

FEA Analysis of FSAE Chassis

¹. Rupesh Patil

1.Engineering Student,
Dept. Mechanical Engineering,
Sinhgad Academy of Engineering
Pune, India

². Vinayak Chikkali

2.Engineering Student,
Dept. Mechanical Engineering
Sinhgad Academy of Engineering
Pune, India

Abstract — This Research Paper mainly presents an analysis of FSAE chassis. This chassis is design by conforming the rules of FSAE. The chassis frame is a main structural component to which all the other parts are attached. The chassis frame acts as the skeleton of the automobile so it should be able to withstand against various load condition. FEA Analysis Conforms ability to assess the forces and safety of design prior to manufacturing. Material Preferred for FSAE chassis is AISI4130 alloy of chromium and Molybdenum which has high strength as well as low weight. Analysis is done on standard Results to get maximum strength and minimum weight possible. Software tools used for analysis is Ansys.

Keywords— FSAE, Modal Analysis, FEA, AISI4130, Torsional Analysis

I. INTRODUCTION

SAE (society of automotive engineers) conducts the competition in which student build, design, and compete in formula student race car these help in providing the educational theory and hands on experience. Rules of SAE are analogous to code of conduct at work force it also incorporates the Team Work, Project management and Finance. The SAE competition is consisting of static and dynamic event where they are judge based on various parameters like conforming rules with performance of vehicle, team work and vehicle compliances. The main philosophy behind the analysis is to assure the driver safety in the cockpit. FEA Analysis is carried out by considering various Loads like bending, Torsional, Twisting, on Side Impact and Front Impact. FSAE chassis undergo various types of forces in dynamic event, FEA analysis helps to decide the manufacturing process has to carried out in order to make chassis as stiffer as possible with minimal cost. Material selection also plays a vital role in terms of strength, torsional rigidity. The paper focuses on the different configuration and actions of the chassis used in the FSAE competition when the chassis is used with front impact, side impact and torsional stiffness by the rules of the Formula SAE competition.

II. METHODOLOGY

FINITE ELEMENT ANALYSIS (FEA) is carried out to determine the stiffness of chassis frame before construction. Conventionally in FEA, the frame is subdivided into elements. Nodes are placed where tubes of frame join. There are many types of elements possible for a structure and every choice the analyst makes can affect the results. The number, orientation and size of elements as well as loads and boundary conditions are all critical to obtain meaningful values of chassis stiffness. Beam elements are normally used to represent tubes. The assumption made in using beam elements is that the welded

tubes have stiffness in bending and torsion. If a truss or link elements were used, the assumption being made would be that the connections do not offer substantial resistance to bending or torsion. Another aspect of beam element is the possibility of including transverse shearing effects. While modeling the stiffness contribution from each part of the frame, method to apply the loads and constrain the frame plays significant role for an accurate analysis. Accurate analysis means to predict the stiffness of frame close to actual stiffness as the frame operates in real conditions. The problem here has normally been how to constrain and load a frame, so to receive multiple load inputs from a suspension, while it has been separated from that suspension and many other such problems. For practical reasons, it is recommended that the load on the chassis frame, including its own weight should be applied at the joints (nodes) of structural members. These point loads were statistically equivalent to the actual distributed load carried by the vehicle. The analysis was done in Ansys R17.2. We choose to do 2D shell analysis as it gives appropriate result. Elements selected were 2D QUADS and 2D R-TRIAS. We also inserted dead mass to account for the inertia of the mass of engine.

III. MATERIAL SELECTION

Material selected for Chassis is AISI4130, also referred to as Chromyl Steel, is a chromium and molybdenum alloy steel Known for its responsiveness to mechanical and heat treatments. It is low carbon steel and offers lighter weight still maintain high strength. The mechanical properties of AISI4130 is listed in the following table.

TABLE 1

Properties	Metric
Tensile Strength, Ultimate	760 MPa
Tensile Strength, Yield	435 MPa
Modulus of elasticity	205 GPa
Bulk Modulus	140 GPa
Poisson's Ratio	0.29

The Chemical Composition of AISI4130 is listed in the following table.

TABLE 2

Element	Content%
Iron, Fe	97.03 - 97.22
Chromium, Cr	0.80 - 1.10
Manganese, Mn	0.40 - 0.60
Molybdenum, Mb	0.15-0.25
Carbon, C	0.280-0.330
Silicon, Si	0.15 - 0.30
Sulphur, S	0.040
Phosphorous, P	0.035

IV. ANALYSIS

Four types of analysis are performed on FSAE chassis. As Chassis undergo Bending due to weight of mounted components on Vehicle like engine, fuel tank, dampers etc. At dynamic event in FSAE, chassis evaluates for acceleration and cornering ability which leads to twist in chassis whereas it can be judge with the help of Torsional analysis. Types of Analysis performed are as follows.

- 1) Static Structural analysis
- 2) Torsional Analysis
- 3) Bending Analysis
- 4) Modal Analysis

Static Structural analysis are performed on two different cases of Chassis.

Two different cases are as follows

Case 1 Side Impact

Case 2 Front Impact

The FSAE car chassis must be rigid enough because it has to withstand vibration, twist, buckling, shock and many other stresses. Chassis must strong enough to allow adequate bending and torsion stiffness in order to improve the handling system.

A. Case 1 Side Impact

The velocity of car is assumed to be 70 km/hr. Side member of roll cage is in contact with wall. One side of the frame was constrained and calculated load was applied on the opposite side impact structure. Worst case considered is 4.5G.

Parameters to be considered are shown in table below.

TABLE 3

Parameters	Values
Velocity	46 Km/hr
Car Weight	220 Kg
Driver Weight	60 Kg

Calculations

The total force acting on the cockpit members was calculated using following empirical relations:

- Final velocity (v) = Initial velocity (u) + acceleration (a) * time (t)
- $11.11 = 0 + \text{acceleration (a)} * 0.3$
- Acceleration (a) = 37.03 m/s² (retarding -ve X axis)
- Total force acting on the body = Total mass of car * Acceleration
- Total force acting on the body = $280 * 37.03\text{N}$
= 10368.4 N (-ve X axis)

This is the total calculated load, which has been acting on the side impact structure of FSAE chassis.

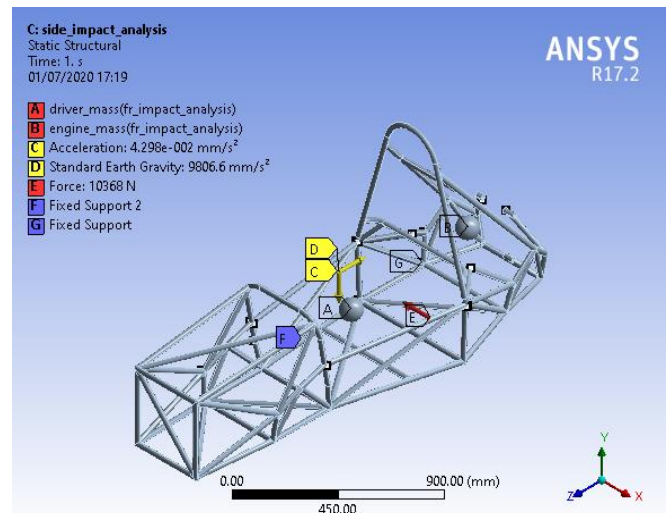


Figure 2 Boundary condition for Side Impact Structure

A boundary condition is a place on a structure where either the external force or the displacement are known at the start of the analysis. The boundary conditions are the places where the structure interacts with the environment either through the application of an external force or through some restraint that is imposing a displacement. For a structural analysis problem to be solvable, every location on the boundary of our structure must have a known boundary condition. For example, we may know that there are locations on our structure that have no external force. A displacement boundary condition that is zero is equivalent to the structure being held in place at that location. They simply impose deformations on boundaries in your model (usually equal to zero). Each point in space can translate in 3 directions and rotate around 3 axes.

TABLE 4

Total Force on Side Impact Structure		10368(on side impact member -ve X axis)
Boundary Conditions	Translational Displacement	Rotational Displacement
Right Side front Suspension	X = 0	X = Fixed
	Y = 0	Y = Fixed
	Z = 0	Z = Fixed
Right Side Rear Suspension	X = 0	X = Fixed
	Y = 0	Y = Fixed
	Z = 0	Z = Fixed

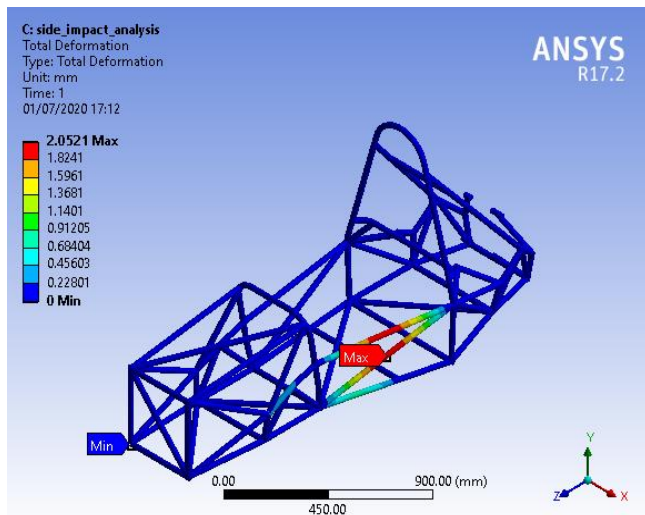


Figure 2 Deformation Analysis of Side Impact Structure

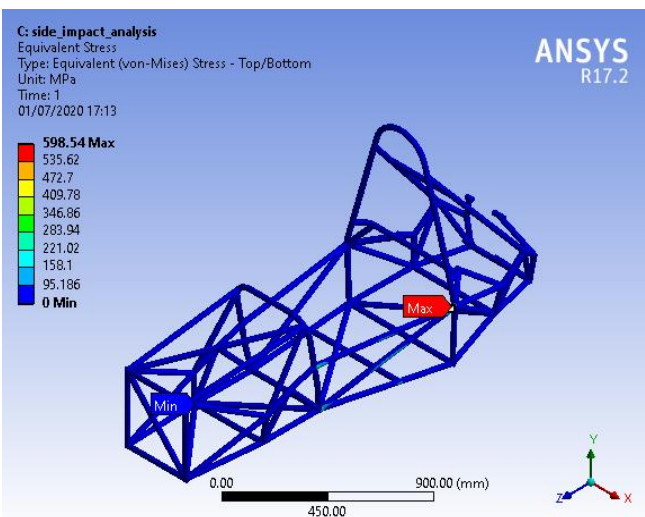


Figure 3 Von Mises Stress Analysis of Side Impact Structure

Result

TABLE 5

Maximum Deformation	2.0521 mm
Von Misses Stress	598.54 MPa
Factor of Safety	$UTS/WS = 670/598.54 = 1.11$

B. Case 2 Front Impact Analysis

The Front Impact Analysis is done to find out the amount of bending stress as well as deformation generated in the FSAE chassis if the car hits a solid body from the front whereas it is more likely to happen in any worst case. In order to Simulate the frontal impact crash, the vehicle was assumed to impact upon a stationary object. Upon research it was found that the maximum force that a human body can withstand is 5G. For front Impact analysis suspension pick up point of the frame was Constrained and loads were applied at the fore most end. Worst case scenario considered for the analysis was 7.6G Parameters to be considered for calculations are given in table below

TABLE 6

Parameters	Values
Velocity	85 Km/ hr
Car Weight	220 Kg
Driver Weight	60 Kg

Calculations

The total force acting on the front bulkhead was calculated using following empirical relations:

- Final velocity (v) = initial velocity (u) + acceleration (a) * time (t)
- $23.61 = 0 + \text{acceleration (a)} * 0.3$
- Acceleration (a) = -78.7 m/s^2 (retarding)
- Total force acting on the body = $280 * 78.7 \text{ N} = 22,036 \text{ N}$ (-ve Z axis)

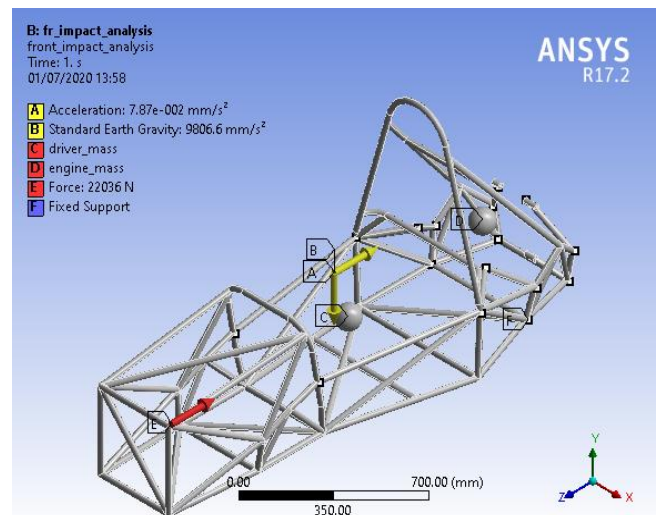


Figure 4 Boundary Conditions of Front Impact Structure

TABLE 7

Total force acting on Front Impact Structure		22036N (on front Impact member-ve Z axis)
Boundary Condition	Translational Displacement	Rotational Displacement
Front Suspension	X = Free	X = Free
	Y = Free	Y = Free
	Z = Free	Z = Free
Rear Suspension	X = 0	X = Fixed
	Y = 0	Y = Fixed
	Z = 0	Z = Fixed

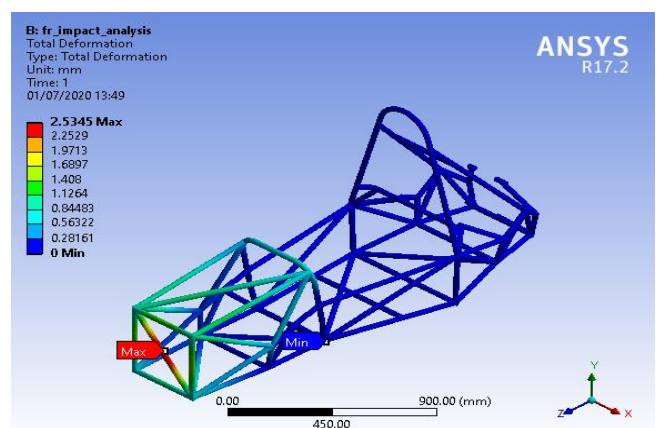


Figure 5 Deformation Analysis of Front Impact structure

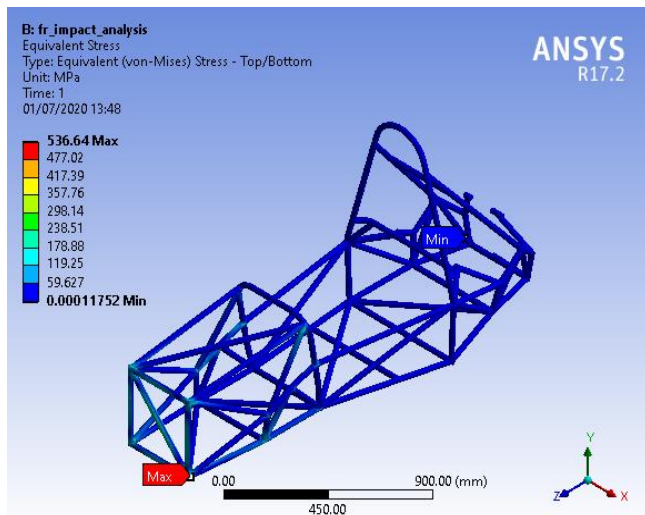


Figure 6 Von Mises Stress Analysis of Front Impact Structure

Result

TABLE 8

Maximum Deformation	2.5345 mm
Von Mises	536.64 MPa
Factor of Safety	UTS/WPS = 670/536.64 = 1.24

V. TORSIONAL ANALYSIS

Torsional stiffness is used to determine the torsional stiffness of structure i.e. how much the structure can resist the twisting. It is a very important parameter of any vehicle to resist torsional stresses and deflection during turning, drifting cornering and undulating road surface. It is good to have high torsional stiffness because it permits to precisely control handling parameters by adjusting the suspension parameters. If car is sufficiently stiff under torsional loading, then it will normally withstand bending and longitudinal/lateral bending. Torsional rigidity validates/rejects the chassis structure. In this case, chassis is assumed to act as a cantilever with one end fixed and other end free and subjected to torque about its longitudinal axis. Clockwise couple of 3000N was applied on front suspension pickup points by fixing rear suspension pickup points.

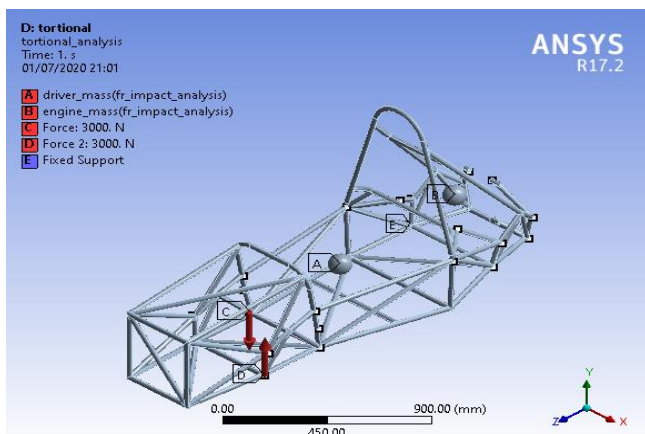


Figure 7 Boundary Conditions for Torsional Analysis

TABLE 9

Total Force acting on Body		3000N (Right side Front Suspension mountings in Y axis) -3000N (Left Side front Suspension mountings in Y axis)
Boundary Condition	Translational Displacement	Rotational Displacement
Front Suspension	X = Free	X = Free
	Y = Free	Y = Free
	Z = Free	Z = Free
Rear Suspension	X = 0	X = Fixed
	Y = 0	Y = Fixed
	Z = 0	Z = Fixed

For torsional analysis force of 3000 N is applied on left side front suspension in up word direction and 3000 N of force is applied on right side front suspension in down word direction. it produces a couple which apply the torque on chassis the rear suspension mountings are fixed in translational and rotational displacement during torsional analysis This approach adopted in finding an ideal stiffness focused around finding the least stiff the chassis could be before major performance or handling losses would arise. This method was chosen because stiffness is so closely related to vehicle weight, and for a Formula SAE vehicle, weight is the most sensitive parameter on the car.

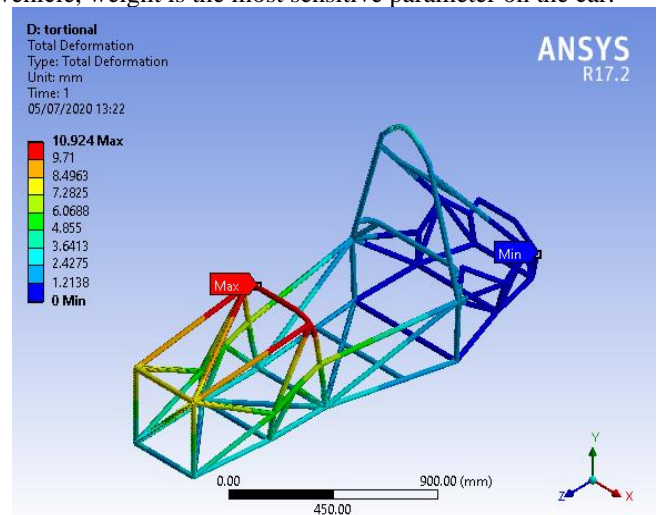


Figure 8 Torsional Deformation Analysis of Chassis

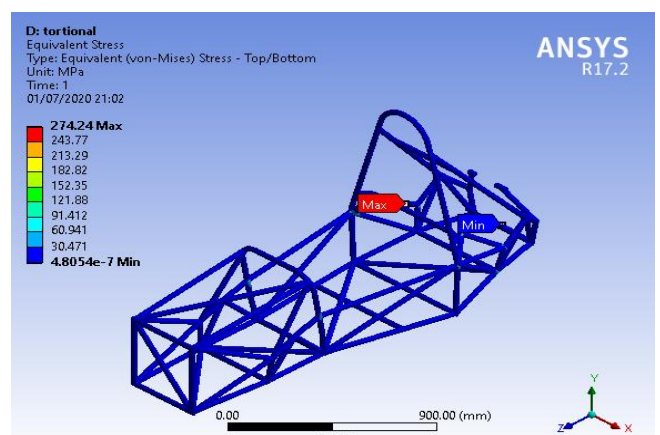


Figure 9 Von Mises Stress due to Torsional deflection

Initial Assumptions

Driver weight = 60 kg

Car Weight = 220 kg

Total weight = 280 kg

Force (F) = 3000 N

Calculations

- Torque = $F \times (1/2) \text{ Track width (1)}$
 $= 3000 \times 0.607$
 $= 1821.00 \text{ Nm}$
- $\theta = \text{Angle of Deflection}$
 $= \tan^{-1} (\text{vertical displacement} / (1/2) \text{ track width})$
 $= \tan^{-1} (10.924 / 607)$
 $= 1.031 \text{ degree}$
- Torsional Stiffness (k) = torque/angle of deflection
 $= 1821.00 / 1.031$
 $= 1766.27 \text{ Nm/degree}$

Result

TABLE 10

Maximum Deformation	10.924 mm
Von Misses Stress	274.24 MPa
Factor of Safety	UTS/WS = 670/274.24 = 2.44

VI. BENDING ANALYSIS

Vertical bending strength is a strength that shows chassis endurance towards other car components 'weight such as engine, body, wing, drivetrain, and driver under gravitation effect. Vertical bending using car weight to determine whether the frame is strong enough to hold car's weight. Vertical bending strength can be found by applying static force with car's components weight. The one of the ways to calculate bending deflection is rear suspension mountings are acting as fixed supports and a force is acting in vertical direction on front suspension mountings

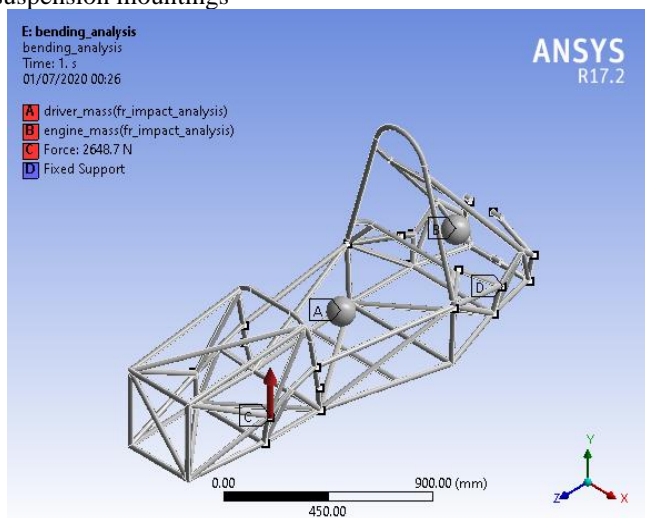


Figure 10 Boundary Conditions for Bending Analysis of Chassis

TABLE 11

Total Force Acting on Chassis (Total Weight)		2648.7 N
Boundary Condition	Translational Deformation	Rotational Deformation
Front Suspension	X = Free	X = Free
	Y = Free	Y = Free
	Z = Free	Z = Free
Rear Suspension	X = 0	X = Fixed
	Y = 0	Y = Fixed
	Z = 0	Z = Fixed

Calculations

Bending stiffness of roll cage is calculated by using formula:

- $K = W/d$
- $K = 2648.7/1.46$
- $K = 1814.17 \text{ N/mm}$

Whereas,

K=Bending stiffness of roll cage.

W= Applied force

d= deformation

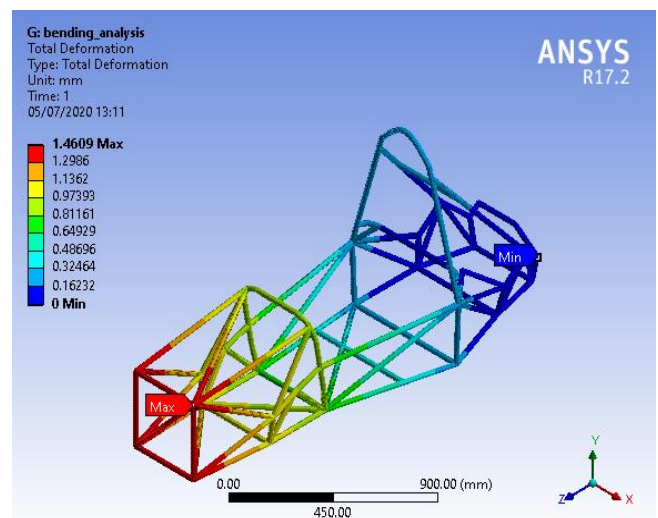


Figure 11 Bending Deformation Analysis of Chassis

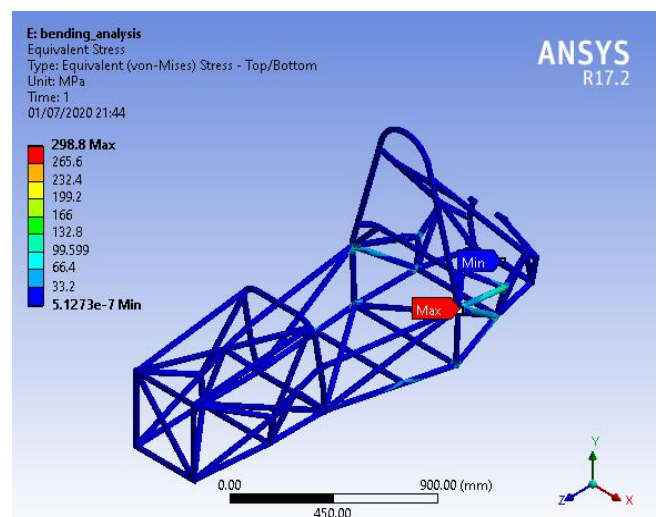


Figure 12 Von Mises Stress due to Bending of chassis

Result

TABLE 12

Maximum Deformation	1.46 mm
Von Mises Stress	298.8 MPa
Factor of Safety	UTS/WS = 670/298.8 = 2.24

VII. MODAL ANALYSIS

Modal analysis is the study of the dynamic properties of structures under vibrational excitation. Modal analysis is the field of measuring and analyzing the dynamic response of structures and or fluids during excitation. In cars it is used to determine natural frequencies of vibrations. Higher the natural frequency better is the structure to sustain Modal analysis determines the mode shape (vibration shape) and frequencies for the particular mode shape of a structure for a free vibration analysis. Normal Modes Analysis, also called eigenvalue analysis or eigenvalue extraction, is technique used to calculate the vibration shapes and associated frequencies that a structure will exhibit. It is important to know these frequencies because if cyclic loads are applied at these frequencies, the structure can go into a resonance condition that will lead to catastrophic failure. It is also important to know the shapes in order to make sure that loads are not applied at points that will cause the resonance condition. Modal analysis has been done to ensure that the natural frequency of the roll cage does not match the natural frequency of the engine.

Natural frequency of engine is 42 Hz.

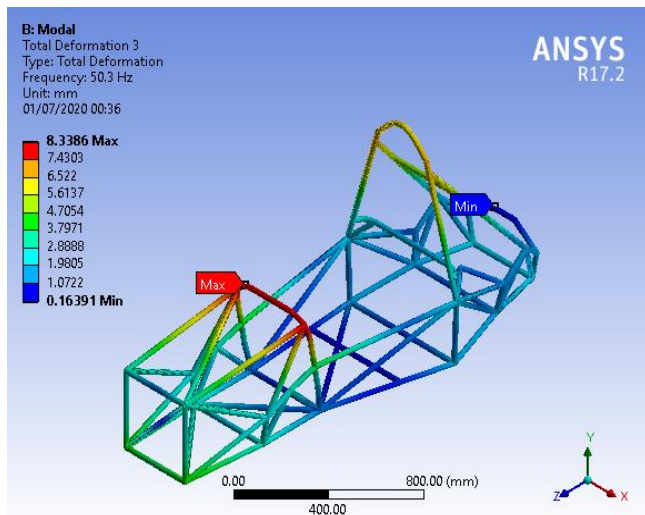


Figure 13 Modal Analysis of Chassis

TABLE 12

No Modes	Frequency (Hz)	Deformation(mm)
1)	50.3	8.3386
2)	81.876	7.0605
3)	87.16	9.3975
4)	102.72	11.666
5)	103.58	11.904
6)	131.01	12.457
7)	143.23	14.145
8)	145.35	10.997
9)	161.00	12.209
10)	168.13	16.877

The fundamental of modal analysis using measured frequency response function data is about curving fitting the data using a predefined mathematical model of the measured structure. This model assumes the number of DOFs of the structure, damping measured frequency range type and possibly the number of vibrations modes within the measured frequency range.

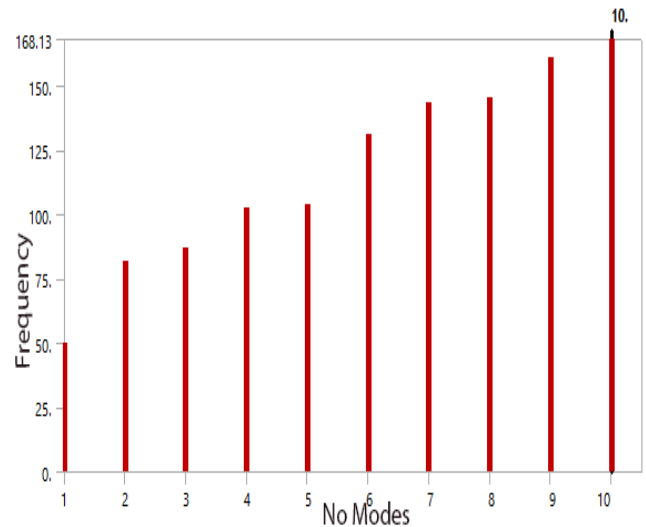


Figure 14 Graph of Frequency Vs No modes

VIII. CONCLUSION

Overall goals for the chassis included, increasing torsional rigidity and decrease weight, deformation. Chassis should perform at an optimum level. The results gained in this section show that the chassis will experience very minimal deflections under race conditions. This is also going to allow the chassis to perform at an optimum level in conjunction with the rest of the vehicle setup. Like any vehicle, the chassis is the backbone of the system. Every critical component relies on the chassis either directly or indirectly. The driver of the vehicle also relies on the chassis for protection in the event of an accident. This dissertation covers the procedures that were used to successfully design and construct a functional chassis

IX. REFERENCES

- [1] William F. Milliken and Douglas L. Milliken, "RACE CAR VEHICLE DYNAMICS", Society of Automotive Engineers Inc., pg. 673-667, 1995
- [2] Daryl L. Logan, "FIRST COURSE IN FINITE ELEMENT ANALYSIS",
- [3] Thomas D. Gillespie. Fundamentals of Vehicle Dynamics. Society of Automotive Engineers, Inc.
- [4] Herbs Adams, "CHASSIS ENGINEERING",
- [5] William B. Riley and Albert R. George, Cornell University, "Design, Analysis and Testing of a Formula SAE Car Chassis", 2002
- [6] Carroll Smith, "TUNE TO WIN", 1978, Inc.
- [7] J Mohammad Al Bukhari Marzuki, Mohd Arzo Abu Bakar and Mohammad Firdaus Mohammed Azmi, "Designing Space Frame Race Car Chassis Structure Using Natural Frequencies Data from Ansys Mode Shape Analysis", International journal of information systems and engineering, April 2015