International Journal for Ignited Minds (IJIMIINDS)

Structural Chassis Design and Finite Element Optimization of a Truck for Road Vibration

Bharath S B[a], Prashantha Kumar S T[b] & Harish H[c]

[a]PG Student, Machine Design, V.V.I.T, Bengaluru, India.
[b]Asst. Prof, Dept. of Mechanical Engineering, V.V.I.T, Bengaluru, India.
[c]Asst. Prof, Dept. of Mechanical Engineering, V.V.I.T, Bengaluru, India.

ABSTRACT

In the present work, two chassis are analyzed to find the optimum chassis structure. Both the models are checked theoretically for certain cross sections and final validation is done through finite element software 'Ansys'. Generally the best designs are identified by lesser stress, lesser displacement and higher natural frequencies. So the analysis is mainly concentrated on these parameters to find the best design. Initially both the models are modeled using three dimensional modeling softwares and are imported to Hypermesh in 'step' file format. After shell meshing with proper quality checks, the meshed models are imported to Ansys in 'inp' file format for further analysis. Totally 5 load cases are analyzed to find the best chassis designs. The 5 load cases are self weight, external weight and three spectrum loads. The results are captured for stress and displacement for each case and are tabulated. The results shows better results with second model compared to the first one as displacement and stresses are less with second model. Also second model shows higher stiffness, rigidity and uniformity for all three spectrum loads. Along with this, modal analysis is carried out and again model 2 shows better dynamic behavior. So the model 2 is better for the given loading considerations.

Keywords *– Chassis, Displacement, Stresses, Spectrum loads, Modal analysis.*

1. INTRODUCTION

Chassis is the main structure of the vehicle which holds the components of vehicle like axel, suspension, cabin etc. it is commonly subjected to cabin weight, inertia force which arises due to road vibration. The stress analysis is made in chassis to determine the critical point in the structure. The critical point is the one where there is high stress concentration. By knowing the maximum magnitude of stress it is possible to predict the life of the chassis and to prevent the failure. The original chassis design taken from horse drawn carriage the most effective shapes are I beam, C beam, Square beam, the can holds the larger weights in the vehicle. These types of sections are used because of simple construction and due to heavy supporting. But it would no longer couple the suspension component under high speed conditions. The body on the chassis was eliminated long before ago, to reduce the cost and weight during production. In order to prevent the draw backs of the beam sections space frames are originated. But due to less production of the space frame it was set to the small and sports cars. Monocoque chassis replaced the space frame because it had a flat surface where large mass is concentrated which is away from the neutral axis. The space frames were replaced light weight honey comb structure. The honey comb structure has very high shear strength and compressive direction are unstable and it easily undergo buckling hence the honey comb structure and is coupled by two layers surrounds the drives has it takes the penetrative load effectively.

2. METHODOLGY

Step 1 : For any problem to solve, it is always better to go with the history of the problem. It gives us idea regarding loads acting on the structure, related problems, improvement methodologies, mathematical formulations etc.
Step 2 : Generally for all the automobile designs, these standards need to be followed to check the safety conditions. These loads are in reality collected from past history of loading on the automobiles. These refer critical loads that act on the structure during its functioning on the real road structure.
Step 3 : In this project, two models are built as per the specifications using Inventor series. But the drafting for the 1st model is done using Solid Edge software. For the second model dimensions are specified in the AutoCAD itself. For better quality mesh on which finite element results depends, Hypermesh is used for quality meshing.

Step 4 : In the actual design process, the design methodology will be to define the problem initially either preliminary or comprehensive type. Later Designer has to select different mechanisms or structures which satisfy the requirements. Among all these designs, the best design needs to be selected for optimum conditions like weight, cost and maintenance etc.

Step 5 : This step is very essential in the present competeting market. The product should work satisfactory along with lesser cost and maintenance. This is possible by setting different optimization parameters without compromising on the structural safety conditions.

Step 6 : Spectrum is very important analysis for the automobile structures. The spectrum is not a definite load, but a probabilistic load. 100% safety can't be decided for the spectrum loads. Maximum of 99.7% safety can be ensured by checking for 3 sigma loads. By considering 6 sigma loads, further safety can be ensured but not 100%. The spectrum loads are defined from vibration measuring equipment.

Step 7 : Finally the documentation is very important and user should understand the procedures adopted for the given problem and this should be literature or useful information for the future generation.

3. SECTIONAL CALCULATION

- Support length= 2250mm (Center of wheel to wheel)
- Total Load, P = $\frac{75000}{2}$ = 37500 N
- Total load considered, P = 37500 N
- If converted to udl load, w = $\frac{37500}{2250}$ =16.7N/mm^2.
- Yield stress of the structural Steel = 550MPa.
- Factor of Safety = 5 [CMTI- for Variable load calculations]
- Design Stress σ_d = $\frac{500}{5}$ =110MPa.
- Maximum Moment M = $\frac{(w*l^2)}{8} = \frac{(16.7*2250^2)}{8}$ =10567968mm^4.
- Section Modulus, $\frac{(b*h^2)}{6} = \frac{M}{\sigma_d}$
- Assuming section of the beam as square

  $h^3 = 6 * \frac{M}{\sigma_d}$

  $= 6 * \frac{10567968}{110}$ = 576434mm^3.
- H= ~83mm.
- Moment of inertia required I= $\frac{h^4}{12}$ = 3954860mm^4.
- Considered section is 104mmX104mm with 6.5mm thickness.

4. FINITE ELEMENT MODEL DEVELOPMENT

4.1 Model-1

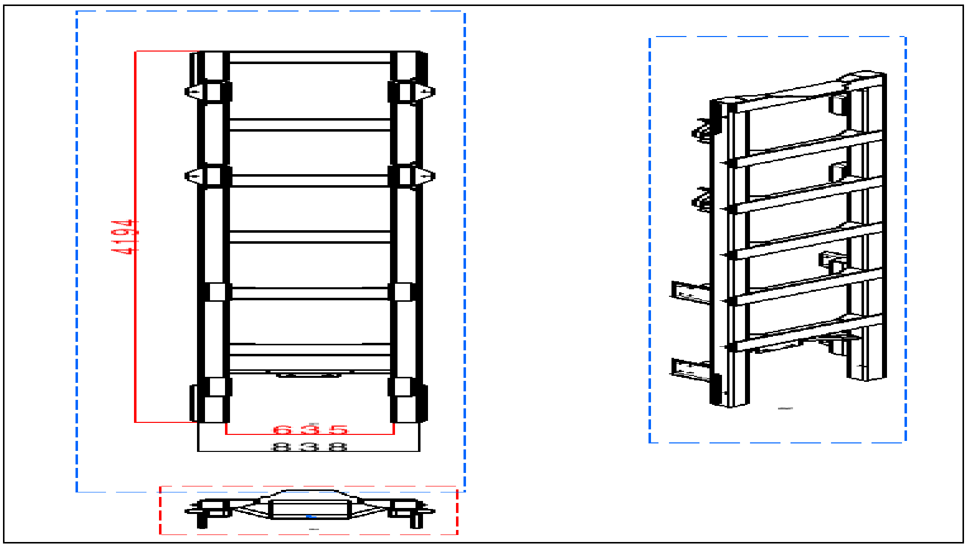


Fig 1: Major Dimensions of the Chassis

The figure 1 shows major dimensions of the developed model. The individual dimensions are represented in the following pictures. Front, top and side views are represented. Also an isometric view is also represented to understand the structure.

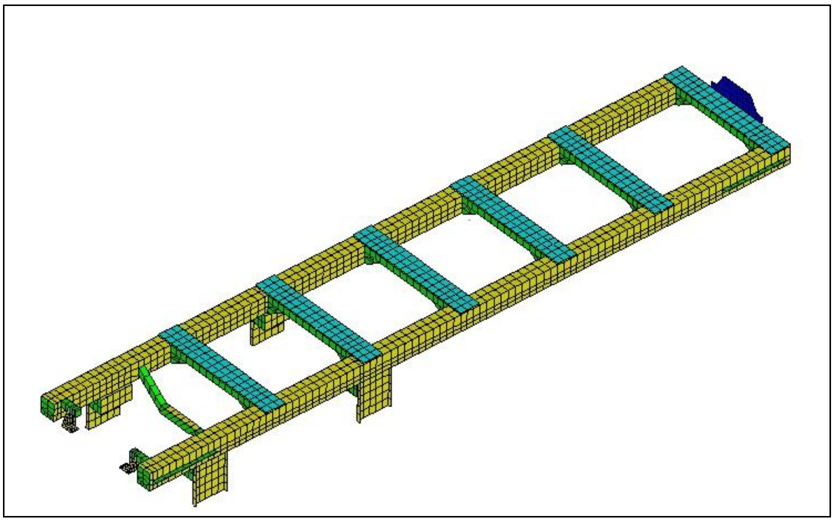


Fig 2: Mesh plot Model-1

4.2 Second Model

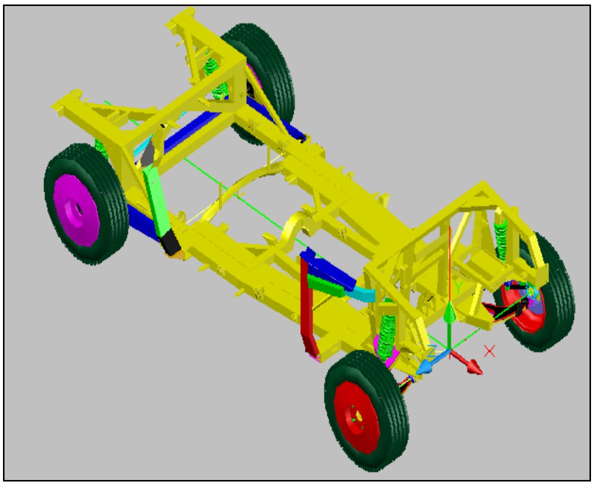


Fig 3: Simulation view of Chassis

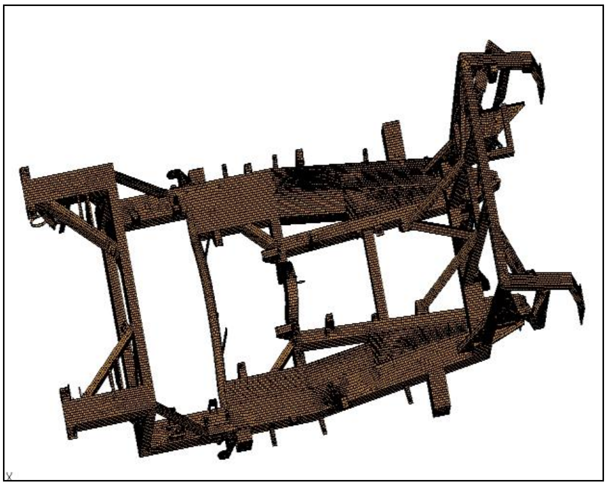


Fig 4: Meshed Model

The figures 3 show geometry of the second model. This model is designed for compact and all the members are accommodated in limited space. Compared to the first design, heavy ribbing is done in the second design in compact space. The chassis has around 2300mm length with a height of 600mm. The length of this chassis is less than the first configuration.

Material: Low Alloy Steel A 710(class 3)

- Young's modulus= 207 GPa
- Poison's ratio= 0.3
- Density= 7800 kg/ m^3
- Yield Stress= 552MPa
- Tensile Strength= 620MPa

5. RESULT FOR ANALYSIS

5.1 Analysis of Model 1 for Self Weight and External Load

Figure 5 shows displacement in the structure as 0.154mm (0.000154meters) and observed at the end portion. This can be attributed to cantilever effect of the longer unsupported length at the end. All the basic principles of mechanics of Solids holds in the software related calculations. The deflection is a critical parameter which represents the stiffness of the structure. Higher the displacement, the stability of the system will be problem.

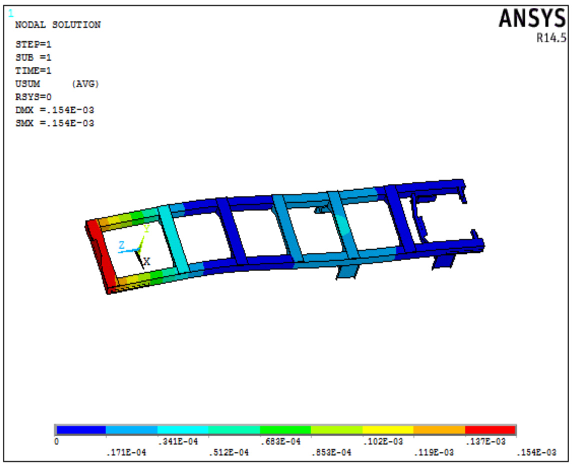


Fig 5: Displacement for Self Weight.

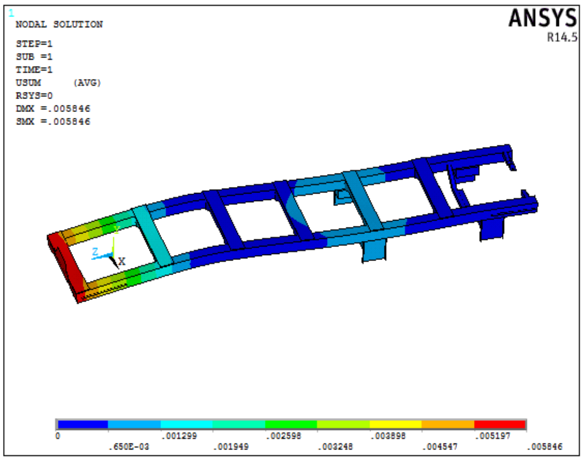


Fig 6: Displacement for External Load.

The figure 6 shows the displacement induced in the structure of 5.846mm or 0.005846m and is noticed at the unsupported end region.

5.2 Model Frequencies

Set No.	Frequency (Hz)
1	31.717
2	46.682
3	96.957
4	98.996
5	116.48

Table 1: Modal Frequencies Plot

The table 1 shows modal frequencies of the problem. The first frequency value is 31.717Hz which is more than the operational frequency of 11.8 Hz for the problem. So the chassis is safe for the dynamic modal conditions.

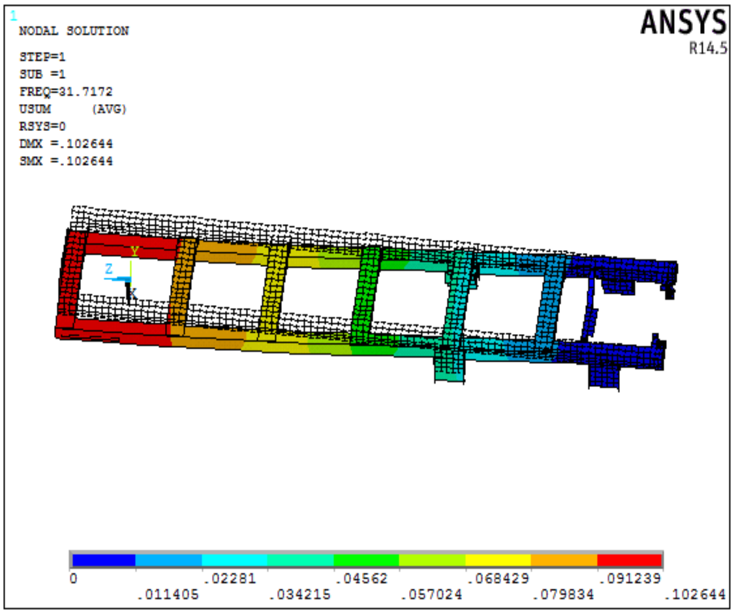


Fig 7: Mode Shape with un-deformed model.

5.3 Analysis of Model 2 for Self Weight and External Load

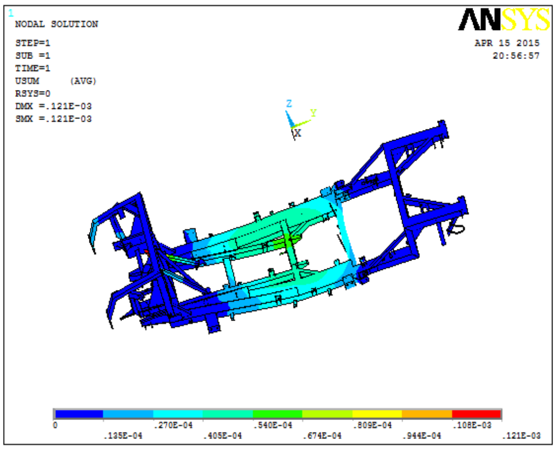


Fig 8: Displacement for Self Weight.

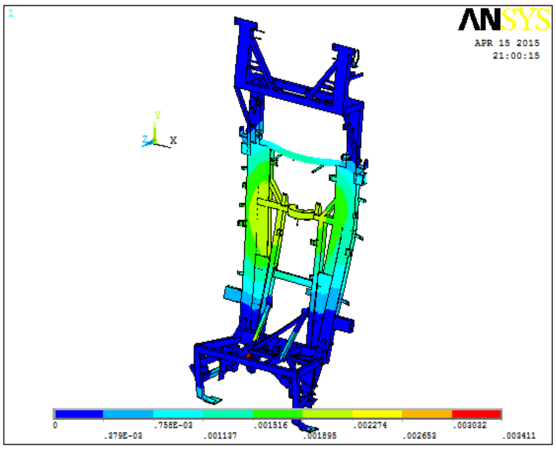


Fig 9: Displacement for External Load.

The figure 8 shows deformation in the problem as 0.121mm or 0.000121m. The deformation is observed at the unsupported overhung tie rod. The displacements are less at the support regions or wheel regions. Other regions, the deformation is shown in the figure. Deformation represents the stiffness of the structure. Lesser the deformation, higher the stiffness of the structure along with higher dynamic natural frequencies which are critical for dynamic behavior of the chassis structures. Deformation can be calculated as force to the stiffness of the structure.

The figure 9 shows developed displacement for the external load as 3.4mm or 0.00034m. This deformation is less than the allowable deflection for the problem. Allowable deflections are calculated based on IS standards. As per the specification for a rigid structure, for every 350mm, one mm is allowed. For the present configuration of 2.3meters, allowable deflection is around 6mm. So structure is safe for the given loading conditions.

5.4 Model Frequencies

Set. No	Frequency(Hz)
1	32.837
2	64.2
3	78.176
4	91.523
5	95.887

Table 2: Modal Frequencies for Second System.

Further modal analysis has been carried out to find the dynamic nature of the second design. The fundamental frequency value is around 32.837Hz which is also higher than the operational frequency of 11.8 Hz. So dynamically the design is safe. Similarly the mode shape at the bottom shows lateral nature of vibration at the center. The mode shapes gives idea for weaker regions along with unconnected parts.

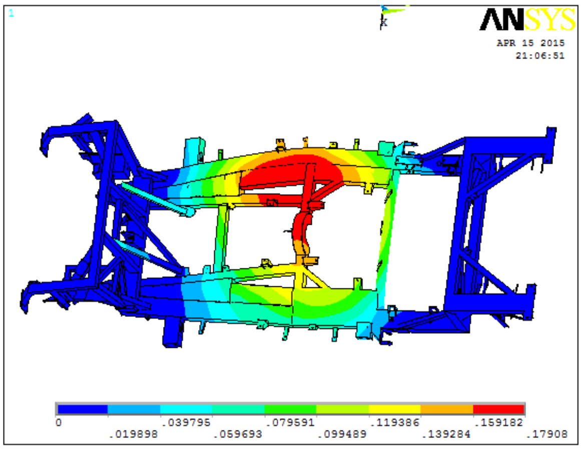
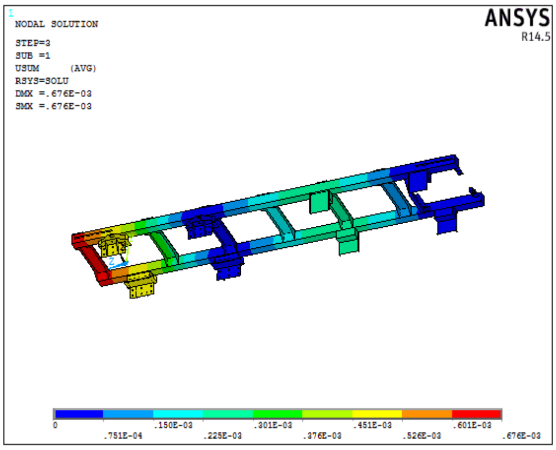


Fig 10: Mode Shape.

5.5 Spectrum Analysis for First Model



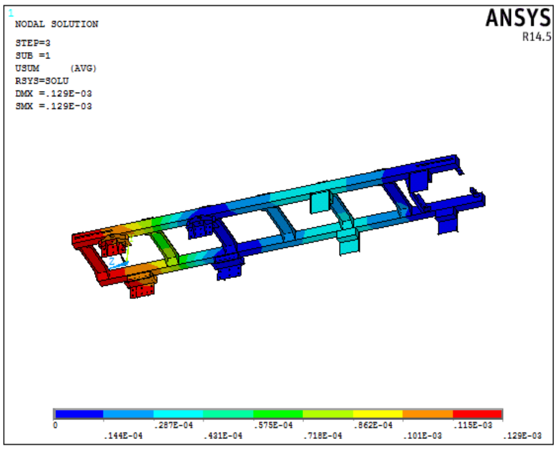
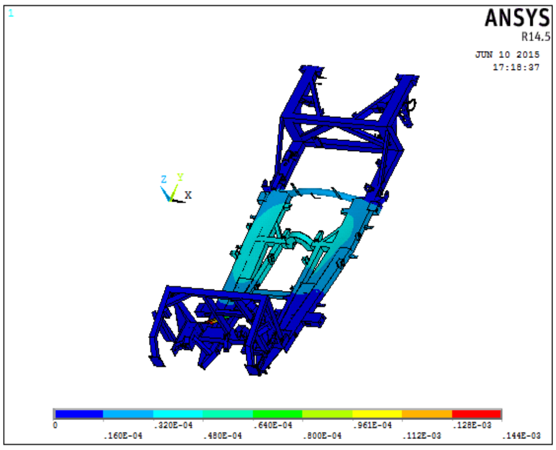


Fig 11: Displacement Plot (Vertical Spectrum) Fig 12: Displacement Plot (Longitudinal Spectrum)

The figure 11 show deformation due to vertical spectrum corresponding 1 sigma loads. The maximum deformation is around 0.676mm which can be multiplied by 3 times to get the 3 sigma results. Even this 3 sigma results (3*0.676=2.028mm) is less then allowable deflection of the problem (6mm). So for vertical spectrum the displacement of the structure is safe.

The figure 12 shows deformation results for 1 sigma results. Maximum deformation is around 0.129mm or 0.000129m. But this deformation is less then allowable deformation for the problem, even when compared with 3 sigma results.

5.6 Spectrum Analysis for Second Model



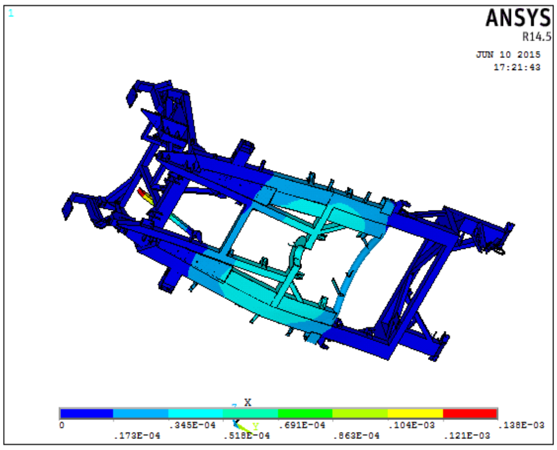


Fig 13: Displacement Plot (Vertical Spectrum) Fig 14: Displacement Plot (Longitudinal Spectrum)

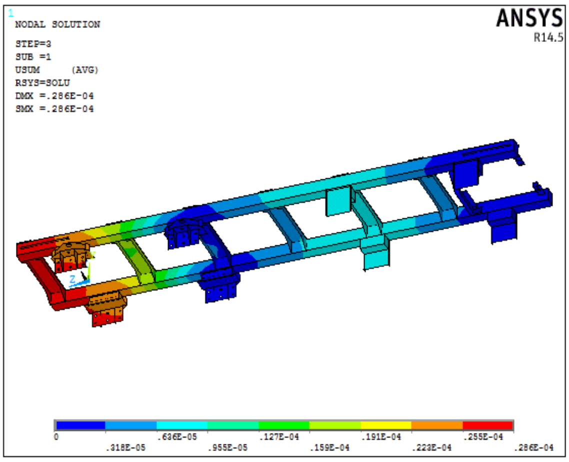


Fig 15: Displacement Plot (Lateral Spectrum)

The figure 13 shows displacement plot for the vertical spectrum analysis and the maximum displacement is found to be 0.144mm for 1 sigma results. To find 3 sigma results, the value should be multiplied with 3. But this value is very small compared to the allowable deflection of the problem. So structure is safe for the vertical spectrum.
The figure 14 shows displacement plot for the longitudinal spectrum analysis. Maximum displacement is around 0.138mm for 1 sigma results. To find 3 sigma results, the value should be multiplied with 3. Since the value is small compared to the allowable deflection the structure is safe for the longitudinal spectrum.

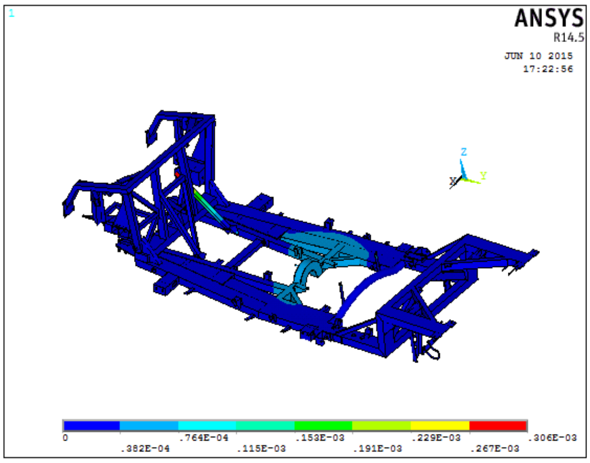


Fig 16: Displacement Plot (Lateral Spectrum)

The figure 16 shows displacement pattern for the lateral spectrum. The spectrum specified in the problem is applied at the constraint regions and solution is obtained for combination of modal and spectrum. Combined effect is obtained and shown in the figure. Maximum deformation is around 0.3mm or 0.0003m as shown in the status bar.

5.7 Discussion

Analysis has been carried out for 2 different chassis structures for 5 load cases. The results are compared as follows.

Description		Displacement(mm)	Von-Mises Stress (Mpa)
Modal 1	Self Weight	0.154	14.7
	External Load(75KN)	5.5	559
	Vertical Spectrum	0.676	82
	Longitudinal Spectrum	0.129	35.8
	Lateral Spectrum	0.0286	8.7

Table 3: Results for Chassis Modal-Type 1.

Description		Displacement(mm)	von-Mises Stress (Mpa)
Modal 2	Self Weight	0.121	8
	External Load(75KN)	3.4	260
	Vertical Spectrum	0.144	26.1
	Longitudinal Spectrum	0.138	28.3
	Lateral Spectrum	0.3	26.4

Table 4: Results for Chassis Modal-Type 2.

Description	Modal Frequency(Hz)
Modal 1	31.717
Modal 2	32.837

Table 5: Results Comparison for Modal Analysis.

6. CONCLUSION

From the above FE Analysis we can make the fallowing conclusions.

- The results show high stresses and deflection development in the first type chassis as compared to second type chassis. This represents lesser rigidity or lesser stiffness of the structure. So Second chassis is the best for the given loading conditions.
- Modal frequencies are also less compared to the second model. For any structural higher natural frequency is desired to obtain better dynamic performance. Hence, second model is the best for the given conditions.
- All the results shows better strength of the second model compared to the first model. So second design is suggested.

REFERENCES

1. CicekKaraoglu, N. SefaKuralay "*Stress analysis of a truck chassis with riveted Joints*", Elsevier Science B.V Finite Elements in Analysis and Design 38 (2002) 1115–1130.
2. RoslanAbdRahman, MohdNasirTamin, OjoKurdi "*Stress analysis of heavy duty Truck chassis as a preliminary data for its fatigue life prediction using FEM*" Jurnal Mekanikal Dec-2008, No. 26, 76 – 85.
3. N.K.Ingole, D.V. Bhope "*Stress analysis of tractor trailer chassis for self weight Reduction*", IJEST, ISSN: 0975-5462 Vol. 3 No. 9 Sept-2011.
4. MohdAzizi Muhammad Nora, HelmiRashida, Wan MohdFaizul Wan Mahyuddin "*Stress Analysis of a Low Loader Chassis*" Elsevier Ltd. Sci Verse Science Direct Procedia Engineering 41 (2012) 995 – 1001.
5. JadavChetan S., PanchalKhushbu C., Patel Fajalhusen "*A Review of the Fatigue Analysis of an Automobile Frames*" IJACR, Vol-2, No-4, Issue-6, Dec-2012.
6. S.A. Oke, M.Sc.1, A.O. Johnson, B.Sc.2, T.A.O. and A.O. Adeyefa, M.Sc. 4 Application of Neurofuzzy in the Development of Road Bump Designs the Pacific Journal of Science and Technology Volume 8. Number1. May 2007 (spring).
7. Chandrupatla, T. R, Belegundu A. D, "*Introduction to Finite Elements in Engineering*", Prentice-Hall of India Private Limited, New Delhi 1999.
8. G.H. Farrahi a, A. Khalaj b, "*Achievements in Materials and Manufacturing Engineering Estimation of fatigue damage caused by actual roads and maneuverson proving ground*" Volume 14, feb-2006.
9. Arthur Tang-Vice President, business development, virtual proving ground-A CAE Tool for Automotive Durability, Ride and Handling and NVH Applications.Aug-2004.
10. A.V. Pestereva, L.A. Bergmanb, C.A. Tanc, B. Yangd, "*Assessing tire forces due to roadway unevenness by the pothole dynamic amplification factor method*", Journal of Sound and Vibration 279 (2005) page 817–841.