JETIR.ORG ISSN: 2349-5162 | ESTD Year : 2014 | Monthly Issue
JOURNAL OF EMERGING TECHNOLOGIES AND INNOVATIVE RESEARCH (JETIR)
An International Scholarly Open Access, Peer-reviewed, Refereed Journal

STUDY OF DYNAMIC LOAD BEHAVIOUR AND ROLLOVER RESISTANCE OF TRUCK CHASSIS AND OPTIMIZATION

[1]K.SUDHAKAR, [2]Prof. K.R.JAGTAP, [3]Asst.Prof. DILEEP.N.KORADE

[1,2,3]Sinhgad Institute of Technology and Science. Pune, Maharashtra India

Abstract: Chassis is the main assembly in the truck body. Many other assemblies in a vehicle, like Engine, Suspension, Exhaust, Propeller shaft, Battery, Fuel Tank and Urea Tank etc, are mounted on Chassis. For that reason, the Stiffness and the strength are the basis for the truck chassis. Bending stiffness and torsion stiffness are the major parameters to consider in the design of Truck Chassis. Insufficient stiffness causes NVH, Ride Handling, Safety and Reliability problems. The design of the chassis with sufficient stiffness and strength is mandatory. The overall performance of a vehicle structure is completely depending on proper design considerations. The present work includes optimization of truck chassis with multiple iterations using Finite Element Analysis approach for different static and dynamic load cases on truck chassis and reduction in weight through topology optimization without affecting the chassis performance. Followed by fabrication of truck chassis and experimental validation with FEA results.

Index terms: Rollover resistance of truck chassis, Optimization, Cornering load, Hypermesh, Nastran.

I. INTRODUCTION

Chassis is the main supporting structure of an automobile vehicle. The main body is easily bolted to the chassis during manufacturing process is called Body on Frame structure. Most of the heavy duty vehicles and commercial vehicles like trucks and buses are using this process only. The vehicle chassis is a frame on which the engine, axle assemblies, wheels, steering mechanism, transmission, brakes, suspension members are mounted. So, the frame should have sufficient strength to sustain impact loads, torsional loads, rollover loads, vibrations and other bending stresses.

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.

In general, there are two approaches to analyze 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.

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.

Fig 1: Truck chassis at laboratory site

II. LITERATURE REVIEW

Kiran Ghodvinde and S.R.Wankhade [1] focused on the static analysis of two different frame automotive chassis, the Chevy truck chassis shows the critical stress at the joints and it is being reduce by increasing the side member thickness, connection plate thickness and connection plate length were varied. Numerical results showed that stresses on the side member can be reduced by increasing the side member thickness locally. If the thickness change is not possible, increasing the connection plate length may be a good alternative to improve the strength. The method used is numerical analysis is finite element technique to find the critical stress. In this dissertation work, analyzed the monologue and ladder frame for static load condition with the stress, deflection bending moment and even the analysis of two different chassis with same as discuss above frame are being analyzed, i.e. the kit car chassis, this is validated with the other analysis details, and the other one is Chevy truck chassis.

Ahmad O. Moaaz and Nouby M. Ghazaly [2] focused on stress analysis of the truck chassis using finite element package ANSYS. The results of reading this paper will give the researcher a summary of some recent and current developments in the field of vehicle design using finite element stress analysis. Finite element stress analysis of a heavy truck chassis plays an important role during design stages. The review of some of the previously conducted work related to vehicle structural design, analysis and optimization using Ansys software is surveyed. It is found that the chassis analysis mainly consists of stress analysis to predict the weak points and fatigue analysis to predict the life of the chassis. Several state of the art papers and even books on chassis stress analysis have been presented in the recently years. This study makes a case for further investigation on the design of truck chassis using FEA Ansys software.

Naveen Ala et al. [3] observed the chassis frame forms the backbone of a heavy vehicle, its principle function is to safely carry the maximum load for all designed operating conditions. This paper describes design and analysis of heavy vehicle chassis. Weight reduction is now the main issue in automobile industries. In the present work, the dimensions of an existing heavy vehicle chassis of a TATA 2515EX vehicle is taken for modeling and analysis of a heavy vehicle chassis with three different composite materials namely, Carbon Steel, HSLA Steel, Structural Steel, High Strength Steel, Alloy Steel, ASTM A710 Steel, Aluminium 6061T6, AISI4130 Alloy subjected to the same pressure as that of a steel chassis. The design constraints were stresses and deflections. All the heavy vehicle chassis have been modeled by considering three different cross-sections, namely C, I and Box type cross sections. For validation the design is done by applying the vertical loads acting on the horizontal different cross sections. Software is used in this work SolidWorks for modelling, ANSYS Workbench 15 for analysis.

Monika S. Agrawal [4] rectified problems associated with structures of a commercial vehicle such as strength, stiffness and fatigue properties along with stress, bending moment and vibrations. 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 aim of this study can be achieved by static and dynamic analysis, combining existing theoretical knowledge and advanced analytical methods. Design of a Chassis is carried by using CATIA. And finite element analysis carried out by using ANSYS.

Nouby M. Ghazaly [5] selectively and briefly discusses some of recent developments on stress analysis of the truck chassis using four finite element packages namely; ABAQUS, ANSYS, NASTRAN and HYPERVIEW. It is observed that most common basic FEA packages are suitable for this analysis. Also, it is found that Most of the existing researchers utilized ABAQUS and ANSYS software to predict stress analysis of the chassis. On the other hand, a few studies using NASTRAN and HYPERVIEW are conducted.

GoollaMurali et al. [6] achieved in the area of improvement of torsion stiffness based on the result gained from the finite element analysis, further enhancement of the current chassis had been done through the chassis FE model in order to improve its torsional stiffness as well as reduce the vibration level. Series of modifications and tests were conducted by adding the stiffener in order to strengthen and improved the chassis stiffness as well as the overall chassis performances.

III. PROBLEM STATEMENT

Chassis behavior and resistance plays a very important role in overall vehicle rigidity and strength, the more optimized the chassis, the less prone to failure due to uneven loading conditions, hence less impact of connecting members in sprung and unsprung mass, moreover an optimized chassis with less weight will increase the power to weight ratio which is desired by every automotive industry.

IV. OBJECTIVES

- Objective of this project is to validate the truck chassis for different load cases like Gravity load, Cornering (Roll over load) and bump load.
- Iterative approach to find the best optimized model.
- Reduction of weight at least by 15%.
- Validation of the results with less variation.

Table 1: Load Types

S.No	Type of Load	Magnitude of load (KN)	Applied Direction	Point of load
1	Gravity+ Cornering	78.9+22.5	Negative Z dir + Lateral	CG
2	Gravity+ Cornering + Bump	78.9+22.5+9.0	Negative Z dir + Lateral + Vertical	CG

V. GEOMETRY

Design and Analysis of truck chassis of dissertation includes design and analysis of existing truck chassis of TATA LPT 813 Cargo Truck. Dimensions of the existing truck chassis have been measured and CAD model of a truck chassis have been prepared in CATIA

V5 R20. The finite element analysis is carried out by using HYPERMESH and NASTRAN.

Figure 2: Dimensions measurement

VI. CAD Model of Truck Chassis:

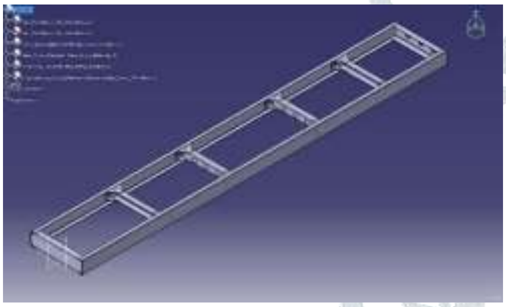

Figure 3: Isometric view of chassis

VII. Finite Element Analysis:

Meshing:
Initially the step file is imported to the meshing software like Hypermesh. The CAD data of the chassis is imported and the surfaces were created and meshed. As all the dimensions of blade are not comparable the element for meshing the chassis is shell element (CQUAD4 & CTRIA3) (2D Mesh Element Type).

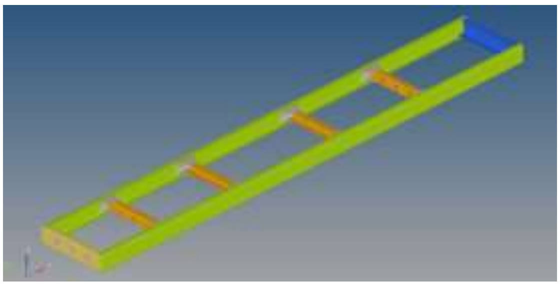

Figure 4: Meshing of Chassis in Hypermesh

Table 2: Meshing Details

Element Type	**CQUAD4 & CTRIA3 (2D element)**
Number of Nodes	**57116**
Number of Elements	**60898**

Table 3: Material Properties

Material	Modulus of elasticity	Density	Poisson ratio
ASTM Low Alloy Steel A 710 C	**207 GPa**	**7.8e^-6 kg/mm^3**	**0.3**

Force calculations:

TATA LPT 813 Specifications:

- Gross Vehicle Weight (GVW) = 8050 kg
- Wheel base = 3800 mm
- Overall width = 2155 mm
- Overall Height = 2341 mm
- Overall length = 6860 mm
- Load body width = 2155 mm
- Load body height = 1590 mm
- Engine Capacity = 5675 cc
- Maximum Power = 95 PS @ 2400 rpm
- Maximum Speed = 92 km/hr
- Location of CG: (from CATIA)
  - CG from front axle = b = 2654 mm
  - CG from rear axle = c = 1096 mm
  - CG height from ground = h = 1755 mm

Now for calculation of forces consider different cases for analysis.

1. Case 1 – Gross vehicle weight as UDL
2. Case 2 – Bump force
3. Case 3 – Torsional force due one side bump
4. case 4 - roll over bending moment

Case 1 – Gross vehicle weight as UDL:

UDL = GVW x Gravitational acceleration………. (1)
= 8050 x 9.81
UDL = <u>78970.5 N</u>

Case 2 – Bump force:

$$FB = \frac{3}{2} \text{ x } \left(\frac{\text{W x c}}{\text{L}} + \frac{\text{m x a x h}}{\text{L}}\right) \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (2)$$

Where,

W = 78970.5 N
c = 1096 mm
m = 8050 kg
h = 1755 mm
L = 3800 mm
a = average acceleration

To calculate acceleration, we have

$$a = 550 \text{ x} \left(\frac{g}{V} \text{x} \frac{HP}{m}\right) ft/s2 \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (3)$$

Where,

$g = 9.81\ m/s^2 = 32.2\ ft/s^2$
V = max. speed = 92 km/hr = 83.85 ft/s
HP = 95 PS = 93.7 HP
m = 8050 kg = 17747.21 lb

Putting above values in equation (3), we get,
$a = 1.115\ ft/s^2 = 0.339\ m/s^2$

Now substituting required values in equation (2), we get,

F_B = 36053.8 N ~ 36054 N

Now this bump force is acting on complete front axle.

Calculate force acting on each wheel and then force acting on each leaf spring mounting.
Bump force acting on each front wheel = F_B / 2 = 18027 N …………. (4)
Bump force acting on each leaf spring mounting = 18702 / 2 = 9013 N.

Case 3 – Torsional force due to one side bump:
For torsional case of the chassis, the bump force will be acting on one side only.
Hence from equation (4),

Bump force acting on one front wheel (say left) = 18027 N
This force will act in vertically upwards direction on left front wheel and same magnitude force will act on right front wheel but in vertically downwards direction.
Hence,

Torsional force acting on chassis = 18027 N

Torsional force acting on each leaf spring mounting = 18027 / 2 = 9013 N

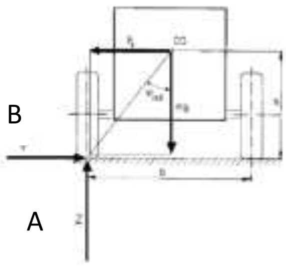

Fig 5: Rolling calculations

taking moment about point A

Z is the reaction force which is W/4= 78970/4 = 19742 N
mg at CG id 78970.5 N
Fy is centrifugal force which is 1/2 x m x r^2
h = 1755mm
b/2 = 2005/2 = 1002.5 mm
putting in equilibrium
mg x b/2 + Fy x h = Z x b
79167926.25 + Fy x 1755 = 39582710
Fy = 22555 N

Finite Element analysis of Existing Chassis:

The calculated loads with practical boundary conditions are applied in HYPERMESH and NASTRAN is used to solve the FEA problem and the results are post processed in HYPERVIEW.

Load Case 1: Roll over bending moment along with Existing UDL

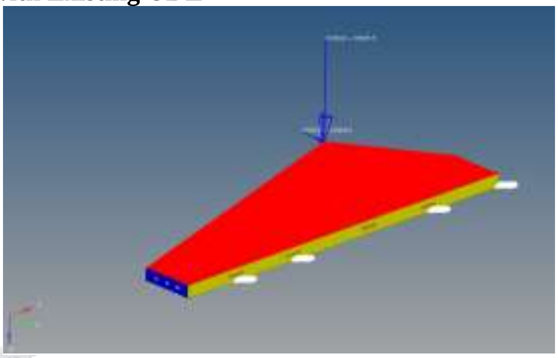

Fig 6: Applied forces and boundary conditions

Deformation plot

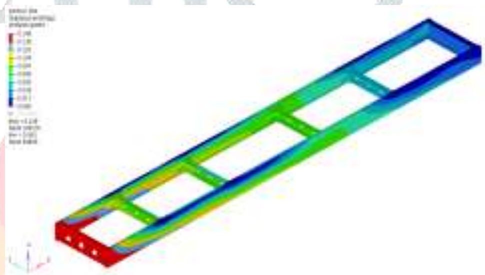

Fig 7: Displacement plot

Stress plot

Fig 8: Stress plot

Maximum displacment of **0.156 mm and** Maximum stress **of 118.20 Mpa** is observed.

Load Case 2: Roll over bending moment along with UDL and Bump load

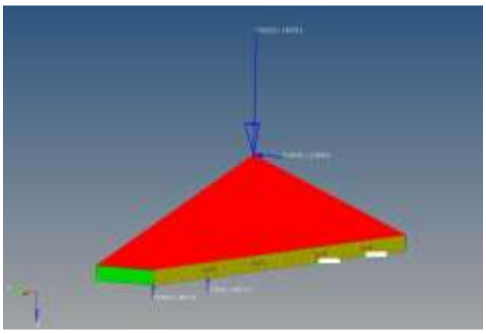

Fig 9: Force and Boundary conditions

Deformation plot

Fig 10: Displacement plot

Stress plot

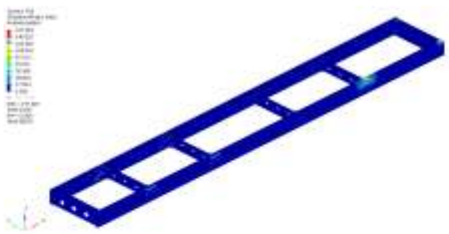

Fig 11: Stress plot

Maximum displacment of **0.562 mm and** Maximum stress **of 157.52 Mpa** is observed.

Stresses in both the load cases are below the Yield limit i.e 550MPa.

OPTIMIZATION OF TRUCK CHASSIS

Iterative approach is used to optimize the chassis to make it lighter without compromising on its strength. And after doing multiple iterations, following design is finalized because of its performance and significant weight reduction.

For Roll over bending moment & UDL

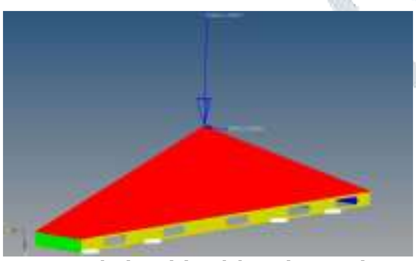

Fig 12: Meshed model with boundary conditions.

Deformation Plot

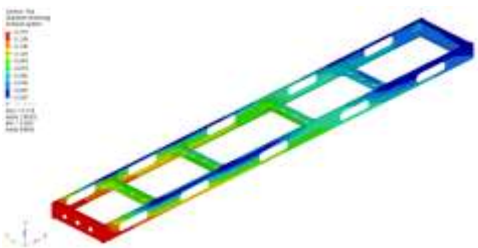

Fig 13: Displacement plot

Stress Plot

Fig 14: Stress plot

Maximum displacement of **0.179 mm and** Maximum stress **of 120.381 Mpa** is observed.

Roll over bending moment & UDL and Bump Load case

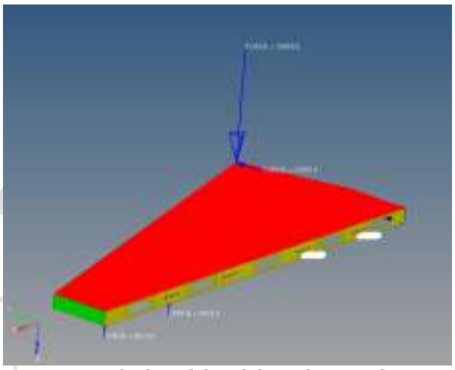

Fig 15: Meshed model with boundary conditions.

Deformation Plot

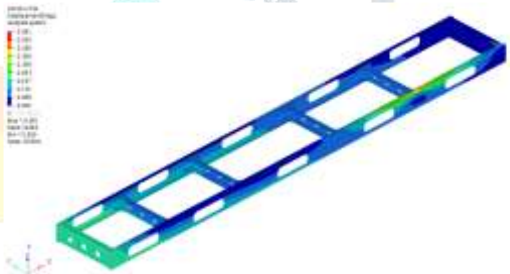

Fig 16: Displacement plot

Stress Plot

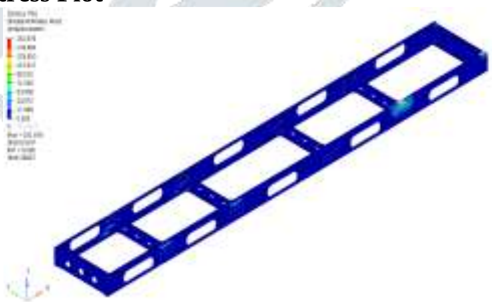

Fig 17: Stress plot

Maximum displacment of **0.591 mm and** Maximum stress **of 161.87 Mpa** is observed.

So we will stop here because further increase in optimization increases stress value near to critical limit. So iteration process is stop here. And this model is selected for manufacturing and testing.

Table 4: Comparison for Stress and Deflection

	UDL + Roll over		
Sr. No.	Von mises stress in Mpa	Deformation in mm	Weight in kg
Existing	118.20	0.156	193.78
Iteration 1	119.42	0.169	182.49
Iteration 2	119.81	0.173	173.54
Iteration 3	120.381	0.179	162.11
	UDL + Roll over + Bump Load		
Sr. No.	Von mises stress in Mpa	Deformation in mm	Weight in kg
Existing	157.52	0.562	193.78
Iteration 1	159.90	0.582	182.49
Iteration 2	160.69	0.586	173.54
Iteration 3	161.87	0.591	162.11

Percentage weight reduction =
(Existing weight – final weight) / Existing weight
= (193.78 – 162.112) / 193.78
= 19.53 %

Table 5: Comparison for modal analysis

Sr. No.	Mode	Frequency (Hz)	Frequency (Hz)
		Existing Truck chassis	Optimized truck chassis
1	1	116.21	112.68
2	2	123.15	123.15
3	3	126.99	126.74
4	4	128.88	128.83
5	5	135.89	135.88
6	6	150.85	145.71

Fabrication and testing

Fabrication

The optimized chassis is fabricated with the scale of 1:4 using ASTM A710 C Alloy Steel material for testing purpose.

Experimental validation

The practical testing is conducted on scaled down model on universal testing machine. Compression loads are applied on the model. The actual vehicle loads are recreated on the fabricated chassis using UTS machine to get the results same as practical conditions. As per scaled down model, the loads are also reduced by one fourth. And maximum effect is produced from Rollover force and Vertical force. Hence vertical force of 19743 N i.e. one fourth of actual load 78970.5 N is applied.

Fig 18: Fabricated Chassis

Fig 19: Chassis Testing

Fig 20: Chassis Testing

Results:

FEA deformation: **0.591 mm**
Experimental deformation: **0.56 mm**

Percentage weight reduction = (FEA deformation – Experimental deformation) / FEA deformation
= (0.591 – 0.56) / 0.56
= 5.2 %

Percentage variation: 5.2%

CONCLUSION

- The comparison between FEA results with Experimental validation, has been performed and observed **5.2%** variation in the results.
- The comparison between modal analysis results of optimized model with existing model shows that the frequencies of vibration of optimized truck chassis in six different modes are lower than that existing truck chassis.
- The percentage weight reduction of **19.53 %** is obtained after the weight optimization of truck chassis without compromising strength of truck chassis.

References:

[1] 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.

[2] Ahmad O. Moaaz and Nouby M. Ghazaly, “Finite Element Stress Analysis Of Truck Chassis Using Ansys: Review”, International

[3] Journal of Advances in Engineering & Technology, Nov., 2014, ISSN: 22311963.

[4] Naveen Ala, K.Tejdeep Reddy , BVSS Bharadwaja, “Static Analysis of Truck Chassis using different materials”, International Journal of Innovative Research in Science, Engineering and Technology, Vol. 5, Issue 1, January 2016, ISSN(Online): 2319-8753, ISSN (Print): 2347-6710.

[5] Monika S. Agrawal, “Finite Element Analysis of Truck Chassis Frame”, International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056 Volume: 02 Issue: 03 | June-2015, p-ISSN: 2395-0072.

[6] 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.

[7] GoollaMurali, Subramanyam.B, Dulam Naveen, “Design Improvement of a Truck Chassis based on Thickness”, Altair Technology Conference, 2013, India.

[8] Anurag, Amrendra Kumar Singh, AkashTripathi, AdityaPratapTiwari, Nitishupadhyay, ShyamBihari La, “DESIGN AND ANALYSIS OF CHASSIS FRAME”, IJRE, Vol. 03 No. 04, April 2016, ISSN 2348-7852 (Print) | ISSN 2348-7860 (Online).

[9] S. Dheeraj and R. Sabarish, “Analysis of Truck Chasis Frame Using FEM”, Middle-East Journal of Scientific Research 20 (5): 656-661, 2014, ISSN 1990-9233.

[10] MukeshPatil, RohitThakare, Aniket Bam. “Analysis of a tanker truck chassis”, International Journal on Mechanical Engineering and Robotics, ISSN (Print): 2321-5747, Volume-3, Issue-6,2015.

[11] Hirak Patel, Khushbu C. Panchal, Chetan S. Jadav, “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 2013.

[12] SandipGodse, Prof. D.A.Patel, “Static Load Analysis Of Tata Ace Ex Chassis And Stress Optimization Using Reinforcement Technique”,

International Journal of Engineering Trends and Technology (IJETT) - Volume4 Issue7- July 2013, ISSN: 2231-5381.