Analysis and Optimization of Heavy Truck Chassis

[1]Mr. Nitin Awate S., [2]Dr. Kiran More

[1,2]Department of Mechanical Engineering, D.Y.Patil Institute of Engineering and Technology Ambi, Talegaon, Savitribai Phule Pune University, Maharashtra, India.

[1]awatenitin01@gmail.com, [2]kiran.more@dyptc.edu.in

Abstract – Chassis is one of the most important parts used in automobile industry. It is a rigid structure that forms a skeleton to hold all the major parts together. Chassis frames are made of "steel section" so that they are strong enough to withstand the load and shock. Chassis must be light in weight to reduce dead weight on the vehicles. Major challenge in today's automobile vehicle industry is to overcome the increasing demands for higher performance, lower weight in order to satisfy fuel economy requirements, and longer life of components, all this at a reasonable cost and in a short period of time. The study is to produce results to rectify problems associated with structures of a commercial vehicle such as strength, stiffness properties along with stress, bending moment and vibrations while optimizing the weight. The FEA and optimization of the chassis is performed in ANSYS. The results obtained from the FEA will be used to manufacture the optimized truck chassis.

***Key Words*-Truck Chassis, FEA, Optimization.**

I. INTRODUCTION

"The truck industry has experienced a high demand in market especially in India whereby the economic growths are very significantly changed from time to time. There are many industrial sectors using this truck for their transportations such as the logistics, agricultures, factories and other industries. However, the development and production of truck industries in India are currently much relying on foreign technology and sometime not fulfill the market demand in term of costs, driving performances and transportations efficiency.

Ladder frame : The history of the ladder frame chassis dates back to the times of the horse drawn carriage. It was used for the construction of 'body on chassis' vehicles, which meant a separately constructed body was mounted on a rolling chassis. The chassis consisted of two parallel beams mounted down each side of the car where the front and rear axles were leaf sprung beam axles. The beams were mainly channeled sections with lateral cross members.

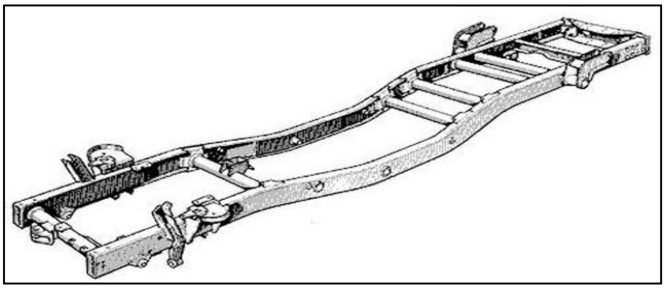

Fig.1. Ladder frame chassis

PROBLEM STATEMENT AND OBJECTIVE

Chassis frames are made of "steel section" so that they are strong enough to withstand the load and shock. Chassis must be light in weight to reduce dead weight on the vehicles. Major challenge in today's automobile vehicle industry is to overcome the increasing demands for higher performance, lower weight in order to satisfy fuel economy requirements, and longer life of components, all this at a reasonable cost and in a short period of time. The study is to produce results to rectify problems associated with structures of a commercial vehicle such as strength, stiffness properties along with stress, bending moment and vibrations while optimizing the weight.

II. LITERATURE REVIEW

Mohd Azizi Muhammad Nor, presented paper on "Stress of a Low Loader Chassis". This paper aims to model, simulate and perform the stress analysis of an actual low loader structure consisting of I-beams design application of 35 tonne trailer designed in-house by Sumai Engineering Sdn. Bhd, (SESB). The material of structure is Low Alloy Steel A710 C (Class 3) with 552 MPa of yield strength and 620 MPa of tensile strength. The scope of this study concerns on structural design of the I-beams for info and data gathering, which will be used for further design improvement. [1]

Chen Yanhong, et.al., present paper on, "The Finite Element Analysis and The Optimization Design of The Yj3128-type Dump Truck's Sub-Frames Based on

DOI : 10.35291/2454-9150.2021.0143

ISSN : 2454-9150 Vol-07, Issue-01, APR 2021

ANSYS". The article mainly studies the YJ3128-type dump trucks sub-frames, for the fatigue crack occurred in the Sub frame witch has worked in bad condition for 3 to 5 months, the trucks working conditions and the load features are researched, and ANSYS is used to analyze the stress of the sub-frame. According the deferent stress, the reason of the fatigue cracks occurring is researched too.[2]

Akash Singh Patel, presented paper on "Modeling, Analysis & Optimization of TATA 2518 TC Truck Chassis Frame using CAE Tools". 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.. [3]

Dharmendrasinh Parmar, et.al, presented paper on, "Design and Weight Optimization of YJ3128 Type Dump Truck's Frame". The thesis consist of an introduction, design of chassis, model, analysis, specific characteristics, a comparison between the results with in obtain by optimization. The mainly studies the YJ3128-type dump truck's frames, for the fatigue crack occurred on frame in worked in bad condition for 3 to 5 months, the truck's working conditions and the load of 16tonne features are researched, and ANSYS is used to analyses the stress of the frame. The weight reduction is achieved by changing section of the side bar and changing materials. Then FEA is performed on that model. After complete FEA, comparing them and get the best solution. [4]

Nouby M. Ghazaly; presented paper on "Applications of Finite Element Stress Analysis of Heavy Truck Chassis: Survey and Recent Development". Nowadays, transportation industry plays a major role in the economy of modern industrialized and developing countries. The goods and materials carried through heavy trucks are dramatically increasing. There are many aspects to consider when designing a heavy trucks chassis, including component packaging, material selection, strength, stiffness and weight. This paper reviews the most important research works, technical journal and conferences papers that have been published in the last thirteen year period (2002-2014). The paper focused on stress analysis of the heavy truck chassis using four finite element packages namely; ABAQUS, ANSYS, NASTRAN and HYPERVIEW.[5]

Mr. Rahul L. Patel, et.al, presented paper on "Weight optimization of chassis frame using Pro-Mechanica". Automotive chassis can be considered as the backbone of any vehicle. Chassis is tasked at holding all the essential components of the vehicle like engine, suspension, gearbox, braking system, propeller shaft, differential etc. To sustain various loads under different working conditions it should be robust in design. Moreover chassis should be stiff and strong enough to resist severe twisting and bending moments to which it is subjected to. The objective is to do weight optimization of Chassis of hydraulic truck (TATA 2516TC). The design is implemented with size optimization using Pro Mechanica software and the studied chassis with capacity 25 tonne is for carrying the load of truck. The basic model will be a good starting point for further studies and developments of final models.[6.]

III. THEORETICAL ANALYSIS

Currently the material used for the chassis (TATA 2518TC) is per IS: - 9345 standard is structural steel with St 37. Structural steel in simple words with the varying chemical composition leads 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% SProperty of the ST37:-

Modulus of Elasticity = 210 GPa = 2.1×10^5 N / mm^2

= 7850 kg/m^3

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

Strength = 260 MPa = 260 N / mm^2

Poisson Ratio = 0.29

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

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 × 65mm × 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:-

 DOI : 10.35291/2454-9150.2021.0143

ISSN : 2454-9150 Vol-07, Issue-01, APR 2021

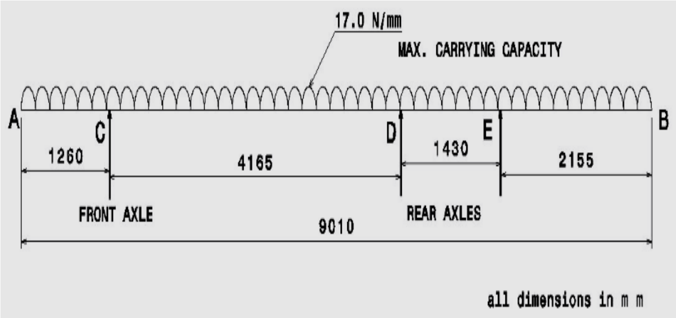

Fig.2 Total load generated on the beam

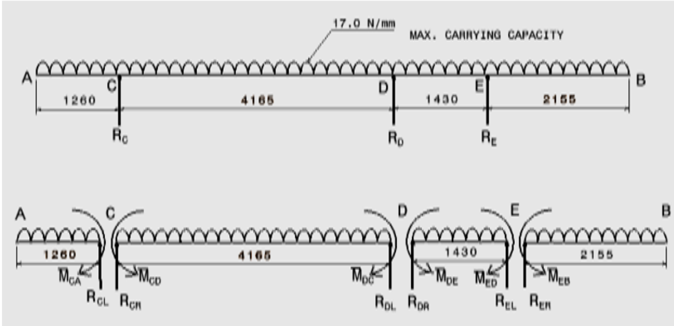

Fig.3 Consideration of fixed end moments

Fixed End Moment

This is the indeterminate structure of beam.

$$\bar{M}_{CA} = \frac{17 \times 1260 \times 1260}{2} = 13494600\ Nmm$$

$$\bar{M}_{CD} = \frac{-17 \times 4165 \times 4165}{12} = -24575235.42\ Nmm$$

$$\bar{M}_{DC} = 24575235.42\ Nmm$$

$$\bar{M}_{DE} = \frac{-17 \times 1430 \times 1430}{12} = -2896941.667\ Nmm$$

$$\bar{M}_{ED} = +2896941.667\ Nmm$$

$$\bar{M}_{EB} = \frac{-17 \times 2155 \times 2155}{2} = -39474212.5\ Nmm$$

Total restraint moment at "C"

$$\overline{M_C} = \overline{M_{CA}} + \overline{M_{CD}} = 13494600 - 24575235.42 = -11080635.42 Nmm$$

Total restraint moment at "D"

$$\overline{M_D} = \overline{M_{DC}} + \overline{M_{DE}} = 24575235.42 - 2896941.667 = 21678293.75 Nmm$$

Total restraint moment at "E"

$$\overline{M_E} = \overline{M_{ED}} + \overline{M_{EB}} = 2896941.667 - 39474212.5 = -36577270.83\ Nmm$$

For the span "CD"

$$M_{CD} = \overline{M_{CA}} = -13494600\ Nmm$$

$$M_{DC} = \overline{M_{DC}} - \frac{\overline{M_C}}{2} + \frac{3EI}{l} i_b$$

$$= 24575235.42 + \frac{11080635.42}{2} + \frac{3EI}{4165} i_b Nmm$$

$$= 30115553.13 + \frac{3EI}{4165} i_b Nmm$$

For the span "DE"

$$M_{DE} = -\bar{M}_{EB} = 39474212.5\ Nmm$$

$$M_{DE} = \bar{M}_{DE} - \frac{\bar{M}_E}{2} + \frac{3Ei}{l} i_b = -2896941.667 + \frac{36577270.83}{2} + \frac{3EI}{1430} i_b = 15391693.75 + \frac{3EI}{1430} i_b$$

Equilibrium condition at "D"

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

$$30115553.13 + \frac{3EI}{4165} i_b + 15391693.75 + \frac{3EI}{1430} i_b = 0$$

$$EIi_b = -1.61476 * 10^{10}$$

Substituting the value of EI i_b

$$M_{DC} = 18484567.78\ Nmm$$

$$M_{DE} = -18484567.77 Nmm$$

Calculation for Reaction

$$R_{CL} = 17 \times 1260 = 21420 (\uparrow)$$

$$R_{CR} = \frac{17 \times 4165}{2} + \frac{13494600 - 18484567.78}{4165} = 34204.4285\ N(\uparrow)$$

$$R_{DL} = 17 \times 4165 - 34204.4285 = 36600.5715\ N(\uparrow)$$

 DOI : 10.35291/2454-9150.2021.0143

$$R_{DR} = \frac{17 \times 4165}{2} + \frac{18484567.77 - 39474212.5}{1430} = 2523.07324N(\downarrow)$$

$$R_{EL} = 17 \times 1430 + 2523.07324 = 26833.07324(\uparrow)$$
$$R_{ER} = 17 \times 2155 = 36635N(\uparrow)$$
$$\therefore R_C = R_{CL} + R_{CR} = 55624.4285\ N(\uparrow)$$
$$R_D = R_{DL} + R_{DR} = 34077.4982\ N(\uparrow)$$
$$R_E = R_{EL} + R_{ER} = 63468.0732N(\uparrow)$$

Calculations for Bending Moment Diagram:-

$$M_A = 0Nmm$$
$$M_C = -\bar{M}_{CA} = -13494600\ Nmm$$
$$M_P = \frac{17 \times 4165 \times 4165}{8} = 36862853.13Nmm$$
$$M_D = M_{DE} = -18484567.77Nmm$$
$$M_Q = \frac{17 \times 1430 \times 1430}{8} = 4345412.5Nmm$$
$$M_E = M_{EB} = -3947412.5Nmm$$
$$M_B = 0N$$

So the maximum bending moment occurs at "E"

$$M_{max} = M_E = -39474212.5Nmm$$

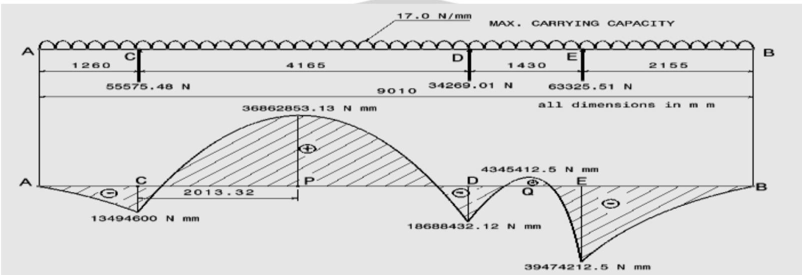

Fig. 4 Bending Moment Diagram

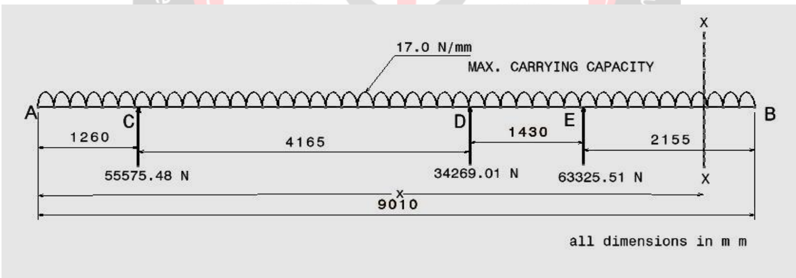

Fig: 5 Reaction generated on the beam

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

Taking moment of all forces about x-x section

Mxx = $-8.5x^2$ + RC(x-1260) + RD(x-5425) + RE (x-6855)

According to Macaulay's theorem

$M_{XX} = EI\frac{d^2y}{dx^2}$ =$-8.5x^2$ + RC(x-1260) + RD(x-5425) + RE (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

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

Applying the boundary conditions

At X=1260 mm , y = 0

$$0 = \frac{-17X^4}{24} + 1260 \times C_1 + C_2 \ldots\ldots\ldots\ldots\ldots 2$$

 DOI : 10.35291/2454-9150.2021.0143

At x = 6855mm, y = 0

$$0 = \frac{-17 \times 6855^4}{24} + C_1 \times 6855 + C_2 + 55624.4285\frac{5595^2}{6} + 34077.49826\frac{1430^2}{6} \dots\dots\dots\dots\dots 3$$

Solving equation 2 and 3 we get

$C_1 = -1.394 \times 10^{10}$

And $C_2 = 1.935 \times 10^{13}$

Putting these values in equation 1 we get

$$y = \frac{1}{EI}(\frac{-17x^4}{24} - 1.394 \times 10^{10}X + 1.935 \times 10^{13} + R_C\frac{(X-1260)^2}{6} + R_D\frac{(X-5425)^2}{6} + R_E\frac{(X-6855)^2}{6} \dots\dots\dots\dots 4$$

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)

$$y_A = \frac{1.935 \times 10^{13}}{EI}$$

Deflection at “B” (i.e. x = 9010mm)

$$y_B = \frac{-9.1408 \times 10^{13}}{EI}$$

So the maximum deflection occurs at “B”

$$y_{max} = y_B = \frac{-9.1408 \times 10^{13}}{EI} \dots\dots\dots\dots\dots 5$$

For C- Section :-

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

y = h /2 = 285 /2 = 142.5 mm

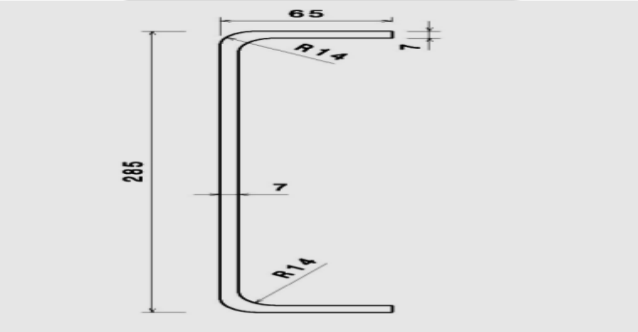

Fig. 6 Main “C-Channel” section

Moment of Inertia around the X – X axis:-

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

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

= 29195623.92 mm4

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:-

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

Maximum Bending Moment acting on the Beam

M_{max} = -39474212.5 N mm

Z = 204881.572 mm3

Stress produced on the Beam

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

from equation 5 Maximum Deflection produced on the Beam

E = 210000 MPa = 2.10 x 105 N / mm2

I =29195623.92 mm4

DOI : 10.35291/2454-9150.2021.0143

ISSN : 2454-9150 Vol-07, Issue-01, APR 2021

$$y_{max} = \frac{-9.1408 \times 10^{13}}{EI} = \frac{-9.1408 \times 10^{13}}{210000 \times 29195623.92} = 14.90\ mm$$

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 14.90 mm is safe

IV. FINITE ELEMENT ANALYSIS

Finite element analysis is performed to find out the optimum shape for the truck chassis of 2518 Tata truck in static loading conditions. For which first baseline model is prepared referring different drawing views and papers published on the same. Please find the 3 D model created using CATIA software in the figure below.

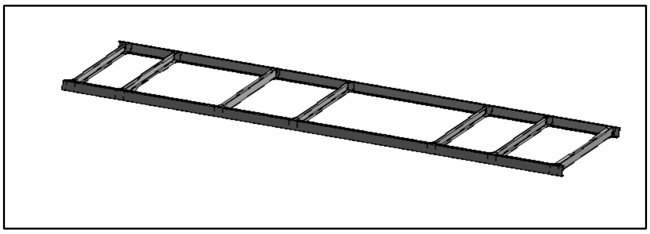

Fig.7: CAD geometry model for Truck chassis of TATA 2518

Geometry is imported in the ANSYS workbench module and static structural analysis is performed on the model with structural load of complete sprung mass on the module. Mass of the truck is considered uniformly distributed on the chassis. Total weight of the truck chassis as specified by the truck manual which is 25 Ton thus we have applied load of 245250N on the chassis, area of the chassis which comes in to the contact with loading is measured and load is divided by the area to calculate pressure applied by the loading on the area of the chassis. By applying pressure boundary condition we ensure that loading is uniformly distributed on the chassis. Pressure Calculations

Table 2: Pressure Calculations

Loading Area	1692900	mm^2
Force applied	245250	N
Pressure	0.14487	Mpa

Then meshing is performed on the module with mesh size of 15 mm with 800600 nodes and 400200 elements. Below are detailed images of the meshing performed on the model. Total weight of the baseline chassis design is 799.27 kg.

Baseline	Total Number of Nodes	Total Number of Elements
	800600	400200

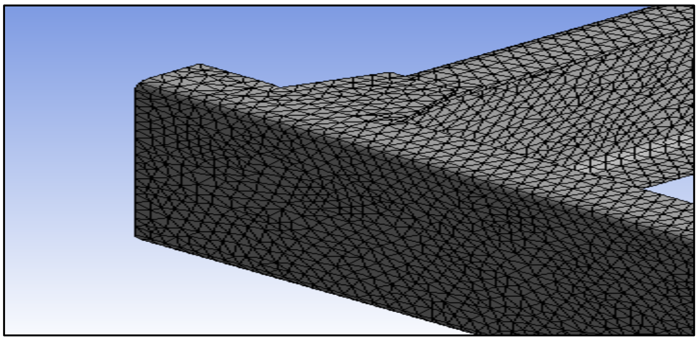

Fig. 8: Details of Meshing

Below is the image showing boundary conditions applied to the chassis to simulate the loading on the chassis and fixing of the chassis on during the static analysis.

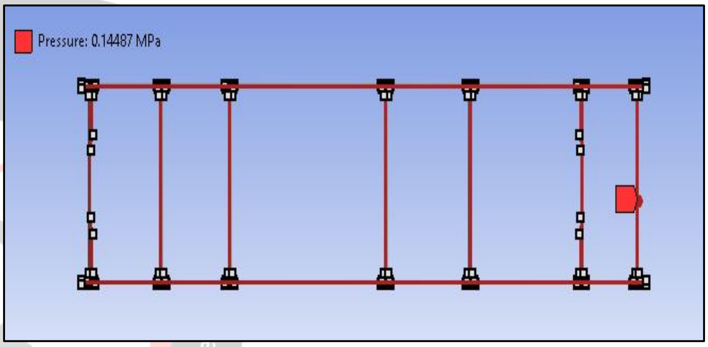

Fig. 9: Pressure loading applied on the top surface of Chassis

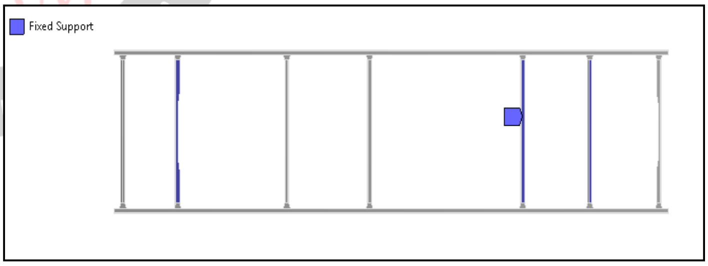

Fig. 10: Fixed boundary condition applied on the three surfaces below

As shown in the figure boundary conditions are applied to the chassis that is to simulate the uniformly loaded chassis we have applied pressure force on the chassis top surface area. From bottom side as shown in the figure three highlighted cross members which are rested on the leaf spring and static analysis run is performed. Stresses and deformation for the run are shown in the images below.

 DOI : 10.35291/2454-9150.2021.0143

ISSN : 2454-9150 Vol-07, Issue-01, APR 2021

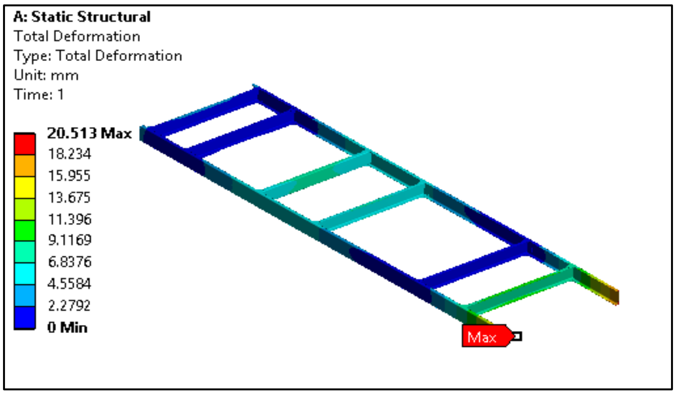

Fig. 11: - Deformation Plot for chassis analysis

Maximum deformation of 20.51 mm is observed at the back end overhang part of the chassis. These deformation values are assuming there is hard mounting of the chassis. In actual case there will be leaf spring below chassis so we can ignore high deformations due to hard mounting.

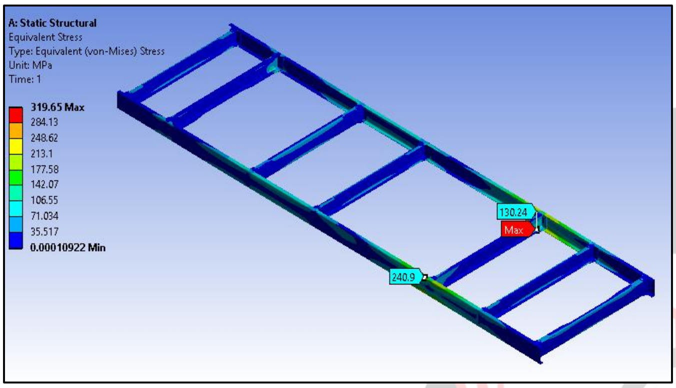

Fig. 12:- von mises stress plot for chassis baseline

Stress concentration shows the maximum stress near 319.65 MPa but just two nodes away stress goes down to 130 MPa, maximum von mises stress observed in the plot is 240.9 MPa on the fillet of side bar.

Iterative study for the shape optimization is to be performed on the chassis design to reduce the weight. We will use simple method of reducing the material with low von mises stress. We have maximum stress observed as 240 MPa. We will make an assumption that removing material which shows 5 and 10 % or less stress than the maximum stress observed in the analysis. Using this method we ensure that we do not chop off the material from the body which is contributing towards the stiffness of the geometry. Wewill plotvon-mises stress plot with capping all the material which has stress less than 12 MPa value. So we can perform iterative study by removing some suitable geometric shape from the capped area of the stress plot which will allow us to perform our next iteration of the design.

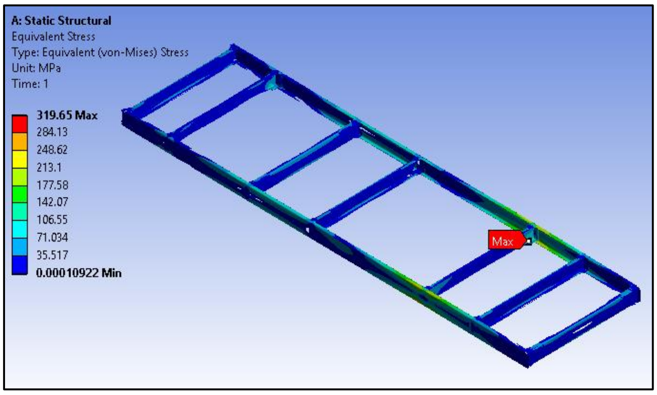

Fig. 13: - Stress plot capping all the area having stress below 12 Mpa

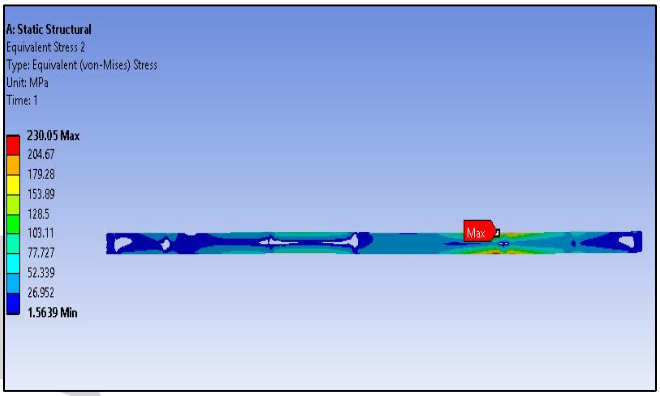

Fig. 14:- 12 MPa capped FEA side rail view

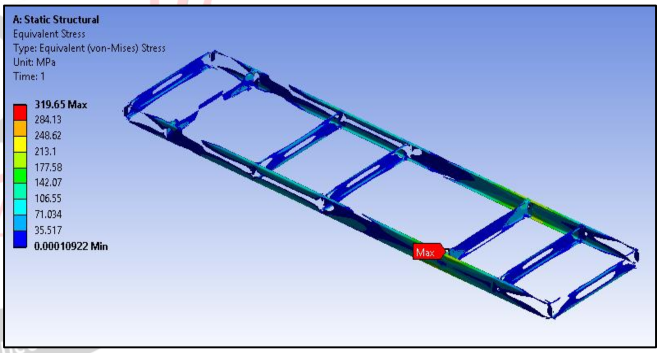

Fig. 15:- 24 MPa Capped Stress plot for Baseline

Iteration 1	Total Number of nodes	Total Number of elements
	770300	386000

From 24 MPa capped view we get clear idea about the shape that can be removed from the material of the frame without losing area of the frame which takes most of the loads. Accordingly changes are made in the design to formulate the iteration 1 design.

Iteration 1

 DOI : 10.35291/2454-9150.2021.0143

International Journal for Research in Engineering Application & Management (IJREAM)
ISSN : 2454-9150 Vol-07, Issue-01, APR 2021

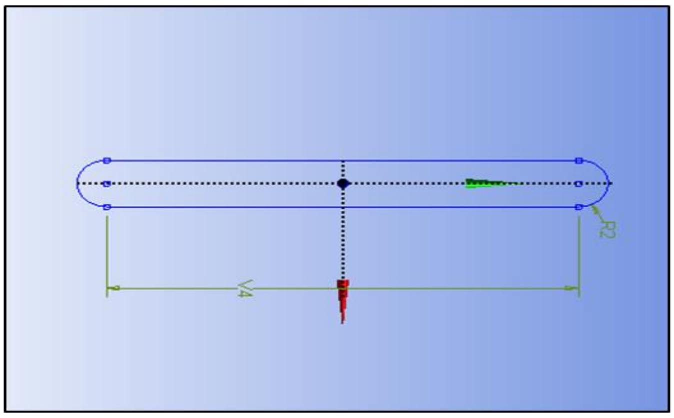

Fig.16:- Sketch for the Iteration 1 material removal

This is the material removal sketch on the some of the cross bars of chassis to reduce the weight of the chassis. Where V4 means width of the slot is 800 mm and R2 means end radius is 50 mm.

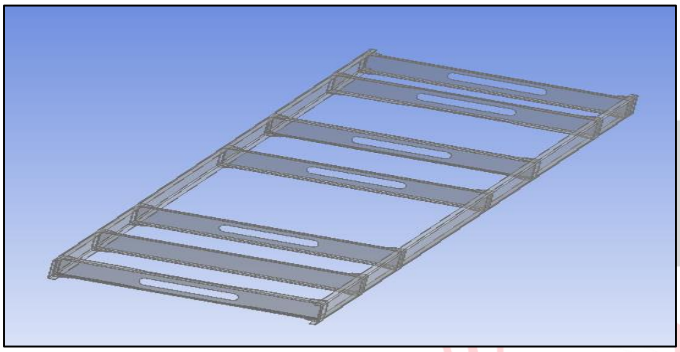
Fig. 17:- Iteration 1 model

6 of the 7 cross members are providing with cut at the center as shown by the sketch. Second cross member in the image above is not cut because of its contribution in the 24 MPa capped stress plot. Similarly as baseline, meshing, constrains and loading is applied on the iteration 1 geometry and results for the iteration 1 design are shown in the images below.

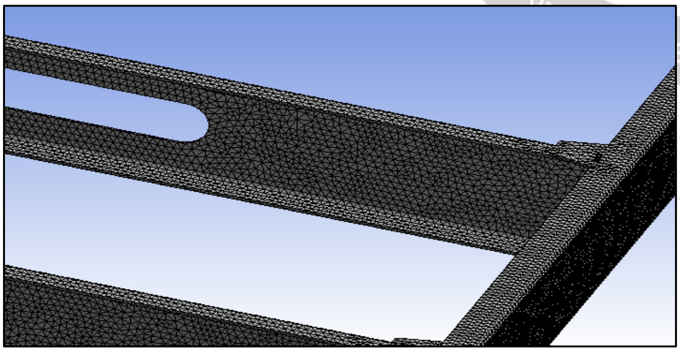
Fig. 18:- Details of Meshing Iteration 1

Figure above shows meshing of iteration 1 model design, total 770300 nodes and 386000elements are used to mesh the body. Total mass of the new design of chassis is observed to be 770.3 kg. Boundary conditions are same as baseline.

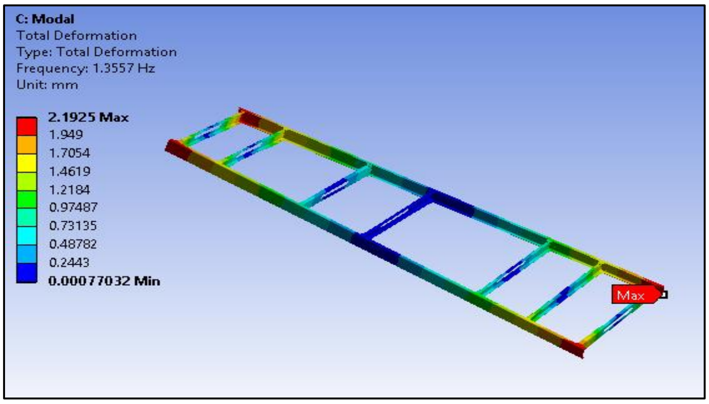

Fig. 19:- Iteration 1 Mode shape plot 1.36 Hz

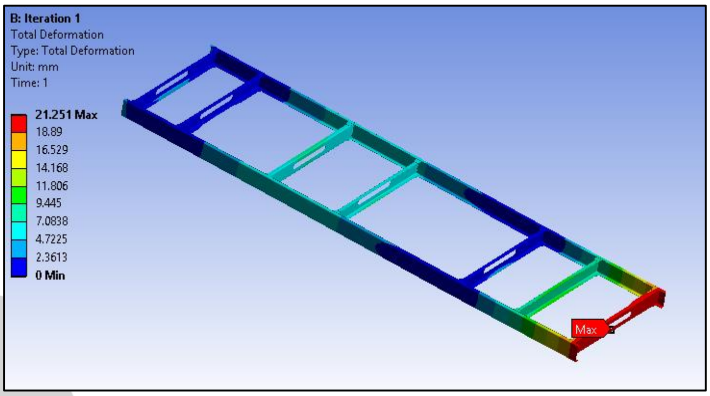

Fig. 20:- Deformation plot Iteration 1 static analysis

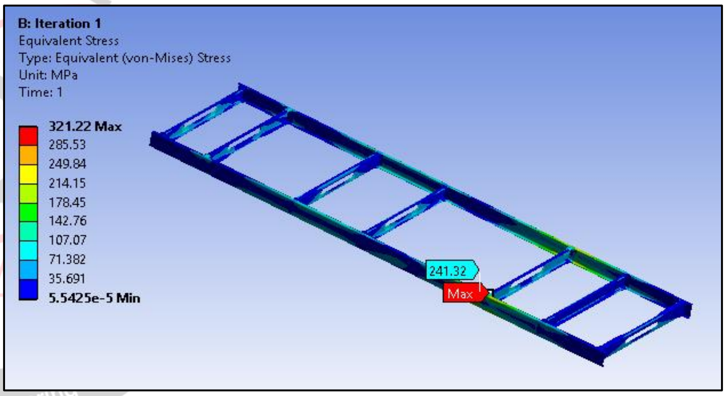

Fig. 21: - Iteration 1 von Mises stress plot static analysis

Again to find out next step for iteration 2 design we will use capped stress plot for iteration 1 results of von mises stress plot. We will cap the results of iteration1 such a way that all the material having stress below 24 MPa should be capped or hidden from the plot. Below is the result for the same.

Iteration 2	Total Number of nodes	Total Number of elements
	769000	382000

 DOI : 10.35291/2454-9150.2021.0143

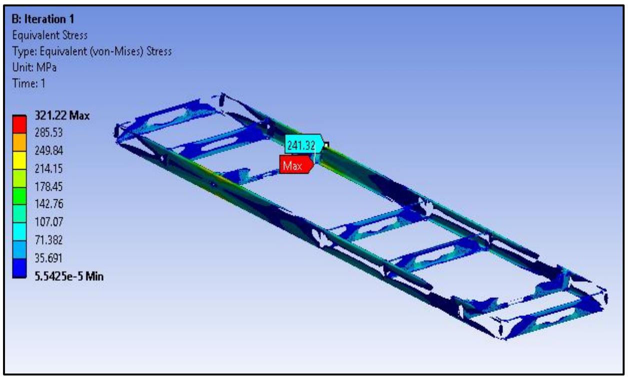

Fig. 22: -24 MPa Capped Stress plot for Iteration 1

Based on the capping of the iteration 1 results iteration 2 sketch for the material removal is created. It can be seen clearly that there is material removal scope in the side members between third and fourth cross member in the image above. Considering that we will remove the material there.

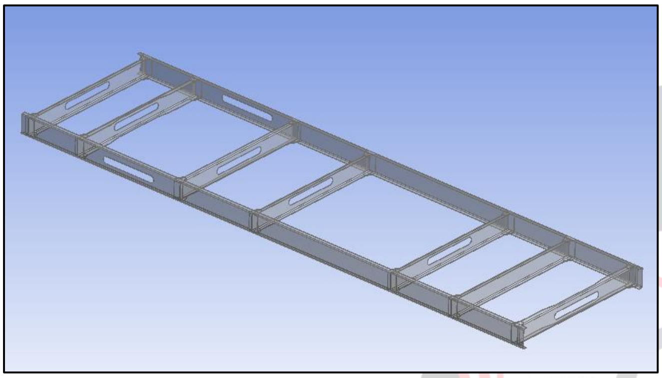

Fig. 23:- Iteration 2 model with side rail material removed

Same sketch used in iteration 1 is used to cut the material on the side rail. Modal and static analysis with the boundary conditions same as Baseline are ran on this model. Results for the same are shown below.

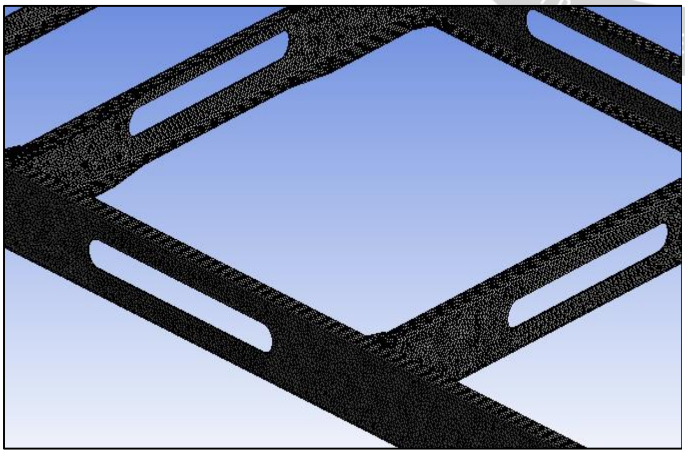

Fig. 24: - Meshing at Modified Area

Meshing is performed same as other two iterations with 15 mm average element size. 769000 nodes and 382000 elements are used to mesh the module of iteration 2. Weight of the module is 760.65 kg.

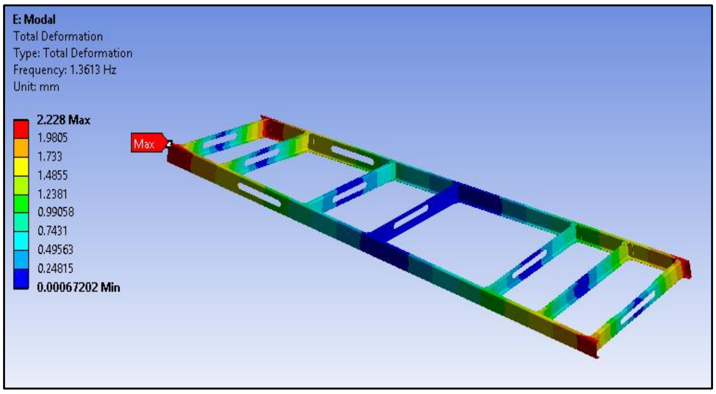

Fig. 25: - Iteration 2 Mode shape plot 1.36 Hz

Iteration 2 mode shape plot shows the twisting mode with same frequency as iteration1.

Static Analysis results are shown in the images below, it can be summarized that iteration 2 maximum deformation observed in the static analysis is 19.93 mm and highest stress away from the stress concentration area is observed as 242 MPa.

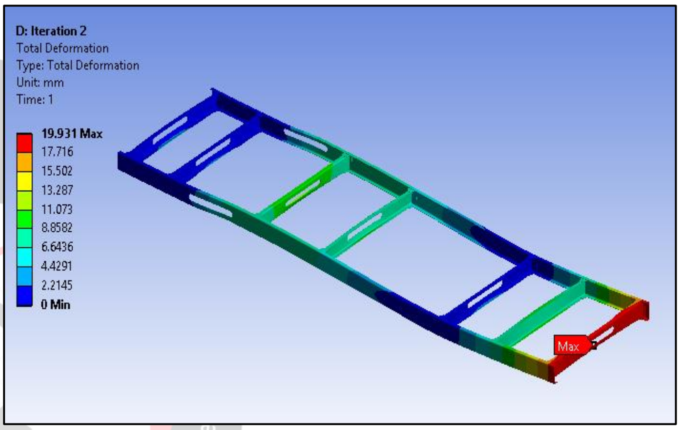

Fig. 26: -Total Deformation Plot Iteration 2

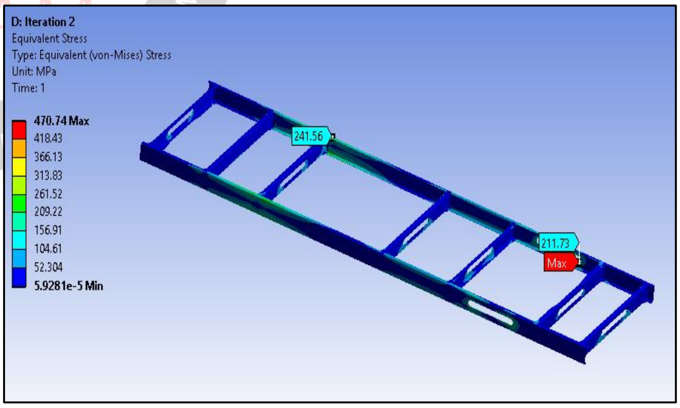

Fig. 27: - von Mises stress plot Iteration 2

New stress concentration zone is created at the newly added feature in iteration 2 design on the side rail which shows stress as high as 471 MPa but if we move just away from the stress concentration area the stress observed is 212 near that region which is within the acceptance limit.

Iteration 3	Total Number of nodes	Total Number of elements
	762000	378000

We will check whether further reduction in the material is possible by capping the iteration 2 design with 24 MPa

 DOI : 10.35291/2454-9150.2021.0143

ISSN : 2454-9150 Vol-07, Issue-01, APR 2021

stress limit, capping iteration 2 stress plot.

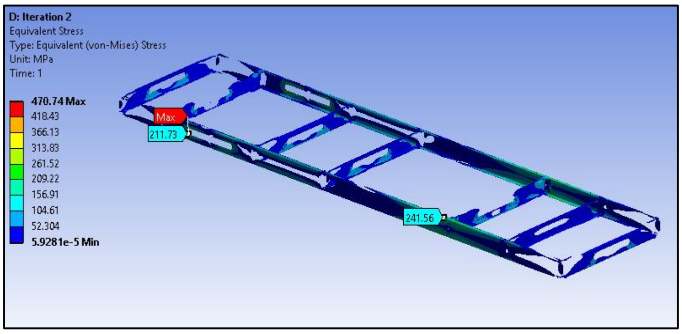

Fig. 28-24 MPa capped von Mises stress plot Iteration 2

This capped stress plot of iteration 2 shows us that we can further try to remove material from the side rail as shown in the image.

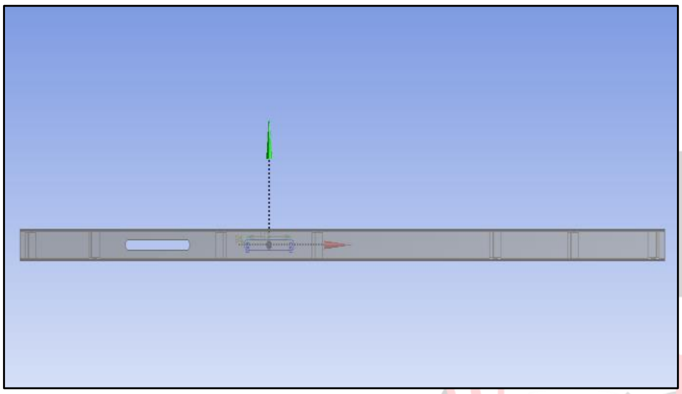

Fig. 29:- Material removal sketch for the Iteration 3 model

Length of the slot is 600 mm and radius is same as previous slots 50 mm. Slot is made on the side rails.

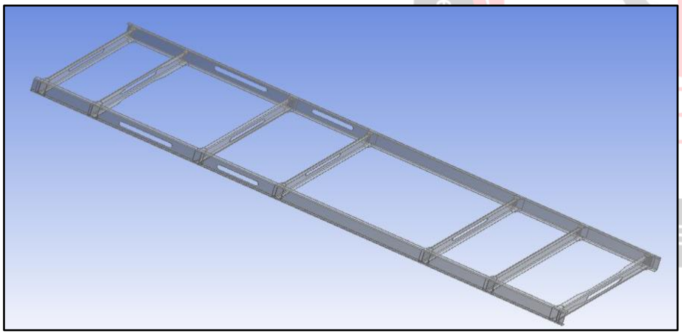

Fig. 30: - Iteration 3 design model

Same procedure is followed for iteration 3 as performed on the other 3 designs previously and modal analysis results and static analysis results are obtained.

Meshing is performed same as other two iterations with 15 mm average element size. 762000 nodes and 378000 elements are used to mesh the module of iteration 3. Weight of the module is 753.19 kg. Modal Analysis Iteration 3 results are shown in the figure below.

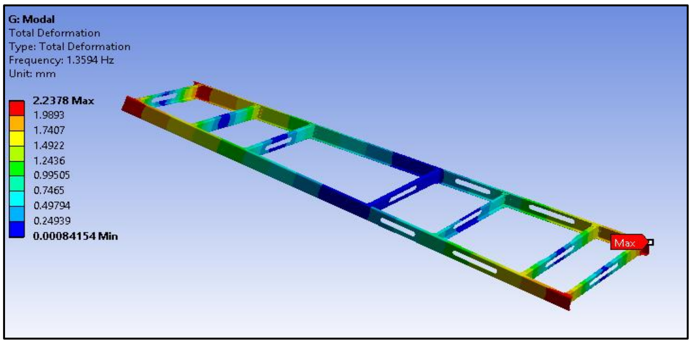

Fig. 31: - Iteration 3 Mode shape plot 1.36 Hz

Iteration 2 mode shape plot shows the twisting mode with same frequency as iteration1.

Static Analysis results are shown in the images below, it can be summarized that iteration 3 maximum deformation observed in the static analysis is 20 mm and highest stress away from the stress concentration area is observed as 240 MPa.

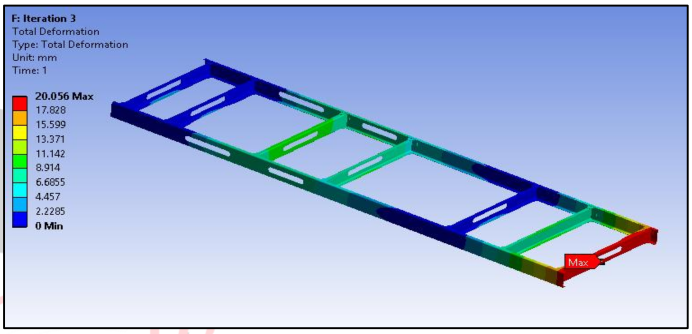

Fig. 32: -Total Deformation Plot Iteration 3

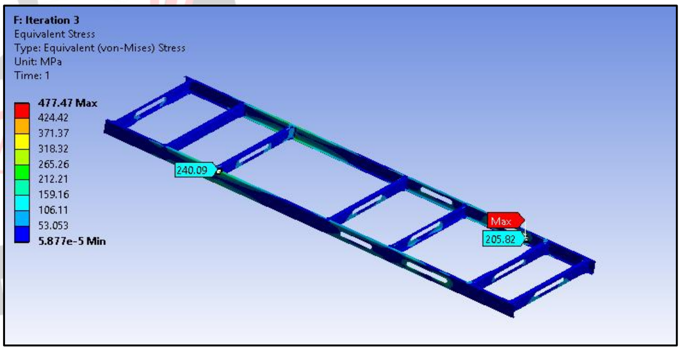

Fig. 33- von Mises stress plot Iteration 3

New stress concentration zone is created at the newly added feature in iteration 3 design on the side rail which shows stress as high as 477 MPa but if we move just away from the stress concentration area the stress observed is 206 near that region which is within the acceptance limit

V. EXPERIMENTAL TESTING

To verify the strength of optimized Truck Chassis experimentally we tested the Chassis on universal testing machine (UTM) in Om Metalabs, Kondhwa, Pune.

In UTM testing the model was mounted on the fixture manufactured especially for testing the Chassis and the fixture was placed between moving jaw and fixed jaw of the UTM. Then slowly moving jaw is moved hydraulically with the displacement vertically downward to load component with compressive loading. The loading is

 DOI : 10.35291/2454-9150.2021.0143

ISSN : 2454-9150 Vol-07, Issue-01, APR 2021

continued until the defined load is taken by the component. The figure below exhibits the testing procedure.

Fig. 34: Chassis scaled model mounted on the fixture and placed on UTM

The results obtained from the UTM Load Testing are plotted in the below graph. It is observed that the Chassis successfully withstood the load with deformation in conformance to FEA deformation. It is observed that design load of the component is safely taken by the scaled model, which is 25 ton, scaled by 10. Load taken by scaled model is 2.50 KN scaling by 10 we get deformation of 1.90 mm

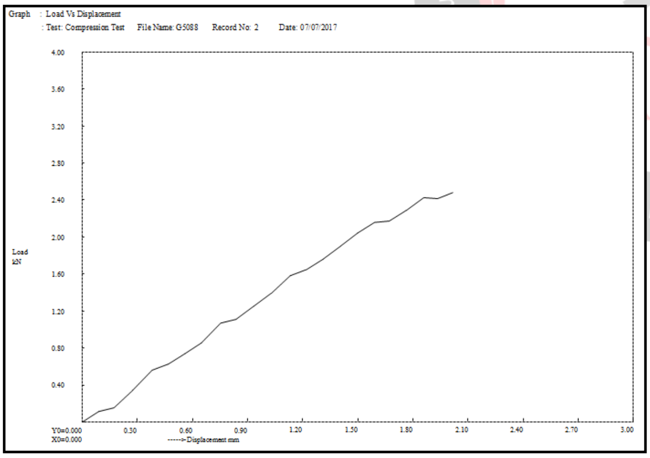

Fig. 35: Graph of Load vs displacement

VI. RESULTS AND DISCUSSION

Table 1:Modal Frequencies of Baseline and Iteration 3

Mode	Baseline Frequencies	Iteration 3
1	0	0
2	1.41E-04	0
3	4.04E-04	2.08E-04
4	7.09E-04	3.85E-04
5	1.79E-03	5.09E-04
6	2.42E-03	1.20E-03
7	1.3835	1.3594
8	8.037	8.2351
9	9.8928	10.032
10	14.653	14.751

Table 2-Results Summary

Design Iteration	**von Mises Stress (MPa)**	**Max Deformatio n (mm)**	**First Modal Frequenc y (Hz)**	**Mass (kg)**	**Mass Reductio n (%)**
Baseline	240	20.51	1.38	799.2 7	0
Iteration 1	241	21.25	1.36	770.3	4%
Iteration 2	242	19.93	1.36	760.6 5	5%
Iteration 3	240	20.1	1.36	753.1 9	6%

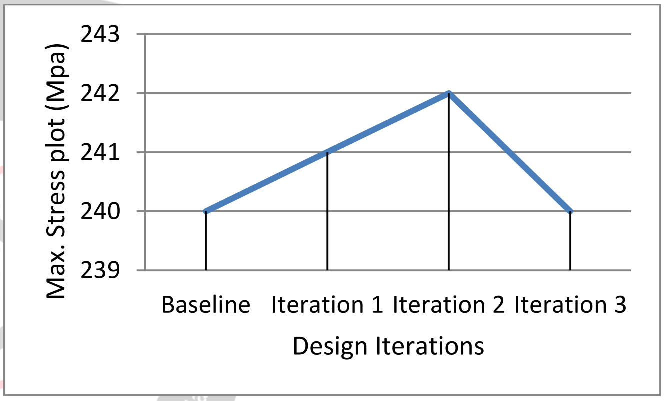

Fig. 36:- Graph of Max. Von Mises stress vs design iteration

Above Graph shows us the relation between design iterations and maximum stress observed on the fillets of the side rail. Location of the maximum stress observed is not changing much according to iterations. Also stress value only varies between 240 to 242 MPa which is almost as good as no change in the max. Stress observed in the different iterations. This graph shows us the effect of material removal on the max stress.

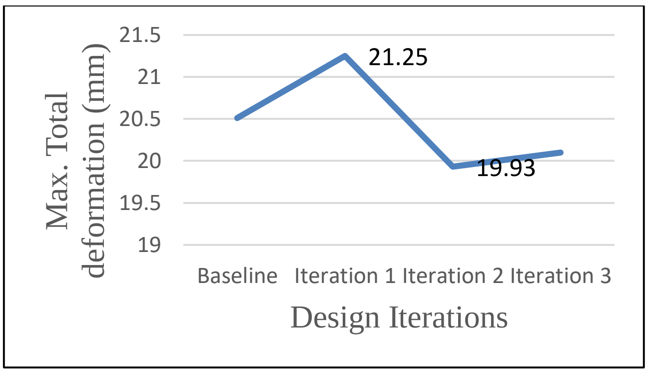

Fig. 37:- Graph of max. Total deformation vs Design Iterations

 DOI : 10.35291/2454-9150.2021.0143

ISSN : 2454-9150 Vol-07, Issue-01, APR 2021

Above Graph shows the effect of design iterations on the maximum deformation observed in the truck chassis during the static analysis. It is very clear that there is notable change in the max total deformation in every iteration. Biggest value of the max deformation is observed in the iteration 1 design 21.25 mm. And lowest value is observed at iteration 2 as 19.93 mm. In iteration it is clear that other than iteration 1 design all other design's maximum deformation values are less than baseline deformation value.

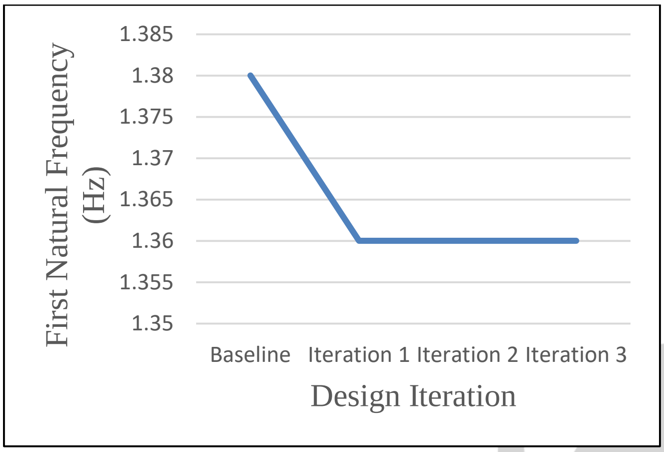

Fig. 38: - First Natural Frequencyvs Design Iterations

Change in first natural frequency with respect to design iterations. It can be seen that first natural frequency only reduces by 0.2 Hz and then remains constant in all the iterations ahead. With Iteration design weight of the chassis goes on decreasing in the graph below.

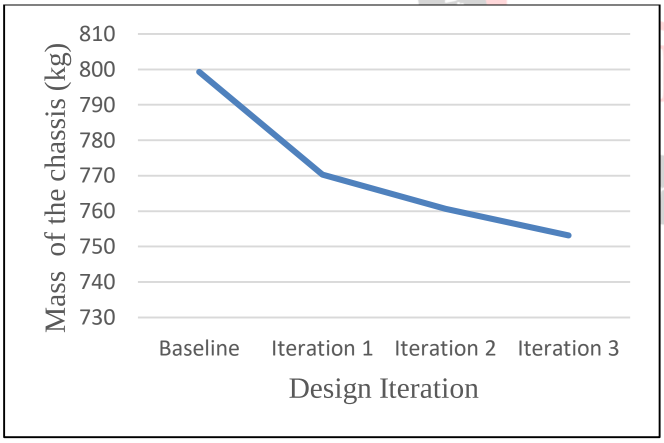

Fig. 39:- Graph of Mass of the chassis vs design iteration

Graph shows us that Mass of Baseline iteration was 799.27 Kg. and after the optimization mass of iteration 3 is 753.19 Kg. It is clearly that mass is reduced by 6 %.

In practical testing it is observed that design load of the component is safely taken by the scaled model, which is 25 ton, scaled by 10. Load taken by scaled model is 2.50 KN, with expected deformation of 2.1 mm. Deformation in FEA was observed to be 20.1 mm for full model scaling by 10 we get deformation of 1.90 mm. So 0.20 mm less deformation is observed on the UTM testing which is hardly 5.5 % error.

VII. CONCLUSION

Chassis frame model for LPK 2518 is created using 3D modeling software and design of it was concluded to be safe under the GVW according to hand calculations. Finite Element analysis was performed to find out first free modal natural frequency of the chassis which is around 1.38 Hz according to FEA results. Baseline model static analysis shows maximum von mises stress of 240 MPa in the model which is within the acceptance limit for the yield strength of steel. Design iterations performed according to stress capping results save around 6 % of the total mass of the chassis which is almost similar percentage of unloaded mass of the vehicle. In iteration 3 first natural frequencies of the chassis and maximum stress observed in the chassis stress plot is almost same as baseline results. So we save 6 % mass of the chassis without affecting the structural performance of the chassis. This proves that design of the chassis for TATA 2518 truck is successfully modified. Total deformation and load safely design load taking capacity of the scaled prototype is tested on the UTM. Total deformation of 1.90 mm and 2.50 KN of design load is safely taken by the prototype. Error of the deformation is observed to be 5.5 %.

FUTURE SCOPE

- The chassis can be manufactured and used in the current vehicle.
- Also, alternate material can be thought of for manufacturing low weight truck chassis.
- Composite materials such as glass fiber and carbon fiber can be tried and tested for manufacturing of the truck chassis.
- Dynamic analysis can be performed on the chassis to make sure that it can withstand the vibration loading applied on it throughout the vehicle's life.
- Fatigue analysis and testing can be performed to verify the effect of optimization on life of the chassis.

ACKNOWLEDGMENT

I take this opportunity to thank Dr. Kiran More and Prof. Pavan Sonawane. for valuable guidance and for providing all the necessary facilities, which were indispensable in completion of this work. Also I sincerely thanks to all the authors who worked on Analysis and optimization of Heavy Truck chassis .

REFERENCES

[1] Mohd Azizi Muhammad Nor et. al., "Stress Analysis of a Low Loader Chassis", Science Direct, International Symposium on Robotics and Intelligent

 DOI : 10.35291/2454-9150.2021.0143

Sensors 2012 (IRIS 2012), doi: 10.1016/j.proeng.2012.07.274.

[2] Chen Yanhong, Zhu Feng, “The Finite Element Analysis and The Optimization Design of The Yj3128-type Dump Truck’s Sub-Frames Based on ANSYS”, Science Direct, doi: 10.1016/j.proeps.2011.09.022

[3] Akash Singh Patel et. al., “Modeling, Analysis & Optimization of TATA 2518 TC Truck Chassis Frame using CAE Tools”, International Journal of Engineering Research & Technology (IJERT) ISSN: 2278-0181 IJERTV5IS100122Vol. 5 Issue 10, October-2016

[4] Dharmendrasinh Parmaret. al, “Design And Weight Optimization Of YJ3128 Type Dump Truck’s Frame”, International Journal of Advance Engineering and Research Development Volume 2,Issue 5, May -2015.

[5] Abdulkadir Yasaret. al, “Design, Analysis and Optimization of Heavy Vehicle Chassis Using Finite Element Analysis”, International Journal of Scientific and Technological , 2422-8702 (Online) Vol 1, No.6, 2015

[6] Hirak Patelet. al, “Structural Analysis of Truck Chassis Frame and Design Optimization for Weight Reduction”, International Journal of Engineering and Advanced Technology (IJEAT) ISSN: 2249 – 8958, Volume-2, Issue-4, April 201.

[7] Abhinn Bajaj et. al, “STATIC AND MODAL ANALYSIS OF TRUCK CHASSIS”, 3rd International Conference on Recent Innovations in Science Engineering and Management, ISBN: 978-81-932074-1-3

[8] Mehdi Mahmoodi-ket. al, “Stress And Dynamic Analysis Of Optimized Trailer Chassis”, ISSN 1330-3651(Print), ISSN 1848-6339 (Online) UDC/UDK 620.172.21:004.942]:629.351.021.21

[9] Patel Vijaykumar V et. al, “Structural Analysis of Automotive Chassis Frame and Design Modification for Weight Reduction”, International Journal of Engineering Research & Technology (IJERT), Vol. 1 Issue 3, May – 2012

[10] IshaTikekar et al, “Weight Reduction Of Heavy Duty Truck Chassis Through Material Optimization”, International Journal of Engineering Research and General Science Volume 4, Issue 3, May-June, 2016, ISSN 2091-2730

[11] K. Venkatesh et al, “Optimization Of Truck Chassis Of Support Stiffness To Improve The Fundamental Natural Frequency”, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e- ISSN: 2278-1684, p-ISSN : 2320–334X

[12] Han Quan-li et al, “Research on Optimization Design of Heavy-duty Truck Frame Based on the Sensitivity”, 978-1-4244-5539-3/10

[13] Rahul L. Patel et al, “Weight optimization of chassis frame using Pro-Mechanica”, SSRG International Journal of Mechanical Engineering (SSRG-IJME) – volume 1 Issue 8–December 2014

[14] Nouby M. Ghazaly, “Applications of Finite Element Stress Analysis of Heavy Truck Chassis: Survey and Recent Development”, Journal of Mechanical Design and Vibration, 2014, Vol. 2, No. 3, 69-73.

 DOI : 10.35291/2454-9150.2021.0143