Analysis of Backhoe Loader Chassis for Weight & Cost Reduction using FEA - A Review Paper

Haresh K. Vaniya
PG-Automobile engineering
Mechanical Engineering Department
Government Engineering College, Bhuj

V. D. Sonara
Assistant Professor
Mechanical Engineering Department
Government Engineering College, Bhuj

Arvind. S. Sorthiya
Associate Professor
Mechanical Engineering Department
Government Engineering College, Bhuj

Abstract

In this study, stress analysis of a backhoe chassis was performed by using FEA. Construction industry is undoubtedly the backbone and propelling force behind our progress. In response to booming construction industry, utilization of earth moving equipment has increased considerably leading to high rate of failure. Backhoe Loader chassis is the skeleton of a commercial vehicles. The main function of the chassis is to support the different components like engine, cabin, transmission, front axle and rear axle. So it is necessary to analyze chassis to avoid failure while it is in working condition. Computer simulation techniques provides a great leverage in design optimization for weight reduction, better material utilization, shorter design cycles and elimination of major part of prototype testing.Static analysis of the chassis shows the equivalent stress and deformation contour when Backhoe Loader is in working condition. Aim of literature review is to find out high stress area at different load condition by stress minimized and weight reduction by giving design changes.
Keywords: Backhoe Chassis; Stress Analysis; FEA.

I. INTRODUCTION

In the era of globalization and tough competition the use of machines is increasing for the earth moving works, considerable attention has been focused on designing of the earthmoving equipments. Thus it is very much necessary for the designers to provide not only a equipment of maximum reliability but also of minimum weight and cost, keeping design safe under all loading conditions by careful stress analysis of the machines.

Backhoe Loaders are used primarily to excavate below the natural surface of the ground on which the machine rests and load it into trucks or tractor pulled wagons or onto conveyor belts. They are capable of excavating all classes of earth, except solid rock, without prior loosing. They are adapted to excavating trenches, pits for the basement, and general grading work, which require precious control of depths.

Chassis of commercial vehicles have almost same appearance since the model was developed. It indicates that there is very slow and stable improvement in the chassis.

II. BACKHOE LOADER CHASSIS

Chassis is one of the most important parts of the entire backhoe loader. It acts as a support for various accessories and mounting of engine, transmission, cab, front axle, rear axle, fuel tank, hydraulic tank etc. It consists of various plates welded together to form the entire structure.

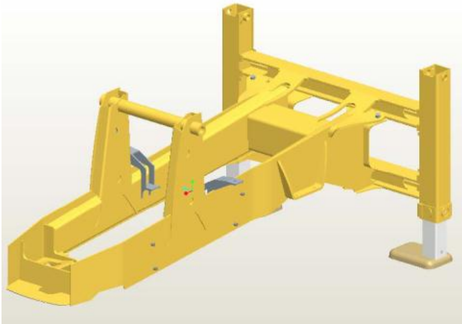

Fig. 1: Backhoe Loader Chassis

III. LITERATURE REVIEW

Abhishek Singhet et. al (2014) [1] have studied Structural analysis & optimization of vehicle chassis with constraints of maximum shear stress and deflection of chassis under maximum load. In the present work, we have taken higher strength as the main issue, so the dimensions of an existing vehicle chassis of a TATA LP 912 Diesel BS4 bus is taken for analysis with materials namely Steel alloy (Austenitic) 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. For validation the design is done by applying the vertical loads acting on the horizontal different cross sections. It is 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 (Intermediate) section is having least deflection i.e., 1.839 mm in all the four type of chassis of different cross section. Finite element analysis is effectively utilized for addressing the conceptualization and formulation for the design stages. The results obtained are quite favorable which was expected. The iterations are carried out in the analysis phase which yields the suitable values for design parameter.

V. Vamsi Krishnam Raj u et. al (2014) [2] have studied modeling and structural analysis of conventional type heavy vehicle frame. Weight reduction is now the main issue in automobile industries. In the present work, the dimensions of an existing heavy vehicle frame of a TATA 1109 EX2 vehicle is taken for modeling and analysis. The vehicle frame is initially modeled by considering 'C' cross section in SOLID WORKS 2011 and then it is imported to ANSYS 13.0. The analysis is done 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 frame. The design constraints are stresses and deformations. The results are then compared to finalize the best among all the four frames. Present used material for chassis is steel. We have considered polymeric composites like Carbon/Epoxy, E-glass/Epoxy and S- glass /Epoxy for chassis material. By employing a polymeric composite heavy vehicle chassis for the same load carrying capacity, there is a reduction in weight of 70% to 80% .Based on the results it was inferred that Carbon/Epoxy polymeric composite heavy vehicle chassis has superior strength, less deformation, less normal stress and less weight compared to steel, E-glass/Epoxy and S- glass /Epoxy. So we conclude that it is better to use Carbon/ Epoxy as a material for frames of heavy vehicle chassis. So that the fuel consumption decreases for the vehicles

Table - 1
Comparison of Results [2]

Material	*Mass(kg)*	*Max. Normal stress (Mpa)*	*Max. Equivalent Stress (Mpa)*	*Max. Deformation (mm)*
Structural steel	*385*	*3359*	*17686*	*5.68*
Carbon/ Epoxy	*79*	*2312*	*16769*	*4.03*
E-glass/Epoxy	*128*	*2888*	*17055*	*9.45*
S-glass/Epoxy	*123*	*2888*	*17055*	*8.64*

Vishal Francis et. al (2014) [3] Automotive chassis frame is an important part of an automobile. The automotive chassis frame is the structural backbone of any vehicle. The main function of chassis frame is to support the body, different parts of an automobile and to payload placed upon it. The chassis frame has to withstand the stresses developed as well as deformation occurs in it and to withstand the shock, twist vibration and other stresses. Its principle function is to carry the maximum load for all designed operating condition safely that should be within a limit. On chassis, frame maximum shear stress and deflection under maximum load are important criteria for design and analysis. In these projects, we Have calculated the von miss stress and shear stress for the chassis frame and the finite element analysis has been done for the validation on the chassis frame model of jeep. We have taken certain material as Mild sheet steel, aluminum alloy and titanium alloy for the rectangular hollow box type to design chassis frame of jeep.

Table - 2
Stress Analysis of Ladder Type Chassis Frame Result [3]

stresses	*Mild sheet steel*		*Aluminum alloy*		*Titanium Alloy*	
	Max. Mpa	*Min. Mpa*	*Max. Mpa*	*Min. Mpa*	*Max. MPa*	*Min. Mpa*
Von misses	*29.8*	*0.032*	*28.96*	*0.0047*	*30.09*	*0.0075*
Stress Shear stress	*16.33*	*0.00173*	*5.19*	*3.57*	*5.84*	*3.64*

Design-Stress	*75.00*	*11.67*	*18.33*

(1) The generated shear stresses are less than the permissible value so the design is safe for all three materials.
(2) Shear stress was found minimum in aluminum alloy and maximum in mild sheet steel under given boundary conditions.
(3) Von Mises stress was found minimum in aluminum alloy and maximum in titanium alloy under given boundary conditions.

Anand Thorat, et. al (2013) [4] have studied on “Vibration analysis of Backhoe Loader chassis 770 model” is to find out natural frequency of the chassis. Also in order to have vibration response at different location chassis harmonic analysis is carried out. The Modal analysis & Harmonic analysis is carried out in Ansys workbench and in order to validate FE results compare with experimental result. Natural frequency of the structural component should not match with the excitation frequency in order to avoid resonance phenomenon. The modal analysis chassis has fixed support at rear axle and front axle. No external force is applied to the chassis. Different mode shape and deformation contour at each frequency is shown below. From modal analysis we can observed that natural frequency varies from 43.33 Hz to 69.252 which is far away from the engine excitation frequency. In the static analysis we can observed that due sharp edges and sharp corners stress generated at some of the location is higher than the permissible stress value. Also from this analysis we can observed the deformation pattern for each load cases. In Ansys harmonic analysis we can observed vibration pattern in terms of acceleration at different location of the chassis and hence conclude that all components of the acceleration are higher around 500 Hz In the experimental analysis we can observed the vibration patterns in terms of acceleration at around 500 Hz are higher than the other frequency same as the Ansys simulation results.

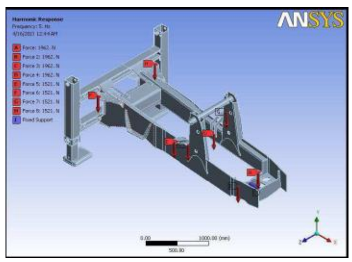

Fig. 2: Different Forces On The Chassis For Harmonic Analysis [4]

G. V. R. Seshagiri Rao et al (2013) [5] have studied on “Static analysis of Backhoe Loader chassis” is to find out stress and deformation pattern for different load cases. Static analysis is carried out for two different load cases in Ansys workbench. 1) Maximum reach condition. 2) Maximum torque condition. The designed mechanism is producing 4020.783 Kg & 4254.273 Kg of breakout force load case 1&2. In both of these above load cases there are two different sub cases. 1A) Stabilizer legs are fixed. 2A) Loader pivot points are fixed. For load case1A we can observed that stress generated are higher at stabilizer legs as compare to other part of the chassis. Also we can observe that deformation is higher at front side of the chassis which has max value of 28.194 mm. For load case2A we can observed that stress generated are higher at stabilizer legs as compare to other part of the chassis as same in the load case1A. Maximum deformation is at front end of the chassis which is 8.5281mm.

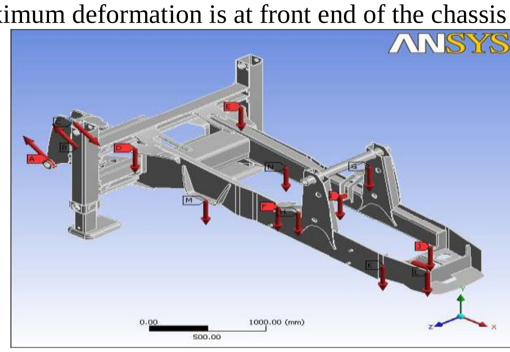

Fig. 3: Different Forces On The Chassis For Load Case1 And Load Case2 [5]

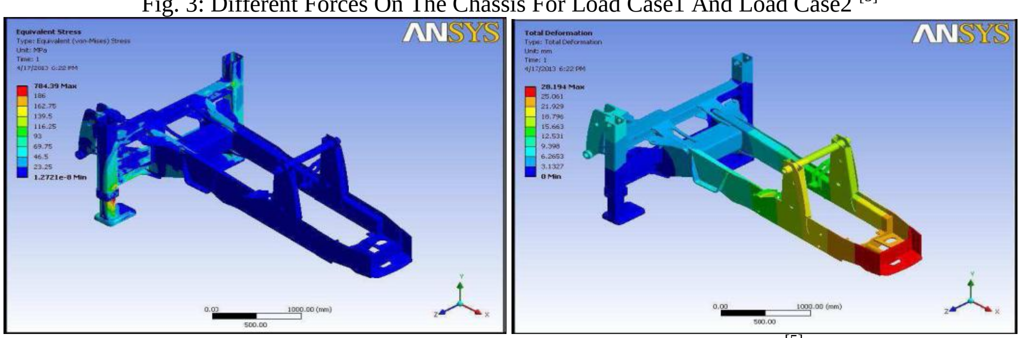

Fig. 4: Von-Mises Stress & Deformation For Load Case1a [5]

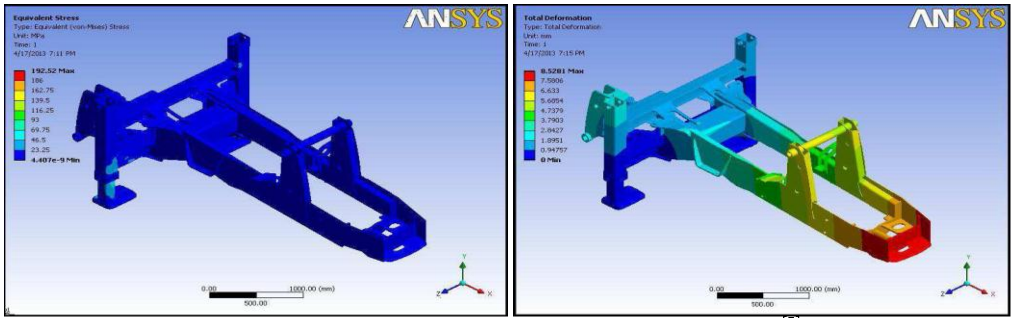

Fig. 5: Von-Mises Stress & Deformation For Load Case2a [5]

Hemant B. Patil et. al (2013) **[6]** Stress analysis of a ladder type low loader truck chassis structure consisting of C-beams design for application of 7.5 tone was performed by using FEM. The commercial finite element package CATIA version 5 was used for the solution of the problem. To reduce the expenses of the chassis of the trucks, the chassis structure design should be changed or the thickness should be decreased. Also determination of the stresses of a truck chassis before manufacturing is important due to the design improvement. In order to achieve a reduction in the magnitude of stress at critical point of the chassis frame, side member thickness is 4mm,5mm,6mm, cross member thickness and position of cross member from rear end were varied difference case 1 to 5. Numerical results showed that if the thickness change is not possible, changing the position of cross member may be a good alternative. Computed results are then compared to analytical calculation, where it is found that the maximum deflection agrees well with theoretical approximation but varies on the magnitude aspect.

The analyses are processed in the static and structural conditions. From comparison for 4mm thickness the highest stress occurred is 123.83 MPa by FE analysis and the calculated maximum shear stress is 123.83Mpa. The maximum displacement of numerical simulation result is 0.288 mm. The difference is caused by simplification of model and uncertainties of numerical calculation and improper meshing Comparing case 5 with case 2 only by increasing thickness of cross member weight is increases by 2.92 Kg stresses are decreased by 43.8 N/mm , and displacement is increased by 0.022mm2 Comparing case 4 with case 2 only by changing the position of cross member weight is not affecting stresses are deceases by 06.7 N/mm , and displacement is increased by 0.026 mm.2 Comparing case 3 with case 2 only by increasing 23.58 Kg stresses are increased by 06.5 N/mm2 and displacement is increased by 0.024mm. Hence 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.

Table - 3
Comparison of Results [6]

Sr. No	*Analytical Method*		*FE Analysis*		
	Displacement (mm)	*Stresses (N/mm^2)*	*Displacement (mm)*	*Stresses (N/mm^2)*	*Weight (Kg)*
Case 1	*1.0845*	*123.830*	*0.288*	*71.2*	*141.48*
Case 2	*1.0271*	*100.83*	*0.203*	*84.6*	*165.06*
Case 3	*0.9780*	*085.57*	*0.227*	*91.1*	*188.64*
Case 4	*1.0271*	*100.83*	*0.229*	*77.9*	*165.06*
Case 5	*0.9300*	*100.83*	*0.225*	*40.8*	*167.98*

Sharad D. Kachave et. al (2013) **[7]** have studied stress analysis and effect of web and flange's thickness on bending stiffness of a ladder type low loader truck chassis structure consisting of C-beams as a cross member, for study the effect on bending stiffness a chassis of 8 tone capacity is consider, chassis structures are comparing by the thicknesses of the profiles. For determining the strength of the frame, structural analyses were performed for these the web and flange thickness are increase and decreases by 1mm The truck chassis was modeled and the finite element analyses were solved in CATIA V5R117. During this four different cases consider to study the effect of thickness on chassis stresses. The commercial finite element package ANSYS was used for the solution of the problem. To reduce the expenses of the chassis of the trucks, the chassis structure design should be changed or the thickness should be optimized.

The maximum deflection and stress is found to be 1.201 mm & 124.02 N/mm^2 in case 2 whereas minimum deflection and stress is found to be 0.97404 mm & 100.31 N/mm^2 in case 3. The maximum bending stiffness found to be 12011.83 N/mm in case 3 and minimum bending stiffness is found to be 9741.88 N/mm in case 2.The maximum weight found to be 557.56 kg in case 3 whereas minimum weight found to be 533.35 kg incase 4. Bending stiffness to weight ratio is found to be maximum 21.54 N/Kg.mm in Case 3 and minimum 18.13 N/Kg.mm in case 2. Therefore from bending stiffness to weight ratio it clears that increase in flange thickness causes more increase in bending stiffness as compared with that causes due to increase in web thickness.

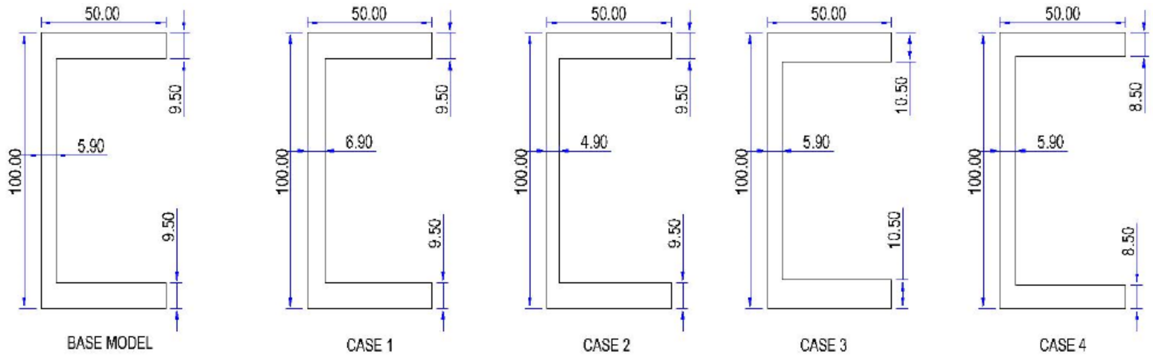

Fig. 6: Dimensions of Section Consider For Analysis [7]

Table - 4
Comparison of Results For All Case [7]

Sr No	*Deflection (mm)*		*Stresses (N/mm^2)*	*Weight (kg)*	*Bending Stiffness (N/mm)*		*Stiffness to weight Ratio*	
	Analytical	*Numerical*			*Analytical*	*Numerical*	*Analytical*	*Numerical*
Base Model	*0.853*	*1.0515*	*111.04*	*544.17*	*13716.30*	*11126.96*	*25.21*	*20.45*
Case 1	*0.846*	*1.0095*	*105.16*	*550.96*	*13829.79*	*11589.90*	*25.10*	*21.04*
Case 2	*0.859*	*1.201*	*124.02*	*537.38*	*13620.49*	*9741.88*	*25.35*	*18.13*
Case 3	*0.831*	*0.97404*	*100.31*	*557.56*	*14079.42*	*12011.83*	*25.25*	*21.54*
Case 3	*0.877*	*1.0468*	*108.36*	*533.35*	*13340.94*	*11176.92*	*25.01*	*20.96*

Monika S. Agrawal et. al (2013) [8] have studied the optimization of the automotive chassis with constraints of maximum shear stress, equivalent stress and deflection of chassis under maximum load. A sensitivity analysis is carried out for weight reduction. So a proper finite element model of the TATA 1612 chassis is to be developed by using actual parameter, the chassis is modeled in CATIA v5. FEA is done on the modeled chassis using the ANSYS Workbench 12; the FEA is carryout for static, dynamic and shape optimization. For the analysis of chassis used Mild steel material. Optimization of design has been achieved There has been considerable decrease in weight of chassis Optimization has been achieved by reducing the thickness of chassis C-section wherever less load is acting and where there are less deformations. The change in the mass of the chassis model and it is reduced from 401.55 kg to 366.76 kg. The maximum shear stress, maximum equivalent stress and displacement are reduced and yield strength of chassis material is so large and if we consider results and yield strength, it is clear that design is safe. From the above result it is clear that the weight is also reduced by 8.49 % of the chassis frame. The paper has looked into the determination of the dynamic characteristic the natural frequencies and the mode shapes of the truck chassis, the eight natural frequencies of the truck chassis are below 100 Hz and vary from 13 to 50 Hz. Generally most truck chassis frequencies are below 50 Hz so design is safe.

Vimalkumar A. Patel et.al (2012) [9] In the linear static analysis backhoe loader chassis, the stress distribution and deformation profile of the Backhoe Loader chassis is observed when backhoe is at maximum reach condition. The material of the backhoe loader chassis is structural steel alloy. The properties of the material are Modulus of Elasticity 200 GPa & Mass Density 7850kg/m3. In this analysis we have taken two different load cases. 1) Maximum reach condition 2) Maximum torque condition. In both of these above load cases there are two different sub cases. For load case 1A i.e. when stabilizer legs are fixed. For load case1A we can observed that stress generated are higher at stabilizer legs as compare to other part of the chassis. Also we can observe that deformation is higher at front side of the chassis which has max value of 28.194mm. For load case 1B i.e. when loader tower points are fixed. For load case1B it can be observed that stress generated are higher at loader tower as compare to other part of the chassis. Also it can be observed that deformation is higher at front side of the chassis which has maximum value of 9.0504mm. Here maximum stress is higher than permissible value due to sharp edges and corners. These can be minimizes by smoothening sharp edges and introducing fillet.

Table - 5
Dead Weight of Different Components [9]

No	*Component*	*Weight(Kg)*	*Load(N)*
1	*Cabin*	*620*	*6082*
2	*Front axle*	*225*	*2207*
3	*Rear axle*	*410*	*4022*
4	*Transmission*	*215*	*2109*
5	*Engine*	*585*	*5739*
6	*Chassis*	*743*	*7289*

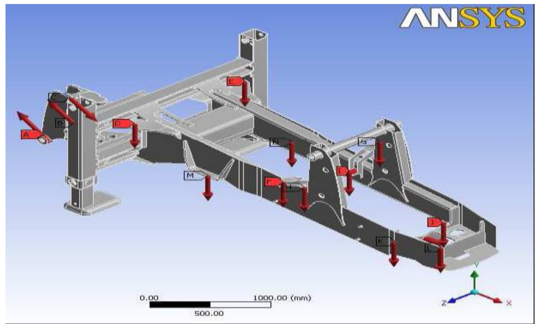

Fig. 7: Different Forces On Chassis [9]

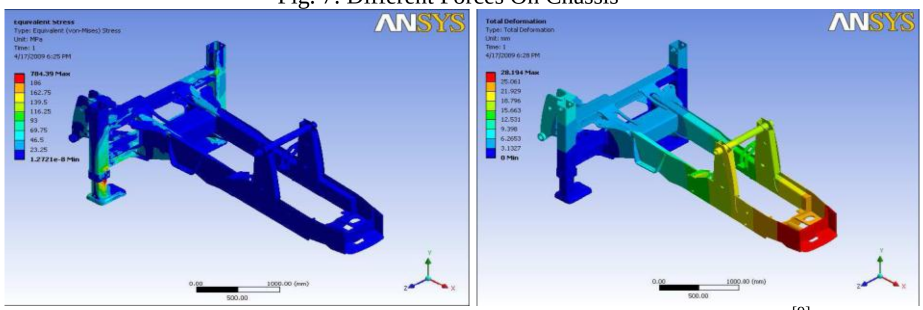

Fig. 8: Von- Mises Stress For Load & Deformation For Load Case 1A [9]

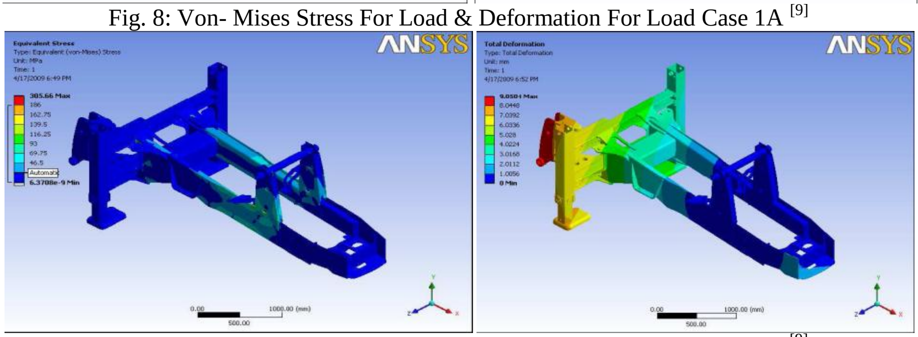

Fig. 9: Von- Mises Stress For Load & Deformation For Load Case 1B [9]

MohdAzizi Muhammad Nor et al (2012) [10] performs the stress analysis of an actual low loader structure consisting of I-beams design application of 35 tone trailer. The material of structure is Low Alloy Steel A 710 C (Class 3) with 552 MPa of yield strength and 620 MPa of tensile strength. He uses modeling software CATIA V5R18. The results of analysis revealed that the location maximum deflection and maximum stress agrees well with theoretical maximum location of simple beam under uniform loading distribution. This study found out that there is discrepancy between the theoretical (2-D) and numerical (3-D FEA) results. It is observed that the maximum deflection is pointed in situated in between BC1 and BC2 with magnitude of 7.79mm. The results of the numerical analysis revealed that the location maximum deflection and maximum stress agrees well with theoretical maximum location of simple beam loaded by uniform force.

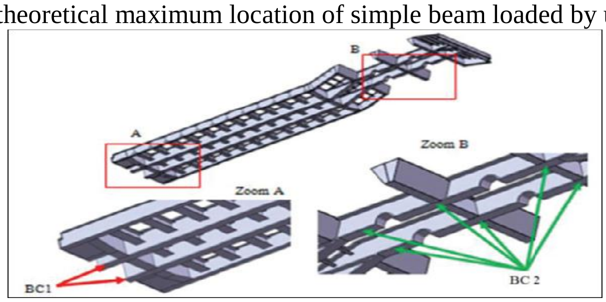

Fig. 10: Boundary Conditions Representation On Chassis of BC1 And BC2 [10]

N. K. Ingole et al (2011) [11] does the modifications in existing model of tractor trailer chassis by 1) Variation in Cross sectional areas of cross members, 2) Variation in cross sectional areas of cross and longitudinal members, 3) Variation in cross sectional areas of cross and longitudinal members and 4) Changing the position of cross members of main frames of chassis, Considering variable cross sectional areas of cross and longitudinal members. It has been found that, maximum stress present in existing chassis is 75 MPa and weight of chassis is 751.82 kg. Case 4 leads to maximum weight reduction of approx 112 kg as compared to case 1, 2 and 3. So modifications as per case 4 are also recommended, case 3 the weight reduction is 88 kg with maximum stress level in range of 25MPa to 66 MPa.

Table - 6
Comparison of Result For Different Cases [11]

S.N	Various Cases	Weight in Kg	Range of Equivalent Stresses on members in Mpa	Final Reduction in weight Kg	Factor of safety under sudden load (without plates)	Factor of Safety under sudden load (with plates)
1	Existing Chassis	751.82	28 to 75	-	1.66	3.37
2	Case1	705.88	17 to 69	45.88	1.78	2.71
3	Case2	674.67	22 to 75	77.15	1.66	2.71
4	Case3	663.87	25 to 66	87.95	1.89	2.31
5	Case4	640.09	42 to 75	111.73	1.66	3.37

Roslan Abd Rahman et. al (2008) [12] does 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.

Cicek Karaoglu et al (2001) **[13]** have does stress analysis of heavy duty truck chassis with riveted joints by utilizing a commercial finite element package ANSYS version 5.3.during this study, he examine the effect of the side member thickness and connection plate thickness with length change, the side member thickness is varied from8 to 12 mm, and the thickness of the connection plate is also varied from 8 to 12 mm by local plate, the connection plate thickness is varied from 7 to 10 mm, and the length of the connection plate (L) is varied from 390 to 430 mm. from this he 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 (L) seems to be best practical solutions for decreasing the stress values.

Sairam kotari et. al (2002) [14] have studied the analysis of chassis frame for improving its payload by adding stiffener and c channel at maximum stress region of chassis frame. The FEM analysis has been carried out with various alternatives. The results illuminate the new creative ways for optimum frame design which makes it more sustainable for structural concerns. This paper analyzed the backbone frame for both dynamic and static load condition with the stress deflection bending moment on the tatra chassis frame. The finite element analysis over Ansys is performed by considering the load cases and boundary conditions for the stress analysis of the chassis. The tatra chassis is being modeled in catia v5 and then it is being imported in the finite element analysis software-Ansys. At present the payload of the tatra is 10.4 tones in this project we enhance the capacity of vehicle to 14 tones from existing chassis as per the requirement. This has been carried out with limited modifications by adding stiffeners and c channel. The necessary design changes required to enhance the load carrying capacity of the vehicle has been recommended. The existing TATRA chassis was analyzed by the finite element analysis for installation of the Antenna and Electronic components and the stress levels are found to be 737.3 N/mm2.After modifications, the TATRA Chassis with suitable reinforcement, increase in thickness, addition of stiffeners, the finite element analysis was carried out, and the stress levels of chassis are found as 173.38 N/mm2, which is less than yield stress 410 N/mm2. From the above Results, it can be concluded that the modified TATRA chassis is capable to carry the loads beyond the previous payload up to 14 tones.

Table - 7
Material Properties of Chassis [14]

No	Material	Yield Strength (σ_y)	Ultimate Tensile Strength (σ_u)	Young's Modulus (E)	Poisson's Ratio (ν)
1	High strength Structural Steel	410 N/mm2	540 N/mm2	2, 00,000 N/mm2	0.3

Table - 8
Specifications of The Chassis [14]

No	Description	Dimension (mm)
1	Length of Chassis	10208
2	Width of Chassis	1000

IV. 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. This study makes a case for further investigation on the design of chassis using FEA Ansys software.
- Finite Element Analysis can be used as a tool to redesign the component if it is already designed by classical design theory.
- Without making the prototype the loading condition can be simulated and make the necessary changes at the design level, if required for the proper functioning of the component.
- Component can be optimized material requirement of parts strengthened can be checked.
- From linear static analysis, maximum deformation of the component and maximum stress can be known and from that the material can be changed if required to meet the loading condition.
- If the component is failing, either the plate thickness can be changed or the curvature can be increased and again check for the loading conditions. After redesign of components, the stress is under the allowable limit.
- Higher stress region can be minimized by removing sharp corners, by providing smooth fillet.
- Optimization of the chassis can be done by removing unnecessary material and providing stiffener to the chassis. Also by providing proper reinforcement to the chassis vibration can be minimized so that there will be minimum vibration transmitted to the operator.
- Fatigue analysis can be done for more accurate results and to predict the life of attachments.
- Feasibility of different design change & material change on stress & weight reduction can be done by FEA00

REFERENCES

[1] Abhishek Singh, Vishal Soni, Aditya Singh " Structural Analysis of Ladder Chassis for Higher Strength" International Journal of Emerging Technology and Advanced Engineering, ISSN 2250 -2459, Volume 4, Issue 2, February 2014
[2] V. Vamsi Krishnam Raj u, B. Durga Prasad, M. Balaramakrishna, Y. Srinivas" Modeling and Structural Analysis of Ladder Type Heavy Vehicle Frame" International Journal of Modern Engineering Research (IJMER), ISSN: 2249–6645, Vol. 4, Issue 5, May 2014.
[3] Vishal Francis, Rajnish Kumar Rai, Anup Kumar Singh, Pratyush Kumar Singh, Himanshu Yadav " Structural Analysis of Ladder Chassis Frame for Jeep Using Ansys " International Journal of Modern Engineering Research (IJMER) ISSN: 2249–6645, Vol. 4, Issue 4, 2014.
[4] Anand Thorat, G.V.R, Seshagiri Rao, Soham Munjal BM College of Technology, Indore (M.P) India " vibration Analysis of Loader Backhoe chassis 770model " International Journal of Engineering Sciences & research Technology(IJESRT), ISSN: 2277-9655, August 2013.
[5] Anand Thorat, G.V.R.Seshagiri Rao" Static Analysis of Loader Backhoe chassis 770 model" International Journal of Engineering Sciences & research Technology (IJESRT), ISSN: 2277-9655, July 2013.
[6] Hemant B.Patil , SharadD.Kachave , Eknath R.Deore" Stress Analysis of Automotive Chassis with Various Thicknesses" IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) , ISSN: 2278-1684, Volume 6, Issue 1, Mar – Apr 2013, PP 44-49
[7] Hemant B.Patil, Sharad D.Kachave "Effects Of Web and Flange Thickness on Bending Stiffness of Chassis" International Journal of Advance Research in Science and Engineering (IJARSE), ISSN-2319-8354, Vol. No.2, Issue No.7, July, 2013.
[8] Monika s. agrawal, md razik "finite element analysis of truck chassis" international journal of engineering sciences & research Technology (ijesrt), ISSN: 2277-9655,
[9] Vimalkumar A Patel " Static structural analysis of Backhoe loader chassis " International Journal Of Advance Science, Engineering and Technology(IJASET),ISSN 2319-5924, Val 1, Issue 1, 2012, pp. 17-20
[10] MohdAzizi Muhammad Nora, HelmiRashida, Wan MohdFaizul Wan Mahyuddin "Stress Analysis of a Low Loader Chassis" Elsevier Ltd. Sic Verse Science Direct Procedia Engineering, July 2012
[11] N.K.Ingole, D.V. Bhope " Stress analysis of tractor trailer chassis for self-weight reduction " International Journal of Engineering Science and Technology (IJEST), ISSN: 0975 -5462 Vol. 3 No, 9 September 2011.
[12] RoslanAbdRahman, MohdNasirTamin, OjoKurdi Faculty of Mechanical Engineering, University Technologi Malaysia Stress analysis of heavy duty truck chassis as a preliminary data for its fatigue life prediction using FEA technique.
[13] CicekKaraoglu, N. SefaKuralay "Stress analysis of a truck chassis with riveted joints "Elsevier Science B.V Finite Elements in Analysis and Design 38 (2002) 1115– 1130.
[14] Sairam kotari, V. Gopinath, 094,QIS College Of Engineering & Technology Ongole, Andhra Pradesh Static and Dynamic analysis of Tatra chassis Vo l.2, Issue.1, pp-086 .