International Journal of Research in Advent Technology (IJRAT)
(E-ISSN: 2321-9637)
Special Issue National Conference "CONVERGENCE 2018", 09 th April 2018

A review on Structural Analysis of Automobile Vehicle Chassis

Miss. Shraddha kuthe[1], Prof. K. R. Sontake[2],
Department of Mechanical Engineering[1,2],PLITMS Buldana[1,2]
Email: shraddhakuthe@gmail.com[1] ,krs_mech@plit.ac.in[2]

Abstract- The Chassis frames are most valuable part in all vehicle compared to all automobile parts. So it must be strong enough to resist the shock, twist, vibration and other stresses. In this paper an effort is made to review the investigations that have been made on the different analysis techniques of automobile frames. Different types of chassis of various vehicles are introduced to study the effect of loading on automobile. A number of analytical and experimental techniques are available for the analysis of the automobile frames. Determination of the different analysis around different condition in an automobile frames has been reported in literature. Some of the authors were works in finding maximum stress and maximum deflection. The main objectives of the authors are to determine the maximum stress, maximum deflection and to recognize critical regions under static loading condition.

Key Words –Structural Analysis, Automobile Chassis, FEM

1. INTRODUCTION

The chassis frame plays a vital role in the automobiles. 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 [2]. 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.

The magnitude of the stress can be used to predict the life span of the chassis. The location of critical stress point is thus important so that the mounting of the components like engine, suspension, transmission and more can be determined and optimized, Finite Element Method (FEM) is one of the method to locate the critical point [3,4]. Safety factor is used to provide a design margin over the theoretical design capacity. This allows consolidation of uncertainties in the design process [5].Jadav Chetan S. et al reviews various factors affecting the fatigue life of an structure like cyclic stress state, geometry, surface quality, material type, residual stresses, size and distribution of internal defects, direction of loading & grain size [6].

2. LITERATURE REVIEW

Ramesh kumar. S, Dhandapani. N. V and et al introduced the numerical method of analysis using ANSYS package for chassis structural design and optimization. The stress analysis and fatigue analysis are the main analysis carried out on the chassis using ANSYS software. This is particularly done for the identification of the weak points and the life of the chassis. Further even more research has to be continued to carry out in the near future for the better enhancement of the chassis structure and design.

Kiran Ghodvinde & S. R.Wankhade were used the numerical analysis as finite element technique to find the critical stress. In this work they have analyzed the monologue and ladder frame for static load condition with the stress, deflection bending moment and even the analysis of two different chassis with same as discuss above frame are being analyzed.

In this the work deal with the static analysis of two different frame automotive chassis, the Chevy truck chassis shows the critical stress at the joints and it is being reduce by increasing the side member thickness, connection plate thickness and connection plate length were varied. Numerical results showed that stresses on the side member can be reduced by increasing the side member thickness locally. If the thickness change is not possible, increasing the connection plate length may be a good alternative to improve the strength. It was shown that an FEA model could be used to simulate the automotive chassis accurately by verification of stresses using the beam model. In this paper the analysis of different frames automotive chassis. In this analysed the car chassis under static

loading condition the analysis is being validate with the cantilever beam analysis. The analysis is purely static and the load is applied is same as compared to the work done in the analysis of beam. The result show the maximum stress at the point of contact and the bending moment can be understood.
Teo Han Fui and Roslan Abd. Rahman presented the study of the vibration characteristics of the truck chassis that include the natural frequencies and mode shapes. The responses of the truck chassis which include the stress distribution and displacement under various loading condition are also observed. The method used in the numerical analysis is finite element technique. The results show that the road excitation is the main disturbance to the truck chassis as the chassis natural frequencies lie within the road excitation frequency range. The mode shape results determine the suitable mounting locations of components like engine and suspension system. Some modifications are also suggested to reduce the vibration and to improve the strength of the truck chassis.

The paper has looked into the determination of the dynamic characteristic (the natural frequencies and the mode shapes) of the truck chassis, investigating the mounting locations of components on the truck chassis and observing the response of the truck chassis under static loading conditions. The first six natural frequencies of the truck chassis are below 100 Hz and vary from 12.68 to 61.64 Hz. For the first four modes, the truck chassis experienced global vibration except for the fifth mode. The global vibrations of the truck chassis include torsion, lateral bending and vertical bending with 2 and 3 nodal points. The local bending vibration occurs at the top hat cross member where the gearbox is mounted on it. The mounting location of the engine and transmission system is along the symmetrical axis of the chassis's first torsion mode where the effect of the first mode is less. For the linear static analysis, the stress distribution and deformation profile of the truck chassis subjected to two loading conditions: truck components loading and asymmetrical loading had been determined. Maximum stress occurred at the mounting brackets of the suspension system while the maximum translation occurred at the location where the symmetry and asymmetry load is acting. The maximum stress of the truck chassis is 490 MPa while the maximum translation is 33.6 mm. These values are acceptable as compared to the yield strength of the chassis material and the tolerance allowed for the chassis.

Patel Vijaykumar V and Prof. R. I. Patel were developed the finite element model of the chassis. The chassis is modeled in PRO-E. FEA is done on the modeled chassis using the ANSYS Workbench. The work performed towards the optimization of the automotive chassis with constraints of maximum shear stress, equivalent stress and deflection of chassis under maximum load. Structural systems like the chassis can be easily analyzed using the finite element techniques. A sensitivity analysis is carried out for weight reduction.

Monika S. Agrawal presented the study which is to produce results to rectify problems associated with structures of a commercial vehicle such as strength, stiffness and fatigue properties along with stress, bending moment and vibrations. This can be achieved by static and dynamic analysis, combining existing theoretical knowledge and advanced analytical methods. Design of a Chassis is carried by using CATIA and finite element analysis will be carried out by using ANSYS.
As conclusion, this study has achieved its core objectives. In Static Analysis, they were determined highly stressed area of truck chassis due to applied load, and analytical shear stress is 13.33% less than FEA values. In Modal Analysis we can determine the total deformation of truck chassis frame at a different frequency range. From the analysis results, the frequency range of Modal Analysis for Free-Free Condition is 16.89 Hz to 46.316Hz. Also frequency range of Modal analysis due to applied load on truck chassis is 13.886 Hz to 43.828.Hz (i.e. for Static Bending with Chassis Load). The Frequency range of both modal analyses for Free-Free Condition and Applied load on truck chassis are in the range 10 to 50 Hz. Almost all of the truck chassis designed were based on this frequency range to avoid resonance during operating conditions, so that the design of truck chassis is safe. By reducing the height of the cross-member of chassis by 8.6%, the weight reduction of chassis is found to be reduced by 8.72%.

A. Hari Kumar & V. Deepanjali presented the work to find out best material and most suitable cross-section for an Eicher E2 TATA Truck ladder chassis with the constraints of maximum shear stress, equivalent stress and deflection of the chassis under maximum load condition. In present the Ladder chassis which are uses for making buses and trucks are C and I cross section type, which are made of Steel alloy (Austenitic). In India number of passengers travel in the bus is not uniform, excess passengers are travelling in the buses daily due to which there are always possibilities of being failure/fracture in the chassis/frame. Therefore Chassis with high strength cross section is needed to minimize the failures including factor of safety in design. In the present work, we have taken higher strength as the main issue, so the dimensions of an existing vehicle chassis of a TATA Eicher E2 (Model no.11.10) Truck is taken for analysis with materials namely ASTM A710 Steel, ASTM A302 Alloy Steel and Aluminum Alloy 6063-T6 subjected to the same load. The different vehicle chassis have been modeled by considering three different cross-sections namely C, I and Rectangular Box (Hollow) type cross sections. The problem to be dealt for this dissertation work is to Design and Analyze using suitable CAE software for ladder chassis. The report is the work performed towards the optimization of the automobile chassis with constraints of stiffness and strength. The modeling is done using Catia, and analysis is done using Ansys. The overhangs of the chassis are calculated for the stresses and deflections analytically are compared with the results obtained with the analysis software.

CONCLUSION:

The main aim of the paper is to review the various works done on the chassis. In this most of the authors works for static analysis with the actual loading conditions. The structural analysis was cried out with finite element methods. Optimization is done by reducing the weight of the chassis by finite element analysis through stress and static analysis. All the work in this literature can be use to better enhance the design of chassis for optimization and reducing the size of chassis.

REFERENCES

[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] 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

[3] Roslan Abd Rahman, Mohd Nasir Tamin, Ojo Kurdi "Stress analysis of heavy duty truck chassis as a preliminary data for its fatigue life prediction using FEM" Jurnal Mekanikal December 2008, No. 26, 76 – 85.

[4] Cicek Karaoglu, N. Sefa Kuralay "Stress analysis of a truck chassis with riveted joints" Elsevier Science B.V Finite Elements in Analysis and Design 38 (2002) 1115–1130.

[5] Mohd Azizi Muhammad Nora,, Helmi Rashida, Wan Mohd Faizul Wan Mahyuddin "Stress Analysis of a Low Loader Chassis" Elsevier Ltd. Sci Verse Science Direct Procedia Engineering 41 (2012) 995 – 1001.

[6] Jadav Chetan S., Panchal Khushbu C., Patel Fajalhusen "A Review of the Fatigue Analysis of an Automobile Frames" International Journal of Advanced Computer Research, Volume-2 Number-4 Issue-6 December-2012, (ISSN (print): 2249-7277 ISSN (online): 2277-7970).

[7] Ramesh kumar. S and et al."Design And Analysis Of Automotive Chassis Frame Using Finite Element Method" International Journal of Pure and Applied Mathematics, Volume 118 No. 20 2018, 961-972, ISSN: 1314-3395

[8] Kiran Ghodvinde & S. R.Wankhade " Structural Stress Analysis of an Automotive Vehicle Chassis" International Journal on Mechanical Engineering and Robotics (IJMER), ISSN (Print) : 2321-5747, Volume-2, Issue-6,2014

[9] Teo Han Fui and Roslan Abd. Rahman Statics and Dynamics Structural Analysis of A 4.5 Ton Truck Chassis, Jurnal Mekanikal December 2007, No. 24, 56 - 67

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

[11] Monika S.Agrawal "Finite Element Analysis of Truck Chassis Frame", International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056 Volume: 02 Issue: 03, June-2015 (A.1)

[12] A.Hari Kumar & V.Deepanjali "Design & Analysis of Automobile Chassis", International Journal of Engineering Science and Innovative Technology (IJESIT) Volume 5, Issue 1, January 2016.