Scientific Journal of Impact Factor (SJIF): 4.72
e-ISSN (O): 2348-4470
p-ISSN (P): 2348-6406

International Journal of Advance Engineering and Research Development

Volume 4, Issue 10, October -2017

TOPOLOGY OPTIMIZATION DESIGN AND FINITE ELEMENT ANALYSIS OF HEAVY DUTY TRUCK CHASSIS FRAME

P.Ravi[1], M.Kumara swamy[2]

[1]*Mechanical Engineering, JNTU Kakinada*
[2]*Mechanical Engineering, JNTU Kakinada*

Abstract- *After a careful analysis of different factors like stress values and deformation etc. by varying cross sections for modeling and analysis. The design and Structural analysis of the heavy vehicle chassis with constraints of maximum stress and deflection of chassis under variousloads. In this paper Design and structural analysis of heavy vehicle chassis (TATA 2518TC)is carried out by considering three different cross-sections, namely C, I, and hollow rectangular (Box) type cross sections subjected to the same conditions, made of four different materials viz., Steel ST37, boron/Epoxy, Carbon/Epoxy, E-glass/Epoxy composites. A three dimensional solid Modelis generated using CATIA and analyzed in ANSYS 15.0. The numerical results obtained are validated with analytical values considering the stress distribution and deformation. The results are then compared to finalize the best among three cross sections. Boron/epoxy with Box section has the minimum deflection and Stress is observed.*

Keywords- *Heavy truck chassis frame, CATIA, ANSYS, FEM, Stress, and Deformation.*

I. INTRODUCTION

The chassis is the backbone of vehicles and integrates the main component systems such as the axles, suspension, and power train and is usually subjected to the weight of cabin, its content, and inertia forces arising due to roughness of road surfaces etc. (i.e. static, dynamic and cyclic loading). It is the most crucial element that gives strength and stability to the vehicle under different conditions. The frame should be strong enough to withstand shock, twist, vibrations and other stresses.The chassis is subjected to stress, bending moment and vibrations due to road roughness and components that mounted on it. To overcome this failure chassis requires appropriate strength, stiffness and fatigue properties of the components to be able to withstand these loads or stresses.
The chassis frame consists of side members attached with a series of cross members Stress analysis using Finite Element Method (FEM) can be used to locate the critical point which has the highest stress.Weight reduction is now the main issue in automobile industries without losing the efficiency.The increase in weight not only causes increase in fuel consumption but also the cost of a vehicle.

1.1. Functions of the chassis

- To carry load of the passengers or goods carried in the body.
- To support the load of the body, engine, gear box etc.,
- To with stand the forces caused due to the sudden braking or acceleration.
- To with stand the stresses caused due to the bad road condition.

1.2 .Chassis Frame Cross Sections:

Different cross section namely C, I and box that are used for chassis frames and their salient features are presented below.

1. "C"-Channel sections

It has good resistance to bending, used in long section of the frame.

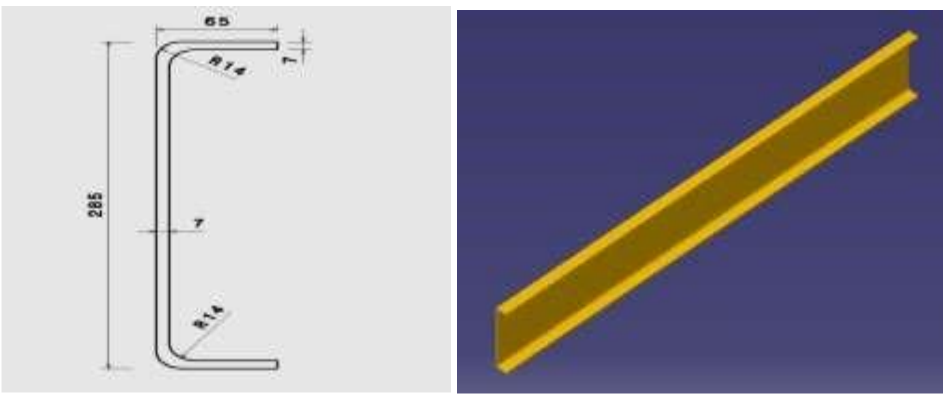


Fig: *1* ***"C"****-Channel sections*

2. "I"- Sections

It has good resistance to both bending and torsion. Due to clamping reason generally "I" section is not used for the practical use.

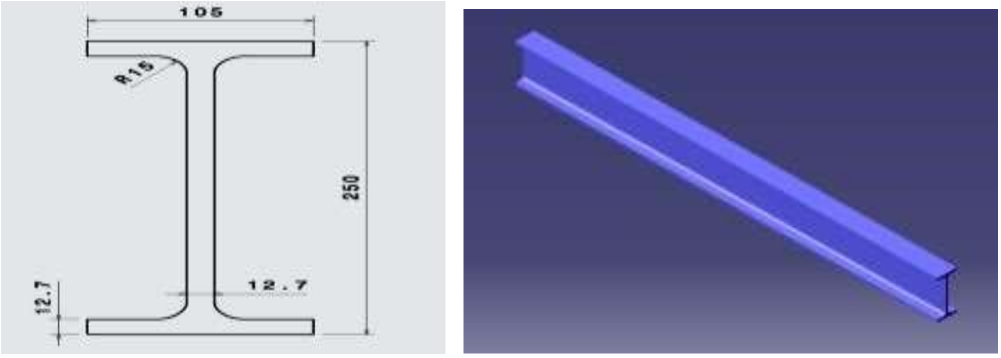
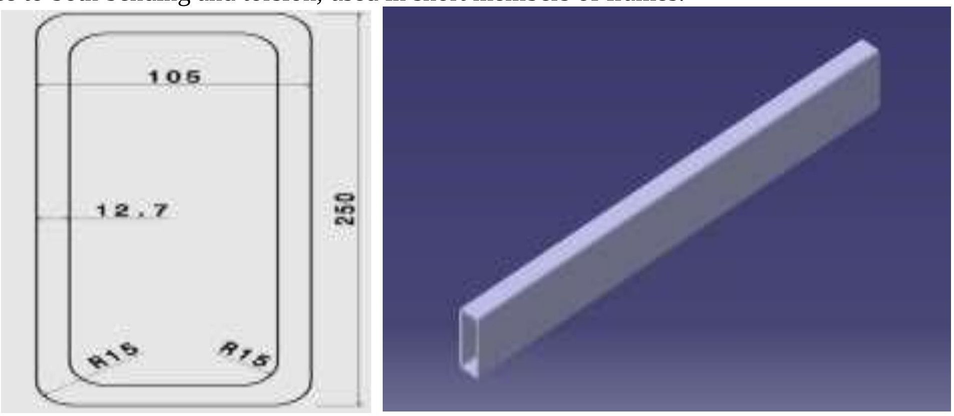


Fig: *2 "I"- Sections*

3. Box sections

It hasgood resistance to both bending and torsion, used in short members of frames.



\ **Fig:** 3Box- Sections

Table: 1 Physical Propertiesof various materials

Material	Modulus of Elasticity (GPa)	Density(kg/m^3)	Poisson Ratio
Steel ST37	210	7850	0.29
Carbon/Epoxy	181	1600	0.28
E-glass/Epoxy	38	1900	0.26
Boron/Epoxy	204	2480	0.23

II. ANALYTICAL CALCULATIONS

In this paper the chassis of heavy vehicle TATA LPT 2518 TC Truck is considered for analysis purpose.
Specification of Existing Heavy Vehicle TATA LPT 2518 TC Truck Chassis frame are shown in table: 2

Table: 2

S. No.	Parameters	Value
1	Total length of the chassis	9010 mm
2	Width of the chassis	2440 mm
3	Wheel Base	4880 mm
4	Front Overhang	1260 mm
5	Rear Overhang	2155 mm
6	Ground Clearance	250 mm
7	Capacity (GVW)	25 ton
8	Kerb Weight	5750 Kgs
9	Payload	19250 Kgs

Fig.4.Shows the live model of the Truck under consideration i.e. TATA LPT 2518 TC

Fig4. TATA LPT 2518 TC Truck Chassis

Loading Conditions

Beam with Shock Absorber and Leaf Spring. Chassis frame may be considered as a continuous Beam .Length of the Beam is 9010 mm.Load (Kerb weight+ Payload) acting on Entire span of the beam is 153281 N. Uniformly Distributed Load acting on the chassis frame is 153281 / 9010 = 17.0 N/mm.

To calculate stress and Deformation first the bending moment and shear force are to be calculated. Whena uniformly distributed load is acting on Continuous Beam.

According to Macaulay's theorem

$$M_{xx} = EI\frac{d^2 y}{dx^2} \quad \text{.................... Eq (1)}$$

By solving the above equation we obtain the max deflection is

$$Y_{max} = \frac{-9.0976*10^{13}}{EI} \quad \text{.................... Eq (2)}$$

From the above equation we get max deflection of chassis.

To find the stress we have basic bending equation

$$\frac{M}{I} = \frac{\sigma}{y} = \frac{E}{R} \quad \text{.................... Eq (3)}$$

From this equation we obtain the stress values.

III. GEOMETRIC MODELING

A three dimensional solid Model of the TATA 2518TC chassis is generated in the CAE software CATIA V5. In order to build the model accurately, the design specifications and measurements needed to be acquired in order to replicate a ladder frame model. To build each part of the chassis (side and cross members) as a separate part in CATIA V5.

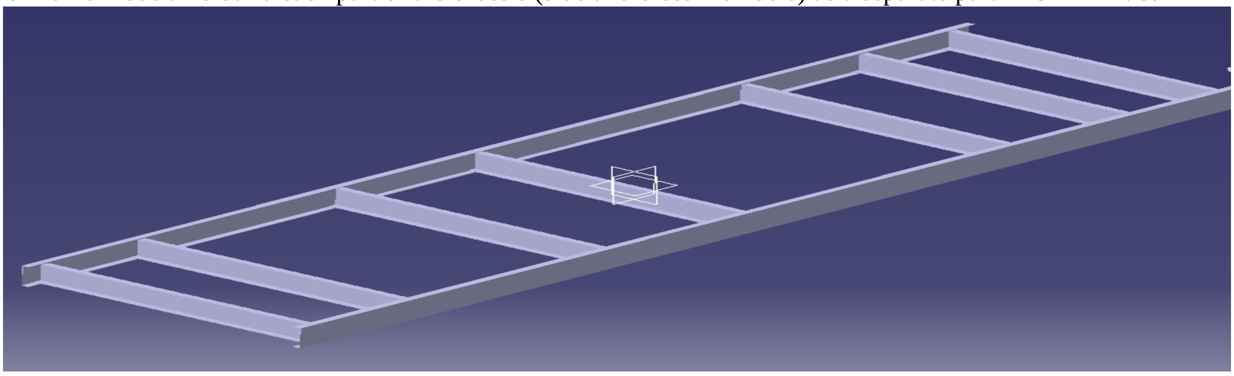


Fig5.C section framed chassis

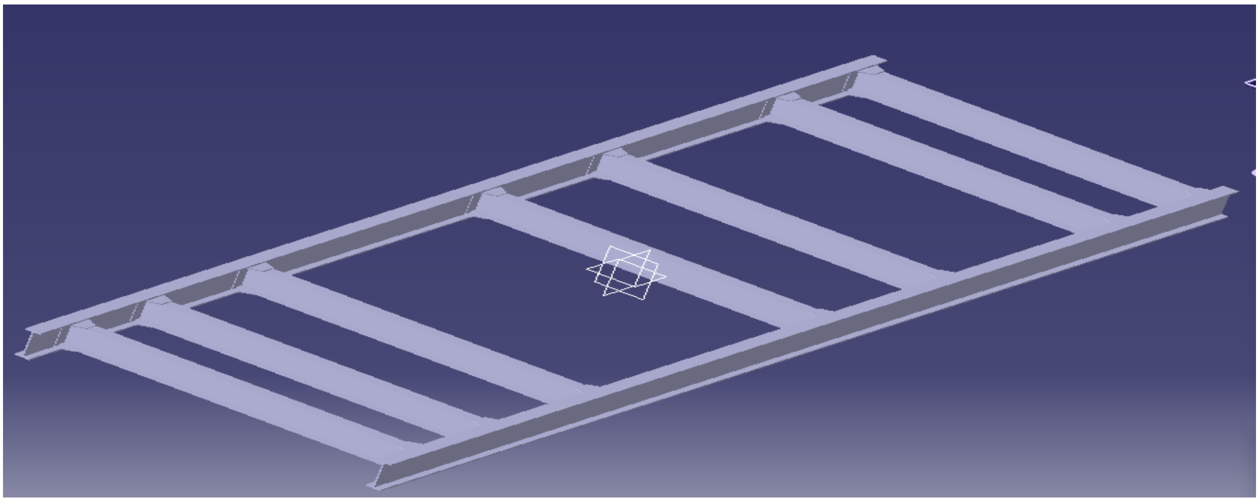


Fig6.I section framed chassis

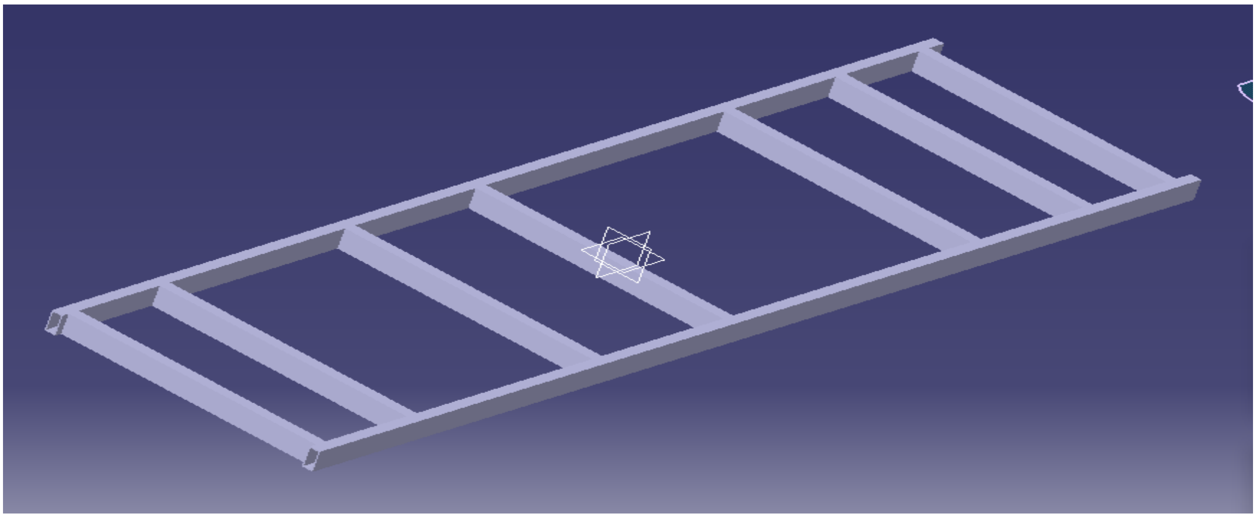


Fig7.Box section framed chassis

IV. STRUCTURAL ANALYSIS

For doing analysis of the model created in CATIA V5, we used the finite element solver ANSYS 15.0 is used.ANSYS is a general purpose finite element analysis (FEA) software package. The geometric model created in CATIA is imported in ANSYS and the analysis is carried out.

4.1. GenericSteps to solving any problem

➢**Build Geometry**

In this paper Model developed using CATIA has been imported into ANSYS for Analysis purpose.

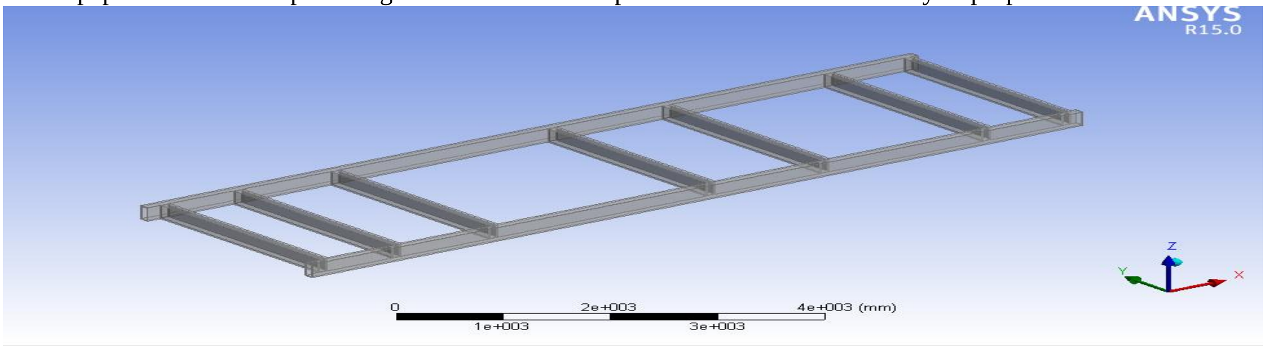


Fig8.Geometric model of Box section

➢**Define Material Properties:**

Now that the part exists, define a library of necessary materials that compose the object modeled. This includes mechanical and thermal properties.

➢ **Generate Mesh:**

At this point ANSYS understands the makeup of the part. Now define how the modeled system should be broken down into finite pieces.

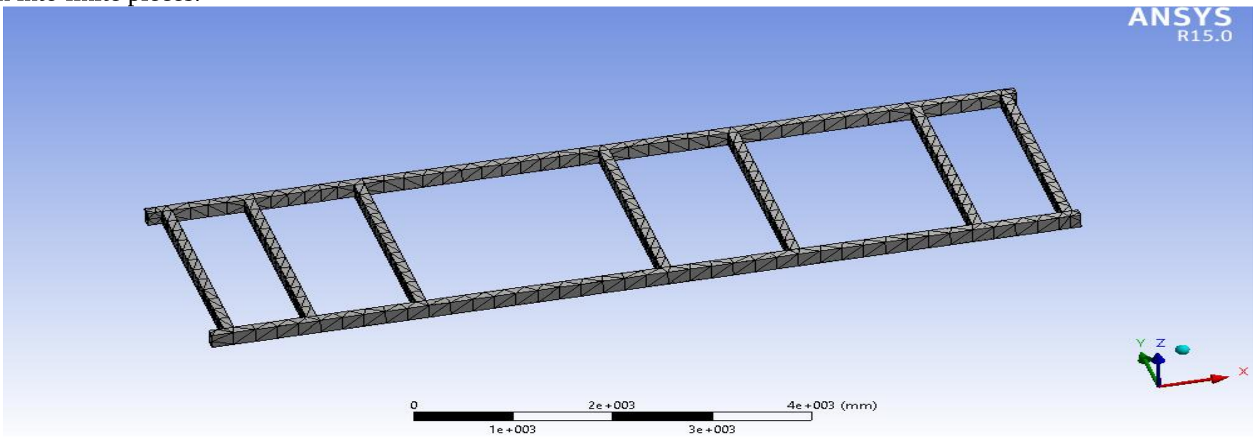


Fig9.Mesh model

➢ **Apply Boundary Conditions:**

Once the system is fully designed, the next task is to burden the system with constraints, such as supports and physical loadings.

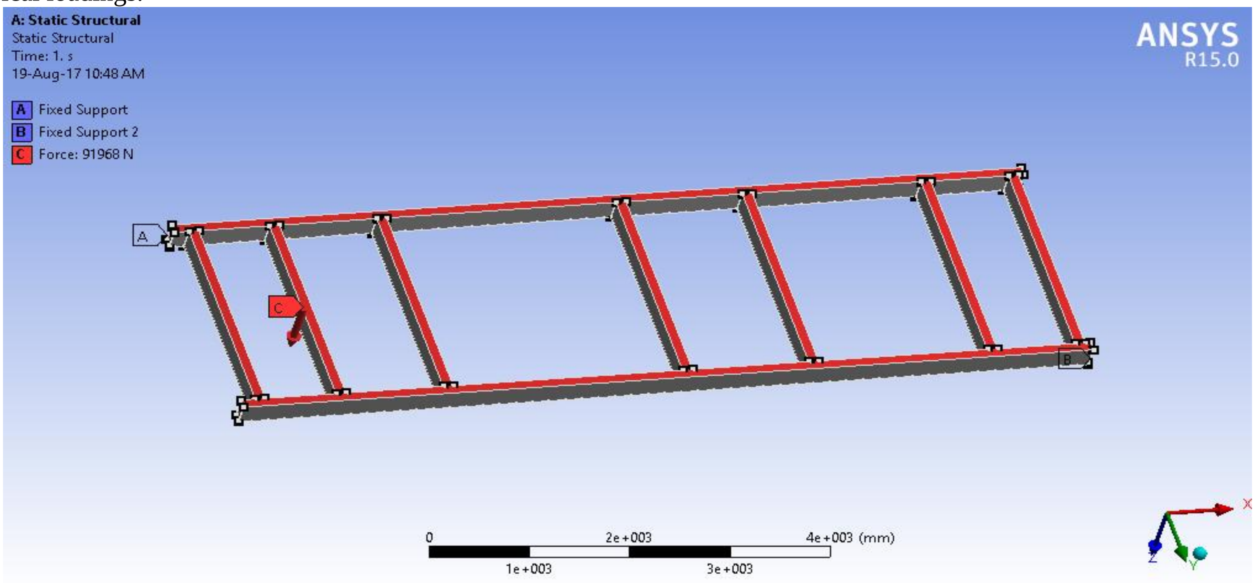


Fig10.Boundary Conditions

- **Obtain Solutions:**

This is actually a step, because ANSYS needs to understand within what state (Steady, transient state…etc.) the problem must be solved.

- **Presentation of the Results:**

Upon the chassis frame chosen the load varying from 91.96 KN, 122.62 KN and 153.28 KN at every loading condition the stress distribution and the deformation are noted in the fig 11 and fig 12.

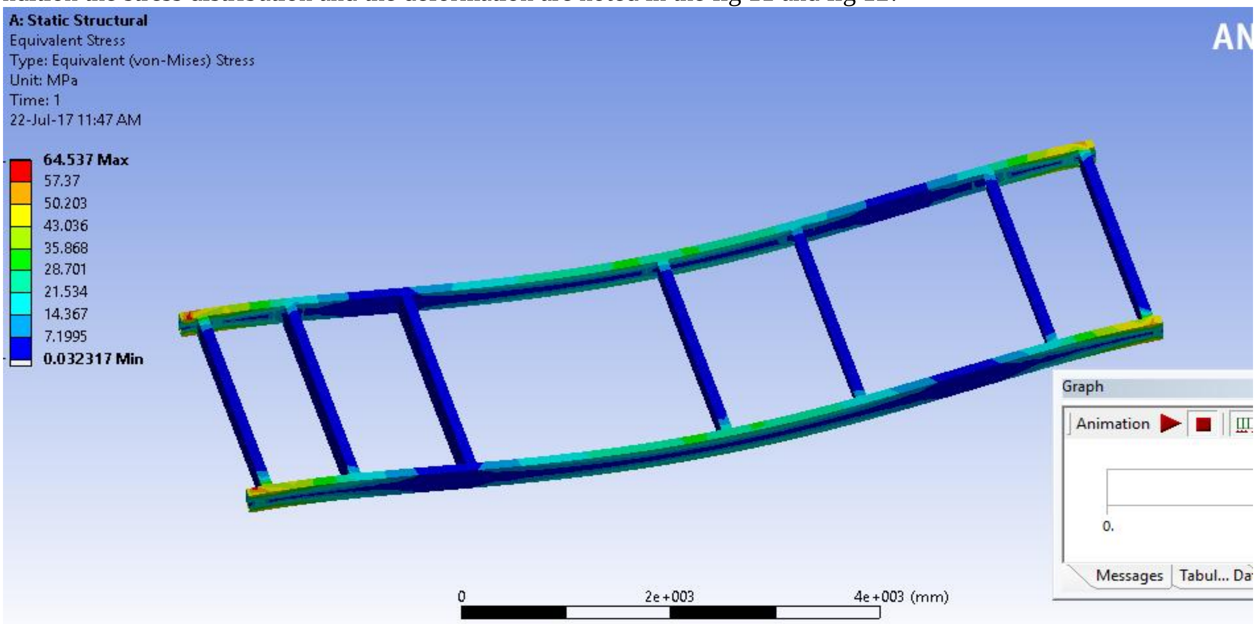


Fig11. Stress distributions in "Box"- Section

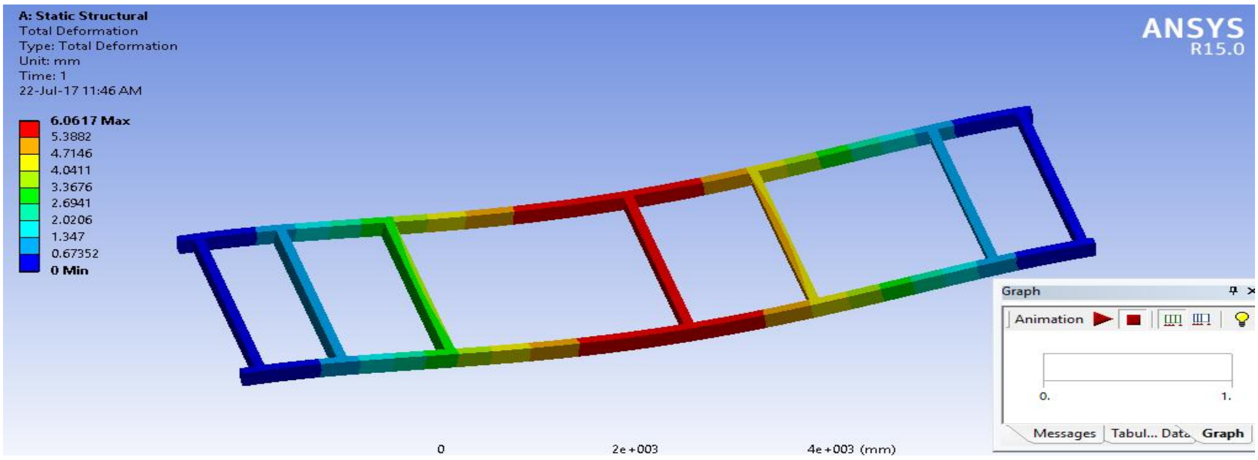


Fig12. Deformation in "Box" section

FEA Model validation:

The stress distributions and deformation at different loading conditions obtained from ANSYS simulations and Analytical results calculated using Eq (1) (2) and (3) for four different materials namely steel ST37, Carbon/Epoxy, E-Glass/Epoxy and Boron/ Epoxy are tabulated in tables 3,4 ,5,6.

Table3: Comparison of Results for steel ST37

Load (KN)	Von misesStress (N/mm^2)						Deformation (mm)					
	Analytical Method			FE Analysis			Analytical Method			FE Analysis		
	C	I	Box	C	I	Box	C	I	Box	C	I	Box
91.96	149.49	108.9	59.36	159.38	117.4	64.53	10.2	6.01	4.84	13.5	7.69	6.06
122.62	201.87	146.9	80.02	212.5	156.61	86.05	13.4	7.9	6.3	18.07	10.26	8.08
153.28	252.32	184.8	101.75	265.6	195.77	107.56	14.83	8.73	7.03	22.58	12.82	10.1

Table3 shows the deformation and stress distribution in Analytical and ANSYS resultsfor steel ST37 in different cross sections under different loads varying from 91.96 KN, 122.62 KN and 153.28 KN.

Table 4: Comparison of Results for Carbon/Epoxy

Load (KN)	Von misesStress (N/mm^2)						Deformation (mm)					
	Analytical Method			FE Analysis			Analytical Method			FE Analysis		
	C	I	Box	C	I	Box	C	I	Box	C	I	Box
91.96	147.79	108.6	59.03	157.9	117.31	64.31	11.8	6.97	5.6	14.9	8.5	6.7
122.62	199.58	146.3	79.5	210.53	156.41	85.7	15.58	9.18	7.23	19.9	11.34	8.93
153.28	246.8	183.59	100.73	263.16	195.52	107.19	17.21	10.13	8.16	24.97	14.17	11.17

Table4 shows the deformation and stress distribution in Analytical and ANSYS results for Carbon/Epoxy in different cross sections under different loads varying from 91.96 KN, 122.62 KN and 153.28 KN.

Table 5: Comparison of Results for E-glass/Epoxy

Load (KN)	Von misesStress (N/mm^2)						Deformation (mm)					
	Analytical Method			FE Analysis			Analytical Method			FE Analysis		
	C	I	Box	C	I	Box	C	I	Box	C	I	Box
91.96	145.85	108.06	58.58	156.5	117.21	64.1	56.4	33.22	26.75	71.38	40.52	31.9
122.62	197.4	145.65	79.1	208.67	156.28	85.47	74.2	43.7	35.2	95.18	54.03	42.6
153.28	244.14	182.65	99.78	260.84	195.35	106.84	82.58	48.29	38.88	118.98	67.53	53.27

Table5 shows the deformation and stress distribution in Analytical and ANSYS results for E-glass/Epoxy in different cross sections under different loads varying from 91.96 KN, 122.62 KN and 153.28 KN.

Table6: Comparison of Results for Boron/Epoxy

Load (KN)	Von misesStress (N/mm^2)						Deformation (mm)					
	Analytical Method			FE Analysis			Analytical Method			FE Analysis		
	C	I	Box	C	I	Box	C	I	Box	C	I	Box
91.96	143.68	107.75	58.18	154.5	117.13	63.8	10.05	5.82	4.68	13.3	7.54	5.9
122.62	194.53	145.2	78.45	206.08	156.18	85.09	13.1	7.36	6.15	17.73	10.06	7.94
153.28	241.37	182.14	99.13	257.6	195.22	106.37	14.19	8.51	6.84	22.17	12.58	9.93

Table6 shows the deformation and stress distribution in Analytical and ANSYS results for Boron/Epoxy in different cross sections under different loads varying from 91.96 KN, 122.62 KN and 153.28 KN.

It can be inferred from the tabulation that the numerical values obtained from ANSYS for the stress and deformation are greater than analytical results; this represents the FEA model values are within permissible limits, so the design is safe. Except E-glass/Epoxy composite material, the deformation and stress values are in the permissible limits.

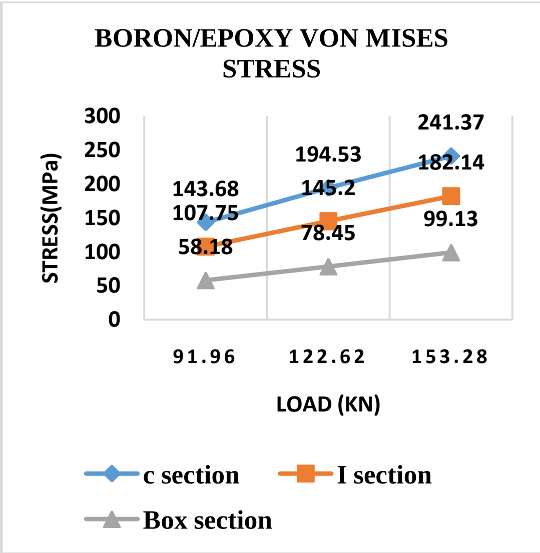


Fig 13.Analytical Results

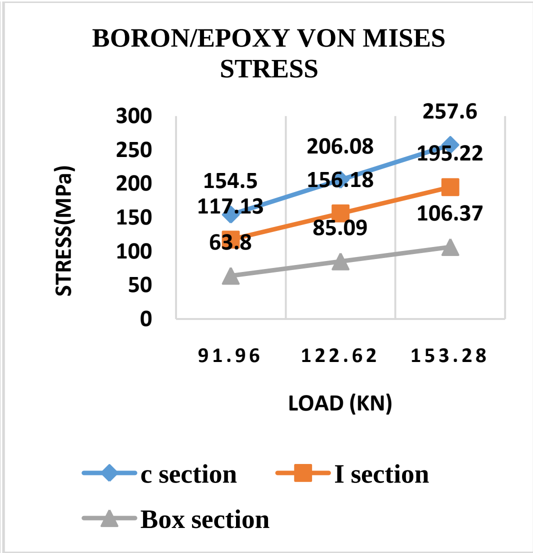


Fig 14.FEA Results

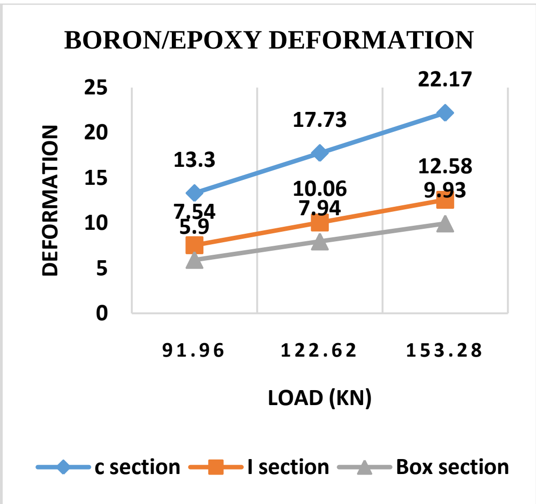


Fig15.FEA results

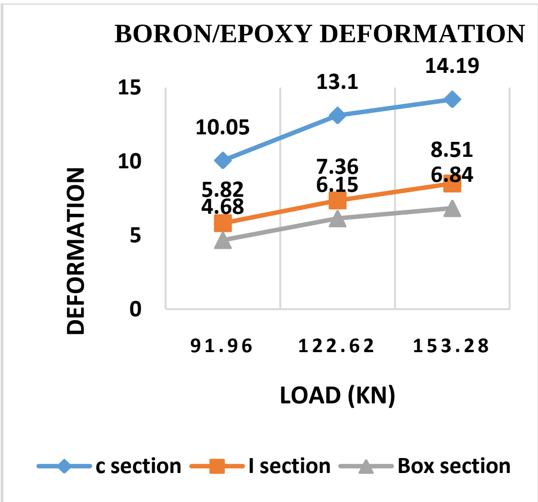


Fig16.Analytical Results

From the above results obtained from both ANSYS and analytical it is observed that the Box section with Boron/Epoxy Material has Best stress and deformation values at given loading conditions.

V.CONCLUSION

Design and analysis of heavy vehicle chassis (TATA 2518TC) have been done by the structural analysis of the heavy vehicle chassis by considering three different cross-sections, namely C, I, and hollow Rectangular (Box) type cross sections subjected to the same conditions, made of four different materials viz., Steel ST37, boron/Epoxy, Carbon/Epoxy, E-glass/Epoxy composites under different load conditions.

- From the results, it is observed that the Rectangular Box (Hollow) section is having more strength compared to C and I sections.
- The Rectangular Box (Hollow) section is having least deflection i.e., 9.93 mm and stress is 106.37 N/mm^2.
- Boron/epoxy material with Box section has the minimum deflection and stress when compared to Steel ST37, E Glass/Epoxy, and carbon/Epoxy.
- So, in different cross sections of the chassis Box-section chassis with Boron/Epoxy material is suitable for the heavy trucks.

REFERENCES

[1] Vijay Kumar V. Patel, "Structural analysis of a ladder chassis frame," World Journal of 2(4)05-08. Vol. 1 Issue 3, May - 2012 ISSN: 2278-0181.

[2]Hemant B.Patil, Sharad D.Kachave, Eknath R.Deore (Mar. - Apr. 2013), "Stress Analysis of Automotive Chassis with Various Thicknesses" International Organization of Scientific Research Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN 2278-1684 Volume 6, Issue 1, PP 44-49, D.O.I. - 10.9790/1684-0614449 .

[3] J. Chandra Sekhar, K. Venkatarao, Design and Analysis of Heavy Vehicle Chassis by Using Composite Materials. Volume 2, issue 11.1428-1427.

[4] Divyesh N. Chaudhari, Vinod M. Makwana, Dharmeet J. Patel Strength and rigidity analysis of heavy vehicle chassis for different frame cross section by analytically and FEA under various loading condition.e-ISSN: 2393-9877, p-ISSN: 2394-2444 Volume 3, Issue 5, May-2016.