International Journal of Research in Advent Technology (IJRAT)
(E-ISSN: 2321-9637)
Special Issue National Conference "CONVERGENCE 2018", 09 th April 2017

Structural Stress Analysis of an Automotive Vehicle Chassis by Finite Element Method

Miss.Shraddha Kuthe[1], Prof. K.R. Sontakke[2],
Department of Mechanical Engineering[1,2], PLITMS Buldana[1,2]
Email: shraddhakuthe@gmail.com[1] ,krs_mech@plit.ac.in[2]

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 Design of the Chassis of vehicles can be analyzed for *Structural strength* to check for vulnerable points where stress will be high, evolves new creative area in which body is analyzed using finite element method for checking out the high point of von misses stress and bending of the frames. The results illuminate the new creative ways for optimum frame design which makes it more sustainable for structural concerns. The method used is numerical analysis is finite element technique to find the critical stress. In this dissertation work we have analyzed the monocoque 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.
Keywords:-Chassis of vehicle, stress on chassis, chassis strength analysis, automotive frame, ladder frame.

1. INTRODUCTION

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]**. The chassis frame assures the supporting connection between front and rear shaft; it takes over all kind of stress which appear during driving traction, bend, oscillation and torsion. The conditions provided for a chassis frame of an off-road motor vehicle are:

- reduced mass, high stiffness to support stress;
- Simple construction which allows an easy mounting and fixing of other units and body;
- the diversity of mounting possibilities shall assure an optimal arrangement of the motor vehicle's components so that the centre of gravity is low and maintains the prescribed ground clearance level; - a low prime cost;- an increased flexibility to twisting strength **Ion Florin Popa et al. [2].**

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 **Cicek Karaoglu, et al. [3]**. The chassis frame includes cross-members located at critical stress points along the side members. To provide a rigid, box-like structure, the cross-members secure the two main rails in a parallel position. The cross-members are usually attached to the side members by connection plates. The joint is riveted or bolted in trucks and is welded in trailers. When rivets are used, the holes in the chassis frame are drilled approximately 1/ 16 in larger than the diameter of the rivet **H.J.Beermann [4].** The rivets are then heated to an incandescent red and driven home by hydraulic or air pressure. The hot rivets conform to the shape of the hole and tighten upon cooling. An advantage of this connection is that it increases the chassis flexibility. Therefore, high stresses are prevented in critical area. The side- and cross-members are usually open-sectioned, because they are cheap and easily attached with rivets

An experimental and numerical analysis of riveted joints was studied by **C.P.Fung and smart [6].** A three-dimensional finite element model of a riveted lap joint was formulated with shell elements and elastic supports to allow for simulation of various levels of load transfer **G.Harrish, et al. [7]. Y. Xiong [8]** used analytical and numerical methods, for the stress analysis of riveted lap joints in Aircraft structures. In the numerical method, finite element analyses were conducted using some commercial packages. Nut and bolt construction are also used in chassis frame as this allows easy removal of components for repair or replacement purposes.

As far as joint modeling techniques are concerned, there are many studies to determine the joint stiffness in bolted connections through detailed finite element modeling.

2. LITERATURE REVIEW

2.1. Introduction

In present day automobile market, with increased emphasis on economy, the need to design structurally efficient cars has become important ant it is becoming increasingly necessary for the designer to be able to analyse a projected structure at the design stage. An automobile application of classical methods of structural analysis can be both inefficient and laborious **H.Alaylioglu [9].**

2.2. Chassis Frame

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. Chassis is generally the usually steel frame, supported on springs and attached to the axles, that holds the body and motor of an automotive vehicle.

Basic (Stripped Chassis) – an incomplete vehicle, without occupant compartment, that requires the addition of an occupant compartment and cargo-carrying, work performing, or load – bearing components to perform its intended functions.

Cutaway chassis- an incomplete vehicle that has the back of the cab cut out for the intended installation of a structure that permits access room the driver's area to the back of the completed vehicle **D.T.Anderson and B.mills [10].**

The chassis systems include

- The frame – structural, load carrying component that supports the car's engine and body, which are in turn supported by the suspension.

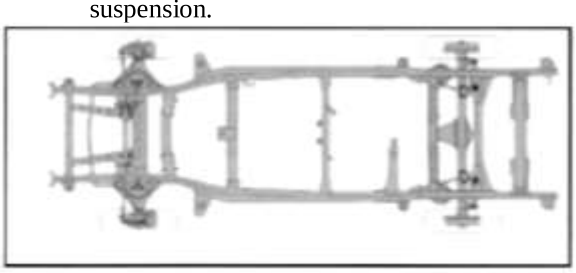

Fig 2.1. Chassis for Toyota Innova MPV

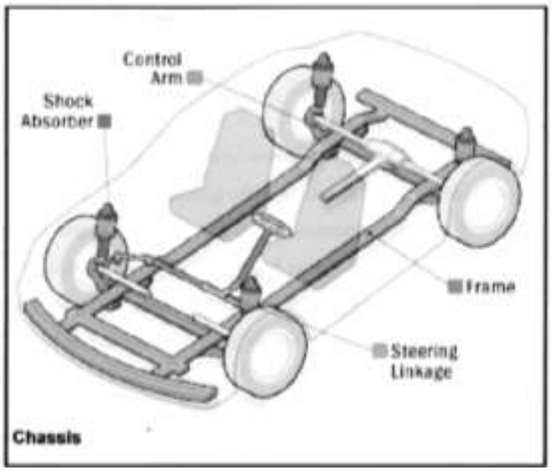

Fig. 2.2.: Basic Chassis Structure

2.3 Chassis Frame and Body

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 Tyres, 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. Layout of Chassis and its main Components is shown in fig 2.3.

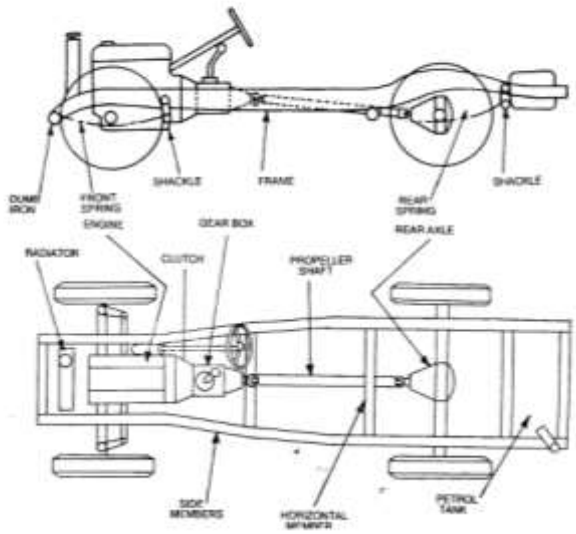

Fig. 2.3. Layout of Chassis and its main Component

The following main components of the Chassis are

1. *Frame:* it is made up of long two members called side members riveted together with the help of number of cross members.
2. *Engine or Power plant:* It provides the source of power

3. *Clutch:* It connects and disconnects the power from the engine fly wheel to the transmission system.
4. *Gear Box*
5. *U Joint*
6. *Propeller Shaft*
7. *Differential*

2.4 Functions of the Chassis Frame

1. To carry load of the passengers or goods carried in the body.
2. To support the load of the body, engine, gear box etc.,
3. To withstand the forces caused due to the sudden braking or acceleration.
4. To withstand the stresses caused due to the bad road condition.
5. To withstand centrifugal force while cornering

2.5 Types of Chassis Frames

There are three types of frames

1. Conventional frame

2. Integral frame

3. Semi-integral frame

Conventional frame: It has two long side members and 5 to 6 cross members joined together with the help of rivets and bolts. The frame sections are used generally.

a. Channel Section - Good resistance to bending

b. Tabular Section - Good resistance to Torsion

c. Box Section - Good resistance to both bending and Torsion

Integral Frame: This frame is used now days in most of the cars. There is no frame and all the assembly units are attached to the body. All the functions of the frame carried out by the body itself. Due to elimination of long frame it is cheaper and due to less weight most economical also. Only disadvantage is repairing is difficult.

Semi - Integral Frame: In some vehicles half frame is fixed in the front end on which engine gear box and front suspension is mounted. It has the advantage when the vehicle is met with accident the front frame can be taken easily to replace the damaged chassis frame. This type of frame is used in FIAT cars and some of the European and American cars **Keith J. Wakeham [12].**

- The suspension system – setup that supports weight absorbs and dampens shock and helps maintain tire contact.
- The steering system – mechanism that enables the driver to guide and direct the vehicle.
- The tires and wheels – components that makes vehicle motion possible by way of grip and or friction with the road **C.C.Swan et al. [11].**

3. FINITE ELEMENT ANALYSIS OF CHASIS

Certain steps in formulating a finite element analysis of a physical problem are common to all such analyses, whether structural, heat transfer, fluid flow, or some other problem. These steps are embodied in commercial finite element software packages (some are mentioned in the following paragraphs) and are implicitly incorporated in this text, although we do not necessarily refer to the steps explicitly in the following chapters. The steps are described as follows.

1. Pre-processing

The pre-processing step is, quite generally, described as defining the model and includes Define the geometric domain of the problem. Define the element type(s) to be used. Define the material properties of the elements. Define the geometric properties of the elements (length, area, and the like). Define the element connectivity's (mesh the model). Define the physical constraints (boundary conditions). Define the loadings. The pre-processing (model definition) step is critical. In no case is there a better example of the computer-related axiom "garbage in, garbage out." A perfectly computed finite element solution is of absolutely no value if it corresponds to the wrong problem.

2. Solution

During the solution phase, finite element software assembles the governing algebraic equations in matrix form and computes the unknown values of the primary field variable(s). The computed values are then used by back substitution to compute additional, derived variables, such as reaction forces, element stresses, and heat flow. As it is not uncommon for a finite element model to be represented by tens of thousands of equations, special solution techniques are used to reduce data storage requirements and computation time. For static, linear problems, a *wave front solver,* based on Gauss elimination, is commonly used.

3. Post processing

Analysis and evaluation of the solution results is referred to as *post processing*. Postprocessor software contains sophisticated routines used for sorting, printing, and plotting selected results from a finite element solution. Examples of operations that can be accomplished include Sort element stresses in order of magnitude. Check equilibrium. Calculate factors of safety. Plot deformed structural shape. Animate dynamic model behaviour. Produce colour-coded temperature plots. While solution data can be manipulated many ways in post processing, the most important objective is to apply sound engineering judgment in determining whether the solution results are physically reasonable

3.1 Boundary Condition for Kit Car Chassis

The kit car chassis used for the dissertation work is shown in fig.5.2. It has the two side rails as shown in fig. 5.2. 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.

3.2 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 i.e. PIPE 16 as this element is suitable for such type of analysis and the analysis is firstly validate by using the cantilever beam.

PIPE 16 is Elastic Straight Pipe. PIPE16 is a uni - axial element with tension-compression, torsion, and bending capabilities. The element has six degrees of freedom at two nodes: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z axes. This element is based on the 3-D beam element (BEAM4), and includes simplifications due to its symmetry and standard pipe geometry.

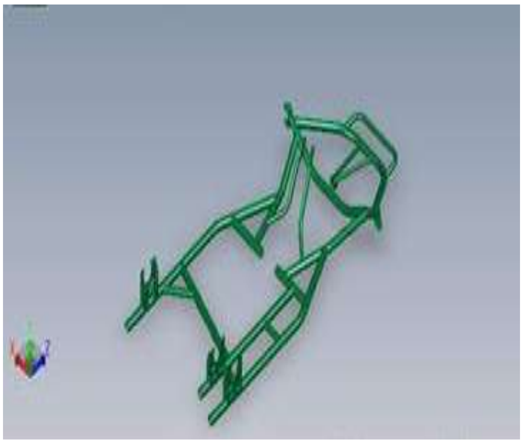

Fig. 3.1. Model of Kit Car Chassis in Solid Works

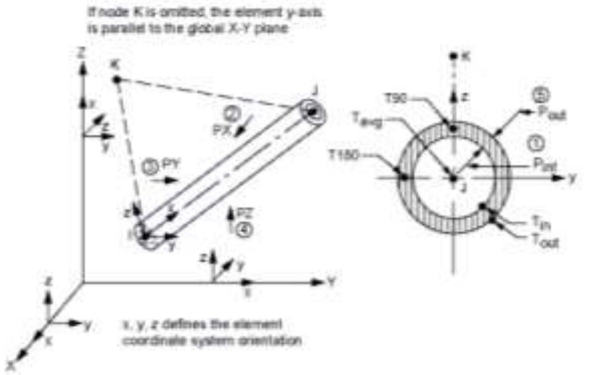

Fig.3.2 PIPE16 Geometry

The input data require for PIPE 16 elements is The geometry, node locations, and the coordinate system for this element are shown in Fig.5.3.: "PIPE16 Geometry". The element input data include two or three nodes, the pipe outer diameter and wall thickness, stress intensification and flexibility factors, internal fluid density, exterior insulation density and thickness, corrosion thickness allowance, insulation surface area, pipe wall mass, axial pipe stiffness, rotor dynamic spin, and the isotropic material properties.

The element X-axis is oriented from node I toward node J. For the two-node option, the element Y-axis is automatically calculated to be parallel to the global X-Y plane. Several orientations are shown in Fig.5.3.: "PIPE16 Geometry". For the case where the element is parallel to the global Z-axis (or within a 0.01 percent slope of it), the element Y-axis is oriented parallel to the global Y-axis (as shown). For user control of the element orientation about the element X-axis, use the third node option. The third node (K), if used, defines a plane (with I and J) containing the element X and Z axes (as shown). Input and output locations around the pipe circumference identified as being at 0° are located along the element Y-axis, and similarly 90° is along the element Z-axis.

The stress intensification factor (SIF) modifies the bending stress. Stress intensification factors may be input at end I (SIFI) and end J (SIFJ), if KEYOPT(2) = 0, or determined by the program using a tee-joint calculation if KEYOPT(2) = 1, 2, or 3. SIF values less than 1.0 are set equal to 1.0. The flexibility factor (FLEX) is divided into the cross-sectional moment of inertia to produce a modified moment of inertia for the bending stiffness calculation. FLEX defaults to 1.0 but may be input as any positive value.

The element mass is calculated from the pipe wall material, the external insulation, and the internal fluid. The insulation and the fluid contribute only to the element mass matrix. The corrosion thickness allowance contributes only to the stress calculations. A positive wall mass real constant overrides the pipe wall mass calculation. A nonzero insulation area real constant overrides the insulation surface area calculation (from the pipe outer diameter and length). A nonzero stiffness real constant overrides the calculated axial pipe stiffness.

Element loads are described in Node and Element Loads. Pressures may be input as surface loads on the element faces as shown by the circled numbers: "PIPE16 Geometry". Internal pressure (PINT) and external pressure (POUT) are input as positive values. The transverse pressures (PX, PY, and PZ) may represent wind or drag loads (per unit length of the pipe) and are defined in the global Cartesian directions. Positive transverse pressures act in the positive coordinate directions. The normal component or the projected full pressure may be used (KEYOPT (5)). Tapered pressures are not recognized. Only constant pressures are supported for this element.

For piping analyses, the PIPE module of PREP7 may be used to generate the input for this element. KEYOPT (4) is used to identify the element type for output labeling and for post processing operations.

KEYOPT (7) is used to compute an unsymmetrical gyroscopic damping matrix (often used for rotor dynamic analyses). The rotational frequency is input with the SPIN real constant (radians/time, positive in the positive element x direction).

The analysis carried out is for linear static analysis and here the section of the chassis is considered as the beam and the loads are actual that are acting on the chassis due to the weight of engine, suspension and other mounting on the chassis. The linear static stress analysis to look into the stress distribution and deformation pattern of the chassis under static load. The boundary conditions for static analysis, used is to represent the real operation environment of the truck chassis. The pinned boundary condition is applied to the suspension mounting-bracket of the chassis since these locations do not allow any translation but allow only rotation about the axis.

In static mode, when the truck is stationary, the loads from the weights of the components like cab, engine, gear box, fuel tank, exhaust and payload is applied to the mounting brackets of the components. Meanwhile the pinned constraints are applied to the suspension mounting brackets. In a symmetry load case which simulates both the truck's front wheels hitting a hump simultaneously, the resulting load is applied to the front suspension mounting brackets. The pinned constraints are applied to the rear suspension mounting brackets. In asymmetry load case which simulates one of the front truck's wheel hitting a hump, the resulting load are applied to the front left suspension mounting brackets with other suspension mounting brackets constraint from any translation.

4 ANALYSIS OF KIT CAR CHASSIS

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 chassis we considered here for analysis is a kit car chassis Shown in fig.5.2. The model of the chassis is formed in solid works but to import the model in Ansys showing the data loss, for this reason the chssis is model in Ansys using the tools of Ansys.

The validation program contains the length 1380 mm beam length and it's a cantilever beam. The analysis with the element PIPE 16, and the nodes are 68 and the elements are 69. The Load acting on the beam is shown in fig. 5.5. and the fig 5.6. shows the deflection of the beam and the fig.5.7. shows the stress on the beam and the fig.5.9. shows the bending moment of the beam. This values is being validate with the analytical formula also.

$$= \frac{3}{3} = \frac{(100)(1380)^3}{3(30000)(\frac{1}{4}(9.5^4 - 7.5^4))} = 584.4$$

$$= \ = \frac{(100)(1380)}{(\frac{1}{4}(9.5^4 - 7.5^4))} = 262.53$$

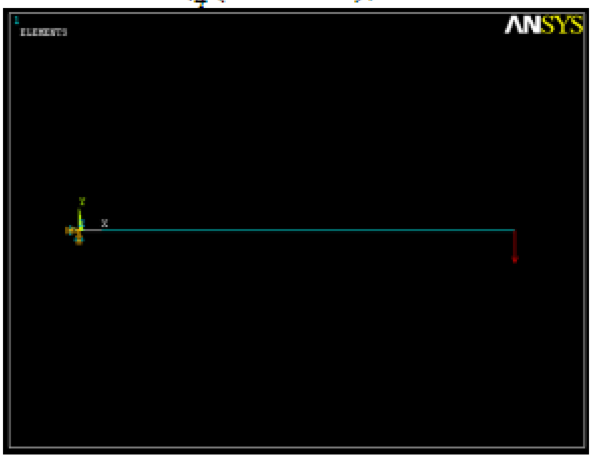

Fig .4.1. Load Applied on Beam

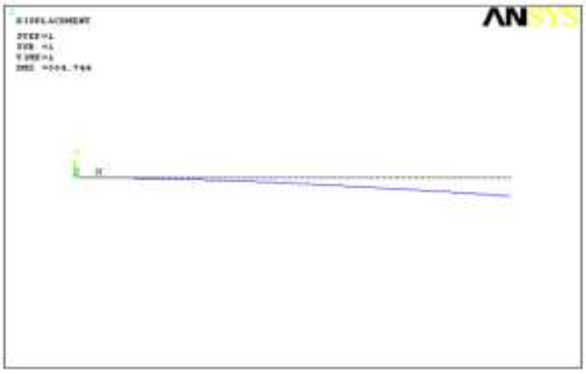

Fig.4.2. Deformation of beam

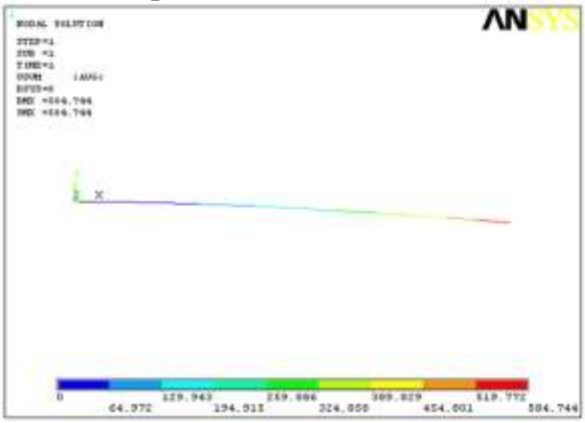

Fig.4.3. Deflection in Beam

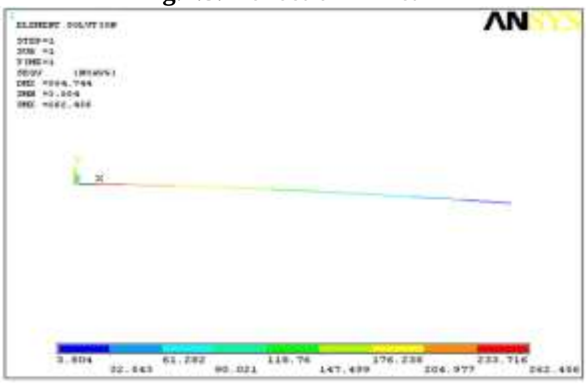

Fig.4.4. Stress Acting on Beam

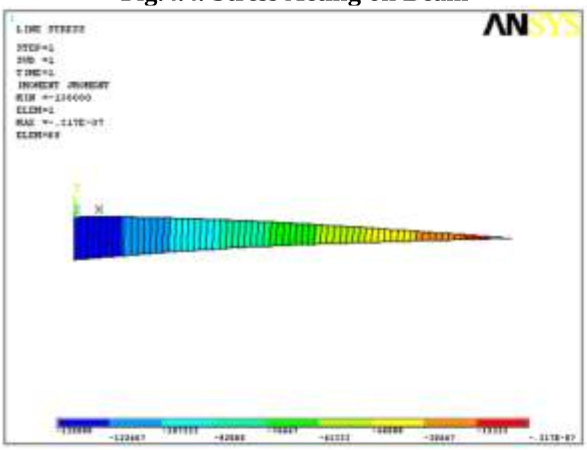

Fig.4.5. Bending Moment on Beam

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 5.9. The analysis contain 237 nodes and 270 elements. Following figure shows the analysis of the chassis of kit car.

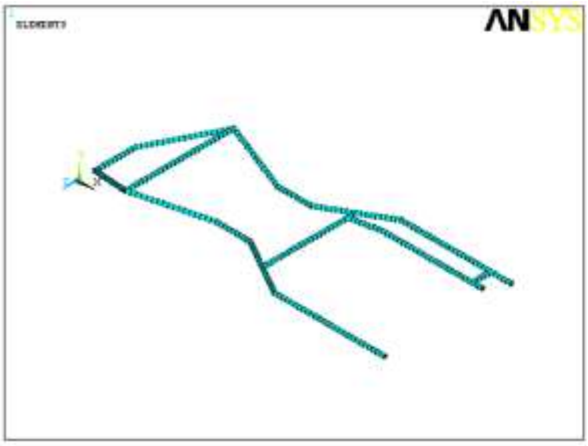

Fig. 4.6. Model of Kit Car Chassis Build in Ansys

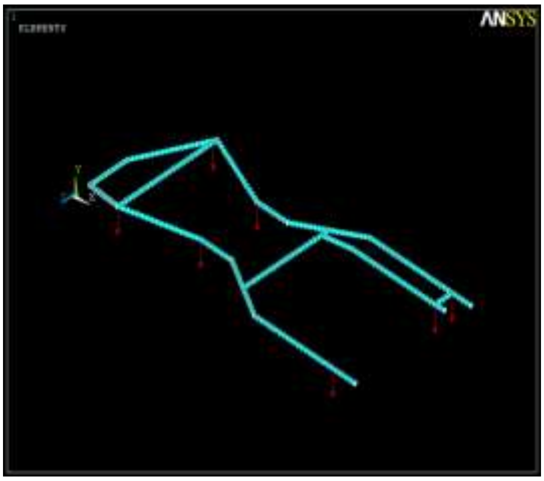

Fig 4.7. Load on the Kit Car Chassis

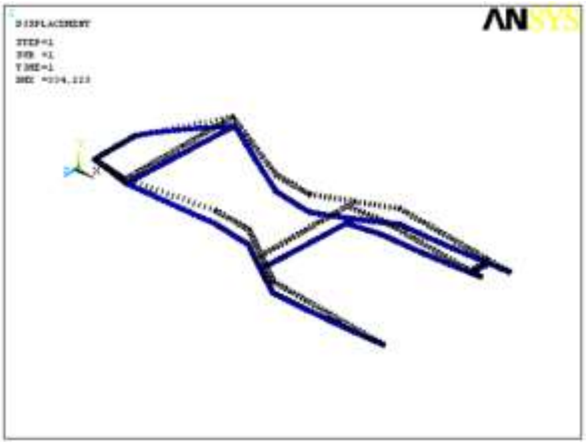

Fig.4.8 Deformation in Chassis

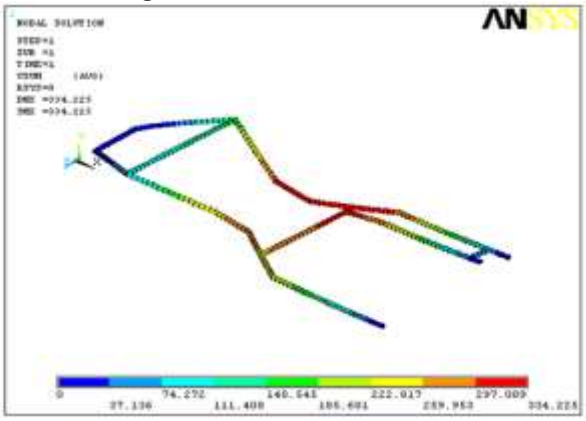

Fig.4.9. Deflection of Chassis

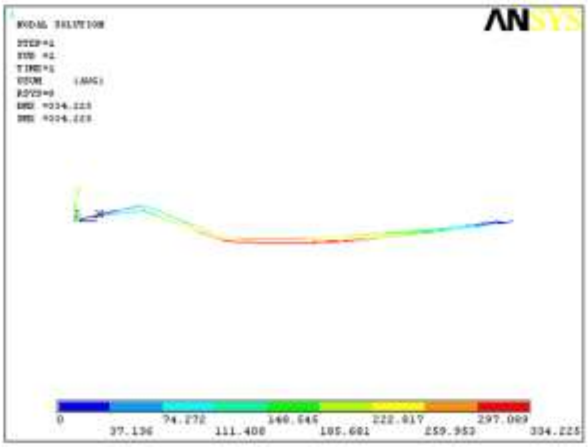

Fig.4.10. Deflection of Chassis Front View.

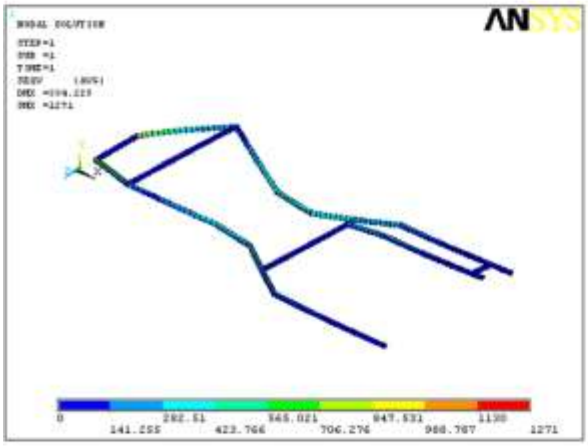

Fig. 4.11. Stress Acting at the Nodal Points

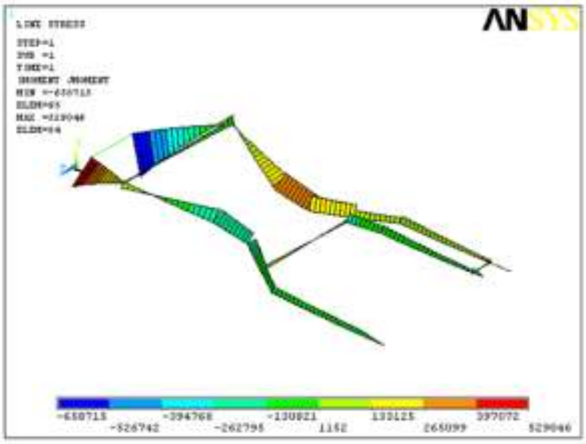

Fig.4.12.Bending moment diagram

5 RESULTS AND DISCUSSION

The above analysis study is done on the kit car chassis formed by monologue frame and the analysis is done to obtain the better strength chassis for an off road vehicle. The stresses found in the above analysis are 1271 N as the maximum stress. The above figures show the details analysis of the kit car chassis and the members of chassis are joined with the round tube. The structural stiffness of the chassis is analyzed and it is found that to increase the structural strength the joints must be proper and if we increase the diameter of the pipe then the result we found is better. The diameter of tube we have taken is 19 O.D. and if we increase the O.D. by 2mm we found that the result are more stiffer than the previous one. The chassis frame can have better strength and stiffness if we provide more tube in between the side members. The simulation we have done is purely static in nature and the loads were due to the weight of the components. The reaction we found at various nodes are

Table 5.1. Reaction on nodes

NO		**73**	**86**		
DE	**1**	**-177.23**	**177.23**	**179**	**184**
FX	**559.**	**2026.8**	**2434.5**	**-**	**136**
FY	**42**	**512.68**	**-463.99**	**16.1**	**9.4**
FZ	**-**	**-17688.**	**0.40645**	**29**	**-**
MX	**12.1**	**-84007.**	**E+06**	**53.8**	**90.3**
MY	**66**	**0.54380**	**50931.**	**15**	**42**
MZ		**E+06**	**0.53055**		
			E+06		

REFERENCES

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

[2] Ion Florin Popa And Romania, Designing a Chassis-Type Structure Form using Beams of Equal Resistance, Proceedings of the 12th WSEAS International Conference on Automatic Control, Modeling & Simulation.

[3] Cicek Karaoglu, N. Sefa Kuralay; (2002), Stress analysis of a truck chassis with riveted joints, Finite Elements in Analysis and Design 38, 1115–1130.

[4] H.J. Beermann; (2009), The Analysis of Commercial Vehicle Structures, Mechanical Engineering Publication Limited, London.

[5] J.W. Fitch, 2001, Motor Truck Handbook, Warrandale, USA.

[6] C.P. Fung, J. Smart; (2004), An experimental and numerical analysis of riveted single lap joints, J. Aerospace Eng. 208, 70–79.

[7] G. Harish, T.N. Farris; (2007), Modeling of skin=rivet contact: application to fretting fatigue, collection of Technical Papers, Proceedings of 38th AIAA=ASME=ASCE Structures, Structural Dynamics and Material Conference, Vol. 4, pp. 2761–277.

[8] Y. Xiong; (2009), Analytical and finite element modeling of riveted lap joints in aircraft structure, AIAA J. 37 - 93.

[9] H. Alaylioglu and R. AliAnalysis; (2008), of an automotive structure using hybrid stress finite elements, Computers and structure vol.8, pp , 237-242 pregamon press, printed I great Britain.

[10] C.C. Swan I. Kosaka; (2007), Reuss and Voigt mixing rules for variable topology material layout: linear elasticity, Int. J. Numer. Methods Eng. 40 3033–3057.

[11] D.T. Anderson and B.Mills; (2002), dynamic analysis of a car chassis frame using the finite element method, Int.J.Mech.sci.pergamon press. Vol.32, pp. 799-808. Printed in great Britain.

[12] Keith J. Wakeham; (2 January 2009), Introduction To Chassis Design Revision 1.0, Mechanical Engineering Undergraduate Memorial University of Newfoundland And Labrador.