International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 08 Issue: 04 | Apr 2021 www.irjet.net p-ISSN: 2395-0072

DESIGN AND STRUCTURAL ANALYSIS OF TRUCK CHASSIS

Naveen G[1], Dr. Prashanth A S[2]

[1]P.G. Student, Dept. of Mechanical Engineering, Dr. Ambedkar Institute of Technology, Bengaluru, India
[2]Asst. Professor, Dept. of Mechanical Engineering, Dr. Ambedkar Institute of Technology, Bengaluru, India

---***---

ABSTRACT – Chassis is the French word was used to denote the frame parts or main structure of vehicle, which is now, denotes the whole vehicle except body in case of heavy vehicles (that is vehicle without body is called chassis). In case of light vehicles of mono construction, it denotes the whole vehicle except additional fittings in the body. Automobile chassis usually refers to the lower body of the vehicle including the tires, engine, frame, driveline and suspension. Out of these, the frame provides necessary support to the vehicle components placed on it. Role of the chassis frame is to provide a structural platform that can connect the front and rear suspension without excessive deflection. Also, it should be rigid enough to withstand the shock, twist, vibration and other stresses caused due to sudden braking, acceleration, shocking road condition, centrifugal force while cornering and forces induced by its components. So, strength and stiffness are two main criteria for the design of the chassis.

Key Words: Truck chassis, High strength, CATIA V5, ANSYS, Ls-DYNA.

1.1 INTRODUCTION

The chassis of a truck is the vehicle's backbone, integrating the majority of component systems such as axles, suspension, gearing, cab and trailer, and is normally subjected to the cabin's load, its contents, and inertia forces resulting from rough road surfaces, among other things (i.e. static, dynamic and cyclic loading). The location of the critical stress point is critical in order to assess and optimize the mounting of components such as the engine, suspension, transmission, and others. The Finite Element Method (FEM) is one of the methods used to identify the juncture. Factor of safety is employed to supply a design margin over the theoretical design capacity.

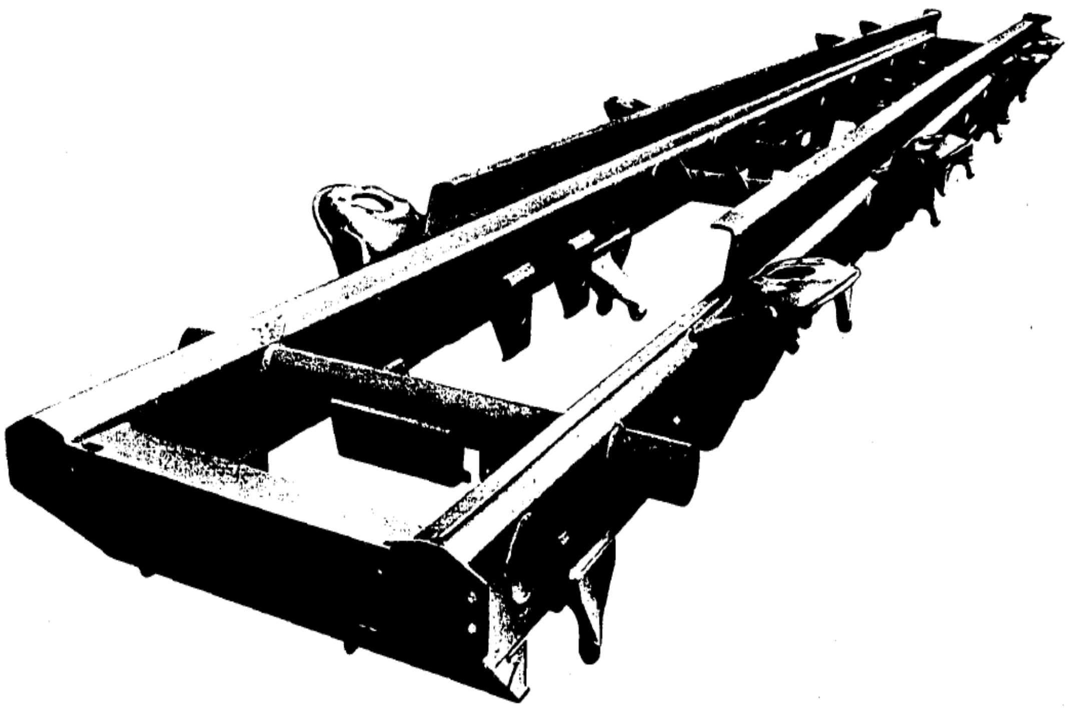

Figure 1: Schematic sketch of Chassis

Before thinking about the frame itself, think about the vehicle's suspension points. After the chassis is designed, designing an ideal suspension for the appliance will make construction impractical or impossible to meet the requirements. Suspension and all of the chassis specifications would necessitate a great deal of compromise. We're just going to look at the "Double Wishbone" suspension in this text.

1.2 LITERATURE SURVEY

RoslanAbdRahman et al [1]; does stress analysis of heavy duty truck chassis by utilizing a billboard finite element package ABAQUS. To work out juncture in order that intentionally modifications the stresses are often reduces to enhance the fatigue lifetime of components.

CicekKaraoglu et al [2]; Stress analysis on heavy duty truck chassis with riveted joints is performed using a billboard finite element package. ANSYS version 5.3.He investigated the impact of side member thickness and connection plate thickness on length change in this study.

MohdAzizi Muhammad Nor et al [3]; performs a strain analysis on a real low loader structure made up of I-beams for a 35-tonne trailer design. Software CATIA V5R18 is the modelling platform he uses. The study's findings showed that the maximum deflection and maximum stress in the situation correspond to the theoretical maximum position of a straigh t beam under uniform loading distribution.

N.K.Ingole et al [4]; does the modifications in existing model of trailer truck chassis by 1) Cross-sectional areas of cross participants vary. 2) Cross and longitudinal members have different cross sectional regi ons. 3) Cross and longitudinal members have different cross sectional regions. 4) Changing the place of cross members in the chassis' main frames.

Patel et al [5]; have Using Pro-mechanical, we investigated and optimised a chassis configuration for weight reduction of the TATA 2516TC chassis frame. They first used ANSYS Software to calculate the assembly weight, maximum stress, strain, and displacement for the prevailing section of chassis, after which they changed the size of existing C -sections and found all again, concluding that the prevailing "C" sections are better than all other sections in terms of strain, displacement, strain, and shear stress except for the load. For load considerations, the modified "C" section is lighter than the other sections studied in this paper.

Murali et al [6]; have investigated the juncture which has the very best stress using Finite Element Method (FEM). One of the variables that will cause fatigue loss is this juncture. For the modifications and analysis, stiffeners were added to the existing truck frame. In bending analysis, the thickness of the model where the greatest deflection occurs was initially increased to a certain value within a reasonable range. Another cross beam was added in the centre of the wheel base to stiffen the model.

B. Ramana Naik and C. Shashikanth [7] have objective to analyse an automobile chassis for a 10 tonne vehicle. The modelling is done using Pro-E, and analysis is done using ANSYS. The overhangs of the chassis are calculated for the stresses and deflections analytically and are compared with the results obtained with the analysis software. Modal Analysis is also done to find the natural frequency of the chassis and seen that it is above than its excitation frequency. The Theoretical calculations and FE analysis results are compared and it is observed that they are within the material properties.

Kamlesh Y. Patil and Eknath R. Deore [8] have studies the Ladder Chassis frame of TATA 912 Diesel Bus and The model of the chassis was created in Pro-E and analysed with ANSYS for Various Cross Sections for same load conditions. They observed that the Rectangular Box (Hollow) section is more strength full than the conventional steel alloy chassis with C and I design specifications. The Rectangular Box (Hollow) section is having least deflection i.e., 2.683 mm and stress is 127 N/mm2 in all the three type of chassis of different cross section.

2. METHODOLOGY

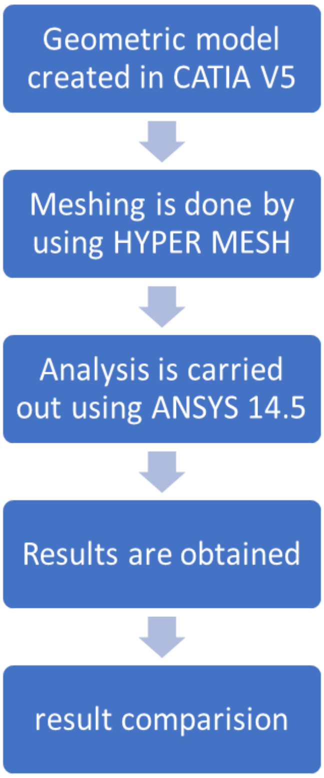

Figure 2: Methodology

2.1. Geometric Model

a) Dimensions of the Model

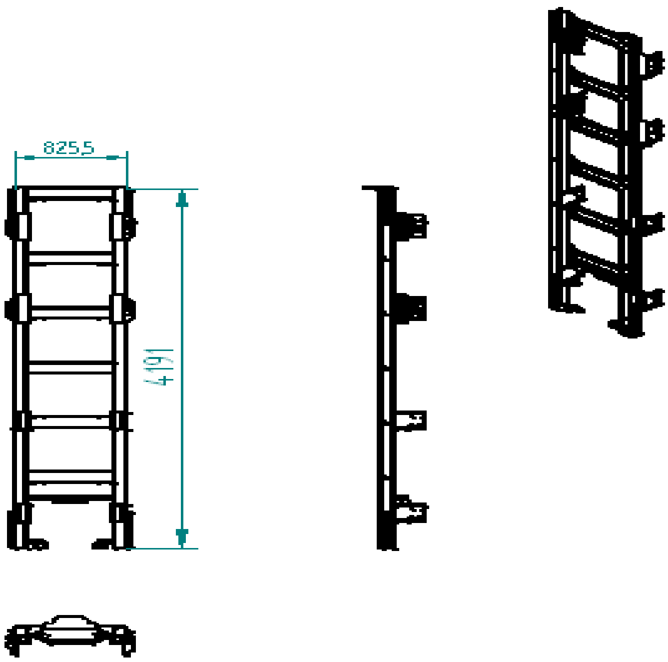

Figure 3: Major Dimensions of the Chassis

The figure 3 shows major dimensions of the developed model. The individual dimensions are represented in the following pictures. Front, top and side views are represented. Also an isometric view is also represented to understand the structure.

b) Geometric Model using CATIA V5

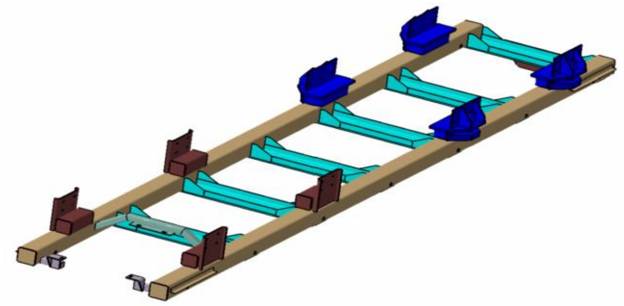

Figure 4: Three Dimensional View1

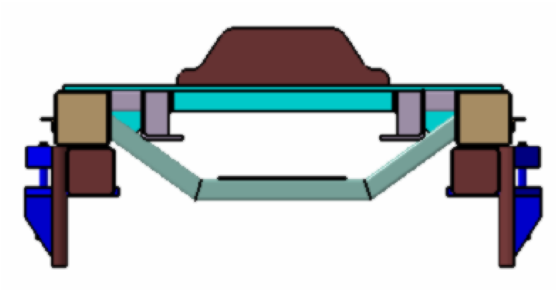

Figure 5: Three Dimensional View2

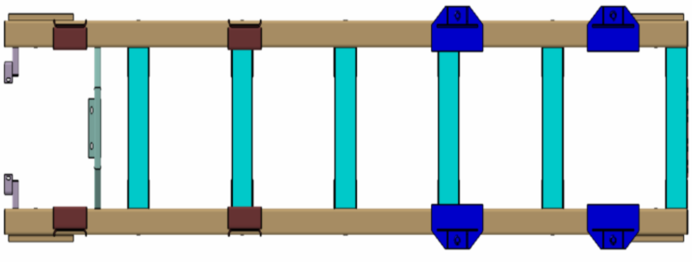

Figure 6: Three Dimensional View3

The figure 4 shows the geometric model of chassis in a three dimensional view and also fig 5 shows the geometrical model of chassis in side view and fig 6 shows the chassis geometrical model in top view. CATIA V5 is used for dimensioning the problem.

2.2. Meshed Model by using Hyper Mesh
2.2.1 Meshing:

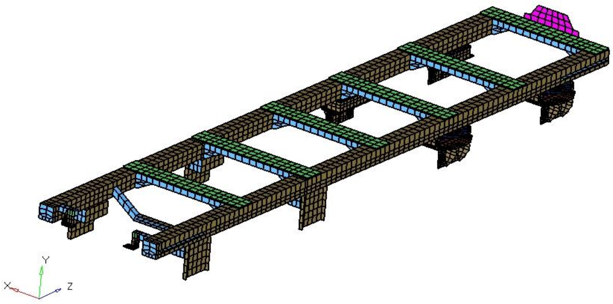

Figure 7: Wing meshed with tetrahedral elements.

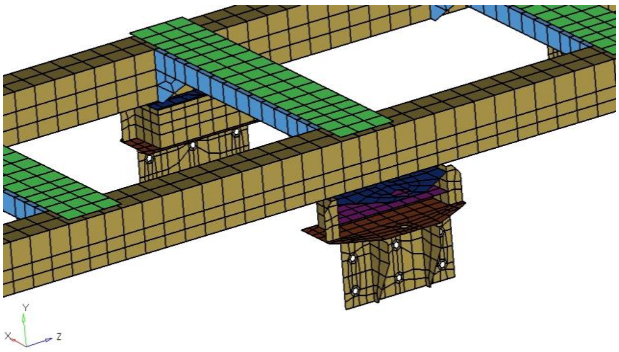

Figure 8: Meshed View of the problem

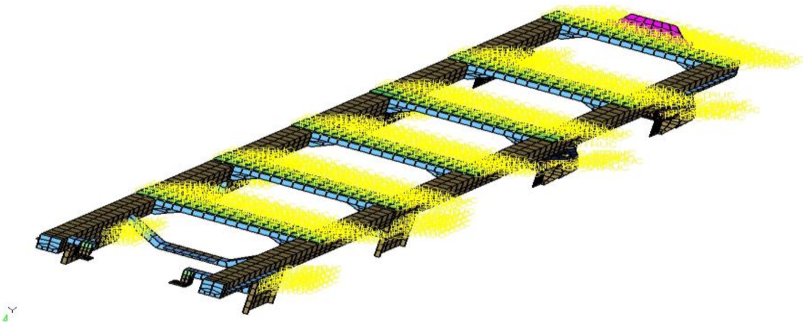

Figure 9: Meshed view with the connections

The figures 7 to 9 shows meshed views of the problem. The number of elements are reduced to execute the problem in Ls-Dyna for impact simulation. 4 nodded shell elements are used for representation of the geometry.

2.2.2 Meshed parameters:

- Type of element = SHELL63
- Total number of elements = 5793
- Total number of nodes = 5932

2.2.3Material:

- Material: Steel
- Steel's Young's Modulus=200Gpa
- Steel's Poison's ratio=0.3
- Density=7800kg/m3.
- Steel's Yield Stress: 350Mpa

3. RESULT AND DISCUSSION

3.1 STRUCTURAL ANALYSIS

Trucks are the most common type of goods transportation. Due to heavy load on the structure, speed limits are applied on the trucks during road curvatures. The trucks are subjected to heavy bending loads and torsion loads during turnings. The following loading conditions are considered for the structural analysis using Ansys.
Case 1: Self Weight analysis
Case 2: Normal running condition (Bending Case)
Case 3: Under humps (Rear wheels on hump and front wheels on ground) (Bending)
Case 4 : Under curvature (Torsion Case)

3.1.1 Self Weight Analysis:

Self-weight analysis is carried out to check the structural strength under normal conditions. The geometry is modelled in meters to get correct results for self-weight. Density and acceleration due to gravity is applied on the structure for self-weight results. The results are as follows.

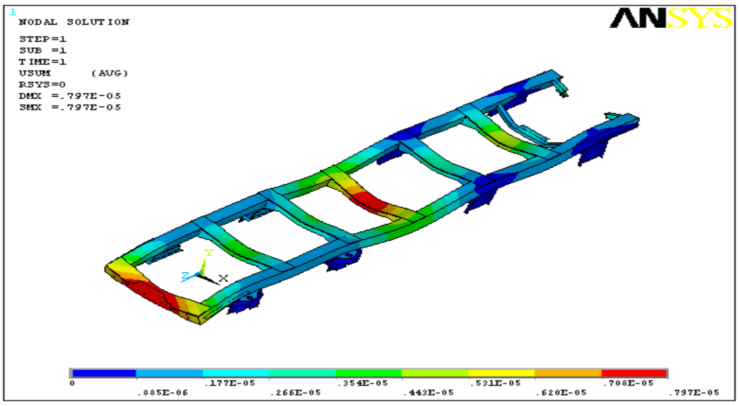

Figure 10: Deflection takes under self-weight

The deflection results shows deflection value of 7.97microns under self-weight. Maximum deflection is observed at the end geometry and at the center of central connection plate. The various colours shows displacement pattern in the structure.

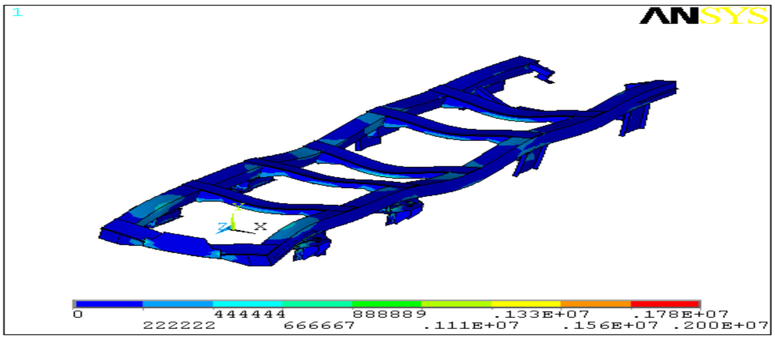

Figure 11: von-misses Stress plot

The figure 11 shows von-misses stress development in the structure due to self-weight. The results shows maximum stress development of 2Mpa as shown in the status bar below the plot.

3.1.2: Normal Running Conditions

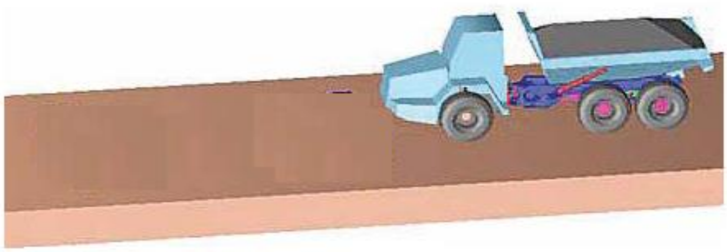

Figure 12: Normal Running Condition

The chassis structure is checked for 15 tons load during the operational conditions along with the self-weight of the member. The load is applied as uniformly distributed load on the longitudinal beams. The results are as follows.

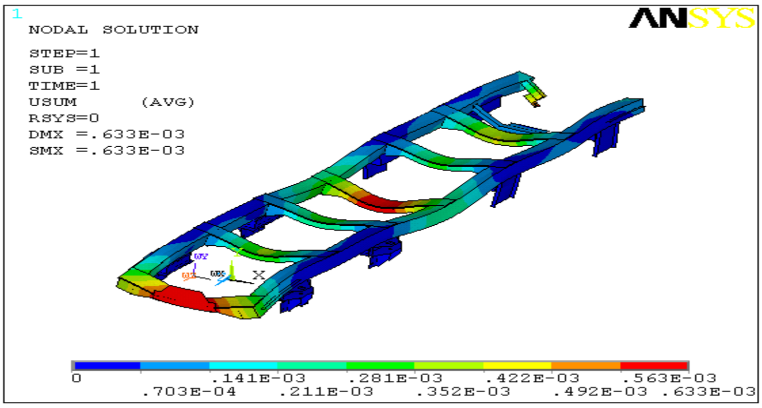

Figure 13: Deflection plot

The figure 13 shows developed deflection in the structure. Maximum deflection is 0.633e-3m or 0.633mm

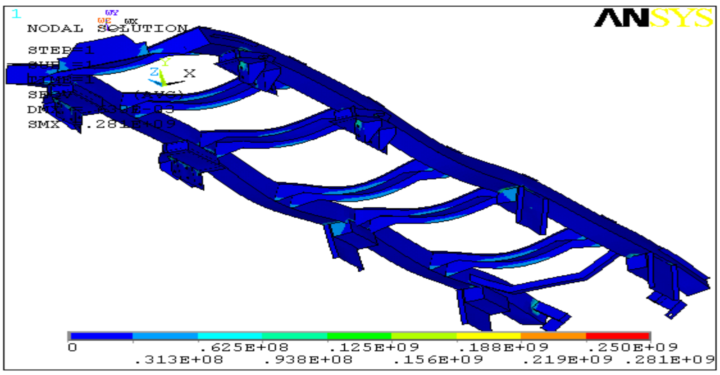

Figure 14: Von-misses stress conditions during fully loaded condition

The figure 14 shows von-misses stress in the structure during fully loaded condition. Maximum stress development is around 281Mpa.

3.1.3 Two trailing wheels on the hump

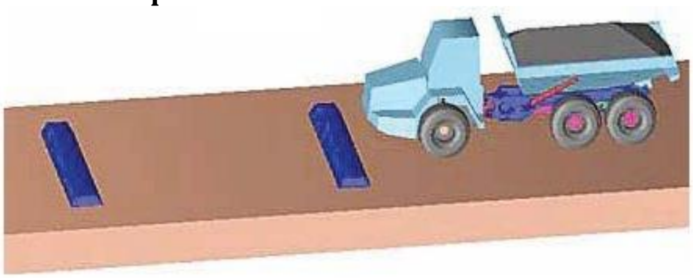

Figure 15: Trailing wheels on the hump

If the back side wheels on the hump, the load distribution is not uniform on the chassis. So, the stresses are find out on the structure for 15tons loading condition.

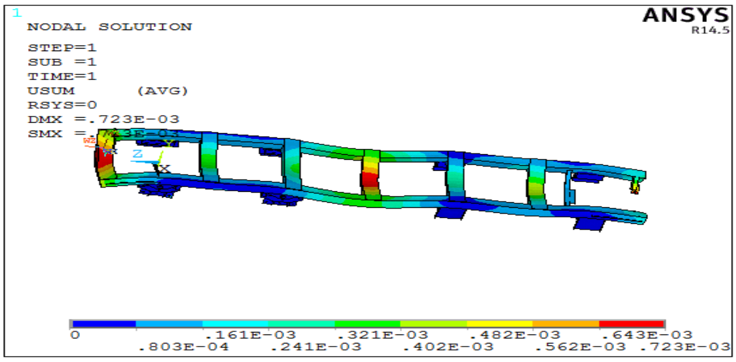

Figure 16: Displacement Plot (Max Displacement: 0.723mm)

The figure 16 Displacement Plot of the trailing wheels on humps, when 15tons weight loaded on truck. The maximum displacement is .723E-03 or 0.723mm.

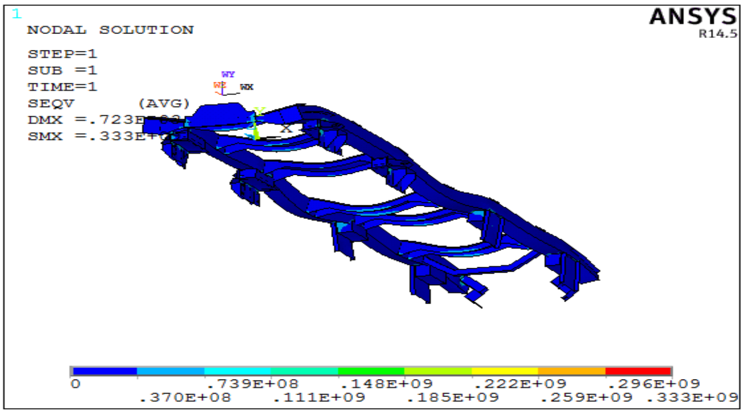

Figure 17: Von-misses Stress Plot (Max Stress: 333Mpa)

The figure 17 Von-misses Stress Plot on the trailing wheels on the humps. The maximum stress is .333E+09 or 333Mpa as

3.1.4 While taking the curve (Torsion load on the structure – 50 KMPH)

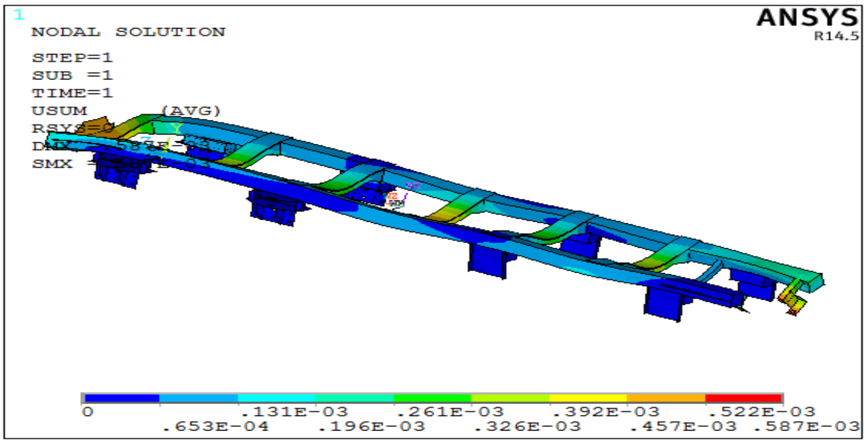

Figure 18: Deflection plot (Maximum Deflection: 0.587mm)

The figure 18 Deflection plot when truck taking a curve, at torsion load on the structure-50kmph. The maximum Deflection is .587E-03 or 0.587mm

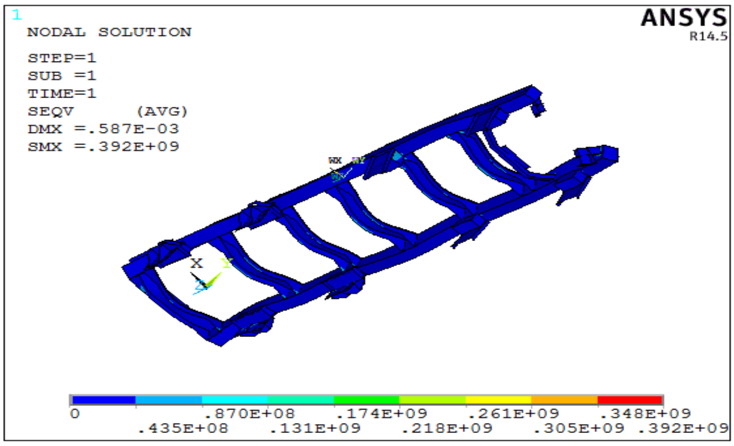

Figure 19: von-misses Stress Plot (Maximum Stress: 392 Mpa)

The figure 19 shows stress exceeding the allowable stress of the material. So structure should be checked for stress condition on various members to identify design changes for the geometry.

3.2 Graphical representation of the structural analysis

1) Deflection v/s self-weight, normal running condition, under hump, one wheel of rear axle on hump and other on ground, under curvature.

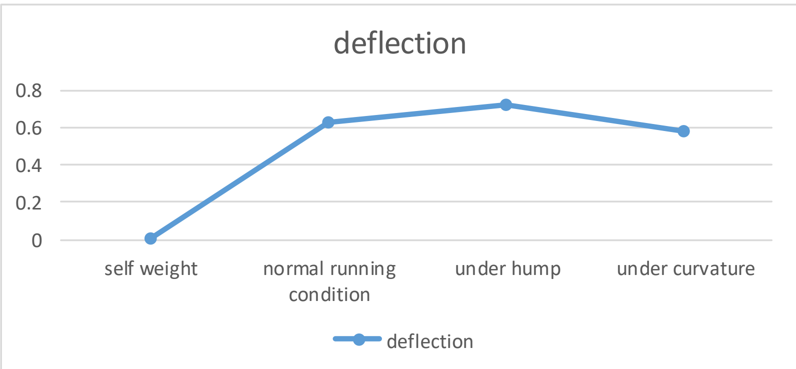

Figure 20: Deflection v/s cases of structural analysis

2) Von misses stress v/s self-weight, normal running condition, under hump, one wheel of rear axle on hump and other on ground, under curvature.

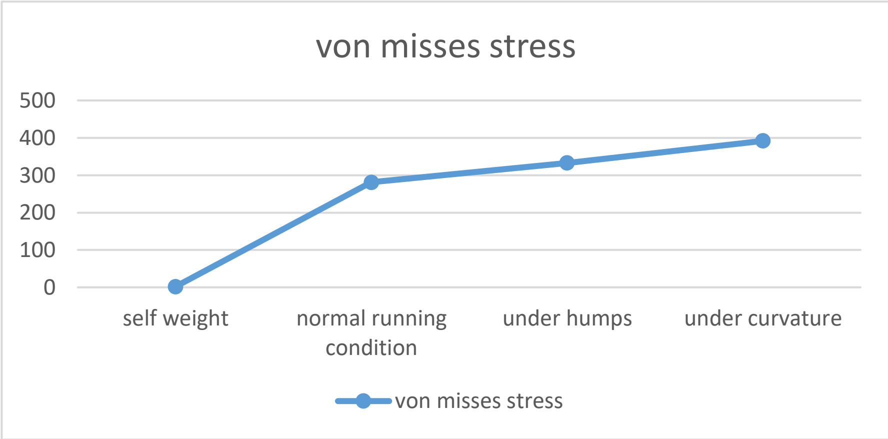

Figure 21: Von misses stress v/s cases of structural analysis

3.3 LS-DYNA SIMULATION RESULTS

Ls-Dyna simulation is used to find the structural behavior of the members. A free analysis is carried out to find the modal nature of the problem. The modal frequencies and corresponding mode shapes are as follows. The modal analysis is carried out to check the resonance condition of the problem.

Mode 1

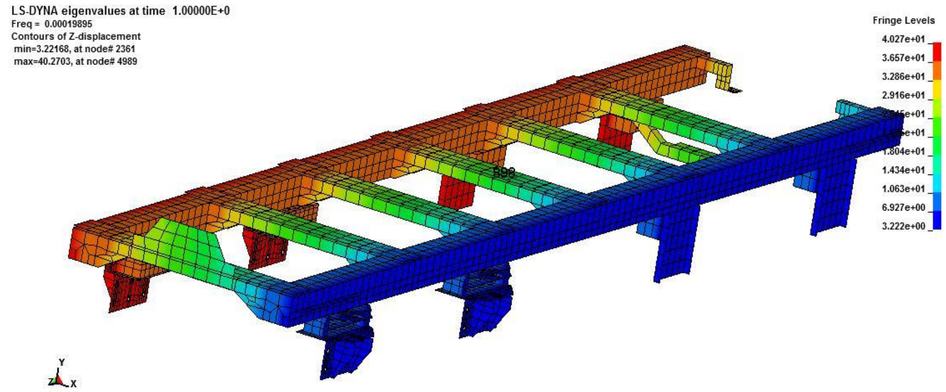

Figure 22: Mode Shape for Frequency (0.00019895Hz)

Figure shows the Mode shape of truck chassis having natural frequency 0.00019895. The maximum deformation is 40.2703mm, showing red in colour and minimum deformation is 3.22168mm.

Mode 2

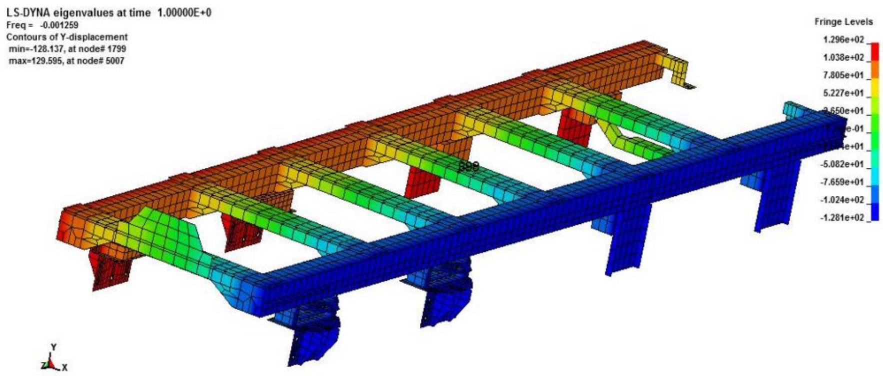

Figure 23: Mode shape for Frequency (0.001259Hz)

Figure shows the Mode shape of truck chassis having natural frequency 0.001259. The maximum deformation is 129.595mm, showing red in colour and minimum deformation is -128.137mm.

Mode 3

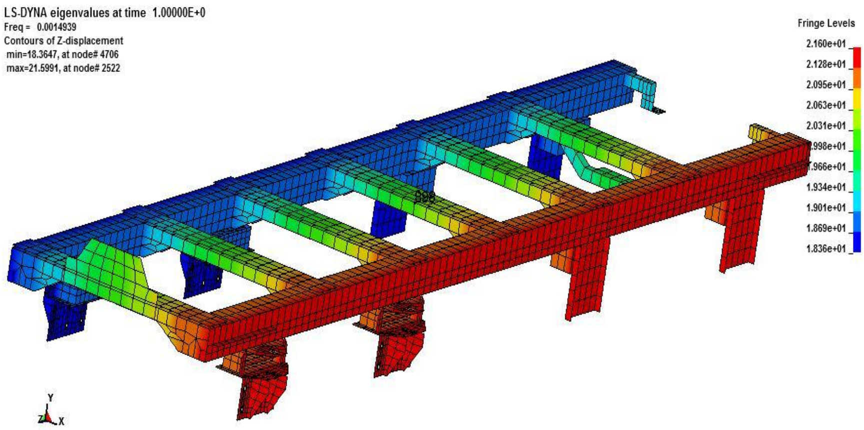

Figure 24: Mode Shape for Frequency (0.0014939Hz)

Figure shows the Mode shape of truck chassis having natural frequency 0.0014939. The maximum deformation is 21.5991mm, showing red in colour and minimum deformation is 18.3647mm.

Mode 4

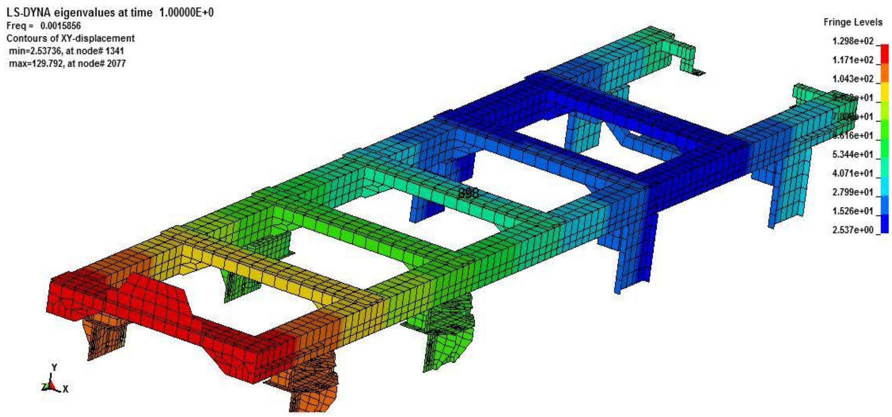

Figure 25: Mode Shape for Frequency (0.0015856Hz)

Figure shows the Mode shape of truck chassis having natural frequency 0.0015856. The maximum deformation is 129.792mm, showing red in colour and minimum deformation is 2.53736mm.

Mode 5

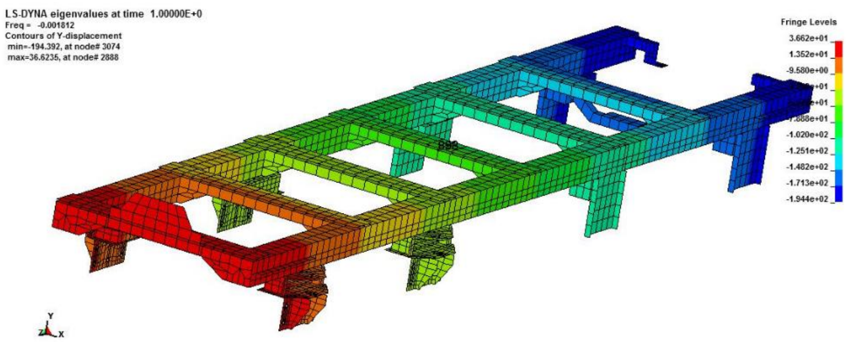

Figure 26: Mode shape for Frequency (0.001812Hz)

Figure shows the Mode shape of truck chassis having natural frequency 0.001812. The maximum deformation is 36.6235mm, showing red in colour and minimum deformation is -194.392.

Mode 6

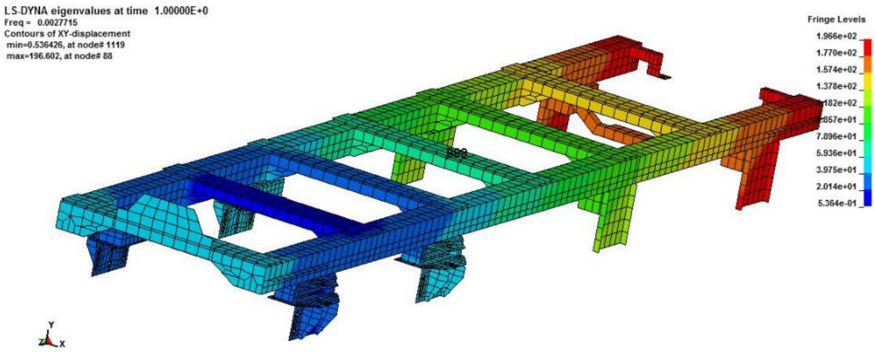

Figure 27: Mode Shape for Frequency (0.0027715Hz)

Figure shows the Mode shape of truck chassis having natural frequency 0.0027715. The maximum deformation is 196.602mm, showing red in colour and minimum deformation is 0.536426mm.

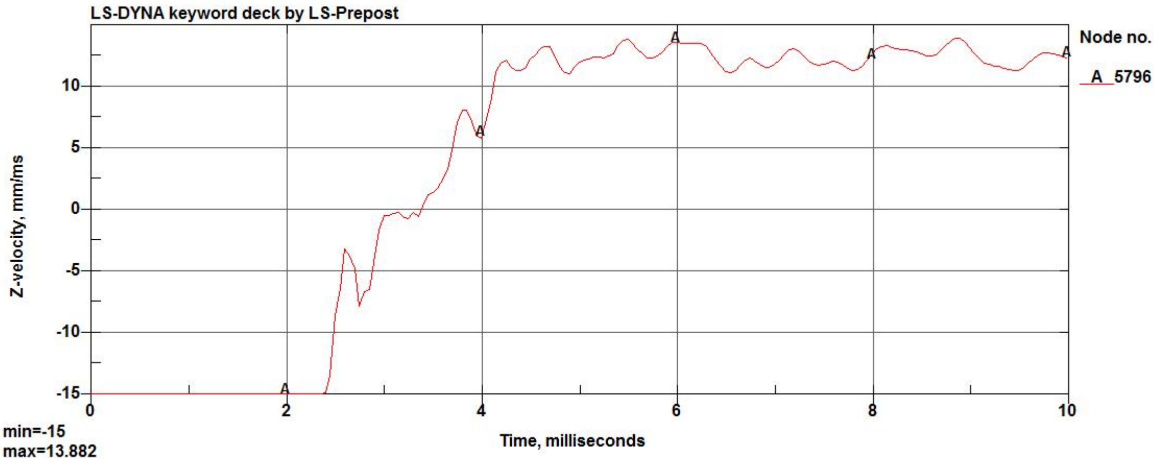

Figure 28: Velocity to Time

The figure shows velocity change with reference to time. The velocity is changing the sign during crash simulation. The node number considered for simulation is also represented.

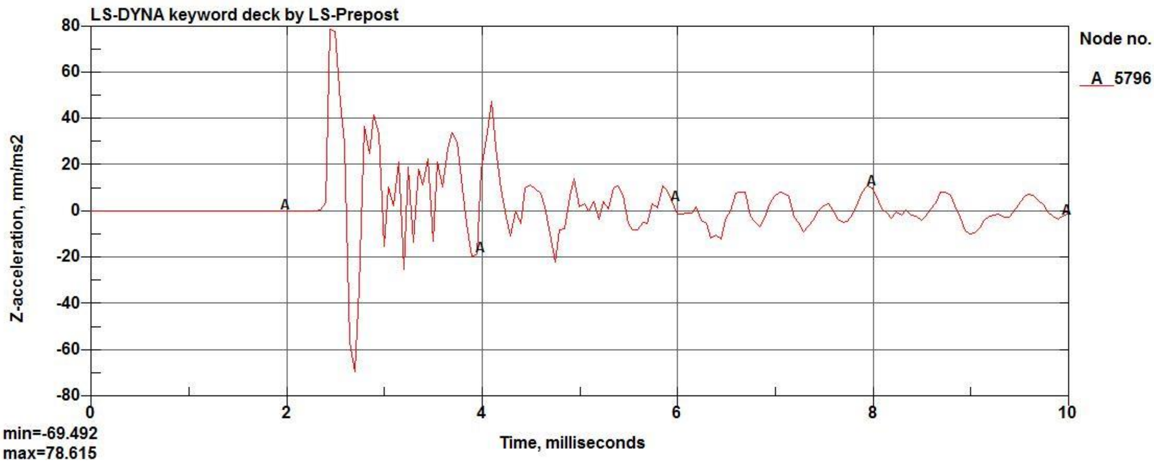

Figure 29: Acceleration plot of the Impact

The figure shows acceleration due to the impact of the problem. Acceleration plot is considered at node number 5796.

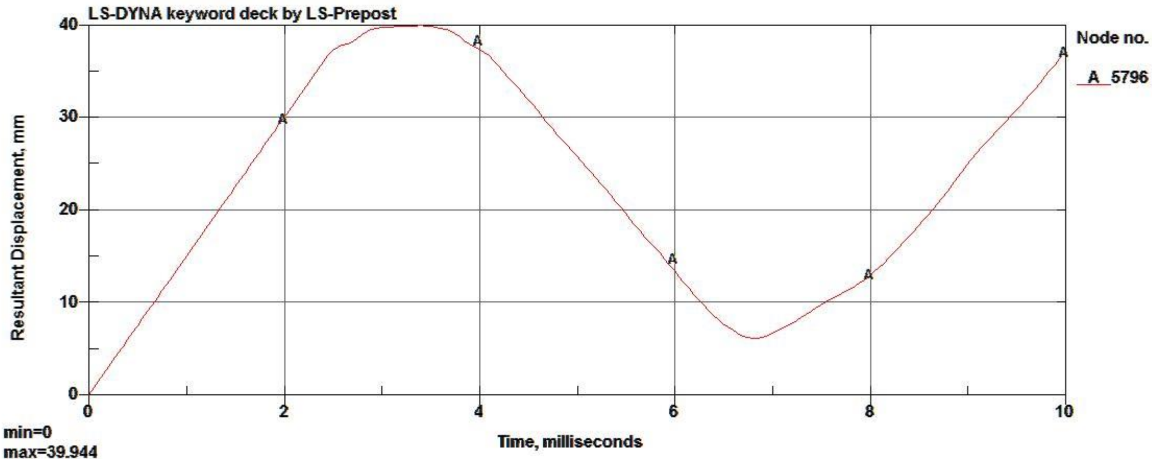

Figure 30: Displacement plot

The figure33 shows displacement plot in the problem. Maximum displacement is around 39.944mm. The peak displacement is observed at 3.2 milliseconds.

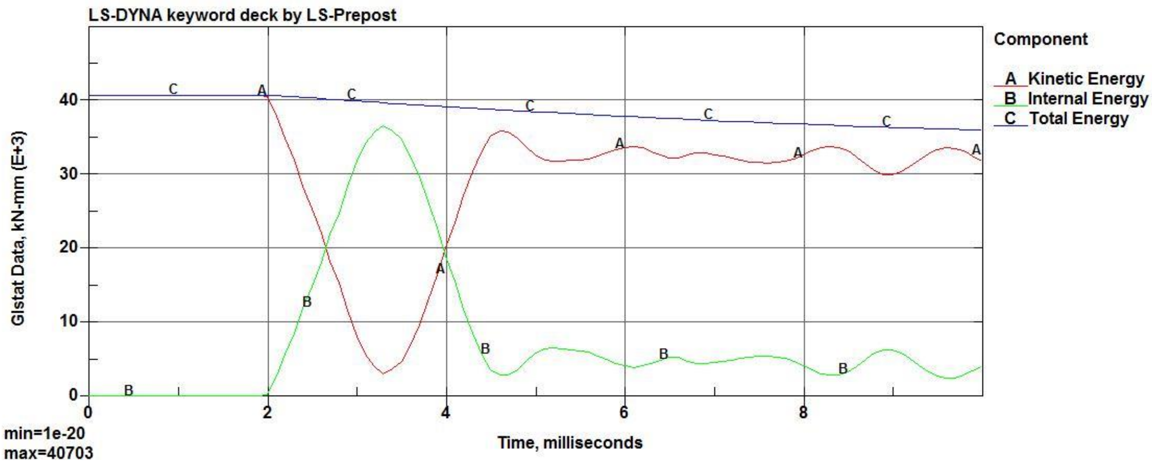

Figure 31: Energy plots

The figure34 shows energy plots in the impact simulation. The graphs shows variation of kinetic energy, Internal energy and total energy in the system. Between 2 4 millisecond increases of internal energy can be observed along with the drop of kinetic energy. So Ls-Dyna simulation helps to predict the stored energies in the system. The stored strain energy is the cause of development of von-misses stress. Von-misses stress is the main stress to find the safety of the structure.

4. CONCLUSIONS & FURTHER SCOPE

4.1 Conclusions:

The chassis is a crucial member of automobile assembly. It supports all the members. The designed structure should take all type of loads during operational and stationary conditions. An analysis has been carried out to check the critical regions of the chassis structure using implicit ansys and explicit Ls-Dyna solver. The overall summary is as follows.

Initially a chassis structure for heavy truck is modelled using Catia software. Catia solid modelling and sketcher options are extensively used to build the geometry.

Initially the chassis is checked for structural conditions. It has been tested for static or self-weight condition. Later the chassis is checked for normal running condition. The results shows complete safety of the problem for the given loads. Both the deformation and stresses are within the allowable limits of the material.

Further analysis has been carried out for different loading conditions. Total 5 cases of static loading is considered. The results are represented for all the load conditions. The results shows higher stresses in the last case (Torsion) where the stresses are exceeding the allowable limits of the material.

Finally, with the change of wheel bracket material, all results show that the issue is completely safe for the given loading conditions.

4.2 Further Scope:

The analysis can be carried out under thermal environment
The pressure contours due to wind load can be determined using CFD simulation.
Complete structure design optimization can be carried out
It is possible to carry out topology optimization.
The structure can be subjected to a spectrum load analysis.

REFERENCES

[1] RoslanAbdRahman, MohdNasirTamin, OjoKurdi "Stress analysis of heavy duty truck chassis as a preliminary data for its fatigue life prediction using FEM" JurnalMekanikal December 2008, No. 26, 76 – 85.
[2] 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.
[3] MohdAzizi Muhammad Nora,, HelmiRashida, Wan MohdFaizul Wan Mahyuddin "Stress Analysis of a Low Loader Chassis" Elsevier Ltd. Sci Verse Science Direct Procedia Engineering 41 (2012) 995 – 1001.
[4] JadavChetan S., PanchalKhushbu C., Patel Fajalhusen "A Review of the Fatigue Analysis of an Automobile Frames" International Journal of Advanced Computer Research, Volume-2 Number-4 Issue-6 December-2012, (ISSN (print): 2249-7277 ISSN (online): 2277-7970).
[5] 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.

BIOGRAPHIES

Mr. Naveen G, holds Bachelor's degree in Mechanical Engineering from Moodlakatte Institute of Technology, Kundapur (udupi) and is currently pursuing Master's in Machine Design from Dr. AIT, Bengaluru.

Dr.Prashanth A S is an Assistant Professor of Mechanical Engineering at Dr. AIT, Bengaluru.