ANSYS of the Tata LPK 2518 Truck Chassis for Stresses & Weight Reduction using Fem Tool

Mr. Zakir Hamid War[1] Gurlal Singh[2]
[2]Assistant Professor
[1,2]Department of Mechanical Engineering
[1,2]Desh Bhagat University, Fatehgarh Sahib, Punjab, India

Abstract— Chassis is one of the important parts that used in automobile industries. It is a rigid structure that forms a skeleton to hold all the major parts together. Chassis is a major component in a vehicle system. It should be rigid enough to withstand the shock, twist, vibration and other stresses. Along with strength, an important consideration in chassis design is to have adequate bending stiffness for better handling characteristics. So, maximum stress, maximum equilateral stress and deflection are important criteria for the design of the chassis. This work involved Transient analysis to determine key characteristics of a chassis. The static characteristics include identifying location of high stress area. This paper presents the Transient load analysis of TATA LPK 2518 truck ladder chassis using ANSYS workbench and stress, Weight reduced by optimization in design. This has been carried out with limited modifications. We have taken Aluminium Alloy 6063-T6 for the chassis. Load applied on chassis is 9.26 N. The necessary design changes required to enhance the load carrying capacity of the vehicle has been recommended successfully.

Key words: Chassis, Solid Works, Transient Structure, ANSYS, FEA, Aluminium Alloy 6063-T6

I. INTRODUCTION

The chassis is thought to be one of the huge structures of a car. It is the edge which holds both the auto body and the power prepare. Different mechanical parts like the motor and the drive prepare, the hub congregations including the wheels, the suspension parts, the brakes, the controlling segments, and so forth., are shot onto the body. The chassis gives the quality expected to supporting the distinctive vehicular parts and in addition the payload and keeps the car unbending and hardened. Subsequently, the case is additionally an imperative part of the general wellbeing framework. Besides, it guarantees low levels of clamor, vibrations and brutality all through the vehicle. Skeleton ought to be sufficiently unbending to withstand the stun, wind, vibration and different burdens. Along the quality, a critical thought is case configuration is to have satisfactory bowing and tensional firmness for better taking care of attributes. Thus, quality and firmness are two imperative criteria for the outline of chassis. The heap conveying structure is the chassis, so the frame must be designed to the point that it needs to withstand the heaps that are coming over it. Car chassis for the most part eludes to the lower some portion of the vehicle including the tires, motor, outline, driveline and chassis. Out of these, the edge gives essential help to the vehicle segments put on it. Likewise the edge ought to be so solid to oppose affect stack, contort, vibrations and other twisting burdens. The skeleton outline comprises of side rails appended with various cross individuals. Alongside the quality; a critical thought in the skeleton configuration is to expand the bowing firmness and torsion solidness. Legitimate tensional solidness is required to have great taking care of qualities. Normally the chassis are outlined based on quality and firmness. In the regular outline system the plan depends on the quality and is then engaged to build the firmness of the body, with little thought to the heaviness of the body. This outline technique includes the adding of auxiliary cross part to the current chassis to enhance its tensional solidness. Thus, weight of the chassis increments. This expansion in weight of the chassis fuel effectiveness is lessened and expands the general cost because of additional material. The outline of the case with appropriate firmness and quality is vital. The outline of a vehicle structure is of crucial significance to the general vehicle execution. The vehicle structure assumes a vital part in the unwavering quality of the vehicle [6]. For the most part, truck is a considerable engine vehicle which is intended for conveying the joined weights, for example, the motor, transmission and suspension and in addition the travelers and payload. The significant concentration in the truck fabricating businesses is to plan vehicles with more payload limit. Utilizing high quality steels than the customary ones are conceivable with comparing increment in payload limit. The chassis of truck which is the fundamental piece of the vehicle that joins the principle truck segment frameworks, for example, the axles, suspension, control prepare taxi and trailer and so forth, is one of the conceivable possibilities for significant weight diminished. The chassis structure is the greater part in the any car vehicle.

II. LITERATURE REVIEW

A.Hari Kumar et al. [1] in the present work, ladder type chassis frame for TATA Turbo Truck was analyzed using ANSYS 14.5 software. From the results, it is observed that the Rectangular Box section is having more strength than C and I Cross-section type of Ladder Chassis. The Rectangular Box Cross-section Ladder Chassis is having least deflection i.e., 2.96 mm and least Von Misses stress and Maximum Shear stress i.e., 54.31MPa & 5.98MPa respectively for Aluminum Alloy 6063-T6 in all the three types of chassis of different cross section.

Anurag et al. [2] proposed that the area where the stress concentration is maximum due to applied load and the portions that has to be considered in the design of chassis frame in order to avoid frequent failures to improve its reliability. Stress analysis of chassis has been done to predict the weak points. Several state of the art papers and even books on chassis stress analysis have been presented in the recent years. This study makes a case for further investigation on the design of truck chassis using Ansys software.

Monika S.Agrawal et al. [3] introduced that the Static Analysis, we can determine highly stressed area of truck chassis due to applied load, and analytical shear stress is 13.33% less than FEA values. In Modal Analysis we can determine the total deformation of truck chassis frame at a different frequency range. From the analysis results, the frequency range of Modal Analysis for Free-Free Condition is 16.89 Hz to 46.316Hz. Also frequency range of Modal analysis due to applied load on truck chassis is 13.886 Hz to 43.828.Hz (i.e. for Static Bending with Chassis Load). The Frequency range of both modal analyses for Free-Free Condition and Applied load on truck chassis are in the range 10 to 50 Hz. Almost all of the truck chassis designed were based on this frequency range to avoid resonance during operating conditions, so that the design of truck chassis is safe. By reducing the height of the cross-member of chassis by 8.6%, the weight reduction of chassis is found to be reduced by 8.72%.

Wang Hai-fei et al. [4] proposed that the inherent frequency value of the first order and second order frame are among sensitive frequency value ranges which we can get from modal analysis. For the vehicle driving in the maximum speed is 30 km/h, that close to the first order natural frequency when driving in macadam if regardless of the middle wheel constraints, thus we should pay attention to avoid resonance while driving. Large displacement and stress mainly concentrated in the central frame by random vibration stress and displacement echogram. And the maximum is 0.02577 m, the biggest stress is 320 MPa, the biggest stress is less than the material yield strength

Mehdi Mahmoodi-k et al. [5] proposed that the Cross-section of chassis reduced the overall weight of chassis by 21 %, which is optimized according to loading; after modifications the finite element analysis was carried out. The stress/strain distribution and natural frequencies were calculated along the chassis. Maximum stress and strain levels are found in front section of chassis, where engine and transmission are installed, and this section should be modified by U-shape stiffeners. In order to improve static and dynamic characteristic of chassis to endure equipment loads, cross section and mass distribution of the chassis were optimized.

Dr.R.Rajappan et al. [6] proposed that the paper has looked into the determination of the dynamic characteristic the natural frequencies and the mode shapes of the truck chassis, investigating the mounting locations of components on the truck chassis and observing the response of the truck chassis under static loading conditions. The first six natural frequencies of the truck chassis are below 100 Hz and vary from 16.24 to 61.64Hz. For the first four modes, the truck chassis experienced global vibration except for the fifth mode. The global vibrations of the truck chassis include torsion, lateral bending and vertical bending with 2 and 3 nodal points. Stress analysis of chassis has been done to predict the weak points.

III. OBJECTIVE OF THE STUDY

1) To study the effect of loads acting on the Chassis under referred boundary conditions.
2) To perform Transient Analysis of Chassis in FEM tool ANSYS assigning the given materials.
3) To identify the set of dimensional parameters with minimum mass that results in minimum stress and deflection.

IV. METHODOLOGY

In our present work 3D models of the chassis of C cross section will be prepared in CAD tool named SOLID WORKS with the proper dimensions and this model will be imported into CAE tool ANSYS in STEP format. Then applying the boundary conditions and the loading conditions, transient structure analysis will be performed to find the von-misses stress value and deformation in the chassis.

V. GEOMETRIC MODELING & FINITE ELEMENT ANALYSIS

A. Sketcher

Sketcher is utilized to make two-dimensional portrayals of related inside the part. We can a harsh layout of bends, and afterward indicate conditions called imperatives to characterize the shapes all the more decisively and catch our outline part. Each bend is alluded to as a portray object. To make a new portray, picked begin to mechanical outline and sketcher at that point select the reference plane or portray plane in which is to be made.

B. Modeling

SOLIDWORKS software is used to create chassis model. SOLIDWORKS is an interactive computer aided designing and manufacturing system. The cad functions automate the normal engineering, design and drafting capabilities found in today's manufacturing companies. Creation of a 3-D model in SOLIDWORKS can be performed using three workbenches i.e.: sketcher, modeling and assembly.

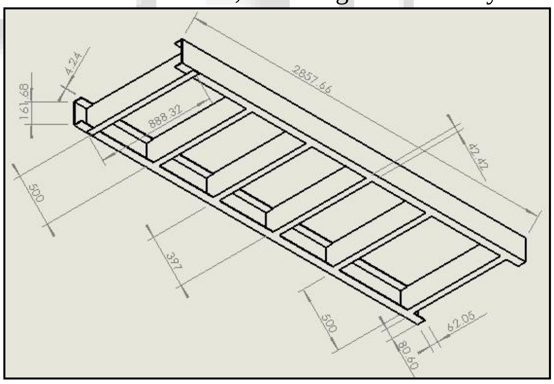

Fig. 1: Chassis Frame

The following dimensions are recommended:

a) Length: - 2857.66mm
b) Width: - 888.32mm
c) Height: -16.68mm
d) C section Thickness: - 4.24mm

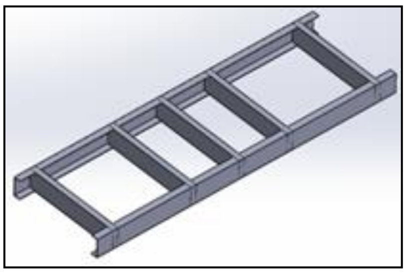

Fig. 2: Chassis Model in SOLIDWORK

Density	2800Kg m^-3
Young's modulus	26000MPa
Poisson ratio	0.3
Tensile yield strength	220MPa
Tensile ultimate strength	250MPa

Table 1: Aluminium alloy 6063-T6 Properties

C. Finite Element Analysis

Finite element modeling of any solid component is the generation of geometry, applying the components material properties using simulation software, meshing the component by appropriate size, defining the load and boundary constraints. These steps will lead to the calculation of stresses and displacements in the component. Constrained Component Investigation (FEA) was first made in 1943 by R. By and by, a paper appropriated in 1956 by Turner et al. developed a more broad importance of numerical examination. The paper concentrated on the "solidness and redirection of complex structures". By the mid 70's, FEA was limited to exorbitant incorporated PC PCs all things considered controlled by the flight, auto, insurance, and nuclear organizations.

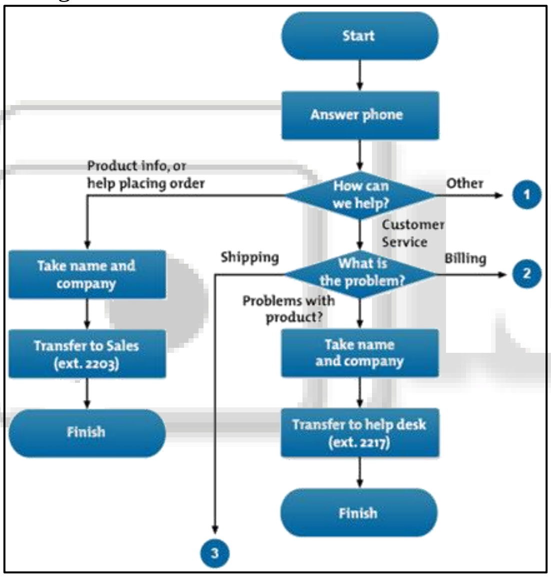

Fig. 3: Finite Element ANSYS Process

D. Mesh Generation

Mesh age is the act of producing a polygonal or polyhedral mesh that approximates a geometric area. The expression "lattice age" is frequently utilized conversely. Normal uses are for rendering to a PC screen or for physical reenactment, for example, limited component examination or computational liquid elements.

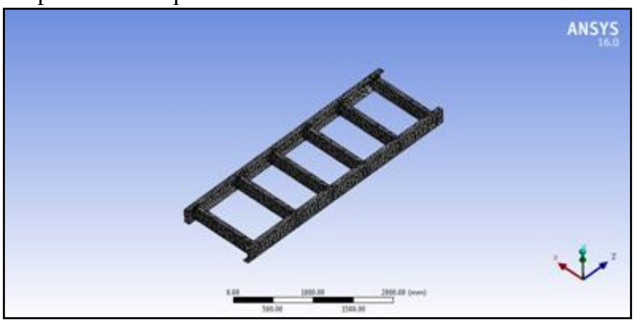

Fig. 4: Meshing Of Chassis Model

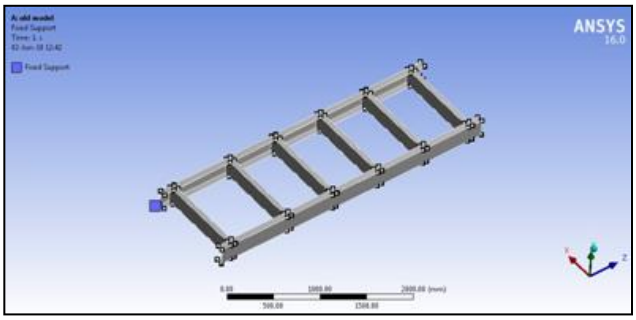

Fig. 5: Fixed Support of Chassis Model

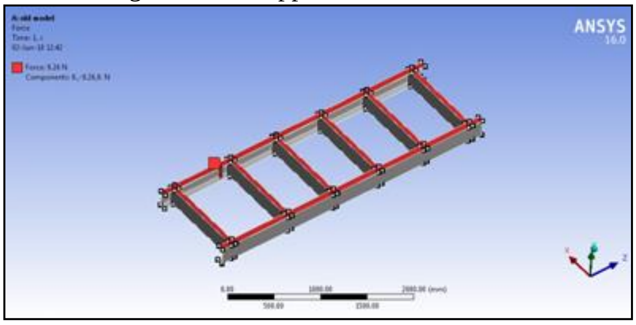

Fig. 6: Force Apply On Chassis Model

VI. RESULTS & DISCUSSIONS

A. ANSYS of Existing Design

After obtaining results of analysis of the conventional Chassis design, the maximum von-Misses stress developed is found to be 0.011652N/mm^2 as shown in Fig. (7).

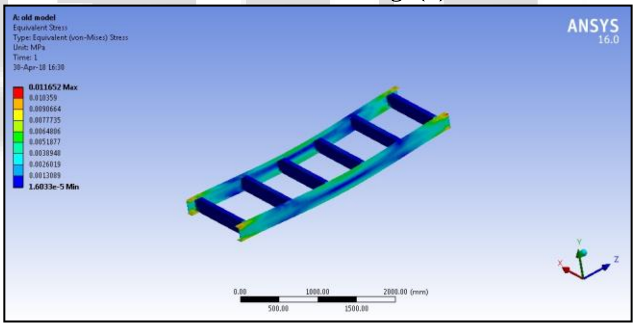

Fig. 7: Max Stress in Chassis

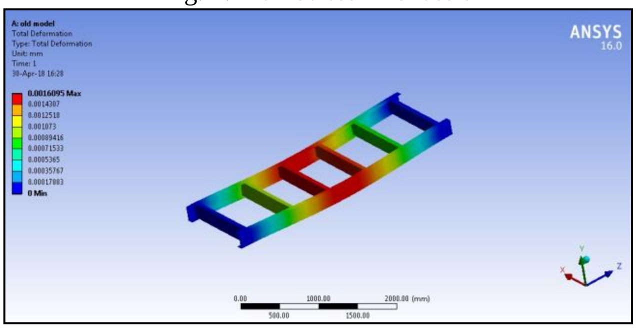

Fig. 8: Max Deformation in Chassis

The maximum deformation is found in the conventional design of chassis frame is 0.0016095mm as shown in Fig. (8).

B. Design Changes (New Design):

In the new design of chassis the maximum stress is found 0.0098905N/mm^2. The force is applied to the top surface of chassis. And the maximum stress is found at the middle of chassis as shown in Fig. (9)

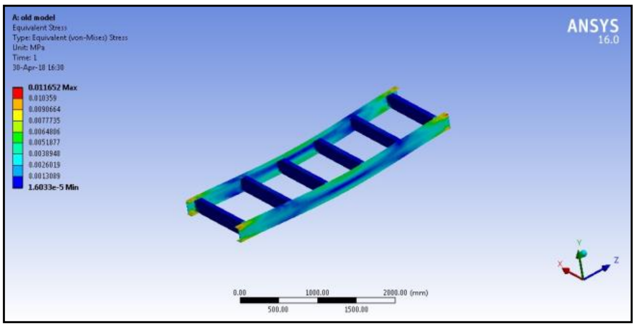

Fig. 9: Max Stress in Chassis

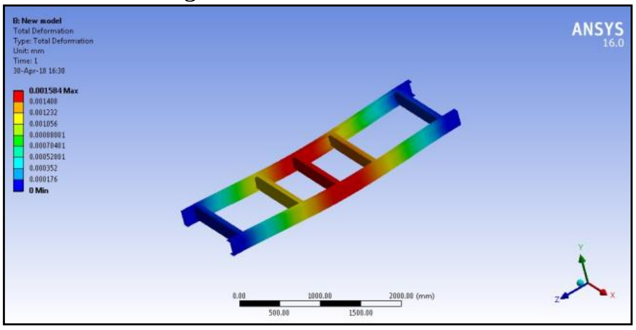

Fig. 10: Max Deformation in Chassis

The maximum deformation is found 0.001584 mm in the new design of chassis as shown in Fig. (10).In the new design of chassis Stress and Deformation is less as compared to the conventional design. And the maximum stress is produced at the mid of chassis frame. The new design has best results as compared to the conventional design.

B. *Graph Comparison*

These graphs shows the comparison results between the stress and deformation produced in the conventional and new Chassis design.

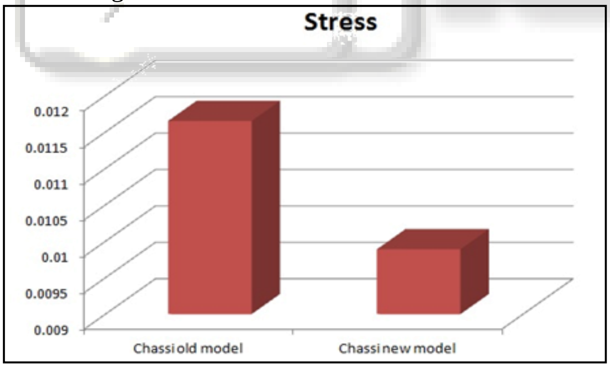

Fig. 11: Stress Comparison

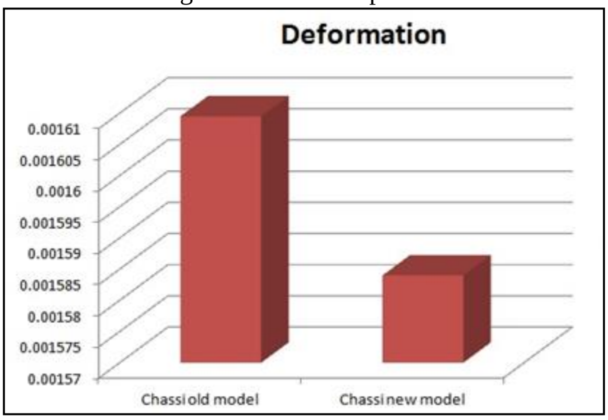

Fig. 12: Deformation Comparison

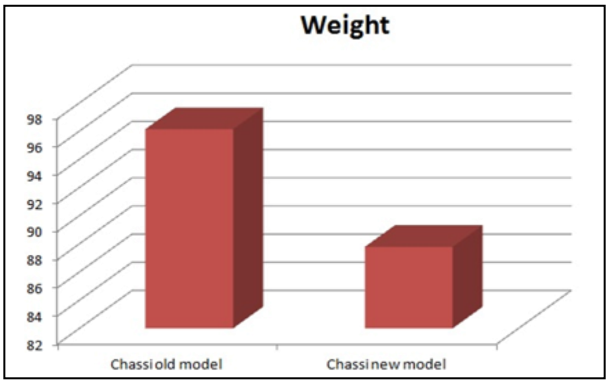

Fig. 13: Weight Comparison

VII. Conclusion

From the above graphs we can see that Deformation and Stress are reduced in the optimization design of chassis.

1) The maximum stress produced in the old design of chassis is 0.011652 and maximum stress in the optimization design of chassis is 0.0098905.Which is less as compared to the old design chassis stress.
2) The Deformation is more in the old design of chassis as compared to the optimization design of chassis.
3) After the optimization of design the weight reduced 8.68% of the chassis.

Therefore at the end we have concluded that new optimization design of the chassis is best as compared to the old design of chassis. We will be preferring new optimization design of the chassis.

A. *Future Scope*

Following Recommendations are there for future scope:

1) The whole analysis can be repeated with more materials used for chassis.
2) The mass of chassis can be further reduced from the insensitive parameters.
3) The vibration analysis of chassis can be helpful for further study of the vibrations produced in the engine.

References

[1] A.Hari Kumar, V.Deepanjali (2016)"Design & analysis of automobile chassis" volume 5, issue 1, January 2016.
[2] Anurag, Amrendra Kumar Singh, Akash Tripathi, Aditya Pratap Tiwari, Nitish Upadhyay, Shyam Bihari Lal (2016)"Design and analysis of chassis frame" vol. 03 no. 04 | April 2016.
[3] Cosme, C.; Ghasemi, A.; Gandevia, J. Application of Computer Aided Engineering in the Design of Heavy-Duty Truck Frames. // SAE Paper 1999-01-3760, International Truck & Bus Meeting & Exposition, Detroit, Michigan, 1999.
[4] Dr.R.Rajappan, 2 M.Vivekanandhan (2013) "Static and modal analysis of chassis by using fea" ||volume|| 2 ||issue|| 2 ||pages|| 63-73 ||2013|| issn: 2319 – 1813 isbn: 2319 – 1805.
[5] Dumber et al.,International Journal of Advanced Engineering Research and Studies E-ISSN2249–8974 .
[6] Liu, Z. S.; Lu, C.; Wang, Y. Y.; Lee, H. P.; Koh, Y. K.; Lee, K. S. Prediction of noise inside tracked vehicles. // Journal of Applied Acoustics. 67, 1(2006), pp. 74-91.

[7] Mehdi Mahmoodi-K, Iraj Davoodabadi, Vinko Višnjić, Amir Afkar (2014) "Stress and dynamic analysis of optimized trailer chassis" tehnički vjesnik 21, 3(2014), 599-608.
[8] Mohd Azizi Muhammad Nor, Helmi Rashida, Wan Mohd Faizul Wan Mahyuddin, Mohd Azuan Mohd Azlan, Jamaluddin Mahmud (2012) "Stress analysis of a low loader chassis" Procedia Engineering 41 (2012) 995 – 1001.
[9] Monika S.Agrawal (2015) "Finite element analysis of truck chassis frame" volume: 02 issue: 03 | june-2015.
[10] M.Vivekanandhan (2013) "Static and Modal Analysis of Chassis by Using FEA" ||Volume|| 2||Issue|| 2 ||Pages|| 63-73 ||2013|| Issn: 2319 – 1813 Isbn: 2319 – 1805.

