International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 05 Issue: 03 | Mar-2018 www.irjet.net p-ISSN: 2395-0072

# DESIGN AND STRUCTURAL ANALYSIS OF AN OFF ROAD VEHICLE

**P. Rastogi[1*], I. Sharma[1], H. Bindal[1]**

[1] *Department of Mechanical Engineering, University of Petroleum and Energy Studies, Dehradun-248007,India*

---

**ABSTRACT:** *This project deals with designing a frame for an ATV and optimizing it, accompanied with a number of iterations. The iterations are done to reduce the stress concentration regions by providing a no. of load paths in the frame keeping the constraint that the weight of the frame should be as less as possible. The reduction of gross weight of the vehicle leads to better fuel economy. To check whether the frame comply with the safety standards different tests including frontal, rear and side impact tests have been performed on the frame. Software used to accomplish the task is CATIA v5 for cad modeling and ANSYS for simulation purpose. The simulation done is 1-D analysis as it's calculation in the blackboard (algorithm defined by the coder) are quite simple and takes less time to process. The advantage for performing FEA analysis on the frame is that it's less time consuming and reduces the cost of the test. Results demonstrate that, increasing the thickness of hollow tubular members decreases the stress concentration. Also, by providing alternate pathway for load decreases the stress concentration in a critical region of failure.*

**Keywords: Space frame, CATIA v5, ANSYS, iterations, FEA.**

## 1. INTRODUCTION

A frame forms the basis of a car structure. Frame structure provides the necessary stiffness and strength to the vehicle. Its primary function is to provide mounting points for the steering mechanism, engine, gearbox, suspension and seating for the occupants. Another function is to ensure occupant safety.

The frame is subjected to various loads, therefore, while designing the frame primary step is to identify different loads acting on the vehicle. These loads are:

1. Longitudinal Torsion: These loads occurs due to roll over bumps acting on diagonally opposite front and rear wheels of a vehicle. Torsional loading affects the handling of the vehicle. Here, the frame can be considered as a torsion spring connecting two ends on which suspension load acts.

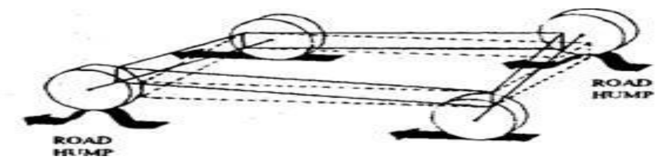


**Fig 1:** Longitudinal torsion

2. Vertical bending: The weight of vehicle components mounted to the frame and driver's weight accounts for vertical bending.

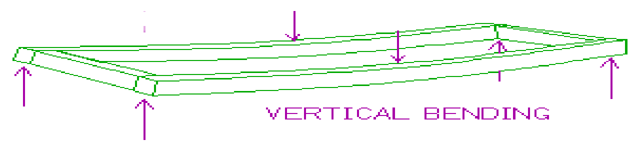


**Fig 2:** Vertical Bending

3. Lateral Bending: This type of bending occurs due to various reasons such as side wind loads, road camber and the centrifugal forces acting due to cornering. These lateral side forces are opposed by adhesive side reactions on the wheels.


**Fig 3:** Lateral Bending

4. Horizontal Lozenging: When diagonally opposite wheels of a vehicle are subjected to forward and backward forces, frame is distorted to parallelogram shape. These loads may be due to vertical variations on the roads.

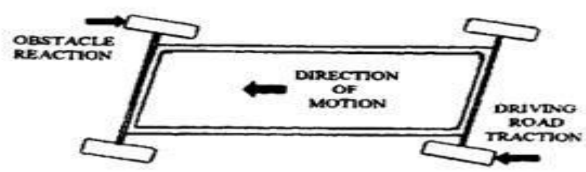


**Fig 4:** Horizontal Lozenging

So, the vehicle must be designed such that it is able to withstand all such loads[1].

Tubular space frame is preferred over other frames as it has *more strength and is lighter in weight than other frame structures* [2]. In tubular 3-D structure frame, hollow tubes are positioned in different directions and welded together which accounts for strength and stiffness. Body panels are just for the sake of aesthetics as they don't take any type of load during its course of usage [3].

After selecting the frame type firstly the material is selected and then CAD model of frame is made using *CATIA v5 Generate Shape Design module* and different tests performed as a part of FEA are:

1. Frontal impact test
2. Rear Impact test
3. Side Impact test

If the intensity of stress concentration is high in critical regions of the frame in the above mentioned tests, then stress concentration in these regions is tried to minimized with introduction of more tubular members in the frame or by replacing the tubular members in high stress concentration zones with greater thickness tubular members as demonstrated by our study.

Similarly, a number of iterations are performed till the maximum stress value in the frame at the critical points reduces to certain degree such that the frame do not fails. These critical points are one of the major reasons for fatigue failure. Life time of the frame can be calculated by stress value [4]. The simulation of the frame is done using FEM software, i.e., ANSYS.

In our current research, we have performed 1-D analysis. So, the meshing and ANSYS solver took less time to process results as the calculations being performed by the algorithm defined in the backstage takes less time to be done.

Physical impact testing of a vehicle is a time consuming and costly task. So, FEM software was introduced as it save money and time both. ANSYS module used in the current project is static structure analysis on which above mentioned safety tests are performed. There are 3 types of elements while performing analysis. 1D elements are preferred when one dimension is greater than other two dimensions. 2D elements are preferred when two dimension are comparable in size and much larger than the third dimension. 3D elements are preferred when all the 3 dimensions are comparable. In case of 1D element, one out of three dimensions is software specified while other two out of three dimensions are specified by the user. In 2D analysis, two out of three dimensions are software specified, while remaining one is user defined. In 3D analysis, all dimensions are software specified and no user input is required [4].

## 2. METHODOLOGY

### 1.1 CAD modeling

The frame is modeled in CATIA v5(Generative Shape Design). Here, triangulation members are provided between longitudinal side members to resist bending of side members.

### 1.2 Material selection

The material selected is AISI 1018. The properties of AISI 1018 are as follows:

**Table 1:** mechanical properties of AISI 1018

| Properties | Value |
|---|---|
| Modulus of elasticity | 205GPa |
| Bulk modulus | 140GPa |
| Shear modulus | 80Mpa |
| Poison's Ratio | 0.29 |
| Density | 7.87 g/cc |
| Yield strength | 370MPa |
| Ultimate tensile strength | 440MPa |

### 1.3 Finite Element Analysis

The wire frame designed in CATIA v5 is exported to ANSYS and tubular member dimensions are specified via ANSYS Design Modeler. Such analysis is called 1D analysis where one dimension is very much larger than other two. So, the larger dimension is software specified (dim. Specified in CATIA v5) while other two are user specified.

### 1.4 Meshing

Meshing is the process of disintegrating a continuous body of infinite D.O.F and the nodes into finite no. of infinitesimally elements having finite D.O.F and nodes. It is the most important step in analysis. If meshing is not done correctly then the results also varies. Meshing converts a non-uniform body to a finite no. of uniform elements. Meshing in 1D analysis takes very less time. Meshing elements are of 3 types:

1. 1D mesh element: line
2. 2D mesh elements: tria and quad
3. 3D mesh elements: tetra and hexa

## 1.5 Boundary conditions and loads

1. Frontal impact test: Rear suspension arm mounting points are fixed and 4G force is applied on the front bar of the frame.
2. Rear impact test: Front suspension arm mounting points are fixed and 4G force is applied on the rear of the frame.
3. Side impact test: Side mounting points of suspension arm are fixed and 4G force is applied on the opposite side.

After specifying boundary conditions and loads, the result parameters which are required must be specified. Then, ANSYS solver , i.e. , ANSYS Multi physics calculates the result.

## 1.6 Iterations

The frame is designed such that the max. stress value at critical points is less than yield tensile strength value of the material. Also, at weld joint it is preferred that the stress value should be less than half of yield tensile strength. Iterations are performed in accordance to avoid any violation of the above stated statements. All the stress value obtained below are in hbar(1hbar=10Mpa).

**Primary Model**

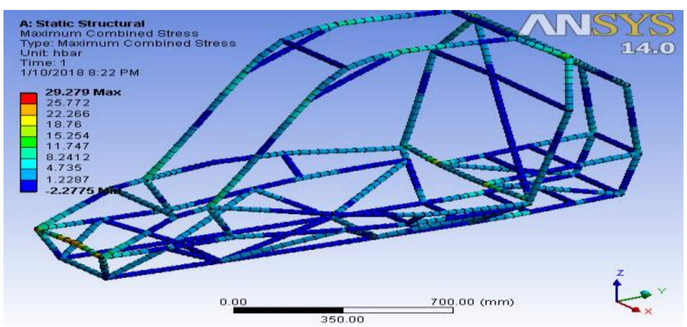


**Fig 5(a)**

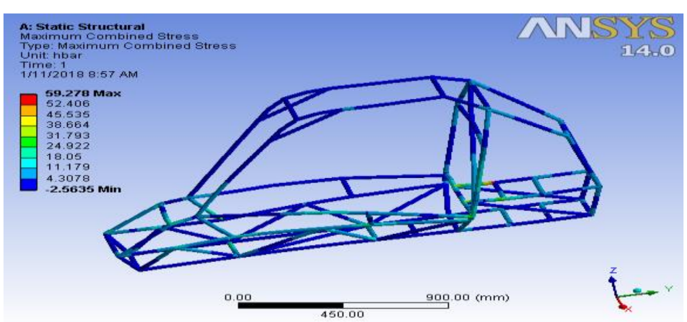


**Fig: 5(b)**

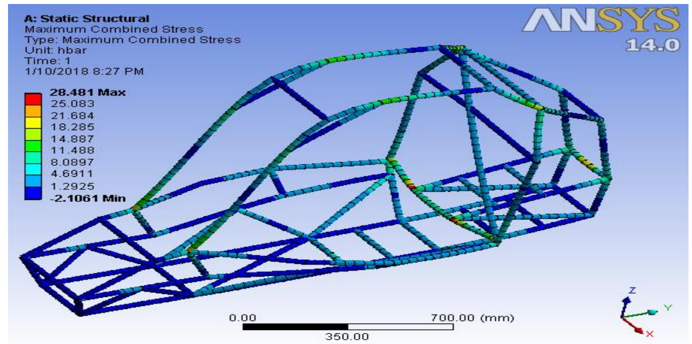


**Fig: 5(c)**

**Figure 5(a),(b) &(c):** frontal, side and rear impact tests of primary model

In side impact test, the maximum value of stress exceeds the yield tensile strength. In rear impact test, maximum stress value is at welded joints which exceeds half the yield tensile strength value. Longitudinal members, vertical members supporting seating and front transverse members are having O.D. (outer diameter) 25.4mm and 2mm thickness. All transverse members, roll over cage and triangulation members are having O.D. 25.4mm and 1.5mm thickness.

**Iteration: 1**

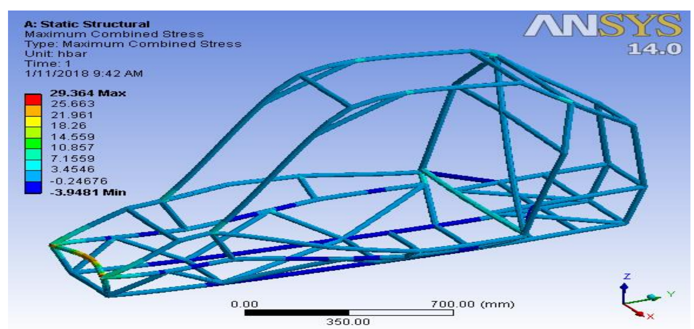


**Fig 6(a)**

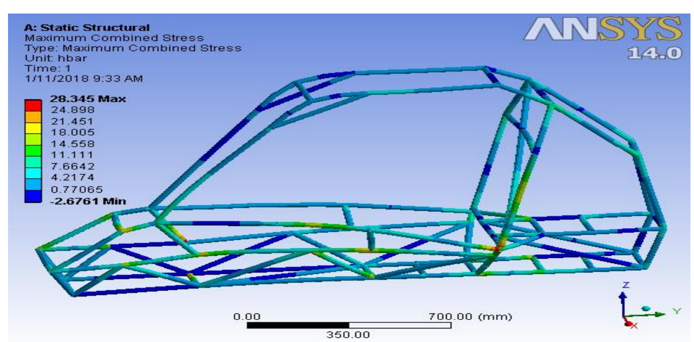


**Fig 6(b)**

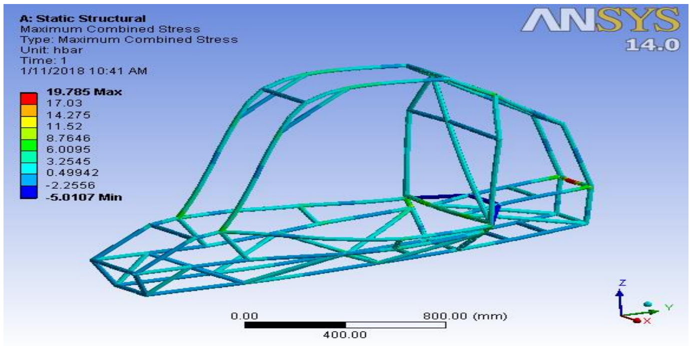


**Fig 6(c)**

**Fig 6(a),(b) &(c):** frontal, side and rear impact tests of 1st iteration model

Here, with introduction of two members A and B side impact test critical points position have changed and so the maximum stress value have decreased drastically (580Mpa to 285 Mpa). These members have O.D. 25.4mm and 1.5mm thickness. Rear most transverse member thickness is increased from 1.5mm to 2mm thickness. So, by increasing thickness of rear most member stress value at critical points decreased to 197Mpa which is less than yield tensile strength value.

### Iteration 2:

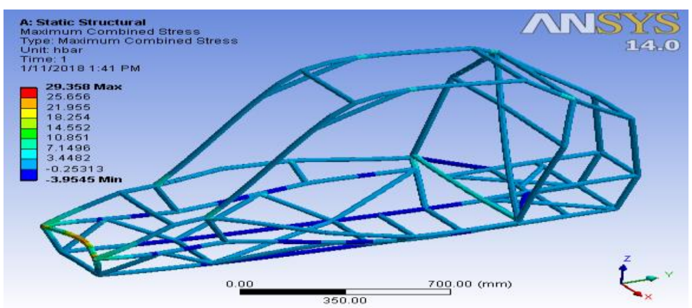


**Fig 7(a)**

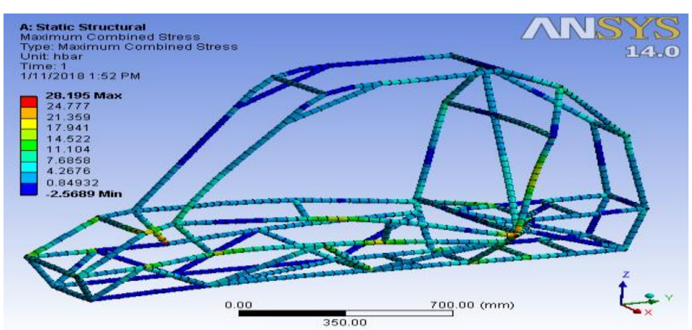


**Fig 7(b)**

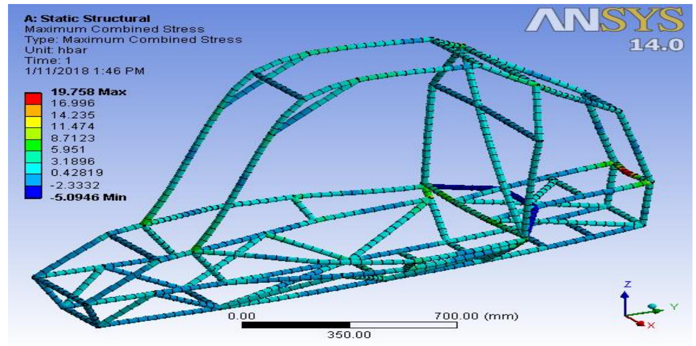


**Fig 7(c)**

**Fig 7(a),(b) &(c):** frontal, side and rear impact tests of 2nd iteration model

With introduction of new transverse diagonal member(C) stress value at critical points in side impact test is reduced to 281Mpa.

### Iteration 3:

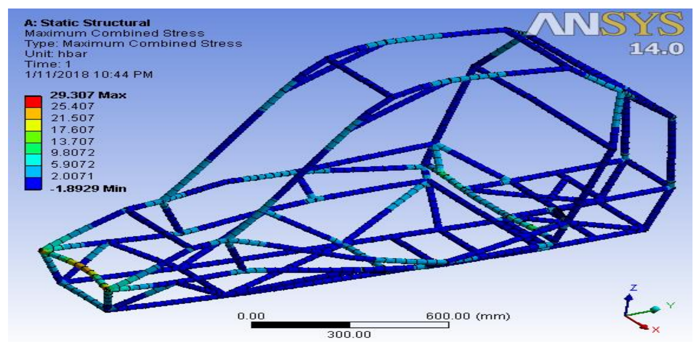


**Fig 8(a)**

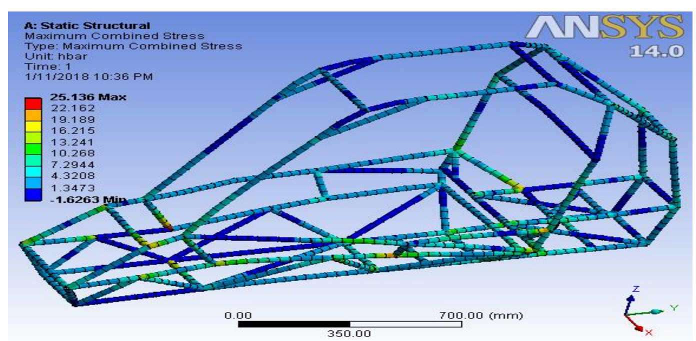


**Fig 8(b)**

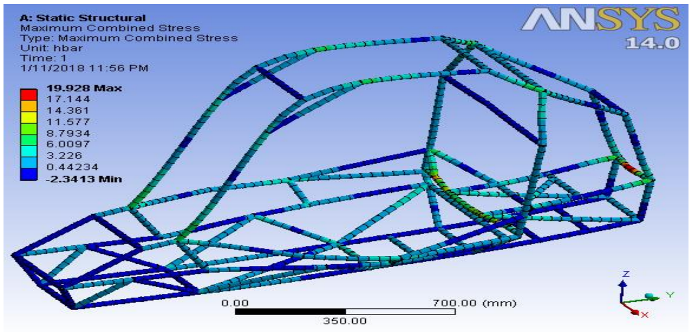


Fig 8(c)

**Fig 8(a),(b) &(c):** frontal, side and rear impact tests of 3rd iteration model

Diagonal member behind driver seat and two inclined members introduced earlier to reduce maximum stress in case of side impact test are eliminated. 6 new inclined members(D and other 3 are same as D but on opposite side) are introduced to decrease stress value at critical points in case of side and rear impact test. These members dimensions are O.D. 25.4mm and 1.5 mm thickness.

### Iteration 4:

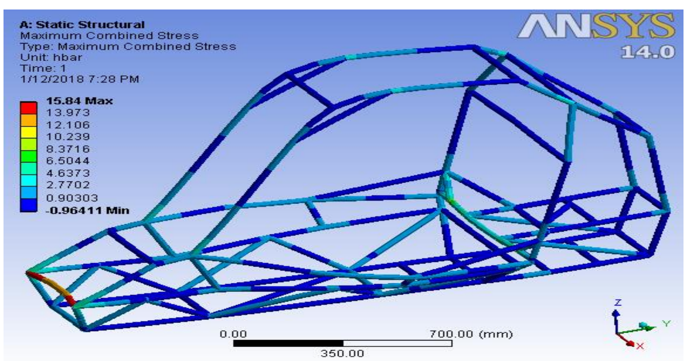


Fig 9(a)

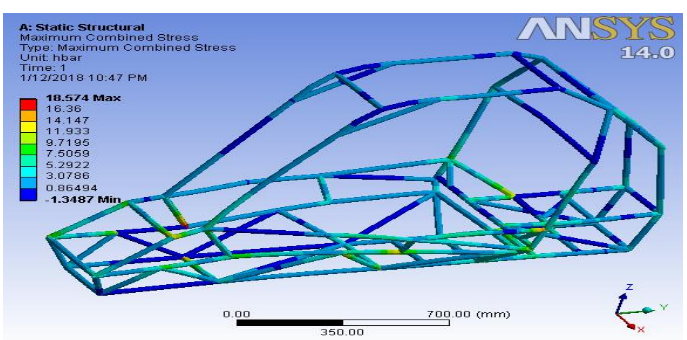


Fig 9(b)

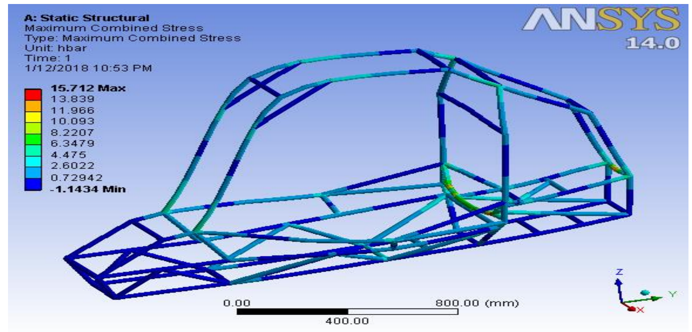


Fig 9(c)

**Fig9(a),(b) &(c):** frontal, side and rear impact tests of 4th iteration model

Here, all members having thickness 2mm was replaced by 3mm thickness and same O.D. 25.4mm. Also, the front most and rear most transverse members were replaced by 3mm thickness tubular members. Side members introduced in previous iteration and side vertical members supporting side longitudinal members were changed to 2mm thickness with same O.D. Here, all the stress value at critical points were under safety consideration but the frame weighs about 50kg which is quite heavy.

### Iteration 5:

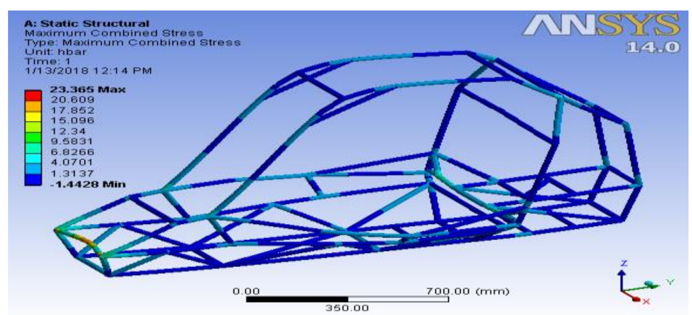


Fig 10(a)

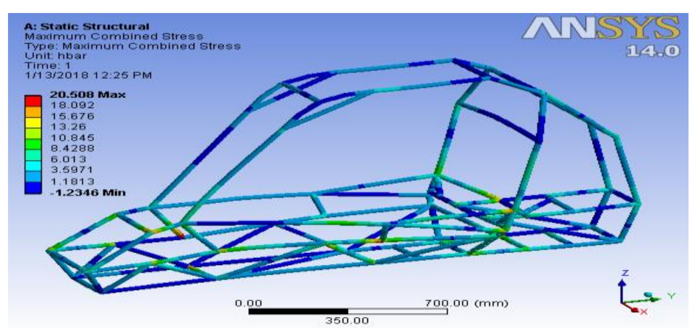


Fig 10(b)

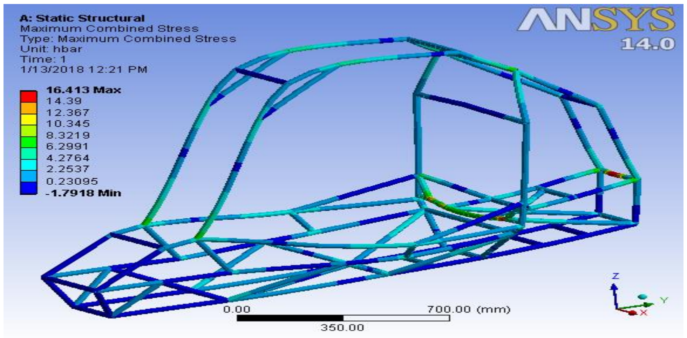


**Fig 10(c)**

**Fig 10(a),(b) &(c):** Frontal, side and rear impact tests of 5th iteration model

Here, all the longitudinal members are made of 2mm thickness. Rear most member is made of 3mm thickness tubular pipe. Diagonal members, side vertical members, side inclined members and lateral members at the base are made 2mm thick with O.D. 25.4mm. Roll cage members are made of 25.4mm O.D. and 1.5 mm thickness. This is done in order to reduce frame weight.

### Iteration 6:

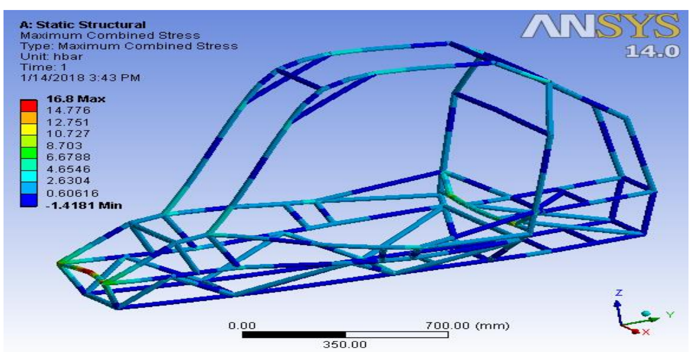


**Fig 11(a)**

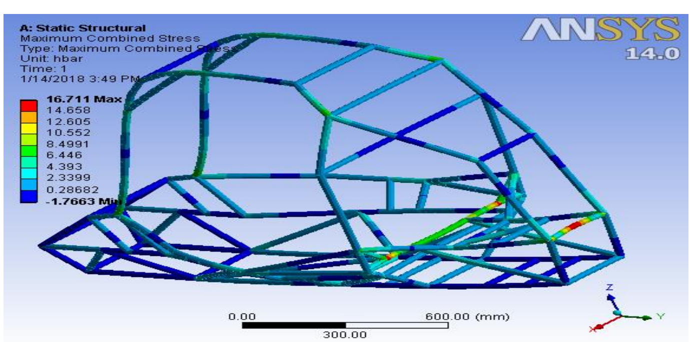


**Fig 11(b)**

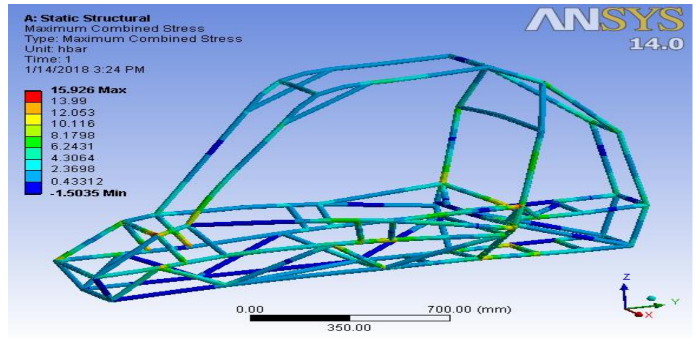


**Fig 11(c)**

**Fig 11(a),(b) &(c):** frontal, side and rear impact tests of 6th iteration model

New, 4 side vertical members are inserted of O.D. 25.4mm and 2mm thickness. Front most member thickness is made 3mm with O.D. same. Transverse member behind driver seat is also made 2mm thick. Now, all the stress value is less than half of yield tensile strength at welded points. So, our frame design is safe.

## 3. RESULTS

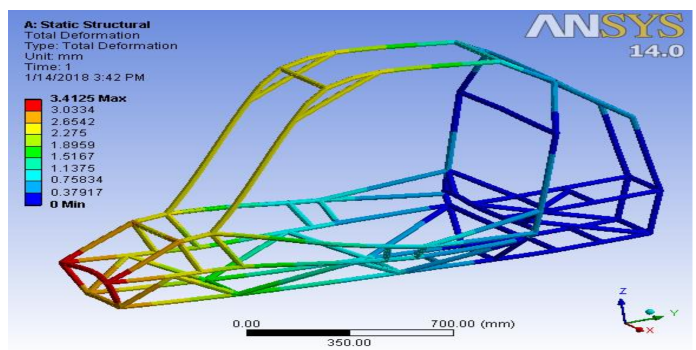


**Fig 12:** Frontal impact total deformation

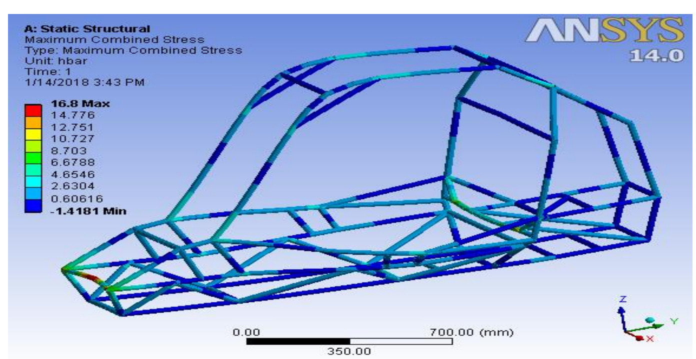


**Fig 13:** Frontal impact stress distribution

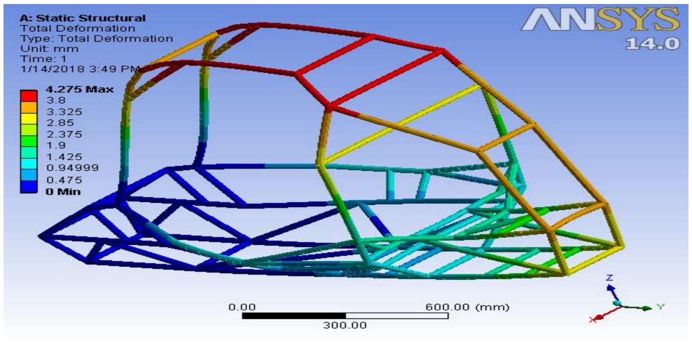


**Fig 14:** Rear impact total deformation

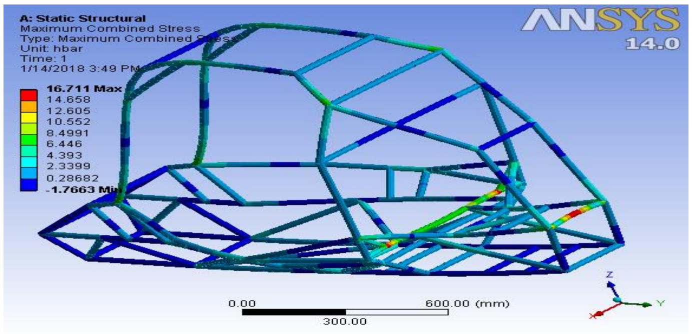


**Fig 15:** Rear impact stress distribution

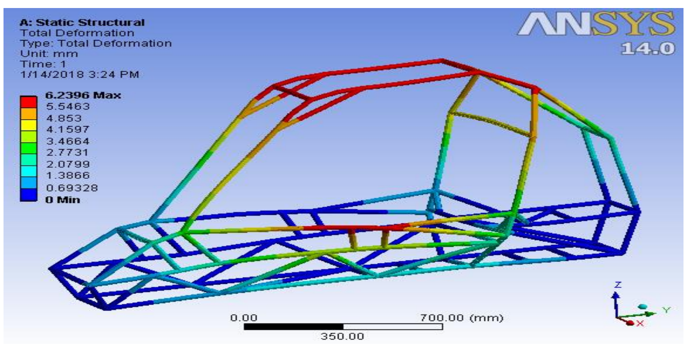


**Fig 16:** Side impact total deformation

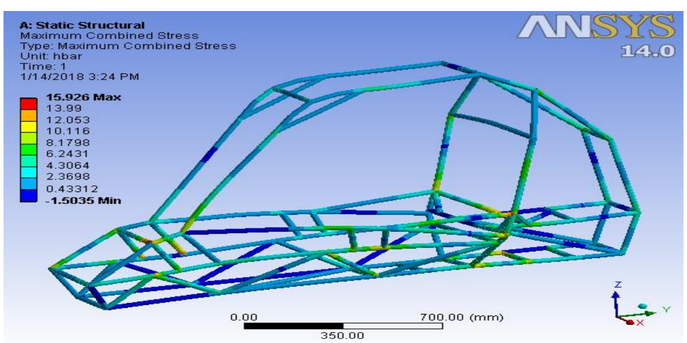


**Fig 17:** Side impact stress distribution

**Table 2:** Results

| | Max. deformation value (mm) | Max. stress value (Mpa) |
|---|---|---|
| Frontal impact test | 3 | 168 |
| Rear impact test | 4.2 | 167 |
| Side impact test | 6.2 | 159 |

## 4. CONCLUSION

The analyses of off road vehicle frame for front impact, rear impact and side impact tests have been performed successfully. All the deformations and critical stress values are under safety limits. The critical stress values at various points can be minimized by increasing the thickness of the tubular pipe and providing an alternating load pathway by introducing new members in the frame. The CAD modeling and analysis of the frame was accomplished using CATIA v5 and ANSYS.

## 5. REFERENCES

[1] Riley WB, George AR. Design Analysis and Testing of a Formula SAE Car Chassis. SAE Technical Paper Series: 013300; 2002.

[2] Lan, T.T. Space Frame Structures. In: Wai-Fah Chen, editor. Structural Engineering Handbook, Boca Raton: CRC Press LLC; 1999.

[3] Patel AS, Chitransh J. Design and analysis of tata 2518TC truck chassis frame with various cross sections using CAE tools. International Journal of Engineering Sciences and Research Technology,2016; 5(9): 692-714.

[4] Gokhale NS, Deshpande SS, Bedekar SB, Thite AN. Practical Finite Element Analysis. 1st ed. Sted: Finite to finite;2008.