DESIGN AND ANALYSIS REPORT OF CHASSIS

Abstract — This Design Report mainly analyzes the **Formula Student Racing Car chassis**. This chassis is designed by conforming to the rules of **SUPRA SAE 2023**. The chassis frame is the main structural component to which all the other parts are attached. The chassis frame acts as the skeleton of the automobile, so it should withstand various load conditions. FEA Analysis conforms to the ability to assess the forces and safety of design before manufacturing. The material Preferred for this chassis is **AISI 1018**, which has good strength and ductility and low weight and is economically viable. The analysis is done on standard Results to get the maximum strength and minimum weight possible. **The software tool used for designing is SOLIDWORKS, and for analysis is ANSYS.**

INTRODUCTION

The SAE competition comprises static and dynamic events, where the teams are judged based on various parameters, such as conforming to the rules, the vehicle's performance, teamwork, and vehicle compliance. The safety of the driver in the cockpit is of utmost importance. Hence, FEA analysis is carried out by considering various loads, such as **bending**, **torsion**, **twisting**, **side impact**, and **front impact**.

The FSAE chassis undergoes various forces during the dynamic event, and FEA analysis helps decide the manufacturing process needed to make the chassis as rigid as possible with minimal cost. Material selection also plays a vital role in strength and torsional rigidity. Therefore, the FEA analysis helps optimize the design and material selection to achieve the desired stiffness and stability while keeping costs low.

The paper focuses on the results of the analysis of the chassis used in the FSAE competition. Here we first explain the **methodology of analysis** and **material property**. Then, we will explain **analysis calculations and simulation results** when the chassis is used with front impact, side impact, rear impact, and torsional stiffness by the rules of the SUPRA SAE competition.

METHODOLOGY

The process typically begins with the team researching and analyzing the regulations set by SUPRA SAE, including weight and size limitations, safety requirements, and performance goals.
We then use this information to create a conceptual chassis design, refined and optimized using CAD software.

Once the design was finalized, we analyzed the chassis using finite element analysis (FEA) through ANSYS to ensure it met strength and stiffness requirements and safety regulations. **FINITE ELEMENT ANALYSIS (FEA)** [1] is carried out to determine the stiffness of the chassis frame before construction. Conventionally in FEA, the frame is subdivided into mesh elements. These meshes are further divided into nodes. Many elements are possible for a structure, and every choice the analyst makes can affect the results. The number, orientation, and size of elements and loads and boundary conditions are critical to obtaining meaningful and accurate values of chassis stiffness. **While doing FEA, we chose tetrahedral elements for meshing and kept the average element size at 10 mm.** We did conduct a mesh convergence study by also testing results at an element size of 5 mm. We didn't observe quite a deviation, so we finalized the element size to be kept as 10 mm for simplification in computation.

After finalizing the design, the designing team works with the manufacturing team to ensure that the chassis can be produced within the specified budget and manufactured to the highest quality Standards.

After our research, we optimized the dimensions of the chassis.

1. Wheelbase = 1589.14 mm
2. Mass = 32.85 kg
3. Material = AISI 1018

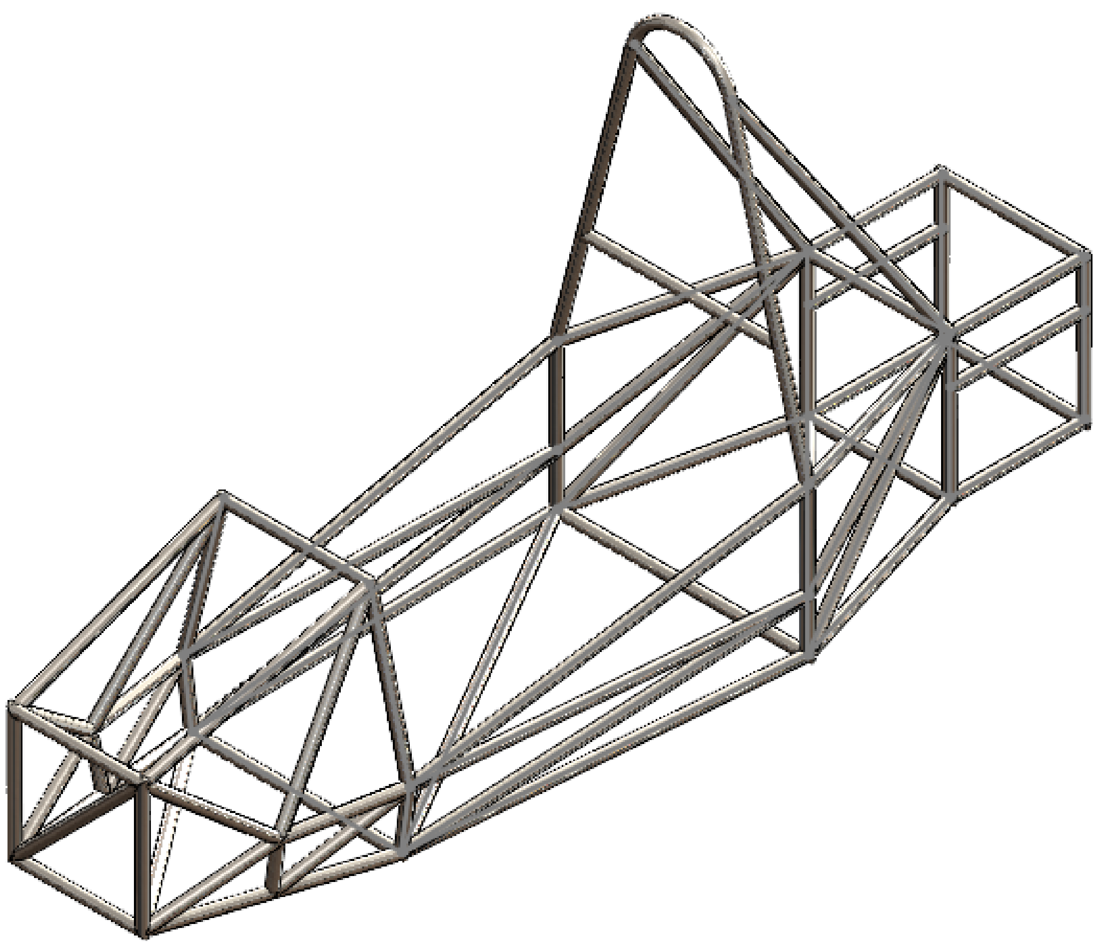

Fig 1: <u>Isometric view of chassis</u>

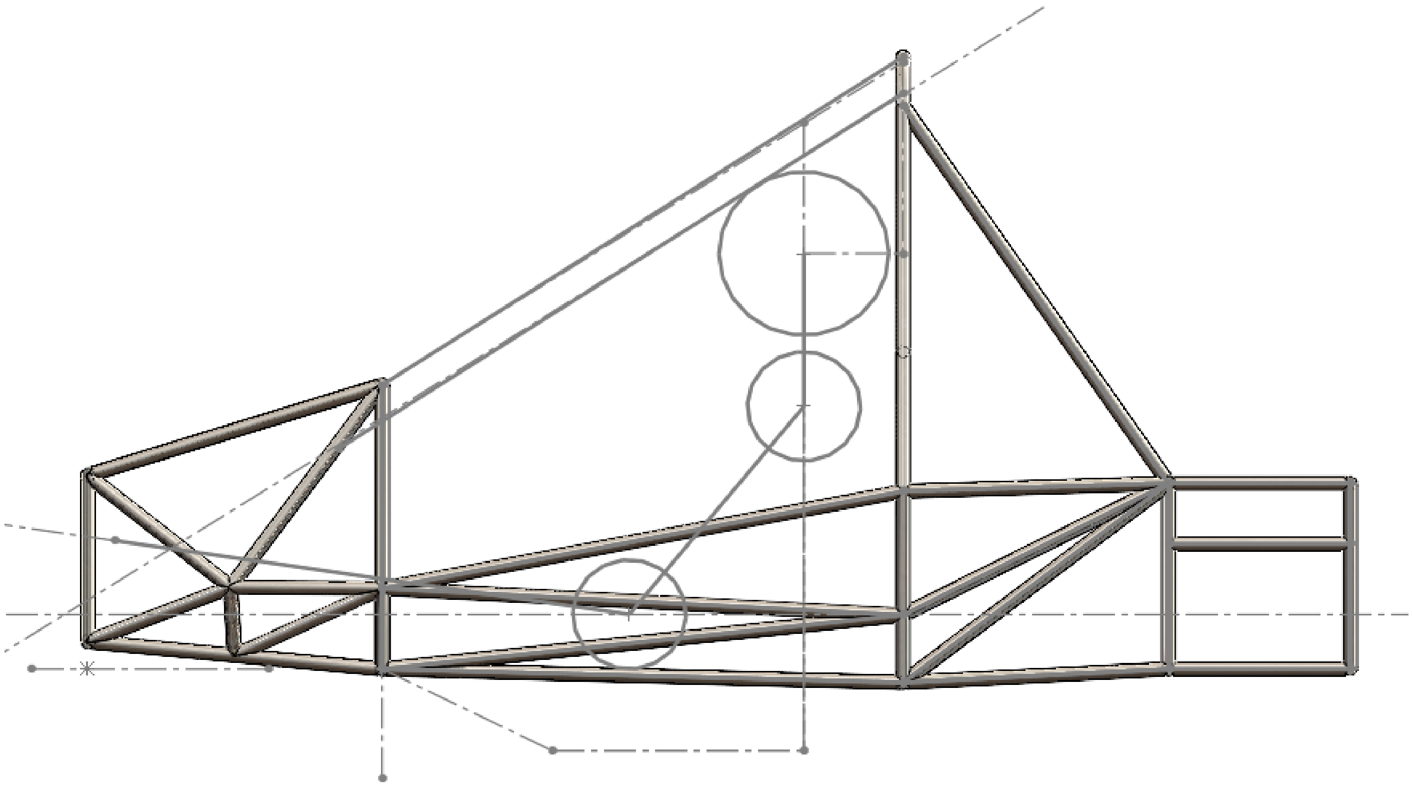

Fig 2.1: **Side view of chassis with a sketch of 95th percentile man**

DESIGN PROCESS

We began designing our chassis in September 2023. The Final draft and all analyses were completed by December 2023. The procurement of material and its manufacturing will be completed by February 2023. Hence in this section, the stages of our design have been discussed in detail.

1.) Understanding constraints and subsequent steps

We first go through the rulebook of SUPRA SAE and understand the constraints like:-

- Triangulation of all members in chassis
- Height of main hoop and front hoop
- Minimum wheelbase
- Size of side impact members
- Angles of the main hoop and front hoop
- Distance between firewall and engine
- Specifications of 95th percentile male and 50 mm rule

1.1) We started our sketch-making wheelbase and respective planes for the main hoop, front hoop, rear hoop, front bulkhead, and impact attenuator. Initially, we took a wheelbase of 1800 mm, but later, altering the positions of the hoops and changing the leg angle to 95th percentile male, we finalized at 1589.14 mm. Then we draw all hoops in their respective planes according to the specifications of the height of hoops given in the rulebook.

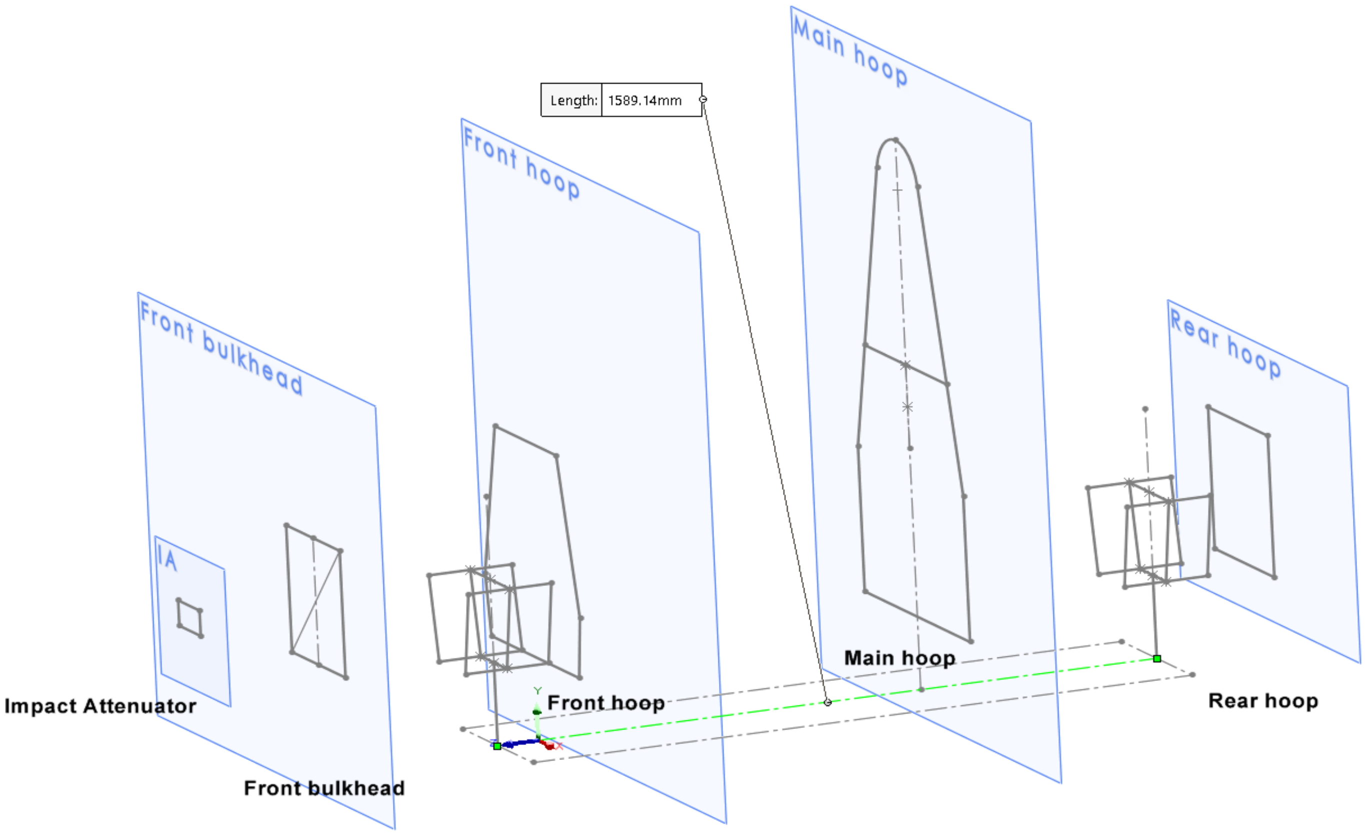

Fig 2.2: Starting sketch with all hoops

Also, we did a market survey for Impact Attenuator (IA), which could satisfy the specifications specified in the rulebook. We finalized this IA [2], which is stated as FSAE standard and have energy-absorbing foam. According to this, we made a construction sketch of IA according to its dimensions for ref

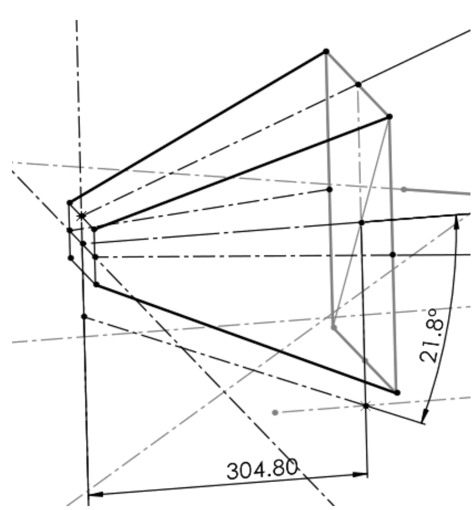

Fig 2.3: Impact Attenuator 3D sketch

1.2). Then we inserted the 95th percentile male and, according to it, made our cockpit. During this step, we took care of all design specifications of the 95th percentile male mentioned in the rulebook. Also, we set the height of our main hoop according to the driver's height to follow the 50 mm rule.

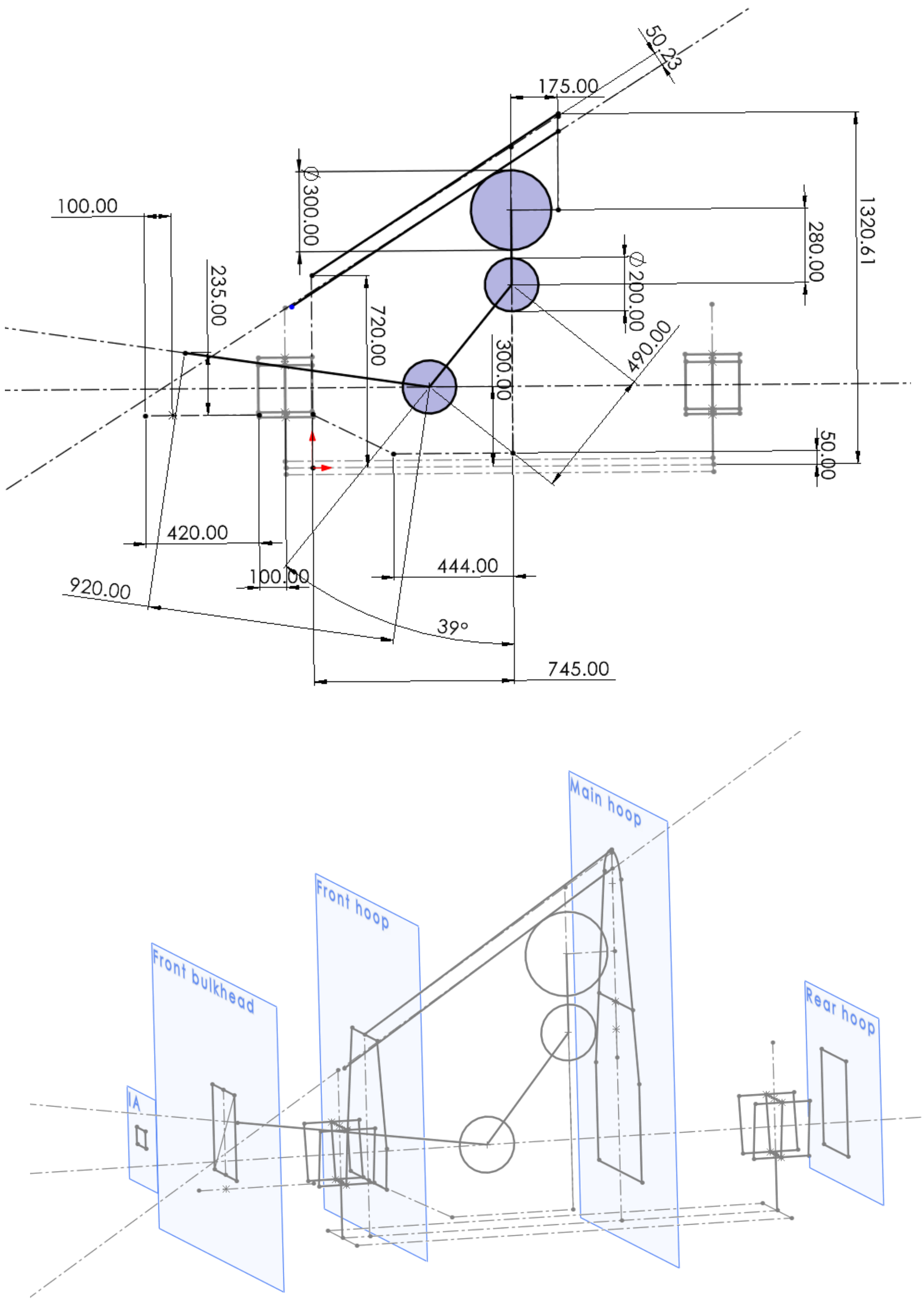

Fig 2.4: **<u>95th percentile male and its associated dimensions with cockpit</u>**

1.3) We draw the trusses in between hoops. We first draw the side impact members between the main and front hoops to complete the cockpit. After this, we complete the trusses between other hoops. The trusses between the front hoop and front bulkhead have been altered after analysis of each cad model in ansys.

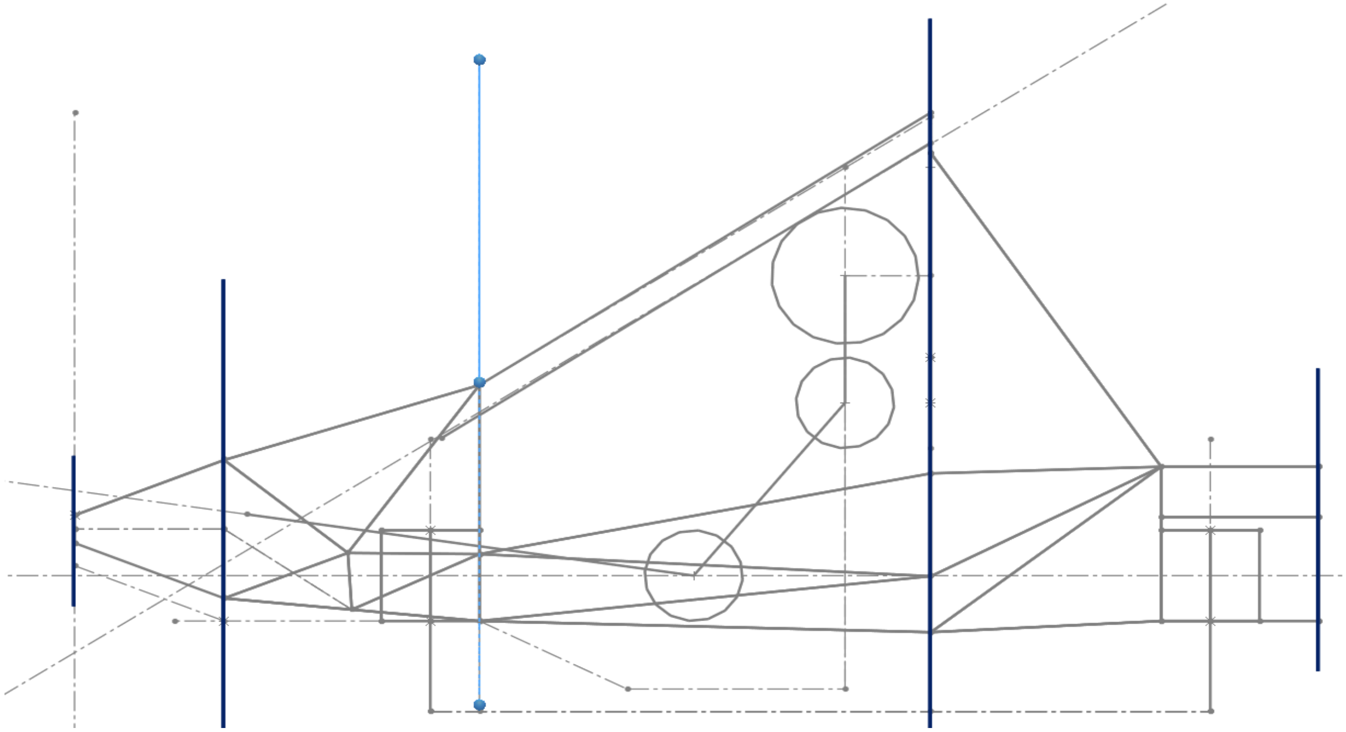

Fig 2.5: **<u>Side view of chassis sketch with a sketch of 95th percentile man</u>**

1.4) As per the rulebook, we use custom weldment profiles to have proper AISI 1018 pipes of the correct diameter and thickness. Then we do corner treatment using the trim/extend feature to refine all chassis corners.

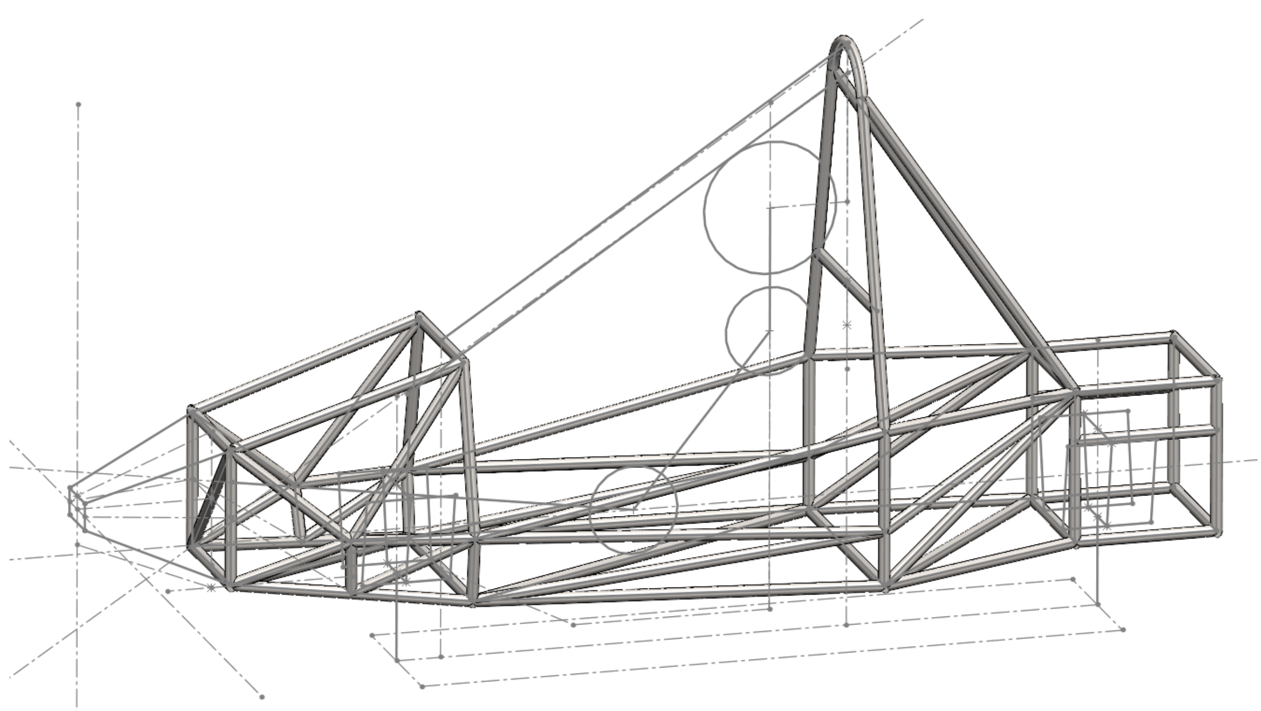

Fig 2.5: **<u>Chassis after insertion of custom weldments and cornertreatments</u>**

2.) Optimization of chassis design

We try to optimize the design of the chassis based on the following parameters: -

- Weight of chassis
- Height of the main hoop
- Aerodynamics

As explained in a later section, we test our CAD models in ANSYS to optimize these parameters. According to the results of those analyses, we made changes in our chassis and finalized these parameters.

2.1) Weight of chassis - We **optimize the weight of chassis by 15%** by reducing additional members of chassis which are not required as stress concentration on them is low.

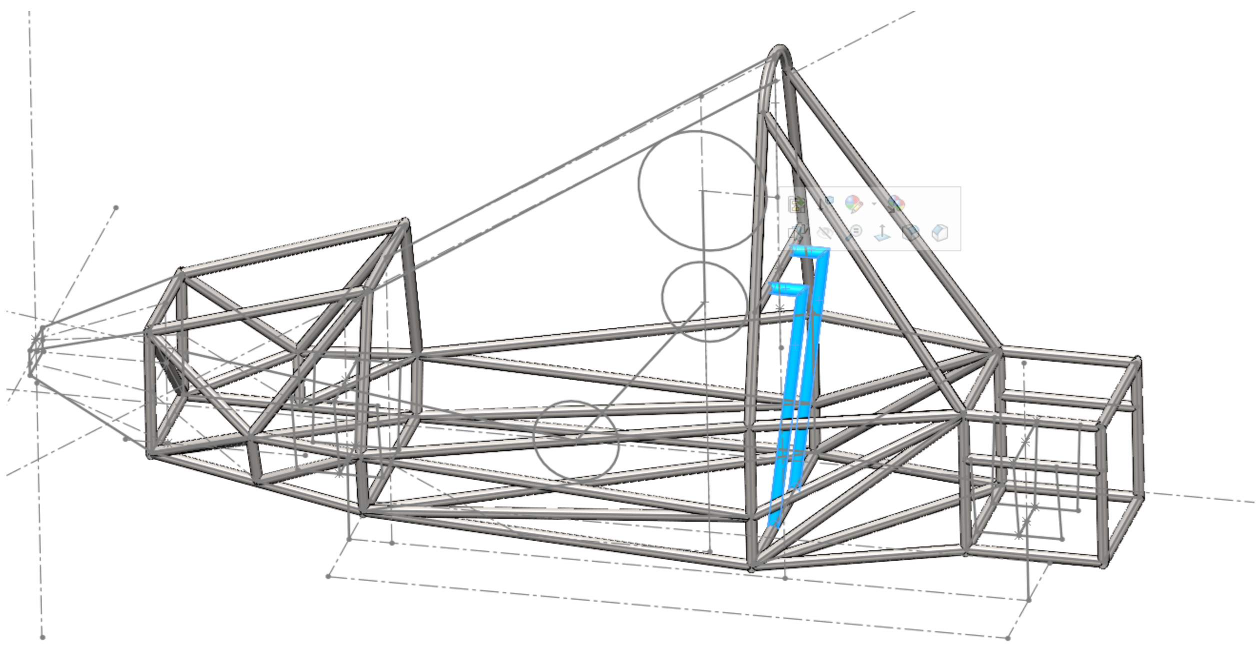

Fig 2.6: Chassis with extra shoulder harnesses in blue colour

For example, we initially designed these highlighted rods to strengthen the shoulder harness. Still, we did a stress-strain analysis on the chassis without these highlighted members and got good results. Hence we removed it and optimized the weight of our chassis.

2.2) Height of the main hoop - We need to optimize the height of the main hoop because as its height increases, the torsional stress also increases [3]. But also, the main hoop height should be enough to accommodate the engine inlet valve. Keeping these two factors in mind, we finalize the height of the main hoop.

MATERIAL SELECTION

The material selected for the Chassis is AISI 1018. AISI 1018 is a low-carbon steel commonly used in manufacturing due to its excellent weldability, machinability, and formability. Here are some properties [4]:

1. **Chemical Composition**: AISI 1018 contains 0.18% carbon, 0.6-0.9% manganese, 0.04% phosphorus, and 0.05% sulfur.

2. **Mechanical Properties**: The tensile strength of AISI 1018 ranges from 410-790 MPa, while its yield strength is between 250-550 MPa. It has a hardness of 121-229 Brinell.

3. **Physical Properties**: AISI 1018 has a density of 7.87 g/cm^3, a melting point of 1425-1540°C, and a thermal conductivity of 51.9 W/mK.

4. **Machinability**: AISI 1018 has excellent machinability due to its low carbon content, which makes it easy to cut, drill, and weld.

5. **Weldability**: AISI 1018 can be welded using various welding methods, including MIG, TIG, and stick welding.

6. **Formability**: AISI 1018 can be easily formed into different shapes using cold-working processes such as bending, punching, and rolling.

7. **Corrosion Resistance**: AISI 1018 has poor corrosion resistance and is susceptible to rusting if not adequately protected.

Properties	**Metric**
Tensile Strength, Ultimate	440 MPa
Tensile Strength, Yield	370 MPa
Modulus of Elasticity	205 GPa
Bulk Modulus	140 GPa
Shear Modulus	80 GPa
Poisson’s Ratio	0.29

Table 1.0 - Properties of AISI 1018 [4]

ANALYSIS

After selecting the material, the next task is to analyze the design. Two types of analysis are done on a vehicle. They are static and dynamic analyses. **Static analysis** is done for the car in the stationary position, and dynamic analysis is done for the vehicle in motion. It will be discussed only on static analysis, as formula student race cars won't be going at such drastic high speeds.

A. Types of analysis

- Front-impact analysis
- Side impact analysis
- Rear impact analysis

B. Steps for performing analysis

Several softwares are available to analyze the object, such as Hyper Mesh, ANSYS, SOLIDWORKS, etc. And here, analysis has been done on the ANSYS. A few basic and simple steps will be followed to perform all the analyses mentioned above on the designed chassis. They are given as follows:

- Import the design in the ANSYS
- Apply the necessary properties of the material
- Mesh the structure completely
- Apply load at the required portion of the chassis of the car and boundary conditions.
- Note the obtained **yield strength value** and calculate its **Factor of safety (FOS)** and other necessary results.

Front Impact analysis

Front Impact Analysis determines the stress distribution throughout the body and the deformation generated in the FSAE chassis if the car hits a rigid body from the front. In contrast, it is more likely to happen in any worst case. To Simulate the frontal impact crash, the vehicle was assumed to impact with a momentum transfer as calculated below.

Calculations:

Parameters	Values
Kerb weight of the car + driver	270 kg

Maximum speed	27.78m/s
Time of retardation	0.4-sec

Table 2.1 - Front Impact Analysis values for calculation

a=(v-u)/t

Hence, a = 69.44 m/s2

F=ma

Therefore, F = 18750 N (into the chassis)

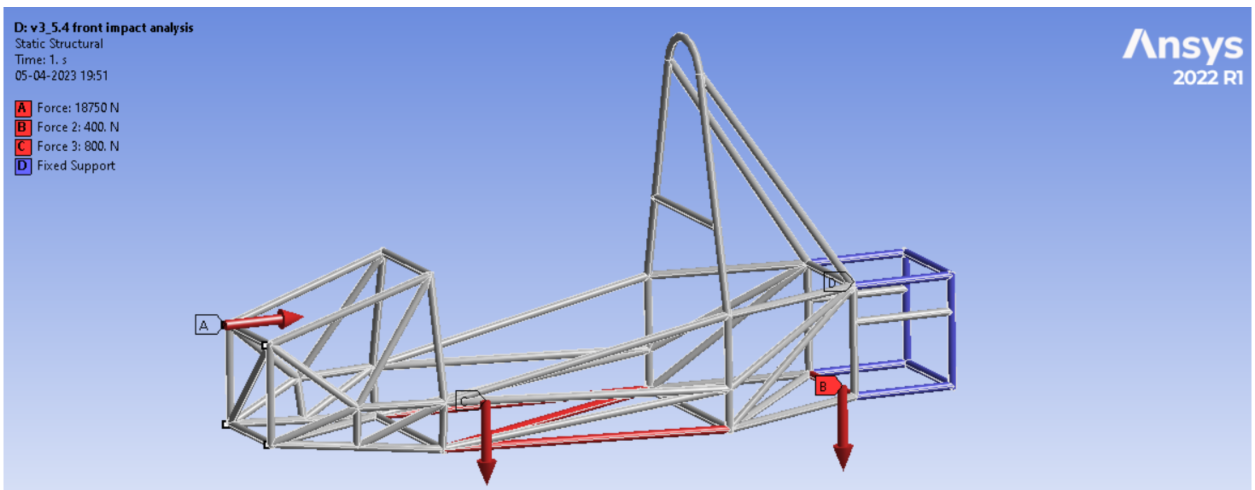

Fig 3.1: Forces setup for Front-Impact Analysis on Chassis

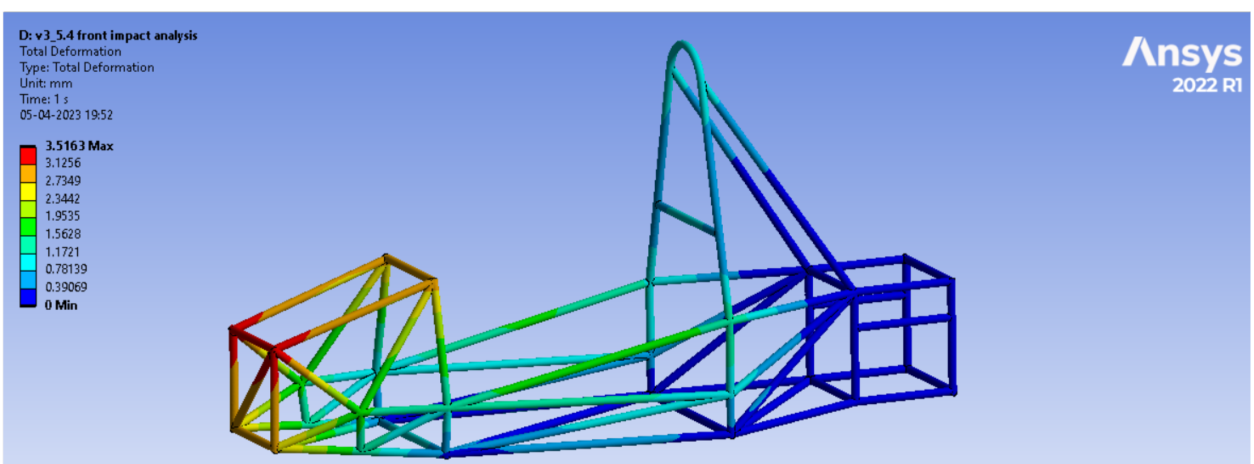

Fig 3.2: Total Deformation of Chassis in Front-Impact Analysis

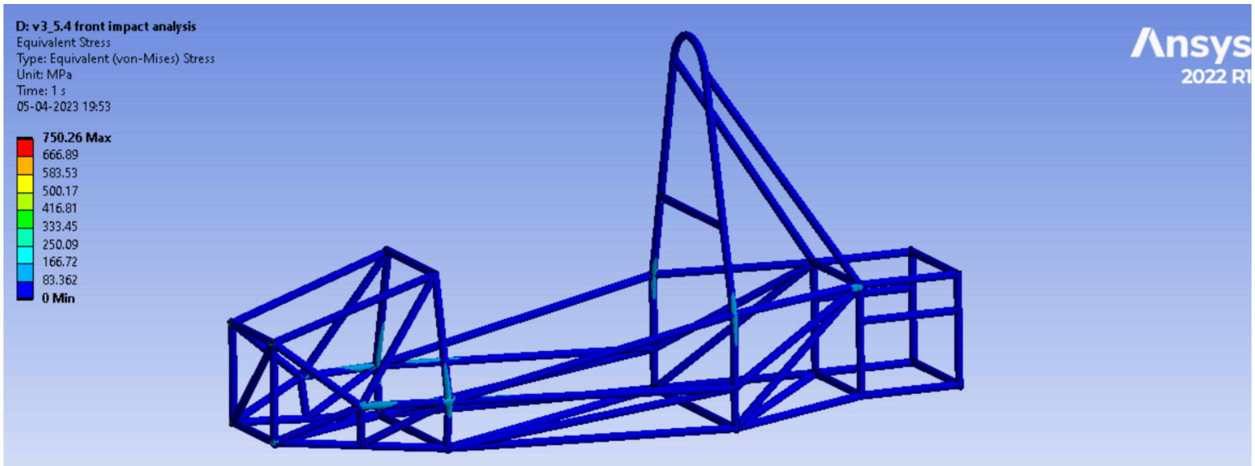

Fig 3.3: Equivalent (Von-Misses) Stress on Chassis in Front Impact Analysis

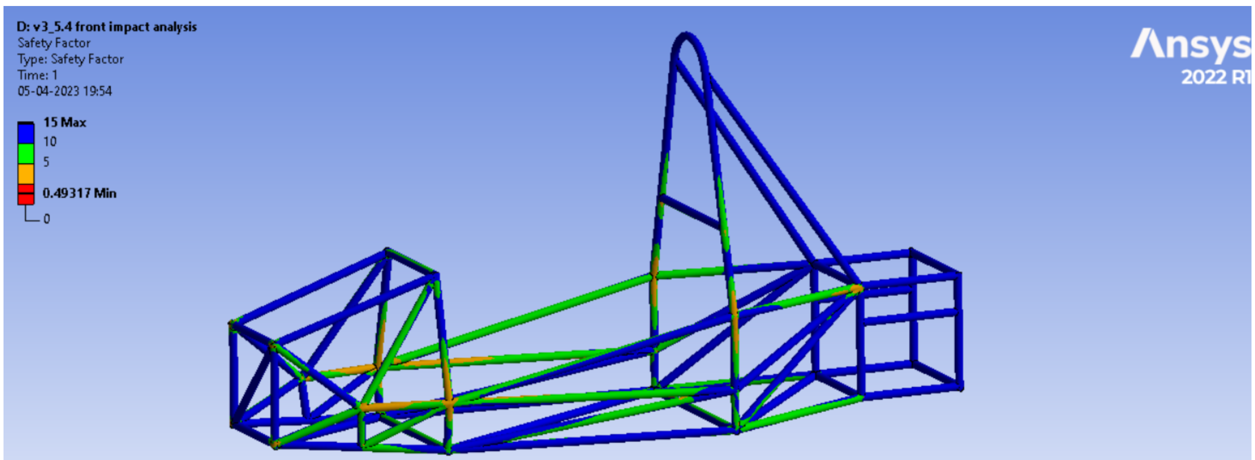

Fig 3.4: Factor of Safety of Chassis in Front Impact Analysis

Parameters	Values
Total Deformation	3.5163 mm (Max)
Equivalent (Von-Misses) Stress	750.26 MPa (Max)
Factor of Safety (FOS)	0.49317 (Min)

Table 2.2 - Front Impact Analysis Results

Side Impact Analysis

The side wall of the chassis is resting against the wall, and another vehicle is supposed to collide against it from the opposite side over the same portion. For constraining the chassis, the suspensions are kept fixed.

Calculations:

Parameters	Values
Mass of the car	270 kg
Maximum speed	27.78m/s
Time of retardation	0.4-sec

Table 2.3 - Side Impact Analysis values for calculation

a=(v-u)/t

Hence, a = 39.68 m/s2

F=ma

Therefore, F = 10714.28 N (into the chassis at an angle of 45 degrees)

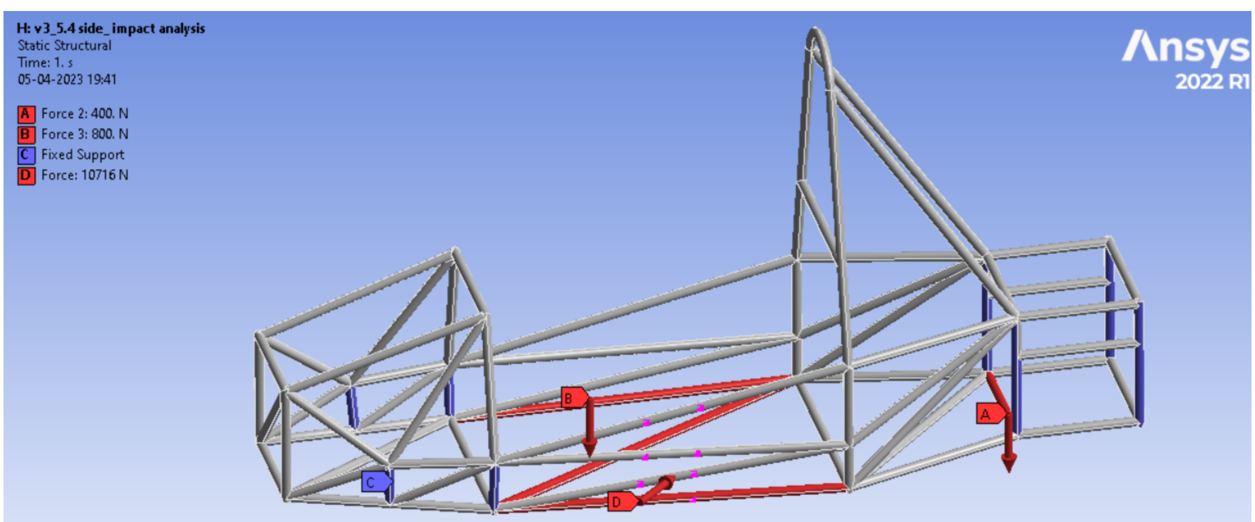

Fig 4.1: Forces setup for Side-Impact Analysis on Chassis

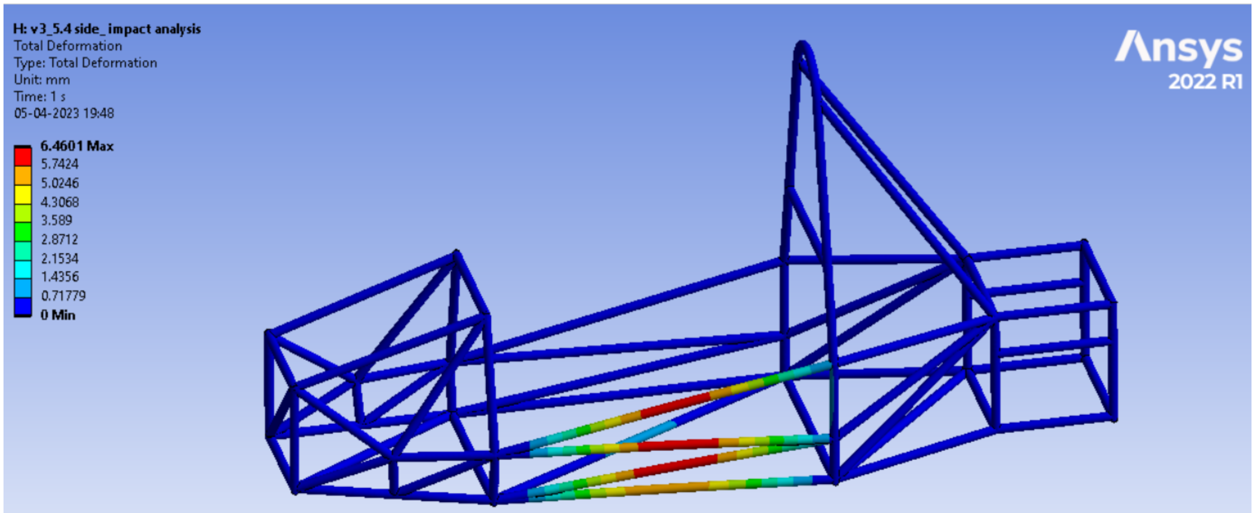

Fig 4.2: <u>Total Deformation of Chassis in Side-Impact Analysis</u>

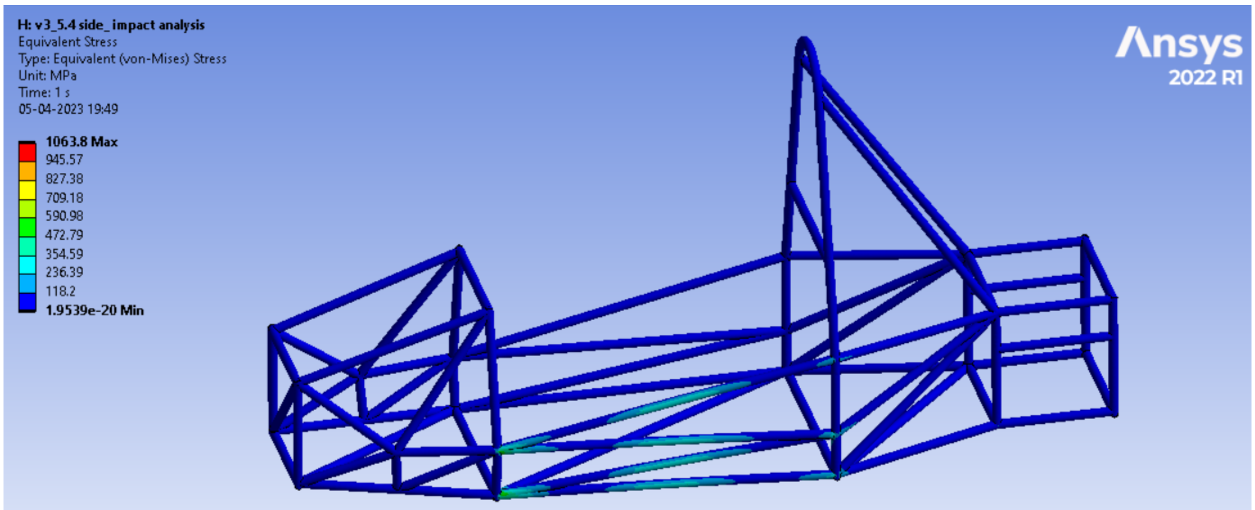

Fig 4.3: <u>Equivalent (Von-Misses) Stress on Chassis in Side Impact Analysis</u>

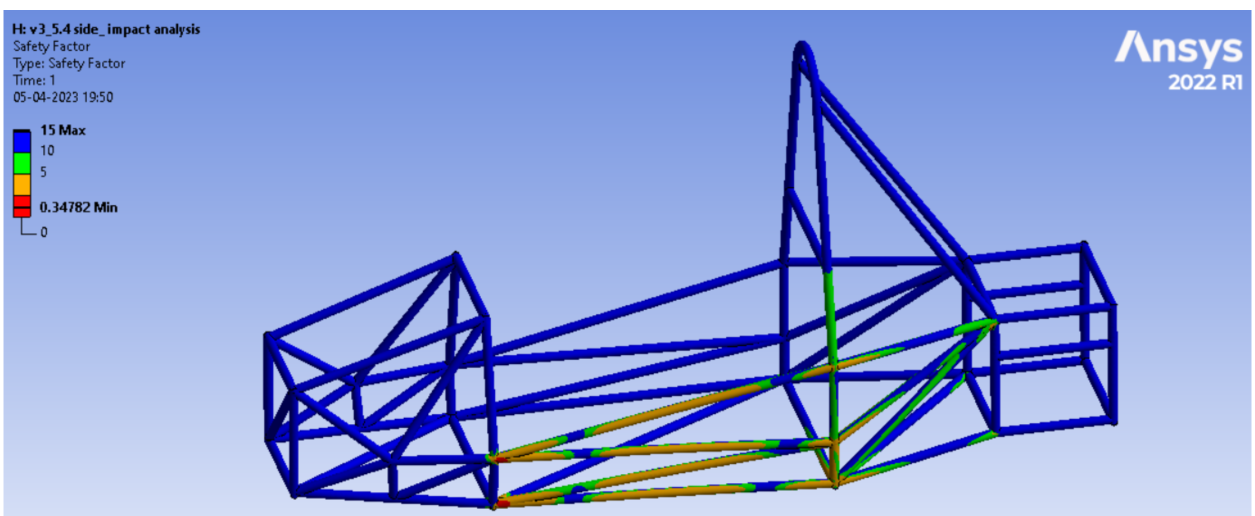

Fig 4.4: <u>Factor of Safety of Chassis in Side Impact Analysis</u>

Parameters	Values
Total Deformation	6.46 mm (Max)
Equivalent (Von-Misses) Stress	1063.8 MPa (Max)
Factor of Safety (FOS)	0.34782 (Min)

Table 2.4 - Side Impact Analysis Results

Rear Impact analysis

The Front Impact Analysis determines the bending stress and deformation generated in the FSAE chassis if the car hits a solid body from the rear end. In contrast, it is more likely to happen in any worst case. To Simulate the rear impact crash, the vehicle was assumed to impact upon a stationary object.

Calculations:

Parameters	Values
Mass of the car	270 kg
Maximum speed	27.78m/s
Time of retardation	0.4-sec

Table 2.5 - Rear Impact Analysis values for calculation

a=(v-u)/t

Hence, a = 69.44 m/s2

F=ma

Therefore, F = 18750 N (into the chassis)

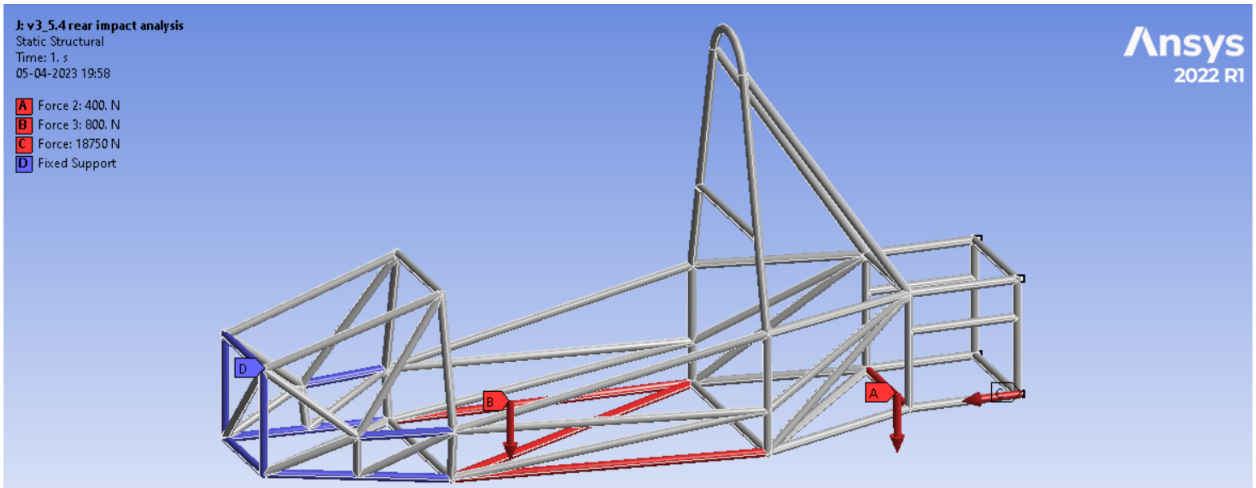

Fig 5.1: Forces setup for Rear-Impact Analysis on Chassis

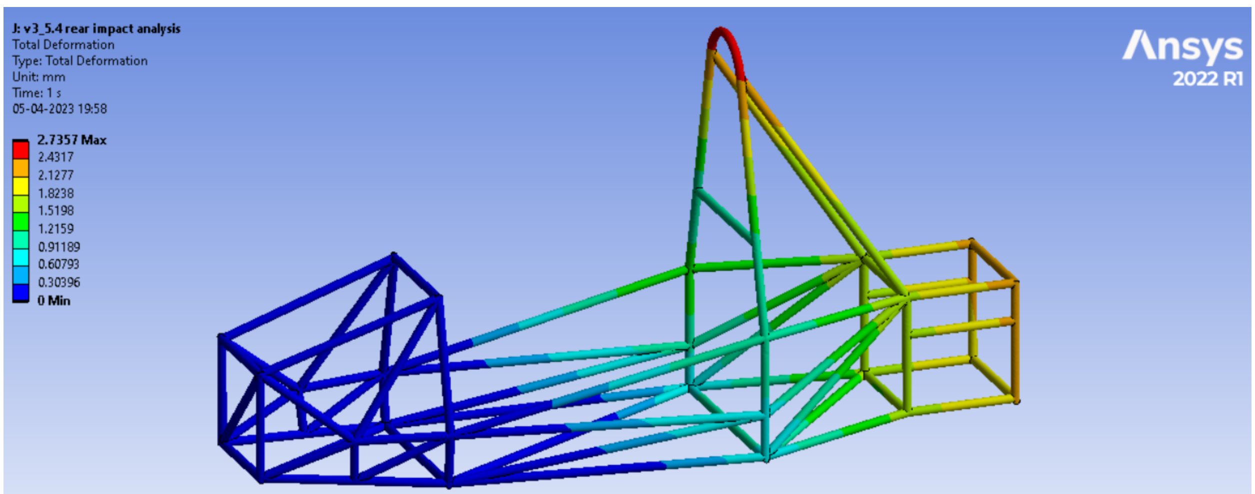

Fig 5.2: Total Deformation of Chassis in Rear-Impact Analysis

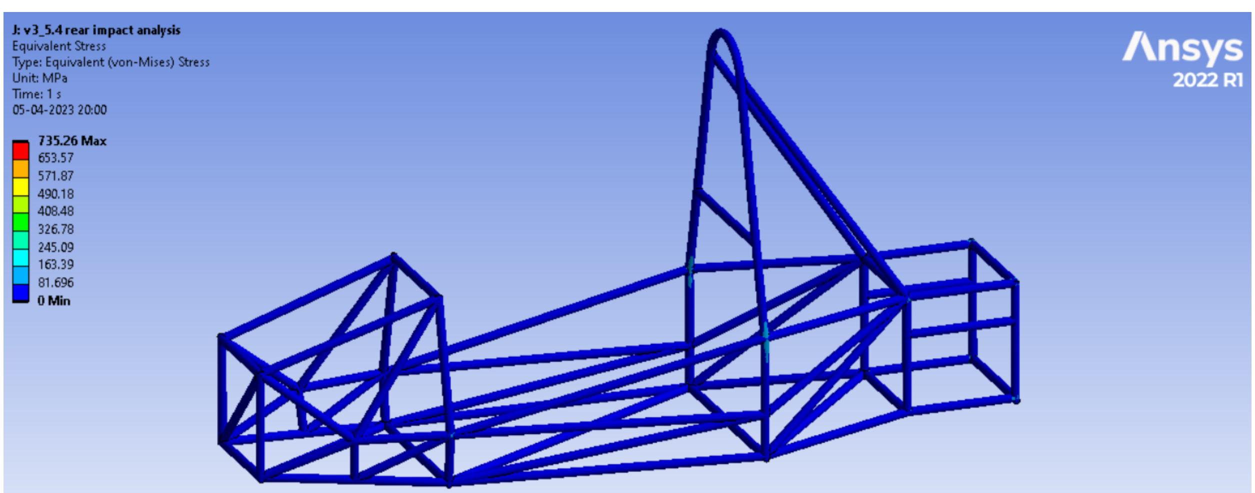

Fig 5.3: Equivalent (Von-Misses) Stress on Chassis in Rear Impact Analysis

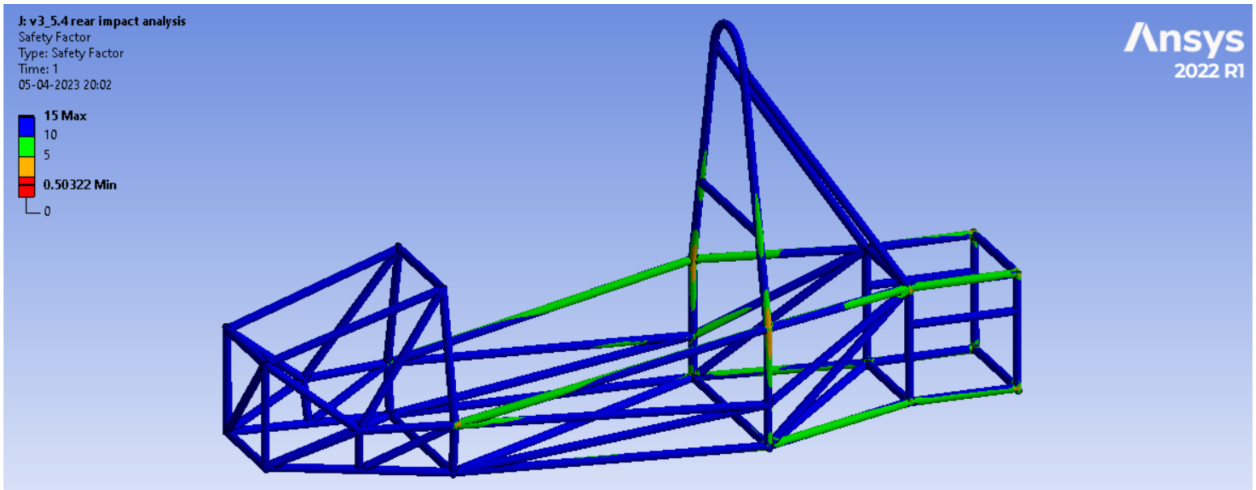

Fig 5.4: Factor of Safety of Chassis in Rear Impact Analysis

Parameters	Values
Total Deformation	2.73 mm (Max)
Equivalent (Von-Mises) Stress	735.26 MPa (Max)
Factor of Safety (FOS)	0.50322 (Min)

Table 2.6 - Rear Impact Analysis Results

REFERENCES

1)https://www.ansys.com/en-in/blog/fundamentals-of-fea-meshing-for-structural-analysis

2)https://pdf.indiamart.com/impdf/16297415162/MY-4624662/fsae-spec-standard-impact-attenuator.pdf

3)https://www.academia.edu/64896073/Modelling_analysis_of_high_effect_of_roll_hoop_main_on_the_strength_of_student_car_formula_chassis

4)https://www.azom.com/article.aspx?ArticleID=6115

5)https://drive.google.com/drive/folders/1oTPhtlNRPRTjwdnBda3W0YMMPyO2GDFK?usp=sharing - Chassis files of Team Trident