RESEARCH SCRIPT
IJRME - International Journal of Research in Mechanical Engineering ISSN: 2349-3860

ANALYSIS OF STRESSES AND MATERIAL SELECTION OF SAE BAJA ATV – A REVIEW

Dhruva Khanzode[1] | Nilay Akre[2] | Akshay Deotale[3]
[1](Mechanical Engineering, U.G. Student, RTMNU, Nagpur, India, anuragisalways@gmail.com)
[2](Mechanical Engineering, U.G. Student, RTMNU, Nagpur, India, nilay.akre@gmail.com)
[3](Mechanical Engineering, U.G. Student, RTMNU, Nagpur, India, deotaleakshay29@gmail.com)

Abstract — *An all-terrain vehicle (ATV), also known as a quad, quad bike, three-wheeler, or four-wheeler, is defined by the American National Standards Institute (ANSI) as a vehicle that travels on low pressure tires, with a seat that is straddled by the operator, along with handlebars for steering control. A roll cage is a skeleton of an ATV. The roll cage not only forms the structural base but also a 3-D shell surrounding the occupant which protects the occupant in case of impact and roll over incidents. The roll cage also adds to the aesthetics of a vehicle. This paper deals with design of roll cage for an ATV and Various loading tests like Front Impact, Side Impact and rear impact have been conducted. The modeling and stress analysis is done by ANSYS software. We have focused on every point of roll cage to improve the performance of vehicle without failure of roll cage.*

Keywords— ATV, Roll Cage, ANSYS, FEA

1. INTRODUCTION

All-terrain vehicles (ATVs) have been popular recreational vehicles since their introduction in the United States in the 1970s. They have experienced increased use worldwide since the early 1990s and newer models weighs up to 600 pounds and can reach speeds of 75 mph [1]. They have widely used for agricultural purposes on farms. Since 1983, there have been about 190 scientific papers and technical reports written about ATV-related injuries and deaths, associated costs, effectiveness of laws, various prevention strategies, and identification of high risk groups. As the no. of accidents is increasing day-by-day on ATV hence safety of the particular ATV is the most important concern and there may be some extra safety precautions from manufacturer side to buyer and rider. To check the strength and safety features of an ATV frame, structural analysis is performed. Structural analysis is the determination of the effect of loads on physical structures and their components. Structural analysis incorporates the fields of applied mechanics, material science and applied mathematics to compute a structure's deformation, internal forces and stresses etc. There are three approaches to the analysis: the mechanics of materials approach, the elasticity theory approach and the finite element approach. The finite element approach is a numerical method for solving differential equations generated by theories of mechanics.Roll Cage can be called as skeleton of a vehicle, besides its purpose being seating the driver, providing safety and incorporating other sub-systems of the vehicle, the main purpose is to form a frame or so called Chassis. We have designed the roll cage keeping in view the safety and aesthetics. These are the two factors which matters us the most, therefore they are given utmost consideration. This paper deals with design of chassis frame for an All-Terrain Vehicle and Various loading tests like Front Impact, Rear Impact & Side Impact test have been conducted on the roll cage.

2. PAST WORK ON DESIGN OF ROLL CAGE OF ATV

CETYS Universidad -Oregon, Mexicali

This report contains the design and analysis of the structure made by team #28 CETYS Universidad, Mexicali.in 2012. The main objective of the design was of maintaining the weight minimum as possible and improving frame resistance. The table shows different Bending Strengths comparing the 1018 vs. the 4130 steel. By selecting the 4130 steel with a larger diameter and a smaller thickness, the inertia moment of the tubes are improved with the plus of a 27% weight reduction per foot. In order to further reduce the weight of the frame, it was decided to use smaller diameter tubes with 1651 mm wall thickness in non-critical parts of the frame, and using the 3.175 cm OD tube only in the main cage members.

Taking in account the new selection of material, thus a weight reduction in the frame was achieved from 61.235 kg to 40.37 kg making a 34% reduction.

TABLE 01. 1018 VS. 4130 STEEL (CES EDUPACK®)

Material	1018 steel	4130 steel	4130 steel
Outside diameter	2.540 cn	2.504 cm	3.175 cm
Wall thickness	0.304 cm	0.304 cm	0.65 cm
Bending stiffness	2791.1 Nm^2	2791.1 Nm^2	3635.1 Nm^2
Bending strength	391.3 Nm	467.4 Nm	487 Nm
Weight per meter	1.686 Kg	1.686 Kg	1.229 Kg

3. FINITE ELEMENT ANALYSIS:

In order to prove the safety of the chassis design , GeoSTAR® was used, due to its low memory requirements. After the static analysis, it was fond that which members of the frame were suffering the most stress and some modifications were made. the second analysis was made after the application of modifications o the lower frame. The stress graph showed the decrement in the overall stress suffered to an acceptable level. To achieve the proper distribution, the lower rear members of the frame were changed from 2.54 cm diameter tube to 3.175 cm diameter tube. Also reinforcement were added to the driver's seat. Further analysis proved an impact factor of 9.5, which meant that the stress during impact that would have to reach more that nine times the normal working condition to cause plastic deformation or break. This could be translated to an impact to the ground from a height of 355.6 cm

Figure 01: Solid works model team #28

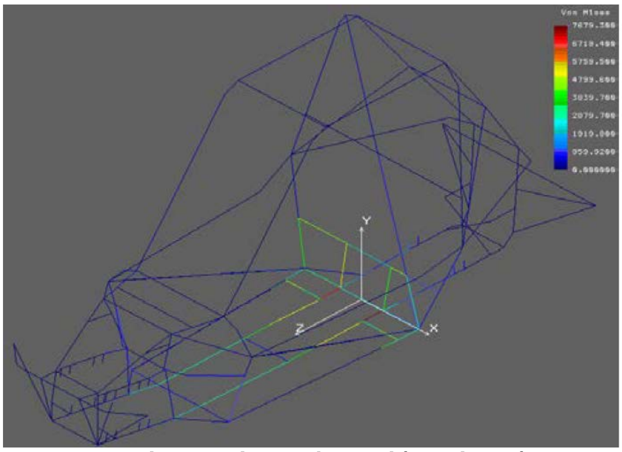

Figure 02: second static analysis, with revised frame design for optimum stress distribution

Sanika Oturkar , Karan Gujarathi

This paper was focused on the computational front of static stress analysis. Here a preliminary design of the frame structure was developed wherein the structural members and their end connections were simply represented by their centre lines and points respectively, in a 3D environment using CATIA V5. The geometries of cross-section and end connections were ignored for simplicity. Parametric modelling was implemented to ensure that future changes could be incorporated easily. A finite element (FE) model was created using the 'Pipe 16' element in ANSYS, on which static analysis was performed. The next step was simulating the problem statement by choosing appropriately the material properties, cross-sectional properties, positional constraints, loading conditions and mesh element size. The analysis showcases the distribution of Von Misses stresses and the deformation of the frame members, when subjected to the applied loads. If the stress generated in the chassis members was found to be above the yield limit of the material, the existing frame was modified for a safe design. The new design was again subjected to the same analysis, and the iterations continued till the stress and deformation was within the desired limit.

In this analysis, circular tubing of AISI 1018 having uniform cross sections was selected, confirming to the rule book. The material properties have been listed in the table below.

TABLE 02: MATERIAL PROPERTIES

Parameter	Value	Unit
AISI 1018	-	
Outer Diameter	25.4	Mm
Thickness	3	Mm
Young's Modulus of Elasticity	250	GPa
Permissible yield stress	365	N/mm^2
Poisson's ratio	0.3	-
Carbon Content	0.18	%

4. ANALYSIS:

The analysis of the chassis was performed in ANSYS APDL. Node to node connectivity between members was ensured to obtain correct readings of the analysis. The line type element 'PIPE 16' was used. It is a uni-axial element with tension-compression, torsion and bending capabilities; and has six degrees of freedom at the two nodes: translation in the nodal X, Y, and Z directions and rotation about the nodal X, Y, and Z axes. With assumptions :

1) The chassis material is considered to be isotropic and homogenous.
2) Chassis tube joints are considered to be perfect joints.
3) The 'Crumple zone' phenomenon is not considered.

5. LOADING POINTS:

The structure is loaded at the points where the Centre of gravity is located (as indicated by the yellow figures on the chassis in Fig. (3). The forces are applied on the frontal part of the chassis (as indicated by red arrows in the figure) as it is the first point of contact in case of a frontal collision.

Failure of mechanical components subjected to bi-axial or tri-axial stresses occurs when the strain energy of distortion per unit volume at any point in the component, becomes equal to the strain energy of distortion per unit volume in a standard tension test specimen during yielding. According to this theory, the yield strength in shear is 0.577 times the yield strength in tension. Experiments have shown that the distortion energy theory is better in agreement for predicting the failure of ductile components than any other theory of failure.

The analysis was carried out using progressively reducing elemental sizes. The elemental size having consecutive stress error less than 5% is generally considered as the optimum size of mesh. It means that any further decrease in size will only negligibly increase the accuracy of the results.

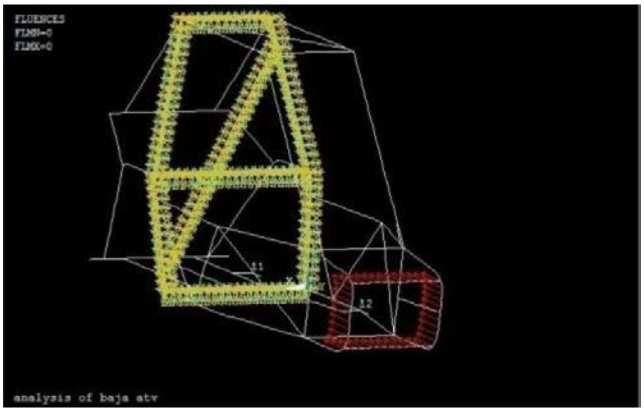

Fig: 03. Loading points and impact points

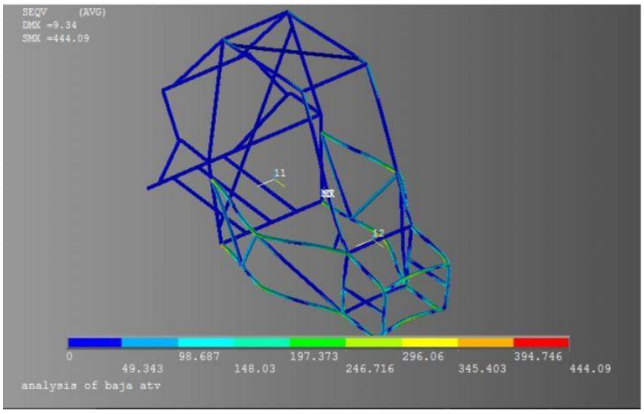

Fig:04. Result for mesh size 10

6. OBSERVATIONS:

TABLE:02.MESH SIZE DEPENDENCE OF VON MISSES STRESS

Parameter	Case I	Case II	Case III	Case IV
Size of mesh (units)	50	10	8	7
No. of nodes	741	3586	4484	5130
No. of elements	7772	3815	4515	5712
No. of nodes selected for fixed constrain	147	588	884	1009
No. of nodes subjected to the force	44	180	262	288
Force on each node	750	183	126	115
Max value of Von Misses stresses	42.775	417.187	443	444.09
Percentage error	-	87.2	6.19	0.24

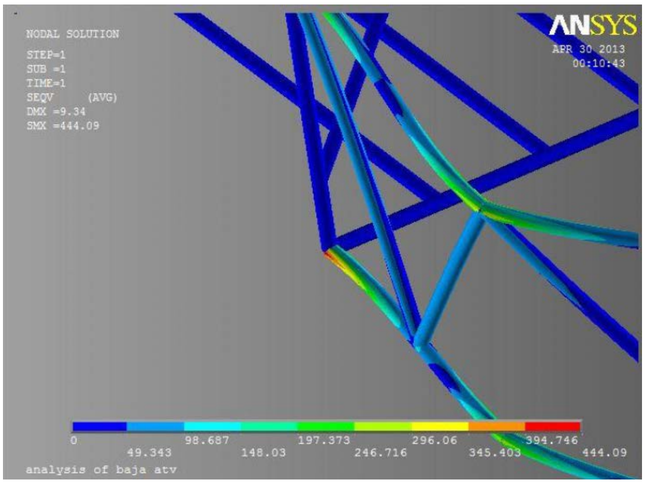

Fig:05. Region of high stress concentration in Fig :04

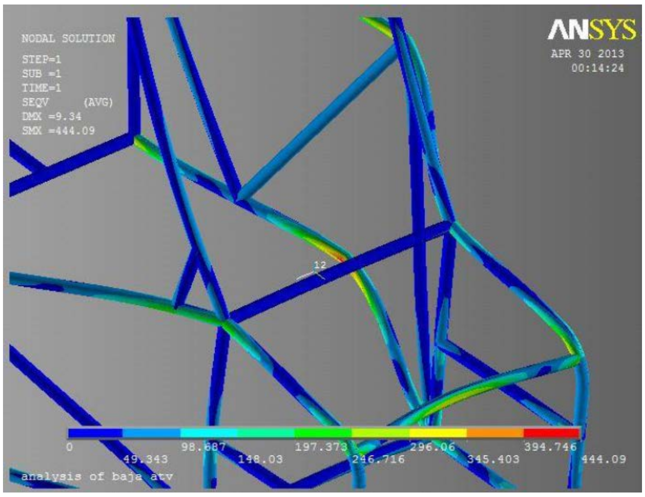

Fig:06. Region of high stress concentration in Fig :04

After all the analysis, the following conclusions were drawn:
1. Maximum stressed regions of the chassis are under safe stress.
2. Two high stress regions were detected as shown in Fig. (5) and Fig. (6)
3. The maximum Von Misses stress was found to be 444.09N/mm2, which exceeds the yield strength.

Ashwa Riders #82

Team “Ashwa Riders” is a team of St.Vincent Palloti College of Engineering and Technology, Nagpur. this report was for the BAJA STUDENT INDIA event for the year 2016. The objective of this project is to use engineering techniques to design and build a prototype of a rugged, single seat, off-road recreational vehicle.

Team Ashwa Riders made it their main objective to focus on safety, weight reduction, manufacturability, serviceability and overall performance. By performing multiple calculations and Finite Element Analysis on design components, the team is confident in the safety of the vehicle. Using various design techniques such as FEA, CATIA V5R20 and PTC CREO 2.0 Software, and traditional hand calculations.

For the 2016 year the team decided to take up the SAE 4130 Steel tubing. The 2015 Baja rules specify using a material greater than or equal to 18% carbon steel which has an Yield strength of 365MP

The frame went through many revisions from the beginning of the design process. By working with the other members of the team the frame has been modified to work with the other sub systems of the car. Once the frame design was set, optimization could be done to reduce weight. The frame analysis was completed using ANSYS Finite Element Analysis (FEA). Using the software the team was able to make changes to the frame to give it a minimum safety factor of 1.8

TABLE 03: FORCE ANALYSIS ON DIFFERENT IMPACTS

Parameter	Front impact	Side impact	Front roll over	Front torsion
Weight of car (Kg)	200	200	200	200
Maximum force (N)	12000	6000	3000	6000
Maximum force (g)	6	3	1.5	3
Max. equivalent stress(MPa)	232	240	305	254
Maximum deformation (mm)	1.7	2.6	2.12	2.2
Factor of safety	2.3	2.2	1.8	2.1

Fig:07 CAD model

Team Jaabaz,VIT University

This report tried to summerise the work done by the students of team JAABAZ #21 of VIT University for the year 2014. This report describes the methodology followed by Team Jaabaz to design, fabricate and test an all terrain vehicle that will compete in Mini Baja event in UTEP. The main design focuses with the completely new vehicle are a lighter and more rigid and ascetics oriented frame, a more robust suspension design, and a more versatile drive train. With gained easy access to a tube bender the team was able to increase the number of bends in the vehicle and in turn use more continuous members. The weight of the car was also reduced by switching to 1 mm thick tube in SIM. Similar robust and durable designs were adopted in suspension, transmission and brakes system.

For the selection of material, multiple materials were compared by the team.

TABLE:04 PROPERTIES OF DIFFERENT MATERIALS

Material	AISI 1018	AISI 4130	Duplex 2205 steel	Duplex 2205 steel
Outside diameter	2.540 cm	2.540 cm	2.540 cm	2.540cm
Wall thickness	0.2 cm	0.2 cm	0.2 cm	0.1 cm
Bending strength	390 Nm	382 Nm	454 Nm	260.4 Nm
Bending stiffness	2791 Nm^2	2791 Nm^2	2171 Nm^2	
Weight/meter	1.6615 kg/m	1.2444 kg/m	1.1475 kg/m	0.8790 kg/m

Since the Duplex steel (2mm tube) has a higher strength than 3mm tube of AISI 1018 steel, therefore the team opted for this steel which ensured better weight savings. The overall weight was further reduced by using 1mm tube as secondary members.

Fig:08. CAD prototype

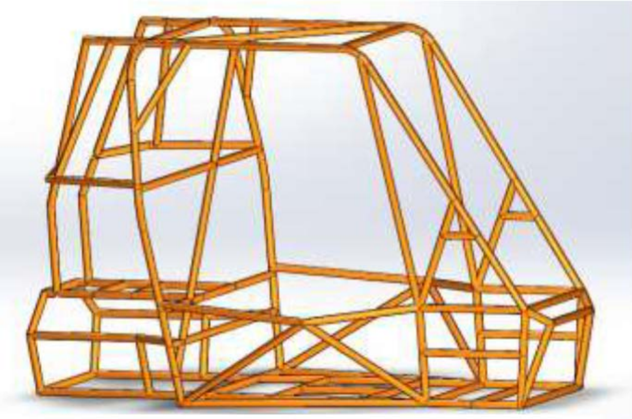

Fig:09 frame for BAJA UTEP 2014

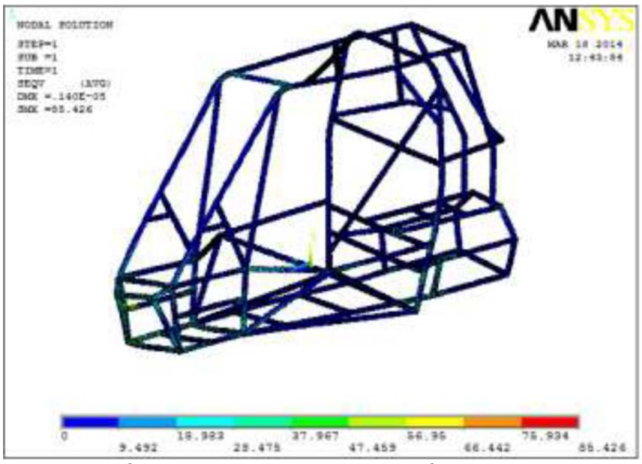

Fig:10.front impact analysis 8500N at front FOS=2.686

Abhinav Sharma,Jujhar Singh,Ashwani Kumar

This paper provides a complete design and analysis of “Baja vehicle” or “All-Terrain Vehicle (ATV)”. While designing this Baja vehicle, all the design aspects were taken as per the rules of Society of Automotive Engineers (SAE)-2014. The main objective of this paper was to design and optimize the roll cage, front and rear suspension system, power train system. The finite element analysis (FEA) is also done on the roll cage for validating the design. Initially, a preliminary design of the roll cage was made as a 3-D model using CAD.

The material used for the required roll cage was circular steel tubing with an outside diameter of 25 mm (1 inch), wall thickness of 3.05 mm (0.120 inch) and a carbon content of at least 0.18 (Baja SAE *et al,* 2014). The research was conducted to choose the best possible material. The choice of material was limited to steel as per SAE rules. The material was selected on the basis of cost, availability, performance and weight of material. After thorough research, two best materials were found for the designing of the roll cage i.e.: Steel AISI 4130 Chromoly alloy and Steel AISI 1018. The reasons for using round tubing (seamless) were it is lighter than square tube as smaller gauge sizes can be used to handle the same stress as a wider square tube and a round tube always out performs the square tube. Table 1.2 shows Mechanical properties of Steel AISI 1018 tube.

TABLE:05. MECHANICAL PROPERTIES OF STEEL AISI 1018 TUBE

Density	0.284 lb/in^3
Ultimate tensile strength	63,800 psi
Yield tensile strength	53,700 psi
Modulus of elasticity	29,700 ksi
Bulk modulus	20,300 ksi
Shear modulus	11,600 ksi
Poisson’s ratio	0.290
Elongation at break(50 mm)	15%
Brinell hardness	126

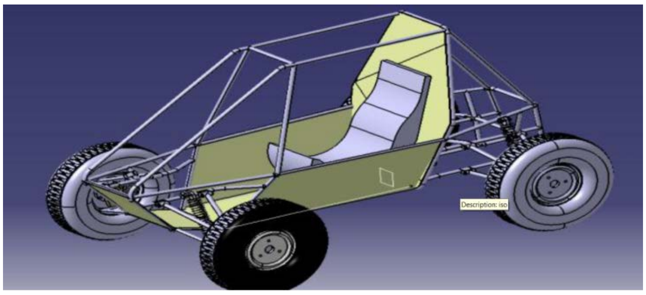

Fig:11.Isometric view of BAJA vehicle

As roll cage was designed by plotting key points, lines and splines, so every member of the roll cage was considered to be properly constrained at every joint. For boundary conditions for frontal impact test, the roll cage had to be fixed from the rear side and the front member will come across the applied load. In the similar way, for side impact test, one side of the roll cage elements was fixed while the other side would be applied with load. In order to determine the safety of our vehicle, deformation produced by applying different impact loads was checked (i.e. forces - 90.718 kgf on front, 45.359 kgf on side members and 90.718 kgf on top in case of roll over) with the help of analysis software. There are few important loading situations that should be analysed. These include frontal impact, side impact and roll over impact. Analysis from figures 1.7 to 1.9 shows the deformations produced by different impact loads. In frontal impact, there is a possibility of vehicle crashing into another vehicle head on during the race (Sanika and Karan *et al,* 2013).

The side impact analysis was carried out as there is a possibility of collision with another vehicle from either direction. Thus, the stresses acting on the side members of the roll cage are analysed.

The rollover impact analysis was carried out by considering the stresses induced on the members of the roll cage when the vehicle topples down from a slope with an angle of 45°. In this impact, the upper and rear members of the vehicle would bear the force.

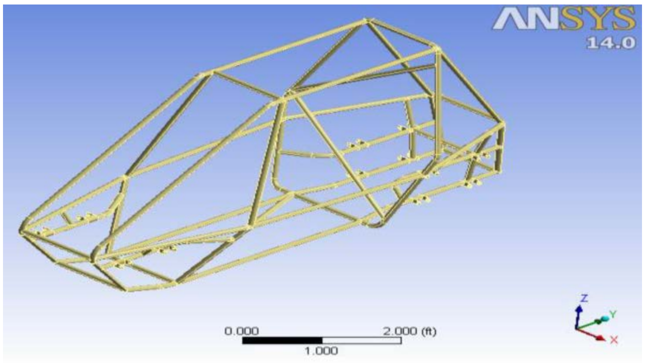

Fig:12 Isometric view of roll cage

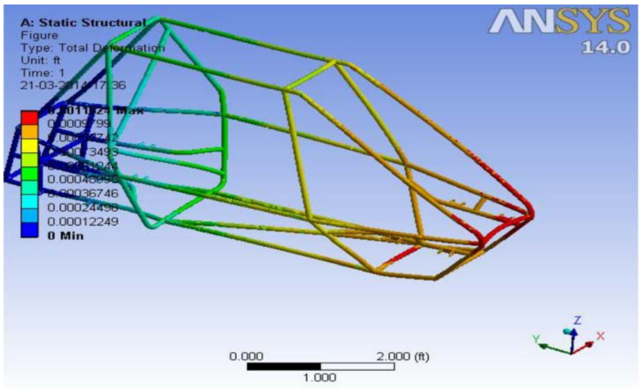

Fig:13 .Deformation due to front impact load

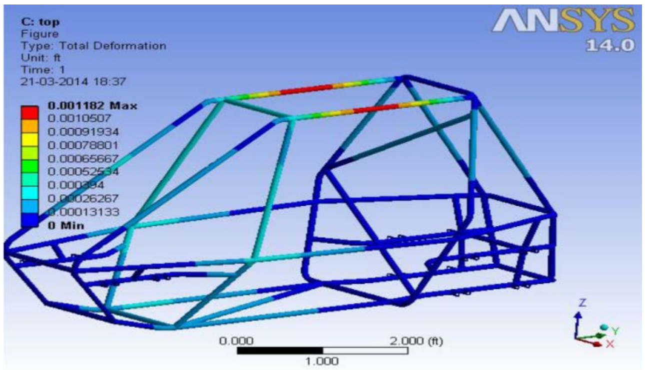

Fig:14. Deformation due to roll over.

7. CONCLUSION

In this study of roll cages designed and manufactured by different BAJA teams across the world, the following aspects should be taken into consideration while designing the roll cage:

- Selection of material: the steel which is to be selected should fulfil the minimum criteria given in the SAE rule book but should have a high yield strength to bear the stresses ,with economical cost so that the budget in maintained.
- Weight of the roll cage: the weight plays a significant role in vehicle dynamics, as reduction in weight leads to higher efficiency with respect to weight fuel ratio and it also helps in increase in speed. So the weight of roll cage should be kept as low as possible.
- Material with multiple cross-section area: it is advisable that bars of different cross section areas should be used at different places. Like material having higher cross section area for primary members and lower cross section area for secondary members. This combination eventually helps in reduction of weight. But it also have some disadvantages as the joints will be undergoing higher stresses than the uniform cross section joints, so that factor should be compensated by proper welding.

REFERENCES

[1] Gustavo Ovies, Andres Magana, Alenjandro Burgas, Ivan Pulido, Ivan Williams, Bernardo Valadez, “CETYS Universidad Baja SAE Design Report”, Mechanical engineering division of CETYS Universidad,Campus Mexicali,2012

[2] Sanika Oturkar,Karan Gujarathi , “ *An Introduction To Computational Frontal Static Stress Analysis Of a Baja Car*”, International Journal of Scientific and Research Publications, Volume 3, Issue 8,ISSN 2250-3153,August 2013.

[3] Shubham Burde, Akash Lanjewar, Tejas Jambhale,”St. Vincent pallotti college of engg & tech. design Report”,Rashtrsant Tukdoji Maharaj Nagpur University,SAE report 2016.

[4] Raghuram Nagubandi,Debidutta Mishra,N.Raghuram”Baja SAE UTEP 2014 Design Report-Team Jaabaz”,VIT university,Vellore,2014

[5] Abhinav Sharma,Jujhar Singh And Ashwini Kumar,”*Optimum Design And Material Selection Of Baja Vehicle”* International Journal Of Current Engineering And Technology, Vol.5 E-ISSN 2277-4106,P-ISSN 2347-5161,June 2015.