International Journal of Science and Research (IJSR)
ISSN: 2319-7064
ResearchGate Impact Factor (2018): 0.28 | SJIF (2018): 7.426

Design, Analysis and Optimization of a BAJA-SAE Frame

Yogesh Chandra

Manipal Institute of Technology, Department of Mechanical and Manufacturing Engineering, Manipal University, Udupi-576103, Karnataka, India

Abstract: ***The report being elaborated below involves the design, analysis and fabrication of a multi-tubular space frame. The main motive of the roll cage is to ensure the structural safety and comfort of the driver under all loading conditions as well as incorporating all the other subsystems together. The roll-cage has been drafted on various designing software's such as Solidworks and CATIA. The analysis and optimization has been done on advanced 3D CAD modelling and FEA software's such as ANSYS, Solidworks and Hyperworks. The material used for fabrication was finalized after extensive market surveys on the basis of cost and strength to weight ratio. Testing and development to meet the design requirements and validation of the specification was undertaken.***

Keywords: BAJA-SAE, CHASSIS, FEA

1. Introduction

The roll-cage is fabricated by a material of same composition but different cross-sections depending on the load. The roll cage has been designed by taking BAJA SAE 2019 RULEBOOK into consideration. The design discussed in the following report involves the incorporation of a 305CC single cylinder BRIGGS AND STRATTON engine. So it becomes more significant to minimize the weight so as to achieve the maximum possible acceleration by balancing the strength to weight ratio. Best possible fabrication techniques have been used to manufacture the roll-cage with importance given to driver ergonomics and

2. Material Selection

The selection of the material is generally on the basis of its strength to weight ratio, elongation properties and availability. An optimum balance of fulfilling design requirements and minimizing weight is crucial for successful design.

The materials generally chosen are AISI 4130 and AISI 1018 steels. The following table compares the two materials.

Table 1: 1018 vs 4130

Material	AISI 1018	AISI 4130
Yield Strength	417 MPa	638 MPa
Ultimate Strength	473 MPa	810 MPa
Bending Strength	402.9 MPa	415 MPa
Preferred Welding Type	MIG Welding	TIG Welding
Availability	Easily Available	Less Available
Cost	Cheap	Expensive

From the above it can be deduced that AISI 4130 has a much better strength to weight ratio. Also by using AISI 4130, we can ensure a straight weight reduction of 17% per tubing length without compromising on its strength.

2.1 Selection of Cross-Section

Primary member

To select the most appropriate section for primary members, an analysis was done for bending strength, bending stiffness and weight per meter of the cross-section. The graphs of Bending Strength and Bending Stiffness versus thickness were plotted in MATLAB. They also include the minimum requirements for bending stiffness, bending strength and minimum wall thickness (1.6 mm) as specified by the rulebook. The cross sections shortlisted were 1 inch, 1.15 inch and 1.25 inch. Through a market survey, it was found that the above mentioned pipe diameters were available in following thicknesses - 1mm, 1.2mm, 1.65mm, 1.8mm and 2.1mm

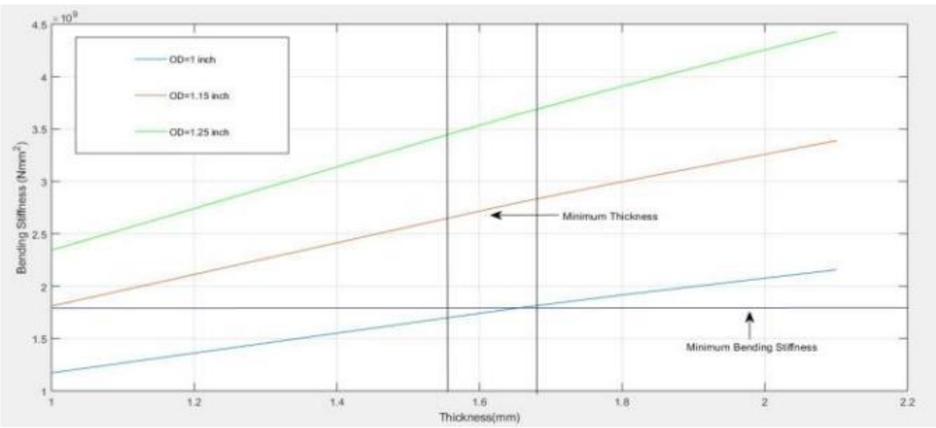

Figure 1: Bending stiffness vs thickness

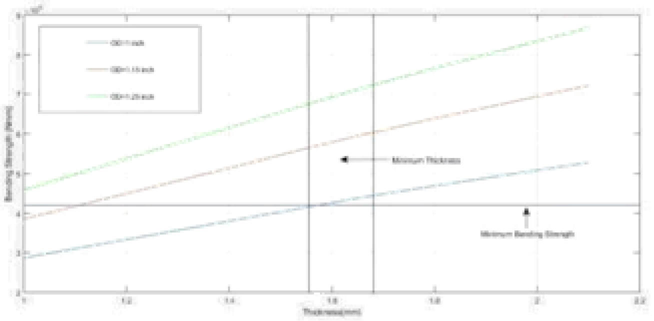

Figure 2: Bending strength vs thickness

Since the cross section 1.15 inch x 1.65 mm gives minimum weight along with satisfying all the conditions, it was selected for the primary members

Secondary Members

Secondary members are provided with an aim of providing structural support to the entire frame and primary members in particular. Since these members are specifically structural they are used of smaller diameter and thickness as compared to the primary ones.

2.2 Welding Technique

Generally TIG and MIG welding is used to weld 4130 steel. By conducting destructive tests on weld samples of these techniques the optimum welding technique that is TIG welding was selected.

CAD Model

Rollcage was designed using Solidworks and CATIA. Analysis and optimization was conducted using ANSYS and ALTAIR Hyperworks. The following parameters were considered while designing the roll-cage:

2.3 Driver Ergonomics

Ergonomics comes into picture with 5 aspects that are to be concentrated upon:
1) Safety
2) Comfort
3) Ease of use

It is important that the driver be comfortable for the endurance race in which he is to drive the vehicle for a time length of continuous 4 hours and thus from this point of consideration for ATV of BAJA SAE, the comfort and safety of the driver are vital in order to reduce the fatigue of the driver and hence increase his efficiency.

The analysis for driver ergonomics was conducted using RULA in CATIA. The emphasis is on the driver comfort and driver vision taking the rulebook constraints into consideration. There is sufficient distance between brake pedal and accelerator for comfortable positioning of both shoe on pedal. There is bracing provided as a support to change position while sitting and prevention from fatigue due to constraint. The manikin was most comfortably placed and different clearances from manikin, its visibility, knee pivot and ankle angle was measured and RULA analysis was conducted.

The cockpit was designed to protect the driver and permit easy egress in an emergency.

The visibility of manikin after placing comfortably on the seat was checked. The kill switch is in the visibility of the driver.

Figure 3: RULA analysis in CATIA

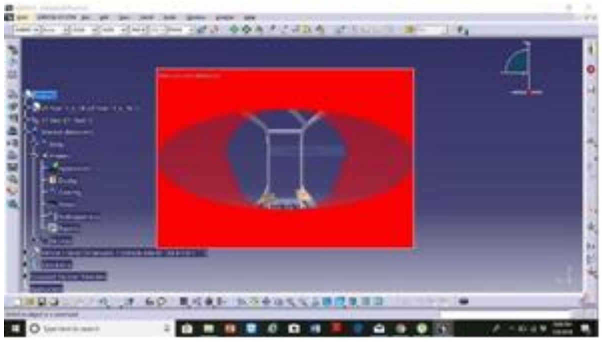

Figure 4: Driver vision cone

CAD Model

Rollcage was designed using Solidworks and CATIA. Analysis and optimization was conducted using ANSYS and ALTAIR Hyperworks.

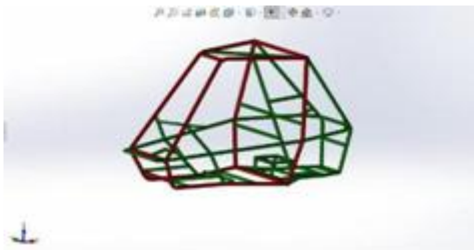

Figure 5: Designed in Solidworks software

Finite Element Analysis

The multi tubular space frame of an all-terrain vehicle should be capable of enduring harsh off road environments. Finite element analysis of the roll-cage was done using ANSYS workbench 19.2 and Hyperworks. The roll-cage was analysed for various conditions like Front impact, Side impact, and Front roll over, Side roll over, Torsional Stiffness with the main focus on driver's safety. The results were studied and the necessary changes were implemented. The results of the simulation were interpreted in HyperView. The analysis determines the intensity and the areas of the highest Von Mises stresses and the deformations that the frame members are subjected for the applied loads.

International Journal of Science and Research (IJSR)
ISSN: 2319-7064
ResearchGate Impact Factor (2018): 0.28 | SJIF (2018): 7.426

Meshing
The Roll-cage mid surfaces and cleanup of the geometry were created in ALTAIR Hyperworks and the .iges file was imported from Solidworks. 2D meshing was carried out since the thickness of the pipe was much less than the diameter of the tube. Shell elements were used for carrying out the 2d meshing of the roll-cage. The element shape used is TRIA and QUAD for 2D mesh.

The quality index had been used to ensure that the fail elements are minimal. Also, it was taken care that the element with a size less than 3 mm is minimal in order to avoid any unnecessary solver time. The meshing element was selected as mixed as per the scenario with the thickness provided as additional input along with material properties and load collectors and boundary conditions for the solver.

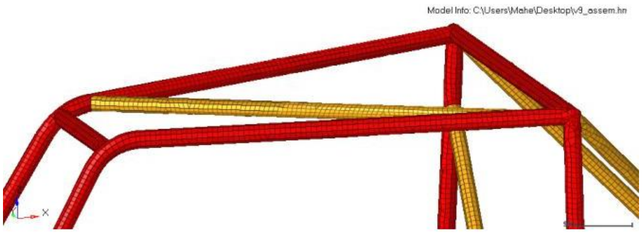

Figure 6: 2D Roll-cage meshing

Components
Primary and secondary tubes were of different thickness and so, they were assigned with different component as shown in fig 4.

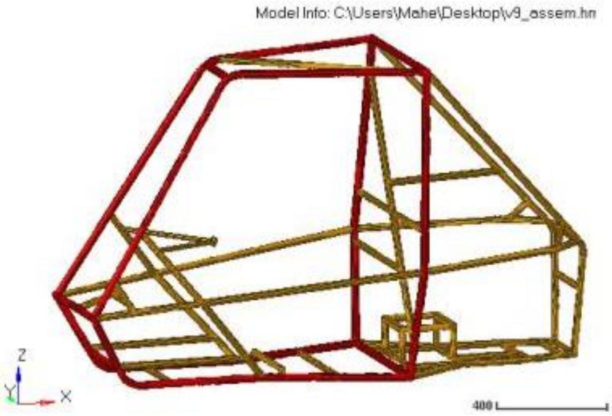

Figure 7: Red color denotes primary members and yellow denotes secondary members.

Quality Criterion
To ensure model accuracy and efficiency, the mesh of the model needs to meet a mesh quality criterion. The quality of the mesh will affect the time step calculations of the simulations and thus the computation time. The time step is directly related to the characteristic length of the elements so the minimum element size is of particular importance. Severely distorted elements will affect the accuracy of the results due to an increase in stiffness of the element due to the distortion. The percentage triangular elements should be less than 5% of the number of elements in the component because the triangular elements impart an artificial stiffness into parts modeled with them. This will cause an unrealistic behavior of the chassis frame. The target element size was 6mm.

The warpage in two-dimensional elements is calculated by splitting a quad into two trias and finding the angle between the two planes which the trias form. The quad is then split again, this time using the opposite corners and forming the second set of trias. The angle between the two planes which the trias form is then found. The maximum angle found between the planes is the warpage of the element. Warpage was maintained below 15 degrees.

The Jacobian ratio is a measure of the deviation of a given element from an ideally shaped element, and it was kept at 0.6. FIG 3 outlines the important mesh quality criteria.

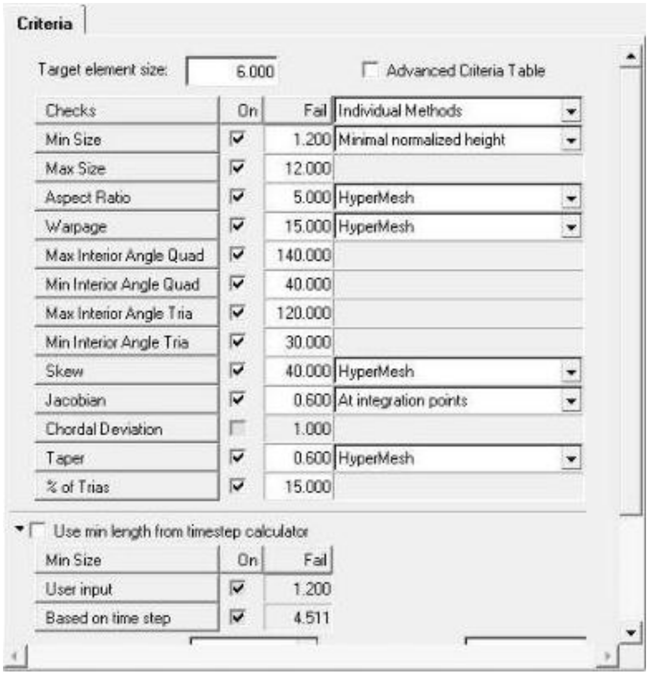

Figure 8: 2D element quality report

Impact Analysis
Linear static analysis was performed that simulates the loads from a frontal impact using Optistruct solver. The front impact analysis is done for analysing the rigidity of a roll cage as well as the safety of the driver in case of a head on collision of the car. Results of interest from this analysis are Von Mises stress and displacements for different loading conditions on the roll-cage structure. If a design cannot survive a linear static stress analysis it has to be fixed before moving on to more complex, time consuming and expensive dynamic or non-linear analysis

Assumptions for frontal impact simulation:
1) The chassis material is considered isotropic and homogeneous
2) Chassis tube joints are assumed to be perfect joints

The Impact forces were calculated using Newton's second law which states that the net force acting on a body is equal to the product of mass and acceleration of the body.

Force= Mass * acceleration
Force= Rate of change of momentum
Impulse= Force * time= Change of momentum = Mass * Change in velocity
Force= M*V / impact time

Velocity was assumed to be 45kmph (for front and rear impact). The weight of car was considered to be 210Kg.

Front Impact: In actual conditions, the car is going to hit a tree, another car or a wall. In the first 2 cases, the tree and the other car are deformable bodies. So the time of impact will be greater, around 0.4 seconds, while the wall is considered as non-deformable i.e. a rigid body. Hence the time of impact will be obviously less than that in the above case. It is obvious that the impact force in the case of wall will be more than the first two cases. The vehicle was considered to be moving with a velocity of 45 kmph and time of impact as 0.1 seconds.

Side Impact: Since both bodies involved are deformable, the time of impact is slightly more than that of front impact. In case of side impact, the vehicle was considered to be in a stationary state. Impact was subjected on the side by an identical vehicle at a speed of 30 kmph. Time of impact is taken as 0.4 seconds because both the bodies are deformable.

Front Impact Analysis

Mesh Features

Mesh type: 2D mesh of mixed type
Total elements: 43152
Element shape: Quad and Tria.

Table 2: Loading conditions of front impact analysis

Maximum Force	6.6KN
Force Applied On	Front Member
Fixed	Suspension mounting points

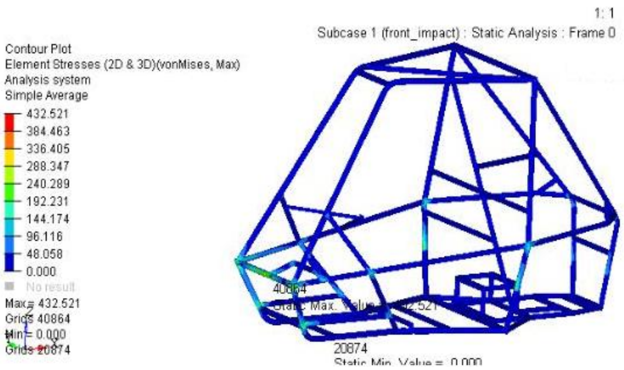

Figure 9: Front Impact Stress distribution

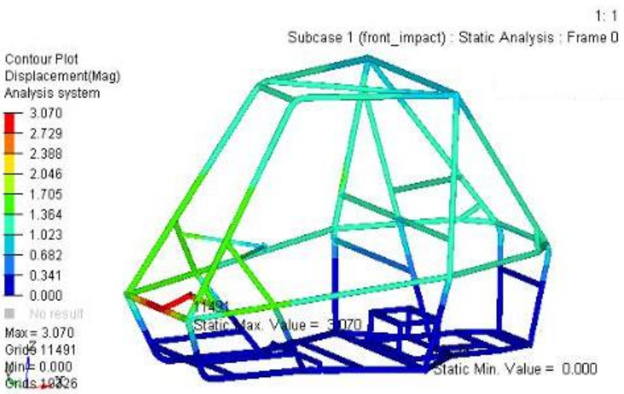

Figure 10: Front Impact deformation

Table 3: Result of front impact analysis

Max Stress	Max Deformation	Factor of Safety
432.52 MPA	3.07 mm	1.47

Side Impact Analysis

Side impact analysis was performed to analyze the strength of the roll-cage in the case of accident involving the vehicle hit by another car from side**.**

Table 4: Loading conditions of side impact analysis

Maximum Force	4.2KN
Force applied on	SIM member
Fixed	Lower suspension mounting points

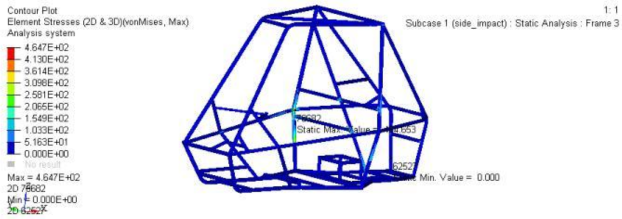

Figure 11: Side impact stress distribution

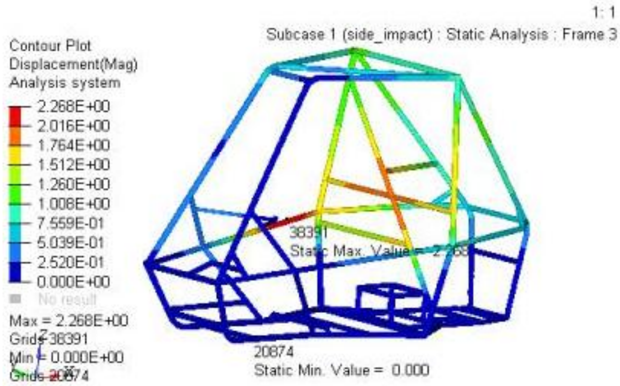

Figure 12: Side impact deformation

Table 5: Results of side impact analysis

Max Stress	Max Deformation	Factor of Safety
464.7MPa	2.268mm	1.37

Rear Impact Analysis

Rear impact analysis was performed to analyze the strength of rear members of the roll-cage on impact by other car, the rear of roll-cage holds and supports drivetrain and crucial suspension components, which should be protected from external force.

Table 6: Loading conditions of rear impact analysis

Maximum Force	6.6KN
Force applied on	Rear
Fixed	Suspension mounting points

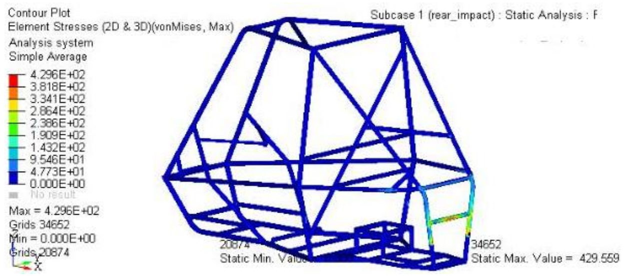

Figure 13: Rear Impact stress distribution

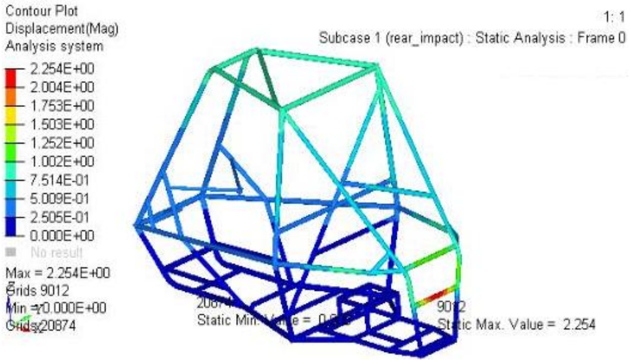

Figure 14: Rear Impact deformation

Table 7: Results of rear impact analysis

Max Stress	Max Deformation	Factor of Safety
429.5 MPa	2.25 mm	1.48

Front Rollover Analysis

In the case of front roll over, the vehicle is considered as toppling while coming down a hill. The roll cage should protect driver under this severe condition.

Table 8: Loading conditions of front rollover analysis

Max Force	2.42KN
Force Applied on	At 45° to FBM-RHO Bends
Fixed	Suspension mounting points

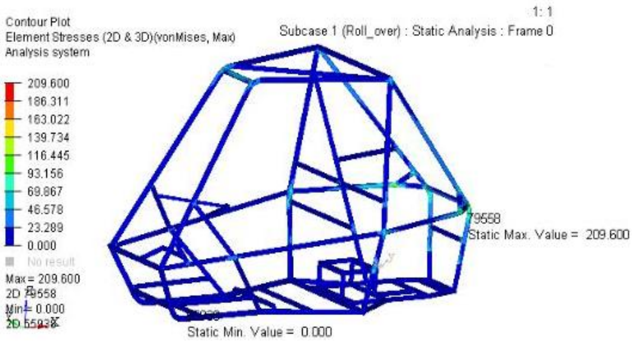

Figure 15: Front rollover stress distribution

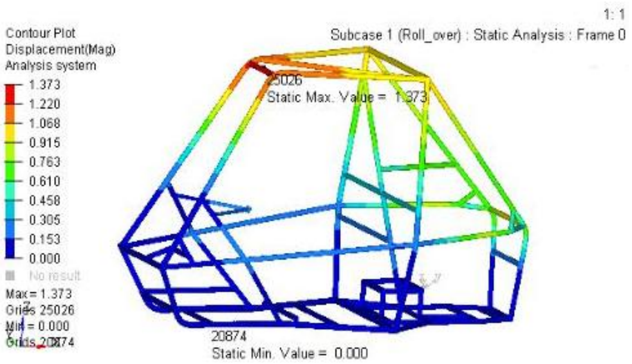

Figure 16: Front rollover deformation

Table 9: Results of front rollover analysis

Max Stress	Max Deformation	Factor of Safety
209.6 MPa	1.37mm	3.04

Side Rollover Analysis

In this case, the analysis is performed for side rollover caused due to the harsh and non-uniform off road condition the car being subjected during the race.

Usually side roll over analysis is not so significant in case of commercial vehicles, since if the vehicle topples while cornering; it will be because of the faulty suspension design. But in case of an ATV, there are chances that the vehicle will topple while encountering a treacherous terrain.

Table 10: Loading conditions of side rollover analysis

Max Force	2.42KN
Force applied on	RHO member
fixed	Suspension mounting points

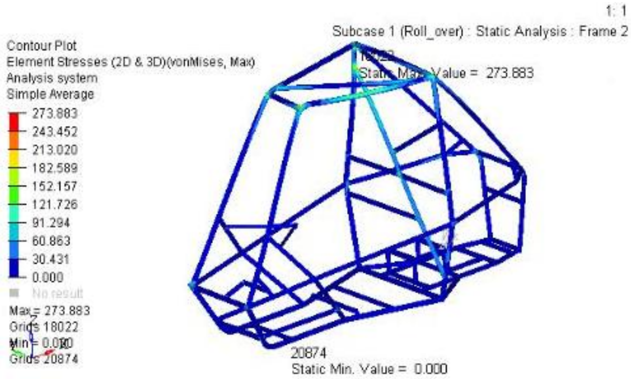

Figure 17: Side Rollover Stress distribution

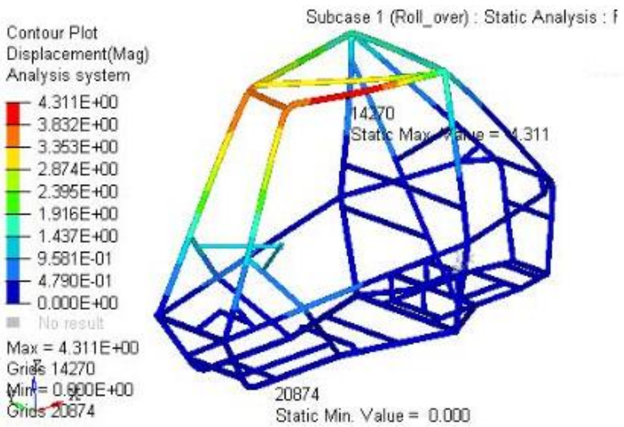

Figure 18: Side rollover deformation

Table 11: Results of side rollover analysis

Max Stress	Max Deformation	Factor of Safety
273.8 MPa	4.31 mm	2.33

Torsion Stiffness Analysis

The chassis should be stiff enough to sustain dynamic suspension loads. When the vehicle is negotiating the bump there might be a case of alternating bumps on left and right wheels. Considering the left wheel is having the upward travel (bounce) and the right wheel is having the downward travel (rebound) the spring forces will act in the opposite direction composing a couple on front of the vehicle.

This couple tries to produce the torsional stress in the chassis. for the worst case scenario the diagonally opposite wheels are having the opposite wheel travel i.e. front right wheel is having the vertically upward travel and at same time rear left wheel is having the vertically downward travel producing a couple diagonally. This couple is responsible for the torsional stresses in the vehicle

Table 12: Loading conditions of torsional analysis

Max Force	2.42KN
Force applied on	Diagonally Opposite Suspension Mounts
Fixed	Diagonally Opposite Suspension Mounts

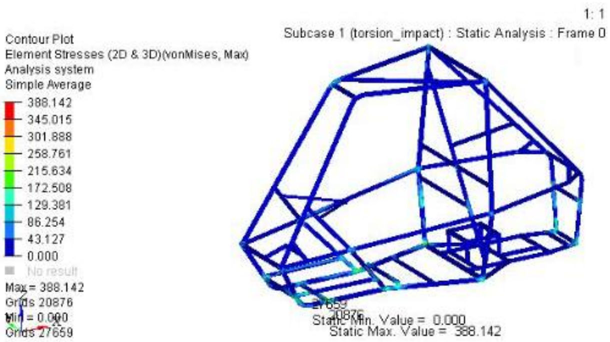

Figure 19: Torsional Stress distribution

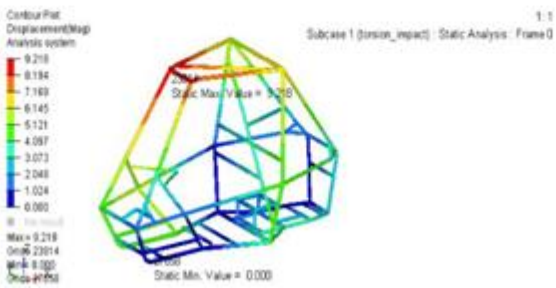

Figure 20: Torsional deformation

Torsional Stiffness

The maximum deformation was at the front RHO bend.
F=2420N
L = Distance between diagonally opposite suspension mounts=730mm
D = Vertical deformation=9.218mm
Θ = Angular deformation Tan (θ) = D/(L/2)
Torsional Stiffness = (F x L) / θ.
D=9.218mm
Torsional Stiffness= 1244 Nm/degree

Modal Analysis

Modal analysis was done to avoid resonance of the roll-cage. Engine is the main source of vibration in the vehicle. Since a 4-stroke single cylinder engine is used, dominant half order excitation frequency of the engine was calculated at idle and maximum rpm. The dynamic characteristics of the chassis were assessed in three steps.

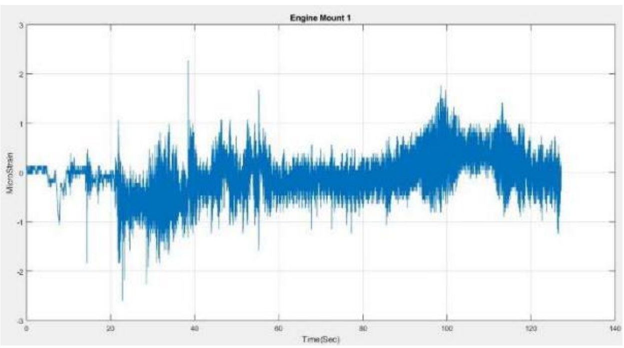

Figure 21: Engine vibration graph from strain

Firstly, a 1D meshed FEM model of the chassis was developed using ANSYS workbench 19.2 modal analysis solver. Natural frequencies and mode shapes were calculated considering a free-free boundary condition and finally comparison of natural frequency with other sources of vibration acted on the roll-cage to avoid resonance condition under any condition.

It was concluded from modal analysis that first six modes of vibrating frequency does not lie between working frequency of the engine and hence resonance will not occur. The vibration frequency of the engine ranges from 15Hz to 31.667Hz.

Mesh characteristics:

Element length: 10mm

No of elements: 3199

No of nodes: 9553

The matrix solver used Block Lanczos extraction method to evaluate eigenvectors from 0Hz to 1000Hz

Element type: BEAM188 is suitable for analyzing slender to moderately stubby/thick beam structures. The element is based on Timoshenko beam theory which includes shear-deformation effects. The element provides options for unrestrained warping and restrained warping of cross-sections.

Model was constrained of nodal displacement and nodal rotation for all degree of freedom at the suspension mounting points.

Table 13: First six natural modes of chassis

Mode	Frequency
1	63.96 Hz
2	84.1 Hz
3	98.89 Hz
4	105.7 Hz
5	107.22 Hz
6	116.76 Hz

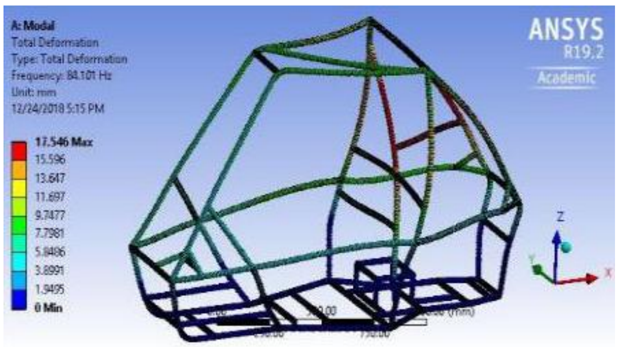

Figure 22: 1st mode deformation

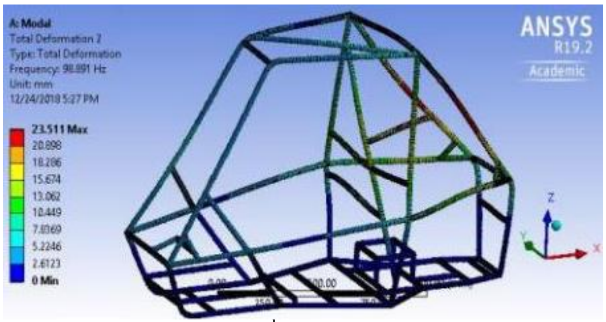

Figure 23: 2nd mode deformation

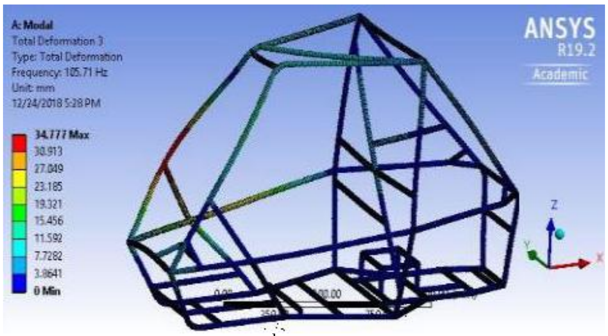

Figure 24: 3rd Mode deformation

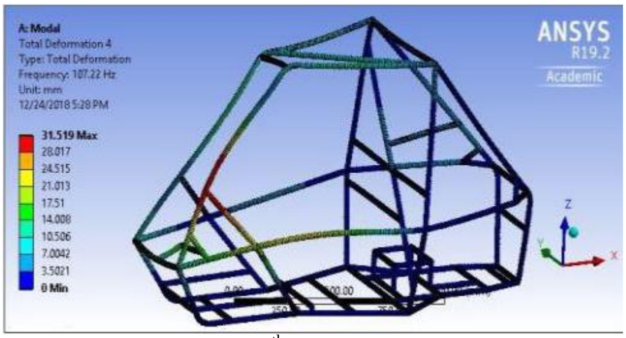

Figure 25: 4th mode deformation

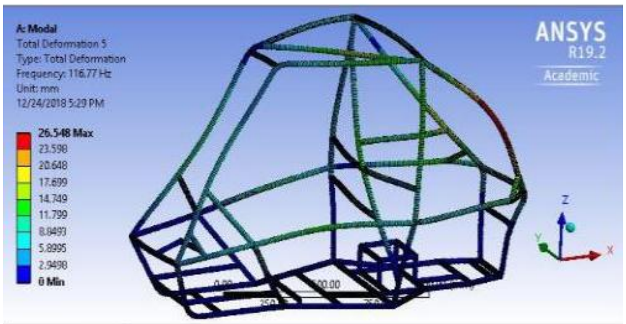

Figure 26: 5th mode deformation

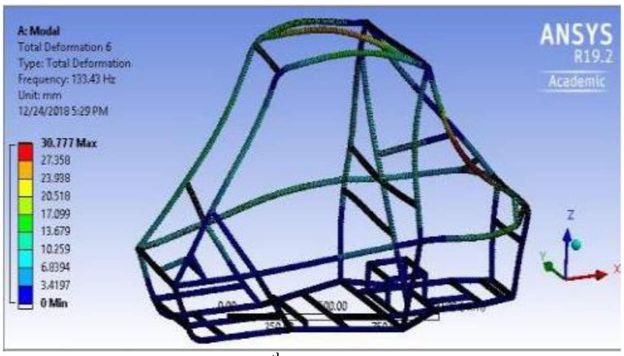

Figure 27: 6th mode deformation

Explicit Dynamic analysis

In mechanics, the static system is the state of a system that is in equilibrium with the action of balanced forces and torque so that they remain at rest. But to get a real case value, dynamic analysis has to be performed. Considering, this is a crash analysis with short duration impact, explicit dynamic analysis had been performed using HyperMesh 17.0. This analysis had been performed for the particular case of impact. Here the case considered is impact with a rigid wall.

The vehicle had to make a head-on collision with the rigid wall from a particular distance from the extreme point of the roll cage.

Components:

Primary and secondary tubes of different dimensions were assigned to different components.

Properties:

Thickness had been decided in this card. Other than that, some important parameters were also decided such as Ishell=24 (QEPH formulation) which reduces the hourglass energy & N=5 (number of integration points).

Material

M36_PLAS_TAB material of Elasto-plastic type had been taken considering the ductile material and the specification of the material had been shown in table.

Table 14: Material Specification

Specification	*Values*
Density(Rho_initial)	7.89e+9 ton/mm3
Young's modulus (E)	210000 N/mm2
Poisson ratio(nu)	0.3
Yield stress (a)	638 MPa

Considering dynamic analysis, a rigid connection had been made for each of the components using RBE2, and the approximate center of mass for each component had been decided in CATIA, which was bolted to the chassis members as they have to be.

Adding rigid mass gave us a clear picture of doing the dynamic analysis as it could be seen in figure 15.

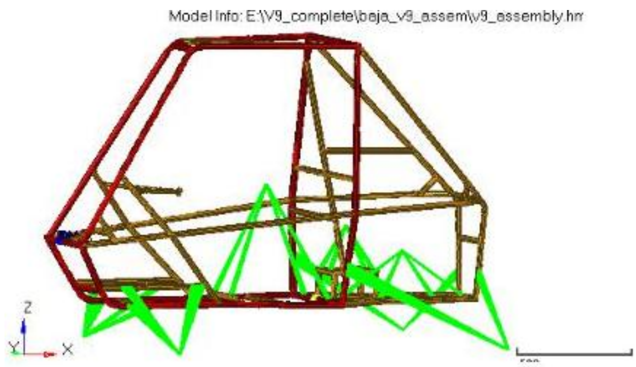

Figure 28: Rigid mass added for each component

The total mass comprising of all components comes out to be 192kg

ISSN: 2319-7064
ResearchGate Impact Factor (2018): 0.28 | SJIF (2018): 7.426

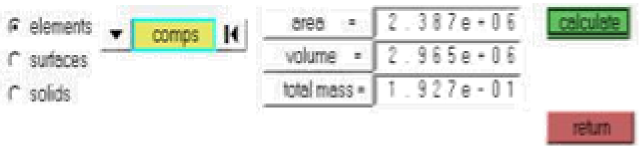

Figure 29: Total mass of the system

Contact

Type 7 contact is created so that the components can interact in the event of impact. Istf value of stiffness scale factor of 4 was considered with gapmin of 1mm, with Inacti taken as 6 for variable gap with consideration of initial penetration.

Load collector

Velocity: - Initial velocity of 45 kmph was imparted to our vehicle using card INIVEL collector and they were made to collide with the rigid wall which is at a distance of 10 mm from the front most part of the vehicle.

Cards

Several cards were made in order to optimize the whole simulation. Some of them were:-

A. ENG_RUN:

T-stop had been calculated and decide here, which is basically total run time for the simulation. For this, the total time taken to hit the second vehicle and coming back to its original position has been taken. Considering the second vehicle at a distance 10mm from the rigid wall, and vehicle moving at speed of 45kmph, the total time would be 0.1 seconds.

B .ENG_TFILE:

The frequency of time after which the animation file would be generated is mentioned here which is taken as 0.001.

C. ENG_ANIM_ELEM:

Those parameters which had to be evaluated other than displacement (default) had been taken here i.e. hourglass energy and Von misses stress.

3. Result and discussion

The post processing had been performed on HyperView. Two results had been inferred from the analysis, Von Mises stress and total displacement.

The maximum stress comes out to be 638 MPa.

An energy curve was also plotted for this analysis on HyperGraph to ensure that the total energy remains constant throughout the process and hourglass energy to be zero.

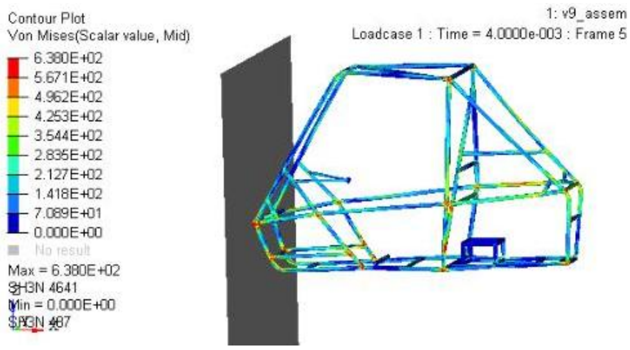

Figure 30: Stress distribution

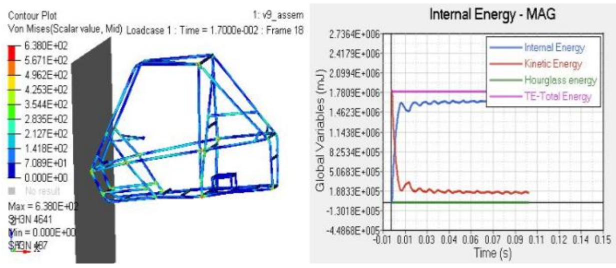

Figure 31: Energy curve vs. time

Tabs:

Various tabs of the AISI 4130 alloy steel grade as that of the roll cage were used to mount the powertrain and the suspension of the car.

These tabs were designed using solidworks and were diligently analyzed on hyperworks for linear static analysis using Optistruct solver.

3d meshing was performed using tetrahedral elements of 4mm size.

The simulations of these tabs are shown below.

Table 15: Shock Absorber Tab

Applied Force	6000N
Deformation	0.69mm
Factor Of Safety	1.8
Max Stress	354.3 MPa

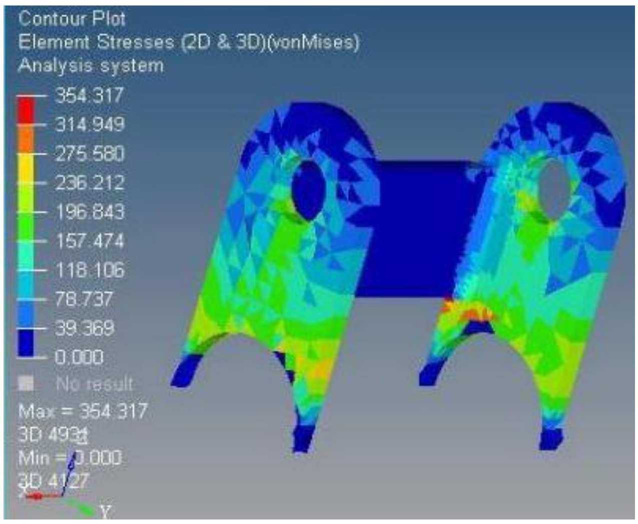

Figure 32: Stress distribution on shock absorber tab

Table 16: Upper A-Arm Tabs

Applied Force	6000N
Deformation	0.78mm
Factor Of Safety	1.73
Max Stress	368.7 MPa

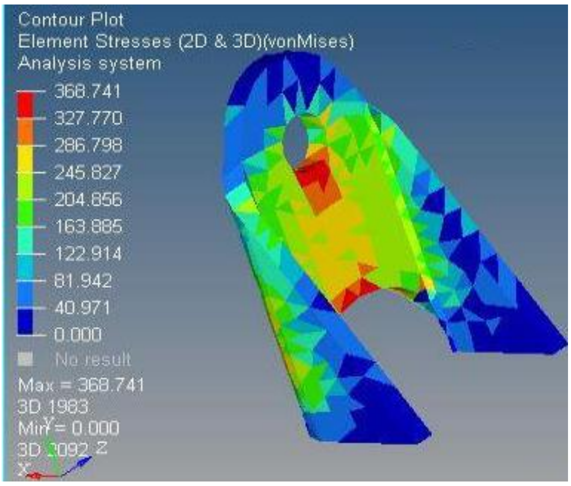

Figure 33: Stress distribution on upper arm tab

Table 17: Lower A-Arm Tabs

Applied Force	6000N
Deformation	0.82mm
Factor of Safety	1.47
Max Stress	434.1 MPa

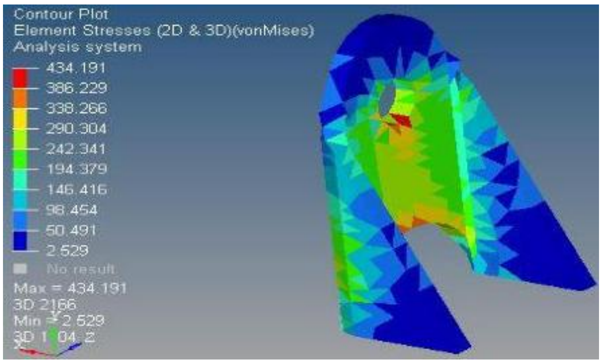

Figure 34: Stress distribution on Lower a-arm tab

Table 18: Gearbox Mounting Tabs

Applied Force	6000N
Deformation	0.73mm
Factor Of Safety	1.12
Max Stress	625.1 MPa

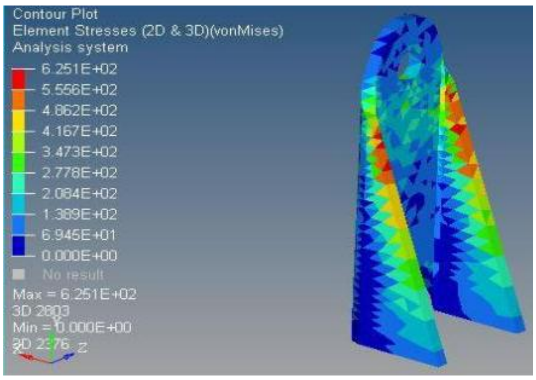

Figure 35: Stress distribution on Gearbox mounting tab

Table 19: H-Arm Mounting Tabs

Applied Force	6000N
Deformation	0.15mm
Factor Of Safety	1.24
Max Stress	511.3 MPa

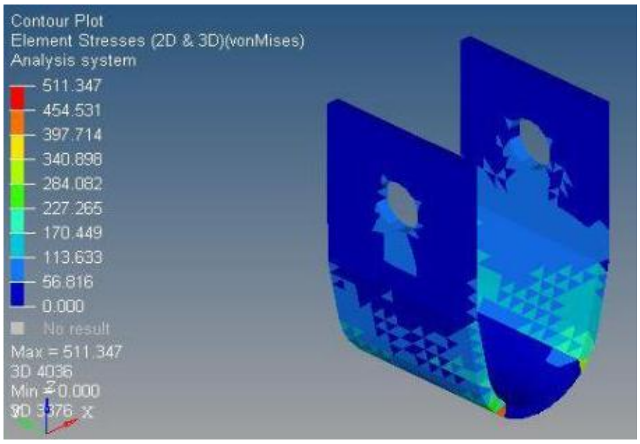

Figure 36: Stress distribution on H-arm mounting tab

SEAT: Glass fiber is used as a reinforcement and hand lay-up technique is used for the manufacturing of the seat. An analysis was run on ANSYS Structural by considering the weight of the driver (80kgs) using ACP (pre) setup.

Material data: Defined Epoxy E-glass wet material of 0.5mm thickness of 6 layers.

Meshing:
No of elements=54114
No of nodes= 64295

Element type is SOLID185, is used for 3-D modeling of solid structures. It is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyperelasticity, stress stiffening, creep, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

Constraints: constrained all the bolt holes

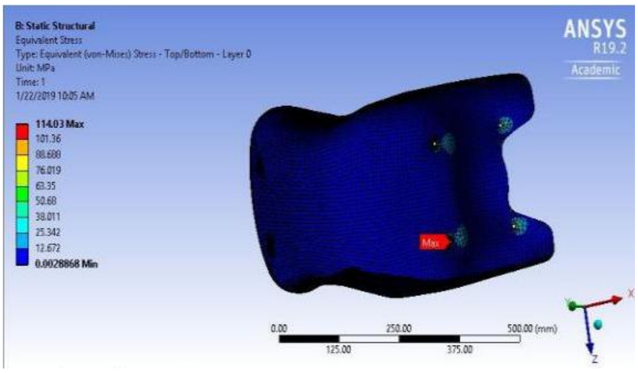

Figure 37: Stress distribution on seat

Table 20: Results of seat analysis

Applied force	1569.6N
Max stress	114.03 MPa
Inverse reverse factor	0.8612
Max deformation	0.5956mm

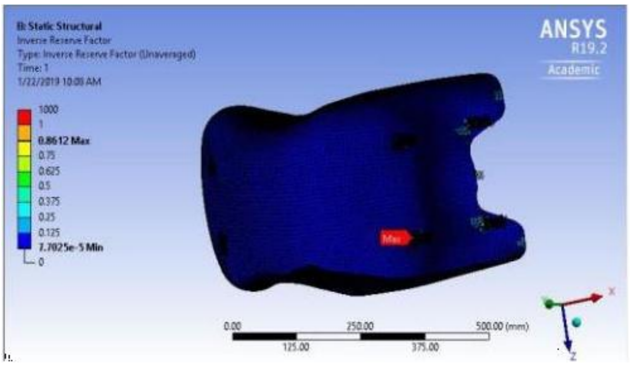

Figure 38: Deformation on seat

4. Conclusions on Fea of Chassis

After performing the Front impact, side impact, and roll over and torsion analyses and making the necessary changes, the following design was finalized.

- The above designed chassis is much stiffer and stronger than the preliminary design. The chassis members were optimized by changing dimensions of the pipes in required positions**.**
- In the case of front impact and side impact analysis, the deformation of the front most member of the roll cage must be less than 10% of the clearance between driver roll cage members ensuring the safety of the driver. Though the factor of safety in front impact is 1.47 and in side impact is 1.37 , deformation is within limit, ensuring that the driver is safe
- For front roll over, the deformation is important than the maximum stress. The deformation is 1.37mm and it is safe for the driver.
- The modal analysis was carried out without any consideration of damping components such as vibration isolators, Panels, etc. If they are included, the frequency will be even much lower.

5. Acknowledgements

Author is grateful to ‘Team Manipal Racing’ for resources and consistent support required for this study**.**

References

[1] Racing and Sports Car Chassis Design, Michael Costin and David Phipps
[2] Thomas D. Gillespie. Fundamentals of Vehicle Dynamics. Society of Automotive Engineers, Inc.
[3] Prof. Dipl.-Ing. Jornsen Reimpell. The automotive chassis: engineering principles (2001). Butterworth Heinemann.
[4] Jonathan Hastle.(2005) Mini-BAJA vehicle design optimisation. North-eastern university.
[5] T. Stolarski, Y. Nakasone & S. Yoshimoto (2006). Engineering Analysis with ANSYS Software. Elsevier Butterworth-Heinemann.
[6] BAJA SAE Rulebook