Journal of Basic and Applied Engineering Research
Print ISSN: 2350-0077; Online ISSN: 2350-0255; Volume 1, Number 12; October-December 2014 pp. 172-175

http://www.krishisanskriti.org/jbaer.html

Geometry Optimization of Automobile Chassis- A Review

Nishant Raman, Abhishek Shukla, Gaurav Shukla and Irfan Ul Haq*

School of Mechanical Engineering, S.M.V.D.U. Katra, Jammu & Kashmir 182320, India
**E-mail: haqmechanical@gmail.com*

Abstract: *With the advent of computers and competitive market trends to reduce product development time, computer aided analysis holds a significant position in the modern product design cycles throughout the industry, in general, and particularly in automotive sector. In order to mitigate the costs incurred in prototype building, on road tests and to come up with optimized designs, automobile industries are focusing more on computer aided simulations. An automotive chassis being the primary load bearing member of an automobile, its proper designing as well as geometric optimization is important so as to ensure the durability of the automobile. Researchers throughout the globe have been working on the virtual analysis of the chassis by subjecting it to various kinds of loads and studying the stress patterns thereof. Our paper presents a review of all the novel work done in this field.*

Keywords: *Chassis, Geometry Optimization, Finite Element Methods, Stress Analysis, Fatigue.*

1. INTRODUCTION

Chassis is the main frame of an automobile which supports all the major components of a vehicle. These components are engine, fuel tank, suspension system, transmission system, steering system, braking system, axel and wheels so it should be able to withstand the weight of all these components. Chassis is also subjected to inertia forces, braking forces, cornering forces, sudden impact, shocks and vibration due to the irregularities of the road so it must be able to withstand the stresses induced due to these forces without an appreciable amount of deflection. There are mainly two types of chassis. First is conventional or space frame type and the second is monocoque. Space frame chassis consists of circular and square tubes, welded together to form a lattice structure. This type of chassis is formed by using compressed steel bars. Monocoque is generally constructed in only one piece without any joints and support the structural loads using external skin. Monocoque requires no separate frame to support the loads as this is constructed in one unit.

As chassis is subjected to different type of loading so it is very necessary to determine the types of loading and the stresses caused by these loadings at various points in a chassis before manufacturing it so that the chassis may be designed on the basis of maximum stress and maximum deflection.

This paper deals with the static and dynamic analysis and also geometrical optimization of different types of chassis using Finite Element Methods.This analysis is the investigation of location of high stress area and torsion stiffness of the chassis.

2. LOAD CASES AND BOUNDARY CONDITIONS

For the finite element analysis of a chassis it is very important to know the different types of load acting on chassis and different boundary conditions. Ashutosh Dubey analyzed the various load cases and boundary conditions for stress analysis of the chassis using finite element analysis. Author used shell elements for the simulation of the longitudinal and cross members, beam elements for the simulation of the various attachments and spring elements for suspension system. Author used FFT analyzer for the power spectrum analysis and reviewed that the worst case of load and overloading must be considered for the analysis of the static load. For dynamic load conditions author has taken some considerations which are, loads on the chassis when vehicle moving on elevation, loads when the vehicle accelerating from low speed, loads on the chassis on application of braking, cornering forces while taking a turn. The researcher took the situation of bumps on diagonally opposite wheels, combination of loads, road bump load case, did power spectrum density analysis and formed the boundary conditions for the analysis of stresses.

3. FATIGUE ANALYSIS

While performing the optimization of chassis, the fatigue behavior of the chassis plays a major role. Considerable work in this field has been reported.

C.L. Petracconi, et al have simulated the fatigue life of the hook assembly of a passenger car. Experiments were performed simulating the actual conditions in the customer environment. S-N curves, Rain flow Counting and Miner rule were used to determine the fatigue damage imposed on the component. The researcher has evaluated the measured strain and stresses, simulated the tests and fatigue life assessments, on the basis of S-N curve.

V.Veloso, et.al have done failure investigation and stress analysis of a longitudinal stringer of an automobile chassis. Experimental tests with an instrumented vehicle were performed to obtain the values of acceleration and forces on the failure region of the component. It was determined an acceleration of 5g and a force of 35 KN at the bumpers fixation points. A finite stress analysis is carried out at the failure region to determine the stress distribution and possible design improvement. The FEM model was found in Hypermesh software. An external reinforcement models were proposed like the best solution to decrease the active stresses on the longitudinal stringer.

4. STATIC AND DYNAMIC ANALYSIS AND GEOMETRY OPTIMIZATION

For the determination of maximum stress and maximum Deflection region before manufacturing the chassis, static and dynamic analysis of a chassis is very important so that an optimized safety factor be given to it.

In September 2008, A. Gauchia, et. al studied the stress distribution of a Real Bus Structure. The structural behavior of the Bus when subjected to weight and torsion was analyzed by using Finite Element Model (FEM). The geometrical optimization was accomplished by Genetic Algorithm without decreasing the vehicle safety. In this Work the researcher have found that the selection of an adequate fitness function greatly influences the obtained solution. By modifying only four beam cross section from a total of 55, a reduction of 4 percentage of the bus weight structure and an increase of 0.23 percentage of torsion stiffness was obtained.

N.K Ingole made a stress analysis of a tractor trailer chassis for self weight reduction. Finite Element Method was implemented on 8 ton four wheeler trailer to modify this chassis which results in reduction in weight with safe stress and manufacturing cost. For FE analysis, he created a solid model of chassis on PRO-E and a FE Model. In this work, static analysis was carried out by considering sudden impact loads. The author has carried out Finite Element Analysis on existing chassis for four different cases. Case 1 was the variation in cross sectional areas of cross members. Case 2 was the variation in cross sectional areas of cross and longitudinal members. Case 3 was the variation in cross sectional areas of cross and longitudinal members and changing the position of cross members. Case 4 was the variation in cross sectional areas of cross and longitudinal members. The results obtained by him are shown in the table below:

(Tensile Yield Strength of Material is 250 MPa)

Table 1; Comparison of Results for different cases

S. N.	Various cases	Weight in Kg	Range of equivalent stresses on members in Mpa	Final reduction in weight in Kg	Factor of safety under sudden load(Without plates)	Factor of safety(with plates)
1	Existing chassis	751.82	28 to 75	-	1.66	3.37
2	Case 1	705.88	17 to 69	45.88	1.78	2.71
3	Case 2	674.67	22 to 75	77.15	1.66	2.78
4	Case 3	663.87	25 to 66	87.95	1.89	2.31
5	Case 4	640.09	42 to 75	111.73	1.66	3.37

Roslan Abd Rahman, et.al investigated stress analysis of heavy duty truck chassis using Finite Element Method. Finite Element result shows that the critical point of stress occurred at opening of chassis which is in contacted with the bolt. Thus they have reduced the stress magnitude at that point.

Mohd Hanif Mat, et. al proposed a light weight steel space framed chassis for a 3 cylinder 660 cc engine single seated “Eco Challenge” race car to increase its fuel efficiency as well as load carrying capacity. Geometrical optimization and weight reduction were done by stress analysis using ANSYS software. The result of the FEA analysis showed that the chassis was able to support all the loading conditions i.e. dead weight of the vehicle and load due to acceleration, braking and cornering forces.

Cicek Karaoglu, et. Al studied stress analysis of a truck chassis with riveted joints by using FEM .In order to achieve a reduction in the magnitude of stress near the riveted joints of chassis frame, side member thickness, connection plate thickness and connection plate length were varied. The results obtained in the analysis depicted that increasing side member thickness can reduce the stresses in joint areas but overall weight of chassis will also increase. Using local plates also only in the joint area can also increase side member thickness and excessive weight of chassis can be prevented. Increasing the connection plate thickness can reduce stresses in the connection plate. When the length of the connection plate is increased, stresses in both side member and connection plate decrease.

Carl Reed, Jaguar Cars Limited Body and Trim CAE Engineering Centre studied the vehicle body structure and optimized the mass and increased the stiffness of the vehicle structure. For this the researcher has used the Altair software. This was achieved by assuming the vehicle structure as a single component. By the help of software the author has distributed the material uniformly throughout the structure, it was done in efficient manner to maximize stiffness and minimize mass. The author has used two types of load cases for optimization:

1. NVH type load cases- encompassing local and global body static stiffness.

2. Crashworthiness load cases- encompassing energy load path management.

The design variables used for the optimization are based upon the dimensions of the beam and gauge of the shell elements. In isolation for the most efficient beam, its external dimension should be as large as possible whilst its wall thickness should be as thin as possible.

Sairam Kotari, et. al have carried out the static and dynamic analysis on Tatra chassis and improved the payload capacity by adding the stiffener and C channel at maximum stress region of chassis frame. They have improved the payload from 10 tones (without modification) to 14 tones (after modification). The authors have analyzed the stress deflection, bending moment for static and dynamic load condition. The authors have also found the maximum deflection region and corresponding stress and bending moment. At that point they have added the stiffener in an efficient manner and reduced the deflection and stress concentration. All of the analysis has performed in ANSYS.

Mohd Azizi Muhammad Nor performed a static analysis on the low loader chassis using finite element method and compared maximum stress and maximum deflection with theoretical results to obtain a preferable design safety factor for the low loader chassis. The results of Finite Element analysis revealed that the location of maximum stress and maximum deflection coincide with the location of analytical maximum stress and maximum deflection.

In the analysis of the chassis, the design of welded and riveted joint plays a major role. M. Zehsaz, et.al analyzed the behavior of the welded and riveted joints by using the three plates of thicknesses 5,8,12 mm at the maximum stress region. The authors have examined and evaluated the stress distribution. The authors found that by increasing the thickness of the connecting plate the amount of stresses in connection plates and chassis were decreased. The researchers have compared the result obtained and experimental results which were obtained previously related to vibration, frequency and modal shapes. After comparing the results they have evaluated that the use of combined welded and riveted joints reduce the stress concentration.

When large oscillations occur in a chassis, it causes the excessive deflection and failure of the chassis cross bars and longitudinal bars. The produced vibration causes high stresses at certain locations, fatigue of the structure, noise creation, loosening of joints and also discomfort to the occupants. To solve these problems R. Rajappan studied the dynamic characteristics of the truck chassis, to develop a new chassis they have determined the torsional stiffness, static and dynamic modal shape of the truck chassis, also improved the static and dynamic behavior of chassis by changing the dimensions and structural properties. All of the analysis has been performed in ANSYS. After modal analysis they have found that first six natural frequencies of the chassis were below the 100 Hz and varied from 16.24 to 61.64 Hz. Truck chassis experienced global vibration for first four mode of vibration except fifth. On the basis of that they reviewed that maximum stress occurred at the mounting brackets of suspension system and maximum translation occurred at the location of load symmetry.

5. CONCLUSION

In this paper an effort is made to review the work that have been made on different types of chassis to analyze the static and dynamic characteristics using Finite Element Methods so that its weight may reduce as well as strength may increase. An attempt has been made in the article to present an overview of various techniques for the geometry optimization of an automobile chassis.

REFERENCE

[1] Ashutosh Dubey and Vivek Dwivedi, “Vehicle chassis analysis: Load Cases and Boundary Conditions for Stress Analysis”.

[2] C.L.Petracconi, S.E. Ferreira, E.S. Palma, “Fatigue Life Simulation of a Rear Tow Hook Assembly of a Passenger Car”, Belo Horizonte, MG, Brazil, September 2009

[3] V.Veloso, H.S. Magalhaes, G.I. Bicalho, E.S. Palma, “Failure Investigation and Stress Analysis of a Longitudinal Stringer of an Automobile Chassis”, Belo Horizonte, MG, Brazil, December 9, 2008

[4] A. Gauchia, V. Diaz, M.J.L. Boada and B.L. Boada, “Torsional Stiffness and Weight Optimization of a Real Bus Structure”, Madrid 28911, Spain, March 2, 2009

[5] N.K Ingole “Stress Analysis of Tractor Trailer Chassis for Self Weight Reduction”, R.C.E.R.T Maharashtra, India, September 9, 2011

[6] Roslan Abd Rahman, Mohd Nasir Tamim, and Ojo Kurdi “Stress Analysis of Heavy Duty Truck Chassis”, December 2008

[7] Mohd Hanif Mat, Amir Radzi Ab. Ghani “Design and Analysis of Eco-Car Chassis”

[8] Cicek Karaoglu, N. Sefa Kuralay “Stress Analysis of Truck Chassis with Riveted Joints” October 8, 2001

[9] Carl Reed, Jaguar Cars Limited Body and Trim CAE Engineering Centre “Application of Optistruct Optimization to Body in White Design”

[10] Sairam Kotari, V.Gopinath “Static and Dynamic Analysis on TATRA Chassis”, IJMER, Vol. 2, Issue. 1, pp-086-094

[11] Mohd Azizi Muhammad Nor, Helmi Rashid, Wan Mohd Faizul Wan Mahyuddin, Mohd Azuan Mohd Azlan, Jamaluddin Mahmud “Stress Analysis of a Low Loader Chassis”, IRIS 2012

[12] M.Zehsaz, F. Vakili Tahimi and F, Esmaeili "The Effect of Connection Plate Thickness on Stress of Truck chassis with Riveted and Welded Joints under Dynamic Loads", Asian Journal of applied Sciences 2 (1): 22-35, 2009, Malaysia

[13] R. Rajappan and M. Vivekanandan "Static and Modal Analysis of Chassis by Using FEA", The International Journal of Engineering and Science, Vol. 2, Issue 1, pp-63-73, 2013

[14] G. Chiandussi, I. Gaviglio, A. Ibba "Topology Optimization of an Automotive Component without Final Volume Constraint Specification", July 2003

[15] M.M. Topac, H. Gunal, N.S. Kuralay "Fatigue Failure Prediction of a Rear Axle Housing Prototype by using Finite Element Analysis", 2009

[16] I.D. Paul, S.M. Sarange, G.P. Bhole and J.R. Chaudhari "Structural Analysis of Truck Chassis using Finite Element Method", IJMRAE, ISSN 0975-7074, Vol. 4, No. I, pp-85-98, January 2012

[17] Tushar M. Patel, Dr. M.G. Bhatt and Harshad K. Patel "Analysis and Validation of Eicher 11.10 Chassis Frame using ANSYS", IJETICS, Vol. 2, Issue 2, March-April 2013

[18] Haval Kamal Asker, Thaker Salih Dawood, Akram Fawzi Said "Stress Analysis of Standard Truck Chassis During Ramping on Block using Finite Element Method" APRN Vol. 7, No. 6, June 2012