International Journal of New Technology and Research (IJNTR)
ISSN: 2454-4116, Volume-6, Issue-11, November 2020 Pages 18-25

Comparative Analysis of Chassis of Various Sections Used In Heavy Vehicle Tata Truck 1618 Using Ansys

Mr. Durgesh Singh Bhadauriya, Prof. Kamlesh Kushwaha

***Abstract*— This work has been done to find out the better option of all part of chassis has been designed and develops with the help of CATIA and ANSYS software. But for check theoretical analysis has also done. All part is safe as a design point of view. Comparative analysis has also been done. Analyzing the results it found chassis that which section is better than various cross sections of chassis. The main objective of this document is to find out best suitable cross section design for TATA 1618 Model chassis and also optimizing the design. The work of this chassis is to design and developed by CATIA and its analyzing is done with ANSYS software and it also cross verified with theoretical mathematical calculation analysis. In this chassis, we use various sections i.e., 'C type', 'I type' and 'box type'. This thesis presents an analysis of the static stress that acting on the upper surface of the truck chassis. In this work we study about the distribution of stress and deformation that acting on the chassis and also for failure of critical parts. In this work used to numerical analysis method is finite element analysis.**

***Index Terms*— chassis, frame work, CATIA, ANSYS, analysis, stress, deformation.**

I. Introduction

Chassis is a French word with means "main structure of vehicles" or "frame parts". Chassis is to be considered as a major factor of an

automotive industry and also called as backbone of vehicle. One of the greatest challenges for an automotive industry is to provide high performance with less weight and longer life of vehicle parts, and all this at a low cost or reasonable prize. A vehicle without body is known as Chassis. The mechanical component of vehicle like engine, suspension, power plant, axles, transmission system, wheels, braking and steering system are bolted on the chassis. So chassis provides stiffness and strength for vehicle comp. to keep rigid and stable. The chassis consists a two member one is cross member (horizontal member) and is long member (vertical member). Stress analysis is done by 'Finite Element Method' (FEM) which is used locate peak stresses at critical point. The critical point is one who responsible for fatigue failure. The magnitude of the stress used to predict the life of span for any truck chassis.

Mr. Durgesh Singh Bhadauriya, Research Scholar, Mechanical Engineering, Maharana Pratap College Gwallior
Prof. Kamlesh kushwaha, Assistant Professor, Mechanical Engineering, Maharana Pratap College Gwallior

CHESSIS

Chassis is a "structure skeleton" of any truck. Chassis is a solid structure that holds wholly load of various part of any vehicle with its self-weight. Chassis is a French word with means "main structure of vehicles" or "frame parts". Automobile chassis indicates the lower body of

Vehicle including the suspension, tires, drive line, frame and engine. If it contains the upper body then it is known as vehicle as per the shape and design of the body.

II. RESEARCH METHODOLOGY

MODELING AND ANALYSIS OF CAE

A Part Modeling of 3D solid Model of the chassis 1618TC TATA 3D solid modeled in the CATIA and the analysis CAE software done in ANSYS

The procedure of modeling and analysis consists of

- To measuring of all the dimensions of TATA LPT 1618 TC chassis frame.
- Part modeling of chassis frame for different cross sections C, I and Box In three different Solid Models in create by CATIA software.
- Each solid model generate in ANSYS for FE Analysis for different value like: 'assembly weight', 'stress' and 'deformation' etc.
- Examination all dimension whether they are within acceptable limit or not for chosen materials.
- Result of Validation and Optimization
- Final results and Conclusions

GEOMETRIC MODELING IN CATIA

The CATIA is CAE (Computer Aided Editing) software in which we can create a solid model with the help various tools present in it. The chassis of TATA 1618 model has been developed or designed in CATIA with accurately dimensioning.

In this, we created both part of chassis frame separately i.e., (Long member and Cross member). Firstly, we had created a 2-D sketch of chassis with the facilitate of commands (circle, line, arc, fillet, conics etc) and used various relation attribute such as symmetry, tangency, concentricity, parallelism etc. After, fully defined 2-D sketch we made a 3-D model or SOLID part in the CATIA.

GEOMETRIC MODELING IN ANSYS- ANSYS software is used for analyzing the part we made in CATIA. We used the finite element solver ANSYS. Finite element analysis is a numerical method in which dividing the model into finite element. This software applies various equations on the model and check it behavior under those conditions. These results then are often present in tabulated or drawing value. This sort of study is usually used for designing and optimization of a structure for too complex to research by hand. Systems which will fit into this grouping are too complex thanks for their scale, geometry, or governing equations. ANSYS software is that the standard FEM tool with in the engineering.

BASIC STEPS FOR SOLVING ANY PROBLEM IN ANSYS:

Generate Geometry: Create a 2 Dimension and 3 Dimension geometry models of our chassis in CATIA Software. But this model can also be executed in ANSYS software also.

Material Apply: Secondly, we are applying the material on the CAD model of chassis of according our design need requirement

Meshing Generate: At this node & element in ANSYS understands the structure of the part.

Apply Boundary Conditions: Once the arrangement is fully calculated, the next task is to burden the scheme with constraints.

Obtain Solutions: This is the fact a step, because ANSYS desires to realize within what position, transient state, Steady etc.

Currently the Results: following the solution have been obtained, there are various ways to current ANSYS results, desire from various options such as graphs, tables, and contour plot.

III. CASE STUDY

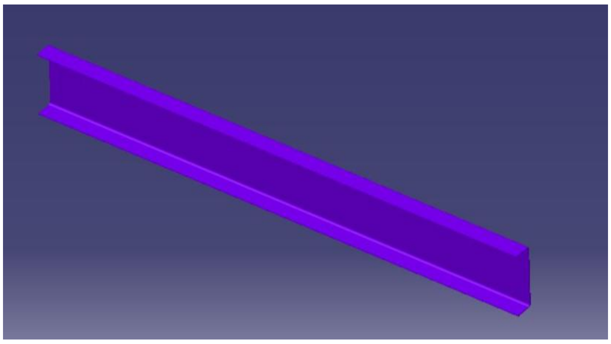
IDENTIFICATION OF PROBLEM

The current study has analyzed the various literatures. Once a careful analysis varied of assorted analysis studies conducted therefore for it have been found that there is the scope of optimizing various factors like weight, stress-strain values and deformation etc. by varied cross sections for modeling and analysis. Its paper describe the look, Structural analysis & optimization of the significant vehicle chassis with constraints of most stress, strain and deflection of chassis below most load. Our work is to vogue and analyze the significant vehicle chassis to cut back weight, stress, strain values and deformation etc.

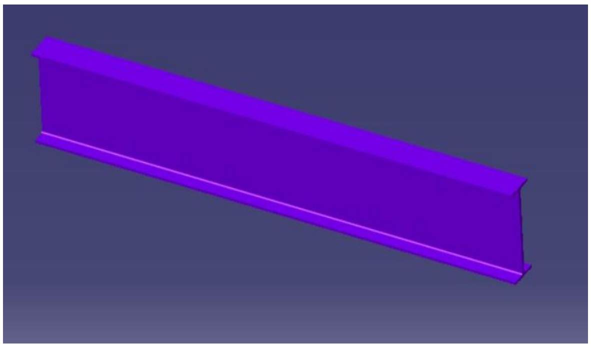
AIM OF WORK

In this thesis, the dimension of the TATA 1618 model. chassis is employed for the structural analysis for the heavy vehicle chassis by taking three different cross-sections, namely I, C and Rectangular Box (Hollow) cross sections subjected to identical loads condition. A 3-dimensional solid designed within the CAD software CATIA and analyzed in ANSYS. The numerical results are going to be valid with analytical calculation considering the stress distribution and deformation.

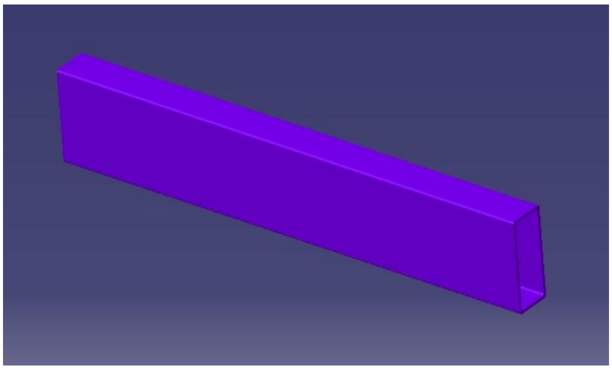
PARTS MODELING IN CATIA



PART MODELING OF “C” SECTION



PART MODELING OF “I” SECTION

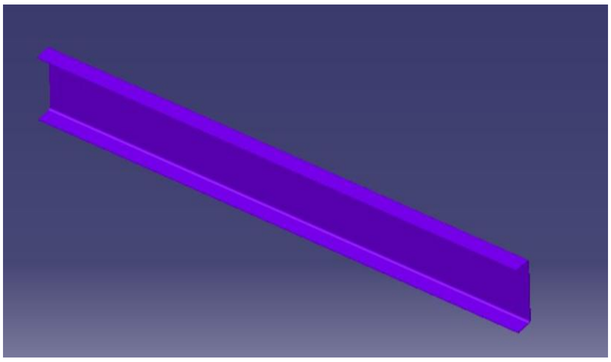


PARTMODELINGOF“BOX”SECTION

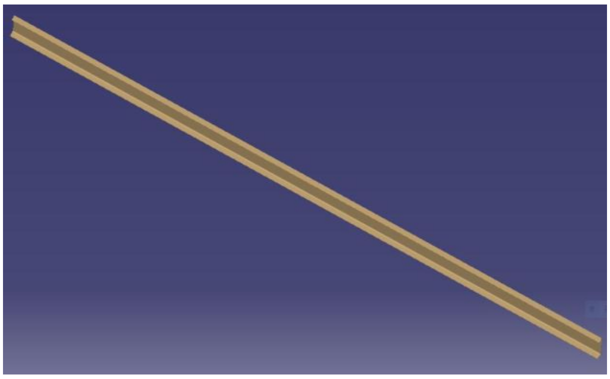
ASSEMBLE IN CATIA

Once sub-part or sub assembly has been completely designed in CATAI then, they are assembled in assembly function. While doing assembly of part (members) we use various relations in mates’ functions such as parallelism, concentricity, width, distance and more tools. With these mate’s relationship individual parts or member of chassis are assembled with equivalent relationship. Hence, our assembly is being ready for analyzing in ANSYS software for loading conditions.

ASSEMBLE IN CATIA OF “C”-SECTION

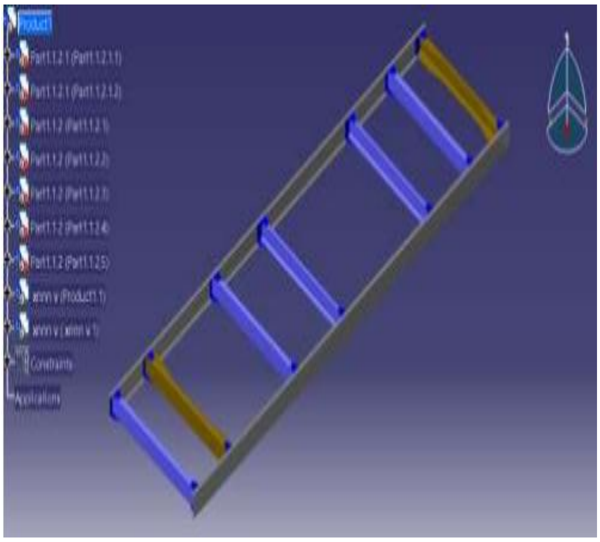


Cross members



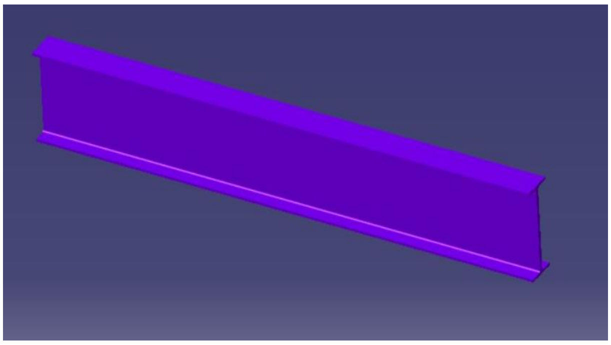
Side members

PART OF “C”SECTION CHASSIS FRAME DESIGN ON CATIA

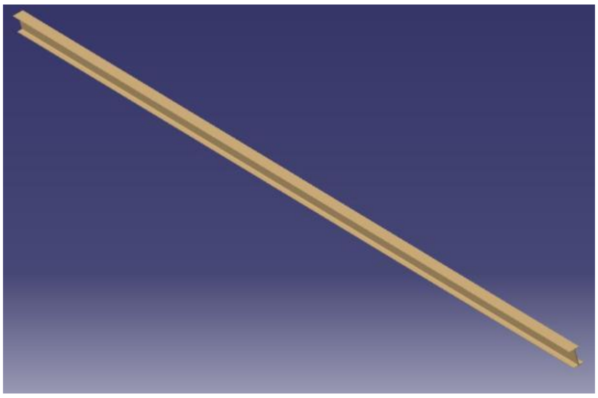


“C”-SECTION CHASSIS FRAME ASSEMBLE ON CATIA

ASSEMBLE IN CATIA OF “I”-SECTION

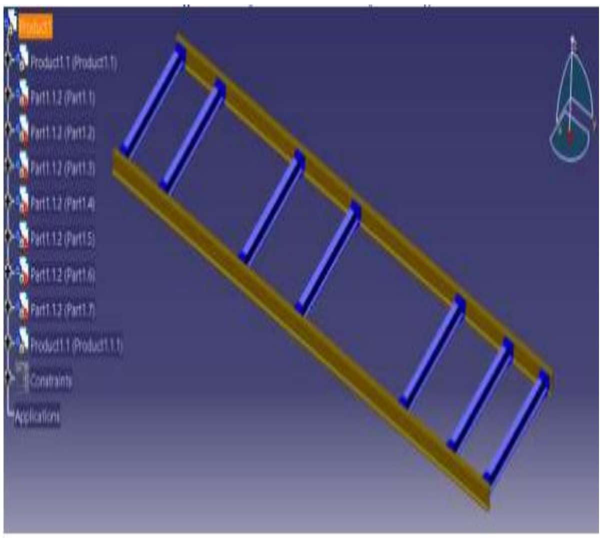


Cross members



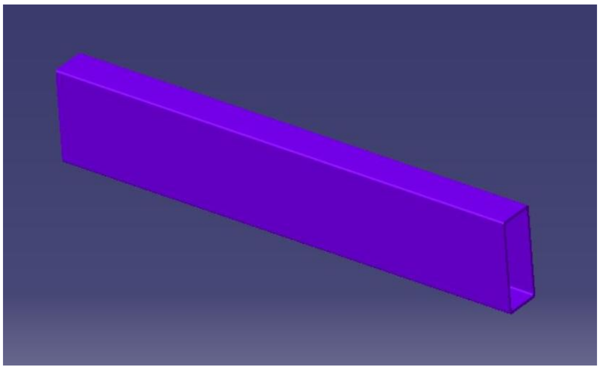
Side members

PART OF “C”SECTION CHASSIS FRAME DESIGN ON CATIA

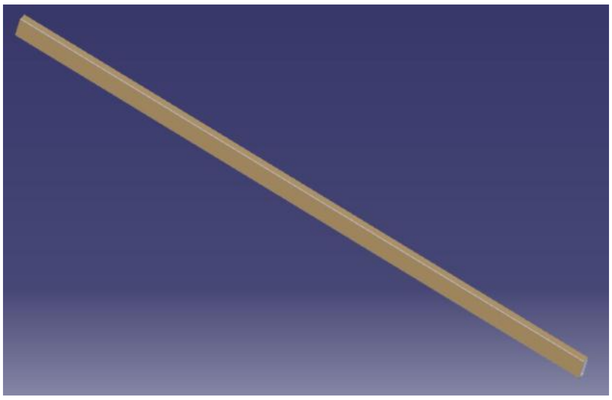


“I”-SECTION CHASSIS FRAME ASSEMBLE ON CATIA

ASSEMBLE IN CATIA OF BOX-SECTION

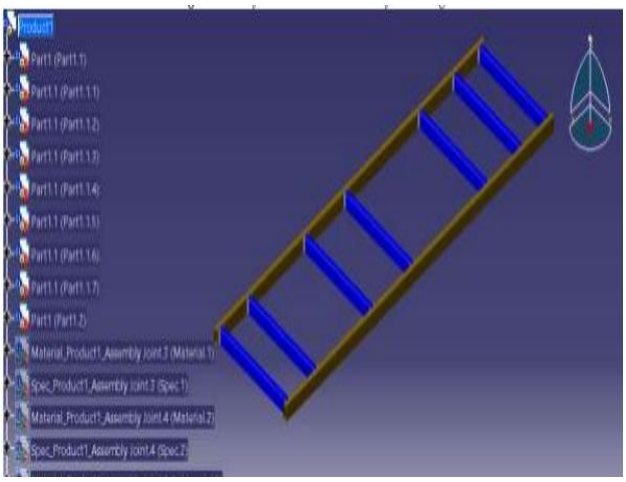


Cross members



Side members

PART OF "BOX"SECTION CHASSIS FRAME DESIGN ON CATIA

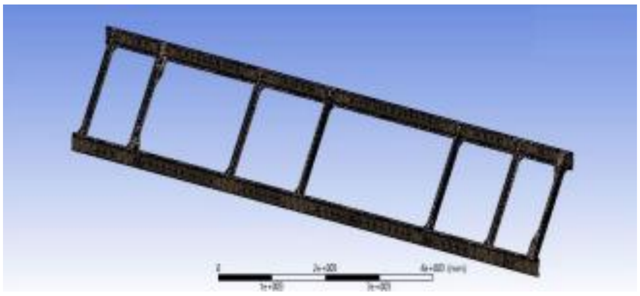


"BOX" SECTION CHASSIS FRAME ASSEMBLE ON CATIA

STATIC ANALYSIS OF CHASSIS IN ANSYS SOFTWARE

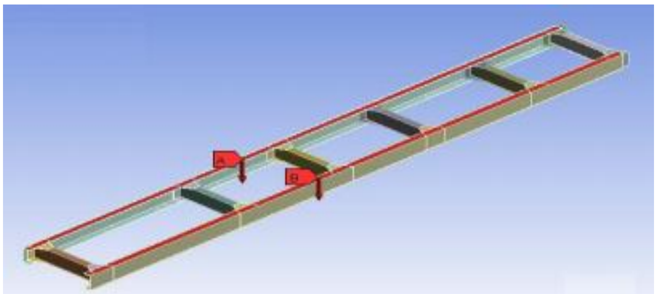
In this work, for doing analysis of our model which were created in CATIA. So, we were using finite element solver (FES) ANSYS. We use ANSYS as general analyzing software (FEA). The geometric model of our chassis is designed in CATIA and analyzed in ANSYS Software. To carry out of the static analysis the part modeling and assembly of chassis is generate in the CATIA after that are import in ANSYS software for static analysis by FEA method of ANSYS Software result plot.

Structural of chassis is Generate Meshing in ANSYS Software



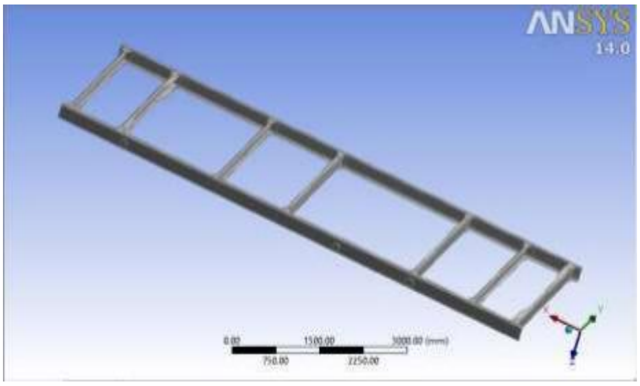
MESH MODEL IN ANSYS SOFTWARE

The truck chassis model is loaded by static forces from the truck body. The magnitude of force on the upper side of chassis is 198652.5 N which is carried by two side bars so load on one side bar is 99326.25 N

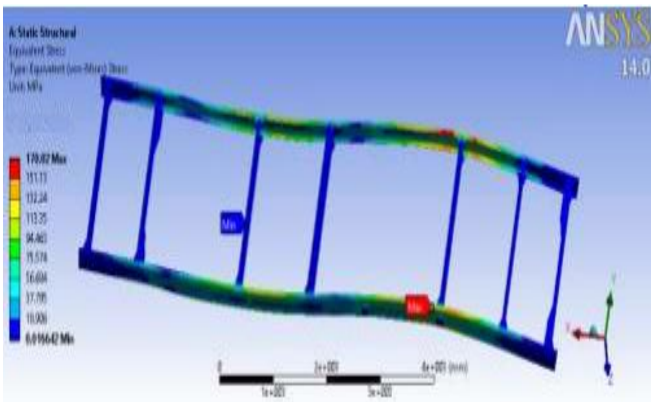


LOAD APPLY CONDITIONS IN ANSYS

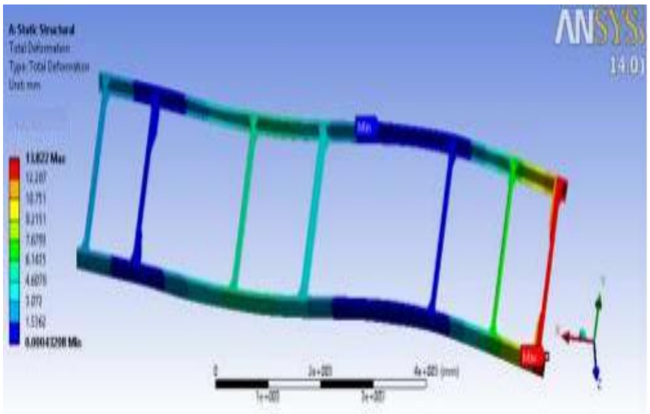
STRUCTURAL ANALYSIS OF "C" SECTION-



GEOMETRIC MODEL OF "C"- SECTION IN ANSYS SOFTWARE

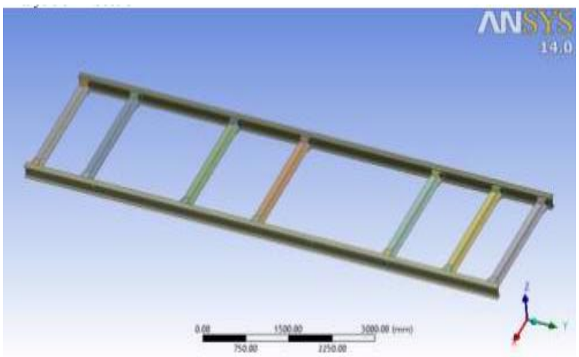


STRESS DISTRIBUTIONS IN "C"- SECTION IN ANSYS SOFTWARE



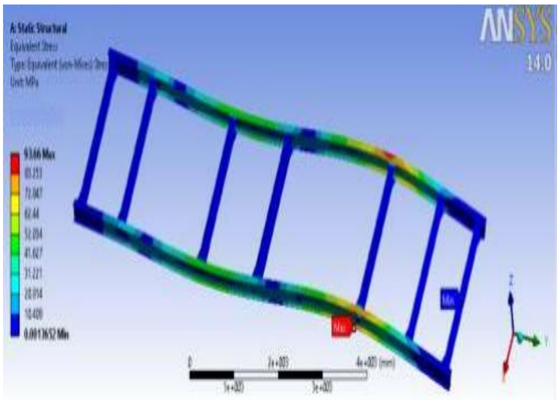
DISPLACEMENT PATTERN IN "C"- SECTION IN ANSYS SOFTWARE

STRUCTURAL ANALYSIS OF I- SECTION

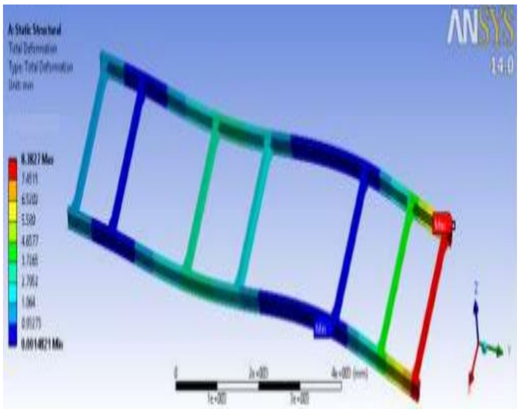


GEOMETRIC MODEL OF I- SECTION IN ANSYS SOFTWARE

International Journal of New Technology and Research (IJNTR)
ISSN: 2454-4116, Volume-6, Issue-11, November 2020 Pages 18-25

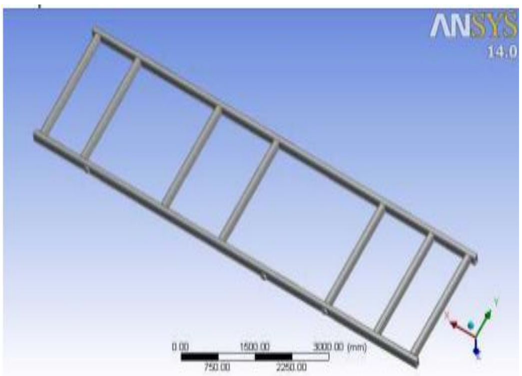


STRESS DISTRIBUTIONS IN I- SECTION IN ANSYS SOFTWARE

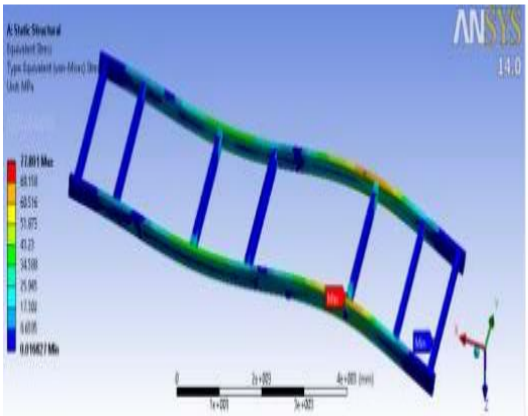


DISPLACEMENT PATTERN IN I- SECTION IN ANSYS SOFTWARE

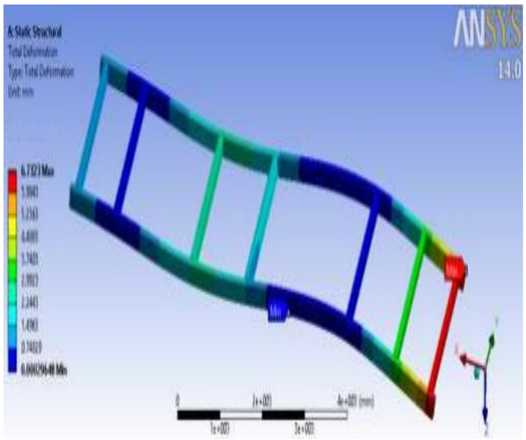
STRUCTURAL ANALYSIS OF BOX- SECTION



GEOMETRIC MODEL OF BOX- SECTION IN ANSYS SOFTWARE



STRESS DISTRIBUTIONS IN BOX- SECTION IN ANSYS SOFTWARE



DISPLACEMENT PATTERN IN BOX- SECTION IN ANSYS SOFTWARE

IV. MATHEMATICAL ANALYSIS

DESIGN AND CALCULATIONS OF EXISTING CHASSIS

Basic Calculation for Chassis:

Model No. = TATA Truck 1618

Specification of chassis as per the IS 9435 for the wheel base 4565mm as mentioned as under

S.NO.	PARAMETERS	VALUES
1.	Total length of the chassis	7720
2.	Width of the chassis	1400
3.	Weight of engine	413 kg.
4.	Front Overhang	1185
5.	Rear Overhang	1809
6.	Capacity(GVW)	16 ton
7.	Kerb Weight	4045 Kg.
8.	Payload	16200 Kg.

Table: 1 Specification of Existing Vehicle TATA 1618 Truck Chassis

Fundamental Calculation for Chassis

Capacity of Truck= 16.20 ton = 16200Kg.= 158922N

Capacity of Truck with 1.25% =158922 x 1.25 N =198652.5 N

Total Load acting on the Chassis = 198652.5 N

Truck chassis has two beam. So load acting on each beam is half of the Total load acting on the chassis. Load acting on the Chassis=Total load acting on the chassis /2

= 198652.50 / 2

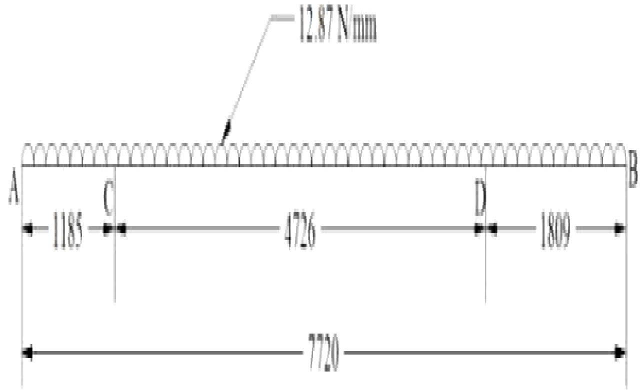
= 99326.25 N / Beam

Calculation for Reaction:-

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 99326.25 N. Length of the Beam is 7720 mm.

Uniformly Distributed Load is 99326.25 / 7720 = 12.87 N

Among these three wheels one wheel / axle are working as a supporting only. Total load reaction generated on the beam is as under:-



For getting the load at reaction C and D, taking the moment about C and we get the reaction load generate at the support D. calculation of the moment are as under. Momentum about C:-

$\frac{12.87 * 1185 x1185}{2}) \quad (\frac{12.87*4726*4726}{2}) - 4726*D+12.87*1809*5630.50$

9036187.88=143726214.10-4726 *D+ 131088343.8

4726D=265778370

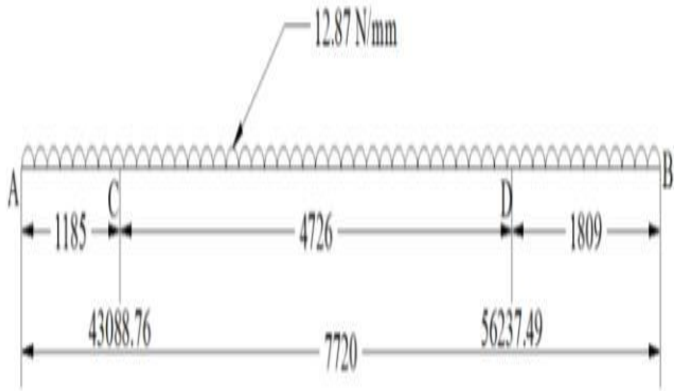
D = 56237.49 N

Total load acting on the beam is 99326.25 N. so load acting on the reaction is as under

C = 99326.25 – D

= 99326.25 - 56237.49

= 43088.76 N



Reaction generated on the beam

C. Calculation for Shear Force Diagram and Bending Moment Diagram:-

Shear Force Diagram:-

FA 0 N

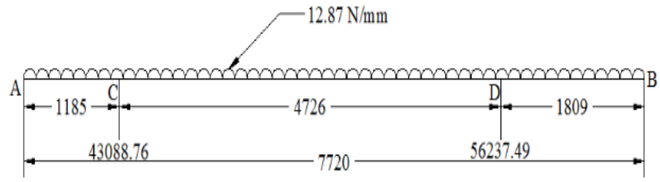
FC = -12.87 x1185+43088

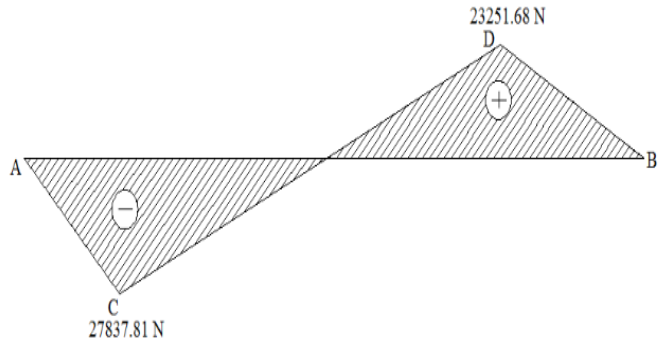
= - 27837.81 N

FD = -12.87 x 5911 + 43088 + 56237

= - 23251.68 N

FB = -12.87x7720 + 43088+ 56237 = 0 N





Shear Force Diagram

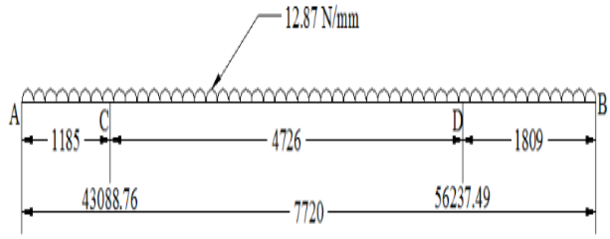
$M_A = 0$

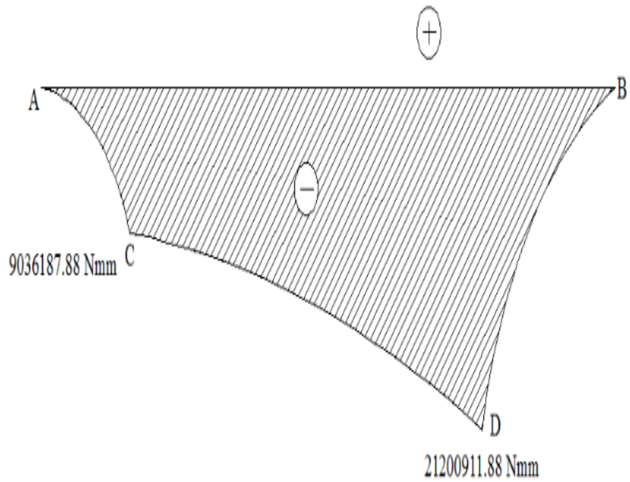
M_c = -(12.87x1185x1185) / 2

= - 9036187.88 Nmm

M_D = -(12.87x5911x5911)/2+43088x4726

M_B =0





Bending Moment Diagram

Calculation for the DEFLECTION:-

M max = **21200911.88 Nmm**

Material of the Chassis is as per IS: - 9345 standard is Structural Steel with St37.

Material Property of the St37:-

Ultimate Tensile Strength= 370 to 490 N/mm2

E= 210000MPa=2.10 x 105 N / mm2

Poisson Ratio = 0.29

Stress and Deflection of Chassis Frame for "C"-Section

b=100, h=225mm, B=90mm, H=205

y=h/2 = 225/2 =112.5mm

Radius of Gyration (R) = 225/2=112.5mm

Moment of Inertia around the X – X axis:-

$$I_{xx} = \frac{(bh^3 - BH^3)}{12} = \frac{(100*225^3 - 100*205^3)}{12}$$

I_{xx} = 23129166.67 mm^3

Section of Modules around the X – X axis:-

$$Zxx = \frac{I_{xx}}{y} = \frac{23129166.66}{112.5}$$

Zxx= =205592.59 mm^3

Maximum Bending Moment acting on the Beam:

Mmax = 21200911.88 Nmm

Zxx = 205592.59mm^3

Stress produced on the Beam is as under:- $\sigma = M / Z$

$$\sigma = \frac{21200911.88}{205592.59}$$

σ = 103.12 N/mm^2

Maximum Deflection produced on the Beam:-

E=210000MPa=2.10x105N/mm^2

I = 23129166.6mm^3

$$y_{max} = \left[\frac{2*w*x_2}{384*E*I}\right] \times \lfloor L^3 - 2LX_1^2 + X_2^3 \rfloor$$

$$y_{max} = \left[\frac{2*12.87*1809}{384*2.10\times10^5*23129166.67}\right] \times \lfloor 7720^3 - 2*7720*1185^2 + 1809^3 \rfloor$$

y_{max} =11.63mm

Stress and Deflection of Chassis Frame for "I"- Section

b=100mm, h=225mm, B= 90mm, H= 205mm,

y=h/2 =225/2 =112.5mm

Radius of Gyration = 225/2 = 112.5 mm

Moment of Inertia around the X–X Axis:

$$I_{xx} = \frac{(bh^3 - BH^3)}{12} = \frac{(100*225^3 - 90*205^3)}{12}$$

Ixx = 30308437.5 mm^3

Section of Modules (around) x–x axis-

$$Z_{xx} = \frac{I_{xx}}{y} = \frac{30308437.5}{112.5}$$

Z_{xx} = 269408.33mm3

Maximum Bending Moment acting on the Beam:

Mmax = 21200911.88 Nmm

Zxx = 269408.33mm3

Stress produced on the Beam is as under:- $\sigma = M / Z$

σ = 21200911.88/ 269408.33mm3

σ = 78.69434431 N/mm^2

Maximum Deflection produced on the Beam:-

E=210000MPa=2.10x105N/mm^2

I = 30308437.5 mm^3

$$y_{max} = \left[\frac{2*w*x_2}{384*E*I}\right] \times \lfloor L^3 - 2LX_1^2 + X_2^3 \rfloor$$

$$y_{max} = \left[\frac{2*12.87*1809}{384*2.10\times10^5*30308437.5}\right] \times \lfloor 7720^3 - 2*7720*1185^2 + 1809^3 \rfloor$$

y_{max} = 8.465387123mm

Stress and Deflection of Chassis Frame for "BOX"-Section

b=100mm, h=225mm, B= 80mm, H= 205mm,

y=h/2 =225/2 =112.5mm

Radius of Gyration = 225/2 = 112.5 mm

Moment of Inertia around the X – X axis:-

$$I_{xx} = \frac{(bh^3 - BH^3)}{12} = \frac{(100*225^3 - 80*205^3)}{12}$$

Ixx = 37487708.33 mm^3

Section of Modules around the X – X axis:-

$$Z_{xx} = \frac{I_{xx}}{y} = \frac{37487708.33}{112.5}$$

Zxx = 333224.0741mm^3

Maximum Bending Moment acting on the Beam:

Mmax = 21200911.88 Nmm

Zxx = 333224.0741mm^3

Stress produced on the Beam is as under:- $\sigma = M / Z$

σ = 21200911.88/ 333224.0741mm^3

σ = 63.62359003 N/mm^2

Maximum Deflection produced on the Beam:-

E=210000MPa=2.10x105N/mm^2

I = 37487708.33 mm^3

$$y_{max} = \left[\frac{2*w*x_2}{384*E*I}\right] \times \lfloor L^3 - 2LX_1^2 + X_2^3 \rfloor$$

$$y_{max} = \left[\frac{2*12.87*1809}{384*2.10\times10^5*37487708.33}\right] \times \lfloor 7720^3 - 2*7720*1185^2 + 1809^3 \rfloor$$

y_{max} =6.844180878 mm

V. RESULTS AND DISCUSSION

The stress distribution parameter and deformation parameter for different cross sections and compared with the analytical values.

Table-5.2 Comparison of Results

Section	Analytical Method		FE Analysis	
	Stress (N/mm^2)	Deformation (mm)	Stress (N/mm^2)	Deformation (mm)
"C"	103.12	11.63	170.02	13.84
"I"	78.69	8.46	93.66	8.38
Box	63.62	6.84	77.80	6.73

It can be inferred from the tabulation that the numerical parameter of the deformation and stress distribution are nearest analytical results; these represents the FE Analysis models results are within permissible limits, so we can say the chassis design are safe

VI. CONCLUSION

The model of the chassis was designed in CATIA and its analyzed is done in ANSYS of different cross sections used in this thesis for similar load conditions. Following analysis a comparison is made between chassis section and other sections in terms of deformation and stresses distribution, to decide on the best one.

The outcome find, 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., 6.84mm and stress is 63.62N/mm2 in various types of chassis of different cross section. So, in different cross sections of the chassis Box-section chassis is suitable for the heavy trucks.

In conclusion the analysis using various 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.

REFERENCES

[1] Monika S.Agrawal "Design and Analysis of Truck Chassis Frame" IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
[2] VinodbabuChintada,ChiranjeevaRao.Seela, VykuntaRaoMatta"Modal and Static Analysis of Automotive Chassis Frame by Using FEA"IEEE Sponsored 2nd International Conference on Innovations in Information,Embedded and Communication systems.
[3] M. Ravi Chandra1, S. Sreenivasulu, Syed AltafHussain, "Modeling and Structural analysis of heavy vehicle chassis made of polymeric composite material by three different cross sections" International Journal of Modern Engineering Research (IJMER)
[4] Ramesh kumar. S, Dhandapani. N. V, Parthiban.S, Kamalraj.D, Meganathan.S, Muthuraja.S"Design And Analysis Of Automotive Chassis Frame Using Finite Element Method" International Journal of Pure and Applied Mathematics
[5] Vijayan, S. N.,Sendhikumar S., Kiranbabu K.M. "analysis of pedal operated wheel brake system of a helicopter using adams software" International Journal of Current Research