International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 10, October-2016

Modeling, Analysis & Optimization of TATA 2518 TC Truck Chassis Frame using CAE Tools

Akash Singh Patel
Scholar of Master of Technology,
Mechanical Engineering Department,
UCER, Allahabad, India

Atul Srivastava
Assistant Professor,
Mechanical Engineering Department,
UCER, Allahabad, India

***Abstract*— Chassis is an important part of automobile. The chassis acts as the backbone of a heavy vehicle which carries the maximum load for all designed operating conditions. Role of the chassis 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 and materials of chassis frame. 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.

In the present work, the dimension of the TATA 2518TC chassis is used for the structural analysis of the heavy vehicle chassis with three different alloys subjected to the same conditions of the steel chassis. The three different material used for the chassis are selected as grey cast iron, AISI4130 alloy steel and ASTM A710 STEEL GRADE A based on their easy availability and known properties. A three dimensional solid Modeled in the CAE software CATIA V5 and analyzed in ANSYS 14.0. The numerical results are validated with analytical calculation considering the stress distribution, deformation. Also two standard optimization techniques (Boxing & Reinforcement) are used to optimize the chassis frame.

Keywords*— *Heavy truck chassis frame, CATIA, ANSYS, FEM, Assembly weight, stress, deformation.

I. 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].

A. Principal Functions

- The chassis frame 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.
- It must absorb engine & driveline torque.

While fulfilling these functions, the chassis should be light enough to reduce inertia and offer satisfactory performance. It should also be tough enough to resist fatigue loads that are produced due to interaction between the driver, engine, power transmission and road conditions [3].

B. Types of Chassis Frames

There are three types of chassis frames:

1) Conventional Frame

It has two long side members and 5 to 6 cross members joined together with the help of rivets and bolts.The frame sections are used generally [4].

- Channel Section - Good resistance to bending.
- Tabular Section - Good resistance to Torsion.
- Box Section - Good resistance to both bending and Torsion.

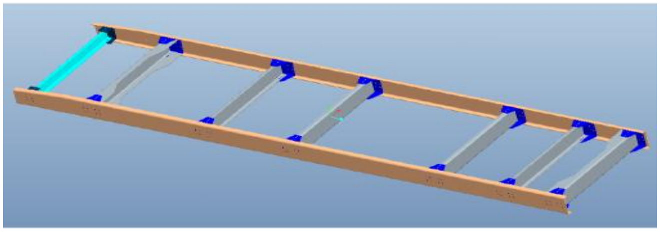


Fig. 1. Conventional frames

2) Integral Frame

This frame is used now days in most of the cars. There is no frame and all the assembly units are attached to the body. All the functions of the frame carried out by the body itself. Due to elimination of long frame it is cheaper and due to less weight most economical. Only disadvantage is repairing is difficult [4].

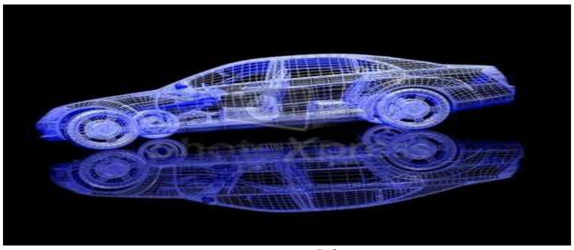


Fig. 2. Integral frames

3) Semi - Integral Frame

In some vehicles half frame is fixed in the front end on which engine gear box and front suspension is mounted. It has the advantage when the vehicle is met with accident the front frame can be taken easily to replace the damaged chassis frame. This type of frame is used in FIAT cars and some of the European and American cars [4].

Fig. 3. Semi - Integral frame

II. LITERATURE REVIEW

A. Literature Survey

There are some 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 (C, I and Box sections) 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 and 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.

Sharma et al [4] 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.

Bajwa et al [6] have studies the Static Load acting on Tata Super Ace Chassis and optimized the stresses Using Standard Techniques. They used two standard methods and the analysis is done to observe the reduction in the stress levels using CATIA and ANSYS Software's. (a.) Optimisation using boxing technique: It is the addition of 3 mm or thicker plate by welding it into the opening of C channel to form a box section. The maximum stress intensity and von mises stress are reduced to 103.47 MPa and 94.641 MPa respectively. The total deformation is also reduced to 1.588 mm. (b.) Optimization using reinforcement technique: It is the practice of providing a cover plate of 3 mm thickness and 180 mm length either internal or external on the side members at the highly stressed regions. In this case, stress intensity and von mises stress are reduced to 80.676 MPa and 74.203 MPa respectively. Total deformation is reduced to 1.4976 mm.

Patel et al [7] designed and analyze the chassis frame of TATA 2516TC. They design the chassis taking C Section using structural steel (St37) material on Pro-E Software and find out maximum von mises, strain, displacement and maximum shear stress using ANSYS Software and compare with allowable strength of material and found within limit therefore, the design is safe.

Singarajan Nagammal Vijayan and Sathiavelu Sendhilkumar [10] have studies and used the dimension of the existing heavy vehicle chassis of EICHER10.9 and taken for design and analysis with different cross sections for different materials like Carbon/Epoxy composites. The model of the chassis was created in Pro-E and analyzed with ANSYS for same load conditions. The results of the steel and polymeric composites material with cross section C, I, and Box are performed. It is inferred that by employing a carbon epoxy composites heavy vehicle chassis for same load carrying capacity, there is a reduction in weight when compared to steel. Carbon epoxy induces low deformation and stress distribution when compared to steel especially in "I" section.

The results show that carbon epoxy composites with "I" section has superior strength to withstand same load.

Babu et al [11] have studies the stresses and deformation developed in chassis frame of EICHER 11.10 by considering three different materials like St52, Ni-Cr Steel and CFRP in each case. The chassis is modeled in PRO-E and finite element analysis has been done in ANSYS. Finally they concluded that when (a.) CFRP is used instead of St52 the deflection is reduced by 63.24% and also stress was reduced by 12.61%. (b.) CFRP is used instead of Ni-Cr the Steel deflection is reduced by 64.70% and also Stress is reduced by 12.40%.

B. 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 and materials 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 and deformation etc. The strength of the chassis is an important issue when the weight of the chassis is reduced.

C. 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 with three different alloys subjected to the same conditions of the steel chassis. The three different material used for the chassis are selected as grey cast iron, AISI4130 alloy steel and ASTM A710 STEEL GRADE A (CLASSIII) based on their easy availability and known properties. 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.

III. 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 0.565%C, 1.8% Si, 0.7%Mn, 0.045%P and 0.045%S. Other materials are [4]:

A. Grey cast iron

The composition of gray cast iron in terms of its entire constituent elements can be explained as follows:
C (up to 4%), S (up to 3%), Mn (0.8%), S (.07%), P (0.2%), Mo (up to 0.75%), Cr (0.35%), V (0.15%).

B. AISI 4130 alloy steel

AISI stands for the American Iron and Steel Institute, has given the designation to the steel alloy with the particular composition of material like AISI 4130, AISI 4140. AISI 4130 is also known as the chrome-moly alloy steel which stands for chromium-molybdenum alloy steel. The chemical composition of the AISI 4130 steel alloy is given as follows [4]:
C (0.28-0.33), Cr (0.8-1.1), Fe (97.3-98.22), Mn (0.4-0.6), Mo (0.15-0.25), P (up to 0.035), S (up to 0.04), Si (0.15-0.35),

C. ASTM A710 STEEL GRADE A (CLASS III)

ASTM stands for American Society for Testing and Materials and has given the designation to the alloy steel the grade of ASTM A710 steel with a particular composition. The chemical composition is given as follows [4]:
C (0.07), Mn (0.4-0.7), P (0.025), S (0.025), Si (0.035), Cr (0.60-0.90), Ni (0.7-1), Mo (0.15-0.25), Cu (1-1.3), Niobium (0.02),

TABLE. I PHYSICAL PROPERTIES OF MATERIALS

Material	Modulus of elasticity (GPa)	Density (kg/m3)	Tensile strength (MPa)	Yield strength (MPa)
Structure Steel ST37	210	7850	460	260
Grey cast iron	110	7200	240	-
AISI 4130 steel alloy	260	7798	1030	910
ASTM A710 Steel	205	7850	515	450

IV. ANALYTICAL CALCULATIONS

TABLE. II 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

A. 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 [5].

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

B. 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

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 10, October-2016

According to loading condition of the beam, a beam has a support of three axle means by three wheel axles C, D and E [5]. Total load reaction generated on the beam is as under:-

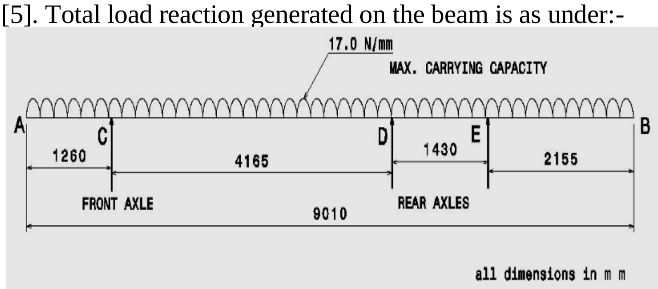


Fig. 4. Total load generated on the beam

C. Fixed End Moment

This is the indeterminate structure of beam. The fixed end moment considered as shown in fig.5 [5].

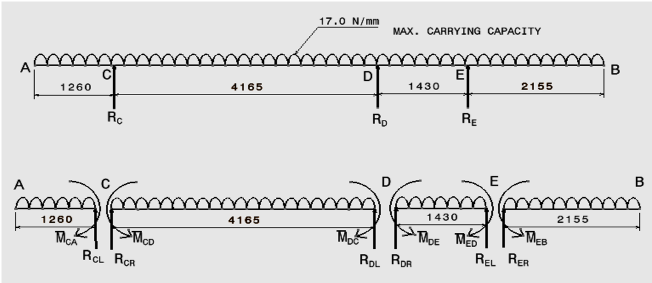


Fig. 5. Consideration of fixed end moments

$\bar{M}_{CA}$ = (17*1260*1260)/2 = 13494600 N mm
$\bar{M}_{CD}$ = (-17*4165*4165)/12 = -24575235.42 N mm
$\bar{M}_{DC}$ = +24575235.42 N mm
$\bar{M}_{DE}$ = (-17*1430*1430)/12 = -2896941.667 N mm
$\bar{M}_{ED}$ = +2896941.667 N mm
$\bar{M}_{EB}$ = (-17*2155*2155)/2 = -39474212.5 N mm

Total restraint moment at "C"
$\bar{M}_C = \bar{M}_{CA} + \bar{M}_{CD}$
= 13494600 -24575235.42 = -11037364.58 N mm

Total restraint moment at "E"
$\bar{M}_E = \bar{M}_{ED} + \bar{M}_{EB}$
= 2896941.667 -39474212.5= -36577270.83 N mm

For the span "CD"
$M_{CD} = -\bar{M}_{CA}$ = -13494600 N mm
$M_{DC} = \bar{M}_{DC} - \bar{M}_C\backslash 2 + 3EI/l\ i_b$
= 24575235.42 + 11037364.58\2 + 3EI/4165 i_b
= 30093917.71 + 3EI/4165 i_b

For the span "DE"
$M_{ED} = -\bar{M}_{EB}$ = 39474212.5 N mm
$M_{DE} = \bar{M}_{DE} - \bar{M}_E\backslash 2 + 3EI/l\ i_b$
= -2896941.667 + 36577270.83\2 + 3EI/1430 i_b
= 15391693.75 + 3EI/1430 i_b

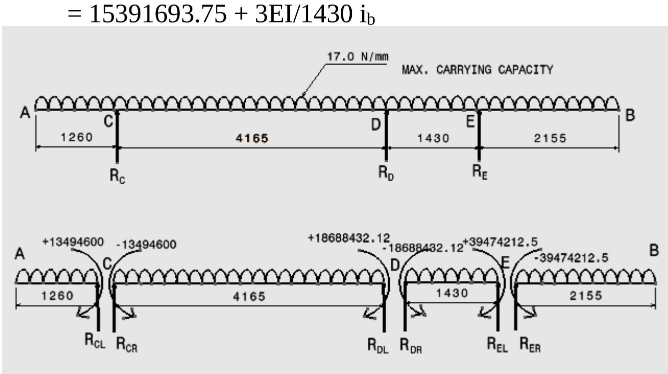


Fig. 6. Fixed end moments

Equilibrium condition at "D"
$\bar{M}_{DC} + \bar{M}_{DE} = 0$
30093917.71 + 3EI/4165 i_b + 15391693.75 + 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

D. Calculations for Reaction and Shear Force Diagram

R_{CL} = 17 * 1260 = 21420 N (↑)
R_{CR} = (17*4165)/2 + (13494600-18688432.12)/4165
= 34155.48 N (↑)
R_{DL} = 17*4165 – 34155.48 = 36649.52 N (↑)
R_{DR} = (17*1430)/2 + (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 (↑)

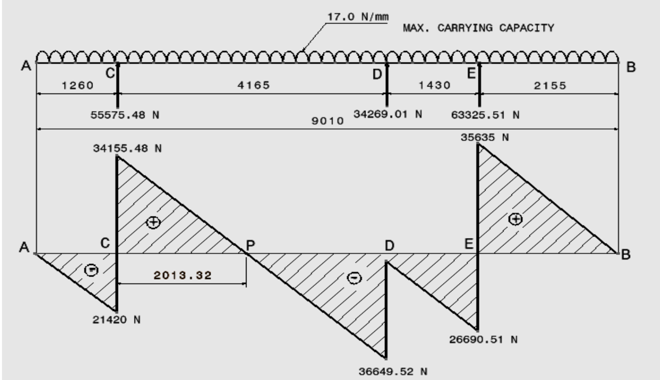


Fig. 7. Shear Force Diagram

E. Calculations for Bending Moment Diagram

M_A = 0 N mm
$M_C = -\bar{M}_{CA}$ = - 13494600 N mm
M_P = (17* 4165* 4165)/8 = 36862853.13 N mm
$M_D = M_{DE}$ = - 18688432.12 N mm
M_Q = (17* 1430* 1430)/8 = 4345412.5 N mm
$M_E = \bar{M}_{EB}$ = -39474212.5 N mm
M_B = 0 N

So the maximum bending moment occurs at "E"

$M_{max} = M_E$ = -39474212.5 N mm

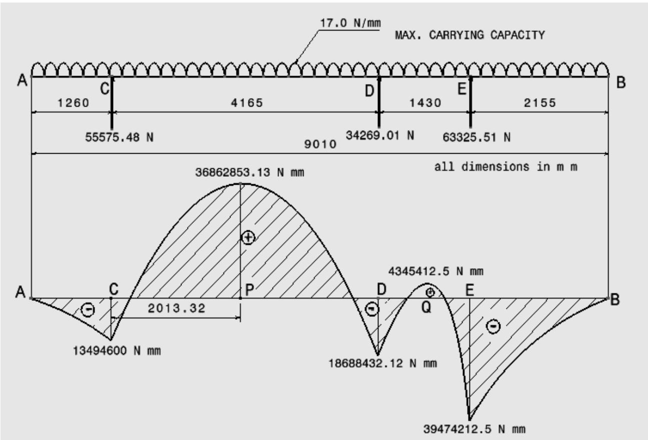


Fig. 8. Bending Moment Diagram

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 10, October-2016

F. Calculations for the Maximum Deflection

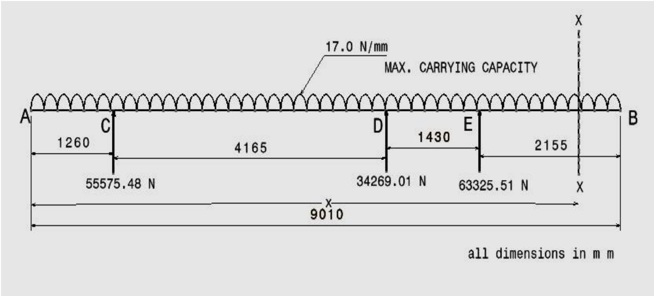


Fig. 9. Reaction generated on the beam

We consider a section x-x in "EB" span at x distance from A.

Taking moment of all forces about x-x section [5]

$M_{xx} = -8.5x^2 + R_C(x-1260) + R_D(x-5425) + R_E(x-6855)$

According to Macaulay's theorem

$M_{xx} = EI\,(d^2y/dx^2)$

$= -8.5x^2 + R_C(x-1260) + R_D(x-5425) + R_E(x-6855)$

On integrating with respect to x we get

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

Again integrating with respect to x we get

$EI\,y = -17x^4/24 + C_1x + C_2 + R_C(x-1260)^3/6 + R_D(x-5425)^3/6 + R_E(x-6855)^3/6$ …… ……………………………………1

Applying the boundary conditions

At x = 1260mm, y = 0

$0 = -17*1260^4/24 + C_1*1260 + C_2$ ……………………………2

At x = 6855mm, y = 0

$0 = -17*6855^4/24 + C_1*6855 + C_2 + 55575.48*5595^3/6 + 34269.01*1430^3/6$ ……………………………… 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

$y = (1/EI)[-17x^4/24 - 1.384*10^{10}x + 1.922*10^{13} + 55575.48(x-1260)^3/6 + 34269.01(x-5425)^3/6 + 63325.51(x-6855)^3/6]$ …… 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)

$y_A = (1.922*10^{13})/EI$

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

$y_B = (-9.0976 * 10^{13})/EI$

So the maximum deflection occurs at "B"

$y_{max} = y_B = (-9.0976 * 10^{13})/EI$ ………………………… 5

G. Stress and Deflection in Chassis Frame

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

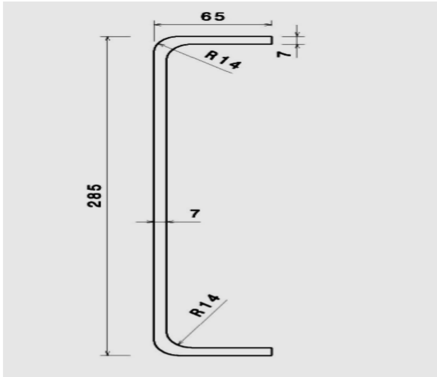


Fig. 10. Main "Channel" section

Moment of Inertia around the X – X axis:-

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

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

$= 29195623.92\ mm^4$

Section of Modules around the X – X axis:-

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

Basic Bending equations are as follow:-

$M/I = \sigma/y = E/R$ …………………………………… 6

Maximum Bending Moment acting on the Beam

$M_{max} = -39474212.5$ N mm

$Z = 204881.572\ mm^3$

Stress produced on the Beam

$\sigma = M/Z = -39474212.5/204881.572 = -192.669\ N/mm^2$

Maximum Deflection produced on the Beam

1) For structural steel ST 37

$E = 210000\ MPa = 2.10 \times 105\ N/mm^2$

$I = 29195623.92\ mm^4$

$y_{max} = -9.0976*10^{13}/EI$

$= -9.0976 * 10^{13} / (210000*29195623.92) = -14.839$ 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.

2) For Grey cast iron

$E = 110000\ N/mm^2$

$y_{max} = -9.0976*10^{13}/(110000*29195623.92) = -28.328$ mm

So 28.328 mm is safe.

3) For AISI 4130 steel alloy

$E = 260000\ N/mm^2$

$y_{max} = -9.0976 * 10^{13}/(260000*29195623.92) = -11.985$ mm

So 11.985 mm is safe.

4) For ASTM A710 steel grade A

$E = 205000\ N/mm^2$

$y_{max} = -9.0976*10^{13}/(205000*29195623.92) = -15.200$ mm

So 15.2 mm is safe.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 10, October-2016

V. 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.11. The procedure of modeling and analysis consists of [5]:

- 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.

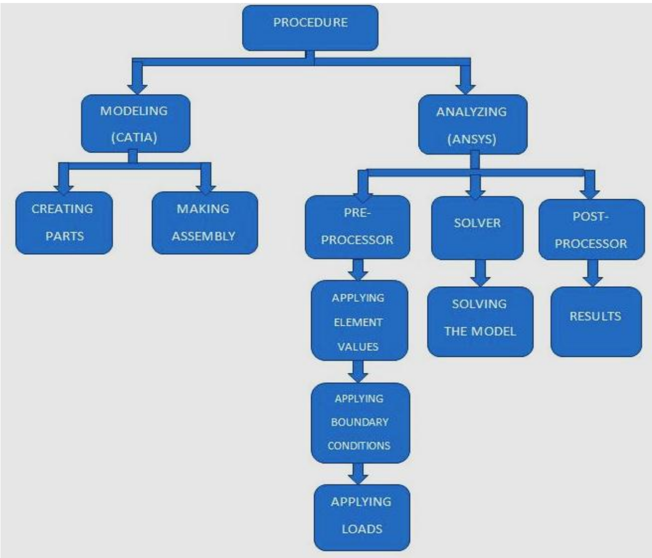


Fig. 11. Procedures for Modeling and Analysis

A. 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. In the design field, CATIA offers a solution to shape design, styling, surfacing workflow and visualization to create, modify, and validate complex innovative shapes from industrial design. The CATIA V3 was the first version, started in 1984 as its main 3D CAD tool. Then in 1992 CATIA V4 version was developed, then in 1995 V5 version. Science in 2008, the Dassault Systems released a latest version of CATIA V6. In 2014, Dassault Systems launched 3D experience Platform R2014x. CATIA is a best CAD tool in 3D and easily familiar to us [5].

B. ANSYS

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 [5].

ANSYS provides a cost-effective way to explore the performance of products or processes in a virtual environment. This type of product development is termed virtual prototyping in which user can iterate various scenarios to optimize the product long before the manufacturing is started. This enables a reduction in the level of risk, and in the cost of ineffective designs.

VI. 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, first the parts of model was build and then they are assembled to make complete geometric model.

A. 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. The parts of "C" section chassis frame are shown in fig.13.

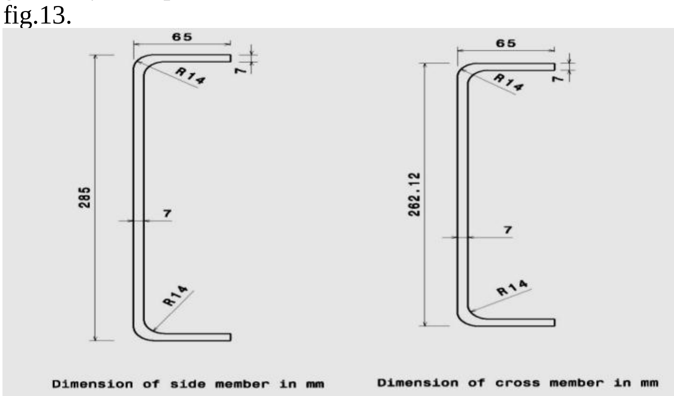
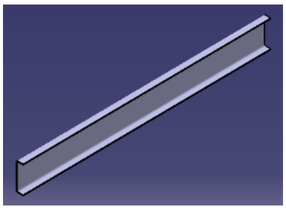
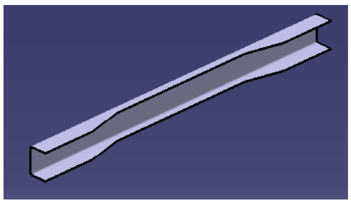


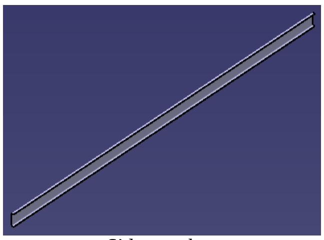
Fig. 12. Dimension of Side and Cross members of Channel section



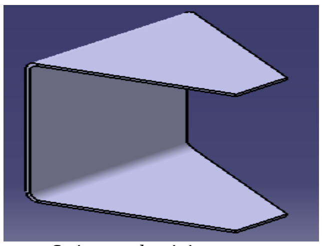
First type Cross members



Second type Cross members



Side members



Strips used at joints

Fig. 13. Parts of "C"- Section Chassis frame design on CATIA

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 10, October-2016

B. Making Assembly

Once each part and sub-assembly was completed they were combined together in an assembly as shown in fig14. 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 [5].

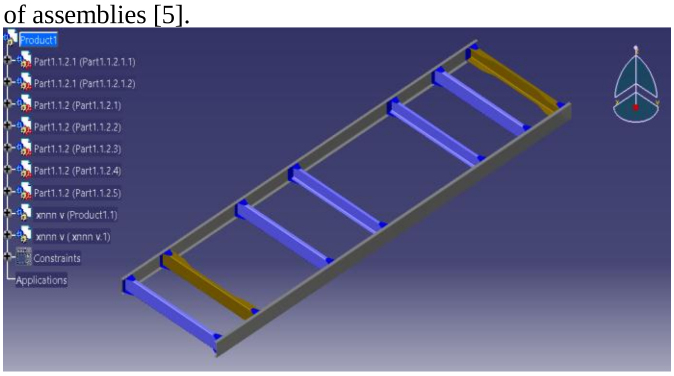


Fig. 14. “C”-Section Chassis Frame Assemble on CATIA

The different colors represent different parts of the model.

C. Drawing and Detailing

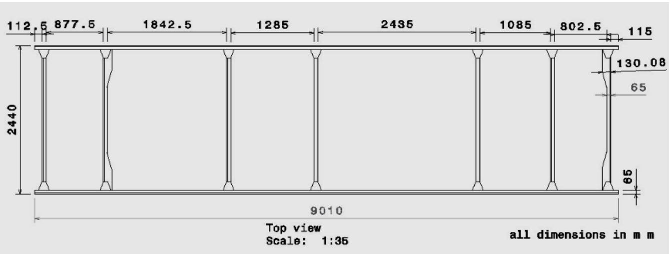


Fig. 15. Top view details

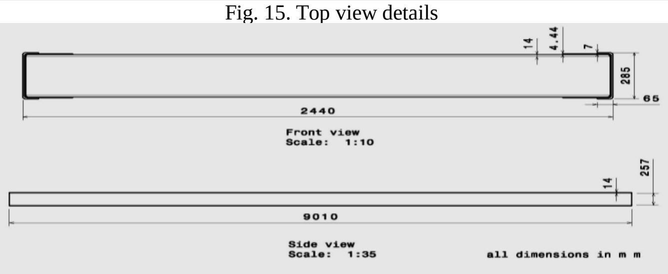


Fig. 16. Front and Side view details

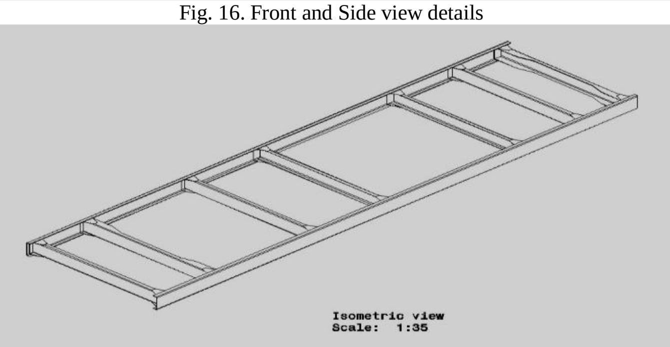


Fig. 17. Isometric views

VII. 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 fig.18.

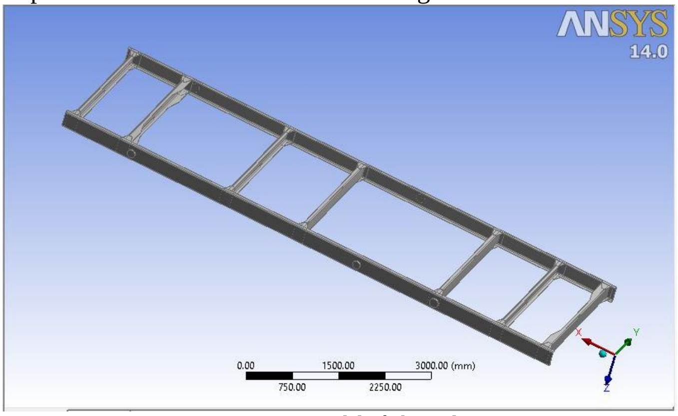


Fig. 18. Geometric model of channel section

A. 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 meshed model of chassis frame is shown in fig.19 [5].

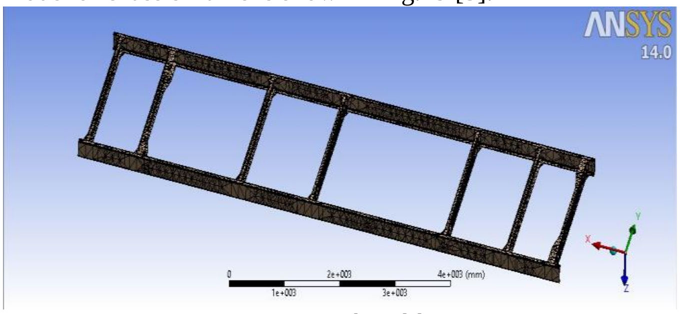


Fig. 19. Mesh model

B. 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.20 [5].

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 10, October-2016

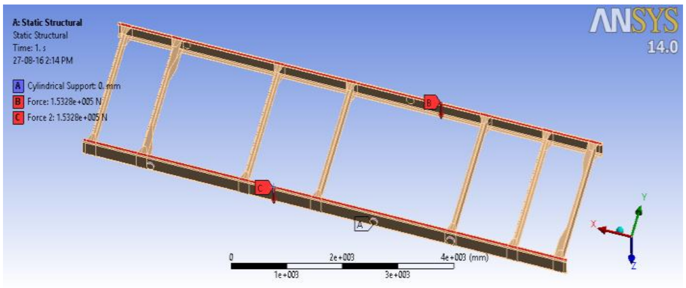


Fig. 20. Boundary Conditions

C. FE Analysis for Structure Steel ST 37:-

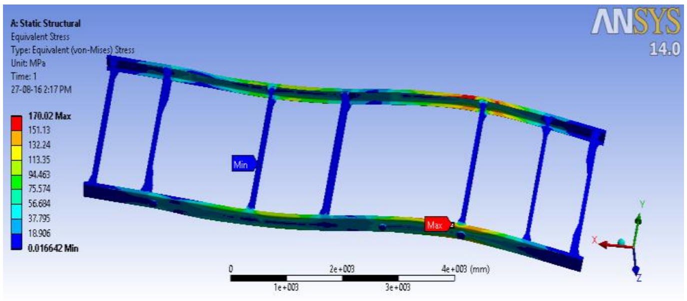


Fig. 21. Stress distributions for Structure Steel ST 37

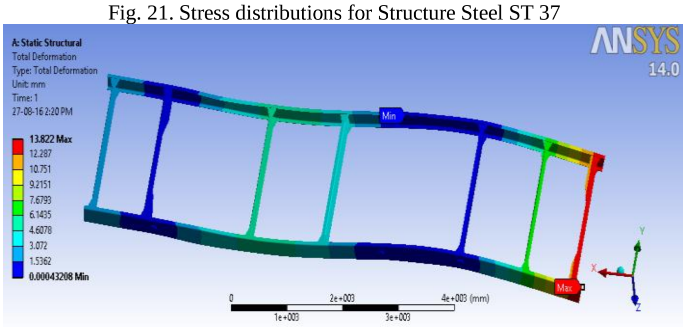


Fig. 22. Displacement pattern for Structure Steel ST 37

D. FE Analysis for Gray Cast Iron:-

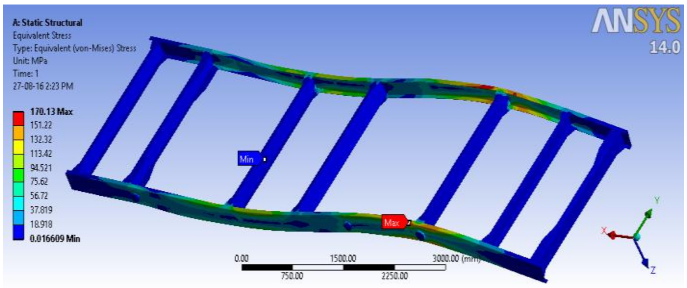


Fig. 23. Stress distributions for Gray Cast Iron

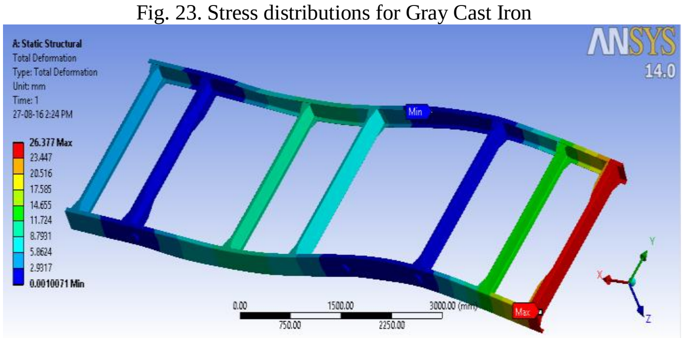


Fig. 24. Displacement pattern for Gray Cast Iron

E. FE Analysis for AISI 4130 Steel Alloy:-

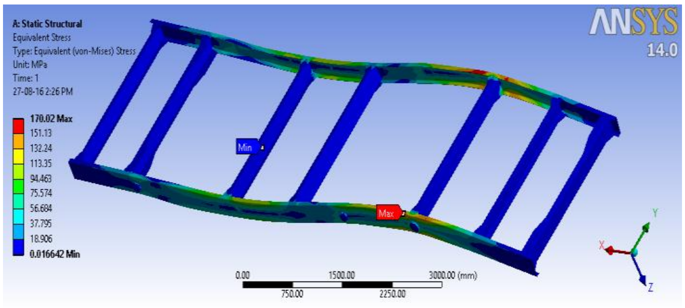


Fig. 25. Stress distributions for AISI 4130 Steel Alloy

Fig. 26. Displacement pattern for AISI 4130 Steel Alloy

F. FE Analysis for ASTM A710 steel grade A:-

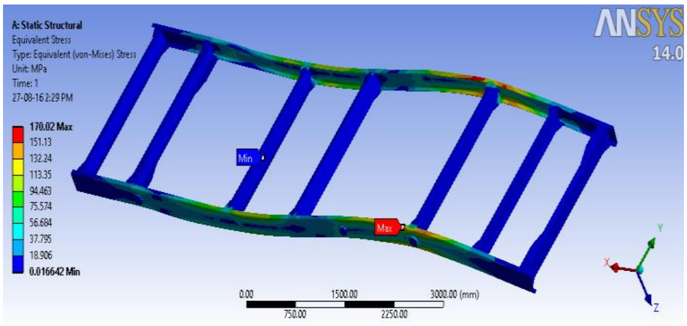


Fig. 27. Stress distributions for ASTM A710 steel grade A

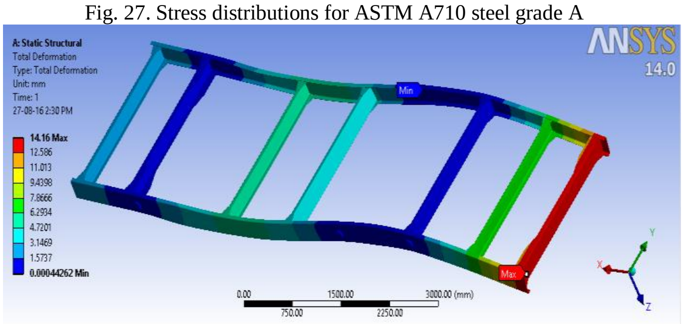


Fig. 28. Displacement pattern for ASTM A710 steel grade A

VIII. RESULTS AND DISCUSSION

The stress distribution and deformation pattern for the C-channel cross section for the different materials are shown in Figs. 21 to 28 for the same loading conditions.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 10, October-2016

TABLE. III COMPARATIVE ANALYSES OF STRUCTURE STEEL HEAVY VEHICLE CHASSIS AND OTHER ALLOY STEEL CHASSIS FOR "C"-CHANNEL SECTION

Section	Parameter	Structural Steel ST37	Grey Cast Iron	AISI 4130 Alloy Steel	ASTM A710 Steel
"C" Channel	Weight (kg)	838.89	769.43	833.33	838.89
	Stress (N/mm2)	170.02	170.13	170.02	170.02
	Deformation (mm)	13.822	26.377	11.167	14.160

The results shows that the default material for the chassis Structure steel ST37 shows strength equal to the AISI 4130 steel alloy but in case of the deformation AISI 4130 alloy is superior to structure steel material for the same loading conditions, which indicates the load carrying capacity of the AISI 4130 alloy steel material is higher than structural steel ST 37. So, for the consideration of alloy for the chassis AISI 4130 alloy is better than others. Also the AISI 4130 alloy is lighter than structural steel ST 37.

A. *FEA Model Validations*

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

TABLE. IV ANALYTICAL AND NUMERICAL VALUES OF STRESS AND DEFORMATION FOR DIFFERENT MATERIALS

Section	Material Type	Stress Distribution (MPa)		Deformation (mm)	
		Analytical	*Numerical*	*Analytical*	*Numerical*
"C" Channel	ST37	192.67	170.02	14.839	13.822
	GCI	192.67	170.13	28.328	26.377
	AISI 4130	192.67	170.02	11.985	11.164
	ASTM A710	192.67	170.02	15.200	14.160

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.

IX. OPTIMIZATION

Different practices are available for chassis modification. Here suitable changes are made in the design using two standard methods and the analysis is done to observe the reduction in the stress levels [6].

A. *Boxing optimization technique*

Boxing is the addition of 3 mm or thicker plate by welding it into the opening of C channel to form a box section as shown in fig.29. Since box section has good resistance to both bending and torsion [6].

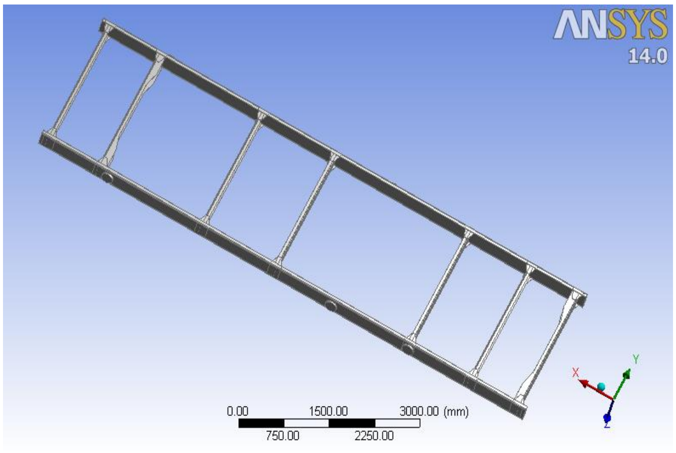


Fig. 29. Optimized Geometrical Model

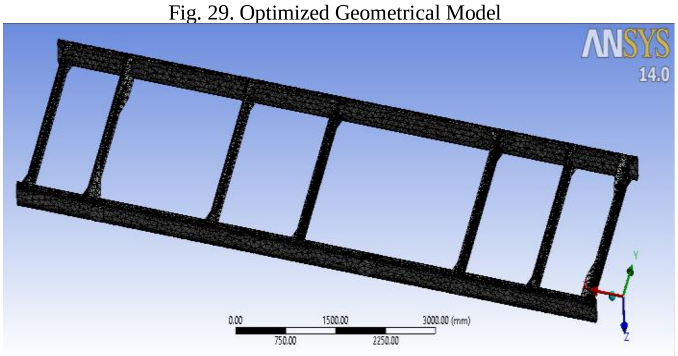


Fig. 30. Mesh Model

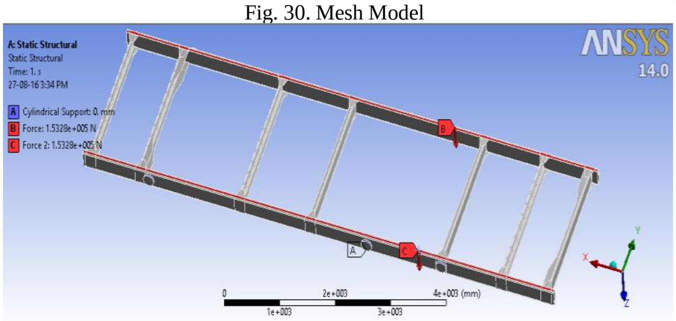


Fig. 31. Boundary Conditions

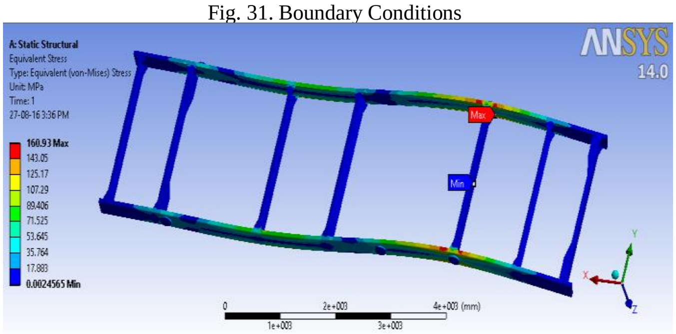


Fig. 32. Stress distributions

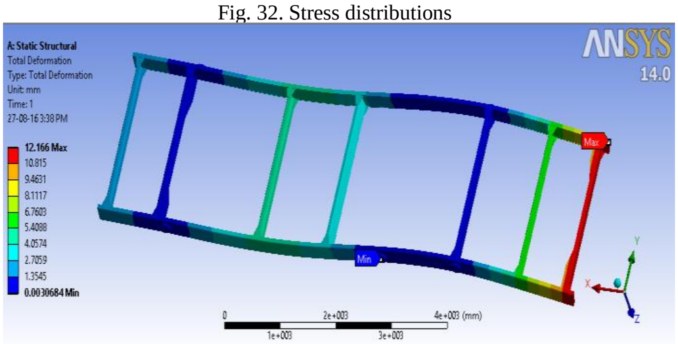


Fig. 33. Displacement pattern

B. Reinforcement Optimization Technique

Reinforcement is the practice of providing a cover plate either internal or external on the side members at the highly stressed regions. Here reinforcement of 3 mm thickness and 180 mm length is provided on the side members where the stress is maximum as shown in fig34 [6].

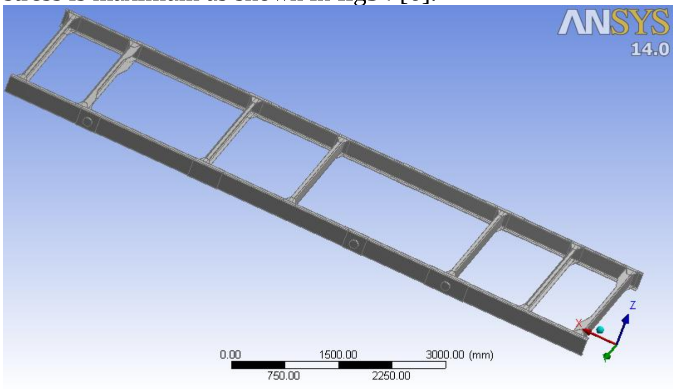


Fig. 34. Optimized Geometrical Model

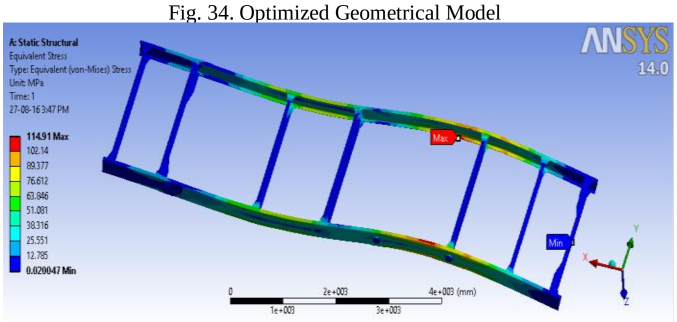


Fig. 35. Stress distributions

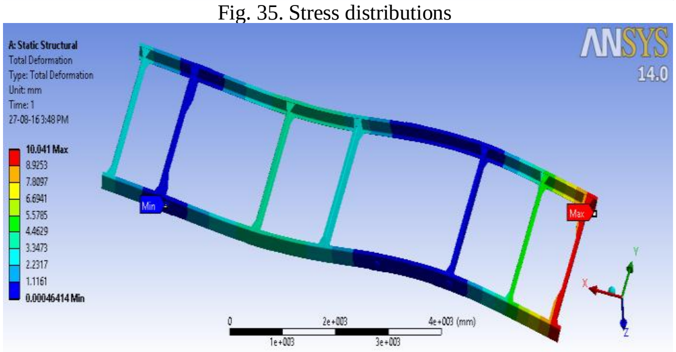


Fig. 36. Displacement pattern

The stress distribution and deformation pattern for the existing C-channel cross section are shown in Figs. 21 and 22 for the structural steel ST37 material. The stress distribution and deformation pattern for modified case 1and 2 are shown in Figs. 32, 33 and Figs. 35, 36 respectively, also numerical values tabulated as below.

TABLE. V STRESS AND DEFORMATION VALUES USING OPTIMIZATION TECHNIQUES

Parameters	Existing "C" Channel Section	Modified Case-1	Modified Case-2
Assembly Weight(kg)	838.89	973.54	889.39
Stress (N/mm2)	170.02	160.93	114.91
Deformation (mm)	13.822	12.166	10.041

The tabulated result shows that the values of stress and deformation reduced 5% (approx.), 12% respectively in case-1, and 32% (approx.), 27% respectively in case-2, so the method of reinforcement is found to be most effective.

X. CONCLUTIONS

The existing heavy vehicle chassis of TATA 2518 TC is taken for design and analysis for different materials. 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 existing structure steel chassis and alloy steel materials in terms of deformation and stresses, to select the best one.

The results shows that for all of the materials that have been tested in this text, AISI 4130 steel alloy shows better performance than all of the other metal alloys. It is seen that the default material for the chassis Structure steel ST37 shows strength equal to the AISI 4130 steel alloy but in case of the deformation AISI 4130 alloy is superior to structure steel material for the same loading conditions, which indicates the load carrying capacity of the AISI 4130 alloy steel is higher than structural steel ST 37. Also the AISI 4130 alloy is lighter than structural steel ST 37.

So, for the consideration of alloy for the chassis AISI 4130 alloys is better than others and it is suitable for the heavy trucks.

Finally the analysis and optimization using different standard optimization techniques 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.

A. Future Scope of Research

In future, for this heavy vehicle future research might attempt 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.

ACKNOWLEDGMENT

I wish to express my sincere thanks to project guide Mr. Atul Srivastava and all Faculties of Mechanical Department for his cordial support in every step and constant encouragement for this paper, help and guidance given by him time to time shall carry me to publish this paper.

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 pp. 1-6, 2013.

[3] S.Prabakaran and K.Gunasekar "Structural Analysis of Chassis Frame and Modification for Weight Reduction" ISSN: 2277-9655,

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 10, October-2016

International Journal of Engineering Sciences & Research Technology (IJESRT), vol. 3 Issue. 5, pp.595-600, May, 2014.

[4] 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.

[5] Akash Singh Patel and Jaideep Chitransh "Design And Analysis of TATA 2518TC Truck Chassis Frame With Various Cross Sections Using CAE Tools" ISSN: 2277-9655, International Journal of Engineering Sciences & Research Technology (IJESRT), vol. 5 Issue 9, pp.692-714, September, 2016.

[6] 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- 3pp. 41-45, Sept-2013.

[7] 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 IIIpp.115-119, March 2014.

[8] 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.

[9] 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 3pp.1-6, May-2012.

[10] 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.

[11] 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 Number 7pp. 315-318, April 2015.

[12] 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 2pp. 85-88, March - April 2013.

[13] Sandip Godse and Prof. D.A.Patel "Static Load Analysis of Tata Ace Ex Chassis and Stress Optimization Using Reinforcement Technique" ISSN: 2231-5381, International Journal of Engineering Trends and Technology (IJETT) Vol. 4 Issue. 7, pp. 3037-3039, July 2013.