ISSN (O): 2393-8609

International Journal of Aerospace and Mechanical Engineering

Volume 2 – No.6, December 2015

Design and Analysis of Space Frame Tubular Chassis to be used in Formula SAE

Raman Nain
B.Tech Automotive Design Engineering
University of Petroleum and Energy Studies
nainraman@yahoo.com

Rahul Sharma
B.Tech Automotive Design Engineering
University of Petroleum and Energy Studies
i.rahulsharma10@gmail.com

ABSTRACT
The objective of this paper is to design and analyse a space frame tubular chassis which has to sustain the racing environment. The work in this paper describes the team Formula Racing UPES's car on how a roll cage is designed using SOLIDWORKS to allow finite element analysis to be performed by ANSYS. It illustrates a systematic approach which considers all the parameters and allows the design to incorporate every part of the car correctly. It also describes the factors on which material selection was done and depicts the results of various static structural tests. Throughout these processes, the design and analysis are based upon FSAE 2015-16 rule book.

Keywords
Space Frame Tubular Chassis, Formula Student, SOLIDWORKS, ANSYS, Finite Element Method.

1. INTRODUCTION
Chassis is an essential structural backbone of an automobile, especially in a racing car. Good designs allow a light and stiff chassis to be fabricated at a minimal expense without compromising the safety of the driver. The main objective of this paper is to design and analysis ofa space frame tubular chassis to be used in FSAE. Formula Student (FS) is Europe's most established educational motorsport competition, run by the Institution of Mechanical Engineers. It is the European counterpart of Formula SAE, which has been held annually in the United States from the year 1978. The competition seeks to challenge university students to conceive, design, build and race their own open wheeled race cars.

2. METHODOLOGY
The chassis design is split into a number of bays: the front bay, driver's bay and engine bay. For a single seater car, such as the FSAE model, the first step is to confirm the suspension pickup points and the location of the steering gear, engine and transmission, seating area, pedals, driver position & controls and other bulky items. Once these, as well as the load estimation, wheelbase, track and wheel/tire sizes have been determined, the next step is to place in the main chassis tubes using CAD and modeling software SolidWorks. These should be organized so that there is the largest space possible through the structure for the remaining components. Lastly material selection, mesh generation and analyzing the frame in ANSYS Workbench under various conditions.

3. DESIGN PROCESS
The frame was designed using CAD software SOLIDWORKS

3.1 Initial Step
All the standard dimensions like wheelbase, track width, height, overall length and size of the tyres were selected. Proceeding further from these dimensions roll hoop location, engine mount location, bulkhead location, and wheel centerlines were fixed. Once these locations were fixed a series of planes were created in SOLIDWORKS at these locations to visualize all these dimensions of the frame.

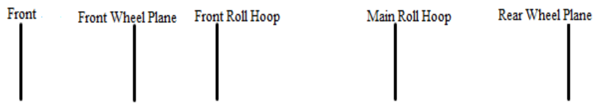

Fig 1: Frame Plane: Side View

3.2 Modeling of Fixed Elements
Roll Hoop, engine mounts, bulkhead, suspension points are the fixed elements which were not moved around during chassis design iteration process so that number of variables to be manipulated can be reduced. It results for a quicker design period.

3.3 Modeling of Variable Elements
Proceeding further, the next milestone was to design the tubes which connect these fixed planes to each other. The arrangement of the tubes was done considering the weight, stiffness and manufacturability of the frame. Node to node triangulation was preferred which results in only tensile or compressive forces in the frame members.

3.4 Final Step
Engine mounting locations are also decided which helped the engine design team to accurately place their engine assembly inside the frame without having to change their parts.

Whole of the frame model was designed of hollow cross section tubes with two different sizes were used during the process. The structure comprises of tube 1" (outer diameter) and 2.5mm wall thickness except main roll hoop and front roll hoop. Both (front and main) hoops are made up of 1" (outer diameter) and 4mm wall thickness. Once shapes of these features are finalized, the tubes are drawn on their respective planes using structural member feature.

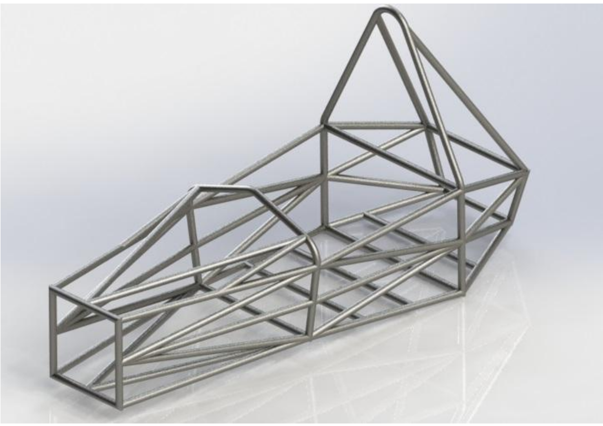

Fig 2: Final Frame modeled in SOLIDWORKS.

4. MATERIAL SELECTION

The material property of the chassis plays an important role while designing and manufacturing the car. Two very commonly used materials for making the space frame chassis are Chromoly 4130 and SAE-AISI 1018. SAE 1018 provides better thermal properties whereas Chromoly is better in terms of strength. It is lighter with thin wall but as strong as thick wall steel tube i.e. higher strength to weight ratio. So frame having less weight can be manufactured using Chromoly 4130.Also considering the safety of the driver as our utmost priority Chromoly 4132 was chosen which has better stiffness and strength.

Table2: Comparison of Mechanical Properties

Material	Chromoly 4130	SAE 1018 Steel
Density	7.8g/cm^3	7.8g/cm^3
Thermal Conductivity	42.7W/m-K	51.9W/m-K
Strength to Weight Ratio	72-133KN-m/Kg	55-60 KN-m/Kg
Brinell Hardness	217	126
Elastic Modulus	190-210GPa	205GPa
Yield Strength	480MPa	370MPa
Ultimate Strength	590MPa	440MPa

5. STATIC STRUCTURAL TESTS

Structural design was followed by its testing and subsequent validity. To determine whether the frame is safe for the driver or not before construction, finite element analysis could serve the purpose. All the tests are performed by using simulation software ANSYS Workbench. The following coordinate system and labeling convention is used within these rules

-Longitudinal(X)

-Transverse(Y)

-Vertical(Z)

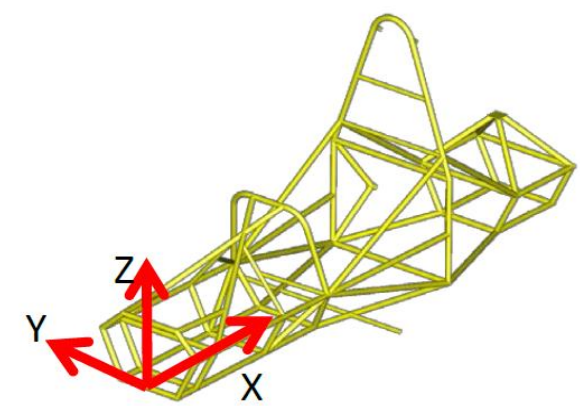

Fig 3: Coordinate system.

5.1 Front Impact Test

Load Applied: F_x= 120000Newton.

Application point: the actual attachment points between the impact attenuator and the front bulkhead

Boundary Condition: Fixed displacement (x,y,z) but not rotation of the bottom nodes of both sides of the main roll hoop and both locations where the main hoop and shoulder harness tube connect.

The deformation and stresses induced are shown below. Maximum combined stress induced is 304MPa which provides FOS of 1.55. Maximum deformation obtained was 2.8mm.

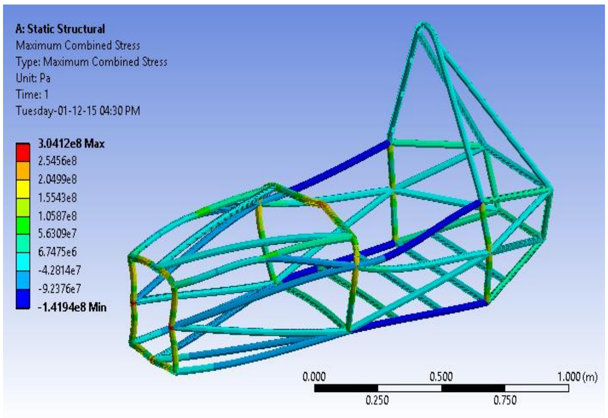

Fig 4: Max. Stress in front impact using ANSYS WB.

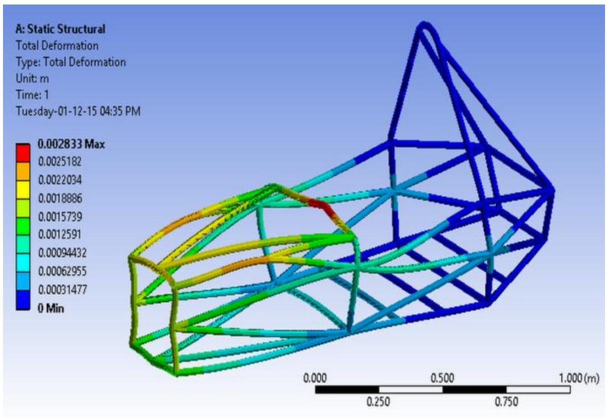

Fig 5: Max. Deformation in front impact using ANSYS WB.

ISSN (O): 2393-8609
International Journal of Aerospace and Mechanical Engineering
Volume 2 – No.6, December 2015

5.2 Side Impact Test

Load Applied: F_y = 7000Newton of lateral load in direction toward the driver.

Application point: All structural locations between front roll hoop and main roll hoop.

Boundary Condition: Fixed displacement (x,y,z) but not rotation of the bottom nodes of both sides of the front and main roll hoops.

The deformation and stresses induced are shown below. Maximum combined stress induced is 135MPa which provides FOS of 3.55. Maximum deformation obtained was 1.9mm.

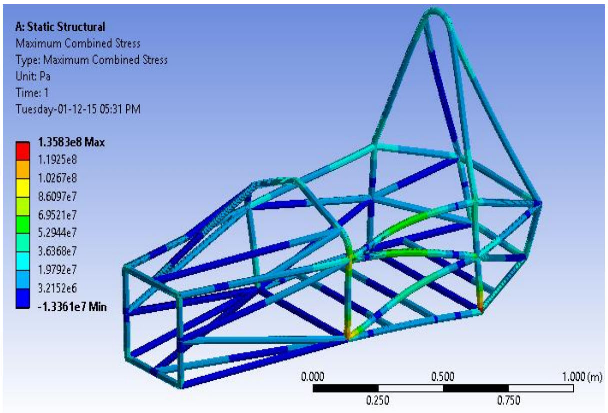

Fig 6: Max. Stress in side impact using ANSYS WB.

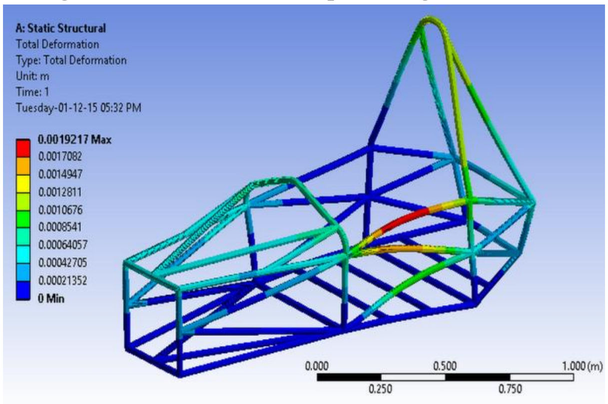

Fig 7: Max. Deformation in side impact using ANSYS WB.

5.3 Front Roll Hoop Test

Load Applied: F_x= 6000 Newton, F_y= 5000 Newton, F_z= -9000 Newton.

Application point: Top of front roll hoop.

Boundary Condition: Fixed displacement (x,y,z) but not rotation of the bottom nodes of both sides of the front and main roll hoops.

The deformation and stresses induced are shown below. Maximum combined stress induced is 351MPa which provides FOS of 1.36. Maximum deformation obtained was 3.4mm.

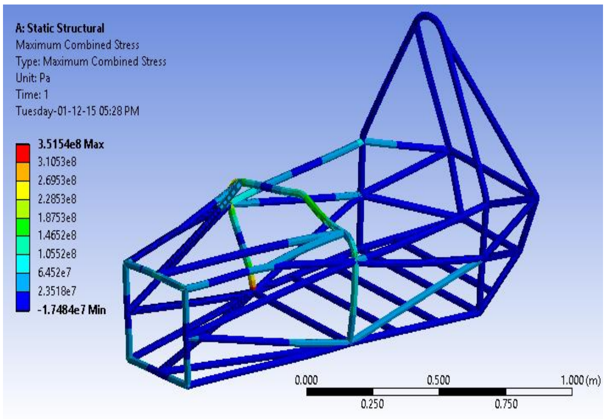

Fig 8: Max. Stress in front roll hoop test using ANSYS WB

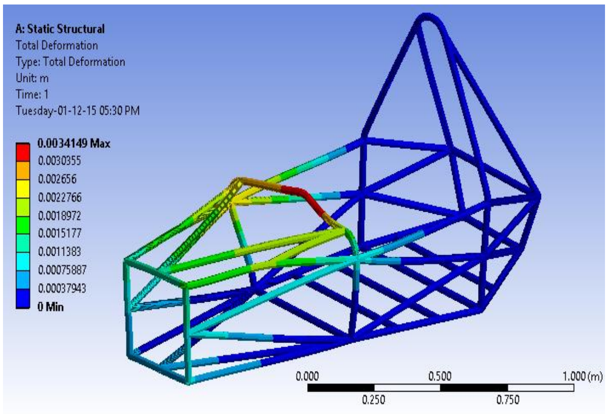

Fig 9: Max. Deformation in front roll hoop test using ANSYS WB.

5.4 Main Roll Hoop Test

Load Applied: F_x= 6000 Newton, F_y= 5000 Newton, F_z= -9000 Newton.

Application point: Top of main roll hoop.

Boundary Condition: Fixed displacement (x,y,z) but not rotation of the bottom nodes of both sides of the front and main roll hoops.

The deformation and stresses induced are shown below. Maximum combined stress induced is 312MPa which provides FOS of 1.53. Maximum deformation obtained was 5.6mm.

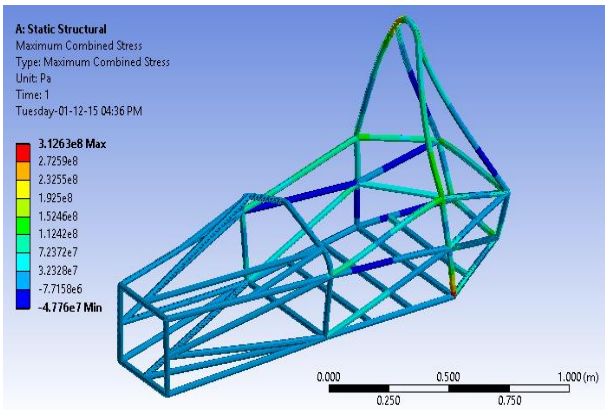

Fig 10: Max. Stress in main roll hoop test using ANSYS WB

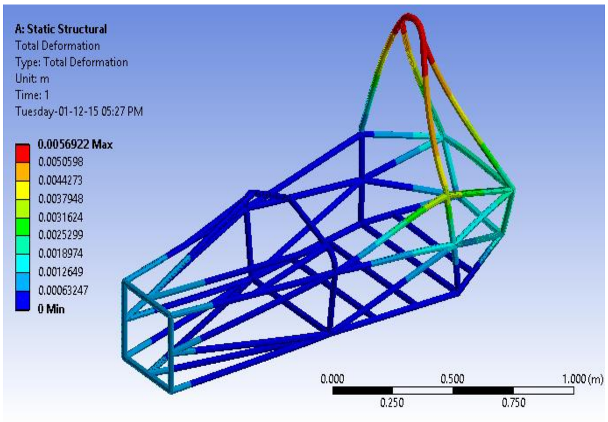

Fig 11: Max. Deformation in main roll hoop test using ANSYS WB.

6. CONCLUSION

Table3: Results

Case	Maximum Stress (MPa)	Maximum Deformation (mm)	FOS
Front Impact	304	2.8	1.55
Side Impact	135	1.9	3.55
Front Roll Hoop	351	3.4	1.36
Main Roll Hoop	312	5.6	1.53

Since the stresses induced in each of the tests performed is less than the yield strength of the material it can be considered that the designed frame is safe for the driver.

7. REFERENCES

[1] 2015 Formula SAE® Rules, SAE International.

[2] R. K. Rajput, (2007). Strength of Materials, 4th Ed.S.Chand Inc.

[3] Nitin S.Gokhale,(2008).Practical Finite Element Analysis, Finite To Infinite.

[4] Richard G Budynas, (2011). Shigley's Mechanical Engineering Design, McGraw-Hill Education (India) Private Limited.