IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 12, Issue 2 Ver. IV (Mar - Apr. 2015), PP 63-71
www.iosrjournals.org

Design and analysis of ladder frame chassis considering support at contact region of leaf spring and chassis frame

Amar R. Contractor[1], Gaurav P. Rathod[2], Tushar M. Patel[3]
[123](*Mechanical Engineering, LDRP-ITR Gandhinagar/Kadi Sarva Vishwavidyalaya, INDIA)*

Abstract: *Truck chassis is the structural backbone of any vehicle which supports the components and payload placed upon it. Also, the chassis should be rigid enough to withstand the shock, twist, vibration and other stresses. A chassis design should have adequate bending stiffness for better handling characteristics along with strength. This paper presents the finite element analysis of the chassis of Eicher 11.10 using ansys workbench and stress computation using standard techniques. Stress determination of the stresses of chassis before manufacturing is vital to improve the design. The design can be improved even before developing the prototype using finite element analysis. In this present work chassis is modelled in Creo Parametric 3.0 and static structural characteristics are analysed using Finite Element models.*
Keywords: *Creo Parametric 3.0, FEA, ladder frame, structural steel, Truck Chassis*

I. Introduction

The chassis of an automobile provide mounting points for the components like engine, driveline, suspension system and wheels. The main functions of the chassis are to support the chassis components and the body to withstand static and dynamic loads without excessive deflection or distortion. The frame must be rigid enough to support or carry all the loads and forces that the vehicle is subjected to in operation. A frame must also be flexible enough to handle shock loads and the twists, bends, sway and sag that it encounters under different road or load conditions. The frame should be able to flex under different situations, while being able to return to its original shape when loads or forces are removed.

From the comparison it has been found that Eicher has lowest height of frame section and Eicher 11.10 has maximum load body length (length of Frame). So this frame is having greatest possibility of bending among all, thus this frame has been considered for the case study [1]. The chassis frame is made of two side members joined with a series of cross members. These cross members provide better handling of the vehicle, prevents deflection and gives torsional strength to resist twisting of the chassis. For the analysis of the chassis frame appropriate model of the chassis is developed and analysed by the Finite Element Method (FEM).

II. Literature review

Structural optimization using Finite Element Analysis (FEA) and other computational tools has become a major part in research and development process in recent years. The method has wide application and enjoys extensive utilization in the structural, thermal and fluid analysis areas. Roopesh (2002) performed the optimization of the automotive chassis with the constraints of stiffness, strength and natural frequency. Structural systems like the chassis can be easily analysed using the finite element techniques. A proper finite element model of the chassis was developed. The chassis was modelled with beam elements and pipe elements in ANSYS. Weight optimization was done on the modelled chassis using the first order optimization methods [2]. Karita, Kohiyama, Kobiki, Ooshima, Hashimoto (2003) observed that aluminium was successfully used as the material for the chassis frame which was a main structural member of heavy-duty trucks, to significantly reduce the truck weight and so allow the payload to be increased. The shape and configuration of the aluminium frame design were optimized while maintaining strength and rigidity equivalent to those of a standard steel frame by using computer-aided-engineering analysis. Using the aluminium frame thus developed [3]. Butdee and Vignat (2008) applied the TRIZ principle and parameters to assist a light weight bus body design which was compared to the existing design. The bus body model was created by CAD and transfer data to CAE using FE analysis. The weight reduction process was then followed up from the analysis. The new light weight bus body design was tested by the same method of FE analysis. The same result of body strength was accepted and used for design and manufacturing. The tested TRIZ method can save material used, production cost and time [4]. Mohd and Abd (2009) presented an analysis of the static stress that acts on the upper surface of the truck chassis. Finite element analysis helped in accelerating the design and development process by minimizing the number of physical tests, thereby reducing the cost and time for analysis. The commercial finite element package Algor was used for this simulation. 3-D model of the truck chassis was drawn by using Solid Works. Results showed the critical part of the chassis and some modifications were also suggested to reduce the stress and to improve the strength of the truck chassis [5]. Nor, Rashid, Mahyuddin, Azlan, Mahmud (2012) performed

Finite element modelling (FEM), simulations and analysis using a modelling software i.e. CATIA V5R18. Firstly, a 3-D model of low loader based on a design from SESB was created by using CATIA. The results of analysis revealed that the location of maximum deflection and maximum stress agree well with the theoretical maximum location of simple beam under uniform load distribution [6].

III. Material of model

The material for the chassis is defined ST 52 which is widely used material for the chassis. The material properties are as shown in Table 1 [7].

Table 1: Material properties of chassis

Material	ST 52
Modulus of Elasticity, E (MPa)	2×10^5
Poisson's Ratio, ν (MPa)	0.3
Tensile Strength (MPa)	520
Yield Strength (MPa)	360

IV. Basic calculation

The calculation of the stress produced in chassis is calculated by the moment distribution method. This method is basically a displacement method of analysis. But this method side steps the calculation of the displacement and instead makes it possible to apply a series of converging corrections that allow direct calculation of the end moments [8].

Model No. = 11.10 (Eicher E2)

Side bar of the chassis are made from "C" Channels with 210mm x 76 mm x 6 mm

Front Overhang (a) = 345 mm

Rear Overhang (c) = 995 mm

Wheel Base (b) = 3800 mm

Capacity of Truck = 8 ton = 8000 kg

= 78480 N

Capacity of Eicher with 1.25% = 78480 N + 19620 N = 98100 N

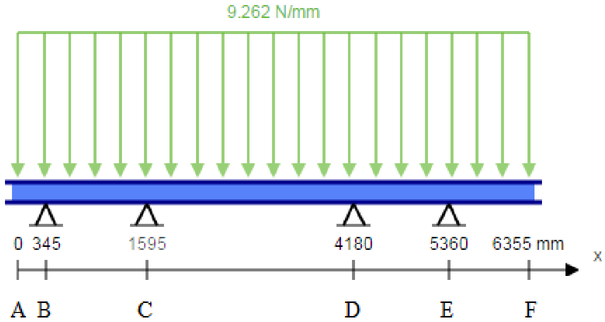

Figure 1 Chassis as a simply supported beam with overhang.

Weight of the body and engine = 2 ton = 2000 kg = 19620 N

Total Load = Capacity of the Chassis + Weight of body and engine

= 98100 + 19620

= 117720 N

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 = $\frac{\text{Total load acting on the chassis}}{2} = \frac{117720}{2}$

= 58860 N / Beam

4.1 Fixed End Moment Calculations:

Fixed end moment for each loaded span are determined assuming both end fixed.

$$\mathbf{M_{FBC}} = -\frac{wl^2}{12} = -\frac{9.262 \times 1250^2}{12} = -1205989.5833 \text{ N.mm}$$

$$\mathbf{M_{FCB}} = \frac{wl^2}{12} = \frac{9.262 \times 1250^2}{12} = 1205989.5833 \text{ N.mm}$$

$$\mathbf{M_{FCD}} = -\frac{wl^2}{12} = -\frac{9.262 \times 2585^2}{12} = -8657600.987 \text{ N.mm}$$

$$\mathbf{M_{FDC}} = \frac{wl^2}{12} = \frac{9.262 \times 2585^2}{12} = 8657600.987 \text{ N.mm}$$

$$\mathbf{M_{FDE}} = -\frac{wl^2}{12} = -\frac{9.262 \times 1180^2}{12} = -1804016.4189 \text{ N.mm}$$

$$\mathbf{M_{FED}} = \frac{wl^2}{12} = \frac{9.262 \times 1180^2}{12} = 1804016.4189 \text{ N.mm}$$

$$\mathbf{M_{FBA}} = \frac{wl^2}{2} = \frac{9.262 \times 345^2}{2} = 551204.775 \text{ N.mm}$$

$$\mathbf{M_{FEF}} = -\frac{wl^2}{2} = -\frac{9.262 \times 995^2}{2} = -4584805.775 \text{ N.mm}$$

The stiffness factors for each span at the joint are calculated. Using these values the distribution factors are determined from the equation (1):

$$DF = K/\sum K \tag{1}$$

Table 2 Distribution Factor

Joint	Member	Relative Stiffness(K)	$\sum K$	$DF = \frac{K}{\sum K}$
C	CB	$\frac{3}{4} \times \frac{I}{L} = 8023.42$	13196.49	0.60799
	CD	$\frac{I}{L} = 5173.06$		0.39200
D	DC	$\frac{I}{L} = 5173.06$	13672.46	0.37835
	DE	$\frac{3}{4} \times \frac{I}{L} = 8499.39$		0.621643

As the chassis is made from the same material the modulus of elasticity E is same for all members, thus the term is removed from the equation which gives the relative stiffness factor: $K_r = I/L$ (for far end fixed) & $K_r = 3I/4L$ (for far end hinged).

Moment distribution is done by assuming that all the joints at which the moments in the connecting spans must be determined are initially locked. Then the moment that is needed to put each joint in equilibrium was determined. The joints were released or unlocked and the counterbalancing moments were distributed into connecting span at each joint using distributing factors. Carry these moments in each span over to its other end by multiplying each moment by carry over factor. By repeating this cycle of locking and unlocking the joints, it will be found that the moment corrections will diminish since the beam tends to achieve the final deflection shape. Each column of fixed end moments, distributed moments and carry over moment should then be to get final moments at the joints [8].

Table 3 Distribution Table

Joint	B		C		D		E	
Members	BA	BC	CB	CD	DC	DE	ED	DF
DF	01	1	0.61	0.39	0.38	0.62	1	01
FEM	551204	-1205989	1205989	-8657600	8657600	-1804016	1804016	-4584805
Balancing		654784					2780789	
C.O.			327392			1390394		
Initial Moments	551204	-551204	1533381	-8657600	8657600	-413621	4584805	-4584805
Balance			4345773	2778445	-3132712	-5111267		
C.O.			--------	-1566356	1389222	--------		
Balance			955477	610878	-527904	-861318		
C.O.			--------	-263952	305439	--------		
Balance			161010	102941	-116066	-189372		
C.O.			--------	-58033	51470	--------		
Balance			35400	22633	-19557	-31911		
Total	551204	-551204	7031044	-7031044	6607491	-6607491	4584805	-4584805

4.2 Reaction at Support:

AB:

$\sum V = 0$

$R_{VB1} - 9.262 \times 345 = 0$

$R_{VB1} = 3195.39$ N

BC:

$\sum$ **V = 0**

$R_{VB2} + R_{VC1} = 1250 \times 9.262$

$\sum M = 0$ @ B

$R_{VC1} \times 1250 + 551204.78 - 7031044.1 - \frac{9.262 \times 1250^2}{2} = 0$

$R_{VC1} = 10972.62$ N

$R_{VB2} = 604.88$ N

CD:

$\sum$ **V = 0**

$R_{VC2} + R_{VD1} = 2585 \times 9.262$

$\sum M = 0$ @ C

$R_{VD1} \times 2585 + 7031044.1 - 6607491.235 - \frac{9.262 \times 2585^2}{2} = 0$

$R_{VD1} = 11807.284$ N

$R_{VC2} = 12134.98$ N

DE:

$\sum$ **V = 0**

$R_{VD2} + R_{VE1} = 1180 \times 9.262$

$\sum M = 0$ @ D

$R_{VE1} \times 1180 + 6607491.235 - 4584805.78 - \frac{9.262 \times 1180^2}{2} = 0$

$R_{VE1} = 3750.44$ N

$R_{VD2} = 7178.72$ N

EF:

$\sum$ **V = 0**

$R_{VE2} - 995 \times 9.262 = 0$

$R_{VE2} = 9215.69$ N

$R_{VB} = R_{VB1} + R_{VB2} = 3800.27$ N

$R_{VC} = R_{VC1} + R_{VC2} = 23107.60$ N

$R_{VD} = R_{VD1} + R_{VD2} = 18986$ N

$R_{VE} = R_{VE1} + R_{VE2} = 12966.13$ N

4.3 Bending Moment:

@ B $= \frac{9.262 \times 345^2}{2} = 551204.77\,N.mm$

@ M $= \left[9.262 \times \frac{(345+625)^2}{2}\right] - (3800.27 \times 625) = -1767860.85\,N.mm$

@ C $= \left[9.262 \times \frac{(345+1250)^2}{2}\right] - (3800.27 \times 1250) = 7031042.27\,N.mm$

@ J $= \left[9.262 \times \frac{(345+1250+1292.5)^2}{2}\right] - [3800.27 \times (1250+1292.5)] - [23107.6 \times 1292.5]$

$= -917073.38\,N.mm$

@ D $= \left[9.262 \times \frac{(345+1250+2528)^2}{2}\right] - [3800.27 \times (1250+2528)] - [23107.6 \times 2528]$

$= 4632400\,N.mm$

@ N $= \left[9.262 \times \frac{(345+1250+2528+590)^2}{2}\right] - [3800.27 \times (1250+2528+590)] -$

$[23107.6 \times (2528+590)] - [18986 \times 590] = 3984115.15\,N.mm$

$$@\,\mathbf{E} \;=\left(9.262\times\frac{995^2}{2}\right)=4584805.77\,N.mm$$

4.4 Stress produced on the Beam is as under

$$\sigma=\frac{M}{Z}$$

$$=\frac{7031042.275}{127356}$$

$$=55.20\ \mathrm{N/mm^2}$$

V. Finite element analysis

The Finite Element Analysis (FEA) is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Simply stated, a boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of the domain [9].

An unsophisticated description of the FE method is that it involves dividing a structure into several elements (pieces of structure), describing the behaviour of each element in a simple way, then reconnecting elements at nodes as if nodes were pins or drops of glue that hold elements together as shown in Fig. 2. This process results in a set of simultaneous algebraic equations. In stress analysis these equation are equilibrium equations of the nodes. There may be several hundred or several thousand such equations, which mean that computer implementation is mandatory [10].

5.1 General Procedure for FEA

There are three basic steps in formulating finite element analysis, viz.: pre-processing, solution and post processing. The pre-processing (model definition) step includes: define the geometric domain of the problem, the element type(s) to be used, the material properties of the elements, the geometric properties of the elements (length, area, and the like), the element connectivity (mesh the model), the physical constraints (boundary conditions) and the loadings [9].

The next step is solution, in this step the governing algebraic equations in matrix form and computes the unknown values of the primary field variable(s) are assembled. The computed results are then used by back substitution to determine additional, derived variables, such as reaction forces, element stresses and heat flow. Actually the features in this step such as matrix manipulation, numerical integration and equation solving are carried out automatically by commercial software [10].

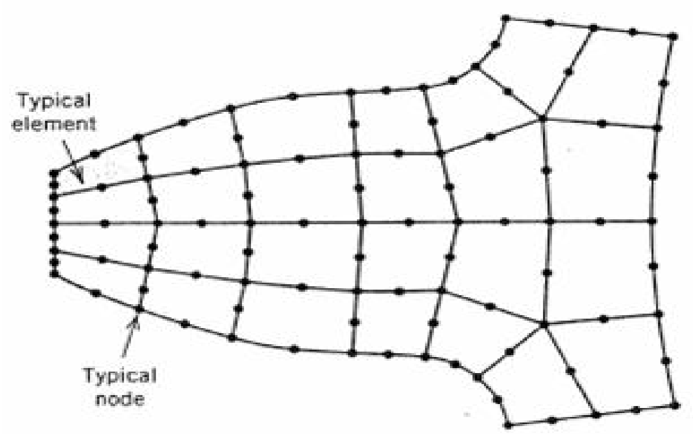

Figure 2. A coarse–mesh, two-dimensional model of gear tooth.

The final step is post processing, the analysis and evaluation of the result is conducted in this step. Examples of operations that can be accomplished includes ort element stresses in order of magnitude, check equilibrium, calculate factors of safety, plot deformed structural shape, animate dynamic model behavior and produce color-coded temperature plots. The large software has a preprocessor and postprocessor to accompany the analysis portion and the both processor can communicate with the other large programs. Specific procedures of pre and post processing are different dependent upon the program [9].

5.2 Modelling of chassis frame

The model of existing chassis as per the dimension is created in Creo 3.0 as shown in Fig. 3. The model is then imported into ANSYS workbench. Fig. 4 shows the imported model in ANSYS workbench.

Figure 3. CAD model of chassis in PTC Creo 3.0

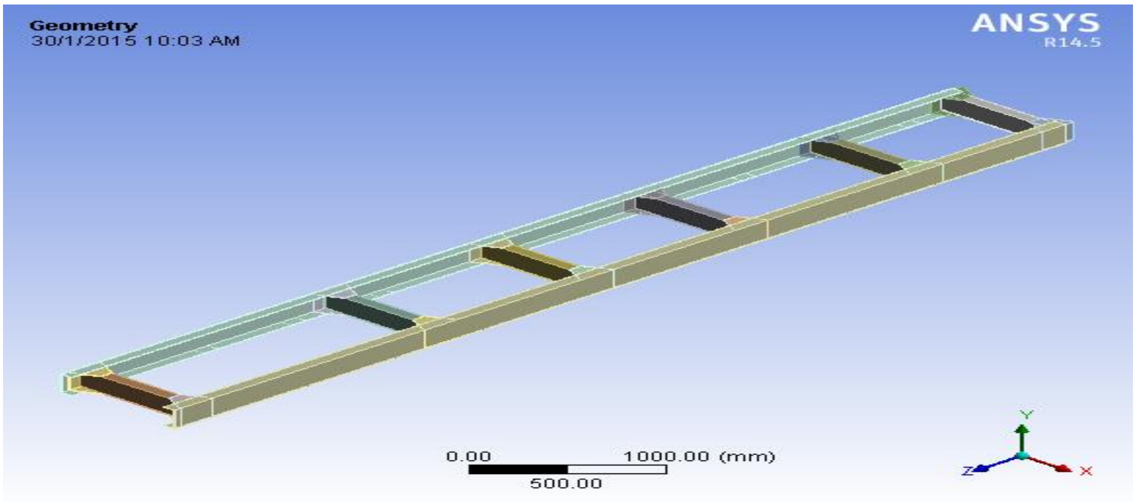

Figure 4. Geometry of chassis frame in ANSYS

5.3 Meshing

FEA software typically uses a CAD representation of the physical model and breaks it down into small pieces called finite "elements" (think of a 3-D puzzle). This process is called "meshing". Higher the quality of the mesh (collection of elements), enhanced the mathematical representation of the physical model. The meshing is done on the model using tetrahedral elements. Fig. 5 and 6 shows tetrahedral element and meshing of model respectively.

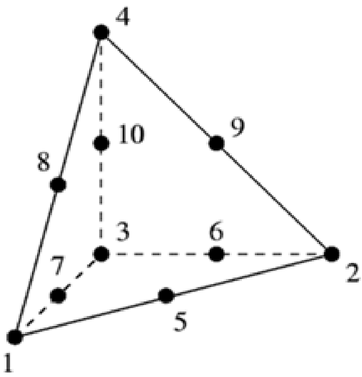

Figure 5. 10 node tetrahedral element

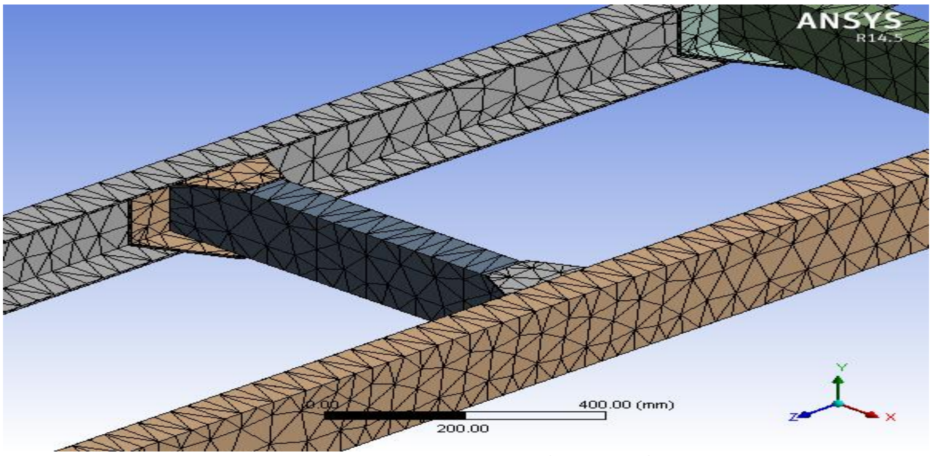

Figure 6. Meshing of chassis frame

5.4 Loading and boundary conditions

The truck chassis model is loaded by static forces from the truck body and cargo.The magnitude of force on the upper side of chassis is 117720 N which is carried by two side bars so load on one side bar is 58860 N. Detail loading of model is shown in Fig. 7. Fix support for analysis purpose are provided at the contact region of leaf spring and frame as shown in Fig. 8.

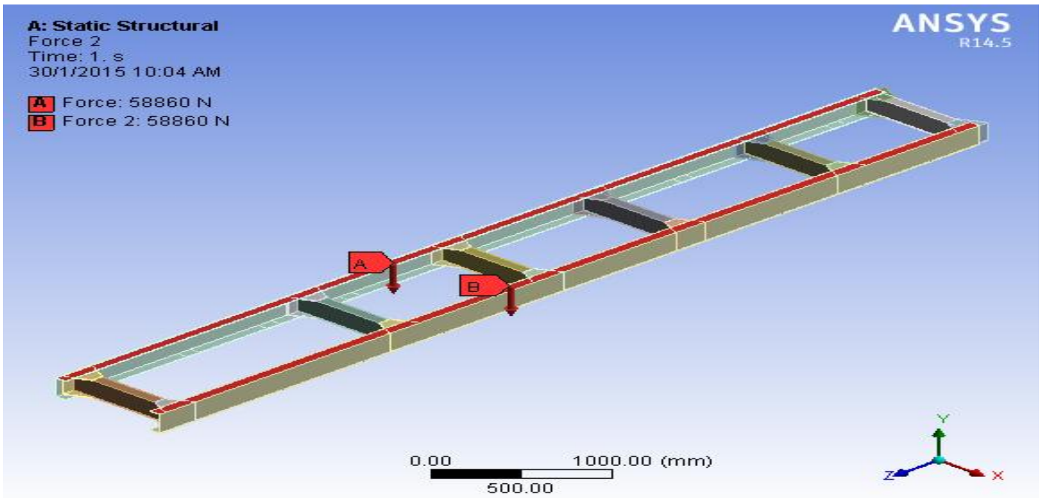

Figure 7. Static load on side bar of chassis frame

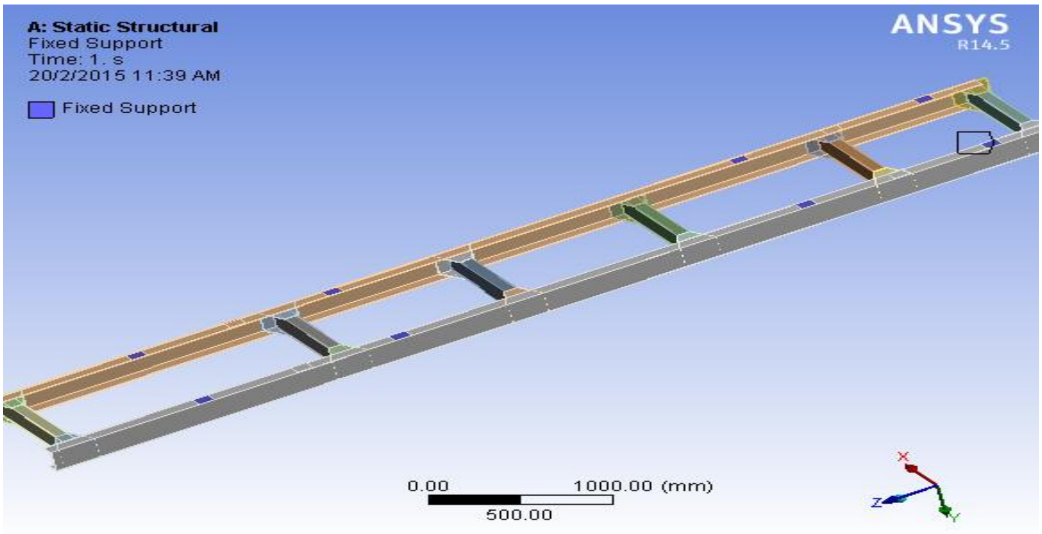

Figure 8. Fixed support of chassis frame

VI. Result of analysis

The analysis done on the chassis model gives the maximum generated von-Mises stress on side bar is 44.49 MPa (Fig 9). The truck chassis can be modified to increase the value of SF especially at critical point area. The permissible value of von-Mises stress for material used is 360/3= 120 MPa (considering factor of safety is 3 for design).

The generated von-Mises stresses are less than the permissible value hence the design is safe. The von-Mises stress and deformation are as shown in Fig. 9 and 10. The weight of the chassis model is 335.14Kg.

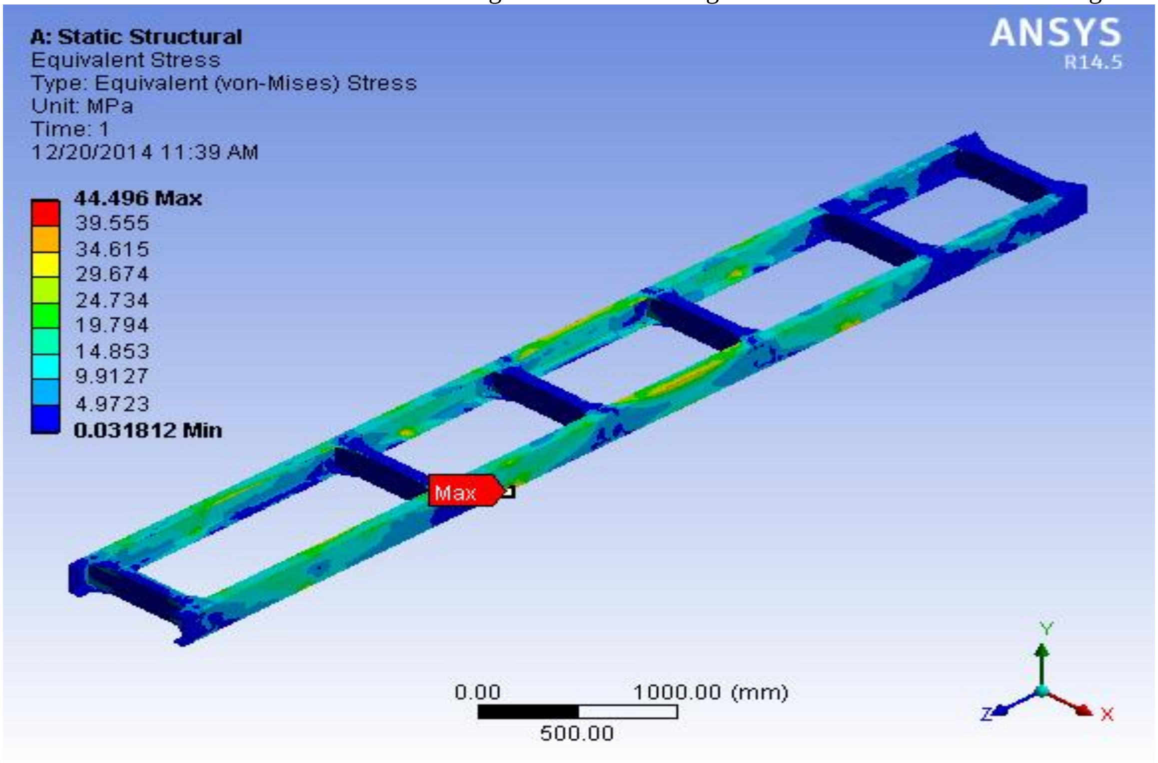

Figure 9. Von Misses stress on chassis frame

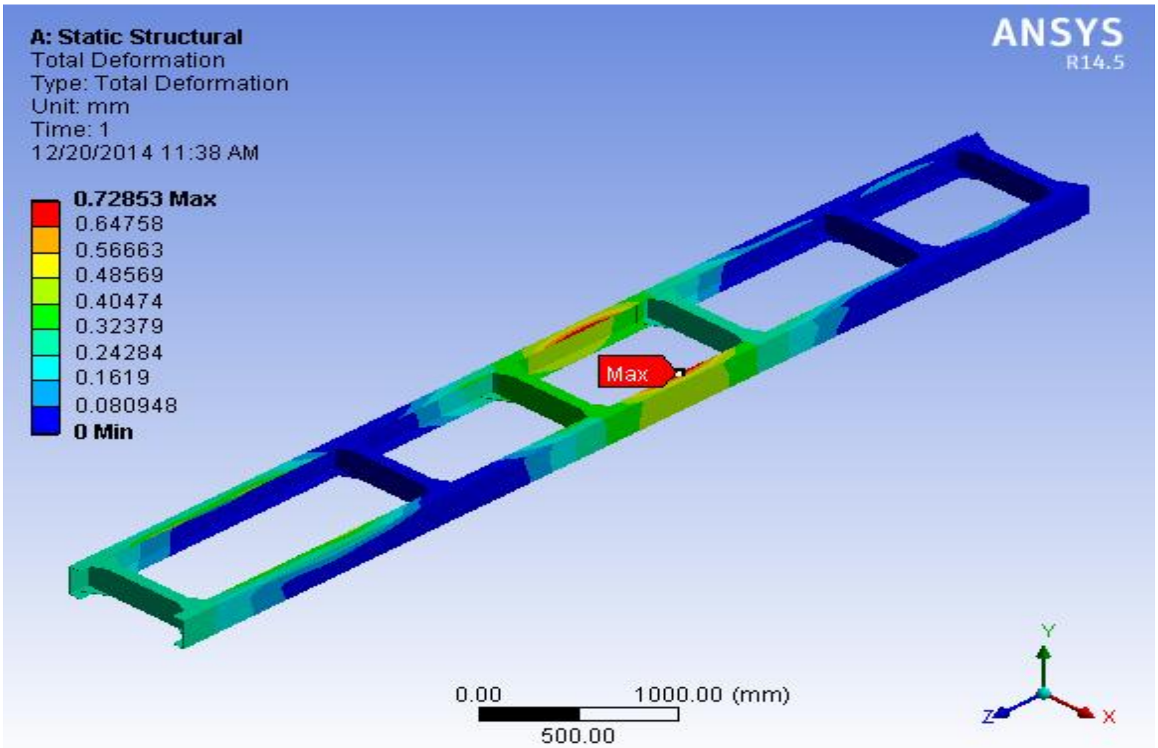

Figure 10. Deformation of chassis frame

Table 2: Results of analysis and calculated results

Criteria	Analysis result of whole chassis frame	Calculated result of sidebar
Von-mises stress	44.49 N/mm^2	55.20 N/mm^2

The stress generated in sidebar is more than that of the whole chassis frame, because of the addition of series of crossbars. This addition of crossbars makes chassis less prone to bending and twisting as well. This variation is caused by simplification of model and uncertainties of numerical calculation.

VII. Conclusion

In this study, the finite element analysis of Eicher 11.10 chassis was carried out. The generated Von-mises stresses are less than the permissible value so the design is safe. The analysis gives maximum equivalent stress and total deformation which are in the desired limit.

References

[1]. Patel, T. M., Bhatt, M. G., & Patel, H. K. (2013). Analysis and validation of Eicher 11.10 chassis frame using Ansys. International Journal of Emerging Trends & Technology in Computer Science,2(2), 85-88.
[2]. Shroff, R. (2002). Structural Optimization of Automotive Chassis (Master's dissertation, Indian Institute of Technology, Bombay).
[3]. Karita, K., Kohiyama, Y., Kobiki, T., Ooshima, K., & Hashimoto, M. (2003). Development of Aluminum Frame for Heavy-Duty Trucks. Technical Review in Japan, 15, 81-84.
[4]. Butdee, S., & Vignat, F. (2008). TRIZ method for light weight bus body structure design. Journal of Achievements in Materials and Manufacturing Engineering, 31(2), 456-462.
[5]. Husaini, M., &Wahab, A. (2009). Stress analysis on truck chassis (Doctoral dissertation,UNIVERSITI MALAYSIA PAHANG).
[6]. Nor, M. A. M., Rashid, H., Mahyuddin, W. M. F. W., Azlan, M. A. M., & Mahmud, J. (2012). Stress Analysis of a Low Loader Chassis. Procedia Engineering, 41, 995-1001.
[7]. Data, D. (2004). PSG Design Data, Coimbatore.
[8]. Chhawsaria (2011). Structural analysis – II. Tech-Max publications, Pune.
[9]. Hutton, D. V., & Wu, J. (2004). Fundamentals of finite element analysis (Vol. 1). New York: McGraw-Hill.
[10]. Cook, R. D. (1995). Finite element modeling for stress analysis (Vol. 8). New York: Wiley.

Appendix

NOMENCLATURES:

DF	Distribution Factor
K	Stiffness Factor
K_r	Relative Stiffness Factor
I	Moment of Inertia
L	Length Span
M_{FBC}	Fixed end moment at BC
w	Uniformly Distributed Load
C.O.	Carry Over
R_{VB1}	Vertical Reaction at B1
σ	Stress
M	Bending Moment
Z	Section Modulus