RESEARCH ARTICLE OPEN ACCESS

Static Analysis, Design Modification and Modal Analysis of Structural Chassis Frame

Madhu Ps[1] and Venugopal T R [2]
[1]M Tech student Mechanical Department, NMAM Institute of Technology, Nitte-574110. India
[2] Assistant Professor Mechanical Engineering, NMAM Institute of Technology, Nitte-574110. India

ABSTRACT
The chassis frame is an important part in a truck and it carries the whole load acting on the truck as well as different parts of the automobile. So it must be strong enough to resist the shock, twist, vibration and other stresses. Maximum stress and maximum deflection are important criteria for design of the chassis. The objective of present is to determine the maximum stress, maximum deflection and to recognize critical regions under static loading condition. Static structural analysis of the chassis frame is carried out by FEA Method. The structural chassis frame is modeled using PRO-E wildfire 4.0 software. The Pre-processing has done with HYPERMESH software; then the problem has been solved through RADIOSS and the post processing was done by HYPERVIEW. The results obtained like maximum shear stress, Von-mises stress and maximum deflections are used for improving design modification. Modal analysis of the chassis frame done using ANSYS WORKBENCH. Through modal analysis, natural frequencies and corresponding vibration mode shapes of the structure are obtained.
Keywords - Static Analysis; Finite Element Model; ladder chassis frame; allowable stress; resonance

I. INTRODUCTION

The chassis frame plays a vital role in the truck. All most all components weight is acting on the chassis frame, thus chassis subjected to static, dynamic and cyclic loading condition on the road, therefore chassis must be rigid enough to resist this loads. Static stress analysis is an important to point out critical (highest stress) regions in the frame. These critical regions may cause fatigue failures [1].

In this study, ladder type chassis frame is analysed. The Chassis consists of side members attached with a series of cross members to complete the ladder like structure, thus its name [5]. They were designed for functionality and provided little torsional stiffness. The FEM is a common tool for stress analysis. FEM with required boundary conditions was used to determine critical regions in the chassis frame.

Static structural analysis is performed to identify critical regions and based on the results obtained design modification has been done.

The modal analysis of the chassis frame is carried out to determine the natural frequency and mode shapes of the system. The rigidity of the system was analysed and their resonance could be avoided.

II. FE ANALYSIS OF CHASSIS FRAME

A vehicle chassis is designed to carry the payload, weights of other structural components mounted on it [4]. For carrying out the FE analysis of Ladder chassis frame has been modeled according to the dimension using PRO-E Wildfire.4 Cad software.

Fig.1 Chassis frame CAD model

2.1 Meshing of geometry:

HYPERMESH software (made for meshing) from Altair is used mesh the solid model. Cad model which is in IGES format is imported to HYPERMESH. To mesh the geometry use the surface first and then solid. While meshing some of the quality parameters as to be maintained, those are aspect ratio, skew, jacobian, minimum element size, warpage.

Element type used: Hexahedron; Number of elements generated: 126266; Number of nodes generated: 260857.

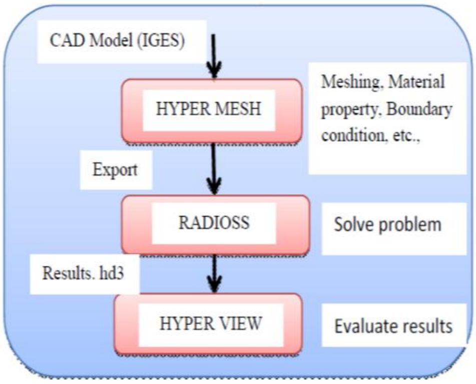

Fig.2 Flow chart of FEA process

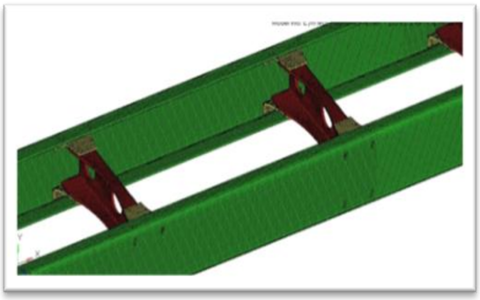

Fig.3 Meshed chassis frame

2.2 Material property:

TABLLE 1
MATERIAL PROPERTY

Material used	Carbon steel, AISI 1080 (tempered @ 205 C, oil quenched)
Young's modulus (E)	215 GPa
Yield strength	800 MPa
Poisons ratio	0.285
Density	7800kg/m^3
Composition	C=0.74-0.88% Fe=98.13-98.66% Mn=0.6-0.9% P= 0.0-0.04% S=0.0-0.05%

2.3 Boundary conditions:

Static analysis is used to determine the displacements, stresses, strains, and forces in structures or components caused by steady loads that do not induce significant inertia and damping effects [2]. Boundary condition involves application of load and defining constraints in the model. In our study Model is fixed in all degrees of freedom near the supports; Rigids are applied at side rail, connecting plates, cross bars. Rigids applied at the side rail which is shown in figure 4.

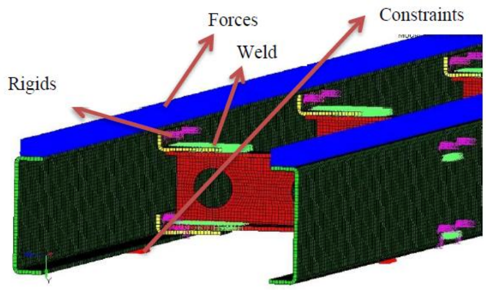

Fig. 4 Forces and constraints in the frame

The total load (123851.25 N) is applied over the top of the chassis frame, which contains 14576 nodes. So load on each node is given by

Force on each node present at top surface of side rail= (total load)/ (number of nodes)

Force on each node = 123851.25/14576

Force per node=8.49N

III. FEA RESULTS

After applying boundary condition, the file is imported to the solver. RADIOSS is used as solver here for the analysis. Finally the post-process is done by using HYPERVIEW software.

The location of maximum Von Mises stress and maximum shear stress are just near the supports and at the joining portion of connecting plates and side rail. The maximum Von mises stress is about 181.69 MPa and the maximum deflection is about 5.65 mm.

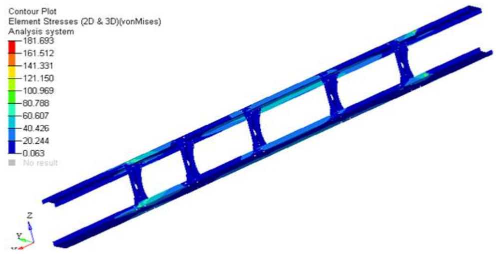

Fig.5 Maximum von-mises stress

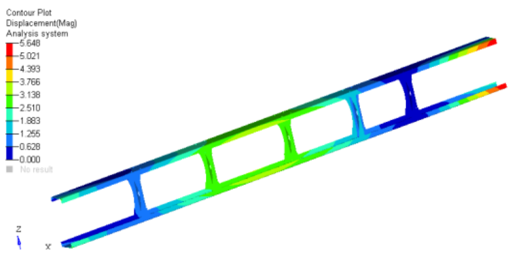

Fig.6 Maximum displacement

IV. DESIGN MODIFICATION

The Design modification (Re-design) is the process of achieving some desired set of specification which minimises the critical factors of model. While modifying the model, the designer must have the knowledge about model, behaviour of the model under given loading condition, and also some experience in similar projects and also previous results of existing model.

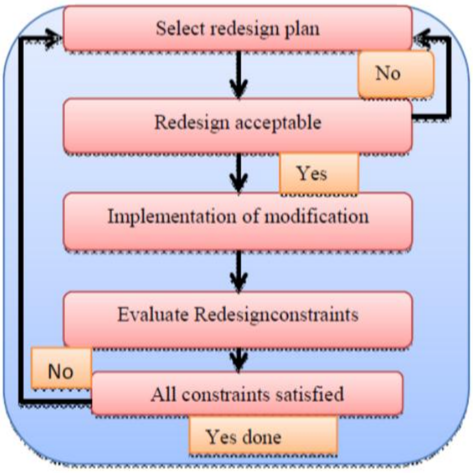

Fig.7 Flow chart for design modification process

In this study, the allowable stress is less than the yield strength, so stresses are not a critical issue, but deflection of chassis frame is about 5.64 mm. Deflection of chassis frame has different standard values for different vehicle frames and also has different values for different type of analysis. Still it is possible to minimize the deflection of chassis frame without increase the stresses.

4.1 Iteration 1:

As mentioned earlier, to minimize deflection we have to study geometry and assembly of the model. Ensure while redesign stresses must within the yield strength or nearer to existing model.

By looking at the geometry of the chassis frame shown in fig.1 can be modified by shifting 3rd cross member away from 2nd, so we are moving the 3rd cross member about 65 mm(away from 2nd cross member). So repeat the previous analysis process to achieve results.

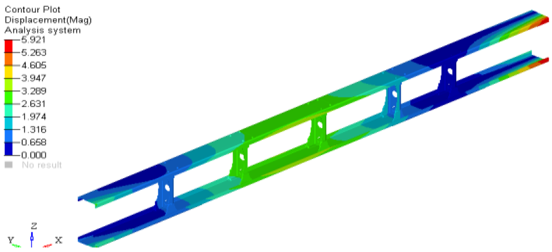

Fig.8 iteration 1 displacements

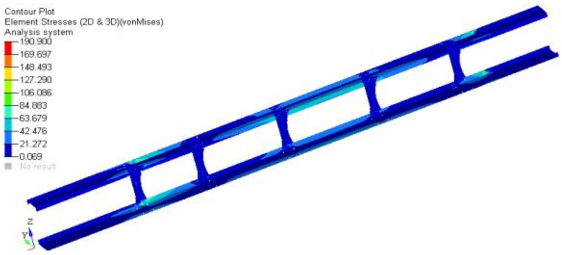

Fig.9 iteration 1 von-mises stress

4.2 Iteration 2:

In the second iteration we are going to add a cross member for the 1st iteration model.

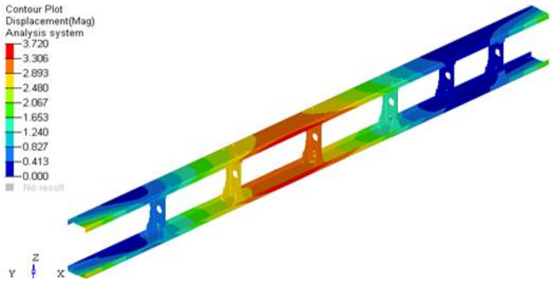

Fig.10 iteration 2 displacements

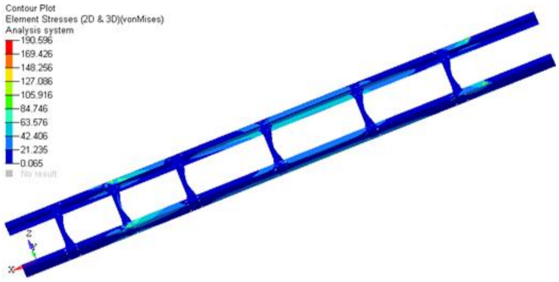

Fig.11 iteration 2 von-mises stress

4.3 Iteration 3:

In the 3rd iteration we are going to add 6th cross member to the existing model.

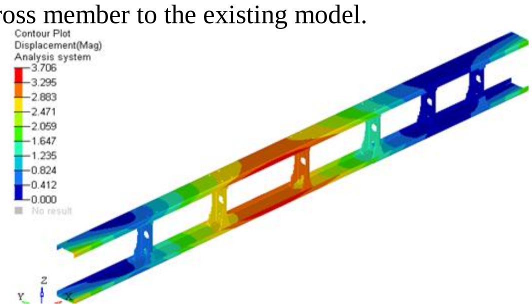

Fig.12 iteration 3rd displacements

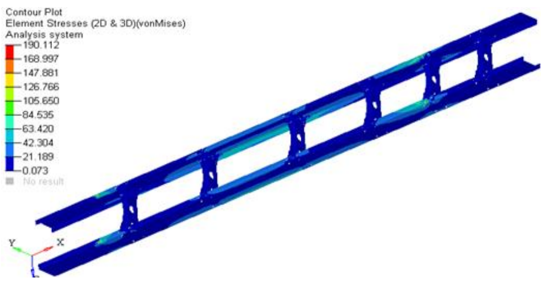

Fig.13 iteration 3rd von-mises stress

TABLE II
DESIGN MODIFICATION RESULTS

Iteration	Von-mises stress in MPA	Displacement in mm
1	190.9	5.921
2	190.596	3.720
3	190.112	3.706

V. MODAL ANALYSIS

The modal analysis is the most basic and important part of analysis of dynamic character [3]. This modern method used find the natural frequency and mode shapes of the structures. The rigidity could be analyzed and the resonance vibration could be avoided. The main characteristics of each mode of the structure can be figured out through the modal analysis, and the actual vibration response under this frequency range can be predicted. The results from modal analysis can be used as reference value for other dynamic analysis like random analysis, harmonic analysis, etc.

In the paper, 3-D FE analysis is carried out on the modal analysis of chassis frame. And the FME software ANSYS WORKBENCH was used to simulate the modal analysis. The results of natural frequencies and mode shape were obtained.

The first 14th-order natural frequencies of a structural frame are extracted. The first, second third and fourth modes are shown in below figures.

TABLE III
NATURAL FREQUENCIES

Mode	Frequency(Hz)
01	3.7377
02	9.5166
03	29.896
04	36.22
05	40.948
06	42.923
07	58.683
08	60.392
09	62.354
10	72.43
11	73.982
12	78.002
13	87.675
14	95.788

Each natural frequency of respective mode shapes is listed in the table no 3. If any one of the natural frequency matches with excitation frequency the frame doesn't satisfies the dynamic characteristics. Out of 14 mode shapes the seventh mode shape with natural frequency 58.683Hz is critical and it will produce maximum stresses under dynamic condition.

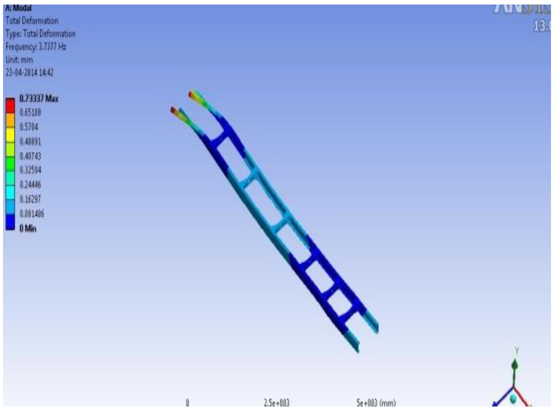

Fig.14 mode shape1

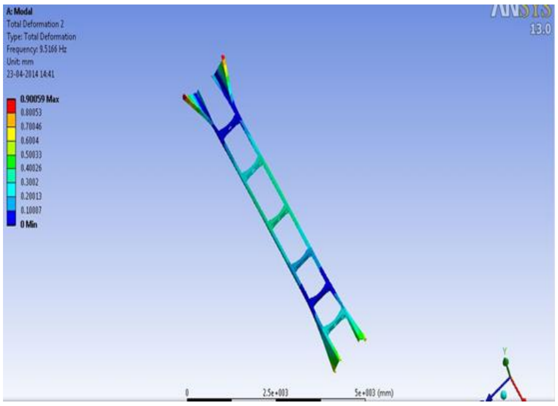

Fig.15 mode shape 2

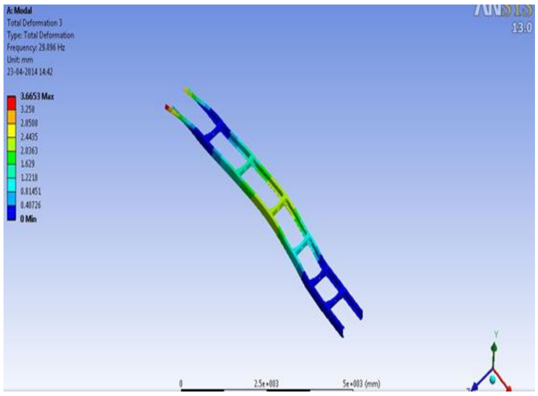

Fig.16 mode shape 3

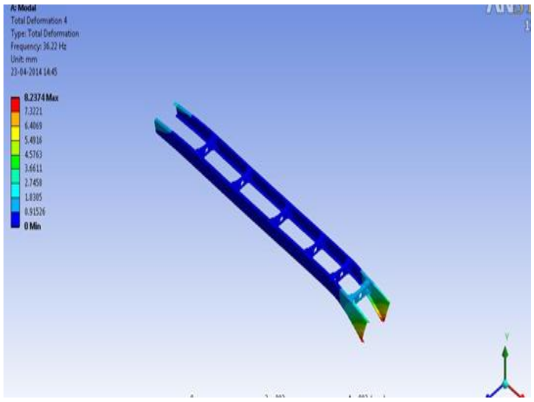

Fig.17 mode shape 4

VI. CONCLUSION

In this paper static analysis, modification of chassis frame based on results and modal analysis of structure is achieved. First modelling of structure done by PRO-E wildfire-4 and static analysis is carried out using following software HYPERMESH, RADIOSS, HYPERVIEW. Results indicate von-mises stress below yield strength of the material, which satisfies the design and deflection is about 5.648mm. Modification of design is carried out to reduce deflection of structure. Deflection of the modified structure is 3.706mm. Therefore chassis frame deflection has been reduced from5.921mm to 3.706mm by modification. Finally the natural frequency and corresponding vibration modes of modified structure are found by modal analysis. Out of all modes shapes seventh mode shape with natural frequency 58.683 Hz is critical.

VII. Acknowledgements

The research work was supported by Merritt Innovative Solution India Pvt Ltd Bangalore (Title: “Static Analysis, Design Modification and Modal Analysis of Structural Chassis Frame”) is gratefully acknowledged.

References

Journal Papers:

[1] Hemant B.Patil, Sharad D.Kachave, Eknath R.Deore “Stress Analysis of Automotive Chassis with Various Thicknesses”, *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)* Vol. 6 Issue 1, 2013,pp 44-49.

[2] Tulasiram Nasikai. Charyulu T.N,"Design and Analysis of Vehicle Chassis Frame", *Indian Streams Research Journal,* West Godavari Aug 2012, Volume 2, Issue. 7,

[3] Shan Xue, Guohua Cao, Qiongying Lv, Hongji Xu" Modal Analysis and Optimization of Novel External Rotor Radar Stabilized Platform", International Conference on Electronic & Mechanical Engineering and Information Technology, 2011, pp 2729- 2733

[4]J. M.Biradar, B.V.Vijay, Kailash Jat,.“Automotive Chassis Sizing Optimisation for Modal and Distortion Criteria” SASTech - Technical Journal, 2008, Volume 7, Issue 2,

[5] Patel Vijaykumar V, Prof. R. I. Patel, “Structural Analysis of Automotive Chassis Frame and Design Modification for Weight Reduction” International Journal of Engineering Research & Technology (IJERT) 2012, Vol. 1 Issue 3