Optimization Of Heavy Vehicle Chassis Using Finite Element Analysis

[1]Hetul J Dalal, [2]prof.Parita Sheth,

[1]Student of Master of Technology (CAD/CAM), [2]Assitant Professor
[1]Department of Mechanical Engineering,
[1]Indus University, Ahmedabad, India

Abstract- Chassis is a most important part of vehicle. The chassis work as the backbone of a vehicle which carries the maximum load for all operating conditions. Work of the chassis is to gives a structural platform that can connect the front and rear suspension. Also, it must be hard enough to bear the shock, twist, vibration and other stresses caused because of sudden braking, acceleration, shocking road condition, centrifugal force while cornering and forces induced by its components. So, strength and stiffness are two main parameter for the design of the chassis.

The current study has analyzed the Different literatures. After a analysis of dissimilar research paper studies carry so for it has been found that there is the scope of optimizing dissimilar point like load, stress-strain values and deformation etc. by varying cross sections and materials of chassis body. This paper describes the design, Structural analysis & optimization of the heavy vehicle chassis with constraints of peak stress, strain and deflection of chassis under maximum load.

In the present work, the measurement of the TATA 2518 chassis is used for the structural evaluation of the heavy vehicle chassis with three different alloys subjected to the same position of the steel chassis. The three different material used for the chassis are selected as grey cast iron, AISI4130 alloy steel and ASTM A710 STEEL based on their easy availability and known characteristic. A three dimensional solid Modeled in the CAE software CREO and analyzed in ANSYS 14.0. The numerical results are validated with analytical calculation considering the stress distribution, deformation. Also two standard optimization techniques (Boxing & Reinforcement) are used to optimize the chassis frame.

Keywords— Heavy truck chassis frame, CREO, ANSYS, FEM, Assembly weight, stress, deformation.

I. INTRODUCTION

The major challenge in today's ground vehicle industry is to overcome the increasing demands for higher performance, lower weight, and longer life of components, all this at a reasonable cost and in a short period of time. The chassis of trucks is the backbone of vehicles and integrates the main truck component systems such as the axles, suspension, power train, cab and trailer. Since the truck chassis is a major component in the vehicle system, it is often identified for refinement. There are many industrial sectors using this truck for their transportations such as the logistics, agricultures, factories and other industries [9].

The chassis frame consists of side members attached with a series of cross members Stress analysis using Finite Element Method (FEM) used to locate the critical point which has the highest stress. This critical point is one of the factors that may cause the fatigue failure. The magnitude of the stress used to predict the life span of the truck chassis [10].

I. BASIC CONCEPT OF FEA

FEA has now become an integral part of Computer Aided Engineering (CAE) and is being extensively used in the analysis of many tedious real time engineering problems. The field of FEA has developed a lot and this analysis greatly depends on rigorous mathematical foundation. Many powerful software tools and packages are available promoting its widespread use in industries. Finite Element Method (FEM) is a computational technique which is employed for achieving approximate solutions for boundary value problems in engineering. Concisely stated, a boundary value problem is a mathematical problem that requires the satisfaction of a differential

equation everywhere within a known domain of independent variables and also the specific conditions on the boundary of the domain via one or more dependent variables. A plain description of the FE method is that it simply involves slicing a structure into several elemental pieces, then reconnecting these elements at nodes; these nodes can be conceptualized as pins or drops of glue that hold the elements together. FEA consists of three main steps, namely, pre-processing, solution, and post processing. Pre-processing (i.e. model definition) includes definition of the geometric domain of the problem, the element type(s) to be employed, the material properties of the elements, the geometric properties of the elements (length, area, etc.), the element connectivity (mesh of the model), the physical constraints (boundary conditions), and the loadings. The solution comprises the governing algebraic equations in matrix form, computes the unknown values of the primary field variable(s), and gathers the findings. The computed results are then employed to determine the additional and the derived variables such as reaction forces, element stresses, and heat flow with the help of a back substitution step. In post processing, analysis and evaluation of the results are carried out stepwise.

II. OPTIMIZATION

1) FE Analysis for Structure Steel ST 37:-

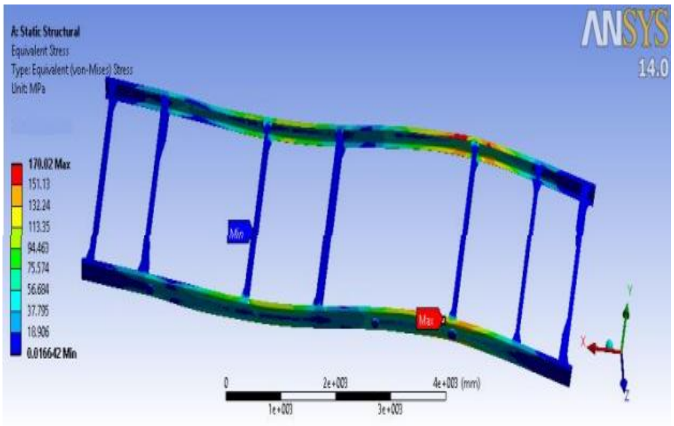

Fig. 1. Stress distributions for Structure Steel ST 37

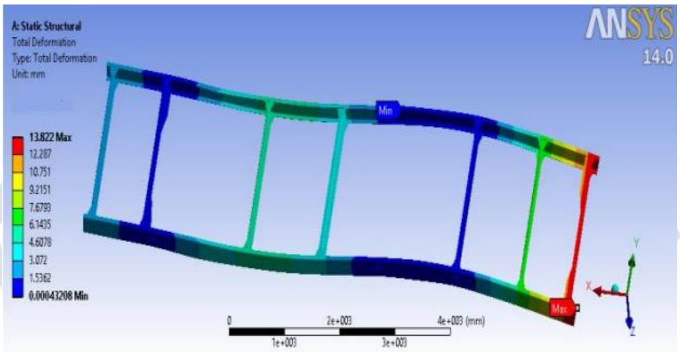

Fig.2. Displacement pattern for Structure Steel ST 37

Different practices are available for chassis modification. Here suitable changes are made in the design using two standard methods and the analysis is done to observe the reduction in the stress levels [1].

2) BOXING OPTIMIZATION TECHNIQUE

Boxing is the addition of 3 mm or thicker plate by welding it into the opening of C channel to form a box section as shown in fig.33. Since box section has good resistance to both bending and torsion [1].

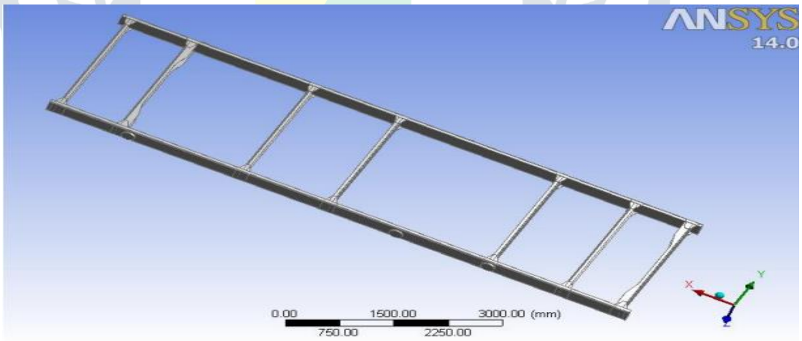

Fig.3. Optimized Geometrical Model

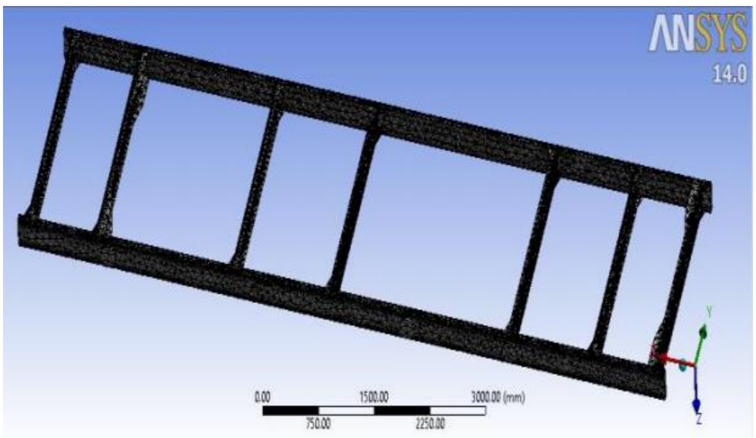

Fig.4. Mesh Model

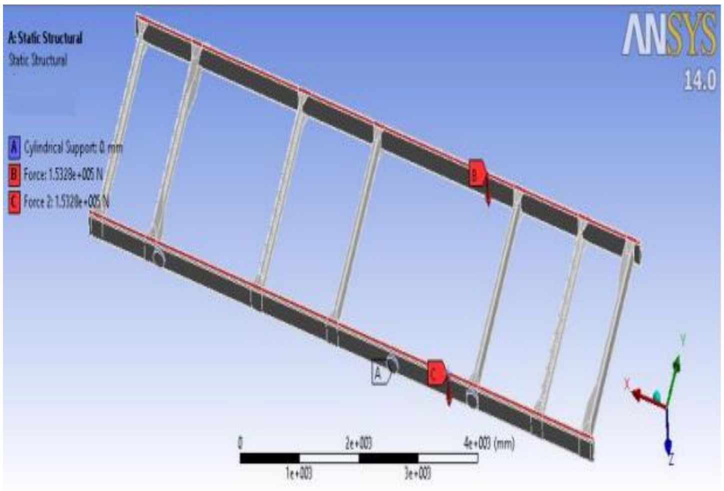

Fig.5. Boundary Conditions

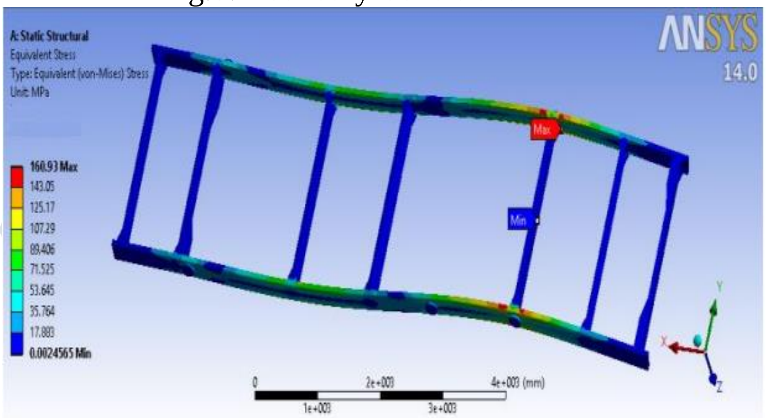

Fig. 6. Stress distributions

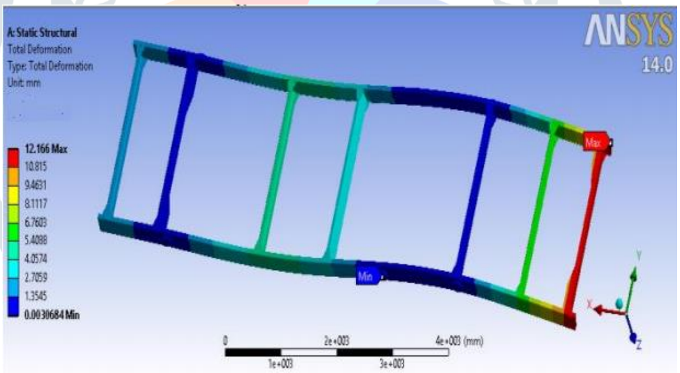

Fig. 7. Displacement pattern

3) REINFORCEMENT OPTIMIZATION TECHNIQUE

Reinforcement is the practice of providing a cover plate either internal or external on the side members at the highly stressed regions. Here reinforcement of 3 mm thickness and 180 mm length is provided on the side members where the stress is maximum as shown in fig.[1]

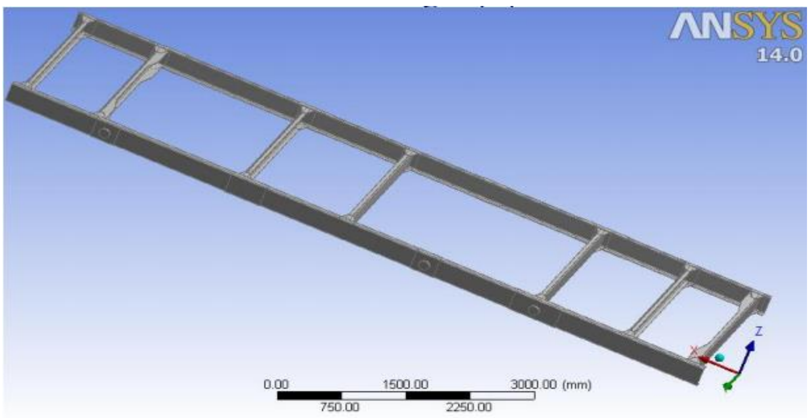

Fig. 8. Optimized Geometrical Model

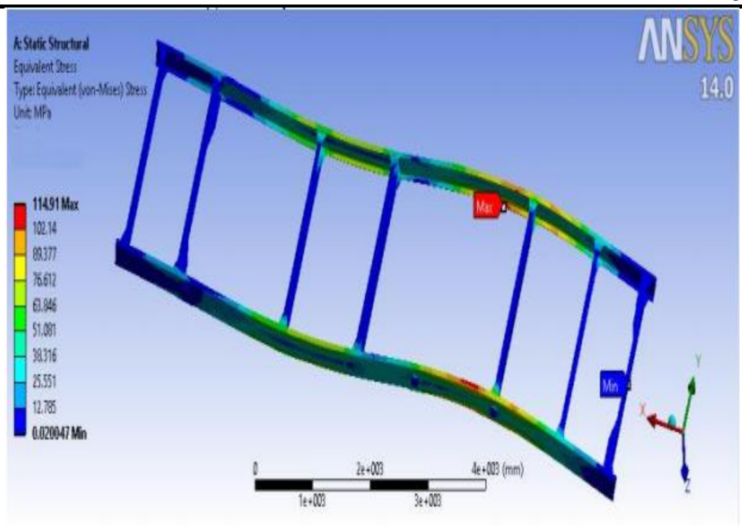

Fig. 9. Stress distributions

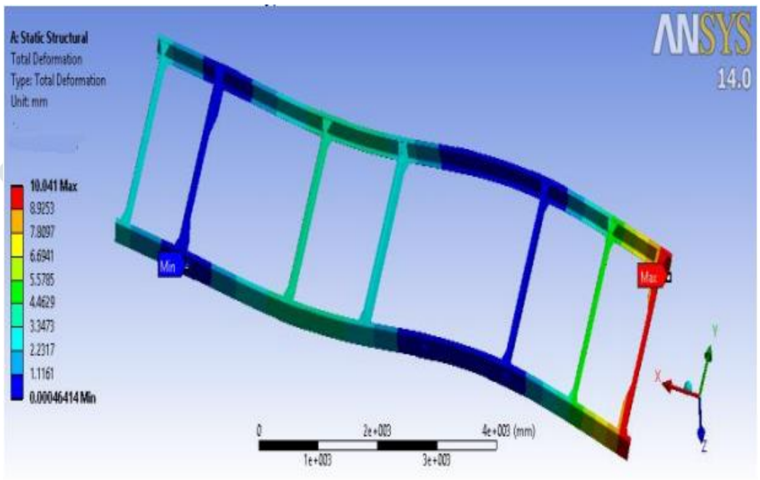

Fig. 10. Displacement pattern

III. CONCLUSION

The stress distribution and deformation pattern for the existing C-channel cross section are shown in Figs. 1 and 2 for the structural steel ST37 material. The stress distribution and deformation pattern for modified case 1and 2 are shown in Figs. 6, 7and Figs. 9, 10 respectively, also numerical values tabulated as below.

Table.1. Stress And Deformation Values Using Optimization Techniques

Parameters	Existing "C" Channel Section	Modified Case-1	Modified Case-2
Assembly Weight(kg)	838.89	973.54	889.39
Stress (N/mm2)	170.02	160.93	114.91
Deformation (mm)	13.822	12.166	10.041

The tabulated result shows that the values of stress and deformation reduced 5% (approx.), 12% respectively in case- 1, and 32% (approx.), 27% respectively in case-2, so the method of reinforcement is found to be most effective. Finally the optimization using different standard optimization techniques has been successfully accomplished. The work not only provides an analysis of the chassis but also presents the scope for its modification in actual. Also the optimized chassis is capable to carry the loads beyond the previous payload.

IV. ACKNOWLEDGMENT

I wish to express my sincere thanks to project guide Prof. Parita Sheth and all Faculties of Mechanical Department for his cordial support in every step and constant encouragement for this paper, help and guidance given by him time to time shall carry me to publish thispaper.

REFERENCES

1. Manpreet Singh Bajwa, Sinthiya Pundir and Amit Joshi “Static Load Analysis of Tata Super Ace Chassis and Stress Optimisation Using Standard Techniques” ISSN: 2320-2092, International Journal of Mechanical and Production Engineering, Vol. 1, Issue- 3pp. 41-45, Sept-2013.

2. R. L. Patel, K.R. Gawande and D.B. Morabiya “Design and analysis of chassis frame of TATA 2516TC” ISSN: 2321-9653, International Journal for Research in Applied Science and Engineering Technology(IJRASET)- Vol. 2 Issue IIIpp.115-119, March 2014.

3. Kamlesh Y. Patil and Eknath R. Deore “Stress Analysis of Ladder Chassis with Various Cross Sections” ISSN: 2278-1684, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) Vol. 12, Issue 4, pp. 111-116, August- 2015.

4. Patel Vijay kumar V, Prof. R. I. Patel “Structural Analysis of Automotive Chassis Frame and Design Modification for Weight Reduction” ISSN: 2278-0181, International Journal of Engineering Research & Technology (IJERT) Vol. 1 Issue 3pp.1-6, May-2012.

5. Singarajan Nagammal Vijayan and Sathiavelu Sendhilkumar “Structural Analysis of Automotive Chassis Considering Cross-Section and Material” International Journal of Mechanical Engineering and Automation Vol. 2, Issue-8, pp. 370-376, August-2015.

6. Chintada.Vinnod babu , Chiranjeeva Rao.Seela and Vykunta Rao.Matta “Structural Analysis of Eicher 11.10 Chassis Frame” ISSN: 2231- 5381, International Journal of Engineering Trends and Technology (IJETT) Vol. 22 Number 7pp. 315-318, April 2015.

7. Tushar M. Patel, Dr. M. G. Bhatt and Harshad K. Patel “Analysis and validation of Eicher 11.10 chassis frame using Ansys” ISSN: 2278- 6856, International Journal of Emerging Trends & Technology in Computer Science (IJETTCS) Vol. 2, Issue 2pp. 85-88, March - April 2013.

8. Sandip Godse and Prof. D.A.Patel “Static Load Analysis of Tata Ace Ex Chassis and Stress Optimization Using Reinforcement Technique” ISSN: 2231-5381, International Journal of Engineering Trends and Technology (IJETT) Vol. 4 Issue. 7, pp. 3037-3039, July 2013.

9. Mr. Rahul L. Patel, Mr. Divyesh B. Morabiya and Mr. Anil N. Rathour “Weight optimization of chassis frame using Pro-Mechanica” ISSN: 2348-8360, SSRG International Journal of Mechanical Engineering (SSRG-IJME) - vol. 1 Issue 8, pp.4-9, December 2014.

10. Goolla Murali, Subramanyam B and Dulam Naveen “Design Improvement of a Truck Chassis based on Thickness” Altair Technology Conference, Sigma Consultancy Service India pp. 1-6, 2013.