Static And Dynamic Stress Analysis Of Auto Chassis And Crank Shaft Using ANSYS

[1]K.S.J PRAKASH,[2]A.SRIKANTH
[1](Mechanical Engineering, Asst. Prof RISE Groups, Ongole, India, suvarna.Prakash2@gmail.com)
[1](Mechanical Engineering, Assoc. Prof RISE Groups, Ongole, India, a_srikanth2002@yahoo.co.in)

Abstract— *In recent days due to development in technology Finite element based static and dynamic stress analysis are very popular in many research applications .Apart from that use of FEM for design and modeling of heavy Auto truck chassis and Crank shaft plays an important prominent role particularly in design stage. However there are many research papers and development programs available in the market especially by the Auto truck and crank shaft manufacturers, these are very much related to the current research work. In this proposed paper mainly focused on static as well as dynamic stress analysis of the Auto truck chassis and crank shaft using finite element based ANSYS. For this proposed scheme results of reading this paper will gives the researcher a summary of some recent Innovational and current research developments in the field of design of Automobile Engines using finite element based static and dynamic stress analysis.*
Keywords—*Finite Element Analysis, AutoTruck Chassis, Crank shaft,ANSYS, Stress Analysis,*

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 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, material selection, 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 ones 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 a 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 un driven and drive 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 stress analysis of truck chassis. A number of analytical, numerical and experimental techniques are considered for the stress analysis of the heavy duty truck frames. Conclusion of the stress analysis in the vehicle chassis has been reported in literature.

Fig.1. Assembled Auto Truck chassis

II.FINITE ELEMENT ANALYSIS OF AUTO 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 analyze multi-disciplinary problems, including but not limited to structures, 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, ABAQUS, COMSOL, ALGOR, HYPERVIEW, LS DYNA, NASTRAN etc) for computations in design of mechanical structures made possible to define more accurately the stress 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. ANSYS STRESS ANALYSIS OF TRUCK CHASSIS

The commercial software package ANSYS used as a FEA tool in the stress analysis. The ANSYS program is a powerful, multi-purpose analysis tool that can be used in a wide variety of engineering disciplines [9]. Using ANSYS software can avoid expensive and time-consuming development loops, so the design period is shortened [10]. 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 in 2002[11], investigated stress analysis of a truck chassis using the commercial finite element package ANSYS. 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, Chinnaraj et 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 commercial finite element package ANSYS, the quasi-static numerical simulations were carried out and compared with experimental results. In 2011, Ingole and Bhope [13] carried out 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. 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. Recently in 2012, Haval Kamal et al. [14] discussed the frame of the standard dump truck supports all types of complicated loads coming from the road and freight being loaded. 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. Weight reduction is now the main issue in automobile industries, Ravi Chandra et al. [15], described analysis of heavy vehicle chassis for weight reduction using PRO–E software for modeling, ANSYS 12.0 for analysis. The dimensions of the heavy vehicle chassis of a TATA 2515EX vehicle was taken for modeling and analysis of a heavy vehicle chassis with three different composite materials namely, Carbon/Epoxy,E-glass/Epoxy and S-glass /Epoxy subjected to the same pressure as that of a steel chassis.

The design constraints were stresses and deflections. The three different composite heavy vehicle chassis have been modeled by considering three different cross-sections. Namely C, I and Box type cross sections. For validation, the design was done by applying the vertical loads acting on the horizontal different cross sections. From the results, it was observed that the polymeric composite heavy vehicle chassis is lighter and more economical than the conventional steel chassis with similar design specifications. Also, PAUL et al. in 2012 [16] analyzed the ladder frame for static load condition with the stress, deflection bending moment on truck chassis frames. The truck chassis was modeled in CATIA V5 R16 and then it imported into Ansys. The side members of the chassis was considered as the beam as it help to simulate various attachments over the chassis, like fuel tank mountings, engine mountings, etc. The model is analyzed on the basis of static load condition due to mountings. The stress is obtained more at the joints where the weld are present. The results illuminate the new creative ways for optimum frame design which makes it more sustainable for structural concerns. More recently in 2013, Hemant Patil et al. [17] and Hemant Patil and Sharad Kachave [1] presented stress analysis of a ladder type low loader truck chassis structure consisting of C-beams design using CATIA V5R10 and ANSYS, as shown in Figure 2. In order to achieve a reduction in the magnitude of stress at critical point of the chassis frame, side member thickness, cross member thickness and position of cross member from rear end were varied. Numerical results showed that if the thickness change is not possible, changing the position of cross member may be a good alternative. 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. They concluded that it is better to change the thickness of cross member at critical stress point than changing the thickness of side member and position of chassis for reduction in stress values and deflection of chassis.

Fig.2. Actual C-beams chassis

Optimization of the automotive chassis with constraints of maximum shear stress, equivalent stress and deflection of chassis was performed by Hirak Patel et al. [18]. 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. Also, MANPREET et al [19] presented the static load analysis of the chassis of TATA super ace using ANSYS workbench and stress optimization using standard techniques of vehicle modification, as shown in Figure 3.

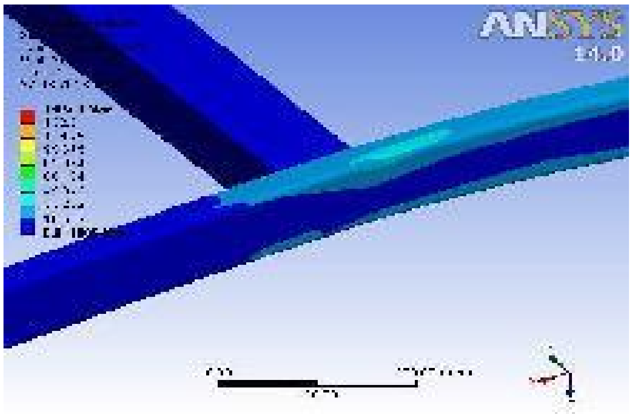


Fig.3. Close view of maximum stress

Darshit Nayak et al. 2014 [20], presented an analysis of the static stress 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 Pro-E before analyzed through ANSYS software. Modal updating of truck chassis model was prepared adjusting the selective properties such as mass density and Poisson’s ratio. 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. More recently some review papers were discussed the fatigue analysis of the frame and the roles of finite element packages on design and analysis the frame [21-22]. Also, Swami and Tuljapure 2014 [23] considered chassis of Eicher 20.16 and the behavior of chassis under same load but varying thickness was studied through ANSYS. They conclusions can be drawn from the same as following:

1. The maximum value of Von Mises Stress is occurred at the Hinged point of at the Right end of the beam near the top edge of Ladder chassis frame.
2. At the free end of the beam, highest deformation has occurred leading to lowest stresses.
3. As the torque decreases, the maximum deformation decreases continuously.
4. As the torque decreases, there is decrease in von mises stress.
5. Deformation & Stresses are directly proportional to load applied.
6. As thickness increases of cross member, there is decrease in von mises stress and deformation and vice versa.

IV. FINITE ELEMENT ANALYSIS OF CRANK SHAFT

For the finite element analysis three different loads i.e., 69kg= 677N, 85kg= 834N, 99kg= 971N are used. . For the analysis triangular plate element with six degrees of freedom per node is considered. The analysis is carried out using *PRO*/E Wildfire 4.0 and ANSYS WORKBENCH 11.0 software. The normal load is applied at the small end of connecting rod keeping big end fixed.

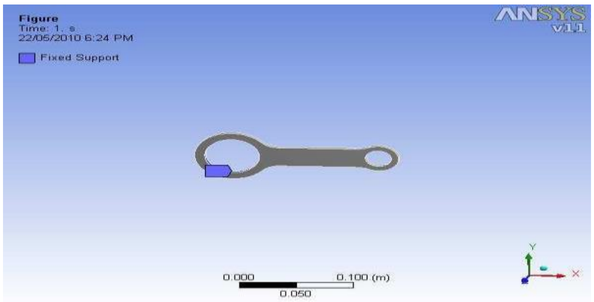


Fig.3. Connecting Rod under Load

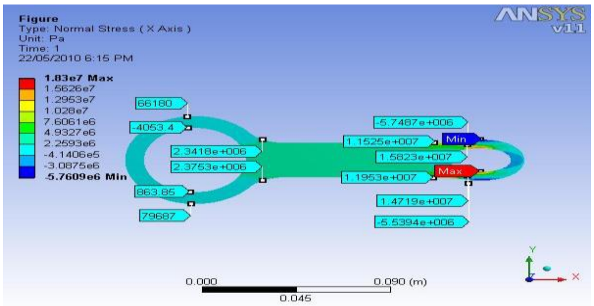
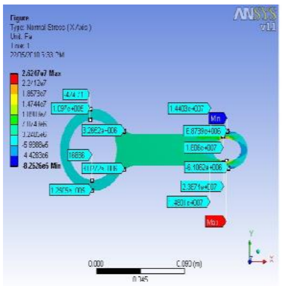


Fig.4. Normal stress at 677N



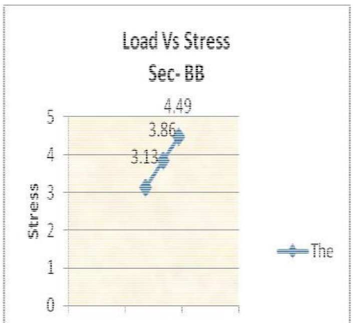


Fig.6 Normal Stress at 971N **Fig.7.Graph between Load & Stress**

Stresses along section C-C

For P = 677N (69kg)
σ = 3.7605N/mm^2
For P = 834N (85kg)
σ = 4.6325N/mm^2
For P = 971N (99kg)
σ = 5.3955N/mm^2

Stresses along section D-D

For P = 677N (69kg)
σ = 17.35N/mm^2
For P = 834N (85kg)
σ = 21.38N/mm^2
For P = 971N (99kg)
σ = 24.90N/mm^2

Stresses along section A-A

*σ = P/(B-D)*T*
For P = 677N (69kg)
σ = 677/(66-48)*6
= 6.2675N/mm^2
For P = 834N (85kg)
σ = 7.720N/mm^2
For P = 971N (99kg)
σ = 8.9925N/mm^2

Stresses along section B-B

For P = 677N (69kg)
σ = 3.13N/mm^2
For P = 834N (85kg)
σ = 3.86N/mm^2
For P = 971N (99kg)
σ = 4.49N/mm^2

V. DISCUSSION AND CONCLUSION

From the theoretical and Finite Element Analysis it is found that

1. The stresses induced in the small end of the connecting rod are greater than the stresses induced at the big end.
2. Therefore, the chances of failure of the connecting rod may be at fillet section of both end.

VI. CONCLUSION

Vehicle structural design and optimization 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 using ANSYS software 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 FEA ANSYS software.

VI.SCOPE OF FUTURE WORK

Future work must focus on life-cycle tests to assess the reliability and longevity of the frame. In addition, the Frame model using ANSYS can further improved and validated using experimental results to correctly predict the stress analysis. Additional efforts are needed to improve the FE model of frame include more details of the cross and long members assembly to account for stress analysis.

REFERENCES

[1] B. Hemant Patil 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.
[2] Kr. Swatantra Porwal 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.
[3]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.
[4]A. Barari, Platanitis, G. Pop-Iliev, R. “Introducing Design Rigidity to Model Unexpected Disturbances In a DSM-Based Design Process” ASME, IDETC-CIE, 2010.
[5]A. R., George, W. B., Riley “Design, Analysis and Testing of a Formula SAE Car Chassis, SAE Motorsports Engineering Conference and Exhibition, 2002.
[6]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.
[7]Nitin S. Gokhale " Practical Finite Element Analysis, 2009.
[8]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.
[9]E. Madenci and I. Guven "The finite element method and applications in engineering using ANSYS"
Springer Publisher. 2007.
[10]S. Dey, R. Chandra Murthy. D., P. Baskar "Structural Analysis of Front axle beam of a Light Commercial Vehicle (LCV) International Journal of Engineering Trends and Technology (IJETT) – Volume 11 Number 5 - May 2014.
[11]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.
[12]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.
[13]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.
[14]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
[15]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, Vol.2, Issue.4, pp-2594-2600, 2012.
[16]I. D. Paul, S. M. Sarange, G. P. Bhole and J. R. Chaudhari "Structural Analysis Of Truck Chassis Using Finite Element Method" International J. of Multidispl. Research & Advcs. in Engg., Vol. 4, No. I (January 2012), pp. 85-98, 2012.
[17]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.
[18]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.
[19]M. Singh Bajwa, S. Pundir, A. Joshi "Static Load Analysis Of Tata Super Ace Chassis And Stress Optimisation Using Standard Techniques' International Journal of Mechanical and Production Engineering, Volume- 1, Issue- 2, Aug-2013.
[20]D. Nayak, P. 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.
[21]M. Nouby Ghazaly "Applications of Finite Element Stress Analysis of Heavy Truck Chassis: Survey and Recent Development" Journal of Mechanical Design and Vibration, 2(3), 69-73, 2014.
[22]O. Ahmad Moaaz, Nouby M. Ghazaly "A Review of the Fatigue Analysis of Heavy Duty Truck Frames" American Journal of Engineering Research (AJER, Volume-3, Issue-10, pp-01-06, 2014.
[23]K. I. Swami and S. B. Tuljapure " Effect of Torque on Ladder Frame Chassis of Eicher 20.16" Journal of Engineering Research and Applications, Vol. 4, Issue 2, February 2014, pp.150-154, 2014.