

Stability Analysis of Chassis Structure in Heavy Load Carrying Vehicles

Siddaram Biradar¹, Vinayak Manur², Dr: S.N.Kurbet³

P.G. Student, Department of Mechanical Engineering, Basaveshwara Engineering College Bagalkot, India^{1,2}.

Professor and Head, Department of Mechanical Engineering, Basaveshwara Engineering College, Bagalkot, Karnataka, India³

ABSTRACT: The chassis is backbone of a heavy load carrying vehicles. Its function is to carry the maximum load for all the designed operating conditions and to support the components placed upon it. The analysis is carried out to improve the structural stability of chassis in heavy load carrying vehicles and to reduce the weight of the chassis without compromising factor of safety. The chassis frame is designed such that the stresses developed in critical areas are well below the yield strength of the material. In this study the heavy load carrying vehicles chassis is analysed for three different worst loading cases such as bumping load case, braking load case and turning load case.

KEYWORDS: Heavy load vehicle chassis, FEM, stress analysis, hypermesh.

I. INTRODUCTION

The chassis is a structural backbone for a heavy load carrying vehicles. Its function is to carry the maximum load for all designed operating conditions and to support the components like engine, tires, axle assemblies, steering, brakes etc. The chassis frames are basically made by steel material and also the aluminium is another raw material that has been highly used for manufacturing the chassis frames. The chassis is subjected to various stress distribution and displacement under various loading condition when vehicle travels along the road. The chassis is considered to be most significant component that gives strength and stability to the vehicle under different operating conditions. Along with the strength the most important consideration in chassis design is to increase the torsion stiffness and bending stiffness.

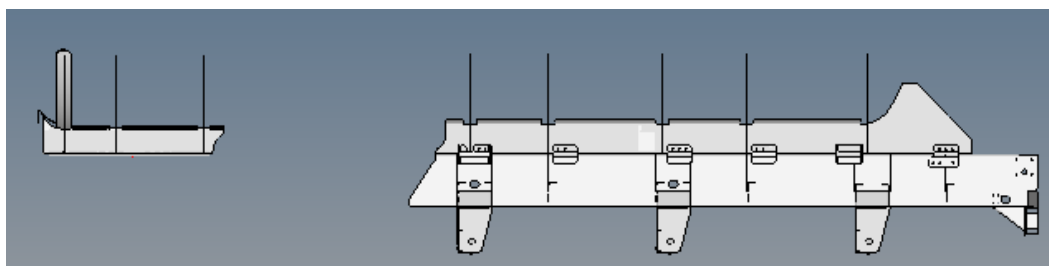


Fig.1.1 Storage tank chassis frame

Commonly chassis frame consists of two side members that are separated apart by cross members. The cross members are used at high stress regions connecting to the side members. Chassis weight can be reduced by minimizing the side member thickness and also having series of holes along the chassis frame where the load is not so high. Increasing the two channel shaped side members thickness can reduce stresses at joint areas, but the overall weight of the frame increases, to overcome this problem using local plates in the joint areas can also increase side member thickness, hence the excessive weight of the chassis frame is prevented. Increasing side member's thickness, adding

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: www.ijirset.com

Vol. 6, Issue 6, June 2017

stiffeners and C channels at maximum stress region of the chassis frame, the present payload of the chassis can be improved.

II. LITERATURE SURVEY

Cicek Karaoglu and N. SefaKuralay (2001): In this study, chassis analysis was carried out by finite element analysis. This analysis showed that increasing the two channel shaped side members thickness can reduce stresses at joint areas, but it is important to realize that the overall weight of the chassis increases. Hence without increasing the side member thickness, providing local plates at joint areas is a good alternative.

Sairam Kothari (2002): Determined that, with limited modifications like increase in thickness, adding stiffeners and 'c' channel at maximum stress region of chassis frame, present payload of the chassis can be improved.

Ahmad O. Moaaz, Nouby M. Ghazaly (2014): Focused on design of truck chassis with more payload capacity and possible less weight and fatigue analysis of chassis frame. This study found that the chassis analysis consists of stress analysis to predict weak point and fatigue analysis to improve the life of the chassis.

Dr: R. Rajappan(2013): Did the finite element analysis to study the static and dynamic load characteristics of chassis. This study found that certain location of high stress area analysing vibration. The modification of truck chassis has been proposed to reduce the vibration.

Divyesh N. Chaudhari et al. (2015): Studied different techniques to reduce stresses on heavy load carrying vehicles chassis frame, finite element method plays a better role in this analysis. Truck chassis is subjected to static, dynamic and also cyclic loading. Hence analysis of chassis frame is very difficult to avoid failure. Study showed that strength of the chassis frame is increased by changing the thickness of web and flange & also providing stiffeners in 'c' channel section.

K. Rajasekar, Dr.R. Saravanan (2014): This study showed that the loads generated by other components of the vehicle are transferred to chassis only. Chassis frame has to be strong enough to withstand all loads of static and dynamic conditions. To minimize the failure in the chassis structure and to ensure greater safety, chassis structure has to be designed such that, variable loads along the length of the vehicle are transferred to variable section of chassis structure.

Kiran Ghodvinde, S. R. Wankhade (2014): The aim of this study is to increase the structural stiffness of chassis frame. Study shows that stresses are high at critical areas, so it is necessary to minimize the stress at joint areas. The maximum deformation takes place at side members when subjected to worst loading conditions. To remove stresses at joint areas and to increase stiffness of the side members it is necessary to provide C member at joint areas. The C frame should be thicker and design will be more durable.

M. Ravi Chandra et al. (2012): This study found that instead of using steel as material for chassis design using polymer composite material reduce chassis weight and also withstand same pressure as that of steel. The polymer composite material is more economic and lighter than the steel material.

Objectives:

- To study the structural stability of chassis in heavy load carrying vehicles.
- To study and compare the stresses and strains and deflection of the chassis during braking, bumping and turning load cases.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: www.ijirset.com

Vol. 6, Issue 6, June 2017

III.METHODOLOGY

The concept of designing chassis rails is based on the theory of deflection of simply supported beam. But in current project the chassis dimensions are considered from the existing reference model. The chassis geometry is modelled in the CAD generating software like solid edge, pro-e, solid works. The CAD model generated is imported as a .step file. The .step is in a neutral file format and can be imported into any other third party software like hypermesh.

Step-1: Finite element model creation: Once the geometry are imported the discretization of the structure is carried out, followed by mid surface extraction. Automesh command is used for meshing with a suitable global element size.

Step-2: Quality check for a FE model: The following element quality checks are done on a 2-D quad and tria element, like Warpage, aspect ratio, Jacobian, minimum length, maximum length, duplicates, minimum and maximum angle.

Step-3: Assigning material properties and section properties: The material used is S355MPa grade with the following properties.

$$E=2.1e^5 \text{ MPa} \quad \rho = 7.85e^{-9} \quad \text{Poisson's ratio } (\nu) = 0.3$$

Step-4: Loads and boundary conditions: The boundary conditions are applied at the centre point of the wheel hub. The centre points of the wheel are constrained in all six degrees of freedom and the front pivot axel is constrained in 3-translation degrees of freedom and 2-rotational degrees of freedom. The rotation about the vertical axis of a vehicle is free to rotate. The load is external load which comes from fluid that stored in the tank.

Step-5: Preparation of abaqus run file (.inp file): The static analysis records are used to define the load cases. The following load cases are defined.

Case1: When the vehicle is de-accelerated (breaking).

Case2: When the vehicle moves on bump.

Case3: When the vehicle takes the turn.

The deck file (.inp file) is imported to submit the run in the abaqus solver. The .inp file is executed (solved) by using following command.

“Abaqus job = chassis .inp”.

Step-6: Post processing: The results from the abaqus will be in the binary format and hence they are imported into abaqus viewer to visualize the stress and the displacement which will help to decide the factors of safety for the given design.

IV.RESULTS AND DISCUSSIONS

The chassis analysis is carried out using the hypermesh and abaqus FEA softwares. Hypermesh is used for the FE modelling and Abaqus is used as solver. The analysis is carried out for three different worst load cases that occur during the moving of the vehicle. The load cases are as follows:

1. Bumping load case: This condition occurs when the vehicle moves over the ditch or bump. 2G static load is applied on the structure in vertical direction.
2. Breaking load case: This condition occurs when the vehicle suddenly comes to rest position. 2.5G static load is applied along the axis of the chassis structure.
3. Turning load case: This condition occurs when the vehicle is taking turn at bend roads. 1.5G load in transverse and 1G load in vertical direction are applied on the chassis structure.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: www.ijirset.com

Vol. 6, Issue 6, June 2017

FE model (CAD Geometry)

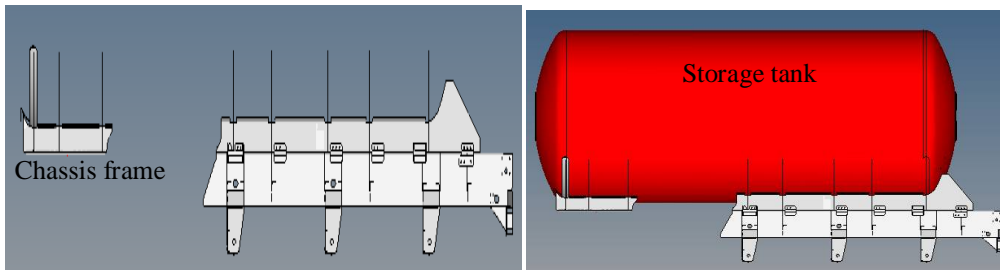


Fig.4.1 Chassis frame and storage tank

The FE model of the chassis is as shown in Fig.4.2 and is modelled with 3 noded triangular and 4 noded quadrilateral shell elements.

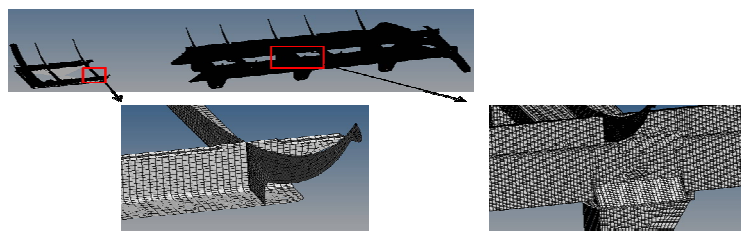


Fig.4.2 FE model

Depending on the cross section and region where stress concentration is more the element size varies. For example, in case of 'plate with a hole' where different size of elements is used, because the element size is very small near the hole compared to the faraway places. Selecting the number of elements is related to the size of the elements in finite element model.

Load case 1: Bumping load in vertical direction

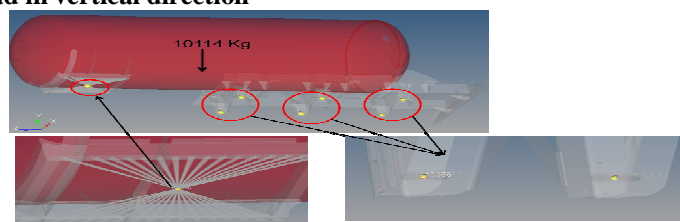


Fig.4.3 Load and boundary conditions for bumping load

The load is external load (10114 kg) which comes from fluid that stored in the tank when the vehicle moves over ditch or bump. The load is applied on the structure in vertical direction. The boundary conditions are applied at the centre point of the wheel hub. The centre points of the wheel are fixed in all degrees of freedom expect rotation about x axis and the front pivot axle is fixed in x and z translation and rotation about x and z.

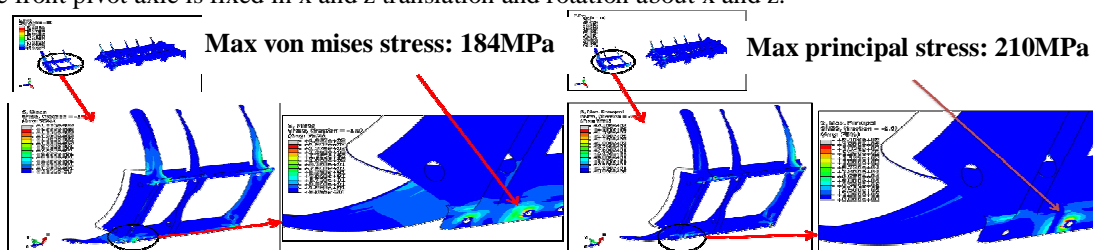


Fig.4.4 Max von mises stress and max principal stress

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: www.ijirset.com

Vol. 6, Issue 6, June 2017

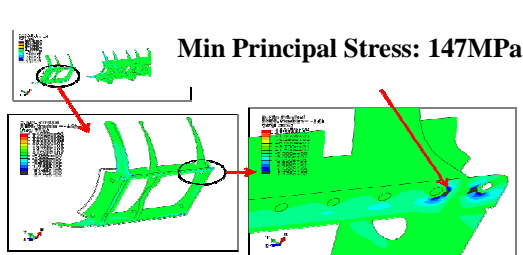


Fig.4.5 Min principal stress

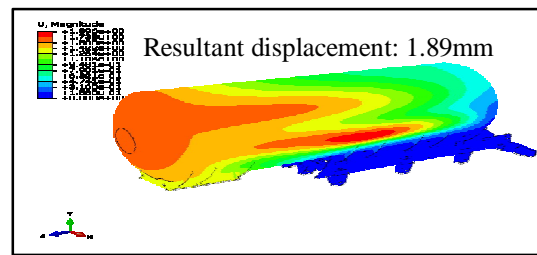


Fig.4.6 Resultant displacement

The von mises stress (184MPa), maximum principal stress (210MPa) and minimum principal stress (147MPa) for bumping load case shown in Fig4.4, and Fig4.5. The stress developed on the chassis structure is well below the yield strength of the material (355MPa). Hence the chassis structure is safe for bumping load case when vehicle moves on ditch or bump. The stress can be minimized by limited modification like adding stiffeners and cross members at joint areas. The resultant displacement 1.89mm is well below the acceptable value.

Load case 2: Breaking load in longitudinal direction

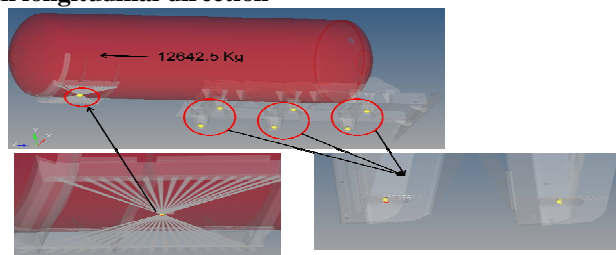


Fig.4.7 Loads and boundary conditions for breaking load

The load (12642.5 kg) applied on the structure in longitudinal direction which occurs when vehicle suddenly comes to rest position. The boundary conditions are applied to the wheel hub and front pivot axle. The load generated is longitudinal direction because pulling action by external load stored in tank. The centre points of the wheel are fixed in all degrees of freedom expect rotation about x axis and the front pivot axle is fixed in x and z translation and rotation about x and z.

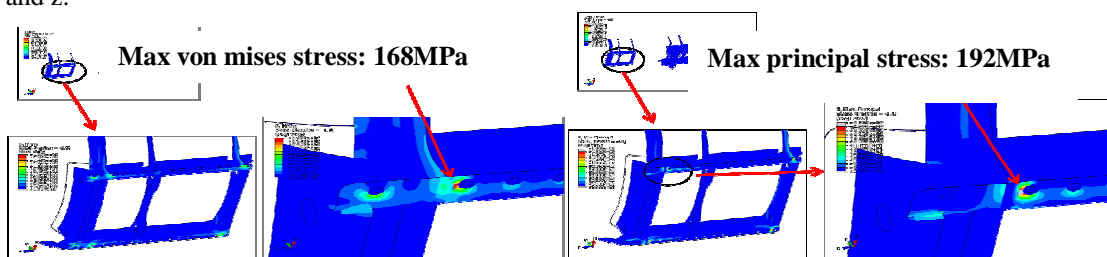


Fig.4.8 Max von mises stress and max principal stress

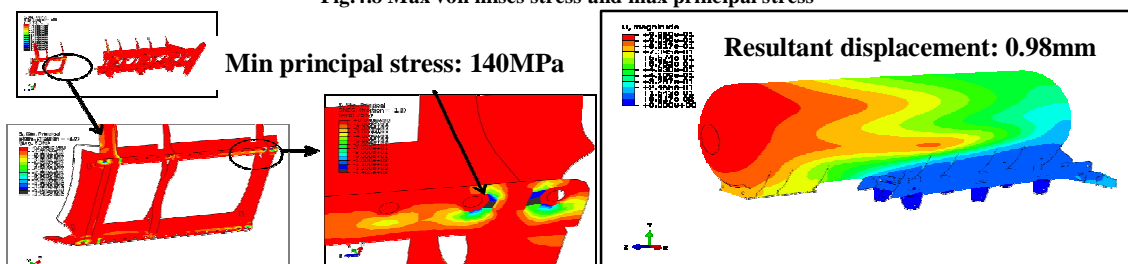


Fig.4.9 Minimum principal stress

Fig.4.10 Resultant displacement

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: www.ijirset.com

Vol. 6, Issue 6, June 2017

The von mises stress (168MPa), maximum principal stress (192MPa) and minimum principal stress (140MPa) for breaking load case shown in Fig4.9 and Fig4.10. The stress developed is well below the yield strength of the material (355MPa). For breaking load condition, when the vehicle suddenly comes to rest position the stresses will be developed at the front part is more compared to rear part of the chassis structure because the load taken by the chassis is more at the front area while breaking.

Load case 3: Turning load in transverse direction

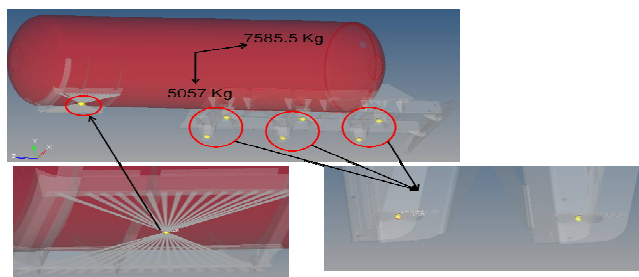


Fig.4.11 Loads and boundary conditions for turning load case

In this case, the maximum load on the chassis structure occurs from transverse direction when vehicle taking turn at bend roads. The boundary condition for turning load case is remains same as that for the bumping load and breaking load cases. The centre points of the wheel are fixed in all degrees of freedom expect rotation about x axis and the front pivot axle is fixed in x and z translation and rotation about x and z.

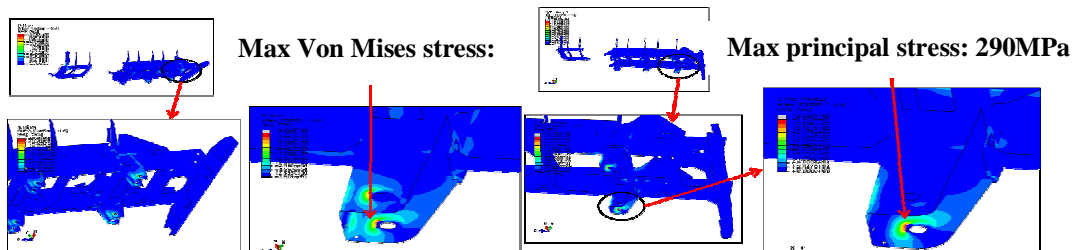


Fig.4.12 Max von mises and max principal stresses

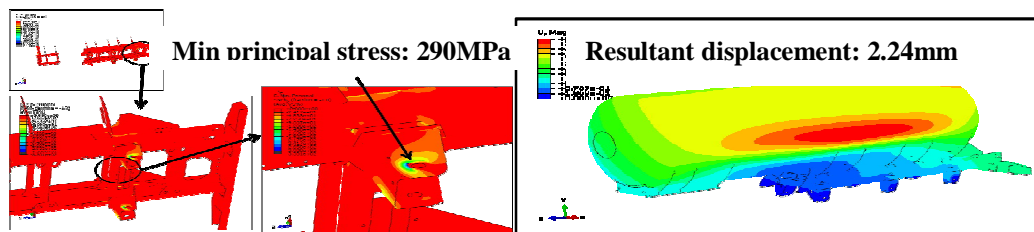


Fig.4.13 Min principal stress and Resultant displacement

The von mises stress (280MPa), maximum principal stress (290MPa) and minimum principal stress (290MPa) for turning load case shown in Fig4.12 and Fig4.13. When vehicle takes turn on corners, the one side of the vehicle slowly lifts then the torsional force act on the chassis. Hence the stresses caused in this case are more concentrated at hinges of the wheel hub. If the stress value obtained from this case as well below the yield strength of the material (355MPa), then the structure is safe for all three load cases. The resultant displacement (2.24mm) observed in this case is maximum compared to the bumping and breaking load case.

Material properties: Steel with YS of 355 MPa. $E = 2.1 \times 10^5$ MPa, Density (tons/mm^3) = 7.85×10^{-9} , Poisson's Ratio = 0.3.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: www.ijirset.com

Vol. 6, Issue 6, June 2017

Strength analysis

Strength analysis report													
Sl. No	Mass of structure (tons)	G Load	Applied Loads on the structure			Reaction force at fixed points			YS (MPa)	Max Principal	Min Principal	Max Stresses	FOS
			X	Y	Z	X	Y	Z					
LC1	5.057	2	0	-99218		0	99213		355	210	147	184	1.7
LC2	5.057	2.5	0	0	124023	0	0	-124016	355	192	140	168	1.8
LC3	5.057	1.5	74414	-49609	0	-74410	49606	0	355	290	290	280	1.2

V. CONCLUSION

From the results it is observed that stresses in the turning load case is maximum of 290MPa. The corresponding factor of safety for turning load case is 1.2. Considering the least factor of safety among the three load cases, the chassis should be designed. The maximum stress is 290 MPa and is well below the yield strength of 355MPa. Hence the structure is safe for all the three load cases and can be recommended for the production.

REFERENCES

- [1]. Cicek Karaoglu, N. Sefa Kuralay, "Stress analysis of a truck chassis with riveted joints" Elsevier Science B.V Finite Elements in Analysis and Design 38 pp. 1115-1130, (2001).
- [2]. Sairam Kotari, "Static and Dynamic Analysis on Tetra Chassis" International Journal of Modern Engineering Research (IJMER) (2002). ISSN: 2249-6645, Vol-2, Issue-1, (2002).
- [3]. Dr.R.Rajappan, "Static and dynamic load characteristics of chassis" (2013)
- [4]. Ahmad O. Moaaz, Nouby M. Ghazaly, A Review of the Fatigue Analysis of Heavy Duty Truck Frames. American Journal of Engineering Research (AJER) e-ISSN : 2320-0847 p-ISSN : 2320-0936 Volume-3, Issue-10, pp-01-06 www.ajer.org (2014).
- [5]. Divyesh N. Chaudhari, Prof. Vinod M. Makwana, and Prof. Dharmmeet J. Patel: A Review Paper on Strength and Rigidity analysis of Heavy Vehicle Chassis for various loading conditions (2015). International Journal of Advance Research in Engineering, Science & Technology (IJAREST), ISSN (O):2393-9877, ISSN (P): 2394-2444, Volume 2, Issue 12, December-2015.
- [6]. K.Rajasekar and Dr.R.Saravanan: "Literature Review on Chassis Design of On-Road Heavy Vehicles" IJISSET - International Journal of Innovative Science, Engineering & Technology, Vol. 1 Issue 7, September 2014.
- [7]. M. Ravi Chandra et al.: "Modeling and Structural analysis of heavy vehicle chassis made of polymeric composite material by three different cross sections. International Journal of Modern Engineering Research (IJMER), Vol.2, Issue.4, July-Aug 2012.
- [8]. Kiran Ghodvinde and S. R.Wankhade : "Structural Stress Analysis of an Automotive Vehicle Chassis." YTIET, Karjat,Raigad,India. (2014).