Structural Analysis of Automotive Chassis & Optimization of TATA ACE

Akash Shree Deo[1] Vinayak Narayan Untwale[2] Abhishek Keskar[3]
[1,2]Junior Scientist
[1,2,3]A2 Innovation PVT. LTD., India

Abstract— Automotive chassis can be considered as backbone of any vehicle. Chassis serves as a frame work for supporting the body and components and hence it should be rigid enough to withstand shock, twist, and vibration and also should have adequate bending stiffness for better handling. The objective of this work is to validate the design of chassis of TATA Super Ace using Finite Element Analysis (FEA). The main objective of this work is to know the maximum stress and deformation location and optimize it. The first stage includes designing CAD model using Solidworks and Ansys Design Modular. Second stage involves analysis on the basis of Structural, Modal, and its optimization using Ansys Tools.

Key words: FEA, Automotive Chassis, Static Structural, Optimization, Modal

I. Introduction

Chassis is tasked at holding all the essential components of the vehicle like engine, suspension, gearbox, breaking 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. It should be strong to withstand vibrations.

The two main goals of the automotive chassis are as follows

- Hold the weight of the components
- To rigidly fix the suspension components together when moving.

The chassis frame consists of side members attached with series of cross members. Stress analysis using finite element analysis (FEA) can be used to locate the critical point which has the highest stress. The critical point is one of the factors that may cause the failure. Along with strength, an important consideration in chassis design is to have adequate bending stiffness for better handling characteristics. So, maximum stress and deflection are important criteria for the design of the chassis. CAD Model:

A. Aim of the Study

The objectives of the work are:

1) Perform stress analysis of base chassis model to know critical stress locations and deflection at maximum load
2) To analyze the sensitivity of frame web height to the change in thickness and vice-versa.
3) To find analyzed chassis dimensions.
4) To find out natural frequencies of the existing as well as modified chassis with the help of ANSYS and validate it with testing.

II. Calculations

- Max Gross vehicle weight= 2255 kg.
- Payload on vehicle= 1000 kg
- FOS= 2
- Total load= (2255+1000) ×9.81×2
- = 63863 N.
- Force On one side member= 31931 N

A. Side Member

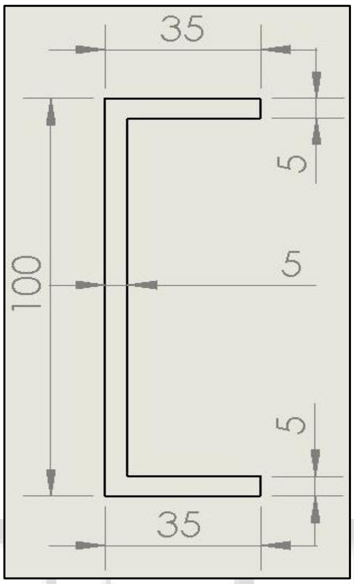


Fig. 1: Side Member

B. Cross Member

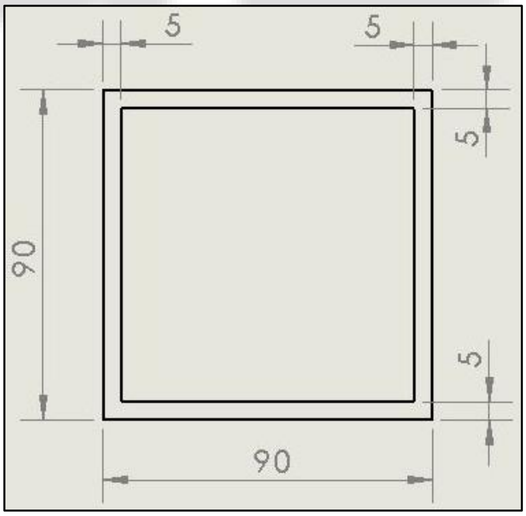


Fig. 2: Cross Member

- Dimensions of the Chassis taken for Calculation

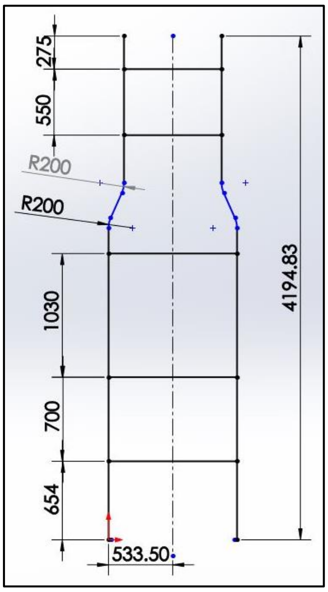


Fig. 3:

C. Theoretical Calculations

Material of the chassis is Steel E = 2.10×10^5 N/mm2
Poisson Ratio = 0.31
The distance between the fixed supports is 2718.8 mm.
The force acting on the one side member as a UDL.= 31931/4218.294(Length on which force is distributed)
UDL= 7.569.
Now using formula for maximum deformation in fixed supported simple beam

$$y_c = \frac{w \ell^4}{384 \ EI}$$

$$y_c = \frac{(7.569)(2718.8)^4}{384 * (200000) * (1094200)}$$

$$y_c = 4.92 \text{ MM.}$$

III. Static Structural Analysis

A. Geometry

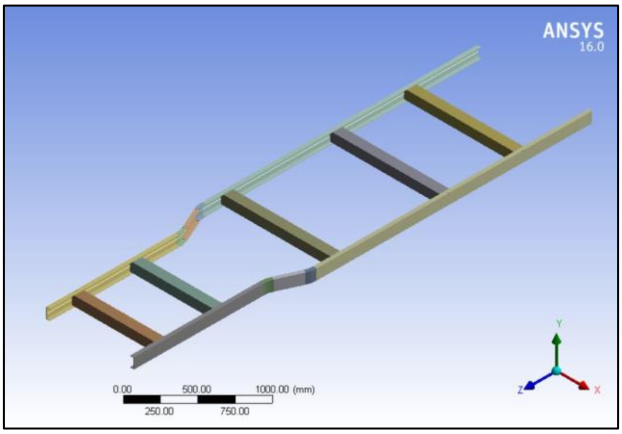


Fig. 4: Geometry

We have used line body and C channel cross section for side members and Box section for cross members.

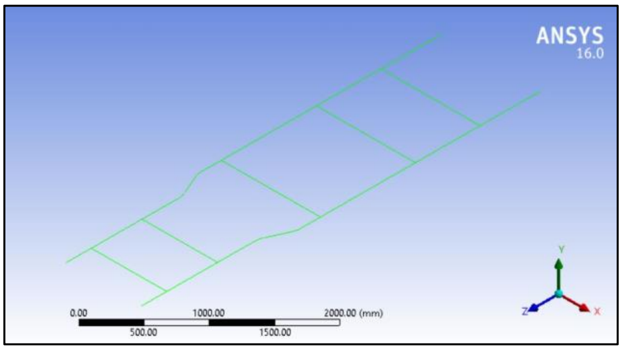


Fig. 5:

B. Meshing

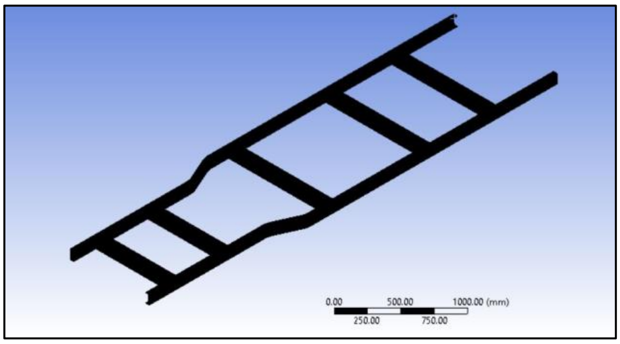


Fig. 6: Meshing

We have used beam element with mesh size of 1mm for all the body meshing.

C. Connections

We have provided spot weld at all the points where cross members and side members are connected in the body. This will help to simulate the real chassis behaviour.

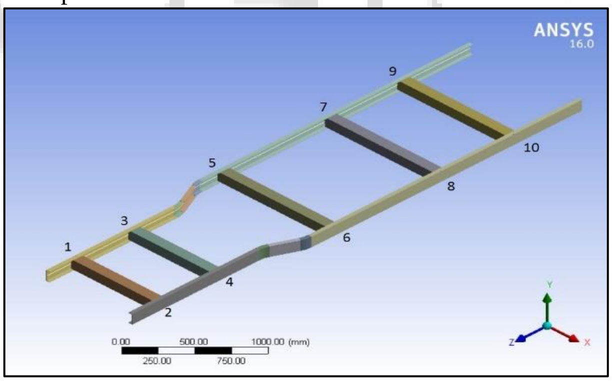


Fig. 7: Connections

D. Boundary Conditions

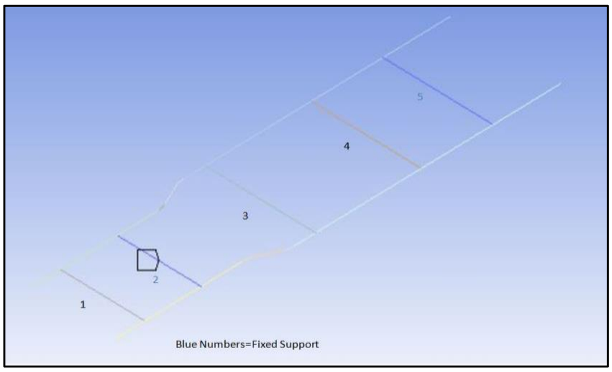


Fig. 8: Boundary Conditions

Here we have provided fixed support to cross member 2 & 5 as we will be considering the case in which the suspension has completed the full travel and now all the force will be transmitted to the chassis body.

Force acting on the body of 63863 is applied to full length side members.

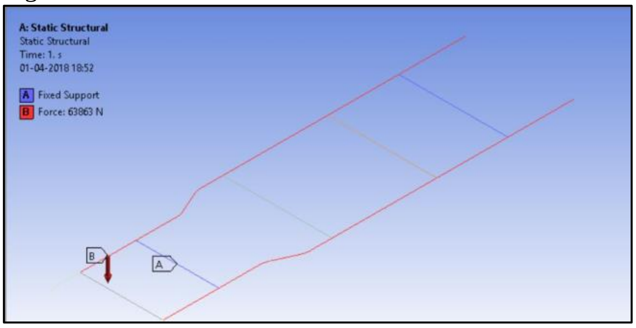


Fig. 9:

E. Results

1) Total Deformation

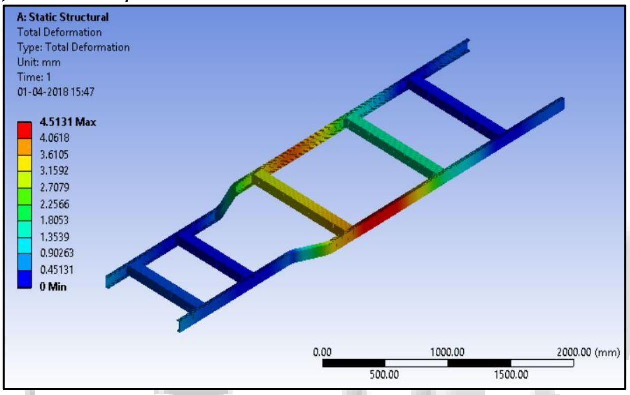


Fig. 10: Total Deformation

The maximum deformation is 4.5131 MM which occurs at the centre of chassis at a distance of 2067mm from small width end.

2) Maximum Combined Stress

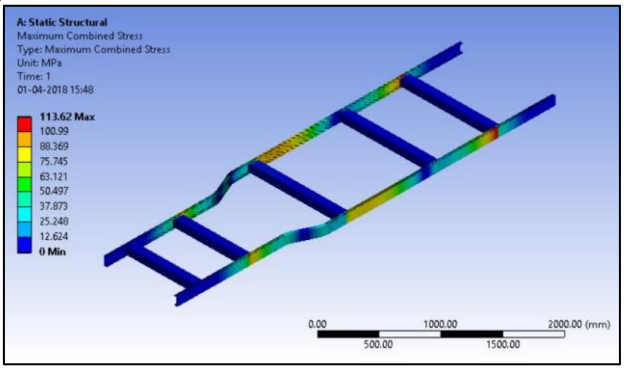


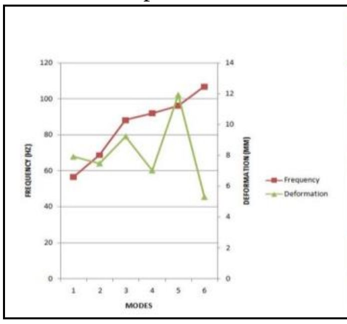
Fig. 11: Maximum Combined Stress

As we can see the maximum combined stress is 113.62 MPa which is well below the permissible limit which occurs mainly at the rear end fixed support.

As we can see the difference between theoretical and FEA result is very close. The error is due to the difference between the original and FEA conditions of the analysis.

IV. Modal Analysis

A modal analysis determines the vibration characteristics (natural frequency and mode shapes) of a structure or a machine component.



MODE	FREQUENCY	DEFORMATION
1	56.754 Hz	7.9388 mm
2	68.941 Hz	7.4999 mm
3	88.329 Hz	9.2596 mm
4	92.146 Hz	7.0406 mm
5	96.33 Hz	11.94 mm
6	107 Hz	5.3262 mm

Fig. 12: Modal Analysis

Here is the summary for the first 6 modes of the chassis. The care should be taken that these frequencies should not be matched with the working frequencies to avoid resonance.

V. Optimization

An optimization problem is a problem in which certain parameters (design variables) needed to be determined to achieve the best measurable performance (objective function) under given constraints.

A. Parameters

An optimization problem is a problem in which certain parameters (design variables) needed to be determined to achieve the best measurable performance (objective function) under given constraints.

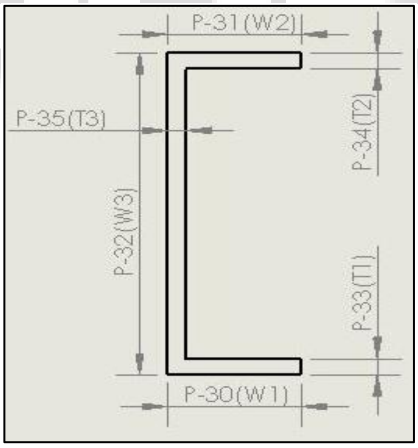


Fig. 13:

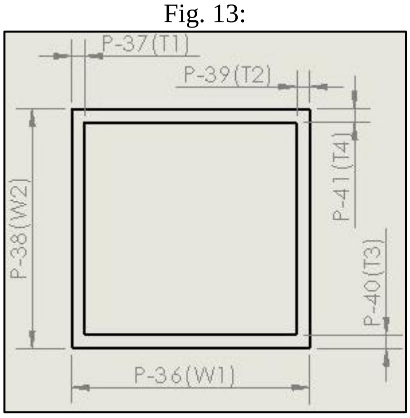


Fig. 14:

Here for research work we have used 12 Parameters as shown above
Input Parameters=Total 12 Parameters As Shown Above.
Output Parameters=
1) P-13 [Total Deformation Maximum]
2) P-29[Maximum Combined Stress]
3) P-42[Geometrical Mass]

Here the input parameters were the Height, Width and thickness at different sections of the chassis members.

These parameters have been optimized with the constraint on the total deformation and maximum combined stress of the chassis.

The output parameters used have a constraint on the maximum combined stress to be less than 250 MPa.

B. Design of Experiment

DOE is used to effectively sample a design space (e.g., all design parameters for the chassis) so that a statistical model can be built to predict responses (e.g., the maximum stress, or the first natural frequency, or the maximum deformation) of a given design.

- We will be using central composite method.
- There were total 280 design points generated from the analysis.

C. Response Surface

Building the response surface also allows us to perform sensitivity analysis, i.e., to see how much the objective changes when each variable changes. When a large number of variables exist, sensitivity analysis allows us to figure out the most important variables to design for, and thus reduce the computational cost of the optimization.

As we have used 2^{nd} order polynomial to fit the data points it is required to check whether the final polynomial gives you the best fit or not. The table given below gives you information about the ideal values required and the actual values of the fit.

	A	B	C	D
1	Name	P13 - Total Deformation Maximum	P29 - Maximum Combined Stress Maximum	P42 - Geometry Mass
2	Goodness Of Fit			
3	Coefficient of Determination (Best Value = 1)	1	0.9933	1
4	Adjusted Coeff of Determination (Best Value = 1)	1	0.99265	1
5	Maximum Relative Residual (Best Value = 0%)	0.025412	1.067	0
6	Root Mean Square Error (Best Value = 0)	0.00030554	0.36961	1.1741E-13
7	Relative Root Mean Square Error (Best Value = 0%)	0	0.32336	0
8	Relative Maximum Absolute Error (Best Value = 0%)	0.41168	28.426	0
9	Relative Average Absolute Error (Best Value = 0%)	0.078642	6.2769	0
10	Goodness Of Fit for Verification Points			
11	Maximum Relative Residual (Best Value = 0%)	0.045393	1.1093	0
12	Root Mean Square Error (Best Value = 0)	0.0014899	0.90417	2.0122E-13
13	Relative Root Mean Square Error (Best Value = 0%)	0.033745	0.87343	0
14	Relative Maximum Absolute Error (Best Value = 0%)	0.67753	24.458	0
15	Relative Average Absolute Error (Best Value = 0%)	0.45462	18.996	0

Fig. 15:

As we can see above the curve fits to the output parameters up to the expectation of the ideal values with slight difference which is well within the requirement for convergence of a solution.

1) Response Surface Chart

This chart includes the effect of the input parameters on the output parameters hence giving us the guidance for the optimization process as per the requirement of the customer.

This is a very important chart as this chart provides us with the required amount of data interpretation for the optimization process.

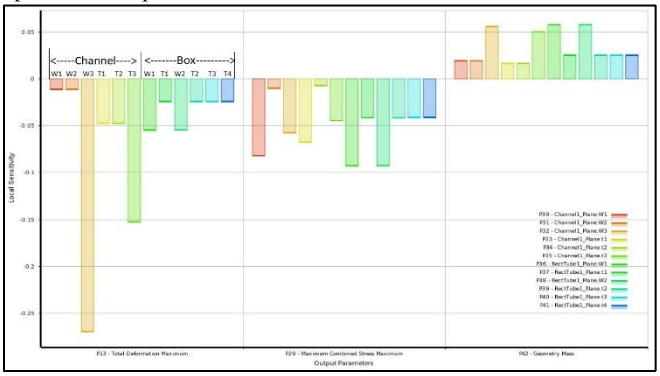


Fig. 16:

As can be seen in the above chart, various conclusions regarding the effect of change in the input parameter value on the output parameter can be seen such as

1) The effect of the change in the height of C section W3 parameter will have the best effect on the reduction of the total deformation.

Hence if user wants to reduce the total deformation as from chart we can see all parameters have inverse relationship with total deformation factor i.e. increase in any parameter will reduce the total deformation and vice-versa.

Hence to reduce the total deformation one must consider first the increment of the height of c section i.e. W3 parameter

2) Similarly we can see that effect of all parameters have been inversely proportional to the Maximum combined stress and hence the most sensitive parameter for the maximum combined stress is W1 (C-section), W1 (BOX section) and W2 (BOX section) should be considered. Hence to reduce the total deformation one must consider first the increment of the height of c section i.e. W1 parameter for both the cross members.
3) Similarly the most important objective for the optimization i.e. Geometric Mass is reduced with the directly proportional relation between all the parameters. But still the most sensitive of the parameter is W3 (C-section).

Hence all the above factors are combined together to obtain the final optimization product which will consider all the constraint of the output while still giving the least possible geometric mass for the given system.

4) Direct Optimization.

Here we have used Sampling method of Optimization which uses various design points generated by *DOE* and gives us the best *CANDIDATE* points which will satisfy the output parameter constraints.

The Constraint for OPTIMIZATION-

1) Total Maximum Deformation-MINIMUM.
2) Maximum Combined Stress-It Should be less than 250MPA (yield stress of Structural Steel)
3) GEOMETRICAL WEIGHT-MINIMUM.

a) Trade off Chart

When an Optimization component is solved, the Tradeoff chart is created. The Tradeoff chart is a 2D or 3D scatter chart representing the generated samples. The colours applied to the points represent the Pareto front they belong to, from red (the worst) to blue (the best points).

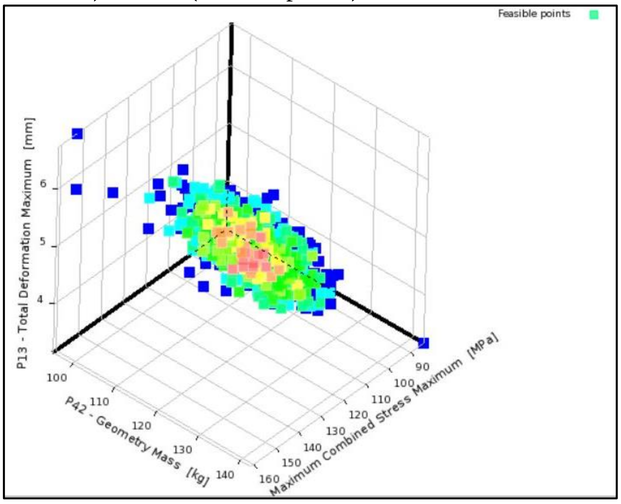


Fig. 17:

b) Candidate Points

These are the points which are available after the optimization process. They show the best values of the parameters for which you can get the optimized best results within the constraint set by the user. These are the best solutions from the samples.

21	Candidate Points			
22		Candidate Point 1	Candidate Point 2	Candidate Point 3
23	P30 - Channel1_Plane.W1 (mm)	31.5	31.511	31.518
24	P31 - Channel1_Plane.W2 (mm)	31.5	35.004	33.254
25	P32 - Channel1_Plane.W3 (mm)	90	96.677	103.34
26	P33 - Channel1_Plane.t1 (mm)	4.5	4.7005	4.9005
27	P34 - Channel1_Plane.t2 (mm)	4.5	4.6434	4.7862
28	P35 - Channel1_Plane.t3 (mm)	4.5	4.5914	4.6823
29	P36 - RectTube1_Plane.W1 (mm)	81	82.394	83.778
30	P37 - RectTube1_Plane.t1 (mm)	4.5	4.5593	4.6181
31	P38 - RectTube1_Plane.W2 (mm)	81	81.956	82.904
32	P39 - RectTube1_Plane.t2 (mm)	4.5	4.544	4.5875
33	P40 - RectTube1_Plane.t3 (mm)	4.5	4.535	4.5695
34	P41 - RectTube1_Plane.t4 (mm)	4.5	4.5328	4.565
35	P13 - Total Deformation Maximum (mm)	6.7364	5.7676	5.055
36	P29 - Maximum Combined Stress Maximum (MPa)	153.06	146.82	137.79
37	P42 - Geometry Mass (kg)	95.441	100.94	104.91

Fig. 18:

From the candidate points diagram as shown above, these are the design points which are best suited for the optimization process with the given constraint. As shown in the above image the candidate point 1 has maximum combined stress of 153.06MPa which is well below the required 250MPa permissible strength criterion. Hence we will use these design point to obtain the result for the optimized chassis.

Given below are the comparison diagram for the optimized cross section and the original cross section.

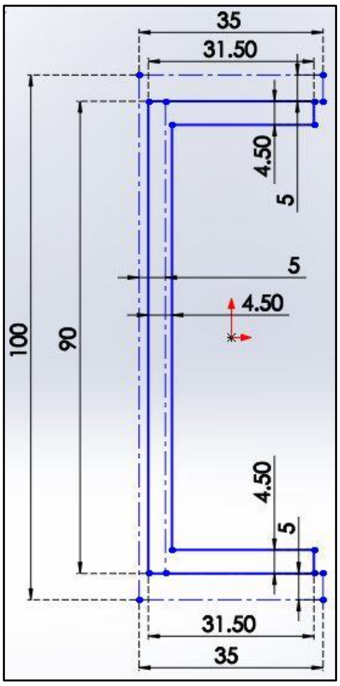


Fig. 19:

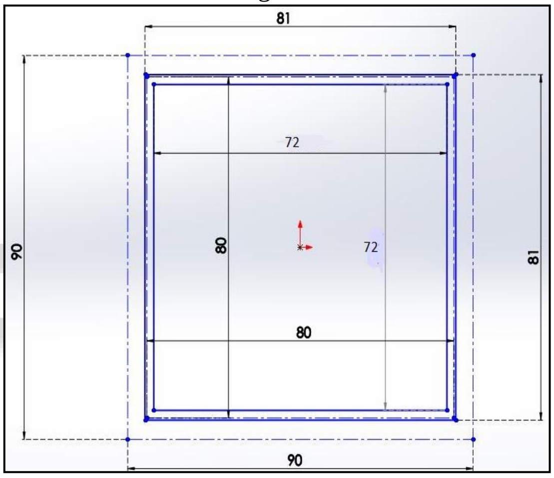


Fig. 20:

VI. COMPARISON

SR.NO.	PARAMETER	ORIGINAL CHASSIS	OPTIMIZED CHASSIS	PERCENTAGE DIFFERENCE
1	Total Deformation	4.5131 mm (LOCATION-2067MM FROM RIGHT)	6.7364 mm (LOCATION-2067MM FROM RIGHT)	-49.26%
2	Geometric Mass	117.83 kg	95.441 kg	19%
3	Maximum Combined stress	113.62 MPa (LOCATION-AT CROSS MEMBER 5)	153.06 MPa (LOCATION-AT CROSS MEMBER 5)	-34.71%

Fig. 21: Comparison

1) The weight reduction of 19% has been achieved which will reduce the weight of the vehicle which will ultimately reduce the cost of fuel required and the cost of manufacturing material required

2) The maximum combined stress in the optimized chassis is 153.06MPa which were within the permissible limit as per the 250MPa strength of the structural steel.
3) The maximum deformation can be used for the design of the stiffeners and the suspension system to reduce its adverse effect on the system in dynamic conditions.

VII. CONCLUSION

From the above work we can conclude that the

1) The effect of the change in the height of C section will have the best effect on the reduction of the total deformation.
2) Hence if user wants to reduce the total deformation as from chart we can see all parameters have inverse relationship with total deformation factor i.e. increase in any parameter will reduce the total deformation and vice-versa.
3) We have also obtained the weight reduction of the 19%, also by keeping the combined stress well below the permissible limit.
4) FEA can be effectively used to find the optimization of the chassis which can further reduce the fuel consumption as well as overall cost of the vehicle.

LITERATURE REVIEW

Reference Books

[1] Ramamrutham S., Strength of materials, First edition 1962, reprint 1970, Dhanpat Rai and sons publication, Pages 528 to 581.
[2] Singh Kripal, Automobile Engineering Vol.1, reprint year 2000, Standard Publishers & distributors.
[3] Ansys Theory reference, release 5.6, 11th edition, November 1999, Ansys.Inc.

Reference Journal Papers

[4] Optimization and Design Exploration of Volvo FM 400 8X4 Tipper with 32 CU.M Coal Body Using Finite Element Analysis Amit Potnis, Rucha Shende, Nilesh Raul Prof. Amit Desale, Abhishek Keskar Marathwada Mitra Mandal's College of Engineering, Pune, India.A2 Innovative Research and Training.
[5] Madhu P.s.and Venugopal T.R., ISSN: 2248-9622, Vol. 4, Issue 5(Version 3) "Static Analysis, Design Modification and Modal Analysis of Structural Chassis Frame'', May 2014.
[6] MOHAMAD SAZUAN BIN SARIFUDIN (UNIVERSITI MALAYSIA PAHANG) BE Thesis. "DESIGN AND ANALYSIS OF CAR CHASSIS" Jan 2012
[7] Patel Vijaykumar V, Prof. R. I. Patel, IJERT Vol. 1 Issue 3,
[8] "Structural Analysis of Automotive Chassis Frame and Design Modification for Weight Reduction'', May 2012
[9] Patil Hemant , Sharad D.Kachave, IJARSE, Vol. No.2, Issue No.7,
[10] "EFFECTS OF WEB AND FLANGE THICKNESS ON BENDING STIFFNESS OF CHASSIS'', July 2013.
[11] Roslan Abd Rahman, Mohd Nasir Tamin, Ojo Kurdi,Jurnal Mekanikal , No. 26, 76 - 85, Stress Analysis of Heavy Duty Truck Chassis as A ppreliminary data for its fatigue life prediction using FEM Tools , Dec 2008.
[12] Ravinder Pal Singh, Jurnal Mekanikal, No. 31, 46 - 61
[13] "STRUCTURAL PERFORMANCE ANALYSIS OF FORMULA SAE CAR'', Dec. 2010.
[14] Vyas Basia, P.B., Prof. Patil M.Y, IJESR/ Vol-3/Issue-5/3018-3022)
[15] "DYNAMIC ANALYSIS AND OPTIMIZATION OF HEAVY MOTOR VEHICLE CHASSIS FRAME", May 2013.

