

# Mechanical Design of Extended Chassis Frame for Defence Vehicle

P.A.Naik and D.M. Kalai

Department of Mechanical Engineering DKTE's TEI, Ichalkaranji, Maharashtra, India  
Email: prashantnaik49@gmail.com

**Abstract** - Generally defence vehicles are considered has the roughed machines, they can withstand very harsh environment. The defence vehicles generally have all wheel drives which enable them to tread difficult terrain and these vehicles are made of high strength materials with high factor of safety. This project is focused to design and analysis of extended chassis frame for such a vehicle. The vehicle chosen in this project is an eight wheeled vehicle with all wheels drive system (8×8) which is connected to the engine through a low deferential gear box. There is a need of adding additional super structure (extra pay loads) on the same vehicle. Extending the present chassis with the help of reinforcement at right locations will be a very effective technique for wide variety of mounting applications. Thus using the extended chassis frame many mounting can be installed/mounted without considerable change in the overall design; this is also economical.

This project work is focused on Re-design of chassis frame by extending it by 900 mm at rear, based on end user requirement to withstand the different super structure loads, the same structure will be statically analyzed by using Finite Element Analysis method to verify the reinforcement of chassis frame and also to study the dynamic characteristics. This work involves static and dynamic analysis to determine key characteristics of a truck chassis. The static characteristics include identifying location of high stress area and the dynamic characteristics of truck chassis such as the natural frequency and mode shape.

**Keywords:** NASTRAN, TATRA chassis, PRO/E wildfire, Dynamic analysis, Static analysis.

## I. INTRODUCTION

Chassis is a French term and was initially used to denote the frame parts or basic structure of the vehicle. It is the back bone of the vehicle. A vehicle without body is called chassis. The components of the vehicle like power plant, transmission system, axles, wheels and tires, suspension, controlling systems like braking, steering etc., and also electrical system parts are mounted on the chassis frame. It is the main mounting for all the components including the body. So it is also called as "carrying unit". The chassis frame is made up of long two members called side members riveted/welded together with the help of number of cross members together forms an integral structure for the support of all chassis equipment and payload.

When the automobiles were first introduced, comparatively little attention was paid to the frame. The other components of the chassis such as power plant, gear set, axles, etc., were held of greater importance. The first type of automobile chassis was made of wood, ash being the material used. The cross members, used for the different unit support, were forged out of angle iron and bolted on the main frame and, in some cases, they were brazed to and made a part of the main frame. After experiencing considerable difficulty, however, due to accidents and other failures which were traced directly to poor-frame design, the automobile engineer found that it was possible to build a frame of great strength with less weight than the types heretofore produced [1].

The improvement in frame design is the result of the tendency to provide perfect alignment of the power plant, clutch, and gearset, making use of what is known as the unit power plant on most models, while on some, particularly on heavier type, flexible mounting of the units has been resorted to. The tendency is toward the use of a flexible mounting of all individual units, at least to some degree, in order to relieve them of the stresses brought about by frame weaving when the road wheels mount an obstacle on the road surface. Although the simplest frame have straight side member in plain and elevation view, packaging requirement for modern, high-powered, vee-type engines frequently require more complex shapes at the engine area.

The work carried out is,

1. Collection of data of material properties and loading details.
2. Design of chassis frame.
3. Static analysis using FEA software.
4. Modal analysis of the chassis frame.
5. Numerical evaluation of results obtained from analysis.

The Finite Element Analysis depicts maximum stress of 163MPa (for two load cases) which is well within the allowable limit of 205 MPa, based on Yield Strength of 410 MPa. It is concluded that, the design of the chassis frame with rear extension and reinforcement is considered adequate with a Factor of safety of 2.52.

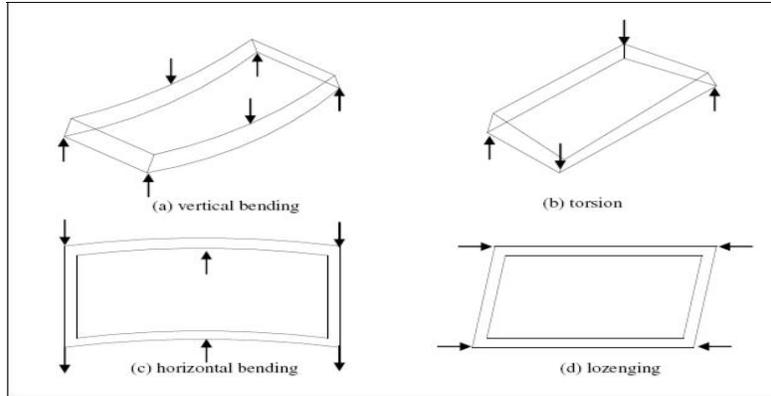


Fig.1 Modes of loading

The investigation on the dynamic behavior of truck chassis has been successfully executed. The dynamic characteristic such as the natural frequencies and mode shapes of the truck chassis are determined and validated numerically and experimentally.

**II. LITRETURE SURVEY**

There are two main objectives, which involves on the development of truck Chassis. Firstly, the appropriate static and dynamic characteristics of the existing chassis have to be determined. Secondly, structural development process in order to achieve high quality of the product. There are many factors which involve and must take into account. These can affect on the vehicle rolling, handling, ride stability and etc.

Today, there are many researches and development programs available by the International truck manufacturers, which are very much related to this study. There are several technical papers from the „Engineering Society for Advancing Mobility Land Sea Air and Space“(SAE) which are reviewed and discussed in the following sections.

**III. METHODOLOGY**

In this work, modal, finite element analyses are used to determine the characteristics of the truck chassis. The analysis results are used to develop virtual model created using FEM tools and the model is updated based on the correlation process.

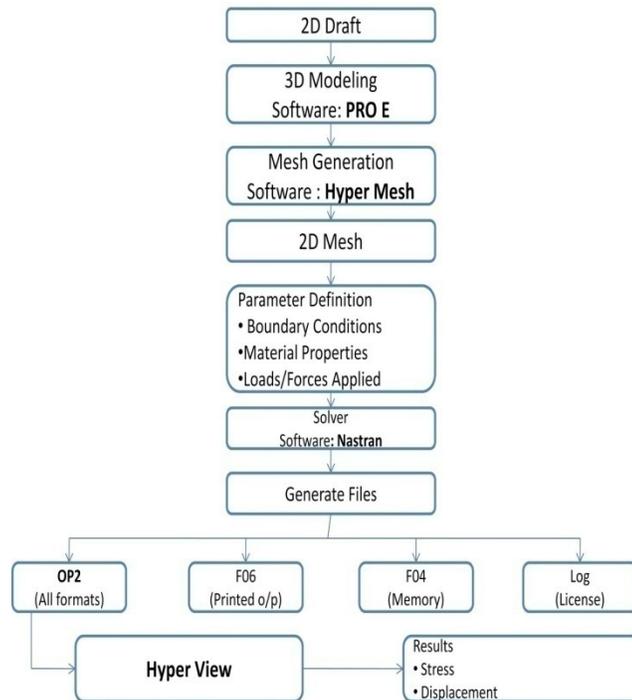


Fig.2 FEM tools

**IV. ASSUMPTIONS**

1. Residual stresses that may exist in the structure because of welding, manufacturing process are not considered and the analysis carried out for stress free structure.
2. Chassis frame is considered as a continuous seamless member for the analysis.
3. The hydraulic crane structure and its connection to chassis frame are considered as rigid, so that load at the end of boom will be completely transmitted to the chassis frame.

**V. LINEAR STATIC ANALYSIS**

**A. Geometric Modeling of Chassis**

The chassis model of TATRA is created by using PRO/E wildfire.

Introduction to Pro/E

*1. Part design*

This application makes it possible to design precise 3D mechanical parts with an intuitive and flexible user interface, from sketching in an assembly context to iterative detailed design. Part Design application will enable you to accommodate design requirements for parts of various complexities, from simple to advanced.

*2. Assembly design*

It allows the design of assemblies with an intuitive and flexible user interface. As a scalable workbench, Assembly Design can be cooperatively used with other current companion products such as Part Design and Generative Drafting.

*3. Interactive drafting*

It is a new generation product that addresses 2D design and drawing production requirements.

Interactive Drafting is a highly productive, intuitive drafting system that can be used in a standalone 2D CAD environment within a backbone system. It also expands the Generative Drafting product with both integrated 2D interactive functionality and an advanced production environment for the dress-up and annotation of drawings. This provides an easy and smooth evolution from 2D to 3D-based design methodologies.

The part modeling of the Front Chassis Frame includes:

- a. Front bumper
- b. Cabin mounting bearing
- c. C-Channel
- d. Engine mounting bracket
- e. Cabin rest

The part modeling of the rear chassis frame includes:

- a. C-Channel
- b. Vertical stiffener
- c. Cross member
- d. Cross header
- e. Rear bumper

In PRO/E wildfire, the part is modeled by using part design workbench

**B. Geometry Details Of Extended Chassis Frame**

- i. The standard cross section of chassis frame including extension of 900 mm is of C-section of dimensions 250 x 102 x 6 mm.
- ii. One number of additional box section included at the extension portion of chassis to improve the torsion rigidity of the chassis frame.

This arrangement of reinforcement details is shown in Frame assembly (Rear) of the chassis frame with 900 mm extension. The specification of the chassis is shown in the Table 1.

TABLE 1 GEOMETRIC SPECIFICATIONS OF CHASSIS

| Sl. No. | Description       | Dimension (mm) |
|---------|-------------------|----------------|
| 1       | Length of Chassis | 8550           |
| 2       | Width of Chassis  | 1000           |
| 3       | Chassis Material  | IS2062         |

The Isometric view of chassis showing the top side of the Conceptual model of TATRA Chassis created from the 2-D drawing in PRO/E is shown in fig.3.

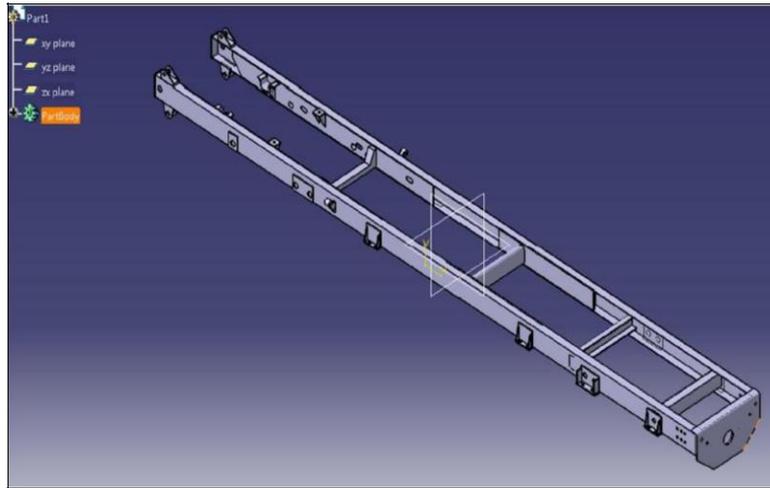


Fig.3 Isometric view of chassis-I

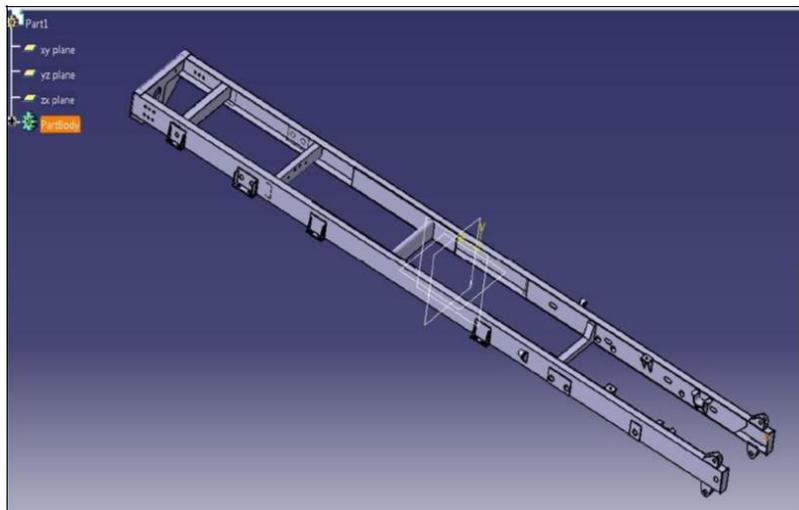


Fig.4 Isometric view of chassis-II

### Mesh Generation Using Hypermesh

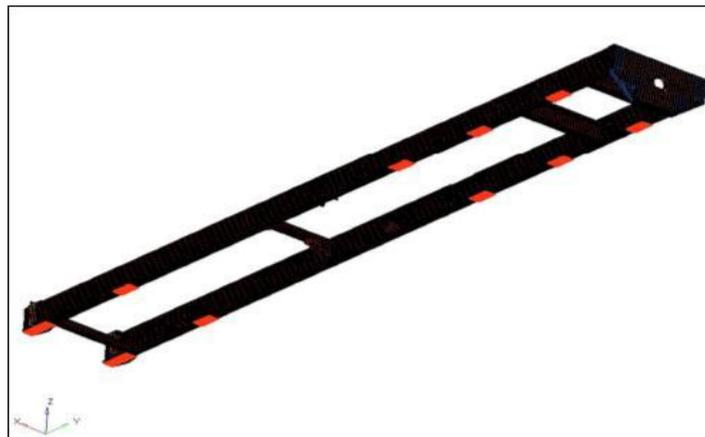


Fig.5 Applied constraints

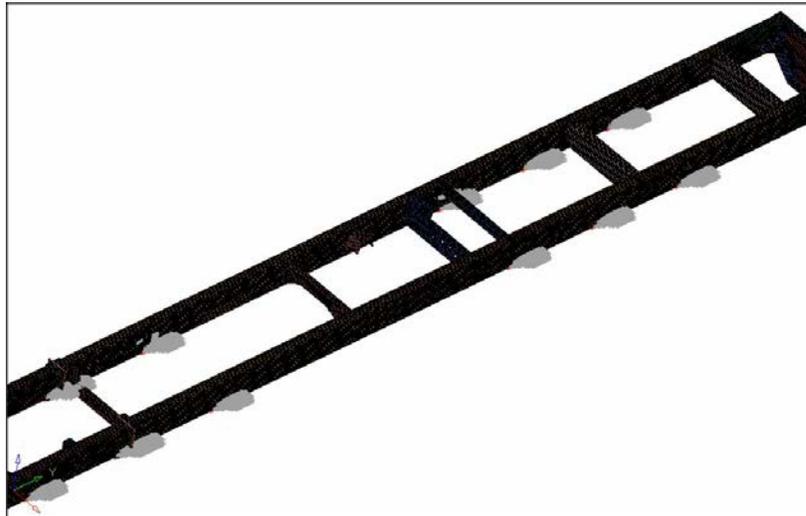


Fig.6 Constrained At Front And Rear Tandem Axle Supports

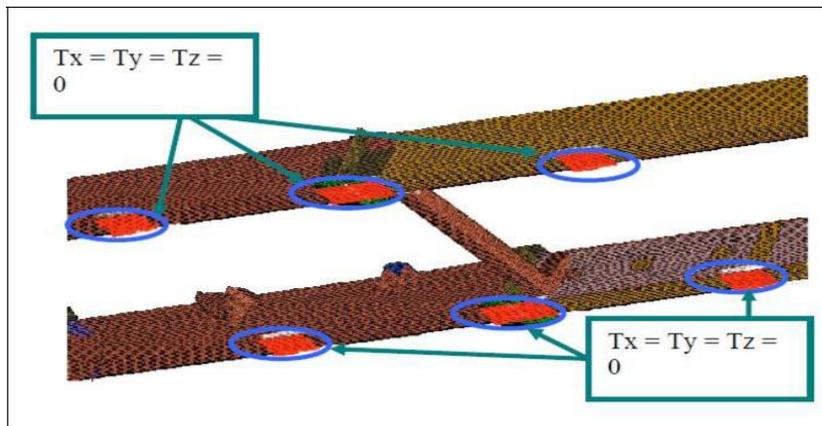


Fig.7 constraints at front tandem axle support

Fig.8 shows the right hand side of chassis frame at 900 mm extension portion, provided with the inner C-channel of 8mm thickness as the reinforcement inside the outer C-channel of thickness 6mm to increase the stiffness of the chassis.

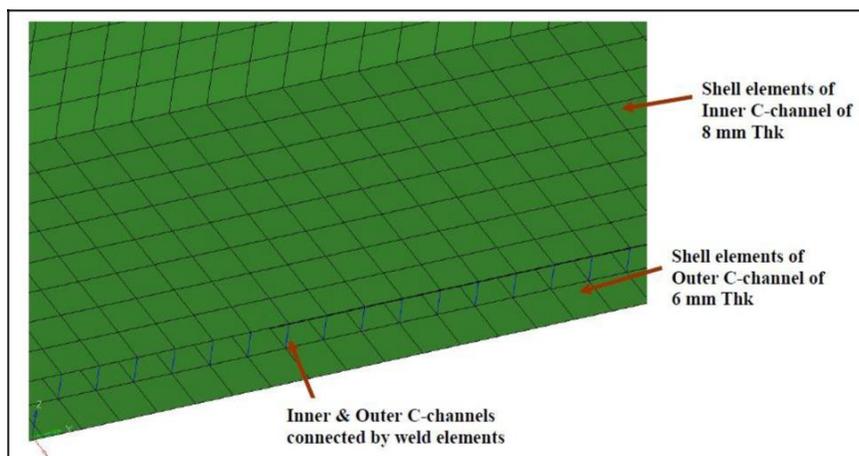


Fig.8 Chassis frame at 900 mm extension portion



Fig. 9 Load applied on the chassis

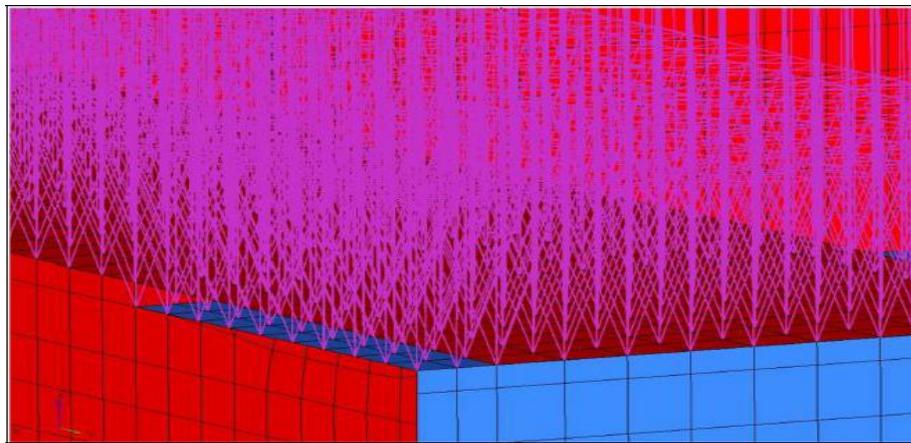


Fig.10 Load applied (magnified view)

Fig.10 shows the magnified view of the load applied over the rear section, the load are applied by selecting the nodes or elements at the corresponding loading locations

## VI. DESIGN ANALYSIS

### NASTRAN

NASTRAN is a Finite Element Analysis program that was originally developed for NASA in the late 1960s under United States government funding for the Aerospace industry.

The MacNeal-Schwendler Corporation (MSC) was one of the principal and original developers of the public domain NASTRAN code. NASTRAN source code is integrated in a number of different software packages, which are distributed by a range of companies.

MSC Nastran is the original commercial Nastran product started by Dr. Richard MacNeal. For many years, MSC maintained a monopoly on the NASTRAN source code,

which ended in June 2003 by the purchase by EDS of MSC.Nastran v2001 source code.

There are three commonly used steps employed in general purpose FEA[9]:

**A. Pre-processing:** This is basically creating the model, inputting geometry, materials properties, boundary conditions and loads. Some software provides automatic mesh generation algorithms but some will require extensive user input to build the desired mesh. Thought must be given to symmetry possibilities, mesh element size, element aspect ratios, geometric limitations etc.

**B. Numerical Analysis:** This is where the software automatically generates element stiffness matrices that represent the behaviour of each element, combines them into the global element stiffness matrix equations that represents the complete structure and solves these equations to determine values at the nodes.

**C. Post Processing:** The results of the FEA are displayed graphically with the user determining which quantities they

wish to look at. Most software will output tables, deformation, stresses and much more.

**Hyper view**

Altair Hyper view is a complete post-processing and visualization environment for finiteelement analysis (FEA), multi-body system simulation, video and engineering data.

Hyperview can visualize data interactively, as well as capture, standardize and automate post-processing activities.

**Stress and Displacement Contours for Load Case – I**

The stress and displacement contours have been obtained for the two load cases mentioned, following figures shows the Stress and displacement contours for load case-I.

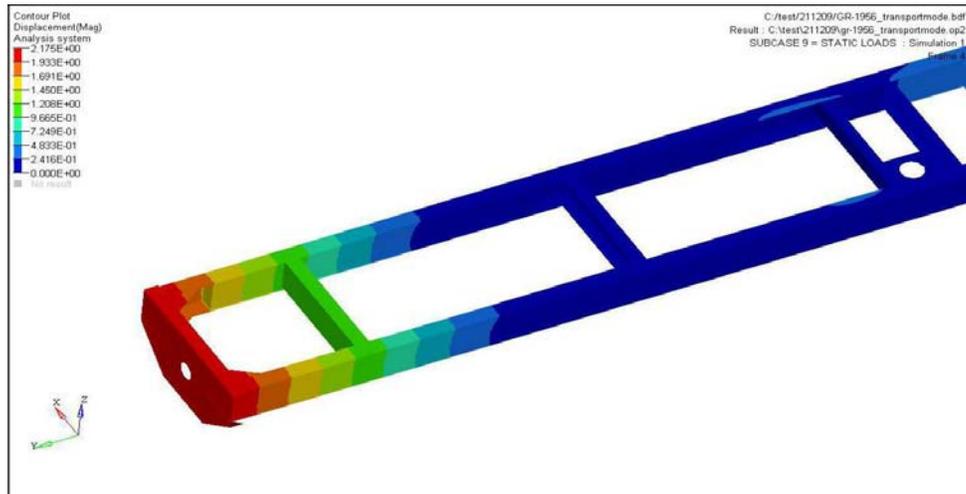


Fig.11 Displacement contour for load case-I

Fig.11 shows the Displacement contour for the first load case i e, closed transport mode. It is found that maximum displacement occurs at the rear end of chassis frame which can be visualized from the red colored portion indicating maximum value

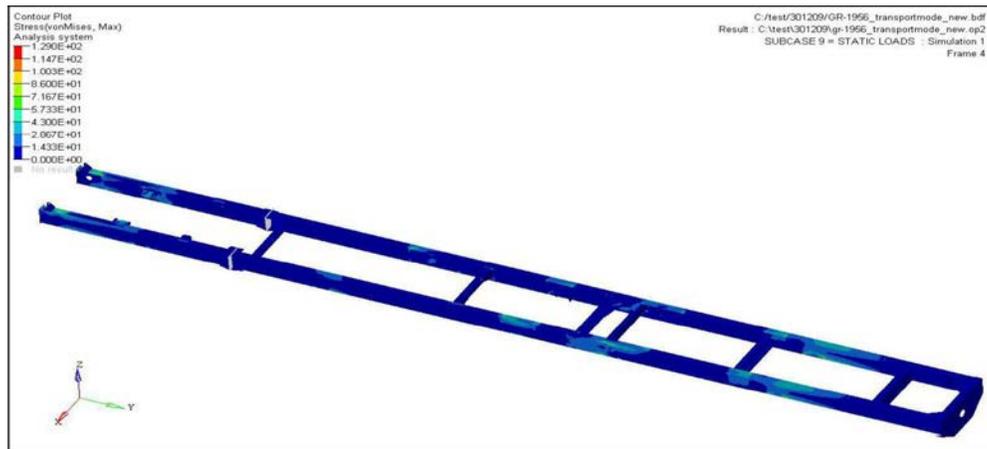


Fig.12 Stress Contour Of Full Chassis Frame For Load Case-I

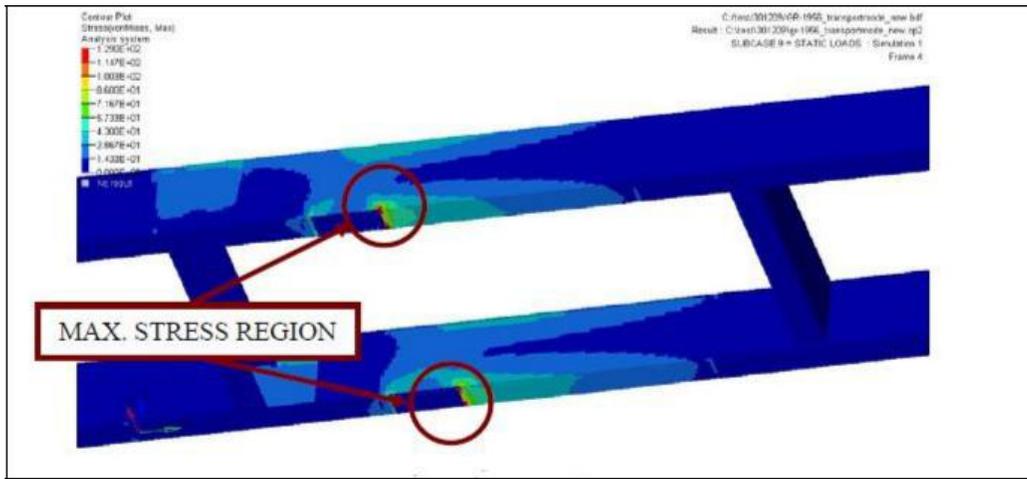


Fig.13 Stress Contour (View From Bottom) For Load Case-I

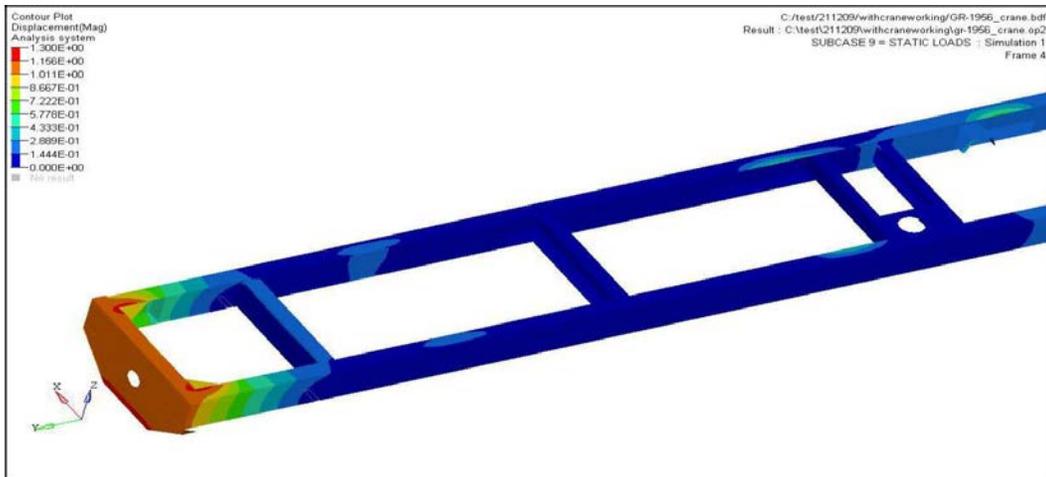


Fig.14 Displacement Contour For Load Case-II

## VII. RESULTS

As the material is ductile, maximum distortion energy theory (Von mises theory) is considered for determining the criterion for failure. Taking a factor of safety of 2.0, the allowable stress for the material is 205 MPa, based on Yield Strength of 410 MPa.

It is concluded that, the maximum stress of 163 MPa (for both of above load cases, over the entire chassis including extended portion) is well within the allowable limit.

Hence, the design of the chassis frame with rear extension and reinforcement is considered adequate with a factor of safety of 2.52.

## VIII. DYNAMIC ANALYSIS

The dynamic analysis helps

1. Account variation of the load with respect to time. Output in the form of stress, displacement, velocity etc wrt time could be predicted by dynamic analysis.
2. The velocity and acceleration can be characterised.
3. The modal analysis can be carried out.

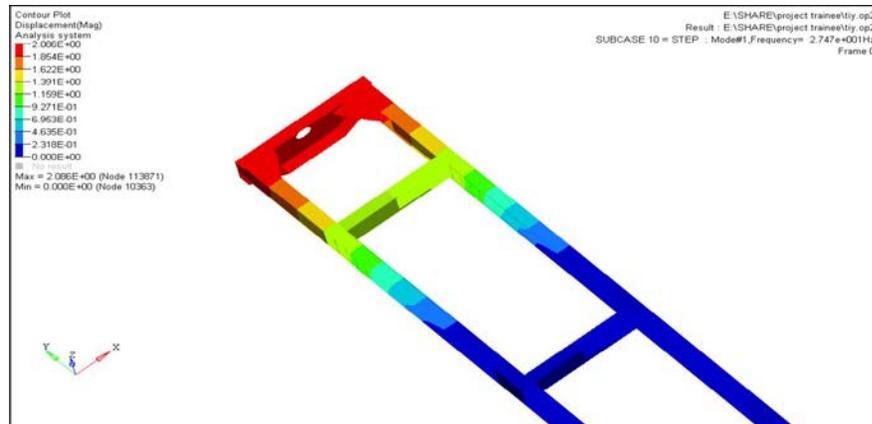


Fig.15 first global mode at 27.47 hz

## IX. RESULTS AND DISCUSSION

This chapter describes the results and discussion of the experiment, theoretical and finite element analysis of the truck chassis in order to determine static and dynamic characteristics. The results of linear static analysis using FEA method is analytically validated and modal analysis and modes shape obtained from FEA and experimental method are compared in order to establish the accuracy and validate the FE modeling.

Experimental results are correlated analysis with FE results. Based on the correlation, the model updating of the truck chassis carried out to acquire better agreement between FE model and experimental model. Finally, this model updating result is used for the further analysis in vibration or dynamic analysis.

### A. Linear Static Analysis

Conceptual model of TATRA Chassis is created from the 2-D drawing in PRO/E, This conceptual model is discretized and converted to Finite Element model using Hypermesh incorporating suitable Elements. Loads and Boundary conditions are imposed and the structure is solved for the given load conditions using MSC Nastran. The exercise is repeated for two load patterns with the addition of reinforcement, to determine the structural response. The same exercise is attempted using analytical method based on Shear force and Bending moment diagram. The Analytical results and Finite Element Analysis are in good agreement. Hence the structural response is as predicted by both models proves that the design of the TATRA Chassis is safe.

As the material is ductile, maximum. Distortion energy theory (Von-mises theory) is considered for determining the criterion for failure. Taking a factor of safety of 2.0, the allowable stress for the material is 205 MPa, based on Yield Strength of 410 MPa. The maximum stress induced is 163

MPa. (For both of above load cases, over the entire chassis including extended portion), which is well within the allowable limit. Hence, the design of the chassis frame with rear extension and reinforcement is considered adequate with a factor of safety of 2.52.

### B. Model Analysis

The investigation on the dynamic behavior of truck chassis is carried out. The application of dynamic correlation technique is performed for verification of the finite element model of truck chassis. Experimental results are used in conjunction with the finite element modal results to predict the dynamic characteristic of truck chassis such as the natural frequency and mode shape. Basically, natural frequency and mode shape are important parameters in design. That is because damage can be occur if the truck chassis is excited at the natural frequency during operation. The dynamic characteristic such as the natural frequencies and mode shapes of the truck chassis is determined numerically and experimentally. The experimental data is used to validate a finite element model representing the real structure. The results indicate that the FE model shows a good correlation with the experimental model for the mode shape but not for the natural frequencies. The FE model presented an average of 10% higher frequencies than the real chassis. This fact is due to fact that FE model is stiffer.

## REFERENCES

- [1] Dave Anderson and Grey Schede, "Development of a Multi- Body Dynamic Modal of a Tractor – Semi trailer for Ride Quality Prediction", International Truck and Engine Corp. 2001.
- [2] Ibrahim I.M.,Crolla D.A and. Barton D.C, "Effect Of Frame Flexibility On The Ride Vibration Of Trucks", Department of Mechanical Engineering, University of Leeds LS2 9JT, U.K. August 1994.
- [3] Pomulo Rossi Pinto Filho, "Automotive Frame Optimization", Universidade Federal de Uberlandia. November 2003.
- [4] Izzudin and Abd. Rahman, Roslan, " Application Of Dynamic Correlation Technique And Model Updating on Truck Chassis" SAE paper-768665 (2006).

- [5] Murali M.R. Krisna, "Chassis Cross-Member Design Using Shape Optimization", International Congress and Exposition Detroit, Michigan. February 23-26,1998.
- [6] Lonny L. Thomson, Pipasu H. Soni, Srikanth Raju, E. Harry Law, "The Effects of Chassis Flexibility on Roll Stiffness of a Winston Cup Race car", Departmental of Mechanical Engineering, Clemson University, 1998.
- [7] "MSC.Nastran for Windows: Quick Start Guide", MSC.Software Corporation, United States of America, 2000.
- [8] William j. sidelko "an objective approach to highway truck frame design", SAE Paper-660162.
- [9] Mehdi Ahmadian And Paul S. Patricio Advanced Vehicle Dynamics Laboratory, "Dynamic Influence Of Frame Stiffness On Heavy Truck Ride Evaluation" Department Of Mechanical Engineering, Virginia Tech, SAE paper-01-2623, 2004.
- [10] Carver G C. "truck chassis frame considerations in equipment mounting" A O. Smith Corp. SAE Paper-760291.
- [11] Avern R D " the text book on Automobile chassis design".