Study of Vehicle Chassis Design

Ankush Yadav[1] Divyanshu Shekhar[2]
[1,2]Department of Automobile Engineering
[1,2]Dronacharya Group of Institution

Abstract— The objective of paper is to find out best material and most suitable cross-section for a 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 used for making buses and trucks are C and I cross section type, which are made up 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 failure including the 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 .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 vehicles 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 the Catia, and analysis is done using the Ansys . The overhangs of the chassis are calculated for the stresses and deflections analytically are compared with the results obtained with the analysis software. Keywords: Automobile chassis, chassis loads, modeling, structural analysis.

Key words: Vehicle Chassis, Structures of an Automobile

I. INTRODUCTION

The chassis is considered to be one of the significant structures of an automobile. It is the frame which holds both the car body and the power train. Various mechanical parts like the engine and the drive train, the axle assemblies including the wheels, the suspension parts, the brakes, the steering components, etc., are bolted onto the chassis. The chassis provides the strength needed for supporting the different vehicular components as well as the payload and helps to keep the automobile rigid and stiff. Consequently, the chassis is also an important component of the overall safety system. Furthermore, it ensures low levels of noise, vibrations and harshness throughout the automobile. Chassis should be rigid enough to withstand the shock, twist, vibration and other stresses. Along the strength, an important consideration is chassis design is to have adequate bending and torsional stiffness for better handling characteristics. So, strength and stiffness are two important criteria for the design of chassis. The load carrying structure is the chassis, so the chassis has to be so designed that it has to withstand the loads that are coming over it.

II. ASPECT OF TRUCK CHASSIS FRAME

The common chassis frame consists of two channel shaped side members that are sustained apart by many cross members, as shown in Figure1. The cross members are placed at points of high stress and are joined to side members. The depth of the channel must be enough to reduce the deflection [1]. Since the load at each point of the frame varies, a weight reduction can be achieved by either minimums of the depth of the channel, or having a series of holes positioned along the axis in the regions where the load is not so high [2]. On the normal road surfaces, the chassis frame is subjected to both bending and torsion distortion. The open-channel sections exhibit excellent resistance to bending, but have very little resistance to twist [3]. From the global torsion analysis, it has been found that the torsion load is more severe than bending load. In order to overcome this problem, a cross bar and material selection are very important to consider during design stage [4]. Therefore, both side and cross-members of the chassis must be designed to resist torsional distortion along their length.

Figure 1

III. FATIGUE ANALYSIS OF TRUCK CHASSIS

Fatigue is the phenomenon in which a repetitively loaded structure fractures at a load level less than its ultimate static strength, as shown in the figure. There are many parameters that contribute to fatigue failures namely: number of load cycles experienced, range of stress experienced in each load cycle, mean stress experienced in each load cycle and presence of local stress concentrations [5]. Also, the fatigue analysis refers to one of three methodologies: local strain or strain life, commonly referred to as the crack initiation method, which is concerned only with crack initiation, stress life that commonly referred to as total life and crack growth or damage tolerance analysis that is concerned with the number of cycles until fracture. The main steps for calculating fatigue life is sometimes called the Five Box Trick, including material, loading, geometry inputs, analysis and results. Determining the stresses of a truck chassis before manufacturing is important due to improvement in design. An important aspect of chassis design and analysis is the stress distribution and fatigue life of prediction process. Chassis analysis mainly consists of static analysis to predict stress distribution and subsequently, the fatigue simulation to predict the life of the chassis. Many researchers carried out study on truck body components. Fermer et al. [6] investigated the fatigue life of Volvo S80 Bi-Fuel using MSC/Fatigue. Also, Fermer and Svensson in 2001 [7] studied the mechanical finite element analysis of welded structures performed on a daily basis in the automotive industry. One objective is to estimate the fatigue strength, which is given mainly by the strength of the joints. This article give some insight into the dimensioning process, with special focus on fatigue analysis of spot welds and seam welds in thin-walled car body structures made of steel. Conle and Chu [8] did research about fatigue analysis and the local stress –strain approach in complex vehicular structures. Moreover, Conle and Mousseau in 1991 [9] described an analytical study of the fatigue life of automobile chassis components using automotive proving ground load history results combined with recent computational advances. This work advances knowledge in two ways: a vehicle dynamics model is used to generate the history of the load vectors acting on the components and the element stress equivalency procedure used until now is improved. It can be concluded that the combination of vehicle dynamics modeling, finite-element analysis and fatigue analysis is a viable technique for the design of automotive components. However, before our durability process can be suitable for applied engineering work a number of improvements are required, which has been outlined. Thompson et al. (1998) [10] discussed the design and Analysis of a Winston Cup Stock Car Chassis for Torsional Stiffness using the Finite Element method. Roll stiffness between sprung and un-sprung masses. The application of the method is demonstrated using two case studies, namely a road tanker and a load haul dumper. In both cases, it was possible to obtain adequately accurate fatigue life prediction results, using simplified loading, static finite element analyses and a stress-life approach to fatigue damage calculations, with material properties available in design codes. Fatigue life analysis and improvements of the auto body in a sports utility vehicle (SUV) were performed by Zhong and Ping in 2006 [11]. The stress distribution under unit displacement excitation was obtained by the finite element (FE) method. A bilateral track model was adopted to obtain load spectra in accordance with the vehicle reliability test. The total life of the auto body was evaluated by the nominal stress method with the assumption of a uni-axial stress state, and thus the critical regions were determined. The life of components with critical regions was further investigated on the basis of multi axial fatigue theory. The results show that some components near to the suspension are easy to damage because they are directly subjected to impact loading from the road. It is also indicated that the result from multi axial fatigue analysis is more reasonable than that from the nominal stress method, which was verified by experimental results. Finally, topological optimization of the spot weld location in the critical region was carried out by the homogenization method to improve its fatigue life. In addition, Hoffmeyer et al. in 2006 [12] discussed some issues in multi axial fatigue and life estimation is presented. While not intended to be comprehensive, these are a relatively broad range of issues which are commonly encountered when dealing with multi axial fatigue. They include damage mechanisms, non-proportional hardening and constitutive behavior, damage parameters and life estimation, variable amplitude loading, cycle counting, damage accumulation, and mixed-mode crack growth. Some simple approximations in capturing some of these effects in multi axial life estimations are also presented. In 2007, Ye and Moan [13] discussed static and fatigue behavior of three types of aluminum box stiffener /web connections are investigated in this study. The main purposes are to provide a connection solution that can reduce the fabrication costs by changing the cutting shapes on the web frame and correspondingly the weld process and meanwhile sufficient fatigue strength can be achieved. Finite element analyses (FEA) show the influence of local geometry and weld parameters on the stress gradient near the fatigue cracking area. The influence of the weld parameters on the structural stress concentration factors is also studied. Twelve specimens of every type were tested and the test data are compared both to a nominal stress based design SN curve Eurocode9/31 and a structural stress based design SN curve Eurocode9/44. RoslanAbd Rahman et al. [14] conducted stress analysis of heavy duty truck chassis by utilizing a commercial finite element package ABAQUS. To determine critical point so that by design modifications the stresses can be reduces to improve the fatigue life of components. During this he uses ASTM low alloy steel a 710 C (Class 3) with 552 MPa of yield strength and 620 MPa of tensile strength for chassis founds the maximum stress 386.9 MPa at critical point occurred at opening of chassis This critical point is located at element 86104 and node 16045, which is in contacted with the bolt from this he concludes that this critical point is an initial to probable failure. Kurdi et al. in 2008, [15] discussed about the one of the most important steps in development of a new truck chassis is the prediction of fatigue life span and durability loading of the chassis frame. The age of many truck chassis in Malaysia are of more than 20 years and there is always a question arising whether the chassis is still safe to use. Thus, fatigue study and life prediction on the chassis is necessary in order to verify the safety of this chassis during its operation. Stress analysis using Finite Element Method (FEM) can be used to locate the critical point which has the highest stress. Critical point is one of the factors that may cause the fatigue failure. The magnitude of the stress can used to predict the life span of the truck chassis. The stress analysis is accomplished using the commercial finite element packaged ABAQUS by Velo so etal. [16] They discussed the failure investigation and stress analysis of a longitudinal stringer of an automobile chassis Fiat Auto motives, Rod. Fernão Dias, km 429, Betim, MG, Brazil Pontifical Catholic University of Minas Gerais

(PUCMinas), Mechanical Engineering, Belo Horizonte, MG, Brazil, A prototype vehicle was submitted to durability test, on road at a proving ground test track. Failures of posterior longitudinal stringers were observed during this test. Cracks were nucleated on these stringers during durability test, before the designed life of these components is reached. These cracks were observed at nearly the bumpers fixation points of the vehicle suspension. Loads are transmitted by wheels to the body of the vehicle through the suspension components. Thus, the longitudinal stringers are subjected to these localized cyclic stresses. Also, Palma et al. in 2009 [17] investigated to analyze the fatigue behavior of an automobile body part, according to the standards of performance. The methodology is based on experiments performed on a rear trailer tow hook pin of a passenger automobile vehicle. Experiments were performed simulating the actual conditions in the customer environment. Stress and strain were experimentally measured by using strain gages, bonded on assembly critical points. Besides, stress analysis was also performed using a finite element program. Fatigue analysis is used to access and to compare the fatigue damage imposed during laboratory experiments. Recently in 2011, Chen and Zhu [18] studied the YJ3128-type dump trucks sub-frames, for the fatigue crack occurred in the Subframe which has worked in bad condition for 3 to 5 months. The sub-frame was analyzed by ANSYS and the reason for the cracking of the frame was found according to the different stress.

IV. CONCLUSION

Fatigue analysis of the track chassis has been the focus of a number of previous works. Some of the previously conducted work related to vehicle structural design, analysis and optimization is surveyed. It is found that the chassis analysis mainly consists of stress analysis to predict the weak points and fatigue analysis to predict the life of the chassis. Several state of the art papers and even books on chassis stress analysis have been presented in the recently years. An attempt has been made in the article to present an overview of various techniques developed for the analysis of automobile frames and results of that analysis due to which further study on the chassis will become easy. Thus study makes a case for further investigation on the design of truck chassis using Fatigue Analysis.

REFERENCES

[1] S.Monika Agrawal, Md. Razik “Finite Element Analysis of Truck Chassis” International Journal of Engineering Sciences & Research Technology, December, 2013, 2(12).
[2] Nouby M. Ghazaly “Applications of Finite Element Stress Analysis of Heavy Truck Chassis: Survey and Recent Development” Journal of Mechanical Design and Vibration. 2014, 2(3), 69-73. DOI: 10.12691/jmdv-2-3-3.
[3] E. Macha and Nieslony A. Critical plane fatigue life models of materials and structures under multiaxial stationary random loading: The state-of-the-art in Opole Research Centre CESTI and directions of future activities. International Journal of Fatigue. 2012;. 39:95- 102.
[4] G. Murail, B. Subramanyam and D. vaveen “Design Improvement of a Truck Chassis based on Thickness” Altair Technology conference , India, 2013.
[5] J. Chetan S., Panchal Khushbu C., Patel Fajalhusen “A Review of the Fatigue Analysis of an Automobile Frames” International Journal of Advanced Computer Research, December-2012, Volume-2 Number-4 Issue-6.
[6] M., Fermer, McInally, G. and Sandin, G. Fatigue Life Analysis of Volvo S80 Bi-Fuel using MSC/Fatigue, Worldwide MSC Automotive Conference, and Germany, 1999.
[7] M., Fermér and Svensson, H. Industrial Experiences of FE-based Fatigue Life Predictions of Welded [8] Automotive Structures,Fatigue &Fracture of Engineering Materials and Structures 24 (7), 2001, 489-500.
[8] F.A.Conle, and Chu, C.C., Fatigue Analysis and the Local Stress-strain Approach in Complex Vehicular Structures, International Journal of Fatigue. 01/1997; DOI:10.1016/S0142-1123(97)00045-5.
[9] F.A. Conle, Mousseau, C.W. Using vehicle dynamics simulations and finite-element results to generate fatigue life contours for chassis components. International Journal of Fatigue.13, 3(1991), pp. 195-205. [10] L. Thompson. Srikanth Raju and E. Harry Law “Design of a Winston Cup Chassis for Torsional Stiffness” SAE, 983053.
[10] DU Zhong-zhe, ZHU Ping,HE Jun, Fatigue lifetime analysis of auto-body structure and spot weld based on finite element method Journal of Automotive Engineering, 2006, Vol. 28, pp. 944-947.
[11] J. Hoffmeyer, Doring, R.; Seeger, T.; Vormwald, M.: Deformation behaviour, short crack growth and fatigue lives ¨undermultiaxial non proportional loading. Int. J. Fat. , 28, 2006, 508 – 520.
[12] N. Ye, and T. Moan, Static and Fatigue Analysis of Three Types of Aluminum Box-Stiffener/Web Frame Connections, International Journal of Fatigue, 2007, 29: 1426-1433.
[13] R. Abd Rahman, Mohd Nasir Tamin, OjoKurdi “Stress analysis of heavy duty truck chassis as a preliminary data for its fatigue life prediction using FEM” JurnalMekanikal December 2008, No. 26, 76 – 85.
[14] O Kurdi R., Abd-Rahman M N Tamin. “Stress analysis of heavy duty truck chassis using finite element method”,Journal of Mechanical engineering, 2008, Vol. 26, pp. 76-85.
[15] V, VelosoMagalhaes H S, Bicalho G Iand Palma E S “Failure Investigation and Stres Analysis of a Longitudinal Stringer of an Automobile Chasis”, Engineering Failure Analysis, 2009, Vol. 16, pp. 1696-1702.
[16] E.S., Palma, Petracconi, C.L. and Ferreira, S.E. “Fatigue behavior analysis of a rear tow hook pin of a passenger vehicle”, Engineering Failure Analysis, 2009, Vol. 16, pp. 2408 - 2416.
[17] C. Yanhong, Zhu Feng “The Finite Element Analysis and The Optimization Design of the Yj3128-type Dump

