Journal of Mechanical Design and Vibration, 2014, Vol. 2, No. 3, 69-73
Available online at http://pubs.sciepub.com/jmdv/2/3/3

DOI:10.12691/jmdv-2-3-3

Applications of Finite Element Stress Analysis of Heavy Truck Chassis: Survey and Recent Development

Nouby M. Ghazaly*

Mechanical Engineering Department, Faculty of Engineering, South Valley University, Qena-83523, Egypt
*Corresponding author: nouby.ghazaly@eng.svu.edu.eg

Received August 19, 2014; Revised August 28, 2014; Accepted August 31, 2014

Abstract Nowadays, transportation industry plays a major role in the economy of modern industrialized and developing countries. The goods and materials carried through heavy trucks are dramatically increasing. There are many aspects to consider when designing a heavy trucks chassis, including component packaging, material selection, strength, stiffness and weight. This paper reviews the most important research works, technical journal and conferences papers that have been published in the last thirteen year period (2002-2014). The paper focused on stress analysis of the heavy truck chassis using four finite element packages namely; ABAQUS, ANSYS, NASTRAN and HYPERVIEW. The results of reading this paper will give the researcher a summary of some recent and current developments in the field of vehicle design using finite element packages.

Keywords: *stress analysis, finite element analysis, truck chassis*

Cite This Article: Nouby M. Ghazaly, "Mechanical Engineering Dept., Faculty of Engineering, South Valley University, Qena-83523, Egypt." *Journal of Mechanical Design and Vibration*, vol. 2, no. 3 (2014): 69-73. doi: 10.12691/jmdv-2-3-3.

1. Introduction

Chassis is one of the important parts that used in automotive industry. This structure is the bigger component in the vehicle as shown in Figure 1. The chassis of trucks is tasked to hold all the components systems together such as the axles, suspension, power train, brake system, cab and trailer etc., while driving and transferring vertical and lateral loads caused by accelerations [1]. The frame and other components not only carry the weight of the vehicle, but its payload as well [2:4]. The chassis structure must safely support the weight of the vehicle components and transmit loads that result from longitudinal, lateral, and vertical accelerations that are experienced in a racing environment without failure. There are many aspects to consider when designing a chassis, including component packaging (including the individual systems), material selection, strength, stiffness and weight [5].

The most significant issue in the truck manufacturing industries is design of vehicles with more pay load. Using higher strength steels than the conventional ones are possible with corresponding increase in pay load capacity. Along with strength, an important consideration in 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 the chassis [6].

An important aspect of chassis design and analysis is the stress distribution and fatigue life of prediction process. Automotive designers need to have complete understanding of various stresses prevalent in different areas of the chassis component. 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 [7].

Figure 1. Chassis of heavy duty truck [22]

Recently, in quite a few of published papers, there are amount technical papers and some other sources which have been showed gradually upward trend. This overview selectively and briefly discusses some of the recent and current developments of the stress analysis of truck chassis. Different finite element packages have been used for predicting and investigating the stress analysis.

2. Finite Element Analysis of Truck Chassis

The Finite Element Analysis (FEA) is becoming increasingly popular among design engineers using it as one of many product design tools. It has been used for the solution of many types of problems. FEA has become an integral part of design process in automotive, aviation, civil construction and various consumer and industrial goods industries, cut throat competition in the market puts tremendous pressure on the corporations to launch reasonably priced products in short time, making them to rely more on virtual tools (CAD/CAE) accelerate the design and development of products [8]. FEA is used to predict multiple types of static and dynamic structural responses. For example, companies in the automotive industry use it to predict, stress, strain, deformations, and failure of many different types of components. FEA reduces the need for costly experiments and allows engineers to optimize parts before they are built and implemented. The wide and most common commercial finite element packages are (ANSYS, ABAQUS, COMSOL, ALGOR, HYPERVIEW, LS DYNA, NASTRAN etc.). The advent of faster computers and robust FEA software allows design engineers to build larger, more refined and complex models resulting in timely, cost-effective, accurate, and informative solutions to customer problems [9]. In the following sections, four finite element packages namely; ABAQUS, ANSYS, NASTRAN and HYPERVIEW will be discussed in more details.

2.1. ABAQUS Stress Analysis of Truck Chassis

ABAQUS consists of four products: Abaqus/CAE, Abaqus/Standard, Abaqus/Explicit and Abaqus/CFD. Abaqus/CAE is a customizable user interface used for designing, modeling, pre- processing and post-processing and visualizing the FEA result. Abaqus/Standard employs solution technology ideal for static and low-speed dynamic events. Abaqus/Explicit uses an explicit integration method to solve high-speed, highly nonlinear and transient response events. Abaqus/CFD is a new application for Abaqus 6.10 which provides advanced computational fluid dynamics capabilities [10].

There are amount of research papers has been conducted using ABAQUS software in order to predict the stress analysis of truck chassis. In 2007, Teo and Roslan [11] predicted the stress analysis of heavy duty truck chassis using ABAQUS software. Also in 2008, Okurdi et al. 2008 [12] performed stress analysis using ABAQUS to locate the critical point which has the highest stress. This 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. To determine critical point so that by design modifications the stresses can be reduces to improve the fatigue life of components, Roslan Abd Rahman et al. 2008 [13] conducted stress analysis of heavy duty truck chassis using ABAQUS. They used ASTM Low Alloy Steel A 710 C material which has 552 MPa of yield strength and 620 MPa of tensile strength. The simulation result showed that the critical point of stress occurred at the opening of chassis which is in contact with the bolt. The stress magnitude of critical point was 386.9 MPa. They reported that the critical point is an initial to probable failure since fatigue failure started from the highest stress point.

Recently, Grzegorz et al. 2011 [14] presented stress analysis on a frame of semi low Loader using Abaqus/Standard software. Two versions of frame design were analyzed. One of them was used in a 4 axle trailer with a load deck that can be extended in length and width to carry oversize and heavy loads. A design solution for a ladder frame, a schematic image of a box-type central beam, a design solution for rear ramps, geometrical models of semitrailer frame with boundary conditions, stress distributions in a non-extended and extended frame model, a section of the rear part of the frame with sliding plates visible and a pin guide bar for blocking frame extension, stress distribution in the front part of the frame, single and double supporting rib, side wall supporting plate and stress distribution after the use of plates and one supporting rib, stress distribution after the employment of plates and two supporting ribs, stress distribution in the alternative gooseneck design, main dimensions of the semitrailer, extended semitrailer are presented. It seems that the analysis made it possible to avoid design errors which could result in damage to the semitrailer during operation.

A numerical technique for the static analysis of preloaded bolted joints using the software ABAQUS/Standard and a mesh modelling with solid type elements was developed by Valladares et al. 2013 [15]. Parting from a new design with bolted joints for the longitudinal beams of an innovative modular semitrailer, the different problems associated to this type of analysis were stated first. Then, the numerical technique developed has been described and all its considerations have been applied to that specific semitrailer bolted joints. Due to high values of Von Mises stress reached at the flanges and at several bolt studs, it has been considered necessary to change some dimensions and materials in the proposed design. This would improve the behaviour of the bolted joints. More recently, Mahmoodi et al. 2014 [16] discussed the stress and dynamic analysis of truck ladder chassis. At the first stage, in order to design a chassis for self-weight reduction, material type and cross section profiles of chassis were selected according to a maximum normal stress and maximum strain theories. Then, the stress analysis of truck chassis has been carried out by ABAQUS software to determine maximum transverse deflection and stress distribution. The stress/strain distribution was calculated along the chassis. Maximum stress and strain levels are found in front section of chassis, where engine and transmission are installed. Moreover, Results showed that open U-shaped profiles are sufficient for weight reduction which can endure loads.

2.2. ANSYS Stress Analysis of Truck Chassis

The ANSYS program is a powerful, multi-purpose analysis tool that can be used in a wide variety of engineering disciplines. Using ANSYS software can avoid expensive and time-consuming development loops, so the design period is shortened [17]. The stress analysis of a truck chassis using ANSYS software carried out by many researchers to reduce the magnitude of stress of the chassis frame. For example; Karaoglu and Kuralay 2002 [18] investigated stress analysis of a truck chassis using the commercial finite element package ANSYS. They examined many geometrical modifications. It was concluded that if the change of the side member thickness using local plates is not possible, due to increase weight of chassis then choosing an optimum connection plate length seems to be best practical solutions for decreasing the stress values. Chinnaraj et al 2008 [19] explained current trend in automotive design to optimize components for weight reduction. To achieve this, the chassis frame assembly of a heavy truck used for long distance goods hauling application was chosen for investigation. A quasi-static approach that approximates the dynamic maneuvers into number of small processes having static equilibriums was followed to carry out the numerical simulation, approximating the dynamic behavior of frame assembly. With the help of commercial finite element package ANSYS, the quasi-static numerical simulations were carried out and compared with experimental results.

Static analysis for the chassis considering sudden load effects to modify a 8 ton 4 wheeler trailer which ultimately results in reduction of weight and manufacturing cost was carried out by [20]. PRO-E was used for modeling the chassis of tractor trailer. ANSYS software was used for analysis and simulation. They did some modifications in model of chassis namely; variation in cross sectional areas of cross and longitudinal members and changing the position of cross members of main frames of chassis. It has been found that the maximum stress in longitudinal member is 75 Mpa. Also, they reported that if sudden loads effects are considered then stress level may rise to approximately to twice that will be approximately 150 Mpa.

The frame of the standard dump truck supports all types of complicated loads coming from the road and freight being loaded was discussed by Haval et al. 2012 [21]. A frame of 6 wheels, standard dump truck has been studied and analyzed using ANSYS software. The static intensity of the frame has been analyzed when exposed to pure bending and torsion stress, within two cases. First case was when the rear wheels zigzag gets over block and the second case was when both wheels get over the block. The simulation results showed that the critical point of stress occurred when the truck zigzag ramp the block. The big effect was given to the case of zigzag wheels of the dump truck ramp the block because there was great difference in the torsion stress values in both two case studies. Recently in 2013, Hemant et al. [22] presented stress analysis of a ladder type low loader truck chassis structure consisting of C-beams design using CATIA V5R10 and ANSYS. Computed results were compared to analytical calculation, where it was found that the maximum deflection agrees well with theoretical approximation but varies on the magnitude aspect.

Optimization of the automotive chassis with constraints of maximum shear stress, equivalent stress and deflection of chassis was performed by Hirak Patel et al 2013 [23]. A sensitivity analysis was carried out for weight reduction. So a proper finite element model of the chassis was to be developed. The chassis was modeled in PRO-E. FEA was done on the modeled chassis using the ANSYS Workbench. The truck chassis design was done analytically and the weight optimization was done by sensitivity analysis. In sensitivity analysis different cross section are used for stress analysis and it was find a 17% weight reduction in the truck chassis. The stress and deformation were also compared for the different cross section. Darshit et al. 2014 [24], presented the static stress analysis that acting on the upper surface of the truck chassis. Critical parts that will lead to failure were also observed. 3-D finite element model of the truck chassis was made using ProE before analyzed through ANSYS software. Numerical results showed that critical part was at the mounting bracket of the tire and also at the front part of the chassis. Some modifications were also suggested to reduce the stress and to improve the strength of the truck chassis. Finally optimum section was suggested by authors.

2.3. NASTRAN Stress Analysis of Truck Chassis

MSC Nastran is a multidisciplinary structural analysis application use to perform static, dynamic, and thermal analysis across the linear and nonlinear domains, complemented with automated structural optimization and award winning embedded fatigue analysis technologies, all enabled by high performance computing [25]. MSC Nastran is also used to improve the economy and passenger comfort of structural designs. MSC Nastran is based on sophisticated numerical methods, the most prominent being the Finite Element Method. However, when a developed model employs a new structure, it is difficult to precisely determine the conditions that are critical for main frames on development stage [26].

Attempts have been made by Masahiro et al. 2004 [27] predicted the strength of main frames of dump truck based on static stress analysis where dynamic loads obtained by measurement with actual machines are replaced with static loads. For the articulated dump truck that was developed recently, it was required to precisely predict the stresses that act on the frames during travel. Therefore, they introduced elastic characteristic of main frame into kinematical analysis models by kinematical analysis software ADAMS and finite element method software NASTRAN in order to calculate frame stress that occurs during travel, which was applied to the rear frame of the articulated dump truck. To reduce the weight of the bracket of chassis and also to achieve safe stress value various modifications were carried out by Balbirsingh et al 2013 [28]. They investigated the critical points of stresses that lead to or induce failure. The finite element packages MSC/NASTRAN with PATRAN were used for modeling and stress analysis. PATRAN FE Pre-post processor was used to model the cross member bracket and included the part modeling environment. The parameters required for the modeling of bracket were contour dimensions height, length, fillet radius, hole diameter. The 3D SOLID was prepared by thickness in the third dimension provided after selecting the 2-D shell element. The bracket model geometry was imported in NASTRAN for further analysis. In design simulation, modifications have been made to current bracket and these modifications have resulted in reduction in stress values leading to safe design. Stress analysis of existing cross member bracket of the heavy truck chassis reveals that the stress level was up to

285MPa at the holes of the central plate which was fixed with the chassis. The remaining flange part of the bracket was under lesser stress levels.

The importance of theoretical bending stress calculation of truck frame side member (FSM) subjected to vertical load which will help the designer to arrive at the major section of FSM during initial stage of the design was explained by Sathish Kumar and Balakrishnan in 2013 [29]. The numerical analysis of truck chassis assembly was applied using commercial software MSC/Nastran. Different concept proposals of FSM were evaluated using these calculations quickly. Based on the experience in validation of FEA results of similar type of vehicles with test results, the calculated theoretical bending stresses were compared with those obtained from numerical analysis. The finite element model of the concrete mixing truck's frame was built by Hai Xia Sun et al., 2014 [30] using shell as basic element, and the process of building the finite element model of the balance suspension was introduced. The frame's stress on five types of typical operating conditions was calculated using NASTRAN. Simulation results showed the dangerous position and the maximum stress position on the frame. The analysis result on structural strength provided the basis for further improving the frame structure.

2.4. HYPERVIEW Stress Analysis of Truck Chassis

Altair HyperView is a complete post-processing and visualization environment for finite element analysis, multi-body system simulation and engineering data. HyperView combines advanced animation and XY plotting features with window synching to enhance results visualization. HyperView also saves 3D animation results in Altair's compact H3D format, so users can visualize and share CAE results within a 3D web environment using Altair HyperView Player [31]. The FEA post processing using HYPERVIEW for evaluation of the stress and displacement of the chassis has been conducted by a few researchers. To determine the maximum stress, maximum deflection and to recognize critical regions under static loading condition that are important criteria for design of the chassis, Madhu and Venugopal 2014 [32] carried out static analysis of the chassis frame by FEA. The structural chassis frame was modeled using PRO-E wildfire 4.0 software. The Pre-processing has done with HYPERMESH software; then the problem has been solved through RADIOSS and the post processing was done by HYPERVIEW. The results indicated that the von-mises stress below yield strength of the material, which satisfies the design and deflection was about 5.648 mm. Modification of design was carried out to reduce deflection of structure. Deflection of the modified structure was 3.706 mm. Therefore chassis frame deflection has been reduced from 5.921 mm to 3.706 mm by modification.

Structural analysis of vehicle chassis with constraints of maximum shear stress and deflection of chassis under maximum load through using Pro-e 4.0 and Altair Hyperworks software was performed by Abhishek et al. 2014 [33]. The dimensions of vehicle chassis of a TATA LP 912 Diesel BS4 bus was taken for analysis with materials namely Steel alloy subjected to the same load. The four different vehicle chassis have been modeled by considering four different cross-sections. Namely C, I, Rectangular Box (Hollow) and Rectangular Box (Intermediate) type cross sections. From the results, it was observed that the Rectangular Box (Intermediate) section is more strength full than the conventional steel alloy chassis with C, I and Rectangular Box (Hollow) section design specifications. The Rectangular Box section was having least deflection in all the four type of chassis of different cross section.

3. Conclusion

Several state of the art papers and even books on stress analysis of the heavy truck chassis through finite element analysis have been presented in the literature. As mentioned above, this paper selectively and briefly discusses some of recently developments on stress analysis of the truck chassis using four finite element packages namely; ABAQUS, ANSYS, NASTRAN and HYPERVIEW. It is observed that most common basic FEA packages are suitable for this analysis. Also, it is found that Most of the existing researchers utilized ABAQUS and ANSYS software to predict stress analysis of the chassis. On the other hand, a few studies using NASTRAN and HYPERVIEW are conducted.

References

[1] K. P. Sirisha, R. Lalith Narayana, A. Gopichand, Ch. Srinivas, G. Ram Balaji Structural and Modal Analysis on A Frame Less Chassis Construction of Heavy Vehicle for Variable Loads" Journal of Engineering Research and Applications, Vol. 3, Issue 4, Jul-Aug 2013, pp. 2318-2323.

[2] M. Ravi Chandra, S. Sreenivasulu, Syed Altaf Hussain, "Modeling and Structural analysis of heavy vehicle chassis made of polymeric composite material by three different cross sections" International Journal of Modern Engineering Research (IJMER), Vol. 2, Issue. 4, 2012 pp-2594-2600.

[3] Mohd Azizi Muhammad Nora, b*, Helmi Rashida, Wan Mohd Faizul Wan Mahyuddinb, Mohd Azuan Mohd Azlanc, Jamaluddin Mahmud "Stress Analysis of a Low Loader Chassis" Procedia Engineering 41 (2012) 995-1001.

[4] Sankararao Vinjavarapu, Unnam Koteswararao, V. Lakshmi Narayana "Design Optimization of Tipper Truck Body" International Journal of Engineering Research and Development, Volume 4, Issue 9 (November 2012), PP. 11-20.

[5] Gauchia, A., Diaz, V., Boada, M.J.L., Boada, B.L. (2010). Torsional stiffness and weight optimization of a real bus structure. International Journal of Automotive Technology, vol. 11, p. 41-47.

[6] Tushar M. Patel and M. G. Bhatt Analysis and Design Modification of a Chassis" LAP Lambert Academic Publishing, 2012.

[7] R. Rajappan, M. Vivekanandhan "Static And Model Analysis Of Chassis By Using FEA" Proceedings of the "National Conference on Emerging Trends In Mechanical Engineering 2013.

[8] Practical Finite Element Analysis (Nitin S. Gokhale 2009).

[9] Siraj Mohammad and Ali Sheikh "Analysis of universal coupling under different torque Condition" International Journal of Engineering Science & Advanced Technology, Volume-2, Issue-3, 690-694.

[10] Abaqus Analysis User's Manual, vol 2-Abaqus 6.10.

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

[12] G. Murail, B. Subramanyam and D. vaveen "Design Improvement of a Truck Chassis based on Thickness" Altair Technology conference, India, 2013.

[13] 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.

[14] 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.

[15] D. Valladares, M. Carrera, L. Castejon, C. Martin "Development of a Numerical Technique for the Static Analysis of Bolted Joints by the FEM" Proceedings of the World Congress on Engineering 2013 Vol III, WCE 2013, July 3-5, 2013, London, U.K.

[16] Mehdi Mahmoodi-k, Iraj Davoodabadi, Vinko Višnjić, Amir Afkar "Stress And Dynamic Analysis Of Optimized Trailer Chassis" Tehnički vjesnik 21, 3 (2014), 599-608

[17] Ji-xin Wang, Guo-qiang Wang, Shi-kui Luo, Dec-heng Zhou "Static and Dynamic Strength Analysis on Rear Axle of Small Payload Off-highway Dump Trucks" Int-ANSYS-Conf. 2004.

[18] C. Karaoglu, N. Sefa Kuralay "Stress analysis of a truck chassis with riveted joints" Elsevier Science B.V Finite Elements in Analysis and Design. Vol 38, pp. 1115-1130, 2002.

[19] 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.

[20] 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.

[21] 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

[22] 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.

[23] Hirak 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.

[24] Darshit Nayak, Dr. Pushpendra 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.

[25] NX Nastran Basic Dynamic Analysis User's Guide, 2003.

[26] MD Nastran Multidiscipline Simulation System for Advanced Engineering Analysis, 2008.

[27] Masahiro Koike, Sanshirou Shimoda, Toshihide Shibuya, Hirofumi Miwa "Development of Kinematical Analysis Method for Vehicle". Komatsu technical report, 2004, VOL. 50 NO. 153.

[28] Balbirsingh R. Guron1, Dr. D.V. Bhope 2, Prof. Y. L. Yenarkar "Finite Element Analysis of Cross Member Bracket of Truck Chassis" IOSR Journal of Engineering, Vol. 3, Issue 3 (Mar. 2013), PP 10-16.

[29] Sathish Kumar P and Balakrishnan M" Theoretical Evaluation and Finite Element Analysis of Commercial Truck Chassis Assembly," SAE Technical Paper 2013-01-1361, 2013.

[30] Hai Xia Sun, Hua Kai Wei, Xiao Fang Zhao, Jia Rui Qi "Finite Element Analysis of Structural Strength of Concrete Mixing Truck's Frame" 2014, Advanced Materials Research, Volumes 945-949, pp 1143-1149. 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.

[31] HyperWorks 12.0 Student Edition-Overview, 2014.

[32] Madhu Ps 1 and Venugopal T R "Static Analysis, Design Modification and Modal Analysis of Structural Chassis Frame" Int. Journal of Engineering Research and Applications, Vol. 4, Issue 5, May 2014, pp. 06-10.

[33] Abhishek Singh, Vishal Soni, Aditya Singh "Structural Analysis of Ladder Chassis for Higher Strength" International Journal of Emerging Technology and Advanced Engineering. Volume 4, Issue 2, February 2014).