Analysis of Low Loader Chassis, Its Prototype Testing and Design Modification for Weight Reduction

Mr. Dhammadip A. Kamble
Sinhgad College of Engineering, Pune
Email:dhammadipkamble_svcp@sinhgad.edu

Prof. Dr.A.P. Tadmalle
Sinhgad College of Engineering, Pune
Email:aptadmalle.scoe@sinhgad.edu

Abstract— **The work presents the analysis of a low loader chassis for the transportation of heavy loads, especially with respect to strength and weight reduction. The existing model of the chassis was analyzed by using FEM and it then modified. Four iterations of the chassis were prepared. This work focuses on the FEM analysis in the design of the chassis where highest stresses were located. A design solution for a chassis, a schematic image of a I type central beam, a design solution for maximum load carrying capacity , geometrical models of optimized frame with different boundary conditions, stress distributions in a non-extended and extended chassis frame model, a section of the rear part of the frame with stiffener plates to with stand and increase load caring capacity, stress distribution in the front part of the chassis frame, single and double supporting rib, side wall supporting plate and stress distribution after the use of various plates and one supporting rib, stress distribution after the employment of plates and two supporting ribs, stress distribution in the alternative finally optimized design, Fabrication details and experimental validation with scale down model are presented in the work.**

Keywords—Low loader chassis, Stress distribution, FEM, Experimental validation.

I. Introduction

Transportation plays a major role in the economy of modern industrialized world and developing countries. The chassis structure must safely good support the weight of the vehicle components and transmit loads that result from different components of accelerations in a vehicle without failure. There are many factors that need to be considering when designing a chassis, including material selection, strength, stiffness and weight. The major focus in the heavy vehicle manufacturing industries is design of vehicles with more pay load. Using higher strength material than the traditional ones are possible with corresponding increase in pay load capacity. The truck chassis which is the backbone of vehicles that integrates the main vehicle component systems such as the axles, suspension, power train, cab and trailer etc. is one of the possible candidates for substantial weight reduction. Along with strength, bending and torsional stiffness are important characteristics. So, strength and stiffness are two important criteria for the design of the chassis of vehicle. During the conceptual design stage, when changes to the design are so easiest to implement and have lower impact on overall project cost, the weight and structural characteristics are mostly unknown since exact vehicle information is unavailable initially. In general, there are two approaches to simulate truck chassis using FEA methods: one is stress analysis to predict the weak points and the other is fatigue analysis to predict life of the frame. Recently, in quite a few of published papers, there are amount technical papers and some other sources which have been showed gradually upward trend. This overview selectively and briefly discusses some of the recent and current developments of the stress analysis of low loader chassis. A number of analytical, numerical and experimental techniques are considered for the stress analysis of the heavy duty truck frames.

II. Literature Survey

[1] This paper aims to model, simulate and perform the stress analysis of an actual low loader structure consisting of I-beams design application of 35 tons trailer designed in-house by Sumai Engineering Sdn. Bhd, (SESB). The material of structure is Low Alloy Steel A710 C (Class 3) with 552 MPa of yield strength and 620 MPa of tensile strength.

[2] The paper presents the process of design of a frame for a semi low loader for the transportation of oversize loads, especially wheeled and tracked machines. After considering different solutions of a frame, a central beam was chosen and an initial design was developed.

[3] In this study, stress analysis of a backhoe chassis was performed by using FEA. Construction industry is undoubtedly the backbone and propelling force behind our progress. In response to booming construction industry, utilization of earth moving equipment has increased considerably leading to high rate of failure.

[4] In this paper the army vehicle chassis has been designed and optimized to reduce the stresses and deflections. Optimization was also done to increase the fundamental natural frequency.

[5] The paper reviews the most important research works, technical journal and conferences papers that have been published in the last thirteen year period (2002-2014). The paper focused on stress analysis of the truck chassis using finite element package ANSYS.

[6] This paper describes Structural analysis & optimization of vehicle chassis with constraints of maximum shear stress and deflection of chassis under maximum load. In the present work, we have taken higher strength as the main issue, so the dimensions of an existing vehicle chassis of a TATA LP 912 Diesel BS4 bus is taken for analysis with materials namely Steel alloy (Austenitic) subjected to the same load.

[7] The paper focused on stress analysis of the heavy truck chassis using four finite element packages namely; ABAQUS, ANSYS, NASTRAN and HYPERVIEW.

[8] This paper presents, stress analysis of a ladder type low loader truck chassis structure consisting of C-beams design for application of 7.5 tons was performed by using FEM.

[9] In this present work static and dynamic load characteristics are analyzed using FE models from this work. It is found that identifying location of high stress area, analyzing vibration, natural frequency and mode shape by using finite element method.

[10] In these projects, we have calculated the von mises stress and shear stress for the chassis frame and the finite element analysis has been done for the validation on the chassis frame model of jeep.

III. Design Of Low Loader Chassis

To perform the FEA of the Existing low loader Chassis, continuum (Chassis model) is discretized into finite number of elements through meshing process and then boundary conditions are applied to the system. Fixed supports are applied to chassis where it comes in contact with the front hinge.

The Pay load capacity of 35 ton (UDL) is converted into pressure then applied on whole chassis surface

Total Pay load capacity considered 35 ton = 35000kg

35000*9.81 =343350 N

Surface area of low loader chassis where load acts = 8803846.153 mm^2

$$\text{Pressure} = \frac{\text{Force}}{\text{Area}} = \frac{343350}{8803846.153} \ldots\ldots\ldots\ldots\ldots\ldots (1)$$

$$\text{Pressure} = 0.039 \frac{N}{mm2}$$

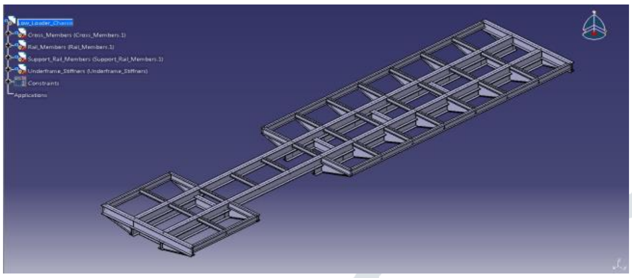


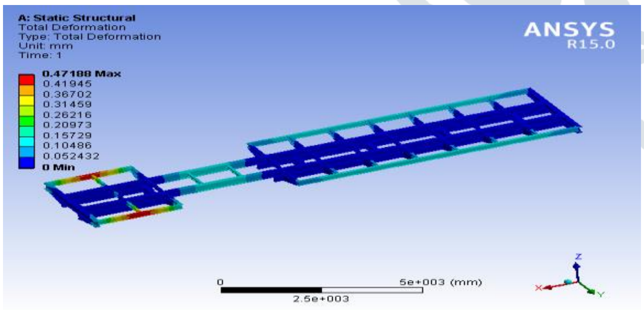
Fig 1. CAD Model

A. Properties of Material

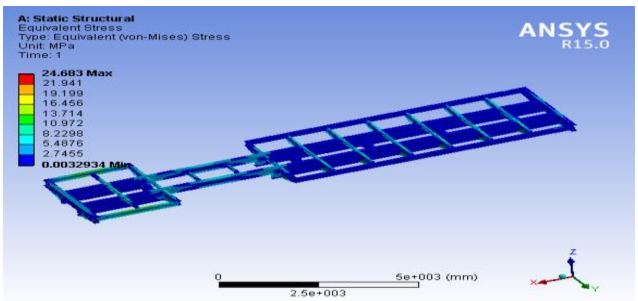
Table 1: Properties of Material

Properties	Values
Tensile Strength	440 Mpa
Tensile Strength (Yeild)	370 Mpa
Elongation At Break	15%
Reduction of Area	40%
Modulus of Elasticity	205 Gpa
Poisson's Ratio	0.29
Machinability	70%

IV. Finite Element Analysis



(a)



(b)

Fig. 2: (a) Deformation plot for Existing Chassis
(b) Stress plot for Existing Chassis

Table 2: Result for Existing chassis

Load(N)	Deformation(mm)	Maximum Stress(Mpa)
343350	0.471	24.68

A. Iteration 1

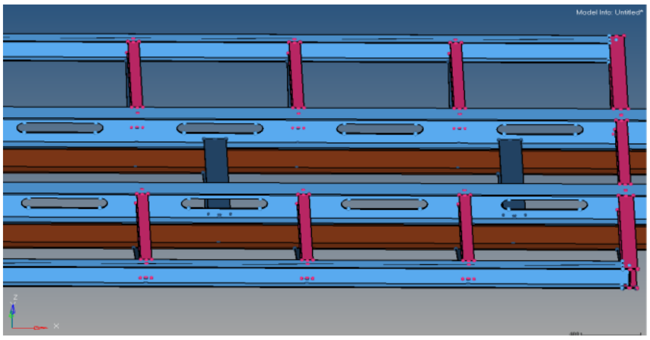


Fig. 3: Material Removed From Area (Iteration 1)

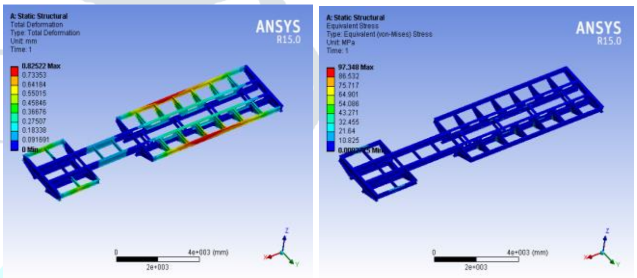


Fig.4: (a) Deformation plot (Iteration 1)
(b) Stress plot (Iteration 1)

B. Iteration 2

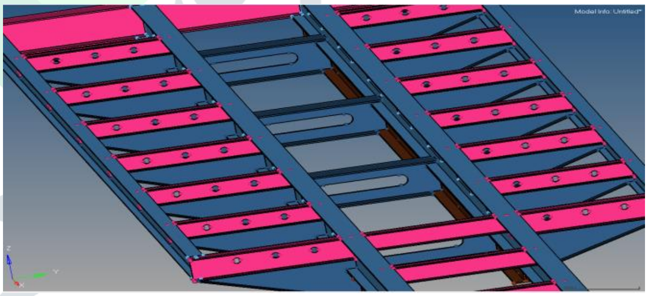


Fig. 5: Material Removed From Area (Iteration 2)

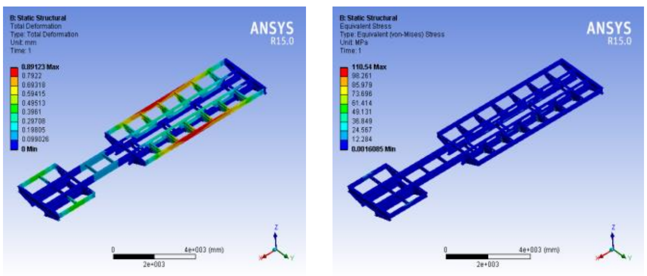


Fig. 6: (a) Deformation plot (Iteration 2)
(b) Stress plot (Iteration 2)

C. Iteration 3

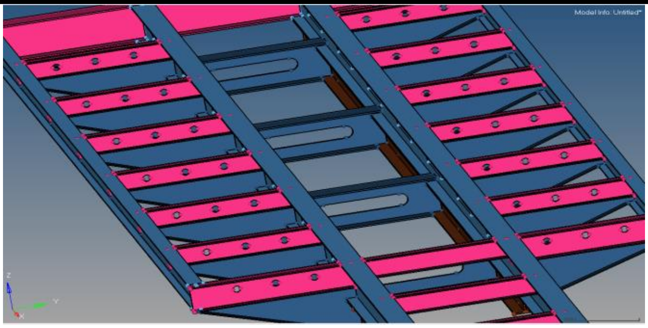


Fig. 7: Material Removed From Area (Iteration 3)

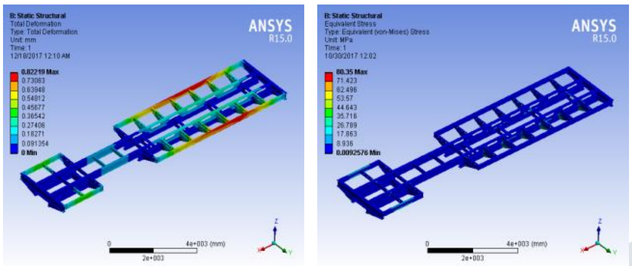


Fig. 8: (a) Deformation plot (Iteration 3)
(b) Stress plot (Iteration 3)

D. Iteration 4

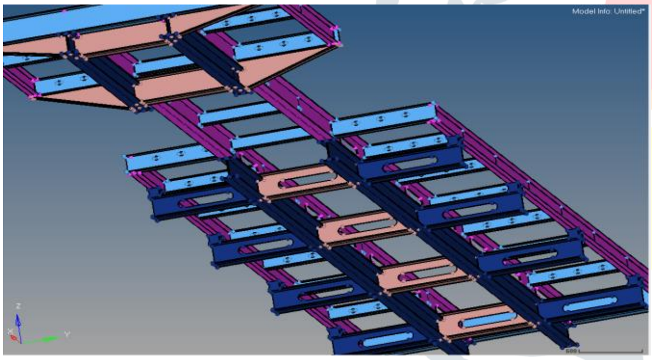


Fig. 9: Material Removed From Area (Iteration 4)

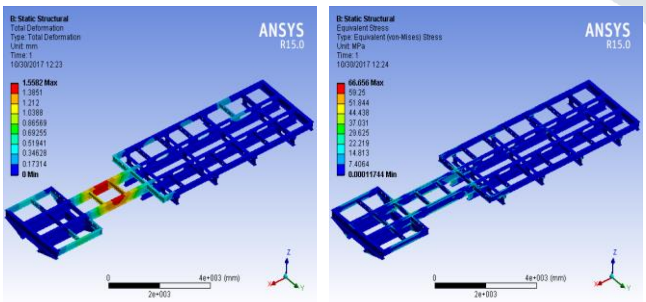


Fig. 10: (a) Deformation plot (Iteration 4)
(b) Stress plot (Iteration 4)

Table 3: Result For Optimization

Case	Total Deformation	Maximum Stress
Existing Model	0.4768 mm	24.68Mpa
Optimized model Iteration 1	0.8252 mm	97.348 Mpa
Optimized model Iteration 2	0.8912 mm	110.54Mpa
Optimized model Iteration 3	0.8221 mm	80.35Mpa
Optimized model Iteration 4	1.558 mm	66.55Mpa

The linear static analysis of optimized low loader chassis (Iteration 3) given us the maximum stress of 80.35 Mpa and maximum deflection of 0.8221 mm. by observing stress and deflection graph, it can be concluded that though stresses and deformation are slightly high compared to existing chassis but the values are well below the critical value of 370 Mpa yielding.

E. Weight Reduction

Fig. shows the weight of chassis prototype. This weight calculation is done in modeling software i.e. CAD. Density of the material is important in the calculation of mass of the chassis. While calculating the mass of chassis prototype we have to select the density of a material, from that data it will directly give the mass of chassis. Mass of scaled down model is 4261.69 Kg.

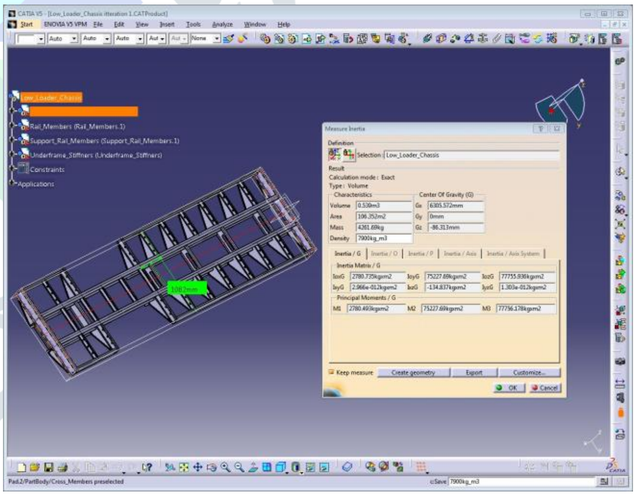


Fig. 11: Mass calculation for scaled down model

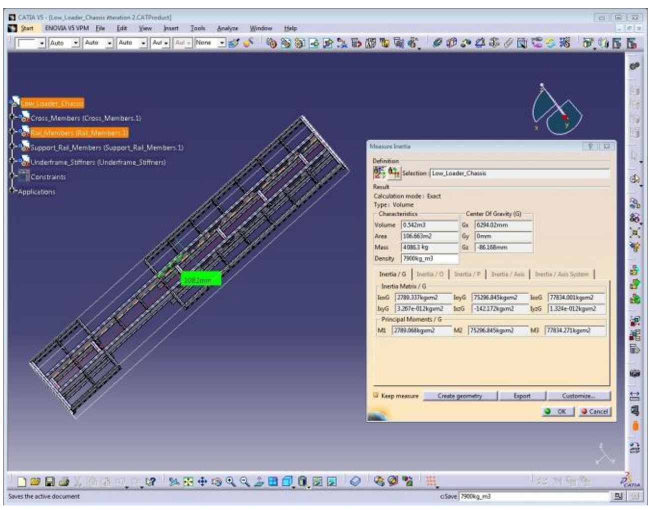


Fig. 12: Mass calculation for optimized model

V. Fabrication Of Chassis Prototype

Low loader chassis has been fabricated as per the drawing generated by scaling it down to 10 times ten original one. Post all members fabrication and optimization all the element has been assemble and welded together by MIG welding machine by maintaining the geometrical dimensions as given in the drawing. The final model is then finish by grinder and removes unnecessary welding slugs and shafts corners

(a) (b)

Fig. 13: Fabrication of chassis Prototype

VI. Experimental Set Up

To reduce the complexity and expenditures for fabrication of huge optimized Chassis. The whole model has been scaled down for ten times with respect to dimensions and loadings and prototype has been created. Then scale downed model is tested for uniformly distributed load case of actual 343.350 KN scaled down to 34.335 KN with proper fixtures mounted and tested in UTM as shown in setup below.

Fig. 14: Experimental set up

VI. Results And Discussion

Fig. shows the optimization for the chassis prototype. Topology optimization is done by removing material from the stress free regions to reduce weight aiming to get optimal design by using minimum material. From the above four iterations third iterations is selected as optimized chassis prototype, because it has maximum stress value within yield stress and minimum deflection without affecting on the strength.

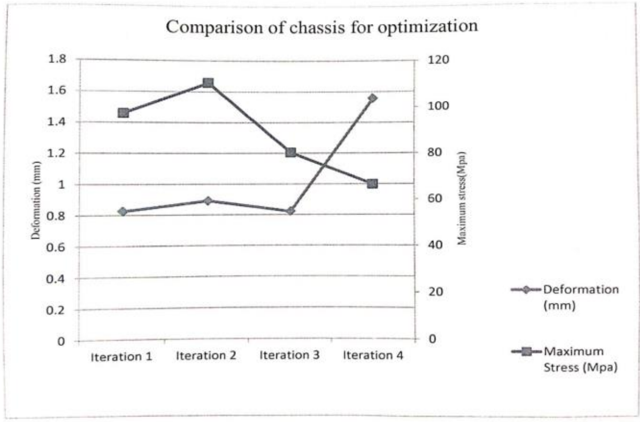


Fig. 15: Comparison of chassis for optimization

Table 4: Comparison of Deformation

Case	Total Deformation(mm)	% Error
Optimized Model Iteration 3	0.8221	6.33
Experimental Scale Down Model	0.77	

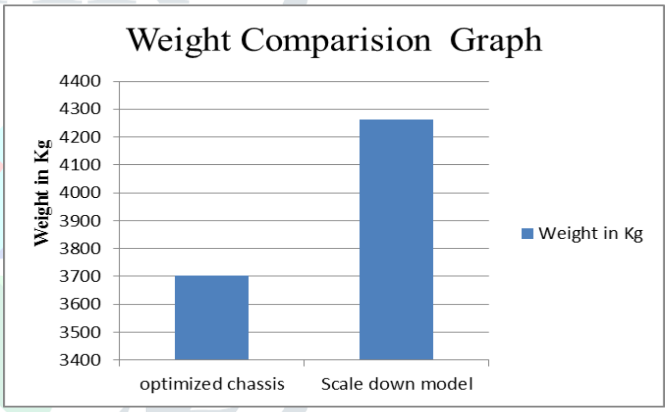


Fig. 16: Weight Comparison graph

Table 5: Weight Comparison

Case	Weight in Kg	% Error
Optimized Chassis	3703.271	13.1
Chassis Prototype	4261.69	

VII. Conclusion

From the above comparision between existing and optimized chassis its observed that optimized model is exibiting slightly high stress and deflection plots for considered loading case but the values are well below the critical value of 370Mpa yielding. So it is concluded that topology optimization is successfully implemented.Hence the optimized low loader chassis will be totally safe under applied loading conditions. Weight reduction of 13.1% is obtained without compromising the strength.

References

[1] MohdAzizi Muhammad Nor, Helmi Rashid, Wan MohdFaizul Wan Mahyuddin, "Stress Analysis of a Low Loader Chassis", International Symposium on Robotics and Intelligent Sensors 2012, Elsevier, 1877-7058.

[2] GrzegorzKoszalka, Hubert Debski, Marek Dziurka, MaciejKaczor, "Design of a Frame to a Semi Low-Loader", Journal of KONES Powertrain and Transport, Vol. 18, No. 2 2011.

[3] Haresh K. Vaniya, V. D. Sonara,Arvind. S. Sorthiya, "Analysis of Backhoe Loader Chassis for Weight & Cost Reduction using FEA - A Review Paper", International Journal for Innovative Research in Science & Technology, Volume 1, Issue 8, January 2015, ISSN (online): 2349-6010.

[4] Tandra Naveen kumar, N.Jeevan Kumar, "Design and Analysis of Army Vehicle Chassis", International Journal of Engineering Trends and Technology, Volume 9, Number 3 - Mar 2014, ISSN: 2231-5381.

[5] Ahmad O. Moaaz, Nouby M. Ghazaly, "Finite Element Stress Analysis of Truck Chassis using Ansys: Review", International Journal of Advances in Engineering & Technology, Nov. 2014. , ISSN: 22311963.

[6] Abhishek Singh, Vishal Soni, Aditya Singh, "Structural Analysis of Ladder Chassis for Higher Strength", International Journal of Emerging Technology and Advanced Engineering, Volume 4, Issue 2, February 2014, ISSN 2250-2459.

[7] Nouby M. Ghazaly, "Applications of Finite Element Stress Analysis of Heavy Truck Chassis: Survey and Recent Development", Journal of Mechanical Design and Vibration, 2014, Vol. 2, No. 3, 69-73.

[8] Hemant B.Patil, Sharad D.Kachave, EknathR.Deore, "Stress Analysis of Automotive Chassis with Various Thicknesses", IOSR Journal of Mechanical and Civil Engineering, Volume 6, Issue 1 (Mar. - Apr. 2013), PP 44-49, e-ISSN: 2278-1684.

[9] Dr.R.Rajappan, M.Vivekanandhan, "Static and Modal Analysis of Chassis by Using Fea", The International Journal Of Engineering And Science,Volume 2, Issue 2, Pages63-73, 2013, Issn: 2319 – 1813.

[10] Vishal Francis, Rajnish Kumar Rai, Anup Kumar Singh, Pratyush Kumar Singh, Himanshu Yadav, "Structural Analysis of Ladder Chassis Frame for Jeep Using Ansys", International Journal Of Modern Engineering Research, Volume 4, Issue 4, Apr. 2014, ISSN: 2249–6645.