figure 2: Chassis Design

The curb weight is mentioned as 16900kg while the payload is mentioned as 33100 kg [16]. Calculating the surface area on which this loading is applied, the pressure load is computed for two cases i.e. for both curb weight and payload. Figure 3 shows the loading pattern that is used for both static as well as harmonic analyses (frequency excitation in addition to loading shown). The pressure due to payload is applied only to the place where the tipper body is located (i.e. only on to rear axle) while the pressure load due to crub weight is applied on total frame along with engine mounts. The mounts supporting the central power transmission housing are fixed.

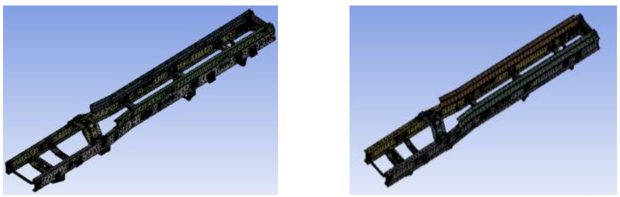


(a) Existing Design (b)Modified Design

Figure 3: Structural Loading

3. MESHED MODEL, LOADING AND BOUNDARY CONDITIONS

Figures 3(a) & (b) show the meshed models of both chassis. Solid 186 and Solid 187 are used for meshing them. Mixed Mode meshing is used. Meshing and analysis are performed in Ansys Workbench. It may be noted that the existing frame model is available in [17]. The modifications are done using Creo software and then imported into Ansys. Meshing of existing solid frame model resulted in 247437 nodes and 119611 elements while that of modified structure resulted in 196595 nodes and 90440 elements



(a) Existing Frame (b) Modified Frame

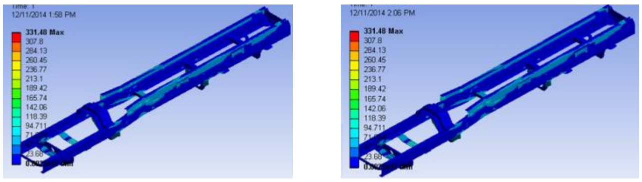
Figure 3: Meshed Models

4. RESULTS & DISCUSSIONS

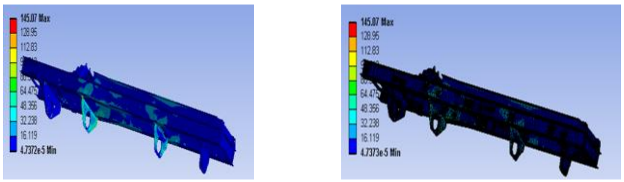
A series of FE analyses are performed on the model with 4 configurations listed in Table 2 are performed. The results are discussed in this section.

4.1 Static Analysis Results

In order to study the effect of vehicle load and cargo on the structural behaviour of the chassis, static analysis is executed. As mentioned earlier, the boundary conditions are applied for all the configs. Figure 4 & shows the results of static analysis and Figures 6 & 7 gives a comparative plot of the results. It may be noted here that the peak stress values are determined taking the degeneracy in to effect.

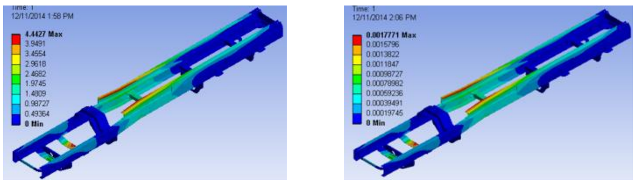


(a) Config 1 (b) Config 2

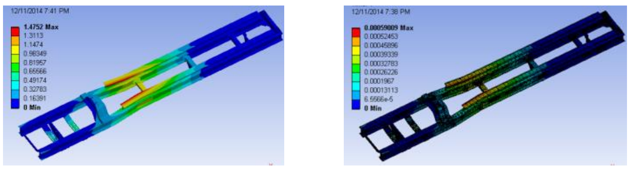


(c) Config 3 (d) Config 4

Figure 4: Equivalent Stress Plots (Units: MPa)



(a) Config-1 (b) Config-2



(c) Config-3 (d) Config 4

Figure 5: Total Deformation plots (Units: mm)

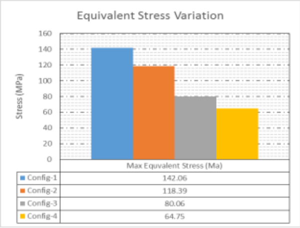


Figure 6: Equivalent Stress Comparative plot

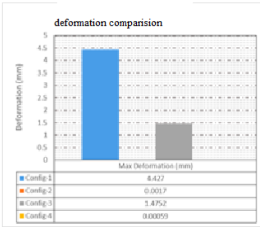


Figure 7: Deformation Comparative plot

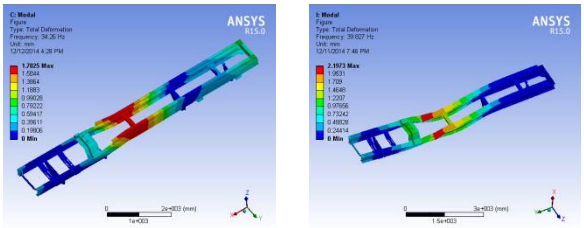
Based on Figure 6, it can be observed that the max stress occurs in config 1 and is 140MPa. Least stress occurs in Config 4. Also comparing both the designs, the max stress is in existing design. From figure 6, it can be understood that max deformation is in Config 1 and is of 4.427mm. Deformation plot for this configuration indicates that the max deformation occurs at two regions: engine support plate and the C section bearing the payload. From figure 6, it can also be observed that for a given material, modified design is having lesser deflection. This indicates that Modified design fares well when compared to existing design.

4.2 Modal Analysis Results

Modal analysis is performed on all configurations to get the natural frequencies of each Configuration. Table 3 shows the extracted natural frequencies. Based on [16] the max engine rpm is 1400. i.e. if an eccentricity exists in the engine, the frequency of excitation will be 23.33Hz. Least natural frequency is observed for Configuration 1. For this the first natural frequency is 34.26 which is quite more than that of the engine excitation frequency. Considering Configuration 2 & 4 (with E-Glass Epoxy material), the natural frequencies are more than 300 times of engine excitation frequencies indicating that they can resist high frequency excitation. Thus it is decided to further investigate only v 1 & 3. Surface area on which this loading is applied, the pressure load is computed for two cases i.e. for both crab weight and payload. Figure 3 shows the loading pattern that is used for both static as well as harmonic analyses (frequency excitation in addition to loading shown). The pressure due to payload is applied only to the place where the tipper body is located (i.e. only on to rear axle) while the pressure load due to curb weight is applied on total frame along with engine mounts. The mounts supporting the central power transmission housing are fixed.

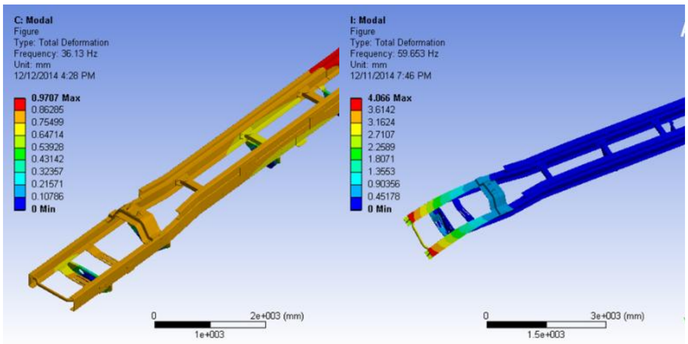
Table 3: Natural frequencies of each Config

#	Config-1	Config-2	Config-3	Config-4
1	34.26	9554.7	39.827	12476
2	36.13	10571	59.653	18686
3	39.993	10640	64.933	20340
4	45.304	11996	65.765	20601
5	52.983	14232	76.626	24003
6	63.681	16597	78.237	24508
7	64.61	17813	82.264	25769
8	76.473	22076	94.864	29716
9	81.694	22316	100.5	31481
10	90.61	23621	103.69	32481



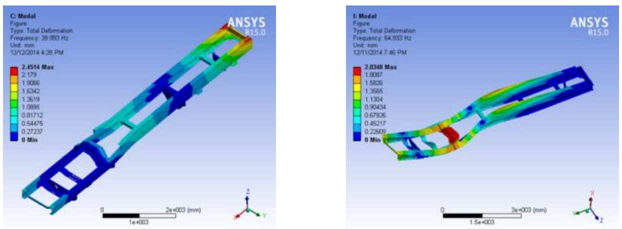
(a) Config-1 (b) Config-3

Figure 8: 1st Mode Shape



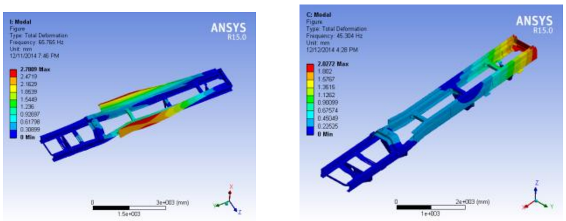
(a) Config-1 (b) Config-3

Figure 9: 2nd Mode shape



(a) Config-1 (b)Config-3

Figure 10: 3rd Mode shape



(a) Config-1 (b)Config-3

Figure 11: 4th Mode shape

4.3 Harmonic Analysis Results

As mentioned in the previous section, harmonic analysis is performed for Configurations -1 & 3 only. The loading is already described in section 3. The variation of normal stress along the chassis axis, deformations in X Y and Z directions with respect to frequency of excitation are studied for both Configurations, during this analysis. Figures 12 to 15 show the variation of these components with frequency. From the normal stress variation graph in Figure 12, it can be observed that there is only one resonant frequency (65.9Hz) for Config-3 (Modified chassis) where a higher stress value is obtained but when Config-1 (Existing chassis) is considered there are multiple such frequencies (36.9Hz, 45.1Hz). Considering the deformation comparison plots in figures 13, 14 and 15 it can be observed that the deformations when Modified chassis (Config-3) is used are quite less and the max value is 5mm at 65.9Hz and in X-Direction only. But when considering existing chassis (Config-1), one should worry for good number of resonant frequencies 36 Hz (45.1mm in X, 9.54mm in Z), 45.1Hz (36.7mm in X, 80.38mm in Z), 64.6 Hz (5.29mm in Y), and 90.6 Hz (44.6mm in Y, 6.52mm in Z). These results indicate that Modified frame with I section beams and modified engine support plates is an optimal solution.

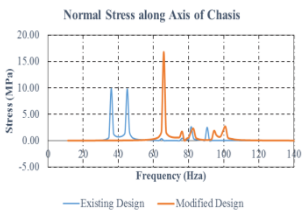


Figure 12: Normal Stress Variation with frequency

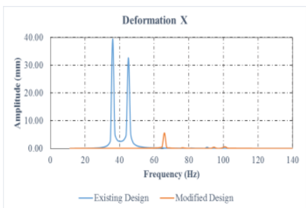


Figure 13: Deformation in X direction w.r.t. frequency

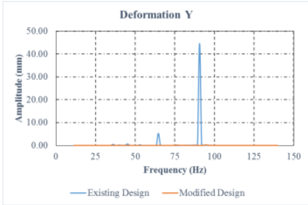


Fig- 14: Deformation in Y direction with respect to frequency

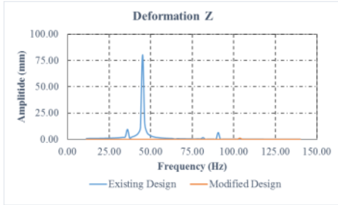


Figure 15: Deformation in Z direction with respect to frequency

5. CONCLUSIONS

In the current work, an attempt has been made to optimize the chassis design for better frequency response. FEA simulations are carried out for this purpose. During the study four different configurations (parameters two different chassis designs and two different materials (steel and E-Glass Epoxy)) are analyzed. Static analysis is carried out to study the response of the chassis under curb weight and payload for all the configurations. The result of the static analysis showed that modified configuration with E-Glass epoxy is a better configuration. Natural frequencies are then computed for each configuration. It is observed that the natural frequencies for both chassis with E-Glass Epoxy material are quite high and Modified chassis has higher natural frequencies when compared for individual materials. Taking the engine excitation frequency into consideration, harmonic analysis is executed for both chasses with steel material. It is observed that when the existing chassis (with C sections) is to be used then there are number of resonant frequencies that the user should worry. But if the modified chassis (with I Sections) are used, there is only one such frequency that the user should worry and if that frequency is made to quickly pass through the vehicle shall be safe. This indicates that the use of I section is more recommended than that of a C Section for heavy truck chassis.

REFERENCES

1. Denish S. Mevawala, Mahesh P. Sharma, Devendra A. Patel, Darshan A. Kapadia (2014), "Stress Analysis of Roll Cage for an All-Terrain Vehicle", IOSR Journal of

Mechanical and Civil Engineering, 2, pp.49-53
2. Goolla Murali, Subramanyam B, Dulam Naveen (2013), "Design Improvement of a Truck Chassis based on Thickness", Altair Technology Conference 2013, India
3. Dr. L. V. Venugopal Rao, Prasanna Priya Chinta (2013), "Dynamic Behavior Of Three-Wheeler Passenger Vehicle Using Finite Element Method, Rigid Body Modelling And Comparison With Intelligent Design Automation", Abhinav- National Monthly Refereed Journal Of Research In Science & Technology , 2(8)
4. Ashif Iqubal,S M Oak, R S Kharatmal (2013), "Analytical Optimization of Chassis Frame for 40ft Dual-Axle Flatbed Trailer Design", IOSR Journal of Mechanical and Civil Engineering , 7(6), pp.74-84
5. Dr S.B.Rane, Harshal Shirodkar, P.Sridhar Reddy (2013), "Finite Element Analysis for Optimization of Fork Lift Chassis", Altair Technology Conference 2013, India
6. Haval Kamal Asker, Thaker Salih Dawood, Arkan Fawzi Said (2012), "Stress Analysis of Standard Truck Chassis during Ramping on Block Using Finite Element Method", ARPN Journal of Engineering and Applied Sciences, 7(6), pp.641-648
7. William Davis, Krysten Carney,Jonathan Leith, Anton Kirschner (2012), "Design & Optimization of a Fourmula SAE Race Car ", Bachlors Thesis, WORCESTER POLYTECHNIC INSTITUTE
8. Ojo Kurdi, Roslan Abdul Rahman (2010), "Finite Element Analysis of Road Roughness Effect on Stress Distribution of Heavy Duty Truck Chassis", International Journal of Technology, 1, pp.57-64
9. Yucheng Liu (2010), "Crashworthiness Analysis of Finite Element Truck Chassis Model Using LS-DYNA®", 11th International LS-DYNA® Users Conference, pp.18-23 - 18-31
10. Ismail Bin Hj. Musa (2009), "Static and Dynamic Analysis of A Ladder Frame Truck Chassis", Master's Thesis, Universiti Teknologi Malaysia
11. J.M. Biradar, B.V. Vijay, Kailash Jat (2008), "Automotive Chassis Sizing Optimisation for Modal and Distortion Criteria", SASTECH, 7(2)
12. Yucheng Liu, M. L. Day (2006), "Development of Simplified Truck Chassis Model for Crashworthiness Analysis", 9th International LS-DYNA Users Conference, Crash/Safety (3), pp.11-43 - 11-56
13. Ashutosh Dubey, Vivek Dwivedi (2003), "Vehicle Chassis Analysis: Load Cases & Boundary Conditions For Stress Analysis", 11th National Conference on Machines and Mechanisms 2003, IIT Delhi
14. Altair Engineering (2003), "Light weight SUV frame", Engineering Report, May 2003
15. P Fischer, W Witteween (2000), "Integrated MB-FE Durability Analysis of Truck Frame Components by Modal Analysis", ADAMS User Meeting 2000, Rome
16. Tatra Phonex T158 – 8P5R46.261 8×8.1R Truck Brochure
17. www.grabcad.com