Structural Stress Analysis of an Automotive Vehicle Chassis

[1]Kiran Ghodvinde, [2]S. R.Wankhade

[1,2]YTIET, Karjat, Raigad, India

Email: [1]kiran.ghodvinde@tasgaonkartech.com, [2]santosh.wankhade@tasgaonkartech.com

Abstract -The automotive chassis forms the structural backbone of a commercial vehicle. The main function of the chassis is to support the components and payload placed upon it. When the vehicle travels along the road, the chassis is subjected to various stress distribution and displacement under various loading condition.

The method used is numerical analysis is finite element technique to find the critical stress. In this dissertation work we have 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.

In this paper is deal with 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.

Keywords: Chassis, chassis of vehicle, stress on chassis, chassis strength analysis, automotive frame, ladder frame, Chevy truck chassis, kit car.

I. INTRODUCTION

The design of chassis is fully on conceptual basis and the objectives are to create conceptual design for an automotive chassis which will utilize standard components. And then Develop CAD drawings for this Concept design of chassis. In this paper we have chosen one automotive chassis for the analysis which is to be done by using Finite Element Analysis (FEA) of the chassis design.

The chassis panel is one of the main parts of the motor vehicle; engine transmission, suspension, shafts steering and body are mounted on the chassis M. P. Bendsoe[1].

Every vehicle has a body, which has to carry both the loads and its own weight. Vehicle body consists of two parts; chassis and body work or superstructure. The conventional chassis frame, which is made of pressed steel members, can be considered structurally as grillages CicekKaraoglu,et al. [2].

The study of the mechanical behavior of the chassis and their welded or rivet joint are becoming a major concern in recent years. For this purpose, finite element analysis methods have been used extensively M.Tanaka[3]. In this study to validate the FEM based model of the chassis; an experimental modal analysis has been carried out.

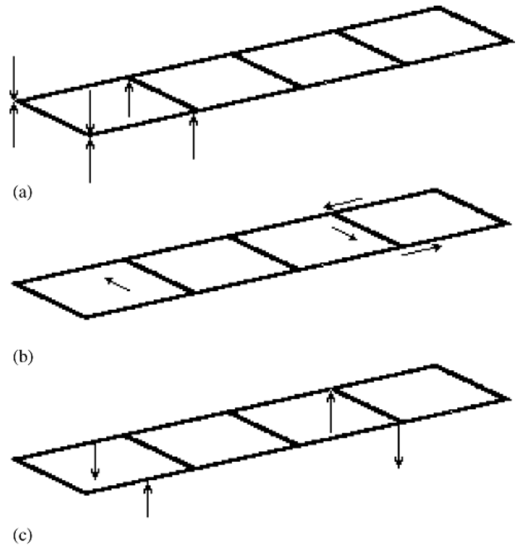

Fig. 1. Main loads on Chassis Frame

1.2. Chassis Design Consideration

The Automotive chassis are meant for two main goals

- Hold the weight of the components
- To rigidly fix the suspension components together when moving

The first item is an easy design solution and is also the basis of the original chassis designs. One of the most effective shapes for supporting point loads fixed at two ends is an I-Beam, a box tube, or a C-Beam. One beam on either side so that a floor could be attached and even the smallest of I or C beams can hold tremendous weight M. Zehsaz [4].

Space frames gave way to the monocoque chassis as large flat surfaces had more mass consistently further away from the neutral axis. Race cars did this first by skinning a space frame to retain a supporting structure. It then eliminated this space frame and replaced it with light weight honeycomb material. These was due to the thin walls being strong in shear but in the compressive direction were unstable and buckled easily. The honeycomb material added a reinforcing structure, and coupling two layers surrounds the driver in a very strong enclosure that is resistant to penetrative loads. Today the main focus is on the material of the structure and layering techniques since the underlying concept is well developed C.J.Cooke [5].

1.3 Various Loads Acting on the Frame

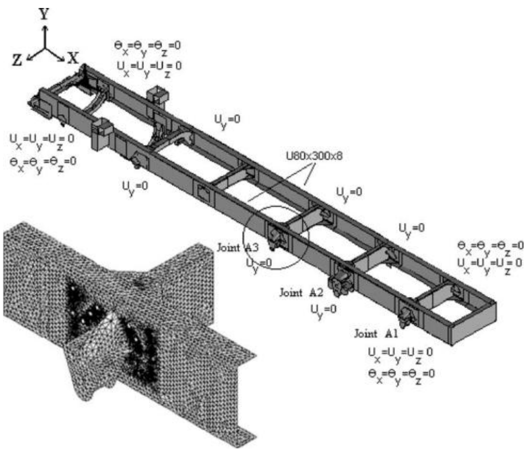

Fig. 2. Loads on chassis.

Various loads acting on the frame are

1. Short duration Load - While crossing a broken patch.

2. Momentary duration Load - While taking a curve.

3. Impact Loads - Due to the collision of the vehicle.

4. Inertia Load - While applying brakes.

5. Static Loads - Loads due to chassis parts.

6. Over Loads - Beyond Design capacity.

II FINITE ELEMENT FORMULATION

The formulation is based on the method due to Pian where an equilibrium stress yield is assumed in the interior of each element and independent displacements are assumed along the boundaries. Instead of using the conventions formulation, a more direct approach was used by the authors to form and minimize the total complementary potential.

Stress field F_i inside each element is assumed in the form of a polynomial with m known stress coefficient β. This can be expressed in tensor notation as

$$F_i = \sum_m A_{im} \eta^{B_{im}} \xi^{C_{im}} \beta_{im} \quad (1)$$

Where η and ξ are non dimensional variables A,B and C are constants. Independently assumed boundary displacements U_{ke} which must be compatible along the inter element boundaries e may also be represented as

$$U_{ke} = \sum_f \sum_p J_{kefp} X^{N_{kefp}} qf \quad (2)$$

qf is the generalized nodal displacement, p is the number of coefficient in the displacement function per nodal displacement, X is coordinate variable η or ξ, J and N are constants. Stress resultants F_{ke}, acting along the boundaries in the direction of the associated displacements is represented by the same notation.

$$F_{ke} = \sum G_{kem} X^{L_{kem}} \beta_m \quad (3)$$

2.1 Steps for Analyzing the Automotive Chassis.

The model is first prepared in solid works and catia and then imported to Ansys for further analysis. The following flow chart tells us the brief procedure of the automotive chassis analysis.

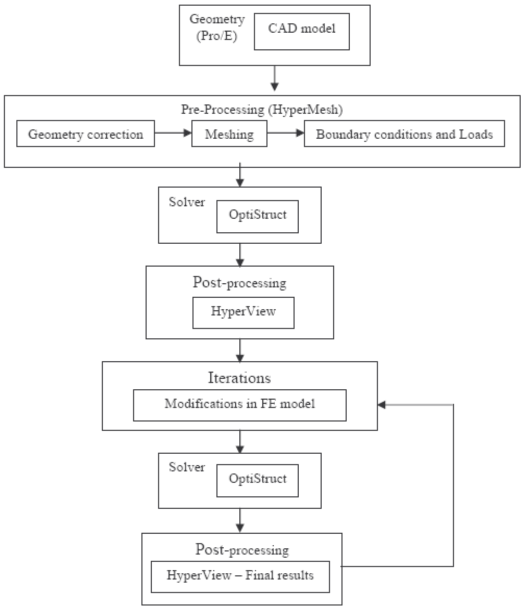

Fig. 3. Overview of Chassis Analyses

2.2. Kit Car

A kit car, also known as a "component car", is an automobile that is available as a set of parts. The definition of a kit car usually indicates that a manufacturer constructs multiple kits of the same vehicle which they then in turn sell.

2.3. Modeling of the Chassis Type Structure Form with Beams of Equal Resistance

The modeling of the chassis type structure form, assumes an optimal dimensioning of the chassis frame components depending on the stress values existing on the whole structure. The modeling assumes following steps:

- Applying the stress obtained by calculation, on the whole length of the chassis beam; - calculation of the cutting force diagrams “T” and of bend moments “M” on the whole length of the chassis beam;

- Calculation of the resistance manner of the “W” beam profile, from the equal resistance condition;

- Modeling of the beam profile geometric dimensions;

The form of the equal resistance beam for a maximum unit are effort is given by following relation:

$$W_z(x) = \frac{M_i(x)}{\sigma_{max}} \quad (1)$$

Where Mi(x) and Wz(x) are functions of the diagram having as the abscissa the position of the section. A certain form of the section being imposed, so that its dimensions shall be determined by a single parameter, from equation (1) the law of variation for this parameter will be obtained throughout the whole length of the beam: for a beam of rectangular section, with constant width b and variable height h (x), related to the beams abscissa, the relation (1) becomes:

$$\frac{b.h^2(x)}{6} = \frac{P.x}{\sigma_{max}} \qquad h(x) = \sqrt{\frac{6.P}{b.\sigma_{max}}.x} \quad (2)$$

The height GER will vary following a parabolic law. Near the free end of the beam, the bending moment has low values. Here the section height results from the resistance condition given by the equation:

$$\frac{3.P}{2.b.h} = \sigma_{max} \quad (3)$$

Where: $h = \frac{3.P}{2.b.\sigma_{max}}$ (4)

The determination of the transversal section dimensions results from the resistance condition:

$$W_{necesar} = \frac{M_{max}}{\sigma_a} \quad (5)$$

2.4. Boundary Condition for Kit Car Chassis

It has the two side rails as shown in fig. 4. The chassis is made from the 4130 steel tube and the outer diameter 30mm and thickness 4 mm. The material of the Kit car chassis is SAE 4130 steel. The properties of the material are listed below:

Chemical Composition by weight, % = 0.28 C, 0.8 Cr, 0.4 Mn, 0.15 Mo

Modulus of Elasticity, E= 30 GPa

Mass Density, ρ= 7798 kg/m^3

Yield Strength = 435.059 MPa

Tensile Strength = 670.17 MPa

The whole chassis is considered as the one beam and the load is applied and the analysis is done with the validation. It is being validated by the cantilever beam analysis done in Ansys. So that we can say that the result we get is correct.

2.4.1 Finite Element Model

The finite element model of the kit car chassis is prepared by using the coordinates system and the points are joined by using the lines and then the cross section is given by using the element as this element is suitable for such type of analysis and the analysis is firstly validate by using the cantilever beam.

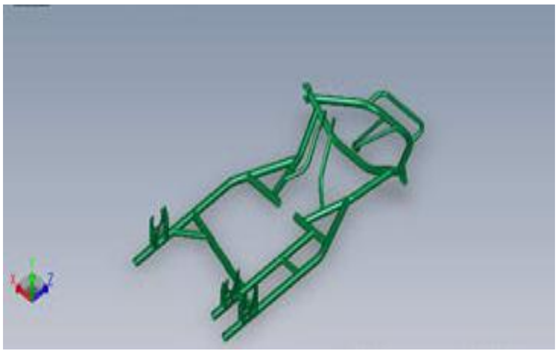

Fig.4. Model of Kit Car Chassis in Solid Works

Before analyzing chassis body in ANSYS we must validate the analysis, as we know that every analysis done in ANSYS must be validate we are considering here the beam of same length i.e. a cantilever beam.

The validation program contains the length 1380 mm beam length and it’s a cantilever beam. The bending moment of the beam. This values is being validate with the analytical formula also.

$$y = \frac{PL^3}{3EI} = \frac{(100)(1380)^3}{3(30000)(\frac{1}{4}\pi(9.5^4 - 7.5^4))} = 584.4\text{mm}$$

$$\sigma_{bend} = \frac{My}{I} = \frac{(100)(1380)}{\left(\frac{1}{4}\pi(9.5^4 - 7.5^4)\right)} = 262.53 \text{ MPa}$$

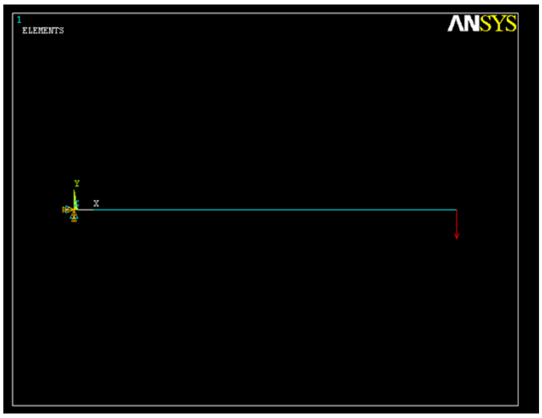

Fig . 5. Load Applied on Beam

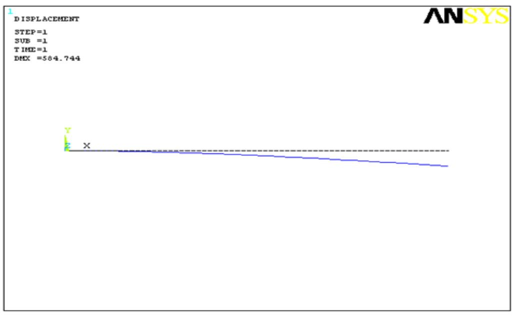

Fig. 6. Deformation of beam

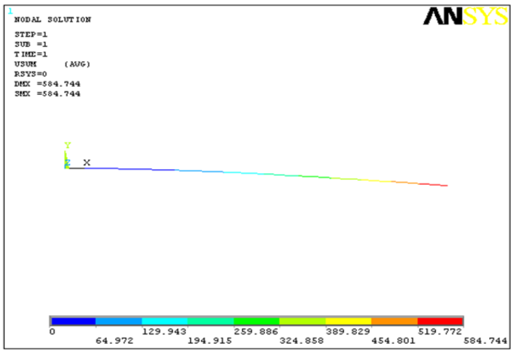

Fig. 7. Deflection in Beam

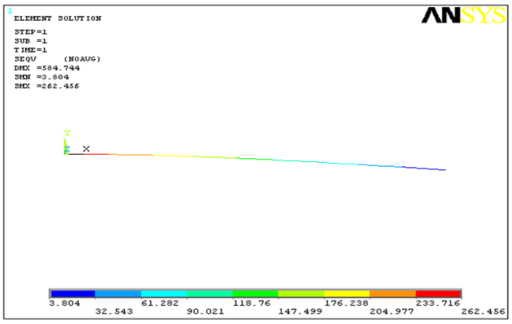

Fig.8. Stress Acting on Beam

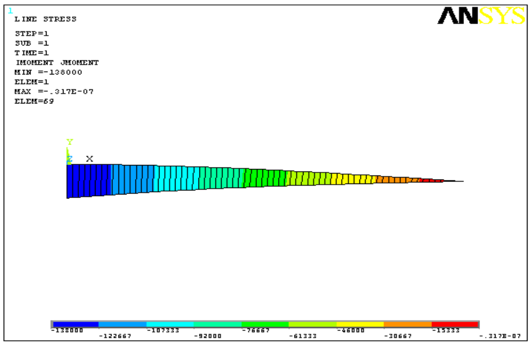

Fig.9. Bending Moment on Beam

The above work shows the analysis of the beam i.e. the validation program of chassis. Now we are going to analysis the kit car chassis body with the same procedure as we have done for the previous one. The chassis is model is prepared in ANSYS to prevent the data loss due to importing and the model look like as shown in fig 10.

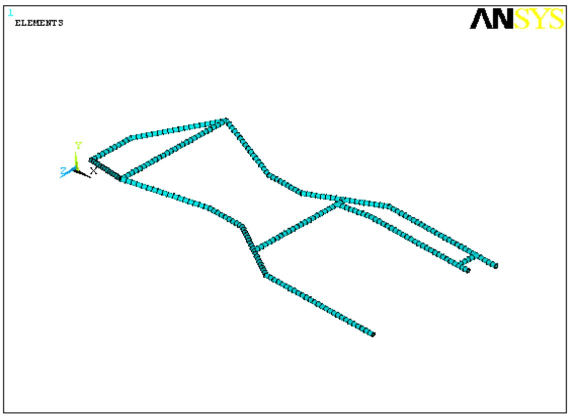

Fig. 10. Model of Kit Car Chassis Build in Ansys

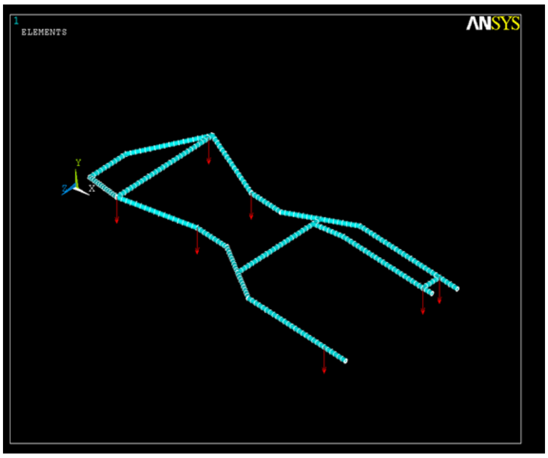

Fig 11. Load on the Kit Car Chassis

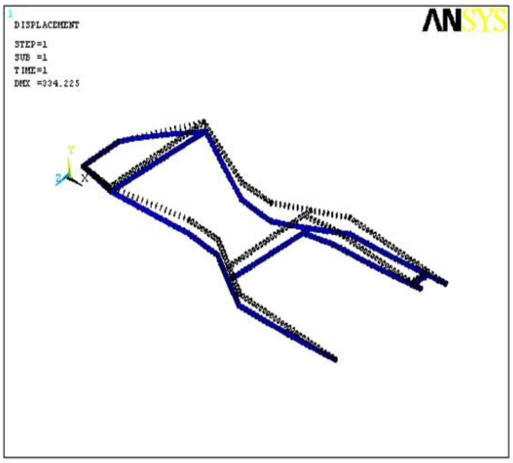

Fig.5.11. Deformation in Chassis

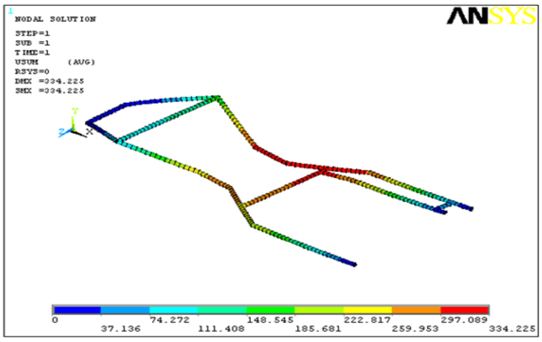

Fig.12. Deflection of Chassis

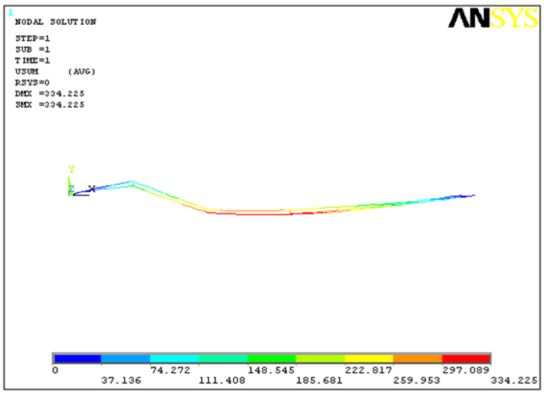

Fig.13. Deflection of Chassis Front View.

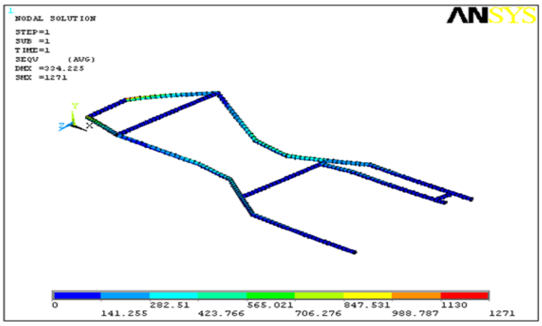

Fig. 14. Stress Acting at the Nodal Points

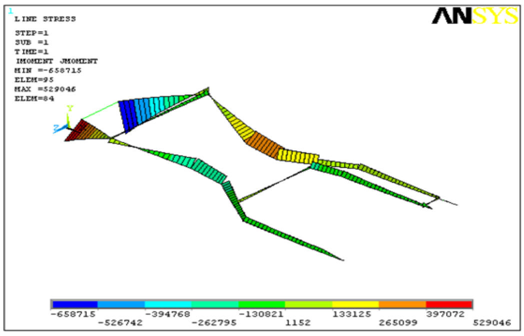

Fig.15. Bending moment diagram

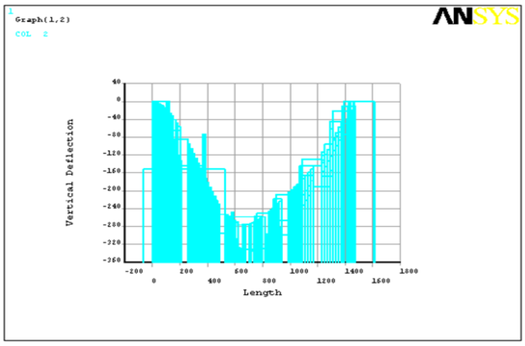

Fig 16. Chart of Deflection vs Length

2.4.2 Chevy Truck Chassis Analysis

The Chevy truck chassis is being model in the CATIA software and then the analysis is done in the ANSYS. The fig .17 shows the model of the truck chassis in CATIA V5. It's having the C member at the side bar and the frame is ladder type.

The automotive chassis used for the study is Chevy Truck Chassis and it has a narrow body with a gross weight of 4.5 ton and a payload of 2500 kg. It consists of 2 C-channel side rails and have 4 cross members along the 2 side rails as shown in Fig.17. There are some additional members like flat and gusset brackets located at the joint between side rails and cross members to strengthen the joints. Towards the middle is a top hat cross member to provide space for mounting of the gear box. The final 2 cross members are C-channel. These are located exactly at the location where the rear suspension is mounted at the side rails. It is to strengthen the chassis frame as the suspension mounting point is a highly stressed area.

The material of the truck chassis is AISI 4130 alloy with quenched and tempered treatment. The properties of the material are listed below:

Chemical Composition by weight, % = 0.30 C, 1.0 Cr, 0.90 Mn, 0.20 Mo

E – Young's Modulus = 29,733,000 psi
G – Shear Modulus = 11,600,000 psi
υ - Poisson's Ratio = 0 .2816
Density (lb / cu. in.) = 0.283
Specific Gravity = 7.8

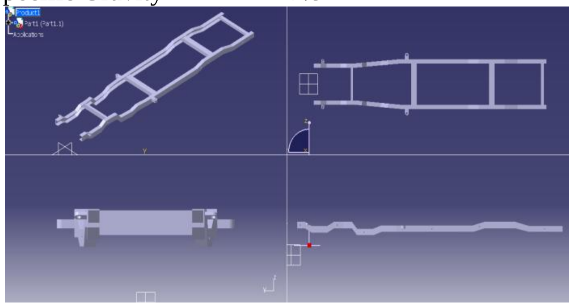

Fig. 17. Chevy Truck Chassis Model in CATIA V5R16.

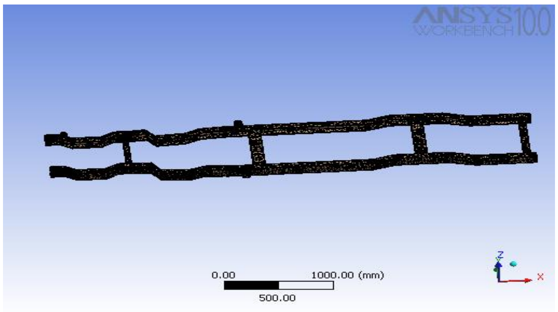

Fig.18. Meshing of the Chevy Truck Chassis

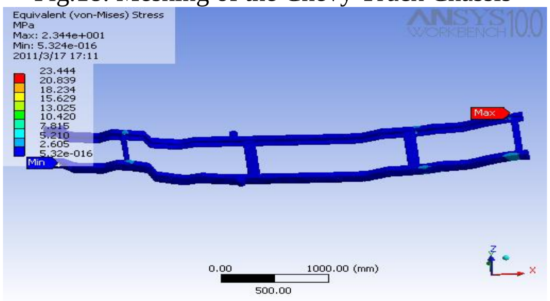

Fig. 19. Von misses Stress on Chevy Truck Chassis

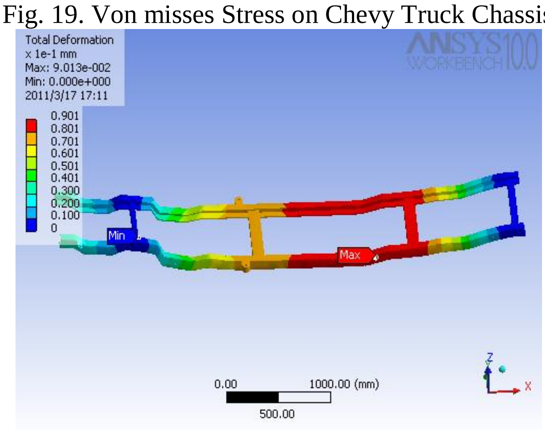

Fig. 20. Total Deformation on Chevy Truck Chassis

III. RESULT & DISCUSSION

Studies have been done to obtain a better chassis design for an off-road vehicle. The torsion stiffness and modal parameters were determined by using ANSYS method and then used to validate the finite element model and finally the chassis was analyzed to increase the structural stiffness. It was noted that the stress found in the above analysis figures are maximum at the joints so there is a need to increase the joint efficiency, and the deformation is maximum at the side member i.e. C frame member so it is required to improve the stiffness of the side member. The stiffness of the side member can be increased by using more thick C frame and with more durable deign as the chassis body is made in such a manner that the front part is narrower then end part. But as the thickness is increases the weight of the chassis is also increases.

The analysis model of Chevy truck chassis consist of 57519 nodes and 33523 elements. The element used is Quadrilateral Dominant. The structural load acting was 4888N and the reaction forces we get are 6321 N.

This simulation is based on the condition of the truck being stationary. The ladder frame chassis was treated as a simply supported beam and loads were due to the weight of components i.e. the pay load is applied to the beam. The support loads from the axles were distributed through spring hangers. The axle’s reaction loads were obtained by resolving forces and taking moments from the weights and positions of the components.

IV. CONCLUSION

It was shown that an FEA model could be used to simulate the automotive chassis accurately by verification of stresses using the beam model. In this paper the analysis of different frames automotive chassis.

In this analysed the kit car chassis under static loading condition the analysis is being validate with the cantilever beam analysis. The analysis is purely static and the load is applied is same as compared to the work done in the analysis of beam. The result show the maximum stress at the point of contact and the bending moment can be understood by the fig. 14. This is for monocoque frame.

While in Chevy truck chassis, analysis is done for stresses due static loads. The analysis is done for the determination of the static behavior of the truck chassis, investigating the mounting locations of components on the truck chassis and observing the responses. It is found that to reduce the stress concentration at the joint the joint must be proper and the reinforcing plate should be more for increasing the twisting moment and the bending moment of the frame. With the increasing side members result will be better than the usual one. But it will also increase the overall weight of the chassis frame and it is recommended to use where the joints or other mounting are to be placed and this will increase the stiffness and will help to reduce the stress in the chassis. We can even use the connection plates for reducing the stresses in both the side members if the change of side member thickness is not possible then by choosing an optimum connection plate length seems to be practical solutions for decreasing the stress values.

The work also content the analysis of kit car chassis and the Chevy truck chassis where we have used the same frame and we found that for car and truck chassis the stress is higher at the joints. How to reduce the maximum stresses and how to get the better chassis. The static analysis of the model is done and the whole FEA program is prepared for the analysis of chassis.

REFERENCES

[1] M. P. Bendsoe, N. Kikuchi; (1988), generates optimal topologies in structural design using a homogenization method, Comp. Meth. Appl. Mech. Eng. 71, 197–224.

[2] CicekKaraoglu, N. SefaKuralay;(2002), Stress analysis of a truck chassis with riveted joints, Finite Elements in Analysis and Design 38, 1115–1130.

[3] M. Tanaka, H. Miyazava, E. Asaba, K. Hongo; (1999), Application of the finite element method to bolt-nut joints-fundamental studies on analysis of bolt-nut joints using the finite element method, Bull. JSME 24, 1064–1071.

[4] M.Zehsaz, F. VakiliTahami and F.Esmaeili; (2009), the effect of connection on stress of truck chassis with riveted and welded joints under dynamics loads. Asian journal of applied sciences 2 (1):22-35, ISSN 1996-3343.

[5] C. J. Cooke; (2005-06), Torsional stiffness analysis of car bodies. Proc.Inst. Mech. Engrs. 180.

◈◈◈