International Journal of Modern Trends in Engineering and Research
www.ijmter.com
e-ISSN No.:2349-9745, Date: 2-4 July, 2015

FINITE ELEMENT STATIC STRUCTURAL ANALYSIS OF 4X2 TRUCK CHASSIS FRAME

Mahvi Malik Shahzad[1], ShaikhRizwan[2], Tarique khan[3]

[1] *(Prof. B.E. Mechanical 9 years of design experience in industry and 4 years of teaching)*
[2]*(Prof &M.E Student SND College of Engineering Yeola)*
[3]*(Prof &M.E Student IBSS College of Engineering Amravati*)

mahvimalikshahzad@gmail.com*1, rizwankshaikh7@gmail.com*2, khantarikanwar@gmail.com

Abstract- Finite element structural analysis of a 4X2 truck chassis plays an important role during design stages. Nowadays, there are many research papers and development programs available in the market especially by the truck manufacturers, which are very much related to this research work. The chassis consist of frame, transmission system, axle, Auxiliary power unit(APU) etc. The paper focused on structural analysis of the truck chassis using finite element package Pro/Mechanica. 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 stress analysis

Keywords- *Pro/E, Pro/Mechanica,*

I. INTRODUCTION

Transportation industry plays a major role in the economy of modern industrialized and developing countries. The total and relative volume of goods carried on heavy trucks is dramatically increasing. The chassis frame 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 racingenvironment without failure. There are many aspects to consider when designing a chassis, including component packaging, materialselection, strength, stiffness and weight. The primary objective of the chassis is to provide a structure that connects the front and rear suspension without excessive deflection. The design of a vehicle structure is of fundamental importance to the overall vehicle performance. The vehicle structure plays an important role in the functionality of the vehicle. Generally, truck is any of various heavy motor vehicles designed for carrying the attached loads, such as the engine, transmission and suspension as well as the passengers and payload. The major focus in the truck manufacturing industries is design of vehicles with more pay load. Using higher strength steels than the conventional once are possible with corresponding increase in pay load capacity. 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., as shown in Figure1, is one of the possible candidates for substantial weight reduction [1].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 [2,3]. Stress-strain relations used to describe deformation of a material are different for the elastic and plastic domain. Consequently, it is important to know if the stress state is in the elastic or plastic domain. For this purpose a yield criterion is used to suggest the limit of elasticity and the initiation of yielding in a material under any combination of stresses. There are several yield criterion used in practice. Some of these are: the maximum shear stress criterion, the maximum principal stress criterion and the von Mises stress criterion. These criteria could be expressed in terms of material constants obtained from different physical tests e.g. a shear or uniaxial tensile test.Automotive designers need to have complete understanding of various stresses prevalent in different areas of the chassis component. During the conceptual design stage, when changes to the design are easiest to implement and have lower impact on overall project cost, the weight and structural characteristics are mostly unknown since detailed vehicle information is unavailable at this early stage [4, 5]. The vehicle design starts up with conceptual studies to define size, number and location of undriven (dead axle) and drive (live axle) axles, type of suspension, engine power, transmission, tire size and axle reduction ratio, cab size and auxiliary equipment. The selected configuration has to be suitable for the considered transportation tasks and should match the existing production line [6].

In general, there are two approaches to simulate truck chassis using FEA methods: one is stress analysis to predict the weak points and the other is fatigue analysis to predict life of the frame. 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 modal analysis of truck chassis. A number of analytical, numerical and experimental techniques are considered for the modal analysis of the 4X2 truck frames. Conclusion of the modal analysis in the vehicle chassis has been reported in literature. Finally, the scope of future work has been discussed after concluding on the obtained in this paper.

Fig 1:

II. FINITE ELEMENT STRUCTURAL ANALYSIS OF TRUCK CHASSIS

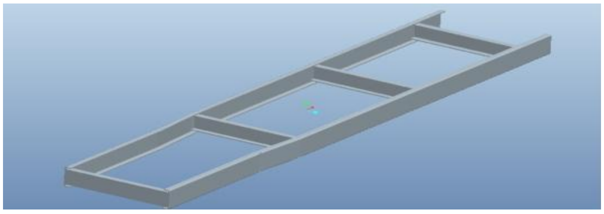
Finite element analysis (FEA) is one of the efficient and well-known numerical methods for various engineering problems. FEA was first developed in 1943 by R. Courant, who utilized the Ritz method of numerical analysis and minimization of variation calculus to obtain approximate solutions to vibration systems as cited by Swatantra and Pradeep [2]. For the last 30 years 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. FEA tools are being used to analyse multi-disciplinary problems, including but not limited to structural, thermal and fluid flow, NVH applications, biotechnology etc. [7]. 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. There are many software packages available to industries that use finite element analysis and computer aided engineering. The wide and universal propagation of commercial finite element packages (ANSYS, Pro/Mechanica, ABAQUS, HYPERMESH, LS DYNA, NASTRAN,ANSYS etc.) for computations in design of mechanical structures made possible to define more accurately the modal analysis of truck chassis. 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 [8]. In the following section, finite element package ANSYS will be discussed in more details.

III. Pro/Mechanica STRUCTURAL ANALYSIS OF TRUCK CHASSIS

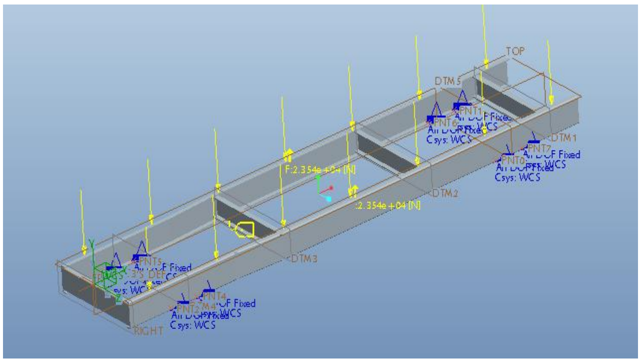
The software package Pro/Mechanica used as a FEA tool in the structural analysis. The Pro/Mechanica program is a powerful, multi-purpose analysis tool that can be used in a wide variety of engineering disciplines [9]. Using Pro/Mechanica software can avoid expensive and time-consuming development loops, so the design period is shortened [10]. The structural analysis of a truck chassis using Pro/Mechanica software carried out by many researchers to reduce the magnitude of stress of the chassis frame. They examined the effect of the geometrical modification through varying the side member thickness from 8 to 12 mm, and the thickness of the connection plate from 8 to 12 mm by local plate, the connection plate thickness from 7 to 10 mm, and the length of the connection plate (L) from 390 to 430 mm. They reported 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 (L) seems to be best practical solutions for decreasing the stress values.In order to understand the dominating stresses in truck frame especially during cornering and braking maneuvers and brings out all geometric locations that may be potential failure initiation locations, Chinnarajet al. [12] 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 finite element package Pro/mechanica, the quasi-static numerical simulations were carried out and compared with experimental results.

IV. Pro/Mechanica Modeling and STRUCTURAL ANALYSIS OF TRUCK CHASSIS

The software package Pro/Mechanica used as a FEA tool in the structural analysis. The Pro/mechanica program is a powerful, multi-purpose analysis tool that can be used in a wide variety of engineering disciplines. Using Pro/mechanica software can avoid expensive and time-consuming development loops, so the design period is shortened. The modeling of chassis frame for FEA analysis is done by using Pro/e software as shown in figure below. The chassis frame has two side members and four cross members out four two are end cross members.



The structural analysis of a truck chassis frame is carried out by taking the 3D model in Pro/Mechanica, The first step is to assign the material we have assign the rolled steel material, after assigning the material next step is to apply the boundary conditions, as the chassis is 4X2 truck hence constraints are given at eight points where suspensions hinge point is located which is 21 inch and 26 inch from the front and rear frame end respectively. As the capacity of truck is 2 tons. The load applied is 2 tons+engine, transmission system, cab assembly etc. loads of truck as shown in figure below.

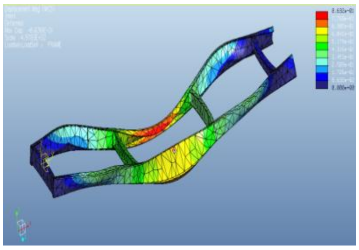


The boundary conditions are applied the next step is analysis, One of specialty of pro/Mechanica is that during analysis software itself will do the meshing of 3D model by using the solid element type. During the analysis software generated the mesh which is having the 4003 solid elements as shown in figure below.

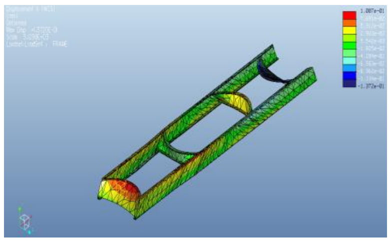
```
Principal System of Units: millimeter Newton Second (mmNs)

Length:          mm
Force:           N
Time:            sec
Temperature:     C

Model Type: Three Dimensional

Points:               1592
Edges:                7193
Faces:                9602

Springs:                 0
Masses:                  0
Beams:                   0
Shells:                  0
Solids:               4003

Elements:             4003
```

The results of structural analysis are shown below.

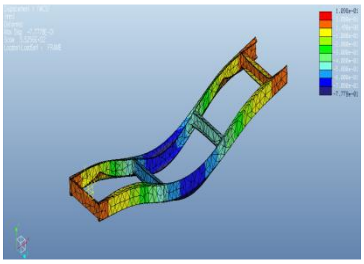
RESULT



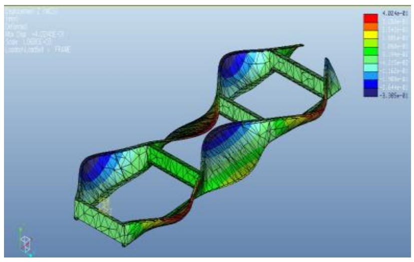
Overall Displacement



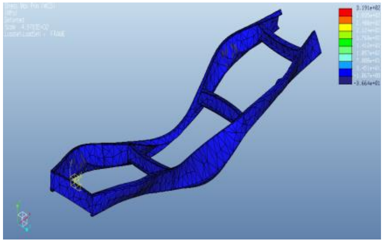
Displacement in X direction



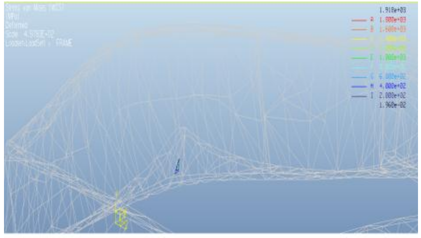
Displacement in Y direction



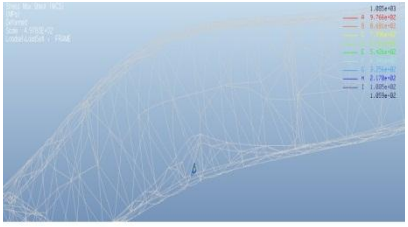
Displacement in Z direction



Principle stress



Von misses stress at SUSPENSION



Maximum shear stress

CONCLUSIONS

Structural analysis using Pro/Mechanica software is done. It is found that the maximum deflection is at the rear side of frame and maximum principle stress, von misses stress and maximum shear stress is at the hinge point of suspension 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 work makes a case for further investigation on the design of truck chassis using FEA Pro/Mechanica software.

REFERENCES

[1] Keith J. Wakeham,Introduction to chassis design, Memorial University of Newfoundland And Labrador.
[2] S. Dey, R. Chandra Murthy. D., P. Baskar "Structural Analysis of Front axle beam of a Light Commercial Commercial Vehicle (LCV) International Journal of Engineering Trends and Technology (IJETT) – Volume 11 Number 5 - May 2014.
[3] C. Karaoglu, N. SefaKuralay "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.
[4] R. L. Patel, K. R. Gawande, D. B. Morabiya "Design and analysis of chassis frame of TATA 2516TC" International Journal for Research in Applied Science and Engineering Technology , Vol. 2 Issue III, March 2014
[5] R. Rajappan and M. Vivekanandhan"Static And Model Analysis Of Chassis By Using FEA" Proceedings of the "National Conference on Emerging Trends In Mechanical Engineering 2013.
[6] B. HemantPatil and D. Sharad Kachave "Effects Of Web And Flange Thickness On Bending Stiffness Of Chassis" International Journal Of Advance Research In Science And Engineering, Vol. No.2, Issue No.7, July, 2013
[7] Peeters, B. and De Roeck,G., Reference-based Stochastic Subspace Identification for Output only
[8] Vold, H. et. al., A Multi-Input Modal Estimation Algorithm for Mini-Computers, SAE Paper No. 820194, 1982
[9] Kr. SwatantraPorwal and Kr.Pradeep. 'Stress Analysis of a Structural Member Using Finite Element Method "VSRD International Journal of Mechanical, Auto. & Prod. Engg, Vol. 1 (3), 104-109, 2011.
[10] Siraj Mohammad Ali Sheikh" Analysis Of Universal Coupling Under Different Torque Condition" International Journal Of Engineering Science & Advanced Technology, Volume-2, Issue-3, 690 – 694, 2013.
[11] 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.

[12] 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

[13] 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.

[14] Vold, H., Crowley, J., Rocklin, G.: A comparison of H1, H2 and HV frequency response functions. Proceedings, IMAC, 1985.

