Review of Off-Road Vehicle Chassis & Challenges in Tractor Chassis – A Review

M. Easwaramoorthi[1] Y. Dinesh[2] S. Karthikeyan[3] S. Kavinkumar[4] C. R. Nithin[5]
[1]Assistant Professor [2,3,4,5]UG Student
[1,2,3,4,5]Nandha Engineering College (Autonomous), Erode-638 052, India

Abstract— The automotive chassis forms the structural backbone of a commercial vehicle. The main function of the chassis is to support the components and payload placed upon it. When the vehicle travels along the road, the chassis is subjected to various stress distribution and displacement under various loading condition. Unstructured road or soil profiles combined with driving speeds and operating environments create the biggest challenge for off-road vehicles. However, the development of the off-road vehicle has received increasing interest and has been widely pursued by researchers. Keeping this in view the present work has been undertaken to develop a compact, reliable and light weight chassis for the mini tractor. Literature related to chassis are investigated and presented in this paper.

Key words: Off-Road Vehicle Chassis, Tractor Chassis

I. INTRODUCTION

The chassis panel is one of the main parts of the motor vehicle; engine transmission, suspension, shafts steering and body are mounted on the chassis. It allows weight to be added in more strategic locations to transfer as much horsepower to the ground.

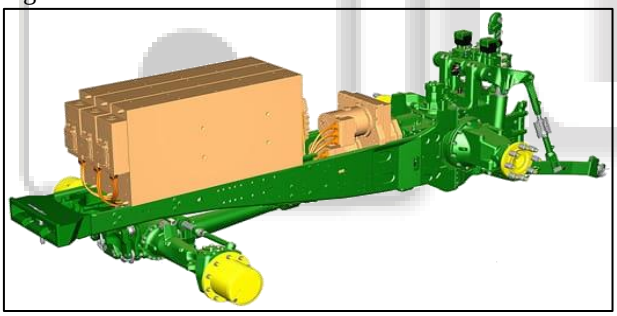

Fig. 1:

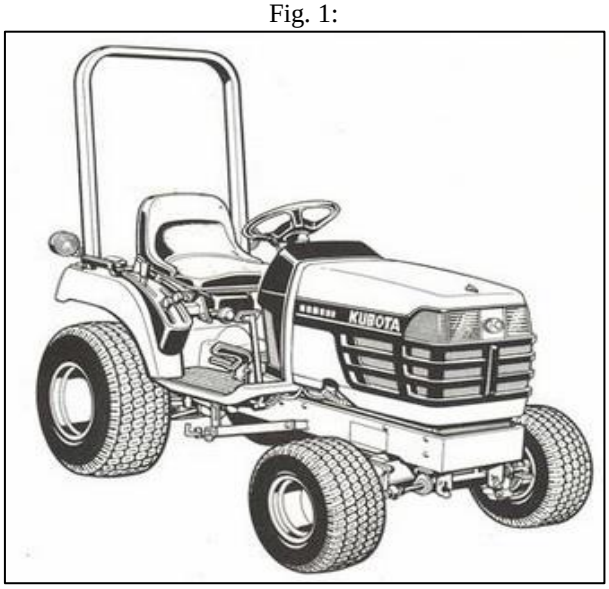

Fig. 2:

There have been many advancements in the design and construction of modified tractor chassis. In order to be as competitive as possible there is now a higher demand for tractors with a higher horsepower to weight ratio. Market demands the faster and higher transportation in a short span of time. In order to meet this market demand, vehicle manufacturers are designing off road vehicles. These heavy load carrying vehicles gives an advantage of faster, heavy transportation in a short span of time. On the other side the safety of the off road vehicle has to be ensured. Based on the historical data Chassis/body is responsible for only 7% of the failure. Chassis design should be cost effective, lesser weight, maximum payload, ensures vehicle safety by withstanding the worst loading conditions. A primary criterion in chassis design is to meet safety. Requirements first and later to reduce weight in order to satisfy fuel economy requirements. It is important to fully understand the primary loads that the vehicle structure must be capable to withstand. These loads must be efficiently transferred through the structure so that the chassis will not be prone to mechanical failure. Hence the present work aims to investigate the literature realted to tractor. The recent literature are reviewed and results are presented.

II. LITERATURE REVIEW

Dhandapani et al. (2012) have used Finite element methods to study the effect of various stress distribution using Ansys software. To investigate the field failure of 100Ton dumper they introduced gussets in failure area. After modification the chassis structure was validated by linear static analysis and found that the modified chassis was safe.

Vijaykumar and Patel (2012) have studied the Ladder chassis frame of Eicher E2 by static structural analysis. For this study, chassis was assumed as simply supported beam with overhang. Pro-E and ANSYS software were used for this work. The study also involves the analytical calculation of chassis. Both software analysis and analytical calculation results were compared and found that the stress values obtained from software analysis is 10% more and also displacement were 5.92% more.

Chinnaraj et al. (2008) have chosen to optimize the weight of chassis frame assembly of a heavy truck used for long distance goods hauling application. Dynamic stress-strain response of the component due to braking and cornering maneuvers were experimentally measured and reported. 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 rail assembly during cornering and braking. Using the commercial finite element package ANSYS, the quasi-static numerical simulations were carried out and compared with experimental results. The study helped in understanding

prevailing stresses in truck frame rails especially during cornering and braking maneuvers and brought out all geometric locations that may be potential failure locations.

Alireza Arab Solghar and Zeinab Arsalanloo (2013) studied and analyzed the chassis of Hyundai Cruz Minibus. ABAQUS Software was used to modeling and simulation. Self-weight of the chassis was considered for static analysis and acceleration, braking and road roughness were considered for dynamic analysis. It's observed that the stresses on chassis caused by braking were more compared with acceleration.

Ojo Kurdi and Roslan Abdul Rahman (2010) studied the road roughness effects on stress distribution of heavy vehicle chassis. They had analyzed Static and Dynamic conditions using Finite Element Analysis software to reduce the cost and get optimum design. The load was assumed as a uniform pressure obtained from the maximum loaded weight divided by the total contact area between cargo and upper surface of chassis. In order to get a better result, locally finer meshing was applied in the region which was suspected to have highest stress. From study it's understood that the dominant loading on the truck chassis comes from cargo and its contents as static loading. The road roughness has not given a significant effect to the stress of component

Kutay Yilmazcoban and Yasar Kahraman (2011) have studied and optimized the thickness of a middle tonnage truck chassis by using Finite Element technique. The main objective of this work was to reduce the material usage through that gaining reduction in material cost. They had analyzed three types of thickness material to chassis and compared the results by stress and displacement. Study reveals that the 4mm thickness is safe enough to carry 15ton load.

Kenji KARITA et al. (2003) had developed a chassis made by Aluminium. The material selected for the frame is 6061-T6. They used the Variable section extrusion method for making the chassis. It's developed with the help of computer Aided Engineering. Aluminium material gives an advantage of weight reduction. From this study authors found that the Aluminium chassis meets the target of weight reduction, strength and rigidity. Also they concluded that the remaining technical issues will be addressed to enable commercial adoption of the aluminum frame

Jangyeol Yoon et al. (2010) have developed Unified Chassis Control (UCC) system to prevent Roll-Over of the vehicle and improve lateral stability by integrating Active Front Steering (AFS) and Electronic Stability Control (ESC). UCC have three modes of control for prevention of roll-over, maneuverability and lateral stability respectively. Maneuverability and lateral stability are improved by reducing the yaw rate error between desired and actual based on steering input of the driver and side slip angle of the vehicle. Roll-Over Index (RI) was used to detect the roll-over and indicating the risk. UCC was also evaluated by obstacle avoidance situation at high speed to ensure the Roll-Over Resistance and Lateral Stability aspects. The study revealed that the UCC can reduce the steering effort, yaw rate error and rolling angle thus reduce the roll-over and improves lateral stability.

Ravichandra et al. (2012) have studied the alternate material for chassis. S-glass/Epoxy, Carbon/Epoxy and E-glass/Epoxy were used as material for chassis in three various cross sections like C-Section, I-Section and Box Section. TATA 2515 EX chassis was taken for analysis. 3D modelling and Finite Element analysis was done in Pro-E and ANSYS software respectively. Their study revealed that the Carbon/Epoxy I-type section has better mechanical properties like stiffness, strength and weight compared to other materials and cross sections.

Kamlesh Patil and Eknath Deore (2015) have studied the TATA 912 diesel bus chassis with different cross sections. Existing C-section along with I-section and hollow rectangular sections were taken for comparative study. The 3D models of the chassis were made in Pro-E software and linear analysis were performed in Hypermesh software. The analytical calculation and finite element analysis in software revealed that the hollow rectangular section has less stress and displacements compared with I-section and C-section.

Teo Han Fui and RoslanAbd. Rahman (2007) have studied a 4.5 ton truck chassis containing two C section side rails and five numbers of cross members against road roughness and excitations. Vibration of the components fixed to the chassis due to excitation and road roughness were studied. Chassis responses were examined by stress distribution and displacement. The suitable locations of engine and suspension systems was determined by the results of mode shape. Analysis results reveal that the major disturbance of the chassis is caused due to road excitation.

CicekKaraoglu and SefaKuralay (2002) have carried out the Finite Element Analysis of truck chassis with riveted joints. The cross member and side rail member were joined by rivets with connection plate. The thickness of the side rail member were varied from 8mm to 12mm and the connection plate thickness was also varied from 7mm to 10mm. Length of the connection plate varied from 390mm to 430mm to study the stress variations. Analysis revealed that the increase of the side rail member thickness can reduce the stresses but weight of the chassis may increase. To avoid this the thickness can be increased locally. Increase in the connection plate length also reduces stress values.

Sane et al. (2008) have analyzed the light Commercial Vehicle chassis using FEM and simulate the failure during testing. Hyper mesh and Opti-struct software were used for analysis and simulation. During the study they introduced local stiffeners to reduce the magnitude of the stress. The modified chassis stress values are reduced by 44%.

Kiran Ghodvinde and Wankhade (2014) have studied the mechanical behavior of the chassis and their welded or rivet joint are becoming a major concern in recent years and for two different frame automotive chassis, the Chevy truck chassis shows the critical stress at the joints and it is being reduce by increasing the side member thickness, connection plate thickness and connection plate length were varied.

Agarwal et al. (2017) Harmonic analysis was carried out to investigate the frequency which produces the maximum deformation of the chassis and have presented a paper on the structural analysis of a ladder chassis for higher strength. Their work includes designing of a TATA bus chassis in PRO-E software with exact dimensioning and material selection. Four different cross-sections of the vehicle

chassis were modelled namely C section, I section, Rectangular Box (Hollow) and Rectangular Box (Intermediate) type cross sections. Finite element analysis was then carried out using Hyper-mesh software and the effects of stress, strain and displacement are computed in the structural analysis under the varying load condition.

III. CONCLUSION

Most of the chassis design were carried out using CAD models and analysed with ANSYS. Most of the researches concentrated on structural design and use of alternative materials.

Literature review revealed that research on chassis structural designs like 'I', 'C', hollow rectangular, ladder type were also done and discussed the challenges related to weight reduction.

Research on tractor chassis design seems to be less in the literature. There is a further scope to improve the chassis design by using alternatve materials and change of structures for tractor chassis.

REFERENCES

[1] Chinnaraj et al. "Experimental Analysis and Quasi-Static Numerical Idealization of Dynamic Stresses on a Heavy Truck Chassis Frame Assembly", Applied Mechanics and Materials, Vol.13-14, PP 271–280, 2008.
[2] Dhandapani et al. "Static analysis of Off-High way vehicle Chassis structure for the effect of various stress distributions", International Journal of Advanced Research in Technology, Vol.2, Issue 1, 2012
[3] Teo Han Fui and Roslan Abd. Rahman, "Statics and Dynamics structural analysis of a 4.5 ton Truck chassis", Jurnal Mekanikal, Vol.24, PP 56-67, 2007.
[4] Vijaykumar Patel and Patel, "Structural analysis of a ladder chassis frame", World Journal of Science and Technology, Vol.2, PP 05-08, 2012.
[5] Kutay YILMAZÇOBAN, Yaşar KAHRAMAN, "Truck Chassis structural thickness optimization with the help of finite element technique", The Online Journal of Science and Technology, Vol. 1 Issue 3, 2011.
[6] Kurdi and Abd-Rahman Tamin. (2008) 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.
[7] Alireza Arab Solghar and ZeinabArsalanloo (2013) studied and analyzed the chassis of Hyundai Cruz Minibus. ABAQUS Software was used to model and simulation.
[8] Kenji Karita et al. (2003) had developed a chassis made by Aluminium. The material selected for the frame is 6061-T6.
[9] Jangyeol Yoon et al. (2010) have developed Unified Chassis Control (UCC) system to prevent Roll-Over of the vehicle and improve lateral stability by integrating Active Front Steering (AFS) and Electronic Stability Control (ESC).
[10] CicekKaraoglu and SefaKuralay (2002) have carried out the Finite Element Analysis of truck chassis with riveted joints. The cross member and side rail member were joined by rivets with connection plate.
[11] Sane et al.(2008) have analyzed the light Commercial Vehicle chassis using FEM and simulate the failure during testing.
[12] Ojo Kurdi and Roslan Abdul Rahman, "Finite Element Analysis of Road Roughness Effect on Stress Distribution of Heavy Duty Truck Chassis", International Journal of Technology, Vol.1, PP 57-64, 2010.
[13] Kiran Ghodvinde and Wankhade (2014) have studied the mechanical behavior of the chassis and their welded or rivet joint.
[14] Agarwal et al. (2017) Harmonic analysis was carried out to investigate the frequency which produces the maximum deformation of the chassis and have presented a paper on the structural analysis of a ladder chassis for higher strength.

