✚IJESRT

INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

DESIGN AND ANALYSIS OF TATA 2518TC TRUCK CHASSIS FRAME WITH VARIOUS CROSS SECTIONS USING CAE TOOLS

Akash Singh Patel *, **Jaideep Chitransh**
Department of Mechanical Engineering, UCER, Allahabad, 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.

The present study has analyzed the various literatures. After a careful analysis of various research studies conducted so for it has been found that there is the scope of optimizing different factors like weight, stress-strain values and deformation etc. by varying cross sections for modeling and analysis. This paper describes the design and Structural analysis of the heavy vehicle chassis with constraints of maximum stress, strain and deflection of chassis under maximum load.

In the present work, the dimension of the TATA 2518TC chassis is used for 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. A three dimensional solid Modeled in the CAE software CATIA and analyzed in ANSYS. The numerical results are validated with analytical calculation considering the stress distribution and deformation.

KEYWORDS: Heavy truck chassis frame, CATIA, ANSYS, FEM, Assembly weight, stress, deformation.

INTRODUCTION

The major challenge in today's ground vehicle industry is to overcome the increasing demands for higher performance, lower weight, and longer life of components, all this at a reasonable cost and in a short period of time. The chassis of trucks is the backbone of vehicles and integrates the main truck component systems such as the axles, suspension, power train, cab and trailer. Since the truck chassis is a major component in the vehicle system, it is often identified for refinement. There are many industrial sectors using this truck for their transportations such as the logistics, agricultures, factories and other industries [1].

The chassis frame consists of side members attached with a series of cross members Stress analysis using Finite Element Method (FEM) used to locate the critical point which has the highest stress. This critical point is one of the factors that may cause the fatigue failure. The magnitude of the stress used to predict the life span of the truck chassis [2].

Principal Functions
Provide mounting points for the suspensions, the steering mechanism, the engine and gearbox, the final drive, the fuel tank and the seating for the occupants. Protect the occupants against external impact,To safety carry the maximum load, Holding all components together while driving, Accommodate twisting on even road surface, Endure shock loading, It must absorb engine and driveline torque [3].

Layout of Chassis
Chassis is an important part of automobile. The chassis serves as a frame for supporting the body and different units of automobile like engine, suspension, gearbox, braking system, steering, propeller shaft, differential, axle assemblies, etc. are welded or bolted as shown in fig.1[4].
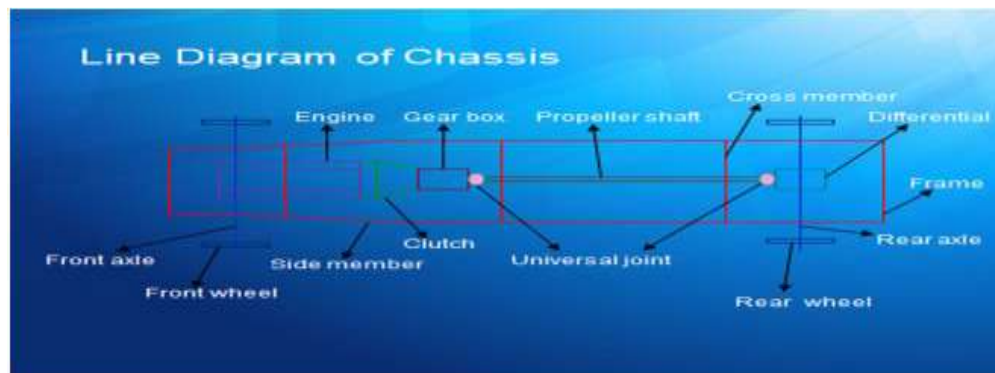

Fig: 1 Line diagrams of chassis

Types of chassis
1.Ladder Frame chassis
 Ladder chassis is considered to be one of the oldest forms of automotive chassis or automobile chassis that is still been used by most of the SUVs till today. It is also resembles a shape of a ladder which having two longitudinal rails inter linked by several lateral and cross braces asshown in fig.2 [5].

Fig: 2 Ladder frame chassis

2. Backbone chassis
The other type of chassis is backbone chassis which has a rectangular tube like backbone and simple in structure. It usually made up of glass fiber that is used for joining front and rear axle together and responsible for most of the mechanical strength of the framework. The space within the structure is used for positioning the drive shaft in case a rear-wheel drive. This type of chassis is strong enough to provide support smaller sports car besides it is easy to make and cost effective [5].
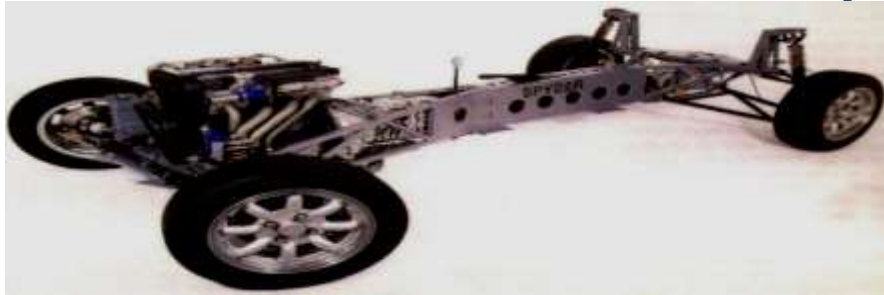
Fig: 3 Backbone chassis

3. Monocoque Chassis

As for monocoque chassis, most modern cars now a days use this type of chassis. A monocoque chassis is a single piece of framework that gives shape to the car. A one-piece chassis is built by welding several pieces together. It is different from the ladder and backbone chassis as unlike them incorporated with the body in a single piece, whereas the former only support the stress members. The demanding of a monocoque chassis highly increased since it is cost effective and suitable for robotized production [5].

Fig: 4 Monocoque chassis

4. Tubular space frame chassis

In this study, it is decided that tubular space frame chassis is used for the urban car. Since ladder chassis is not strong enough, motor racing engineers have developed a 3-dimensional design which known as tubular space frame. Tubular space frame chassis employs dozens of circular-section tubes (some may use square section tubes for easier connection to the body panels though circular section provides the maximum strength), position in different directions to provide mechanical strength against forces from anywhere. These tubes are welded together and form a complex structure [5].

Fig: 5 Space frame chassis

LITERATURE REVIEW

Literature Survey

Many researchers carried out study on truck chassis as follows:

Patel et al [1] have investigated and optimized a chassis design for Weight reduction of TATA 2516TC chassis frame using Pro-Mechanica. Thy first find out the assembly weight, maximum stress, strain and displacement for the existing section of chassis by using ANSYS Software after then they modified the dimensions of existing C-sections and again find all and concluded that the existing "C" sections is better than all the sections with respect to the Stress, Displacement, Strain and Shear stress except the weight. For the weight consideration modified "C" section has less weight than the all sections which are studying in this paper. Finally By the use of modified "C" section,

105.50 Kg (11 %) weight is saved per chassis assembly an.d in same manner cost may also be reduced approximately 11%. From the results, modified "C" sections are used as an optimized section.

Murali et al [2] have investigated the critical point which has the highest stress using Finite Element Method (FEM). This critical point is one of the factors that may cause the fatigue failure. For the modifications and analysis, the existing truck chassis were added with stiffeners. Initially the thickness of the model, where the maximum deflection occurs in bending analysis was increased to certain value with acceptable limit. And one more cross beam was added at the center of the wheel base to add stiffness to the model. Series of modifications and tests were conducted by adding the stiffener in order to strengthen and improved the chassis stiffness as well as the overall chassis performances.

S. Prabakaran and K. Gunasekar [3] have studies the Structural Analysis of EICHER E2 (or 11.10) Chassis Frame for the existing C-section. They first find out the assembly weight, maximum shear stress, maximum equivalent stress and displacement for the existing C-section of chassis by using SOLID WORKS and ANSYS Software and then they modified the existing C-section taking three different cases and find out the parameters for all cases. They have investigated that the weight, maximum shear stress, maximum equivalent stress and displacement for the third case are reduced respectively 6.68%, 12.14 %, 8.55 % and 11.20 %. So they concluded that by using FEM software we can optimize the weight of the chassis frame and it is possible to analyze modified chassis frame before manufacturing.

B. Ramana Naik and C. Shashikanth [4] have objective to analyze an automobile chassis for a 10 tonne vehicle. The modeling 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. This frequency is more than 4 times the highest frequency of the excitation (i.e. 33 Hz) hence the chassis can faithfully transmit the input excitation to the vehicle body without any amplification.

Kamlesh Y. Patil and Eknath R. Deore [5] have studies the Ladder Chassis frame of TATA 912 Diesel Bus and The model of the chassis was created in Pro-E and analyzed 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.

Sharma et al [6] have studies the chassis of a Heavy Vehicle TATA LPS 2515 EX with three different alloys subjected to the same conditions of the steel chassis. The three material used for the chassis are grey cast iron, AISI4130 alloy steel and ASTM A710 STEEL GRADE A. The three different vehicle chassis have been modeled by considering three different cross-sections. Namely C, I, and Box type cross sections. A three dimensional solid Model was built in the CAE software CATIA V5 parametric and the analysis was done in ANSYS-14.5. The results shows that the default material for the chassis i.e. A709M Grade 345 W Structure steel shows strength equal to the AISI 4130 steel alloy but in case of the deformation AISI 4130 alloy is superior to structure steel. So, for the consideration of alloy for the chassis AISI 4130 alloys is better than others and for different cross sections of the chassis C-section chassis is suitable for the heavy trucks.

Lenin et al [7] modeled a chassis used in a TATA ACE using CATIA. Structural and modal analyses are done on chassis using ANSYS. The analysis is done using three materials Cast Iron, Aluminum and E-GLASS EPOXY. By observing structural analysis results, the stress values for Glass Epoxy and E –Glass Epoxy are less than their respective allowable stress values. So using composites for chassis is safe. From the obtained ANSYS results it was found that the suitable proposed material for automotive chassis is Glass Fiber Reinforced Plastic Materials. But, when compared with GFRP the cost of CFRP will be higher therefore here considered that, the GFRP material is suitable for automotive chassis. Also when compared different section it is found that the I-section will be suitable for this application.

Vijay Kumar V. Patel and Prof. R. I. Patel [11] have investigated and optimized a chassis design of EICHER E2 (or 11.10) for Weight Reduction same manner as done by S. Prabakaran and K. Gunasekar but they used Pro-E softer for modeling and find the same results.

Patel et al [14] have investigated the stress and deformation developed in chassis frame of EICHER 11.10. The model of the chassis has been developed in SOLID WORKS 2009 and static structural analysis has been done in ANSYS workbench. The analysis gives maximum shear stress and total deformation which are in desired limit, so the design is safe.

Potdar et al [15] used approximate dimensions to model the N1 type TATA Ace chassis. Software used was CATIA V5 for modeling and ANSYS Workbench for analysis. Concluded that the deflection due to bending of the Chassis Members can be evaluated using Static Finite Element Analysis instead of conventional time consuming calculations. The overall bending stiffness of the Chassis is 13724 N/m which is under 22000N/m, permissible range for N1 Vehicle Category.

Problem Formulation
The present study has analyzed the various literatures. After a careful analysis of various research studies conducted so for it has been found that there is the scope of optimizing different factors like weight, stress-strain values and deformation etc. by varying cross sections for modeling and analysis. This paper describes the design, Structural analysis & optimization of the heavy vehicle chassis with constraints of maximum stress, strain and deflection of chassis under maximum load. Our work is to design and analyze the heavy vehicle chassis to reduce weight, stress-strain values and deformation etc.

Objectives of proposed work
In the present work, the dimension of the TATA 2518TC chassis is used for the structural analysis of the heavy vehicle chassis modeled by considering three different cross-sections, namely C, I, and Rectangular Box (Hollow) type cross sections subjected to the same condition. A three dimensional solid Modeled in the CAE software CATIA V5 and analyzed in ANSYS 14.0. The numerical results will be validated with analytical calculation considering the stress distribution, deformation.

CHASSIS FRAME CROSS SECTIONS
To appreciate the design and construction of a vehicle's chassis an understanding of the operational environment is necessary. Once the operating conditions are known, a comparison of the different available chassis-member cross-section shapes can be made. The flowing sections examine and illustrate the basic requirements of the chassis.

1. "C"-Channel sections
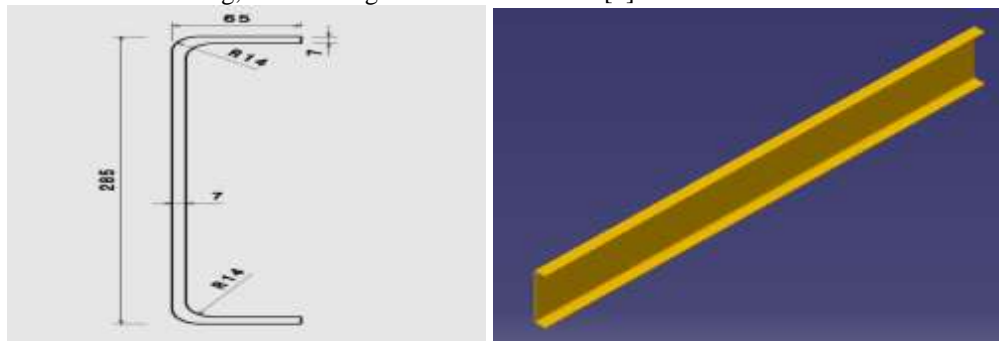It has good resistance to bending, used in long section of the frame [1].

Fig: 6 "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.
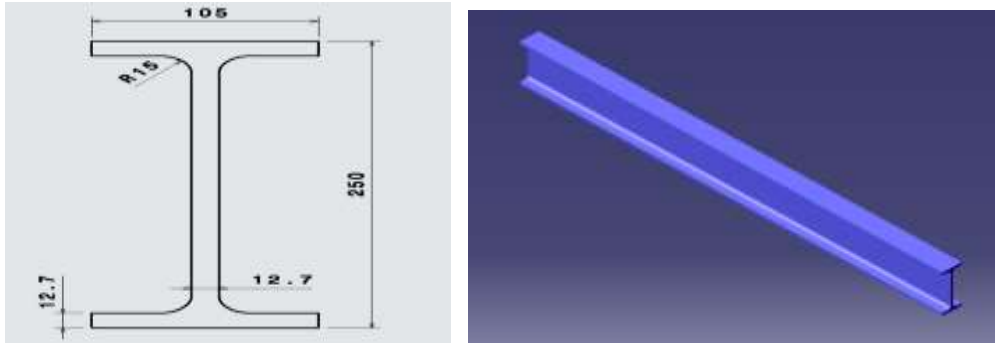

***Fig:** 7 "I"- Sections*

3. Box sections

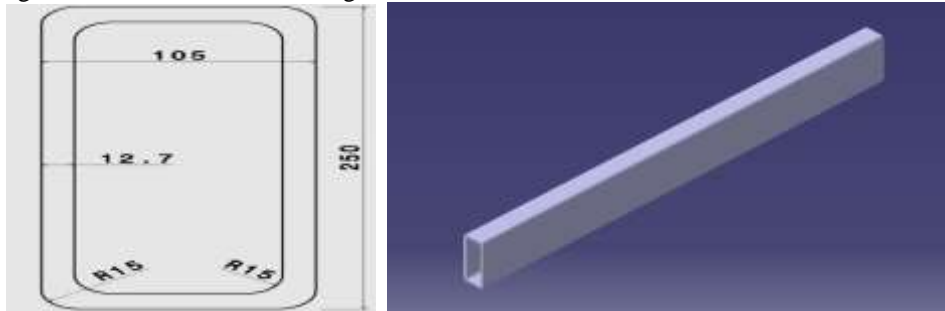It has good resistance to both bending and torsion, used in short members of frames.

***Fig:** 8 Box- Sections*

4. Tubular sections

It has good resistance to torsion. Tubular section is used these days in three wheelers, scooters pick-ups and bicycle.

***Fig:** 9 Tubular sections*

CHASSIS FRAME MATERIAL

Currently the material used for the chassis (TATA 2518TC) is as per IS: - 9345 standard is structural steel with St 37. Structural steel in simple words with the varying chemical composition leading to changes in names. The typical chemical composition of the material is[6]:

0.565%C, 1.8% Si, 0.7%Mn, 0.045%P and 0.045%S.

Physical Property of the ST37:-

Modulus of Elasticity = 210 GPa = 2.10 x 105 N / mm^2

Density = 7850 kg/m^3

Ultimate Tensile Strength = 460 MPa = 460 N / mm^2

Yield Strength = 260 MPa = 260 N / mm^2

Poisson Ratio = 0.29

ANALYTICAL CALCULATIONS

Table: 1 Specification of Existing Heavy Vehicle TATA LPT 2518 TC Truck Chassis frame

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

Side bar of the existing chassis frame are made from "C" Channels with Height (H) = 285 mm, Width (B) = 65mm, Thickness (t) = 7 mm

Basic Calculation for Chassis Frame

Model No. = LPT 2518 TC (TATA)
Capacity of Truck = 25 ton (Kerb Weight+ Payload)
= 25000 kg = 245250 N
Capacity of Truck with 1.25% = 245250 * 1.25 N = 306562 N
Total Load acting on the Chassis = 306562 N

All parts of the chassis are made from "C" Channels with 285mm x 65mm x 7mm. Each Truck chassis has two beams. So load acting on each beam is half of the Total load acting on the chassis.

Load acting on the single frame = Total load acting on the chassis / 2
= 306562 /2
=153281 N / Beam

Loading Conditions

Beam is simply clamp with Shock Absorber and Leaf Spring. So Beam is a Simply Supported Beam with uniformly distributed load. Load acting on Entire span of the beam is 153281 N. Length of the Beam is 9010 mm.
Uniformly Distributed Load is 153281 / 9010 = 17.0 N/mm

According to loading condition of the beam, a beam has a support of three axle means by three wheel axles C, D and E. Total load reaction generated on the beam is as under:-
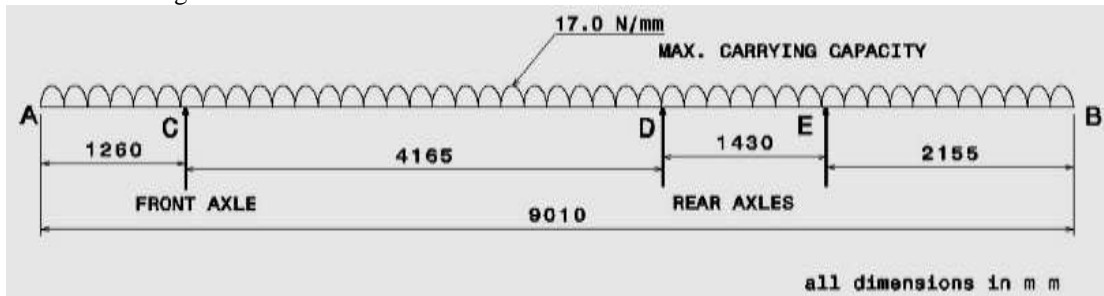
Fig: 10 Total load generated on the beam

Fixed End Moment

This is the indeterminate structure of beam.
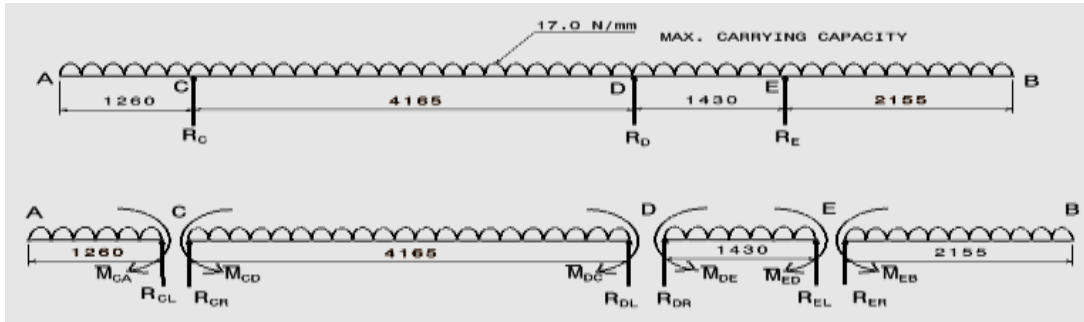
Fig: 11 Consideration of fixed end moments

$\overline{M}_{CA} = \frac{17*1260*1260}{2} = 13494600$ N mm

$\overline{M}_{CD} = \frac{-17*4165*4165}{12} = -24575235.42$ N mm

$\overline{M}_{DC} = +24575235.42$ N mm

$\overline{M}_{DE} = \frac{-17*1430*1430}{12} = -2896941.667$ N mm

$\overline{M}_{ED} = +2896941.667$ N mm
$\overline{M}_{EB} = \frac{-17*2155*2155}{2} = -39474212.5$ N mm

Total restraint moment at "C"

$\overline{M}_C = \overline{M}_{CA} + \overline{M}_{CD} = 13494600 -24575235.42 = -11037364.58$ N mm

Total restraint moment at "E"

$\overline{M}_E = \overline{M}_{ED} + \overline{M}_{EB} = 2896941.667 -39474212.5 = -36577270.83$ N mm

For the span "CD"

$M_{CD} = -\overline{M}_{CA} = -13494600$ N mm

$M_{DC} = \overline{M}_{DC} - \overline{M}_C\backslash 2 + \frac{3EI}{l} i_b = 24575235.42 + 11037364.58\backslash 2 + \frac{3EI}{4165} i_b$

$\qquad = 30093917.71 + \frac{3EI}{4165} i_b$

For the span "DE"

$M_{ED} = -\overline{M}_{EB} = 39474212.5$ N mm

$M_{DE} = \overline{M}_{DE} - \overline{M}_E\backslash 2 + \frac{3EI}{l} i_b = -2896941.667 + 36577270.83\backslash 2 + \frac{3EI}{1430} i_b$

$\qquad = 15391693.75 + \frac{3EI}{1430} i_b$

Equilibrium condition at "D"

$\overline{M}_{DC} + \overline{M}_{DE} = 0$

$30093917.71 + \frac{3EI}{4165} i_b + 15391693.75 + \frac{3EI}{1430} i_b = 0$

EI $i_b = -1.624486 * 10^{10}$

Substituting the value of EI i_b

$M_{DC} = 18392938.12$ N mm

$M_{DE} = -18688432.12$ N mm
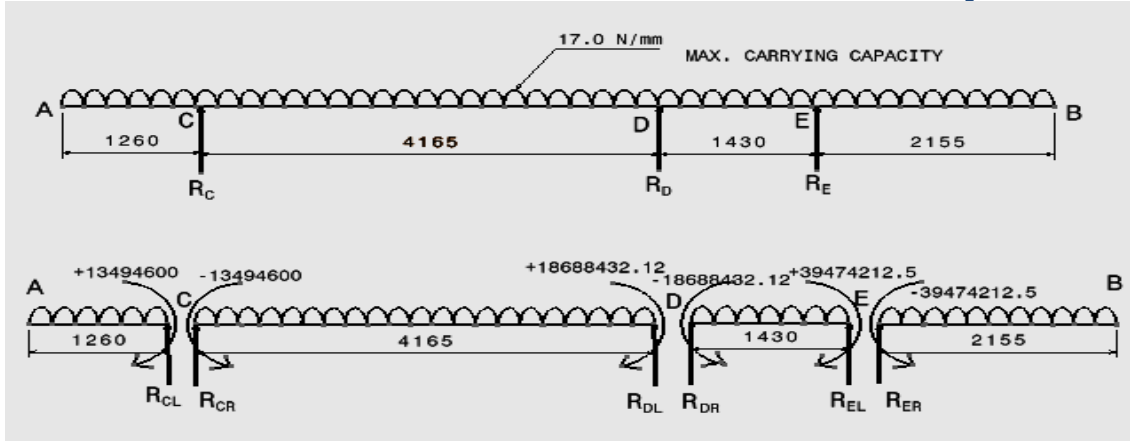
Fig:12 Fixed end moments

Calculations for Reaction and Shear Force Diagram

$R_{CL} = 17 * 1260 = 21420$ N (↑)

$R_{CR} = \dfrac{17*4165}{2} + \dfrac{13494600 - 18688432.12}{4165} = 34155.48$ N (↑)

$R_{DL} = 17*4165 - 34155.48 = 36649.52$ N (↑)

$R_{DR} = \dfrac{17*1430}{2} + \dfrac{18688432.12 - 39474212.5}{1430} = - 2380.51$ N (↓)

$R_{EL} = 17*1430 + 2380.51 = 26690.51$ N (↑)

$R_{ER} = 17*2155 = 36635$ N (↑)

∴ $R_C = R_{CL} + R_{CR} = 55575.48$ N (↑)

$R_D = R_{DL} + R_{DR} = 34269.01$ N (↑)

$R_E = R_{EL} + R_{ER} = 63325.51$ N (↑)
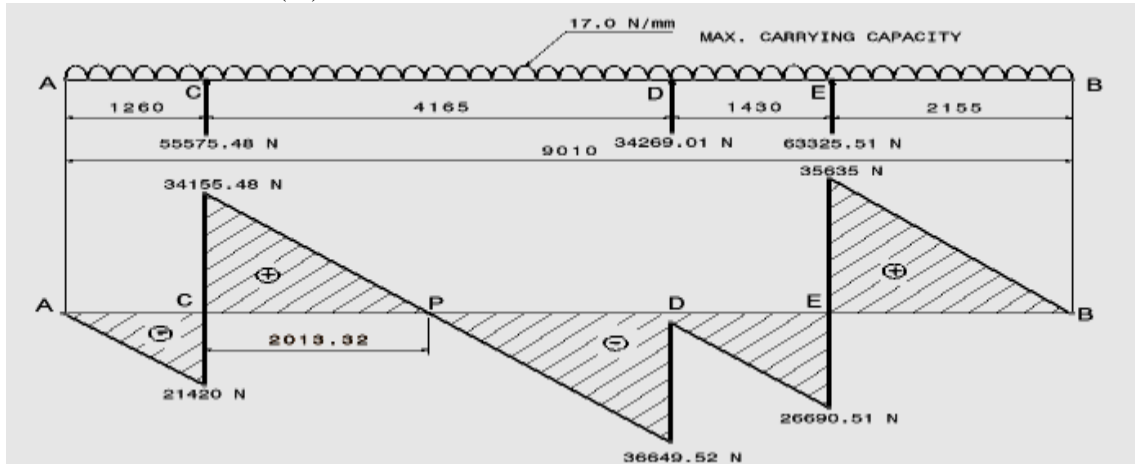
Fig: 13 Shear Force Diagram

Calculations for Bending Moment Diagram:-

$M_A = 0$ N mm

$M_C = - \overline{M}_{CA} = - 13494600$ N mm

$M_P = \dfrac{17* 4165* 4165}{8} = 36862853.13$ N mm

$M_D = M_{DE} = - 18688432.12$ N mm

$M_Q = \dfrac{17* 1430* 1430}{8} = 4345412.5$ N mm

$M_E = \overline{M}_{EB}$ = -39474212.5 N mm
M_B = 0 N
So the maximum bending moment occurs at "E"

$M_{max} = M_E$ = -39474212.5 N mm
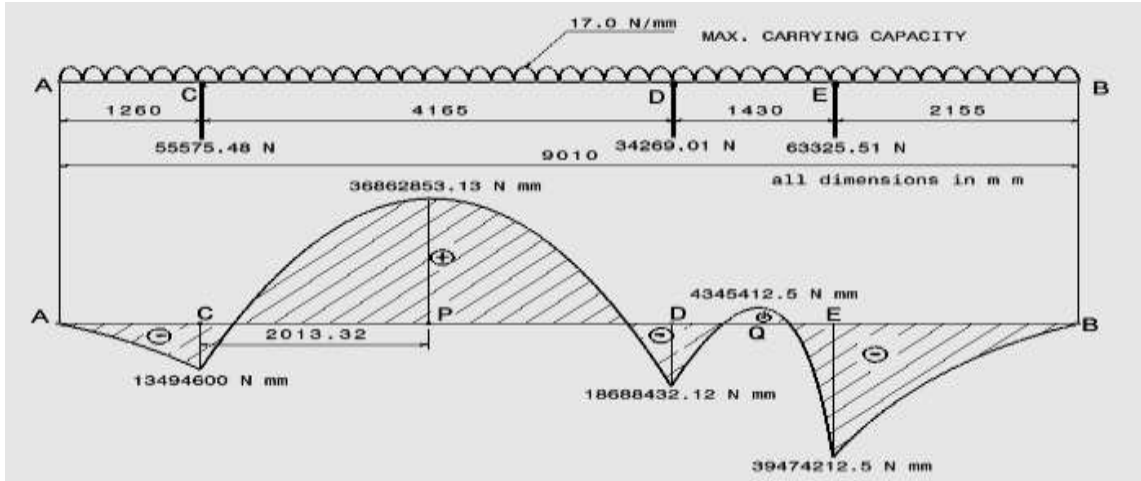
Fig: 14 *Bending Moment Diagram*

Calculations for the Maximum Deflection
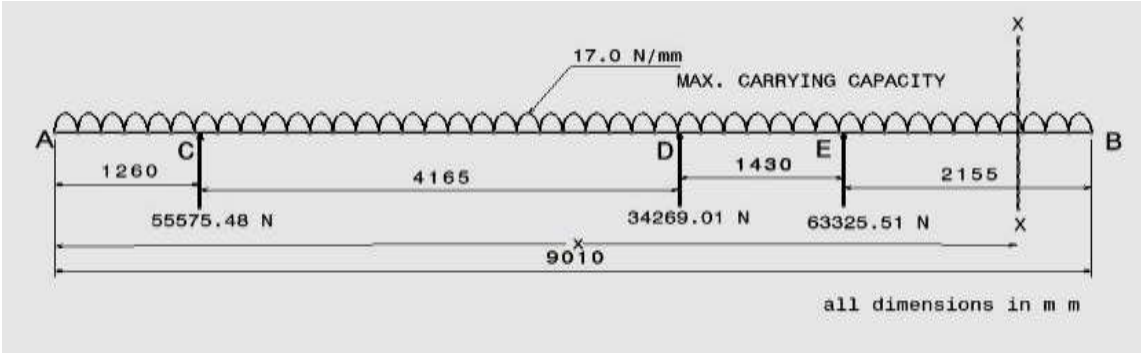
Fig: 15 *Reaction generated on the beam*

We consider a section x-x in "EB" span at x distance from A.
Taking moment of all forces about x-x section
M_{xx} = -8.5x^2 + R_C(x-1260) + R_D(x-5425) + R_E (x-6855)
According to Macaulay's theorem
M_{xx} = EI $\frac{d^2y}{dx^2}$ = -8.5x^2 + R_C(x-1260) + R_D(x-5425) + R_E (x-6855)
On integrating with respect to x we get

$$EI\frac{dy}{dx} = \frac{-17x^3}{6} + C_1 + R_C\frac{(x-1260)^2}{2} + R_D\frac{(x-5425)^2}{2} + R_E\frac{(x-6855)^2}{2}$$

Again integrating with respect to x we get

$$EIy = \frac{-17x^4}{24} + C_1x + C_2 + R_C\frac{(x-1260)^3}{6} + R_D\frac{(x-5425)^3}{6} + R_E\frac{(x-6855)^3}{6} \quad \dots\dots\dots 1$$

Applying the boundary conditions
At x = 1260mm, y = 0
$$0 = \frac{-17*1260^4}{24} + C_1 * 1260 + C_2 \quad \dots\dots\dots\dots\dots\dots\dots\dots\dots\dots 2$$

At x = 6855mm, y = 0

$$0 = \frac{-17*6855^4}{24} + C_1 * 6855 + C_2 + 55575.48\frac{(5595)^3}{6} + 34269.01\frac{(1430)^3}{6} \quad \cdots 3$$

Solving equation 2 and 3 we get

$C_1 = -1.384*10^{10}$

And $C_2 = 1.922 * 10^{13}$

Putting these values in equation 1 we get

$$y = \frac{1}{EI}\left[\frac{-17x^4}{24} - 1.384*10^{10}\,x + 1.922*10^{13} + 55575.48\,\frac{(x-1260)^3}{6} + 34269.01\,\frac{(x-5425)^3}{6} + 63325.51\,\frac{(x-6855)^3}{6}\right] \cdots\cdots\cdots\cdots 4$$

The above equation is the general equation for deflection in chassis. The deflections at the supports (C, D, and E) are zero.

Deflection at "A" (i.e. x = 0)

$$y_A = \frac{1.922*10^{13}}{EI}$$

Deflection at "B" (i.e. x = 9010mm)

$$y_B = \frac{-9.0976*10^{13}}{EI}$$

So the maximum deflection occurs at "B"

$$y_{max} = y_B = \frac{-9.0976*10^{13}}{EI} \quad \cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots 5$$

For C- Section

Radius of Gyration R = (285 / 2) = 142.5 mm

b =65mm, h =285mm, b_1 =58mm, h_1 =271mm,

y = h /2 = 285 /2 = 142.5 mm
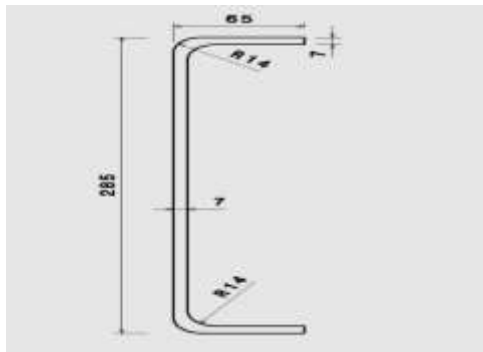
Fig: 16 Main "Channel" section

Moment of Inertia around the X – X axis:-

$I_{XX} = [bh^3 - b_1 h_1^3] / 12$

$\quad = [65* 285^3 - 58* 271^3] / 12$

$\quad = 29195623.92 \text{ mm}^4$

Section of Modules around the X – X axis:-

$Z_{XX} = I_{XX} / y = 29195623.92/ 142.5 = 204881.572 \text{ mm}^3$

Basic Bending equations are as follow:-

$$\frac{M}{I} = \frac{\sigma}{y} = \frac{E}{R} \quad \cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots 6$$

Maximum Bending Moment acting on the Beam

$M_{max} = -39474212.5 \text{ N mm}$

$Z = 204881.572 \text{ mm}^3$

Stress produced on the Beam

$\sigma = \frac{M}{Z} = \frac{-39474212.5}{204881.572} = -192.669 \text{ N/ mm}^2$

Maximum Deflection produced on the Beam

$E = 210000 \text{ MPa} = 2.10 \times 10^5 \text{ N / mm}^2$

I =29195623.92 mm^4

$y_{max} = \dfrac{-9.0976 * 10^{13}}{EI} = \dfrac{-9.0976 * 10^{13}}{210000*29195623.92} = $ - 14.839 mm

According deflection span ratio is allowable for simply supported beam is 1/ 300

According to 1 / 300 for 9010 length

= 9010 / 300 = 30.03 mm, So 14.839 mm is safe.

For I- Section

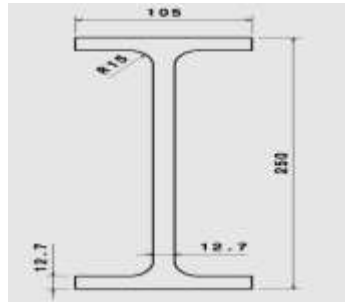b =105mm, h =250mm, b1 =46.15mm, h1 =224.6mm,

y = h /2 = 250 /2 = 125 mm

Figure.17 Main "I" section

Moment of Inertia around the X – X axis:-

I $_{XX}$ = [bh3- 2b1 h13] / 12

= [105* 2503- 2* 46.15* 224.63] / 12

= 49572297.92 mm4

Section of Modules around the X – X axis:-

Z $_{XX}$ = I $_{XX}$ / y = 49572297.92 / 125 = 396578.4 mm3

Stress produced on the Beam

$\sigma = \dfrac{M}{Z} = \dfrac{-39474212.5}{396578.4} = $ - 99.537 N/ mm^2

Maximum Deflection produced on the Beam

$y_{max} = \dfrac{-9.0976 * 10^{13}}{210000*49572297.92} = $ - 8.739 mm

So 8.739 mm is safe

For Hollow Rectangular- Section
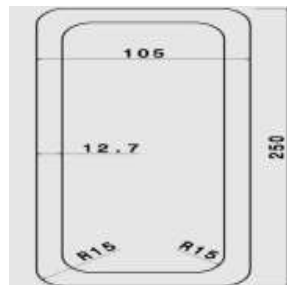
***Fig:** 18 Main "Hollow Rectangular" section*

b =105mm, h =250mm, b$_1$ =79.6mm, h$_1$ =224.6mm,

y = h /2 = 250 /2 = 125 mm

Moment of Inertia around the X – X axis:-

I $_{XX}$ = [bh^3- b$_1$ h$_1^3$] / 12

= [105* 250^3- 79.6* 224.6^3] / 12

= 61563196.525 mm^4

Section of Modules around the X – X axis:-

Z $_{XX}$ = I $_{XX}$ / y = 61563196.525 / 125 = 492505.572 mm^3

Stress produced on the Beam

$\sigma = \dfrac{M}{Z} = \dfrac{-39474212.5}{492505.572} = $ - 80.150 N/ mm^2

Maximum Deflection produced on the Beam

$$y_{max} = \frac{-9.0976 * 10^{13}}{210000 * 61563196.525} = -7.037 \text{ mm}$$

So 7.037 mm is safe

METHODOLOGY FOR MODELING AND ANALYSIS

A three dimensional solid Model of the TATA 2518TC chassis modeled in the CAE software CATIA and the analysis done in ANSYS as shown in fig.19. The procedure of modeling and analysis consists of [7]:

- ➢ Collection of the dimensions of TATA LPT 2518 TC chassis frame.
- ➢ Design of three different Computer Models of chassis frame using CATIA for different cross sections C, I and Box.
- ➢ Each model implemented in ANSYS for FE Analysis for different parameters like: 'assembly weight', 'stress' and 'deformation' etc.
- ➢ Checking all parameters whether they are within permissible limit or not for selected materials.
- ➢ Optimization and Validation of result.
- ➢ Final results and Conclusions.
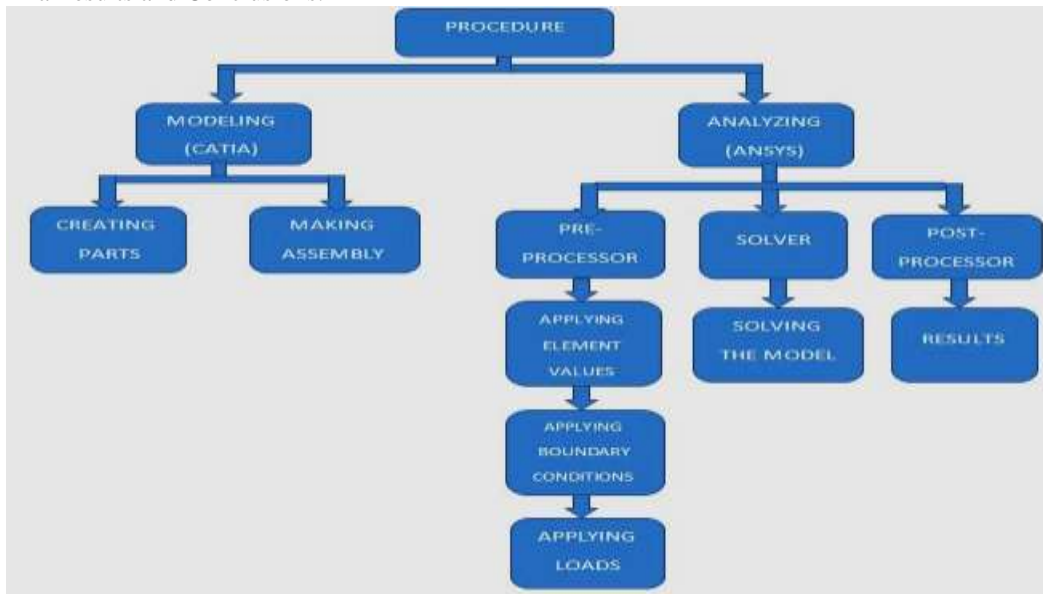
Fig: 19 Procedures for Modeling and Analysis

CATIA
CATIA (Computer Aided Three-Dimensional Interactive Application) is a multi-platform CAD/CAM/CAE commercial software suit developed by the French company Dassault Systems directed by Bernard Charles. It is basically used in the design of aerospace, automobile, shipbuilding, architecture and construction, power plant and other industries. Firstly named CATI (Conception Assisted Tridimensional Interactive), it was renamed CATIA in 1981 when Dassault created a subsidiary to develop and shell the software.

CATIA provides 3D experience platform including surfacing and shape design, electrical fluid and electronics systems design, mechanical engineering and systems design. It is basically developed for the field such as systems Architecture Engineering, System Dynamics and Performance Engineering, 3D Master Conceptual Design, Mechanical and Shape Design, Sheet Metal Design Engineering and Virtual to Real Optimization etc.

ANSYS
For doing analysis of the model created in CATIA V5, we used the finite element solver ANSYS 14.0. ANSYS is a general purpose finite element analysis (FEA) software package. Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces (of user-designated size) called elements. The software implements equations that govern the behavior of these elements and solve them all creating a comprehensive

explanation of how the system acts as a whole. These results then can be presented in tabulated or graphical forms. This type of analysis is typically used for design and optimization of a system for too complex to analyze by hand. Systems that may fit into this category are too complex due to their geometry, scale, or governing equations. ANSYS is the standard FEA teaching tool within the mechanical engineering.

Generic Steps to solving any problem in ANSYS:

➢ **Build Geometry**

Construct a two or three dimensional model of the object using work plane coordinate system within ANSYS. The model of object created in CATIA may also direct implemented in ANSYS geometry modeler.

➢ **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.

➢ **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.

➢ **Obtain Solutions:**

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

➢ **Present the Results:**

After the solution has been obtained, there are many ways to present ANSYS results, choose from many options such as graphs, tables, and contour plots.

GEOMETRIC MODELING

A three dimensional solid Model of the TATA 2518TC chassis modeled 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.

Creating Parts

The approach used was to build each part of the chassis (side and cross members) as a separate part in CATIA V5. Consequently building a model in CATIA V5 usually starts with building 2D sketches. The sketch consists of geometry such as points, lines, arcs, conics, and splines. Dimensions are added to the sketch to define the size and location of the geometry. Relations are used to define attributes such as tangency, parallelism, perpendicularity, and concentricity. The parts of "C" section chassis frame are shown in fig.21.
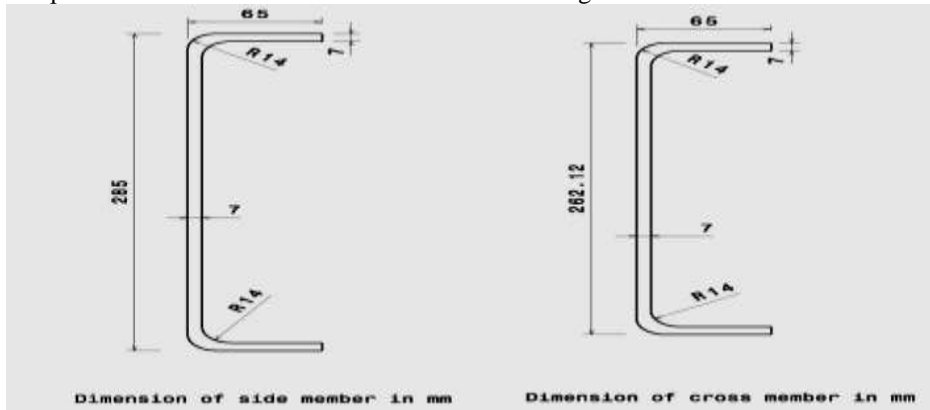

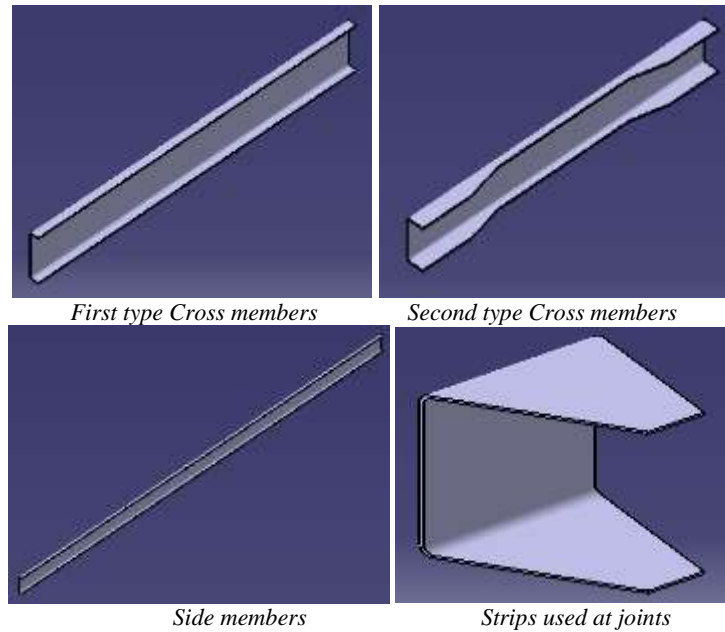*Fig: 20 Dimension of Side and Cross members of Channel section*

First type Cross members   *Second type Cross members*

Side members   *Strips used at joints*

***Fig:** 21 Parts of "C"- Section Chassis frame design on CATIA*

Making Assembly
Once each part and sub-assembly was completed they were combined together in an assembly as shown in fig15. In an assembly, the equivalents to sketch relations are mates. Just as sketch relations define conditions such as tangency, parallelism, and concentricity with respect to sketch geometry, assembly mates define equivalent relations with respect to the individual parts or components, allowing the easy construction of assemblies.
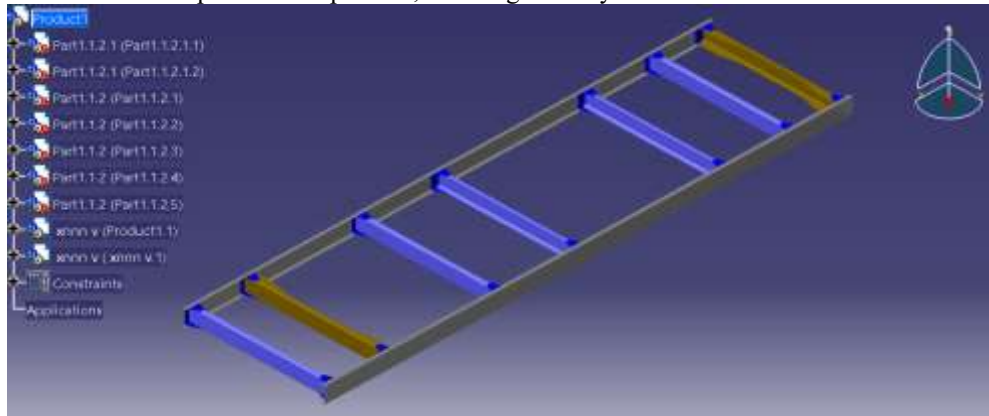
***Fig:**22 "C"-Section Chassis Frame Assemble on CATIA*

The different colors represent different components/parts of the model.

Drawing and Detailing
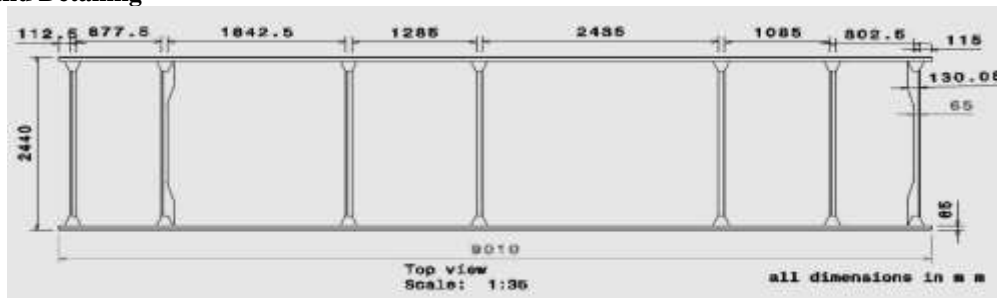
***Fig:** 23 Top view details*
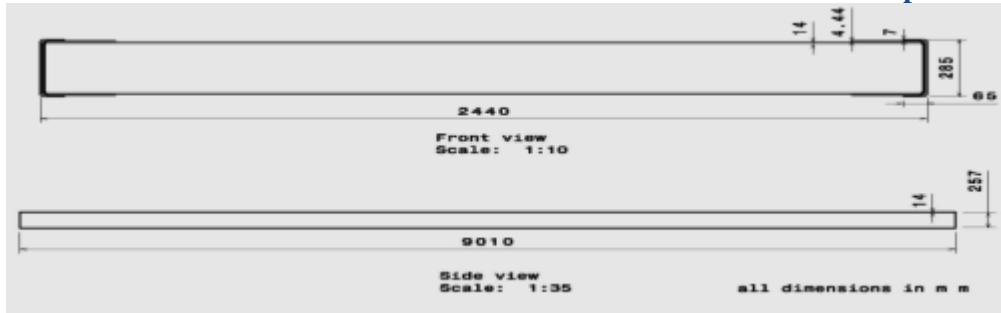
Fig: 24 Front and Side view details

"I"-Section Chassis Frame
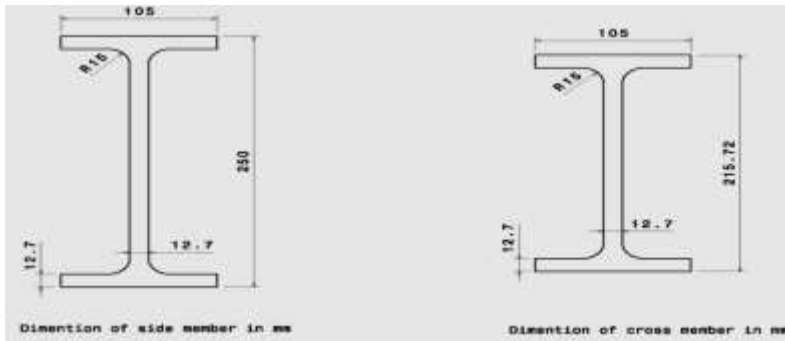
Fig: 25 Dimension of Side and Cross members of I- section
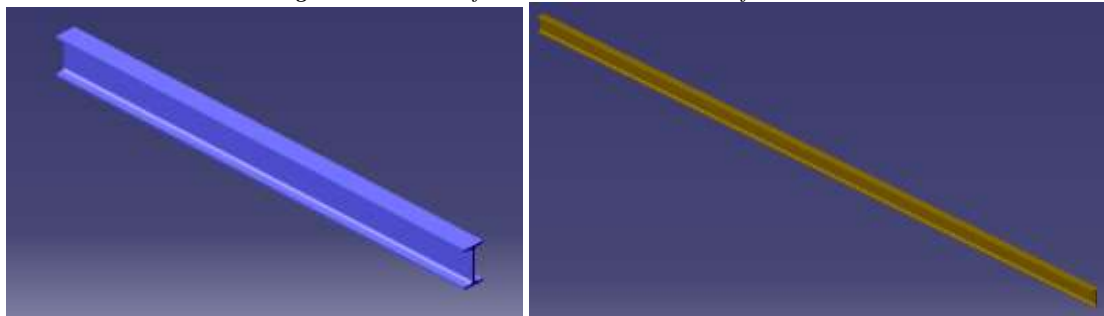
Cross members   *Side members*
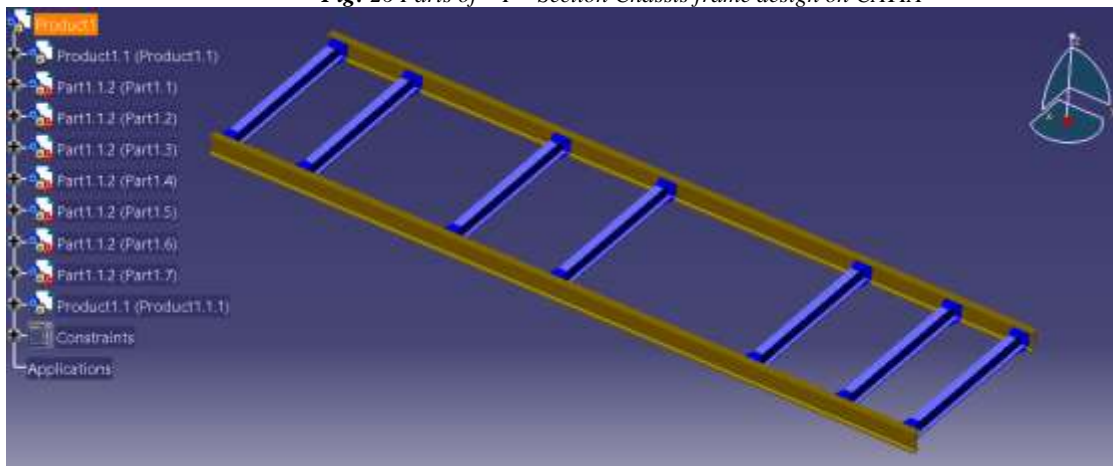*Fig: 26 Parts of "I"- Section Chassis frame design on CATIA*

Fig: 27 Complete Assemble of parts on CATIA

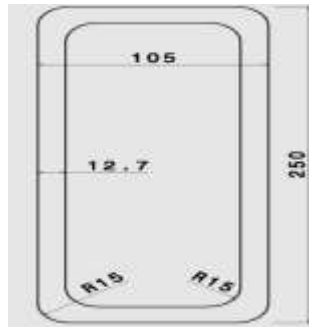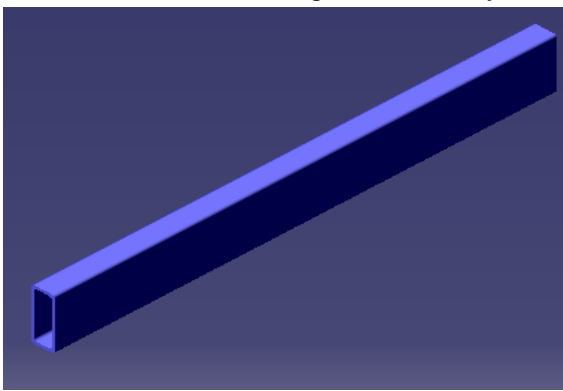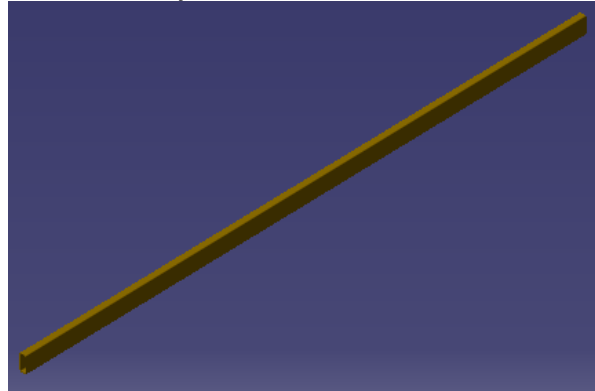The different colors represent different components/parts of the model.

Hollow rectangular (box)-section chassis

Fig: 28 Dimension of Side and Cross members of Box- section

Cross members   Side members

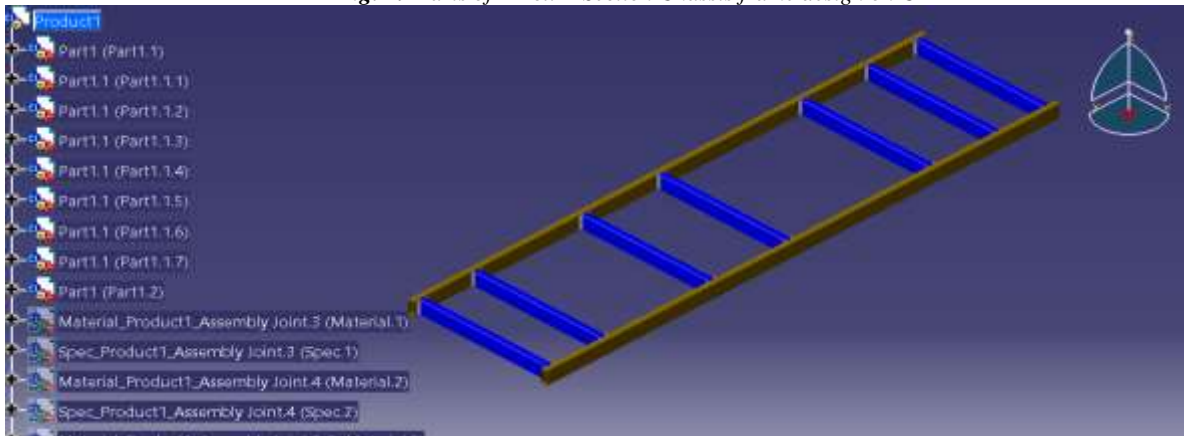*Fig: 29 Parts of "Box"- Section Chassis frame design on CATIA*

Fig: 30 Complete Assemble of parts on CATIA

The different colors represent different components/parts of the model.

STRUCTURAL ANALYSIS

For doing analysis of the model created in CATIA V5, we used the finite element solver ANSYS 14.0. ANSYS is a general purpose finite element analysis (FEA) software package. The geometric model created in CATIA is implemented in ANSYS as shown in figs.31, 36 and 39.

Meshing

At this point ANSYS understands the makeup of the part. Now define how the modeled system should be broken down into finite pieces. A meshing plan was determined to outline a continuous mesh. Using planar shell elements, the finite elements were meshed from all the geometric 2D surfaces of each component into their corresponding

finite element component. The elements used for the meshing were 2D higher order triangle or quadrilateral elements. The uses for these elements were in the calculations of plane strain and plane stress. This choice of element provides information on local stress and strain for thin walled structures, such as the ladder frame. The meshed model of chassis frame is shown in fig.32.

Loads and Boundary Conditions

Loads and boundary conditions are used to create loading and testing parameters needed to simulate realistic driving conditions of the vehicle. The loads applied to the vehicle consider the highest tolerable forces to a chassis structure that would cause irreversible damage. The highest tolerable forces should cover bending, torsional, lateral and vibrational formats to cover the full spectrum of potential loads on a vehicle. These will be used to simulate driving cases such as driving over potholes, bumpy roads, aggressive cornering and large accelerations (including braking). The constraints of the model depend on both the connectivity of the vehicle components and the particular loading case. For this model the total load acting on chassis frame is 306562 N as shown in fig.33.
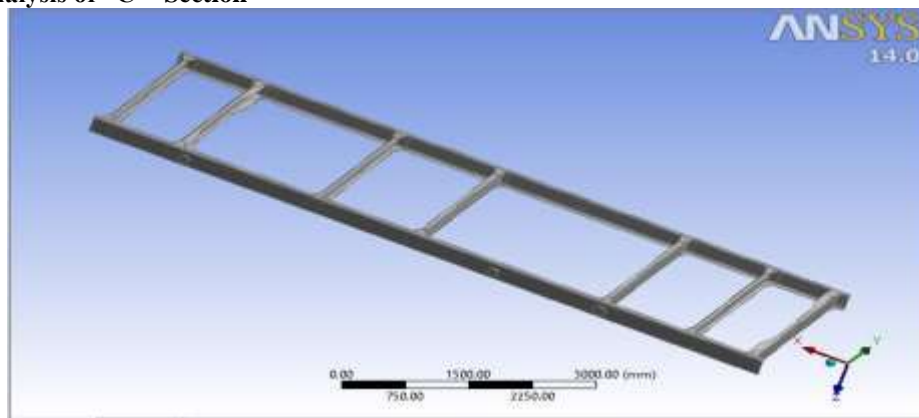
Structural Analysis of "C"- Section
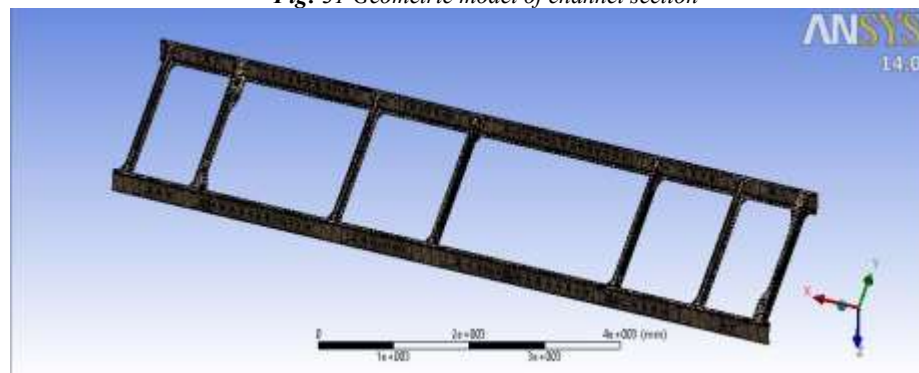

Fig: 31 Geometric model of channel section
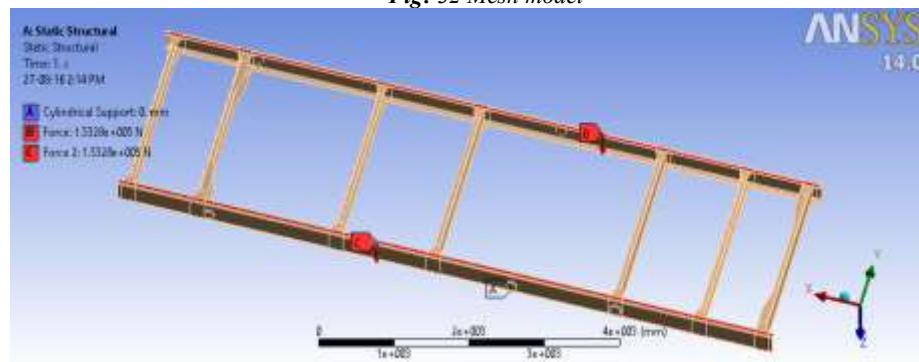

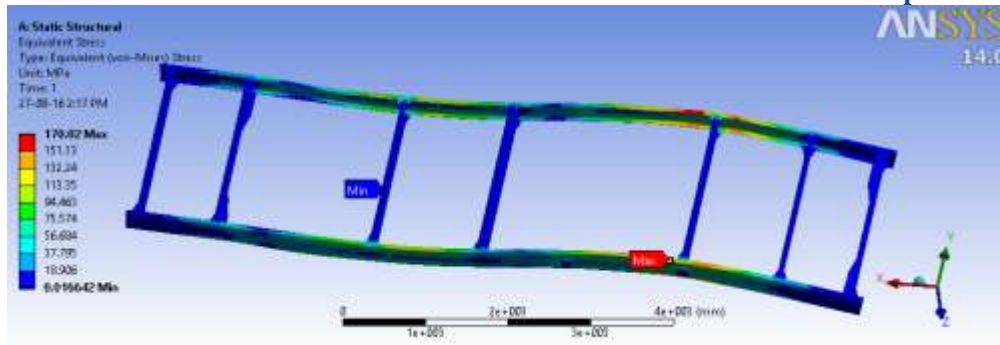*Fig: 32 Mesh model*

Fig: 33 Boundary Conditions

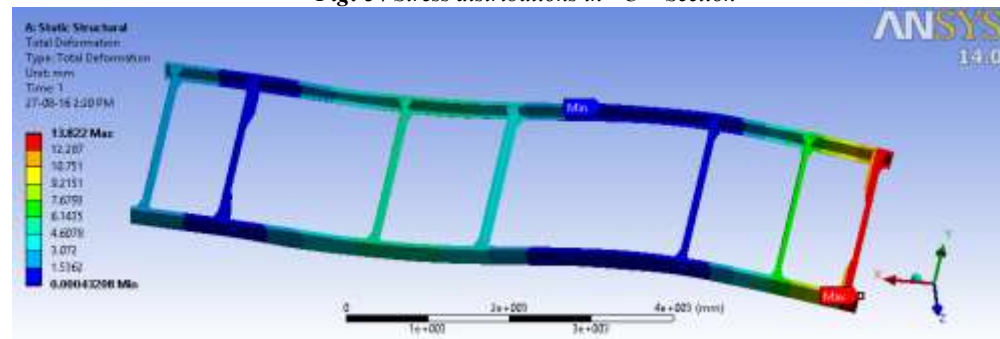*Fig:* 34 Stress distributions in "C"- Section

Fig: 35 Displacement pattern in "C"- Section
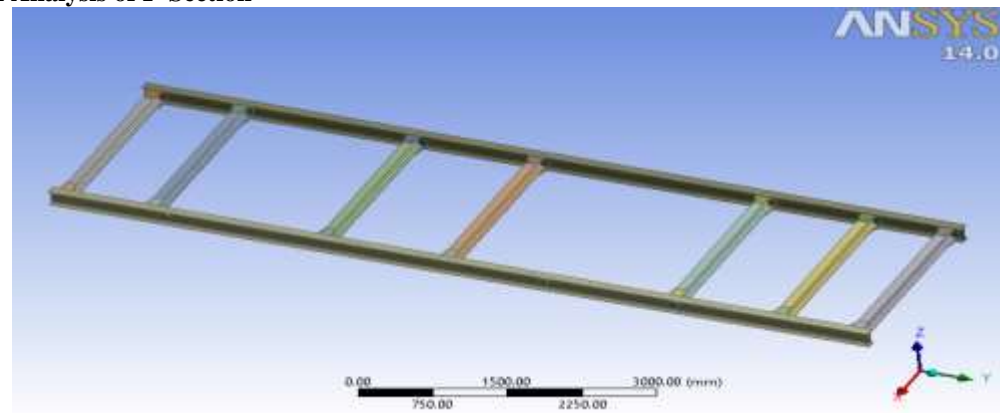
Structural Analysis of I- Section
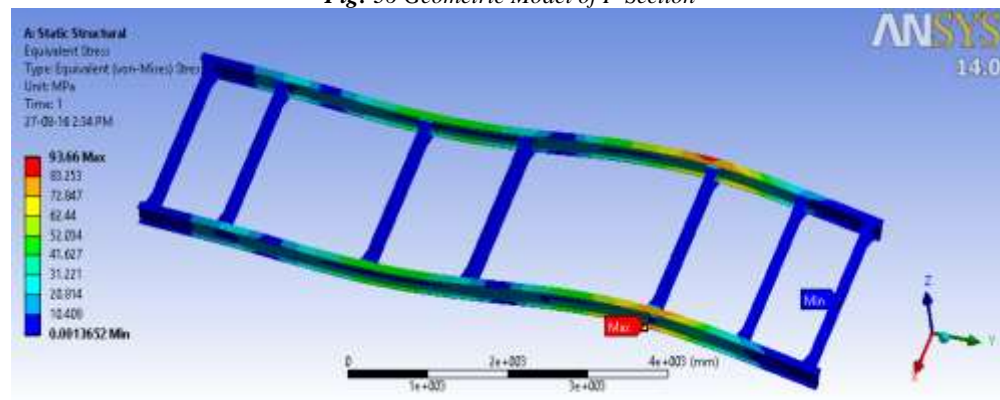

Fig: 36 Geometric Model of I- Section

Fig: 37 Stress distributions in I- Section

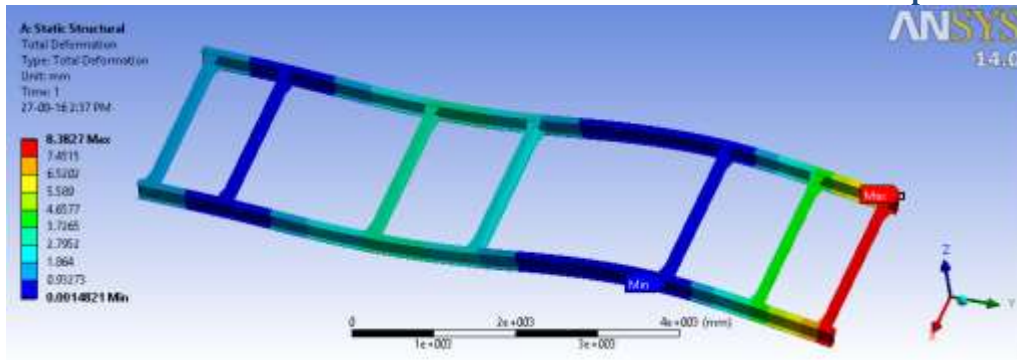***Fig:*** *38 Displacement pattern in I- Section*
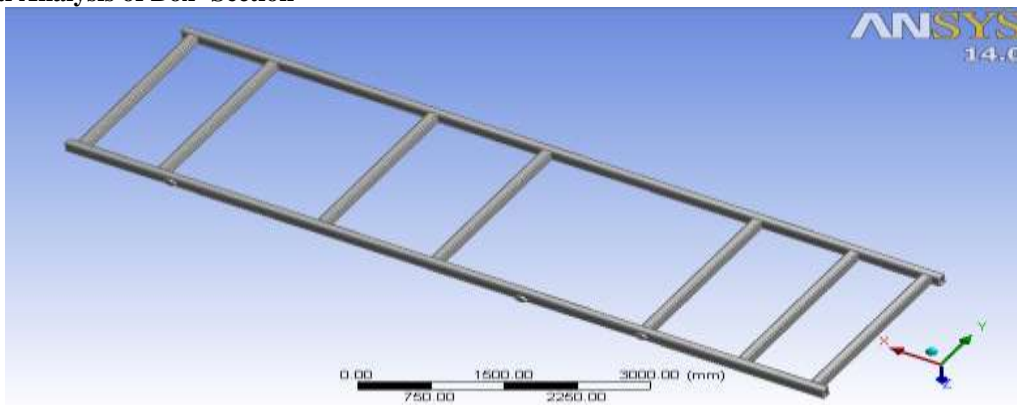
Structural Analysis of Box- Section

Fig: *39 Geometric Model of Box- Section*
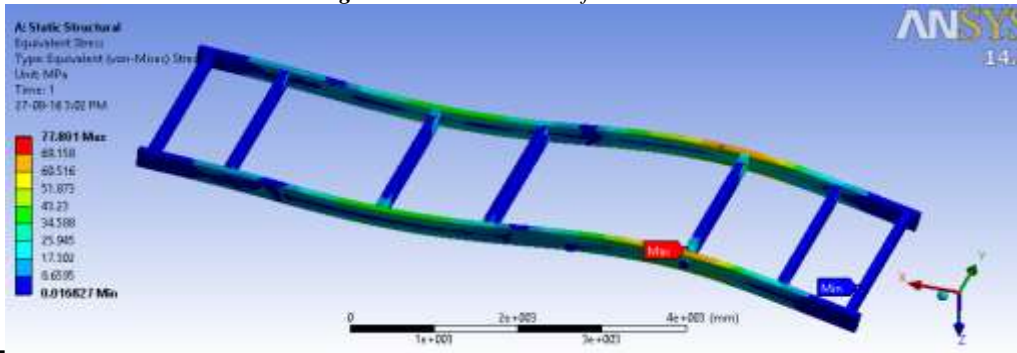

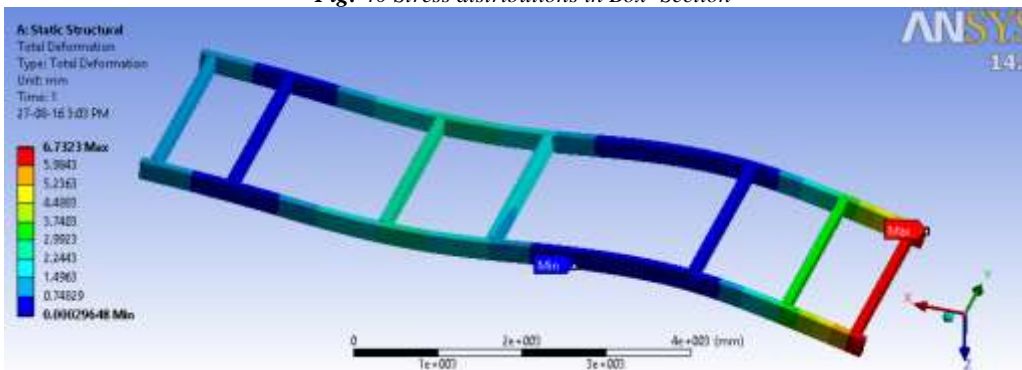***Fig:*** *40 Stress distributions in Box- Section*

Fig: *41 Displacement pattern in Box- Section*

RESULTS AND DISCUSSION

The stress distribution and deformation pattern for the C-channel cross section are shown in Figs. 34 and 35.The stress distribution and deformation pattern for the I-Cross section is depicted in Figs. 37 and 38. The stress distribution and deformation pattern for the Box-Cross section is depicted in Figs. 40 and 41.

Table 2: FE Analysis Results

Section/ Parameter	Weight(kg)	Stress(N/mm^2)	Deformation(mm)
"C"	838.89	170.02	13.822
"I"	1548.2	93.660	8.3827
Box	2228.4	77.801	6.7323

From the results, it is 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., 6.7323 mm and stress is 77.801 N/mm^2 in all the three type of chassis of different cross section.

FEA Model Validations

The analytical results of deformation and stress distribution computed using Eqs. (5) and (6) respectively as tabulated in Table 3. Table 3 shows the stress distribution and deformation values for different cross sections and compared with the analytical values.

Table 3: Comparsion of Results

Section	Analytical Method		FE Analysis	
	Stress (N/mm^2)	Deformation (mm)	Stress (N/mm^2)	Deformation (mm)
"C"	192.67	14.839	170.02	13.822
"I"	99.54	8.739	93.660	8.3827
Box	80.1498	7.037	77.801	6.7323

It can be inferred from the tabulation that the numerical values of the stress and deformation are less than analytical results; these represents the FEA models are within permissible limits, so the all design are safe.

CONCLUSION

The existing heavy vehicle chassis of TATA 2518 TC is taken for design and analysis with different cross sections. The model of the chassis was created in CATIA V5 and analyzed with ANSYS 14.0 for same load conditions. After analysis a comparison is made between chassis section and other sections in terms of deformation and stresses, to select the best one.

From the results, it is 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., 6.7323 mm and stress is 77.801 N/mm^2 in all the three type of chassis of different cross section. So, in different cross sections of the chassis Box-section chassis is suitable for the heavy trucks.

Finally the analysis using different cross sections has been successfully accomplished. The work not only provides an analysis of the chassis but also presents the scope for its modification in actual. Also the optimized chassis is capable to carry the loads beyond the previous payload.

Future Scope of Research

In future, for this heavy vehicle future research might attempt to consider different materials of chassis frame and considering different parameters such as:

➢ Normal strain, shear strain, shear stress, thermal stress, strain energy, stiffness (both bending and torsion) and fatigue life etc.

Some other analysis would be done like: Modal analysis and fatigue failure analysis etc.
Also the chassis frame would be optimized by modifying the dimensions of chassis frame cross sections.

REFERENCES
[1] Mr. Rahul L. Patel, Mr. Divyesh B. Morabiya and Mr. Anil N. Rathour "Weight optimization of chassis frame using Pro-Mechanica" ISSN: 2348-8360, SSRG *International Journal of Mechanical Engineering (SSRG-IJME)* - vol. 1 Issue 8, pp.4-9, December 2014.

[2] Goolla Murali, Subramanyam B and Dulam Naveen "Design Improvement of a Truck Chassis based on Thickness"*Altair Technology Conference, Sigma Consultancy Service India* pp1-6, 2013.

[3] S.Prabakaran and K.Gunasekar "Structural Analysis of Chassis Frame and Modification for Weight Reduction" ISSN: 2277-9655, *International Journal of Engineering Sciences & Research Technology (IJESRT)* - vol. 3 Issue 5, pp.595-600, May, 2014.

[4] B. Ramana Naik and C. Shashikanth "Strength Analysis on Automobile Chassis" ISSN: 2348-4845, *International Journal & Magazine of Engineering, Technology, Management and Research* Vol. 2 Issue 11, pp. 1494-1501, Nov.-2015.

[5] Kamlesh Y. Patil and Eknath R. Deore "Stress Analysis of Ladder Chassis with Various Cross Sections" ISSN: 2278-1684, *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)* Vol. 12, Issue 4, pp. 111-116, August- 2015.

[6] Abhishek Sharma, Mr. Pramod Kumar, Abdul Jabbar and Mohammad Mamoon Khan "Structural Analysis of a Heavy Vehicle Chassis Made of Different Alloys by Different Cross Sections" ISSN: 2278-0181, *International Journal of Engineering Research & Technology (IJERT)* Vol. 3 Issue 6, pp. 1778 1785, June-2014.

[7] N. Lenin Rakesh, K. Gowtham Kumar and Dr. J. Hameed Hussain "Design and Analysis of Ashok Leyland Chassis Frame Under 25 Ton Loading Condition" ISSN: 2319-8753, *International Journal of Innovative Research in Science, Engineering and Technology*, Vol. 3, Issue 11, , pp. 17546-17551, November 2014.

[8] Roslan Abd Rahman, Mohd Nasir Tamin and Ojo Kurdi "Stress Analysis of Heavy Duty Truck Chassis as a Preliminary Data for its fatigue life Prediction using FEM" *Journal Mekanikal*, vol. 26,pp. 76-85, December-2008.

[9] Manpreet Singh Bajwa, Sinthiya Pundir and Amit Joshi "Static Load Analysis of Tata Super Ace Chassis and Stress Optimisation Using Standard Techniques" ISSN: 2320-2092*, International Journal of Mechanical and Production Engineering*, Vol. 1, Issue- 3, pp. 41-45, Sept-2013.

[10] R. L. Patel, K.R. Gawande and D.B. Morabiya "Design and analysis of chassis frame of TATA 2516TC" ISSN: 2321-9653, *International Journal for Research in Applied Science and Engineering Technology(IJRASET)*- Vol. 2 Issue III, pp.115-119, March 2014.

[11] Patel Vijay kumar V, Prof. R. I. Patel "Structural Analysis of Automotive Chassis Frame and Design Modification for Weight Reduction" ISSN: 2278-0181*, International Journal of Engineering Research & Technology (IJERT)* Vol. 1 Issue 3, pp.1-6, May-2012.

[12] Singarajan Nagammal Vijayan and Sathiavelu Sendhilkumar "Structural Analysis of Automotive Chassis Considering Cross-Section and Material" *International Journal of Mechanical Engineering and Automation* Vol. 2, Issue-8, pp. 370-376, August-2015.

[13] Chintada.Vinnod babu , Chiranjeeva Rao.Seela and Vykunta Rao.Matta "Structural Analysis of Eicher 11.10 Chassis Frame" ISSN: 2231-5381, *International Journal of Engineering Trends and Technology (IJETT)* Vol. 22, pp. 315-318, Number 7- April 2015.

[14] Tushar M. Patel, Dr. M. G. Bhatt and Harshad K. Patel "Analysis and validation of Eicher 11.10 chassis frame using Ansys" ISSN: 2278-6856, *International Journal of Emerging Trends & Technology in Computer Science (IJETTCS)* Vol. 2, Issue 2, pp. 85-88, March - April 2013.

[15] Chinmay Potdar, Amey Pise, Aishwarya Dubey and Sushrut Jadhav "Evaluation of Deflection Due to Bending of N1 Type Vehicle Chassis Members Using Static Finite Element Analysis" *Proceedings of 46th IRF International Conference, Pune, India*, ISBN: 978-93-85832-97 pp.78-81, 27th December 2015.