American Journal of Engineering Research (AJER)
e-ISSN : 2320-0847 p-ISSN : 2320-0936
Volume-3, Issue-10, pp-01-06
www.ajer.org

Research Paper Open Access

A Review of the Fatigue Analysis of Heavy Duty Truck Frames

Ahmad O. Moaaz[1], Nouby M. Ghazaly[2]

[1]*Mechanical Engineering Dept., Faculty of Engineering, Beni-Suef University, Beni Suef-62511, Egypt.*
[2]*Mechanical Engineering Dept., Faculty of Engineering, South Valley University, Qena-83523, Egypt.*

ABSTRACT: *Heavy duty vehicle plays a more important role in highway transportation. The major focus in the truck manufacturing industries is design of truck chassis with more pay load capacity and possible less weight. An important aspect of chassis' design and analysis is the stress distribution and fatigue life of prediction process. Fatigue is one of the most important parameters to consider when designing truck components. The components are typically subjected to dynamic loads when in service. In this paper, an effort is made to review the investigations that have been made on the different fatigue analysis techniques of heavy duty truck frames. A number of analytical and numerical techniques are available for the fatigue analysis of the heavy duty truck frames has been reported in literature.*

KEYWORDS *: Heavy duty truck, Chassis frames, Fatigue analysis.*

I. INTRODUCTION

Transportation industry plays a major role in the economy of modern industrialized and developing countries. The early damages caused by heavy load reduce the road service life and ride comfort. Highway maintenance brings huge economic losses, while serious road failure threats traffic safety. Chassis is one of the most significant components in automotive industry and is considered as the bigger parts in the vehicle. The chassis of trucks is tasked with holding all the components systems together; hence it should be rigid enough to withstand the shock, twist, vibration and other stresses. Along with strength, an important consideration in chassis design is to have adequate bending stiffness for better handling characteristics. So, maximum stress, maximum equilateral stress and deflection are important criteria for the design of the chassis. The greater the energy absorbed by the chassis on impact the lower the energy levels transmitted to a vehicles occupants and surroundings, lowering the chances of injury [1]. The chassis of trucks which is the backbone of vehicles that integrates the main truck component systems such as the axles, suspension, power train, cab and trailer etc., is one of the possible candidates for significant weight reduction [2].

In general, the chassis experiences several loading situations that include vertical bending, longitudinal torsion, lateral bending, torsion loading and fatigue loading [3]. In additions, the chassis design includes the selection of suitable shapes and cross-section of chassis-members. Moreover, the design should consider the reinforcement of the chassis side- and cross member joints, and the various methods of fastening them together. There are number of parameters which affect the fatigue life of the structure as listed: cyclic stress state, geometry, surface quality, material type, residual stresses, size and distribution of internal defects, direction of loading & grain size. Many researchers in the automotive industry have taken this opportunity to be involved in the chassis manufacturing technology and development [2].The objective of this review paper is to present the fatigue analysis techniques of heavy duty truck frames. Prevention of fatigue failure is one of the most important parameters to when designing truck components. The components are typically subjected to dynamic loads when in service. A number of analytical, numerical and experimental techniques are considered for the fatigue analysis of the heavy duty truck frames. Conclusion of the fatigue analysis in the vehicle chassis has been reported in literature.

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. Since the load at each point of the frame varies, a weight reduction can be achieved by either minimums 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. On the normal road surfaces, the chassis frame is subjected to both bending and torsional distortion. The open-channel sections exhibit excellent resistance to bending, but have very little resistance to twist. 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.

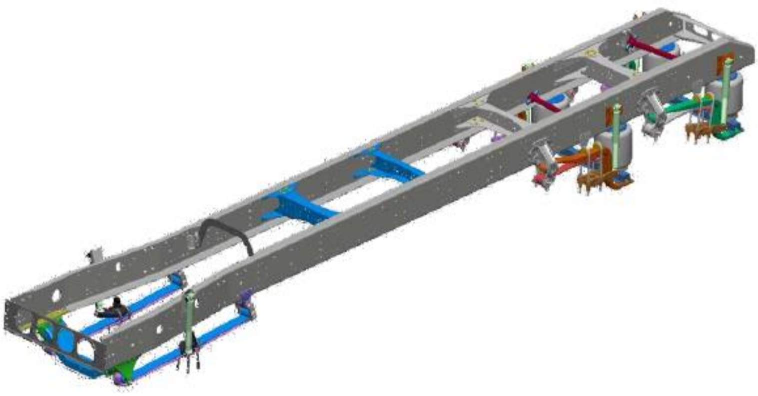

Figure1 Construction of the common chassis frame

Generally for heavy commercial vehicle channel section is preferred over hollow tube due to high torsional stiffness. The chassis frame, however, is not designed for complete rigidity, but for the combination of both strength and flexibility to some degree. The chassis frame supports the various components and the body, and keeps them in correct positions. The frame must be light, but sufficiently strong to withstand the weight and rated load of the vehicle without having appreciable distortion. It must also be rigid enough to safeguard the components against the action of different forces. The chassis design includes the selection of suitable shapes and cross-section of chassis-members. Moreover, the design should consider the reinforcement of the chassis side- and cross member joints, and the various methods of fastening them together [3].

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 Figure 2. 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.

Figure 2 Fatigue crack of the sub-frame

The stress analysis is important in fatigue study and life prediction of components to determine the highest stress point commonly known as critical point which initiates to probable failure, this critical point is one of the factors that may cause the fatigue failure. 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 methods to locate the critical point. Safety factor is used to provide a design margin over the theoretical design capacity. This allows consolidation of uncertainties in the design process [3]. Chetan et al. [5] reviews various factors affecting the fatigue life of a structure like cyclic stress state, geometry, surface quality, material type, residual stresses, size and distribution of internal defects, direction of loading & grain size.

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 analyses 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. Experiences from daily use at Volvo Car Corporation, limitations of methods, future and on-going work are discussed.

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 unsprung 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 autobody was evaluated by the nominal stress method with the assumption of a uniaxial stress state, and thus the critical regions were determined. The life of components with critical regions was further investigated on the basis of multiaxial 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 multiaxial 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 Veloso et al. [16]. They discussed the failure investigation and stress analysis of a longitudinal stringer of an automobile chassis Fiat Automóveis, Rod. Fernão Dias, km 429, Betim, MG, Brazil Pontifical Catholic University of Minas Gerais (PUC Minas), 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 truck's sub-frames, for the fatigue crack occurred in the Sub-frame 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. At last an improvement and optimization to the structures of the frame was provided. Also Hengji et al. in 2012 [19] explained the fatigue life for frame of the 220t mining dump truck, a fatigue life analysis method was presented by integrating multi body dynamic analysis and finite element method. The forces of main joints at frame were measured from the multi body dynamic model, whose road was restructured. The dynamic stress test of the whole truck was implemented to obtain the peak stress of the mainly forced area, which was compared with the simulated stress. It was found out that the error was allowable so that the accuracy of the finite element model was definitely ensured. The quasi-static stress analysis method was employed to acquire stress influence coefficient under unit load, which was associated with load histories of the frame to get the dangerous stress area. The fatigue life of the frame was calculated on the basis of Palmgren–Miner damage theory. It was turned out that the minimum life area of the frame is located at the frame joints of suspension, which matches the practice. More recently, Bhat et al. in 2014 [20], redesigned a modified chassis for tractor trolley. The existing trolley chassis designed by industry uses 'C' Cross section having dimension 200mm x 75mm x 7mm and the material used was mild steel. By keeping the material and dimension similar and using 'I' cross section area instead of 'C' resulted in more safer stresses than 'C' and 31.79kg reduction in weight. They concluded that the Reduction in weight shows that raw material required for manufacturing of the Chassis was reduced. Also, they obtained safer stresses in new suggested design and increase in factor of safety obtained in new suggested design.

IV. CONCLUSION

Fatigue analysis of the track chassis has been the focus of a number of previous works. The review of 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. This 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 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.

[11] 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.
[12] J. Hoffmeyer, Doring, R.; Seeger, T.; Vormwald, M.: Deformation behaviour, short crack growth and fatigue lives ¨under multiaxial non proportional loading. Int. J. Fat. , 28, 2006, 508 – 520.
[13] N. Ye, and T. Moan, Static and Fatigue Analysis of Three Types of Aluminium Box-Stiffener/Web Frame Connections, International Journal of Fatigue, 2007, 29: 1426-1433.
[14] 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.
[15] 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.
[16] V, VelosoMagalhaes H S, Bicalho G Iand Palma E S "Failure Investigation and Stres Analysis of a Longitudinal Stringer of an Automobile Chasis", Enginering Failure Analysis, 2009, Vol. 16, pp. 1696-1702.
[17] 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.
[18] C. Yanhong, Zhu Feng "The Finite Element Analysis and The Optimization Design of the Yj3128-type Dump Truck's Sub-Frames Based on ANSYS" Procedia Earth and Planetary Science, 2011, 133 – 138.
[19] M. Hengji, ZhengqiGu ,Qingquan Yang, DuzhongNie,. Et al, "Frame fatigue life assessment of a mining dump truck based on finite element method and multi body dynamic analysis", Engineering Failure Analysis, 2012, Vol. 23 pp. 1-90.
[20] K. Bhat, UntawaleSp, KatoreHv "Failure Analysis and Optimization of Tractor Trolley Chassis: An Approach Using Finite Element Analysis" International Journal of Pure and Applied Research in Engineering and Technology, 2014; volume 2 (12): 71-84.