Design and Structural Analysis of Chassis

[1]B. Ramya, [2]Sridhar Reddy, [3]Dr. C. Udaya Kiran
[1]PG Student, [2]Associate Professor, [3]Professor and Head of the Department
[1]Mechanical Engineering, J B Institute of Engineering and Technology, Hyderabad, India

---

***Abstract* - A Vehicle is a Structural Assembly which consists of many components coupled together to make the vehicle to run on initial conditions as well as under various load conditions. Vehicles are designed to carry loads like trucks, buses and cars. Different types of vehicles with wide variety of its applications are present in the market which capacity varies from 1Tonne to 40Tonnes and more. The present scenario in automotive industry is an increase in demand of trucks not only on the cost and weight aspects but also on improved complete vehicle features and overall work performance. The chassis plays an important role in the design of any truck. The chassis design in general is a complex methodology and to arrive at a solution which yields a good performance is a tedious task. Since the chassis has a complex geometry and loading patterns, there is no well-defined analytical procedure to analyze the chassis. So the numerical method of analysis is adopted, in which Finite Element Technique is most widely used method. Vehicle chassis is an important part which supports the major load of the vehicle assembly. As vehicle chassis plays a vital role, its design has to be subjected to Structural Analysis to validate against all the possible cases of load applications and failures to strengthen the design.**

A Truck's chassis Frame forms the structural backbone of a commercial vehicle. The main function of the truck chassis frame is to support the components and payload placed upon it. When the truck travels along the road, the chassis is subjected to vibration induced by road roughness and excitation by vibrating components mounted on it. This paper presents the study of the vibration characteristics of the truck chassis that include the natural frequencies and mode shapes. The responses of the truck chassis which include the stress distribution and displacement under various loading condition are also observed. The method used in the numerical analysis is finite element technique. The results show that the road excitation is the main disturbance to the truck chassis as the chassis natural frequencies lie within the road excitation frequency range. The mode shape results determine the suitable mounting locations of components like engine and suspension system. Some modifications are also suggested to reduce the vibration and to improve the Strength of the truck chassis.

I. INTRODUCTION

Chassis is a French term and was initially used to denote the frame parts or Basic Structure of the vehicle. It is the back bone of the vehicle. A vehicle without body is called Chassis. The components of the vehicle like Power plant, Transmission System, Axles, Wheels and Tyres, Suspension, Controlling Systems like Braking, Steering etc., and also electrical system parts are mounted on the Chassis frame. It is the main mounting for all the components including the body. So it is also called as Carrying Unit.

The frame supports the cab, engine, transmission, axles, and the various other chassis components. The cross-members control axial rotation (another way of saying twist) and longitudinal motion of the rails. They reduce torsional stress transmitted from one rail to the other. Cross-members also are used for vehicle component mounting and protecting the wires, hoses, and tubing that are routed from one side of the vehicle to the other.

The following main components of the Chassis are:

- Frame: it is made up of long two members called side members riveted together with the help of number of cross members.
- Engine or Power plant: It provides the source of power
- Clutch: It connects and disconnects the power from the engine fly wheel to the transmission system.
- Gear Box
- U Joint
- Propeller Shaft
- Differential

Functions of the Chassis Frame

1. To carry load of the passengers or goods carried in the body.
2. To support the load of the body, engine, gear box etc.,
3. To withstand the forces caused due to the sudden braking or acceleration
4. To withstand the stresses caused due to the bad road condition.
5. To withstand centrifugal force while cornering

II. PROBLEM DEFINITION

The scope of the Project is to Design and Analyze a Typica Chassis Frame (and to perform optimization process if required to produce an Optimistic Design). To evaluate whether the design is safe design or not.

III. WORK METHODLOGY

3.1 Model Geometry

The given 3D model was modified in CATIA, 3D design software to suit the FEA requirement i.e. very small edges and holes are disfeatured. The given geometry is shown in the Fig.1 below.

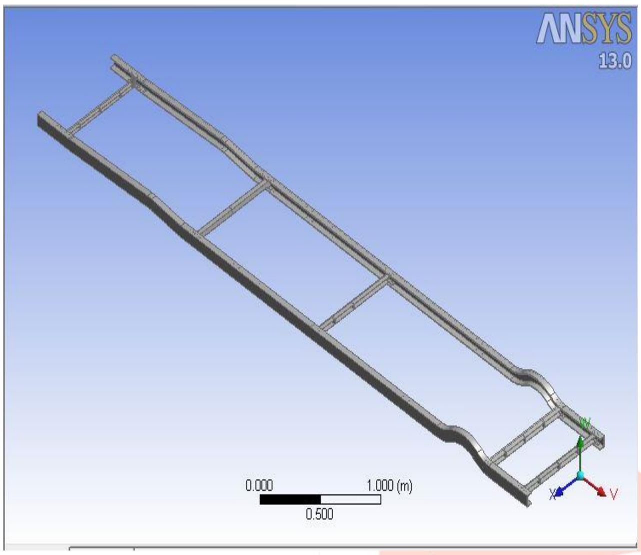

Fig 3.1: Geometric model of Chassis

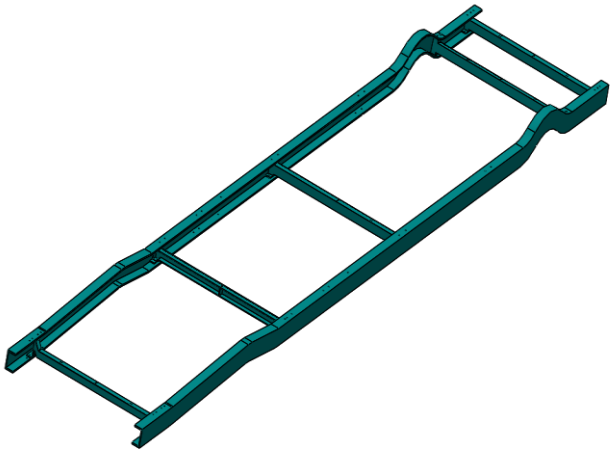

Fig 3.2: Truck Chassis Frame Along With Cross Members

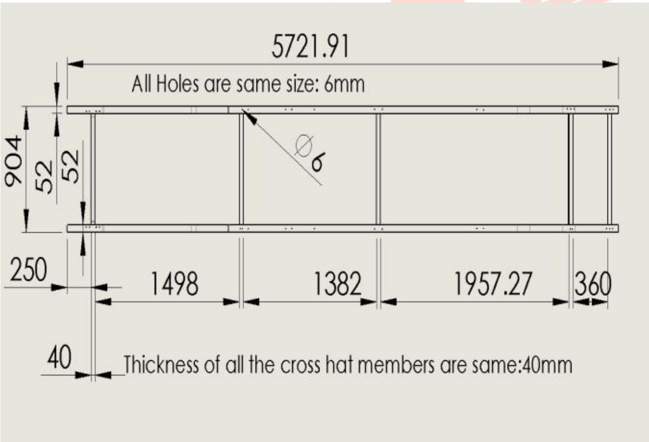

Fig 3.3: Top view of Chassis

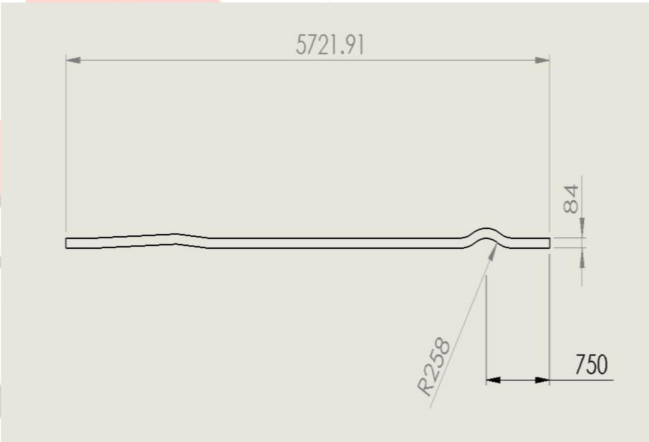

Fig 3.4: Front view of Chassis

3.2 Typical Fem Procedure to Carry Out This Analysis

The truck chassis was modelled by using 10-noded tetrahedral (Tet-10) solid elements. Experimental and numerical studies on a simple hollow rectangular straight beam suggested that Tet-10 element can mesh the truck chassis geometry and give accurate results of the natural frequencies and the mode shapes. There are two types of analysis carried out; normal mode analysis to determine the natural frequencies and the mode shapes, and the linear static stress analysis to look into the stress distribution and deformation pattern of the chassis under static load. For normal mode analysis, the chassis model was meshed with 15648 Tet-10 elements and 32846 nodes, while for linear static analysis, 37014 Tet-10 elements and 77548 nodes were used.

The boundary conditions are different for each analysis. In normal mode analysis, free-free boundary condition will be applied to the truck chassis model, with no constraint applied to the chassis model. For static analysis, the boundary condition used is to represent the real operation environment of the truck chassis. The pinned boundary condition is applied to the suspension mounting-bracket of the chassis since these locations do not allow any translation but allow only rotation about the axis.

In normal mode analysis, no load is applied to the truck chassis model. In static mode, when the truck is stationary, the loads from the weights of the components like cab, engine, gear box, fuel tank, exhaust and payload is applied to the mounting brackets of the components. Mean while the pinned constraints are applied to the suspension mounting brackets. In a symmetry load case which simulates both the truck's front wheels hitting a hump simultaneously, the resulting load is applied to the front suspension mounting brackets. The pinned constraints are applied to the rear suspension mounting brackets. In asymmetry load case which simulates one of the front truck's wheel hitting a hump, the resulting load are applied to the front left suspension mounting brackets with other suspension mounting brackets constraint from any translation (pinned).

1. Dividing the solid model into number of elements and then connecting these elements with each other meshing and thereby obtaining the finite element model.
2. Applying boundary conditions.
3. Solving the problem.

Table 3.1: Material Properties

Material	4130 Steel Alloy
Young's Modulus (GPa)	205
Poisson's ratio	0.29
Density (Kg/m^3)	7850
Yield strength (MPa)	435
Ultimate Tensile Strength (MPa)	670

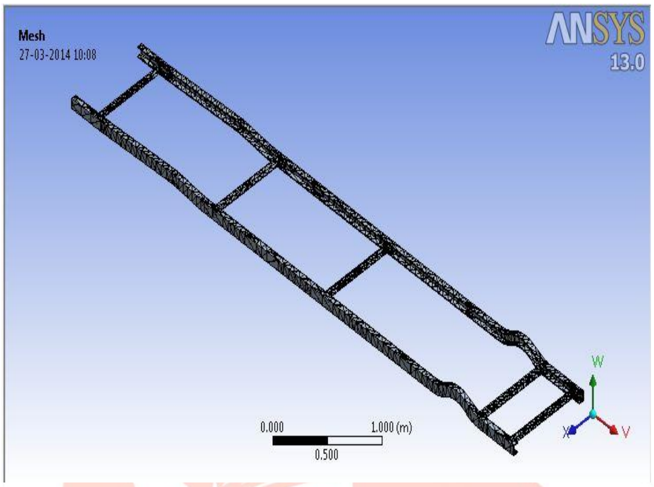

Fig: 3.5: Meshed model

Table 3.2: Mesh Details

S.NO	Element name	Element order	Number of elements
1	Tetrahedral	Quadratic	37014

3.3 Loads and Boundary Conditions

Static Analysis

Static analysis calculates the effect of steady load conditions on a structure while ignoring inertia and damping effects, such as those caused by time varying loads. A static analysis therefore can include steady inertia loads (such as gravitational and rotational velocity). Static analysis is used to determine the displacements, stresses, strains and forces in the structures or components caused by loads that do not induce significant inertia and damping effects. The kind of loading that can be included in static analysis are

1. Externally applied forces and pressure
2. Steady state inertia forces (such as gravity and rotational velocity)

Loads are applied at 3 locations as cumulative loads

Load1: Cab Weight (250Kg) + Engine (150Kg) + Payload1 (417*2=834Kg)+Chassis Wt (200/3=60Kg) =1294Kg
Load2: Gear Box (50Kg) + Chassis Wt (70) + Payload (834) + Fuel Tank (40Kg) =994Kg
Load3: Payload (834) + Chassis Wt (70) + Exhaust (20kg) =924Kg
Total Load as =3212Kg

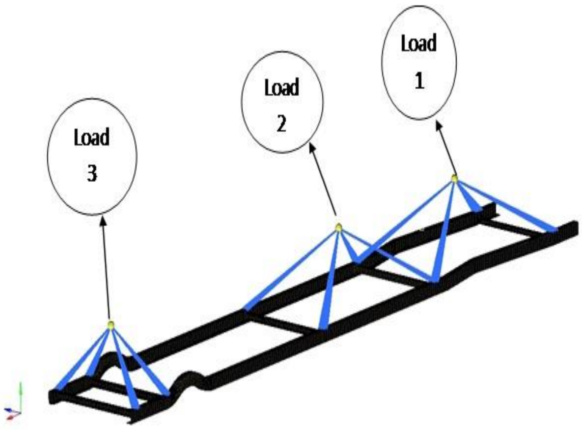

Fig 3.6: Applying Loads

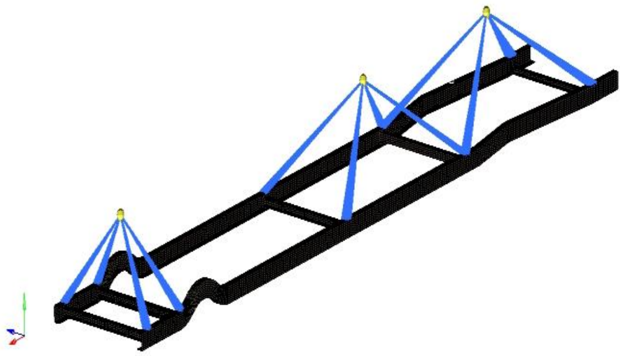

Fig 3.7: Boundary Conditions

IV. RESULTS AND DISCUSSIONS

4.1 Static Analysis

After applying the boundary conditions, the problem was solved by the ANSYS Iterative Solver. ANSYS solver formulates the governing structural stress strain equations for each and every element and these formulated governing equations were solved for the deformations from which all the other quantities such stresses, strains etc can be calculated.

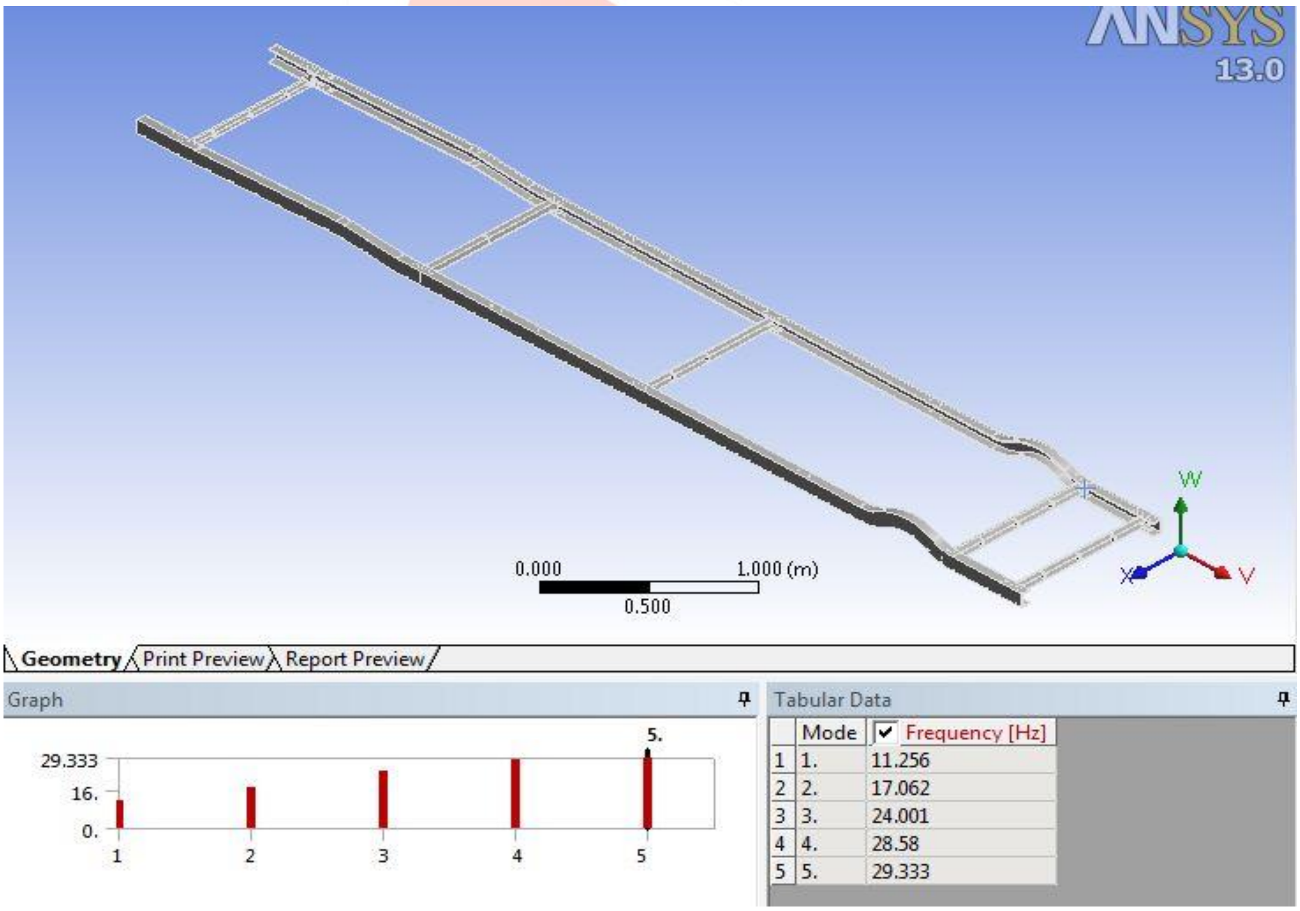

Fig 4.1 Vibrational Failure

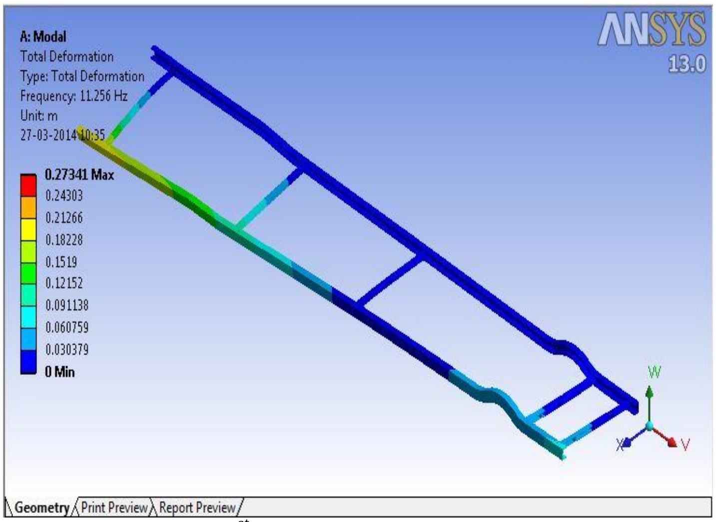

Fig 4.2 Deformation 1st Mode
Frequency is 11.256Hz and
Maximum displacement is 0.27341m for Fig 4.2

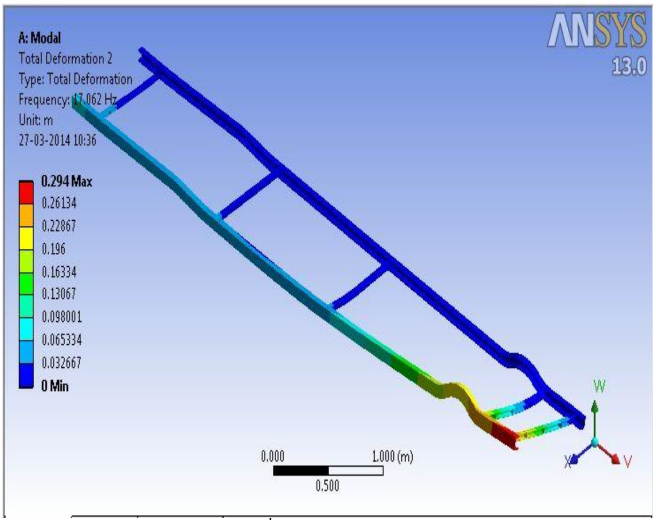

Fig 4.3 Deformation 2nd Mode
Frequency is 17.062Hz and
Maximum displacement is 0.294m for Fig 4.3

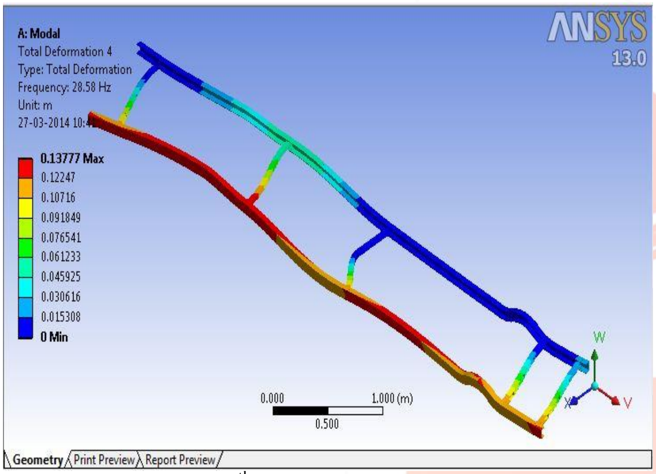

Fig 4.5 Deformation 4th Mode
Frequency is 28.58Hz and
Maximum displacement is 0.13777m for Fig 4.5

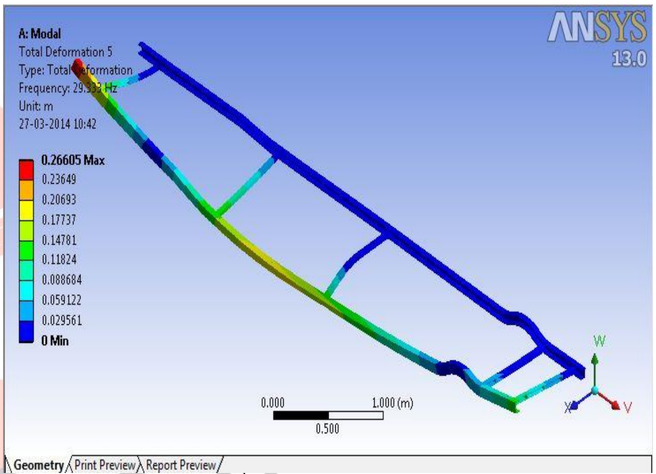

Fig 4.6 Deformation 5th Mode
Frequency is 29.333Hz and
Maximum displacement is 0.26605m for Fig 4.6

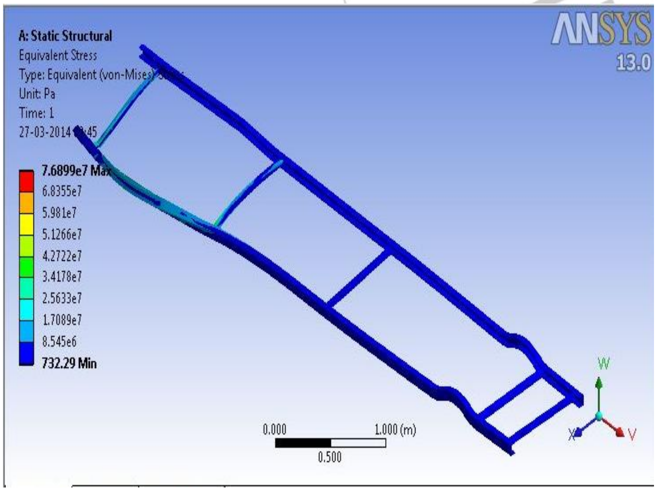

Fig 4.7: Stress

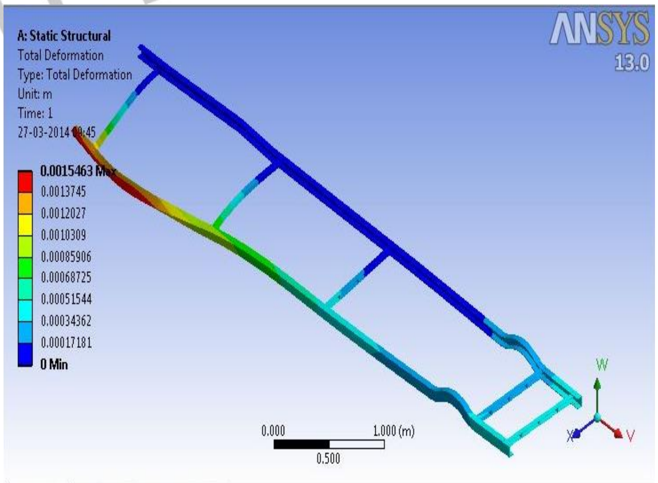

Fig 4.8: Displacement

4.2 Comparison Of Results

Static Analysis:

Loads applied are all applicable loads on vehicle chassis (Engine weight, Chassis weight and other accessories weights as mass elements) along with Gravity.

– The vonmises stress value obtained is **76.89MPa** which far less than the Yield strength value (**435Mpa**).

– Maximum Displacement is **1.546mm** which is very less compared to the dimensions of the chassis.

Frequency Analysis:
Following tabular column shows values obtained from frequency analysis
.

S.No	Mode	Frequency (Hz)	Displacement
1	1	11.25	0.273
2	2	17.06	0.294
3	3	24	0.183
4	4	28.58	0.137
5	5	29.33	0.266

V. CONCLUSION

In our design the Maximum Stress is 76.89MPa where its Yield Stress is 435Mpa, and the Factor of Safety obtained is 5.65 [Factor of Safety= Yield Stress/ Maximum Stress induced], So the design is safe side. And the natural frequencies of the structure are in a better range where we can stiffen the structure by adding some ribs at specific locations.

VI. FUTURE SCOPE

As we have a safe factor of safety of 5.65 which shows it as a safe design and we can reduce the FOS to a lower optimum value. We can optimize the dimensions and mass of the chassis by reducing the dimensions and removing the excess material from low stress concentration areas through iterative process, Authors and Affiliations

REFERENCES

[1] "Fundamentals of Finite Element Analysis" by David V. Hutton
[2] Applied Finite Element Analysis by "Rama Murthy"
[3] Finite Element Methods by "Chandrupatla"
[4] ASME Internationals Journals
[5] http://matweb.com (Material Library)
[6] http://unitconversion.com