INTERNATIONAL JOURNAL OF ADVANCE RESEARCH, IDEAS AND INNOVATIONS IN TECHNOLOGY

IJARIIT

ISSN: 2454-132X
Impact Factor: 6.078

(Volume 7, Issue 3 - V7I3-1775)
Available online at: https://www.ijariit.com

Static Structural Analysis of SAE Baja Chassis

Siddh Shah
siddh.21810348@viit.ac.in
Vishwakarma Institute of Information Technology, Pune, Maharashtra

ABSTRACT

The roll cage of Baja vehicle can be of various types, comprising a tubular structure of various dimensions, but all complying with the official rulebook issued by Baja SAE India. Since the vehicle is expected to perform in off-road conditions, various static and dynamic forces are applied on the chassis, and during these off-road conditions, the main motive of roll cage is to ensure total safety of the driver and to assemble and mount other vehicle components. The chassis of the vehicle is first designed in Solidworks 2018 and then analysed in ANSYS 2019.

Keywords: *Baja Chassis, FEA, Roll Cage Analysis, ANSYS*

1. INTRODUCTION

Baja SAE Indi is a national level undergraduate student competition in which teams from all over India design, analyse, fabricate and test their own All Terrain Vehicle. The competition consists of various static and dynamic events to judge the best team. The vehicle is designed and fabricated in such a way that it can be tested over off-road conditions and hence various forces act on the vehicle. A rigid and strong chassis is required in order to sustain these forces.

The chassis or the roll cage of the vehicle is very important in order to protect the driver from external impact forces as well as the other components and sub-systems of the vehicle.

2. RULES AND CONSTRAINTS FOR THE CHASSIS

According to the rulebook issued by Baja SAE India[1], the tubular frame should be made with a material of minimum strength as AISI 1018 with outside diameter of 25mm and wall thickness of minimum 1.571mm. The bending stiffness and bending strength must be calculated about a central axis which is given by –

A] Bending Stiffness = E * I

Where,

E = Modulus of elasticity
I = Second moment of area

B] Bending Strength = $\frac{Sy * I}{C}$

Where,

Sy = Yield Strength

c = Distance from central axis to extreme point[2]

3. ASSUMPTIONS

A] Material

We have chosen AISI 4130 chromyl steel (Yield Strength = 460MPA) since it has relatively high strength and less density (ρ = 7865 kg/m^3)

B] Weight Distribution

The weight of vehicle is assumed to be 280kg (including driver) for static as well as dynamic conditions. The front part of vehicle consists of everything ahead of RRH and the rear part consists of everything behind RRH. The weight distribution is assumed to be 42% - 58% (Front:Rear).

C] Dynamic Conditions

All types of impacts are considered to be instantaneous. Hence period of impact is assumed to be 0.15s [3] and all the forces acting on the vehicle are instantaneous forces. The maximum velocity of the vehicle during any impact test is assumed to be 18m/s (Approximately 65 km/hr).

4. CAD MODEL

The CAD Model consists of primary, secondary and tertiary members which constrained strictly in accordance with the rulebook issued by Baja SAE India[1]. It is designed in order to minimize material (for weight reduction) and optimize the structure in order to protect the driver and other components of the vehicle in the worst expected scenarios like rollover, front impact, side impact, rear impact and in bump test.

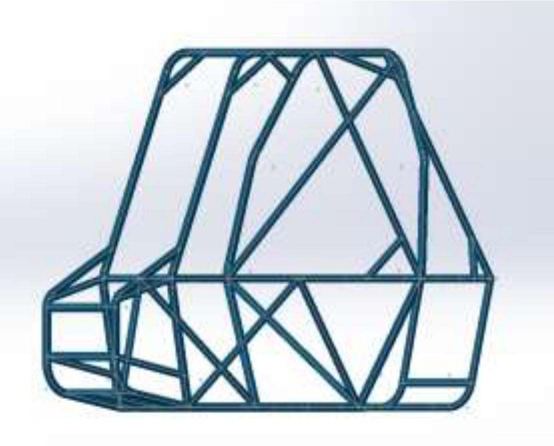

Fig. 1: CAD Model of Chassis on Solidworks 2018

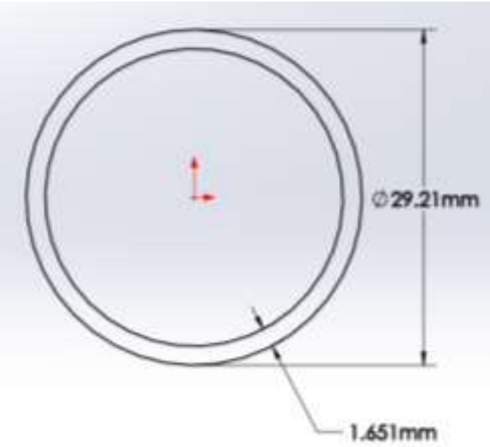

Fig. 2: Cross-section of frame member

5. MESHED MODEL

The finite element mesh is used to subdivide the CAD model into smaller domains called *elements*, over which a set of partial differential equations are solved. These equations approximately represent the governing equation of interest via a set of polynomial functions defined over each element. As these elements are made smaller and smaller, as the mesh is refined, the computed solution will approach the true solution. The selected element is a three-node beam element in 3-D. With default settings, six degrees of freedom occur at each node; these include translations in the x, y, and z directions and rotations about the x, y, and z directions.[4]

Here, we make use of unstructured tetrahedron mesh since there are a lot of curvatures and uniform mesh won't be helpful in this case. Tetrahedron is best for discretizing complex geometries, although it takes more computational time.

No of Nodes – 3,77,167
No of Elements – 1,25,722
Mesh Type – Tetrahedron and Hexahedron

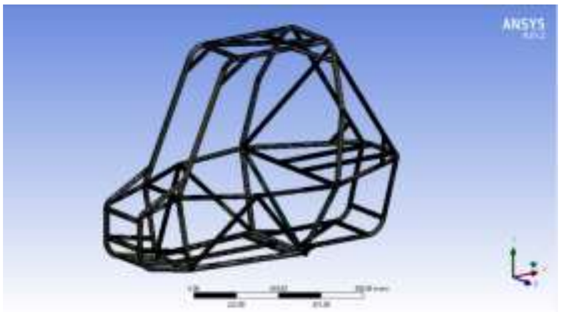

Fig. 3: Meshed Model

6. INFLUENCE OF TYPE OF MESH

The mesh size and type has significant influence on the analysis. Hexahedron type of meshing is more accurate. But the skewness is better in hexahedron meshing only in plane geometries. Tetrahedron type of meshing is done at the curved surfaces. In the chassis, hexahedron type of meshing is done for all the straight members having a length of above 300mm. All the curvatures and small members with lengths less than 300mm are meshed with Tetrahedron Meshing since it reduces errors to a greater extent.[5]

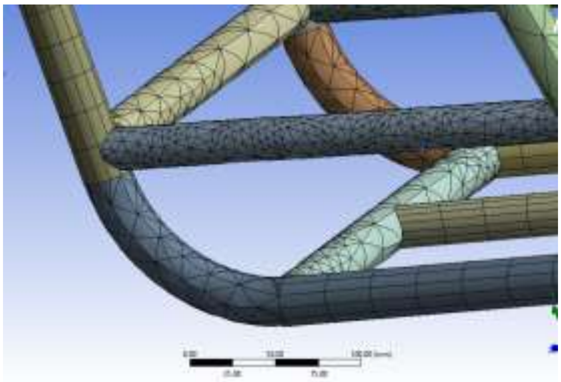

Fig. 4: Detailed view of Mesh

7. MATERIAL PROPERTIES

The material AISI 4130 Chromyl Steel is selected due to its high strength to weight ratio. Following are few characteristics of the selected material –

Table 1: Properties of AISI 4130 Chromyl Steel

Modulus of Elasticity	205GPA
Ultimate Tensile Strength	670MPa
Tensile Yield Strength	460MPa
Carbon Content	0.30%
Density	7865 kg/m^3
Poisson's Ratio	0.285

8. FRONT IMPACT ANALYSIS

Front Impact Analysis is done in order to study the deformation and stresses developed in the vehicle when there is either a sudden impact on the front side of the vehicle or it collides with some other vehicle or something else. The frame is designed so as to have least possible failure and prevent the driver from injuries.

It is assumed that the mass of the vehicle along with driver is 280kg and maximum velocity is 18m/s.

m = 280kg

v = 18m/s

t = 0.15s

$$a = \frac{Vf - Vi}{t} = \frac{0 - 18}{0.15} = 120 m/s^2$$

F = m * a = 280*120 = 33.333kN

F_{fi} = 33333N

The load of 33.333kN is uniformly distributed on the front bracing members, keeping rear suspension points as fixed support.

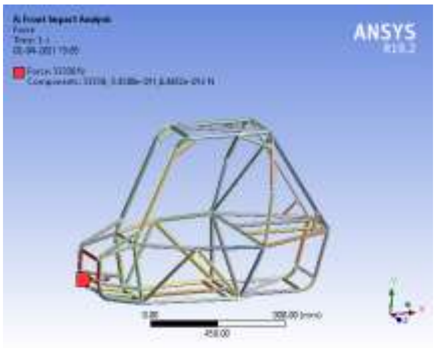

Fig. 5: Boundary Conditions for Front Impact

From the results, it can be seen that the chassis undergoes a maximum deformation of 7.72mm which is acceptable.

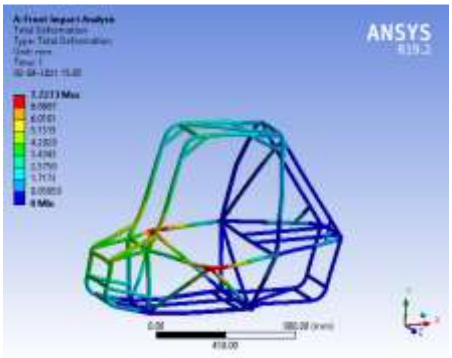

Fig. 6: Results of Front Impact

9. REAR IMPACT ANALYSIS

The Rear Impact Analysis is performed in order to study the deformation and the stress chassis undergoes in the case if there is sudden impact or collision to the vehicle from the rear side. The results are optimized so as to protect the electrical components such as battery pack, controller, motor and electrical harness as well as the powertrain components like gearbox, pulleys and half-shafts.

m = 280kg

v = 18m/s

t = 0.15s

$$a = \frac{Vf - Vi}{t} = \frac{0 - 18}{0.15} = 120 m/s^2$$

F = m * a = 280*120 = 33.333kN

F_{ri} = 33333N

The load of 33.333kN is uniformly distributed on the rear members, keeping the front suspension points as fixed support.

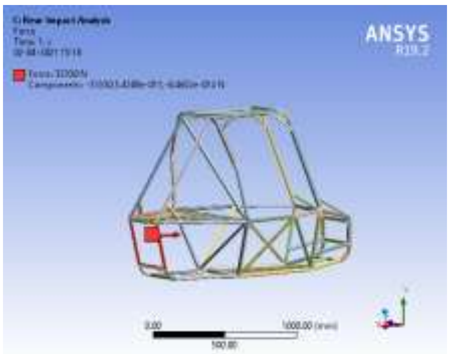

Fig. 7: Boundary Conditions for Rear Impact

From the results, it can be seen that the chassis undergoes a maximum deformation of 7.72mm which is acceptable.

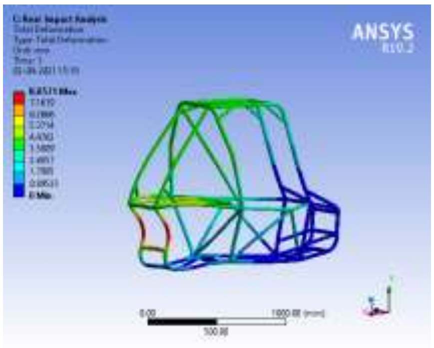

Fig. 8: Results of Rear Impact

10. SIDE IMPACT ANALYSIS

The side impact test is done to understand the and analyse when the vehicle is hit from either of its side. In this case, additional momentum comes into the picture since the side opposite to the side of impact won't be fully constrained. Due to this momentum, we assume the time of impact as 0.3 seconds and not 0.15 seconds as in the previous cases.

m = 280kg

v = 18m/s

t = 0.3s

$$a = \frac{Vf - Vi}{t} = \frac{0 - 18}{0.3} = 120 m/s^2$$

F = m * a = 280*120 = 16.667kN

F_{ri} = 16667N

The load of 16667N is applied on the Side Impact Members (SIM) and on the Lower Frame Side members (LFS)

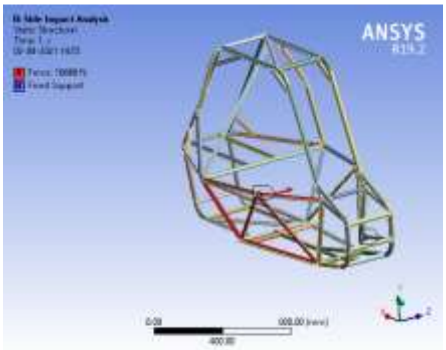

Fig. 9: Boundary Conditions for Side Impact

The maximum deformation of maximum 8.2309mm takes place at the point of impact on SIM. This deformation is acceptable.

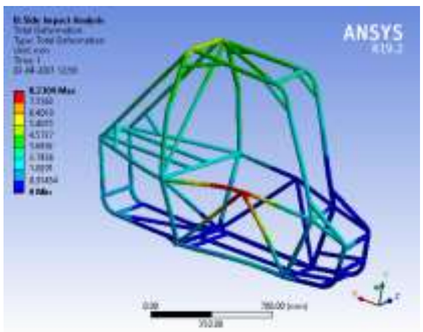

Fig. 10: Results of Side Impact

11. ROLLOVER ANALYSIS

The vehicle needs to be designed and validated for the most extreme off-road conditions including a complete roll-over. The chances of a roll-over is very high during the suspension and traction event, rock crawl or during the Final Endurance Race. This test needs to be performed with utmost care, since the driver should be safe in this extreme condition.

The load acting on the vehicle during a roll-over condition is obtained by considering 25% of the load acting on the front impact.

F = 0.25 * Front Impact Force
= 0.25 * 16.666kN
= 8.33kn

F_{ro} = 8333N

The force of 8.33kN is uniformly distributed along the negative Z-Axis on the top 4 members of RHO. The suspension mounting points are assumed to be fixed supports.

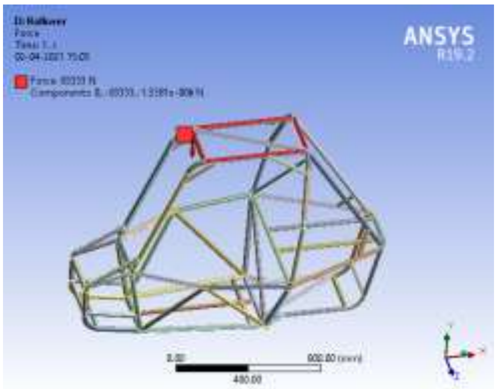

Fig. 11: Boundary Conditions for Roll-Over

According to the results, maximum deformation of 12.8mm occurs on the RHO Members.

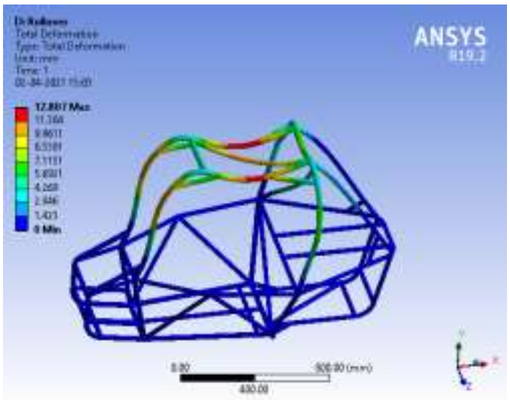

Fig. 12: Results of Roll-Over

12. SIDE BUMP ANALYSIS

The vehicle may experience a bump force on either of its side. This might happen when there is heavy reaction force and the damper is not able to sustain it. Greater forces may lead to side flip of the vehicle. The general weight distribution of the vehicle is assumed to be 42% - 58% (Front : Rear). Hence, the forces are also considered in the same proportion.

Front – 2.4kN

Rear – 3.3kN

The forces are applied on the negative Z direction at the suspension mounting points and the suspension points on the opposite side are kept as fixed points.

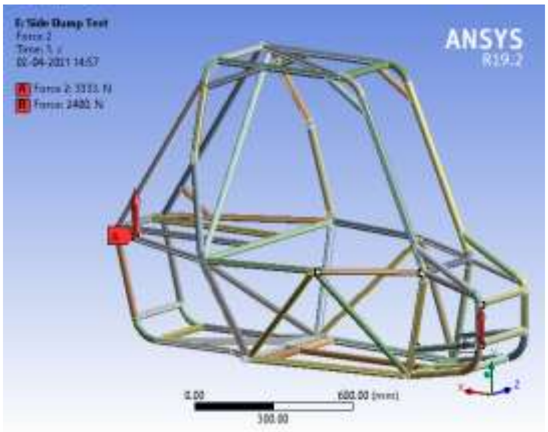

Fig. 13: Boundary Conditions for side bump analysis

According to the results, a maximum deformation of 4.54mm occurring at the rear suspension mounting points. Since the deformation occurs at suspension mounting it is critical, but the magnitude is within the tolerance level.

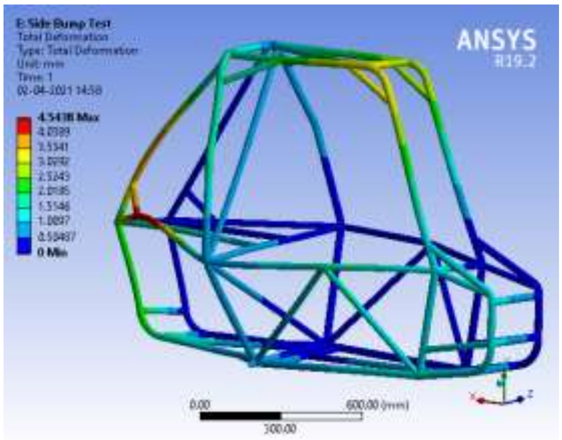

Fig. 14: Results of Side Bump Analysis

13. FRONT BUMP ANALYSIS

The vehicle experiences a torsional moment when passing through off-road conditions. Considering only the front bump, we can determine the stresses and deformations occurring in the chassis.

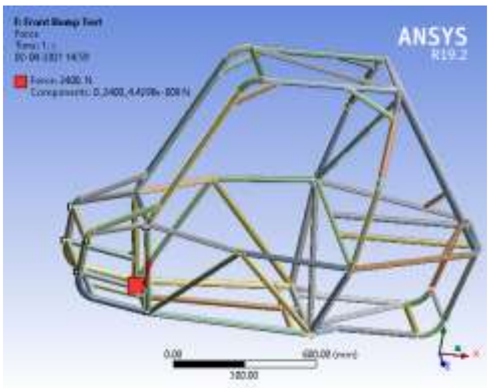

Fig. 15: Boundary Conditions for front bump analysis

It is assumed that a force of 2.4kN is acting on the front suspension mounting points and the rear suspension mounting points are fixed support. In this case, a maximum deformation of 2.4mm occurs.

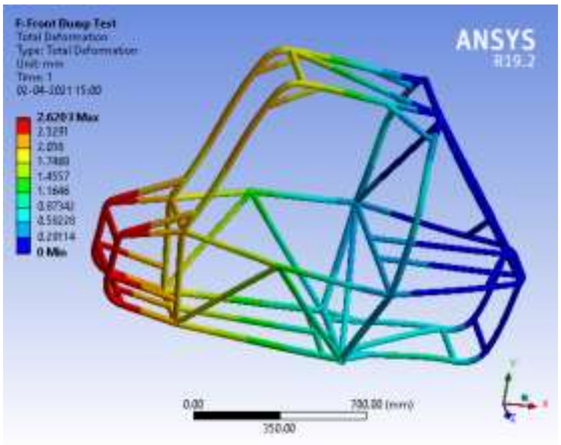

Fig. 16: Results of front bump analysis

14. REAR BUMP ANALYSIS

The rear bump analysis is similar to the previous front bump analysis, just the fixed supports and applied forces are reversed. Since majority of the vehicle weight is concentrated at the back, a force of 3.3kN acts on the rear suspension mounting points during the event of rear bump.

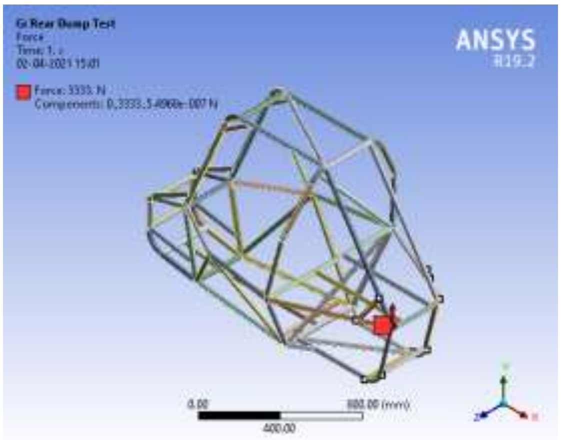

Fig. 17: Boundary Conditions for rear bump analysis

In the results it can be seen that a maximum deformation of 8.9mm occurs in the rear members as well as in the RHO members.

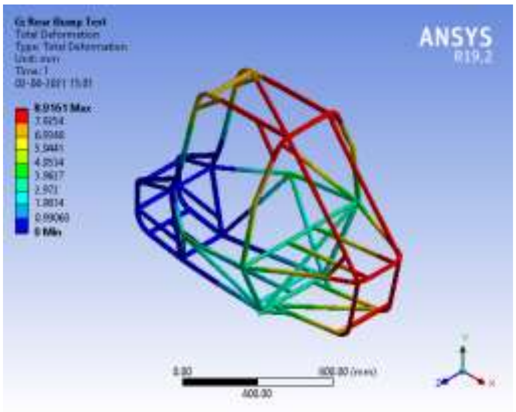

Fig. 18: Results of rear bump analysis

15. CONCLUSION

Hence the FEA of SAE Baja Chassis is done and is proved as safe since none of the deformations exceed 15mm in any of the scenarios considering a factor of safety of 1.3. Given all other sub-systems of the vehicle are designed and manufactured using sound engineering practices, the vehicle is safe to drive.

16. REFERENCES

[1] Rulebook 2021 – Baja SAE India
[2] A Vibration Analysis Of Vehicle Frame by Yogendra S. Rajput
[3] Static Analysis and Optimisation of Chassis and Suspension of an All-Terrain Vehicle by Thanneru Raghu Krishna Prasad
[4] Stress Analysis of Roll cage for an All Terrain Vehicle by Mahesh .P. Sharma
[5] STRUCTURAL OPTIMIZATION OF SAE BAJA CAR FRAME - MAE 598- 2016
[6] A Comparison of All Hexagonal and All Tetrahedral Finite Element Meshes for Elastic and Elasto-plastic Analysis - Greg Sjaardama Sandia National Laboratories Albuquerque, NM.
[7] International Research Journal of Engineering and Technology (IRJET) - Volume: 07 Issue: 03 | Mar 2020 - Design and Analysis of BAJA Rollcage

