Design, Analysis& Optimization of Truck chassis- Rail & Cross member

Mr. Jinto Joju Thaikkattil[1], Gayatri Patil[2]

[1]*PGScholar, Department of Mechanical Engg., KJCOEMR, Pune,* **jjt7171@gmail.com**
[2]*Assistant Professor, Department of Mechanical Engg. KJCOEMR, Pune,*
gayatripatil2411@gmail.com

ABSTRACT

A chassis consists of an internal framework that supports a man-made object. It is analogous to an animal's skeleton. An example of a chassis is the under part of a motor vehicle, consisting of the frame (on which the body is mounted) with the wheels and machinery. Finite Element Analysis (FEA) is the most powerful technique for strength calculations of the structures working under known load and boundary conditions.In this paper we are going to do the finite element analysis of the rail and cross members. FEA approach can be applied for the optimization. We are going to do optimization on the rail & cross members of the chassis on the Eicher 407 model. 3D model of rail and cross membersof chassis will be drawn in CATIA V5R19, Meshing will be carried out in Hypermesh, and Ansys will be used for solutions. Optimization will be carried out in iterations.

Keywords-*Rail and Cross member, Finite Element Analysis, Chassis, Optimization*

---

1. INTRODUCTION

The Automobile chassis usually refers to the lower part of the vehicle including the tires, engine, frame, driveline and suspension. Out of these, the frame gives necessary support to the vehicle components placed on it. Also the frame should be so strong to resist impact load, twist, vibrations and other bending stresses. The chassis frame consists of side rails attached with a number of cross members. Along with the strength, an important consideration in the chassis design is to increase the bending stiffness and torsion stiffness. Proper torsional stiffness is required to have good handling characteristics. Commonly the chassis are designed on the basis of strength and stiffness. In the conventional design procedure the design is based on the strength and is then focused to increase the stiffness of the chassis, with very small consideration to the weight of the chassis. This design procedure involves the adding of structural cross member to the existing chassis to improve its torsional stiffness. As a result, weight of the chassis increases. This increase in weight of the chassis fuel efficiency is reduced and increases the overall cost due to extra material. The design of the chassis with proper stiffness and strength is necessary. The design of a vehicle structure is of fundamental importance to the overall vehicle performance. The vehicle structure plays an important role in the reliability of the vehicle. Generally, truck is a heavy motor vehicle which is designed for carrying the attached weights, such as the engine, transmission and suspension as well as the passengers and payload. The major focus in the truck manufacturing industries is to design vehicles with more payload capacity. Using high strength steels than the conventional ones are possible with corresponding increase in payload capacity. The chassis of truck which is the main part of the vehicle that combines the main truck component systems such as the axles, suspension, power train, cab and trailer etc.

Automotive designers need to have complete understanding of various stresses prevalent in different areas of the chassis component. During the conceptual design stage, when changes to the design is easy to implement and have less impact on overall project cost, the weight and structural characteristics are mostly unknown since detailed and overall vehicle information is not available at the early stage . The vehicle design starts up with conceptual studies to define size, number and location of un-driven and drive axles, type of suspension, engine power, transmission, tire size and axle reduction ratio, cab size and auxiliary equipment. The selected configuration has to be more precise and accurate for the considered transportation tasks and should match the existing production line. In general, there are two approaches to analyse truck chassis: one is stress analysis to predict the weak points and the other is fatigue analysis to predict life cycle of the frame. This overview selectively and briefly discusses some of the recent and current developments of the stress analysis of truck chassis. A number of analytical, numerical and experimental methods are kept in mind for the stress analysis of the heavy duty truck frames. Conclusion of the stress analysis in the vehicle chassis has been reported in literature. Finally, the scope of future work has been discussed after concluding on the obtained results.

2. **LITERATURE REVIEW**ANURAG, AMRENDRA KUMAR SINGH, AKASH TRIPATHI, ADITYA PRATAP TIWARI, NITISH UPADHYAY, SHYAM BIHARI LA, "DESIGN AND ANALYSIS OF CHASSIS FRAME", IJRE, VOL. 03 NO. 04, APRIL 2016,

ISSN 2348-7852 (PRINT) | ISSN 2348-7860 (ONLINE), HAS OVERVIEW SELECTIVELY AND BRIEFLY DISCUSSES SOME OF THE RECENT AND CURRENT DEVELOPMENTS OF THE STRESS ANALYSIS OF TRUCK CHASSIS.

S. Dheeraj and R. Sabarish, "Analysis of Truck Chassis Frame Using FEM", Middle-East Journal of Scientific Research 20 (5): 656-661, 2014, ISSN 1990-9233, has made an attempt to find the stresses in a tipper truck frame by analyzing stressconcentration points where the displacement and frequencies are high at the time of loading and unloading.

Mukesh Patil, Rohit Thakare, Aniket Bam. "Analysis of a tanker truck chassis",International Journal on Mechanical Engineering and Robotics, ISSN (Print): 2321-5747, Volume-3, Issue-6, 2015 has presented an analysis of the static stress acting on the upper surface of the truck chassis.

3. CALCULATION OF FORCES

Now for calculation of forces we consider different cases for analysis viz.

1. Case 1 – Gross vehicle weight as UDL
2. Case 2 – Bump force

Table: 1 Material Properties of High Strength Low Alloyed Steel (HSLA)

Material	HSLA Steel
Modulus of Elasticity E	2.6 x 105*N/mm2*
Poisson's Ratio	0.3
Tensile Yield Strength	310 MPa
Tensile Ultimate Strength	448 MPa
Density	7800 kg/m3

Force calculations:

Table: 2 Eicher Truck Specifications:

Sr. No.	**Parameter**	**Value**
1	Gross Vehicle Weight (GVW)	8250 kg
2	Engine Displacement	3298 cc
3	Maximum Power	95 PS @ 3200 rpm
4	Maximum Speed	92 km/hr.
5	Overall Width	2005 mm
6	Overall Height	2340 mm
7	Wheel Base (W_B)	3765 mm
11	CG from front axle (f)	2654 mm
12	CG from rear axle (r)	1096 mm

Case 1 – Gross vehicle weight as UDL:

$(UDL)_{total}$ = GVW x Gravitational acceleration……….. (1)

= 8250 x 9.81

$(UDL)_{total}$= 80932.5 N

For selected element of the rail & cross members, (this is around rear mounting of rear leaf spring).

Length of rail members = 600 mm

Total length of rail members = 5875 mm

$$(UDL)_{selected} = \frac{600}{5875} x\ (UDL)_{total}$$

$$= \frac{600}{5875} x\ 80932.5\ N$$

$(UDL)_{selected}$ = 8265.4468 N ~ <u>8266 N</u>

Case 2 – Bump force:

$$FB = \frac{\mathbf{m\ x\ g\ x\ f}}{\mathbf{WB}} \dots\dots\dots\dots\dots\dots\dots\dots (2)$$

Where,

m = 8250 kg= 18188.14 lb

g = 9.81 m/s^2

f = 2654 mm

W_B = 3765 mm

F_B = 57050.43 N

Now this bump force is acting on complete rear axle. We must calculate force acting on each wheel and then force acting on each leaf spring mounting at rear.

Bump force acting on each rear wheel = F_B / 2 = 28525.215 N

Bump force acting on each leaf spring mounting (rear) = 28525.215 / 2

= 14262.6 N ~ <u>14263 N</u>.

4.GENERATION OF CAD MODEL

Dimensions required for modelling are obtained by taking reading directly from Rail & Cross members. Further CAD model have been prepared in CATIA. And importing those readings, 3D model is prepared as shown in fig.1.

ISSN: 2321-8169
257 – 263

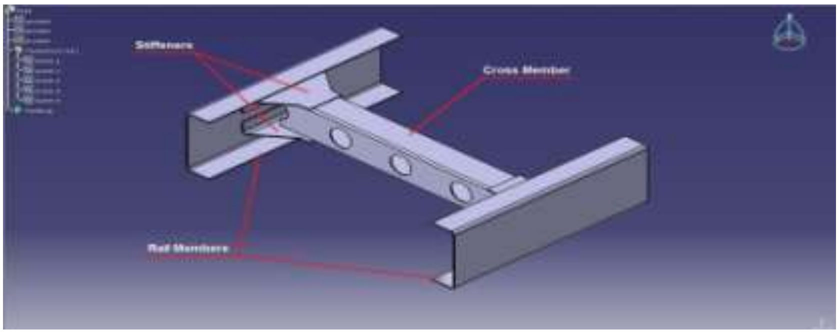

Fig.1- 3D Model of Rail & Cross members

5. FINITE ELEMENT MODELLING OF RAIL & CROSS MEMBER

Initially the .igs file from CATIA is imported to the meshing software Hypermesh. The CAD data of the Rail & Cross member is imported and the surfaces were created and meshed. The model is imported in ANSYS to perform linear analysisand maximum displacement and von-mises stress are obtained.

1. Finite Element analysis of Existing Rail and Cross Member

To perform the FEA of the Existing rail and cross member, continuum (rail and cross member)is discretized into finite number of elements through meshing process and then boundary conditions are applied to the system. Fixed supports are applied to chassis where it comes in contact with the leaf spring systems. Then, theloading is done for two cases

Case 1: Pay load capacity and self weight is collectively applied as uniformly distributed load (UDL) of 8266 N throughout the rail and cross member

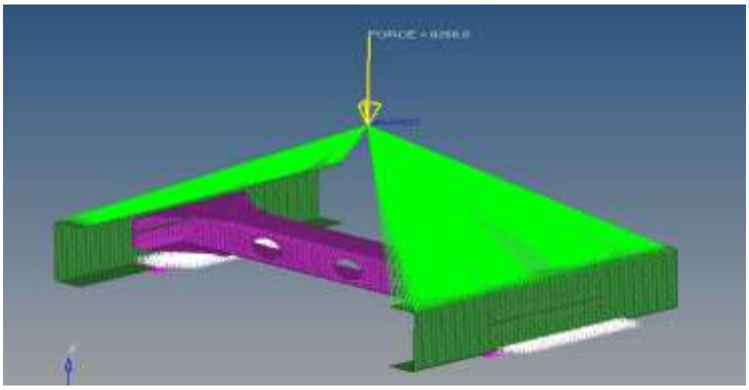

Fig. 2- Applied forces and boundary conditions

Deformation Plot

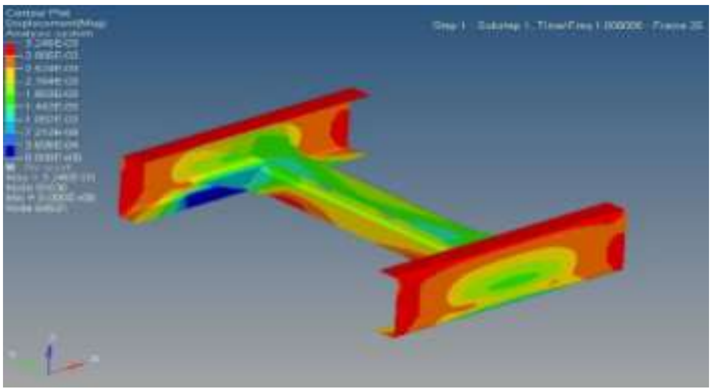

Fig. 3- Maximum displacment of **0.0032 mm** is obsreved which is very less.

Stress plot

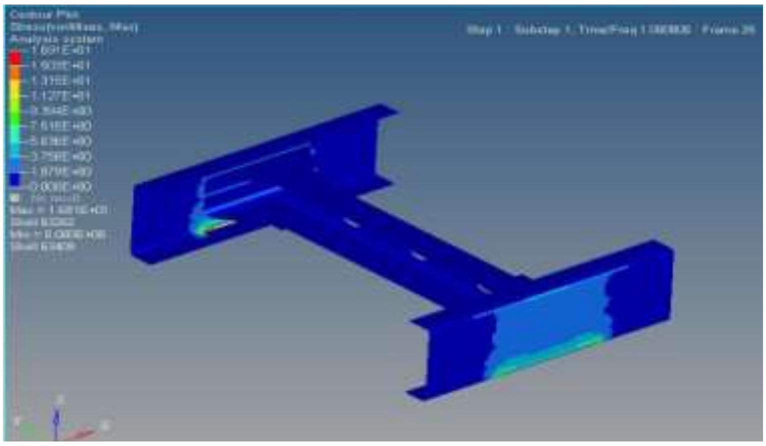

Fig. 4- Maximum Stress of of **16.91 MPa** is obsreved which is very less.

Closure:

The linear static analysis of the existing rail and cross member given the maximum stress of 16.91 MPa which is well below the critical value and maximum deflection of 0.0032 mm . By observing these stress and deflection plots, it can be concluded that design is safe.

Case 2: Considering Maximum dynamic forces of total 14263 N due to bump acting on the rail and cross member.

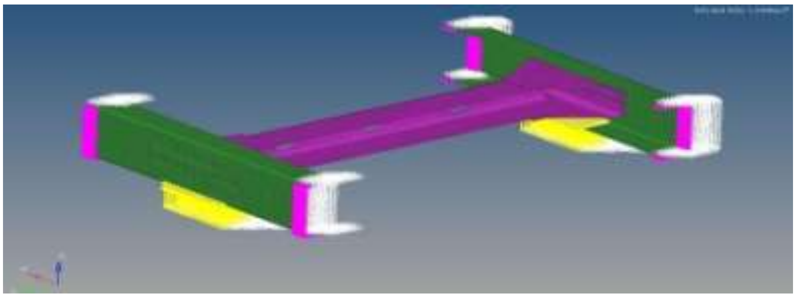

Fig. 5- Applied forces and boundary conditions

Deformation plot

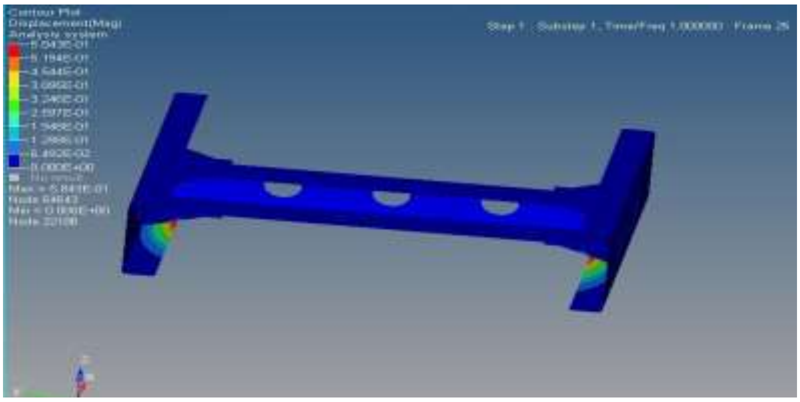

Fig. 6- Maximum displacment of **0.58 mm** is obsreved.

Stress plot

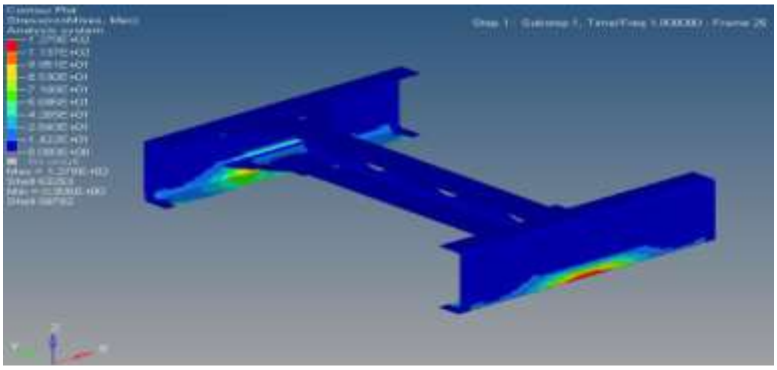

Fig. 7- Maximum Stress of of **127.9 MPa** is obsreved.

Closure:

The finite element analysis of the existing rail and cross member for maximum dynamic forces of total 14263 N due to bump given the maximum stress of 127.9 MPa and maximum deflection of 0.58 mm**.**

By observing these stress and deflection plots of these two cases, it can be concluded that since the obsreved deformation and stresses are less below the critical value of 310MPa of yielding and we have great scope for optimization **of reducing weight of 22.446 kg rail and cross member assembly.**

After observing FEA results of existing rail and cross member and above discuessed optimization techniques. Here an attempt is made to optimize rail and cross member with two iterations considering same loading conditions as in existing rail and cross member

Iterations 1: Aiming for weight optimization, HSLA Steel cross member is replaced with Aluminium

Case 1: Pay load capacity and self-weight is collectively applied as uniformly distributed load (UDL) of 8266 N throughout the rail and cross member

Closure:

The linear static analysis of optimized HSLA Steel cross member replaced with Aluminium cross member given the maximum stress of 16.91 MPa and maximum deflection of 0.0032 mm.

Case 2: Considering Maximum dynamic forces of total 14263 N due to bump acting on the rail and cross member.

Closure:

The finite element analysis of the optimized rail and cross member for maximum dynamic forces of total 14263 N due to bump given the maximum stress of 127.9 MPa and maximum deflection of 0.58 mm.

the maximum stress and deformation plots of HSLA Steel cross member replaced with Aluminium cross member which are same as in case of existing chassis **but weight of rail and cross member assembly has considerably reduced to 19.53 kg** because of Aluminium.

Iterations 2: Topology optimization of HSLA Steel rail member with Aluminium cross member aiming optimum material layout according to the design space and loading case

Case 1: Pay load capacity and self-weight is collectively applied as uniformly distributed load (UDL) of 8266 N throughout the rail and cross member

Closure:

The linear static analysis of topology optimized HSLA Steel cross member replaced with Aluminium cross member given the maximum stress of 21.44 MPa and maximum deflection of 0.0055 mm.

Case 2: Considering Maximum dynamic forces of total 14263 N due to bump acting on the rail and cross member.

Closure:

The finite element analysis of the topology optimized rail and cross member for maximum dynamic forces of total 14263 N due to bump given the maximum stress of 136.9 MPa and maximum deflection of 0.72 mm.

The maximum stress and deformation plots of topology optimized HSLA Steel cross member replaced with Aluminium cross member are little high compared to all the cases but are very less below the critical vale of 310 MPa yielding. **Weight of topology optimized rail and cross member assembly has further reduced to 19.03 kg**.

6. CONCLUSION (COMPARISON OF RESULTS):

Case	Case 1		Case 2		
	Total Deformation	**Maximum Stress**	**Total Deformation**	**Maximum Stress**	**Weight of rail and cross assembly**
Existing Model	0.0032 mm	16.91 MPa	0.58 mm	127.9 MPa	22.446 kg
Optimized model (Iterations 1)	0.0032 mm	16.91 MPa	0.58 mm	127.9 MPa	**19.53kg**
Optimized model (Iterations 2)	0.0055 mm	21.44 MPa	0.72 mm	136.9 MPa	**19.03kg**

Table: 3 Comparison of Results

7. REFERENCES

[1]. Anurag, Amrendra Kumar Singh, Akash Tripathi, Aditya Pratap Tiwari, Nitish Upadhyay, Shyam Bihari La, "DESIGN AND ANALYSIS OF CHASSIS FRAME", IJRE, Vol. 03 No. 04, April 2016, ISSN 2348-7852 (Print) | ISSN 2348-7860 (Online).
[2]. S. Dheeraj and R. Sabarish, "Analysis of Truck Chassis Frame Using FEM", Middle-East Journal of Scientific Research 20 (5): 656-661, 2014, ISSN 1990-9233.
[3]. Mukesh Patil, Rohit Thakare, Aniket Bam. "Analysis of a tanker truck chassis", International Journal on Mechanical Engineering and Robotics, ISSN (Print): 2321-5747, Volume-3, Issue-6,2015.
[4]. KiranGhodvinde, S. R.Wankhade, "Structural Stress Analysis of an Automotive Vehicle Chassis", International Journal on Mechanical Engineering and Robotics, ISSN (Print) : 2321-5747, Volume-2, Issue-6,2014.
[5]. Ahmad O. Moaaz and Nouby M. Ghazaly, "FINITE ELEMENT STRESS ANALYSIS OF TRUCK CHASSIS USING ANSYS: REVIEW", International Journal of Advances in Engineering & Technology, Nov., 2014, ISSN: 22311963.