Stress Analysis and Weight Reduction of Automobile Chassis-A Review

G.Vignesh
P.G. Scholar
Department of Mechanical Engineering
Adhiyamaan College of Engineering Hosur – 635109

Halesh Koti
Assistant Professor
Department of Mechanical Engineering
Adhiyamaan College of Engineering Hosur – 635109

Dr. Channankaiah
Head of the Department
Department of Mechanical Engineering
Adhiyamaan College of Engineering Hosur – 635109

Abstract

Finite element stress analysis of a light vehicle chassis plays an important role during design stages. Nowadays, there are many research papers and advancing programs available in the market especially by the light vehicle manufacturers, which are very much similar to this research work. The paper reviews the most important research works, a technical journal, and conferences papers that have been published. The paper focused on stress analysis of the light vehicle chassis using finite element assortment ANSYS. The results of reading this paper will give the researcher a review of some recent and current developments in the range of vehicle design using finite element stress analysis.

Keywords: Chassis, FEA, Analysis, CATIA V5 R20

I. INTRODUCTION

The basic structure & the frame of the vehicle are called as chassis. In a vehicle, chassis is the part of the vehicle without a body. All the important systems of the vehicle like the engine, transmissions, brake, axles; suspensions, etc. climb up on the chassis. Chassis can also be termed as carrying unit as it provides for the mounting of all units including the body. Hence, the chassis can be referred to as the backbone of the vehicle. Various loads acting on the chassis are:

- Loads induced due to cracked patch termed as short duration loads,
- Cornering loads when vehicle takes a turn,
- Impact loads due to destruction of vehicle,
- Loads induced due to acceleration and braking termed as inertia loads
- Loads due to the overwhelming condition considered behind vehicle capacity.

Structural investigation of the chassis can be observed by finite element analysis method

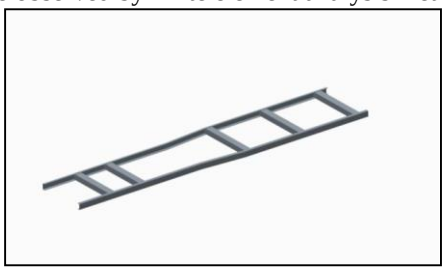

Fig. 1: Cad Model of Chassis

Normally software packages used for the analysis are Hyper Works, ANSYS, NASTRAN, etc.

II. LITERATURE REVIEW

Patel Vijaykumar V et al. [1] focused their work towards weight reduction of chassis by performing structural analysis. Basic calculations for the chassis frame were done systematically based on the bending theory and values of stress and deflection was obtained. Finite element analysis for the existing chassis was performed for overload condition and stress and deflection values were received. For weight reduction, design modifications were made by doing a sensitivity analysis. In a sensitivity analysis, part modulus and flange width were kept constant. Three cases were considered for weight reduction in which thickness and height of the flange were diverse. Comparison of the results showed that out of three cases the third case resulted in a weight reduction of about 7% and stress and deflection values were reduced by about 12% and 11% respectively.

All rights reserved by *www.ijirst.org*

S. S. Sane et al. [2] performed stress analysis of a light commercial vehicle chassis by using finite element method. Initially analysis of the actual chassis of the vehicle was performed and values of the stress and deflection were gathered. A total of nine different load cases were considered for the analysis of the chassis. To reduce the stress values stiffeners of changing height were added at the critical locations. The chassis was analyzed by finite element method using Altair Hyper Works software package. Identification of the results showed that stresses at the critical location were reduced up to 44% by addition of stiffener.

Sandip Godse et al. [3] performed a static load analysis of TATA Ace Ex Chassis using ANSYS Workbench. In their work the authors performed static analysis of the actual chassis of the vehicle and found out the maximum value of stress and deflection. The author performed optimization exercise by the method of reinforcement techniques to reduce the value of stresses and increase the payload. After inclusion of reinforcement's static analysis was performed and it was initiated that the stress values were reduced by about 38% thus increasing the payload of the chassis. Thus, the author terminates that the modification made by optimization of the chassis through reinforcement techniques the chassis is capable of carrying loads beyond the previous payload.

Hirak Patel et al. [4] carried out a sensitivity analysis for weight reduction of different cross-sections of truck chassis. The important benchmark for the analysis of the chassis was maximum stress, maximum equilateral stress, and deflection. The author performed his work against the optimization of the automotive chassis for which a proper finite element model of the chassis was advanced in PRO-E and finite element analysis was carried out using the ANSYS Workbench. The truck chassis search design was done analytically and the weight optimization was done by sensitivity analysis and a 17% weight reduction was achieved.

S. Prabhakaran et al. [5] focused their work towards weight reduction of chassis by performing structural analysis. Basic calculations for the chassis frame obtain done analytically based on the bending theory and values of stress and deflection was obtained. Finite element analysis for the existing chassis was performed for overload condition and stress and deflection values were achieved. For weight reduction, design modifications were made by doing a sensitivity analysis. In sensitivity analysis section modulus and flange width were stored constant. Three cases were considered for weight reduction in which thickness and height of the flange were diverse. Comparison of the results showed that out of three cases third case resulted in about 6.7% of weight reduction.

Marco Cavazzuti et al. [6] focused their study on weight reduction of the automotive chassis by using structural optimization method linked with finite element analysis. Various optimization techniques were explained. The methods were briefly introduced, and some applications were presented and discussed with the aim of showing their potential. A particular focus was given to weight reduction in automotive chassis design applications. The author provided a quick overview of structural optimization methods. The author explained how topology and topometry optimizations were more convenient for an early development stage, whose outcome could be further refined through size and shape optimizations.

Chinnaraj et al. [7] explained current trend in automotive design to optimize components for weight reduction. To achieve this, the chassis frame body of a heavy truck used for long distance goods hauling application was chosen for investigation. A quasi-static access that approximates the dynamic maneuvers into a number of small processes having static equilibriums was followed to bear out the numerical simulation, approximating the dynamic behavior of frame assembly. With the help of economical finite element package ANSYS, the quasi-static numerical simulations were carried out and compared with experimental results.

Ingole and Bhope [8] carried out static analysis for the chassis considering sudden load effects to modify a 8 ton 4 wheeled trailer which ultimately results in a reduction of weight and manufacturing cost. PRO-E was used for modeling the chassis of a tractor trailer. ANSYS software was used for analysis and simulation. They did some modifications in a model of chassis namely; variation in cross-sectional areas of the cross and longitudinal members and changing the position of cross members of mainframes of chassis. It has been found that the maximum stress in the longitudinal member is 75 Mpa. Also, they reported that if quick loads effects are considered then stress level may rise to approximately to twice that will be approximately 150 Mpa.

Haval Kamal et al. [9] discussed the frame of the standard dump truck supports all types of complicated loads coming from the road and carriage being loaded. A frame of 6 wheels, a standard dump truck has been studied and analyzed using ANSYS software. The static energy of the frame has been analyzed when exposed to pure bending and torsion stress, within two cases. The first case was when the rear wheels zigzag gets over the block and the second case was when both wheels get over the block. The reroduction results showed that the critical point of stress occurred when the truck zigzag ramp the block. The big issue was given to the case of zigzag wheels of the dump truck ramp the block because there was a great difference in the torsion stress values in both two case studies.

Ravi Chandra et al. [10], described the analysis of heavy vehicle chassis for weight reduction using PRO–E software for designing, ANSYS 12.0 for analysis. The dimensions of the heavy vehicle chassis of a TATA 2515EX vehicle was taken for modeling and analysis of a heavy vehicle chassis among three different composite materials namely, Carbon/Epoxy, E-glass/Epoxy, and S-glass /Epoxy subjected to the equal pressure as that of a steel chassis. The design constraints were stresses and deflections. The three various composite heavy vehicle chassis have been modeled by considering three various cross-sections. Namely, C, I and Box type cross sections. For validation, the design was done by applying the vertical loads acting on the horizontal various cross sections. From the results, it was observed that the polymeric composite heavy vehicle chassis is lighter and more practical than the conventional steel chassis with similar design specifications.

PAUL et al. [11] analyzed the ladder frame for static load condition with the stress, deflection bending moment on truck chassis frames. The truck chassis was modeled in CATIA V5 R16 and then it imported into Ansys. The side members of the

All rights reserved by *www.ijirst.org*

chassis were considered as the beam as it helps to simulate various attachments over the chassis, like fuel tank mountings, engine mountings, etc. The model is analyzed on the basis of static load condition due to mountings. The stress is achieved more at the joints where the weld is present. The results illuminate the new creative ways for optimum frame design which generate it more sustainable for structural concerns.

Hemant Patil et al. [12] presented stress analysis of a ladder type low loader truck chassis structure consisting of C-beams design using CATIA V5R10 and ANSYS, as shown in Figure 2. In order to achieve a reduction in the magnitude of stress at the critical point of the chassis frame, side member thickness, cross member thickness and position of a cross member from rear end were varied. Numerical results showed that if the thickness change is not possible, changing the location of cross member may be a good alternative. Computed results were compared to analytical calculation, where it was constructed that the maximum deflection agrees well with theoretical approximation but varies on the magnitude aspect. They concluded that it is better to change the thickness of cross member at critical stress point than changing the thickness of side member and position of chassis for a reduction in stress values and deflection of chassis.

Darshit Nayak et al. [13], presented an analysis of the static stress that acting on the upper surface of the truck chassis. Critical parts that will heavy to failure were also observed. 3-D finite element model of the truck chassis was made using Pro-E before analyzed through ANSYS software. Model updating of truck chassis model was prepared to adjust the selective plots such as mass density and Poisson's ratio. Numerical results showed that critical part was at the arise joint of the tire and also at the front part of the chassis. Some modifications were also suggested to reduce the stress and to increase the strength of the truck chassis.

Madan Mohan Reddy et al. [14] investigated the modeling and analysis of container chassis using FEM to improve load bearing capacity and reducing the failure of chassis with bending by adding stiffeners. The rectangular stiffeners to be placed in between the cross affiliates and fastened to chassis by means of bolts. The analysis results of Ansys-14 display that there is a reduction in von misses stress in chassis with stiffener up to the extent of 37.11% compared to without stiffener while stress magnitude reduced up to 36.23% and deflection reduced by 36.16%.

Swami K.I et al. [15] investigated the static structural analysis of truck chassis with the help of ANSYS software. Here the chassis of Eicher 20.16 is of ladder frame category which has two side affiliates or longitudinal members of C- cross section and seven transverse division called cross members of C- cross section. The results from graph show that as the side affiliate thickness increases, initially there is a slight decrease in the maximum value of von misses stress but afterward, it starts increasing. The rate decreases in just before the end and again increases at the end.

III. CONCLUSION

The present study has analyzed the various literatures. After a careful analysis of various research studies conducted so for it has been found that satisfactory studies have not been conducted on variable section chassis concept. Hence, in order to plenty the gap future, research studies may be conducted on variable section chassis concept in automobiles.

REFERENCES

[1] Patel Vijaykumar V., Prof. R. I. Patel, "Structural Analysis of Automotive Chassis Frame and Design Modification for Weight Reduction" International Journal of Engineering Research and Technology ISSN: 2278-0181, Volume-1, Issue-3, May 2012, pp. 1-6.
[2] S. S. Sane, Ghanashyam Jadhav, Anandraj H., "Stress Analysis of a Light Commercial Vehicle Chassis by FEM" Piaggio Vehicles, HTC 08, pp.1-5.
[3] Sandip Godse, Prof. D. A. Patel, "Static Load Analysis of Tata Ace Ex Chassis and Stress Optimization Using Reinforcement Technique" International Journal of Engineering Trends and Technology, Volume-4, Issue-7, July-2013, pp. 3037-3039.
[4] Hirak Patel, Khushbu C. Panchal, Chetan S. Jadhav, "Structural Analysis of Truck Chassis Frame and Design Optimization for Weight Reduction" International Journal of Engineering and Advanced Technology ISSN: 2249 – 8958, volume-2, Issue-4,2013.
[5] S. Prabakaran, K. Gunasekar, "Structural Analysis of Chassis Frame and Modification for Weight Reduction" International Journal of Engineering Sciences and Research Technology, ISSN: 2277-9655, Volume-3(5), pp. 595-600.
[6] Marco Cavazuuti, Luca Splendid, "Structural Optimization of Automotive Chassis: Theory, Setup, Design" Structural and Multidisciplinary Optimization, 2011 pp. 1-3.
[7] K Chinnaraj, M Sathya Prasad, and C Lakshmana Rao, "Experimental Analysis and Quasi-Static Numerical Idealization of Dynamic Stresses on a Heavy Truck Chassis Frame Assembly" Applied Mechanics and Materials, Vols.13-14, pp. 271-280, 2008.
[8] N. K. Ingole and D.V. Bhope "Stress analysis of tractor trailer chassis for self-weight reduction" International Journal of Engineering Science and Technology, Vol. 3 No. 9 September 2011.
[9] H. Kamal Asker, T. Salih Dawood, A. Fawzi Said, ―Stress Analysis of standard Truck Chassis during ramping on Block using Finite Element Method", ARPN Journal of Engineering and Applied Sciences, Vol. 7, NO. 6, June 2012.
[10] M. Ravi Chandra, S. Sreenivasulu, Syed Altaf Hussain, "Modeling and Structural analysis of heavy vehicle chassis made of a polymeric composite material by three different cross sections" International Journal of Modern Engineering Research, Vol.2, Issue.4, pp-2594-2600, 2012.
[11] I. D. Paul, S. M. Sarange, G. P. Bhole and J. R. Chaudhari "Structural Analysis Of Truck Chassis Using Finite Element Method" International J. of Multidisc. Research & Advice. In Engg. Vol. 4, No. I (January 2012), pp. 85-98, 2012.
[12] B. Hemant Patil, Sharad D. Kachave, Eknath R. 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, 2013.
[13] H. Patel, Khushbu C. Panchal, Chetan S. Jadav "Structural Analysis of Truck Chassis Frame and Design Optimization for Weight Reduction" International Journal of Engineering and Advanced Technology, Volume-2, Issue-4, April 2013.
[14] D. Nayak, P. Kumar Sharma, Ashish parkhe "Modelling And Analysis Of Existing And Modified Chassis In Tata Truck " International Journal of Advanced Technology in Engineering and Science, Volume No.02, Issue No. 05, May 2014.
[15] Madan Mohan Reddy and Lakshmi Kanta Reddy, "Modeling and Analysis of container chassis using FEM", International Organization of Scientific Research Journal of Engineering (IOSRJEN), Vol. 04, Issue 01 (January. 2014), pp. 34-37.
[16] Swami K.I.and Tuljapure S.B, "Analysis of Ladder Chassis of Eicher 20.16 Using FEM", IOSR Journal of Applied Geology and Geophysics (IOSR-JAGG), Vol. 2, Issue 1 Ver. I. (Jan. 2014), pp.06-13.

All rights reserved by *www.ijirst.org*