



University of Wales Trinity Saint David
School of Engineering

Optimization of Formula Racing Frame Based on FEA

Zhangwei Guo
1805078

BEng Automotive Engineering
05/02/2020
Major Project M6X00417

1 Contents

1	Introduction	1
2	Measurement	2
2.1	Methods	3
2.2	Divisions	4
2.3	Special Tools	6
3	Drawing Work	14
3.1	Original model	14
3.2	Mesh Problem of First Model	16
3.3	Improved model	17
3.4	Simplified Model	22
3.4.1	Front Part	22
3.4.2	Middle Part	22
3.4.3	Rear Part	24
3.4.4	Structure of chassis	25
4	Literature Research	27
4.1	Type of Frame	27
4.2	Frame material	28
4.3	Stiffness analysis	30
4.3.1	Bending stiffness	30
4.3.2	Torsional stiffness	31

4.4	Strength analysis	31
4.5	Cost	31
4.6	Weight	32
4.7	Establishment of FEA Model	32
4.7.1	Unit Selection	32
4.7.2	Meshing.....	33
4.7.3	Boundary Conditions and Load Handling	34
5	Analysis Work	35
5.1	Improved model.....	35
5.1.1	Improvement of rear part	37
5.1.2	Improvement of middle part.....	38
5.2	Bending stiffness	39
5.2.1	Bending stiffness of middle part.....	39
5.2.2	Bending stiffness of rear part.....	41
5.3	Torsional stiffness.....	43
5.3.1	Real test.....	43
5.3.2	FEA.....	46
5.4	Strength analysis	50
5.5	Weight and center of gravity	50
6	Future Direction	52
7	Conclusion	53
8	Reference	54

9	Appendix.....	56
---	---------------	----

2 Introduction

Based on the basic understanding of the vehicle chassis structure, the measurement, model drawing and analysis of the existing chassis frame are carried out. The reference plane method is used to measure the structure of the vehicle chassis, accurately measure the size of the workbench and the chassis structure of the vehicle, after that determine the central axis of the vehicle to fix the vehicle chassis structure to the workbench. Since the top level on the workbench is taken as the reference plane drawn by the vehicle chassis structure model and the entire vehicle chassis structure is divided into sections and then layered, the measurement of the vehicle chassis structure becomes much easier. After obtaining the measurement data, the basic model of the vehicle chassis structure is established in SOLIDWORKS to prepare for the analysis and simulation. The analysis of the vehicle chassis structure is carried out in terms of strength, stiffness, cost and weight within both FEA and the real world through adding and removing parts of the structure trying to make it stronger, stiffer, lighter and cheaper. The stiffness analysis is mainly for Bending stiffness and Torsional stiffness. This article focuses on the effects of different parts have on the strength, stiffness, weight and cost and trying to find the balance for them. Due to the campus temporary close to students on account of the outbreak of the COVID-19 virus, some of the expected tasks have not been completed and the expected goals have not been achieved as well.

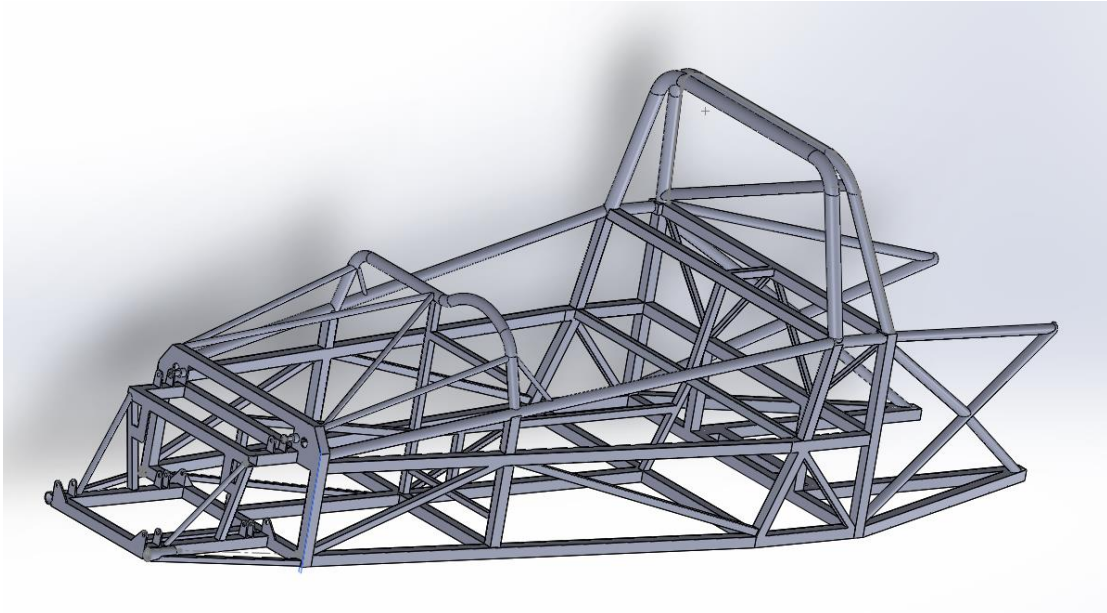


Figure 1-Model of chassis

3 Measurement

Before started the measurement, it takes two and half days to remove all the rivets and skinning to show the structure clearly. Marking every part and taking photographs costs another two and half days. After all of these, methods and divisions were both decided on a group meeting.

All the parameters of front, rear parts and each level of middle part have been measured and recorded.

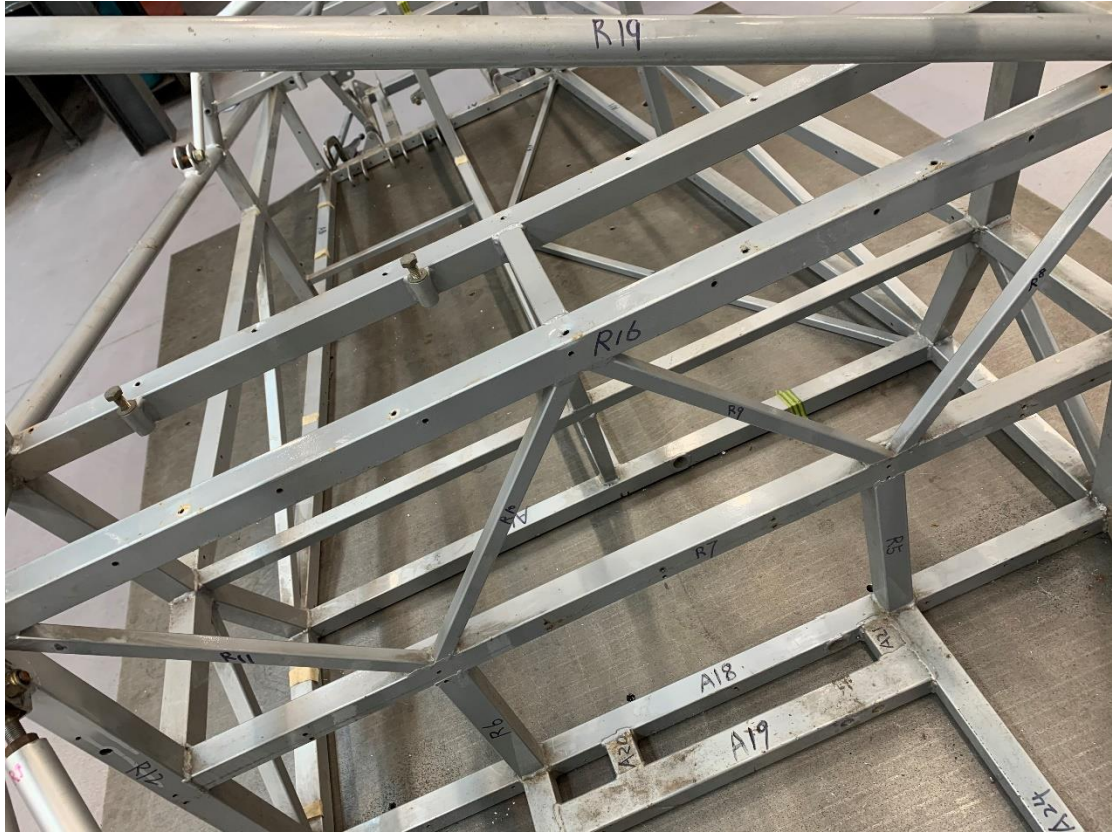


Figure 2-Rear Part

3.1 Methods

The whole chassis has been divided into three main parts which are front part, middle part and rear part. There are several levels within the middle part and all parts in their level have been marked as shown in figure 3, so that every part could be measured and drawn separately.



Figure 3-Levels in middle part

3.2 Divisions

The whole chassis has been divided into three main parts as front, middle and rear part. All the structures on the ground floor have been divided into level A which has 25 components in middle part as the reference for whole chassis.

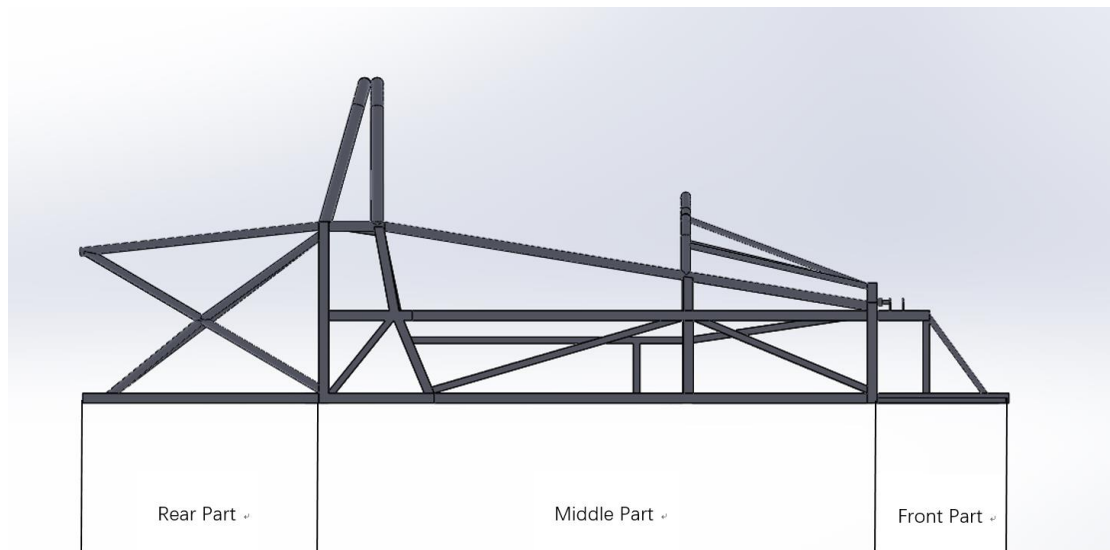


Figure 4-Three main parts

The front part comes with eight main components and several fixed points while the base of the front part has been marked from A1 to A7 in level A.



Figure 5-Front part

There are six levels from A to F within the middle part. Each level comes with 5 to 25 components as all the structures on the ground floor as been divided into level A.

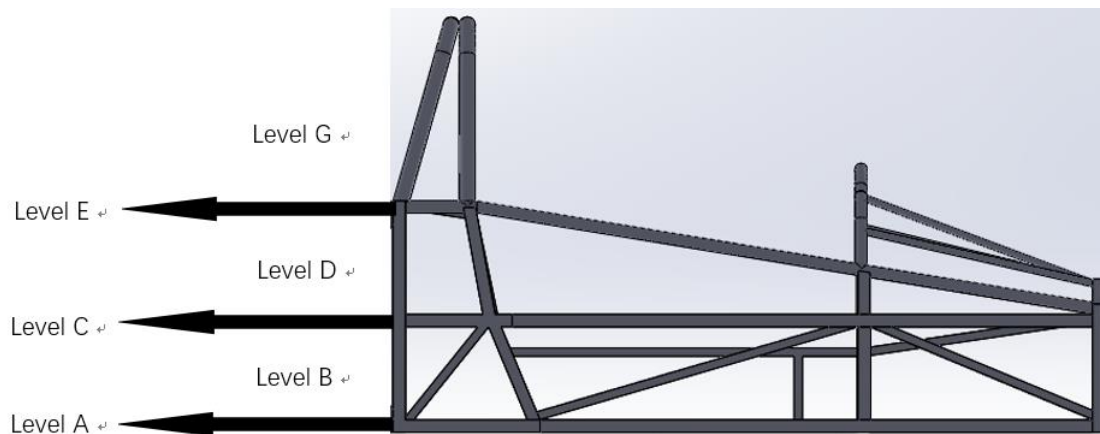


Figure 6-Middle part

Rear part has 22 components with two additional tubes which are marked as 21 and 22 on the top to strengthen the stucture.



Figure 7-Rear part

3.3 Special Tools

A. Vernier caliper

Vernier caliper is a measuring tool for measuring length, inner and outer diameter, and depth. The Vernier caliper is composed of a main ruler and a sliding cursor attached to the main ruler. The main ruler is generally in millimeters, and the cursor has 10, 20 or 50 divisions. According to the division, the Vernier caliper can be divided into tenth degree Vernier caliper, twenty degree Vernier caliper, fifty degree Vernier caliper, etc. The cursor is 9mm for 10 divisions, 19mm for 20 divisions, and 49mm for 50 divisions. There are two pairs of movable claws on the main ruler and the Vernier caliper, which are the inner measuring claw and the outer measuring claw. The inner measuring claw is usually used to measure the inner diameter and the outer measuring claw is usually used to measure the length and outer diameter. Application range

Vernier calipers are a commonly used measuring tool, which can be specifically applied in the following four aspects:

- 1) measure the width of the workpiece
- 2) measure the outer diameter of the workpiece
- 3) measure the inner diameter of the workpiece
- 4) measure the depth of the workpiece

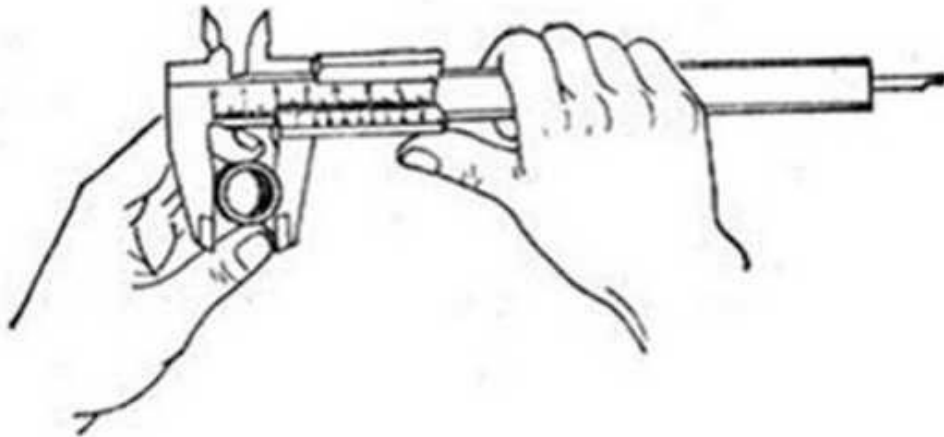


Figure 8-Vernier caliper

B. Square ruler

The square ruler is a measuring tool commonly used in inspection and scribing work. It is used to detect the verticality of the workpiece and the relative position of the workpiece. It is a professional measuring tool, suitable for the verticality inspection of equipment and parts, installation and processing positioning, Marking, etc. It is an important measuring tool in the machinery industry. Angle rulers can be divided into cast iron right angle rulers, magnesium aluminum right angle rulers and granite right angle rulers. The aluminum-magnesium square ruler used in this measurement is mainly used for the positioning and auxiliary measurement of the frame during the measurement.

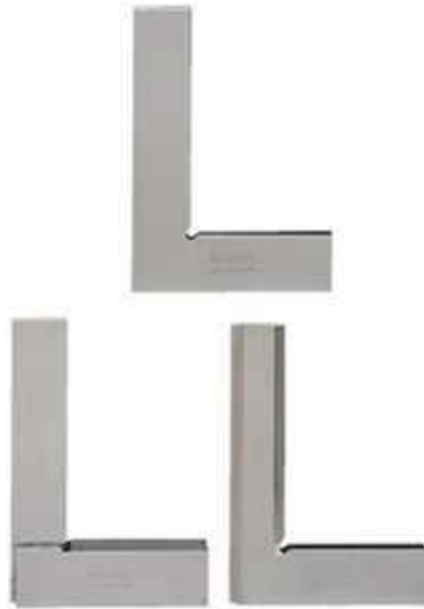


Figure 9-Square ruler

C. Combined angle ruler

The combined angle ruler is a common measuring instrument used for angle, length, height measurement and center positioning in mechanical processing and wood mold processing. The ruler and center gauge can be combined to quickly determine the center of a round workpiece. The outer circle of the protractor has a scale of 0° to 180° from left to right, and the inner circle of the protractor has a scale of 180° to 0° from right to left; there are two hinges in the middle of the length of the ruler, it The ruler can be folded in half, there is a hollow peach shape on the upper end of the ruler, the peach-shaped peach tip is in the middle of the ruler, and the lower half of the ruler has a hollow of 2 to 3 mm wide; It is connected by a movable axis. The utility model can be used as a protractor or a ruler. When used as a protractor, it can measure angles greater than 180° . When not in use, just fold the ruler, rotate the ruler so that the ruler and the bottom of the protractor overlap, so that the two measuring tools become one. Used to measure the inclination of angled rods on the frame.

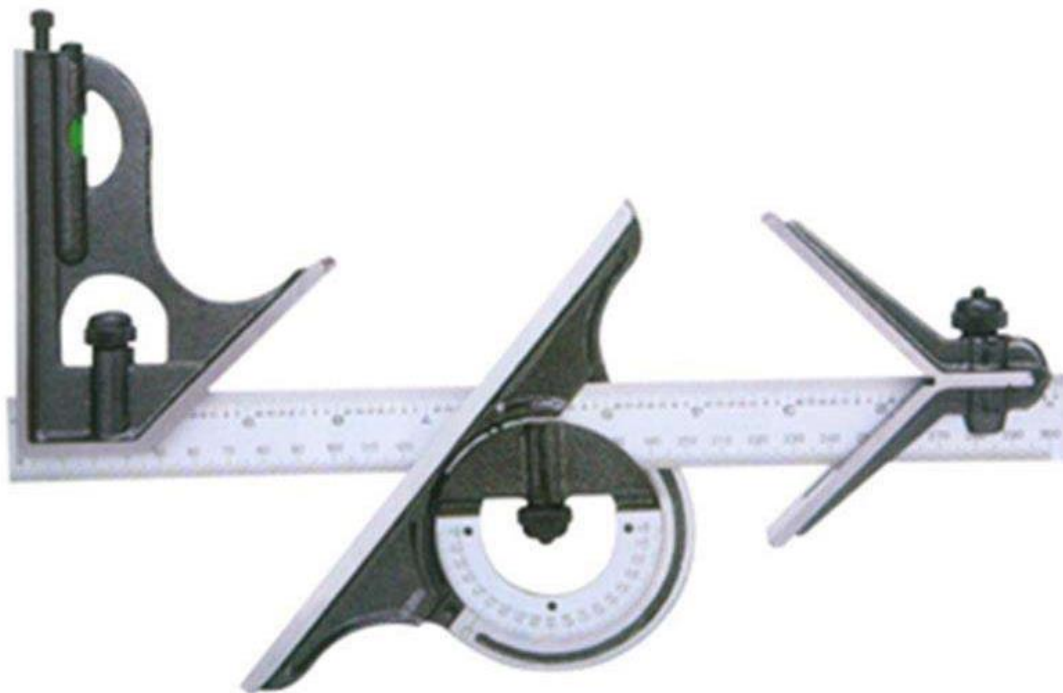


Figure 10-Combined angle ruler

D. Line sag

Line sag is a kind of vertical measuring tool. It is made of iron, with a small pile of money hanging in the middle. Hold the top of your hand and use your eyes to aim at the vertical deviation between the line and the object. It is mostly used to judge whether the object is perpendicular to the ground. This project is used to determine the specific position of the frame spatial structure in the two-dimensional reference plane, so as to further determine the three-dimensional spatial position of each point.



Figure11-Line sag

E. Height ruler

Height ruler is short for Height Vernier Caliper. As the name implies, its main purpose is to measure the height of the workpiece, and it is also often used to measure shape and position tolerance dimensions, and sometimes it is also used to score lines.

According to the different reading forms, height VERNIER calipers can be divided into two categories: ordinary VERNIER and electronic digital display. In the measurement process, both types of height gauges are used.

According to different conditions and uses, it can be divided into single column type and double column type. Single-column height VERNIER calipers are often used in applications where accuracy is not required; dual-column types are mainly used in more precise or larger measurement ranges. The common VERNIER ruler and electronic digital display ruler used in this measurement are both single-column height VERNIER calipers.

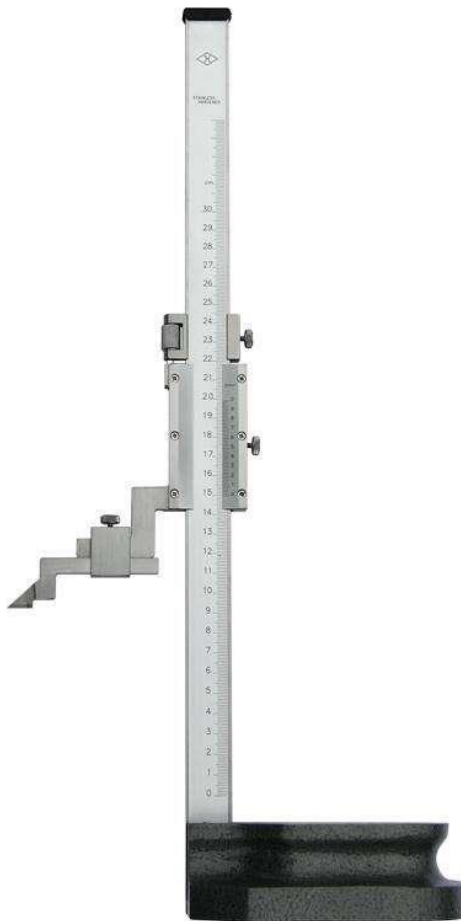


Figure12-Height ruler

F. Dial indicator

Dial indicator is a length measuring instrument that converts the general linear displacement (linear motion) into the rotary motion of the pointer through a gear or lever, and then reads on the dial. The dial indicator is an indicator gauge that is widely used in precision measurement.

The indication range of the dial indicator is generally 0 to 10 mm, and the larger one can reach 100 mm.

The dial indicator has a simple structure, the transmission mechanism is a gear train, the outer size is small, the weight is light, the transmission mechanism is small inertia, the transmission is relatively large, the circumferential scale can be used, and there is a large measurement range, not only for comparative measurement, Absolute measurements can also be made. The working principle of the dial indicator is to move the measuring rod slightly linearly caused by the measured size, and enlarge it through the gear transmission to turn the finger on the dial to read the size of the measured size. The structure of the dial indicator is mainly composed of three parts: the body part, the transmission system, and the reading device. The dial indicator is used to measure the vertical displacement of the measuring point on the marker rod during load loading, and the torsion angle of the frame in this cross section can be calculated in combination with the distance from the measuring point to the central axis.



Figure13-Dial indicator

G. Hand drill

A hand drill is a drilling tool powered by an AC power source or a DC battery. It is a type of hand-held power tool. The main components of the hand electric drill: drill chuck, output shaft, gear, rotor, stator, casing, switch. Some models are equipped with rechargeable batteries, which can work normally without an external power supply within a certain period of time.

During the preparation process, a hand drill was used to remove the rivets securing the peripheral baffle. In the measurement process, it is used not only to drill rods with spatial nodes to obtain the spatial position of the nodes, but also to drill rods without pores, combined with the outer diameter of the rods measured by VERNIER calipers. Calculate the diameter and depth.



Figure 14-Hand drill

H. Portable axle weight electronic scale

The portable axle weight electronic scale is used to monitor the magnitude of the applied load in real time during the experiment and control the consistency of the applied load in the same set of experiments.



Figure 15-Portable axle weight electronic scale

4 Drawing Work

4.1 Original model

The newest software SOLIDWORKS 2019 has been installed on laptop to make sure the vision is the same. Stretch of level A was done firstly as the reference for the structure.

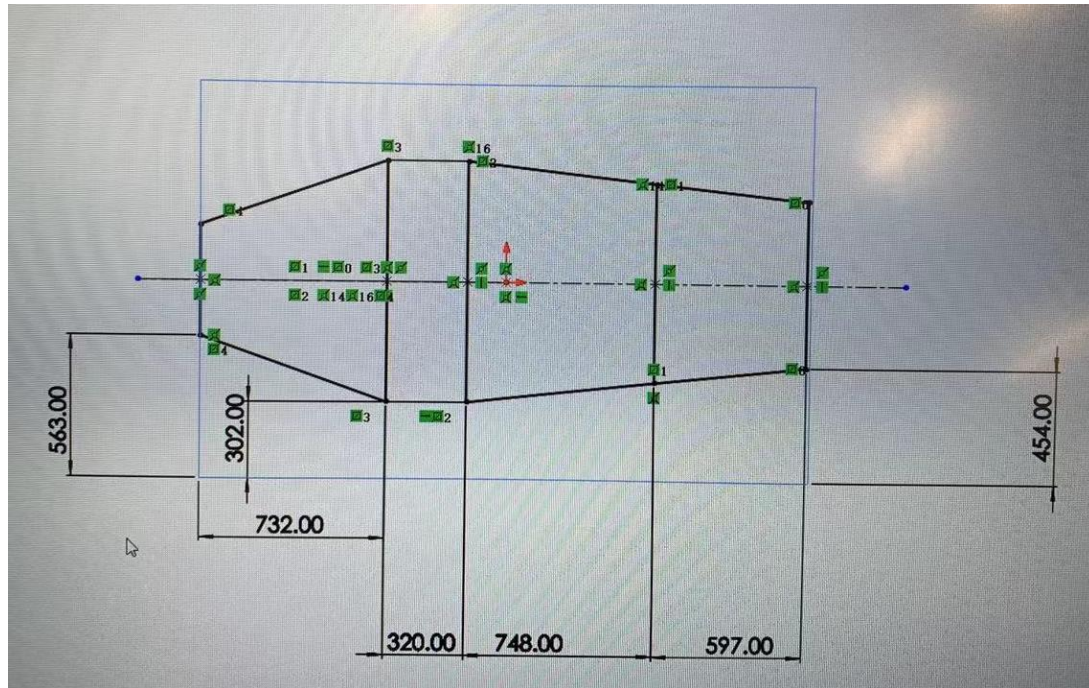


Figure 16-Stretch of Level A

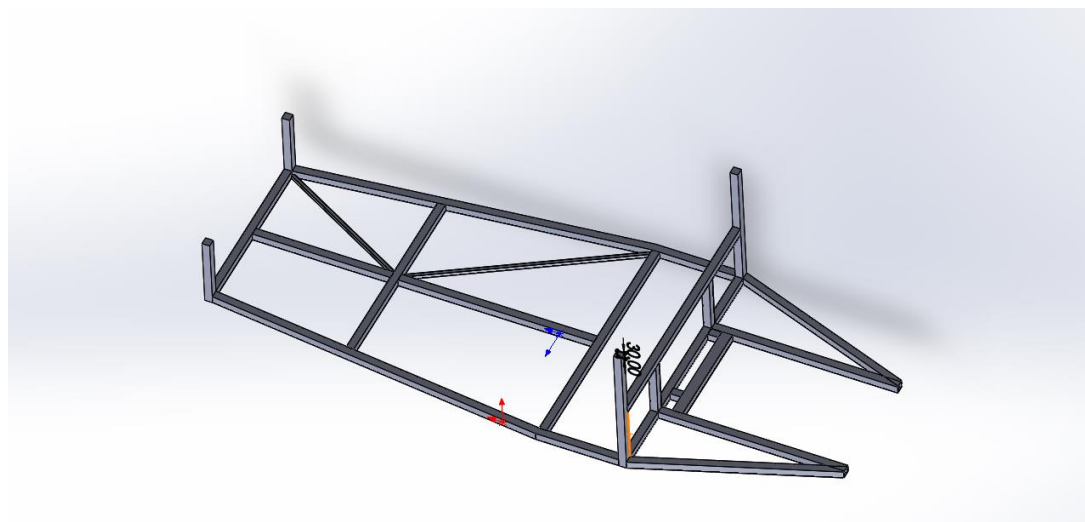


Figure 17-Part Finished Base

Level B, C, D and more structures of rear part have been drawn within next two following weeks.

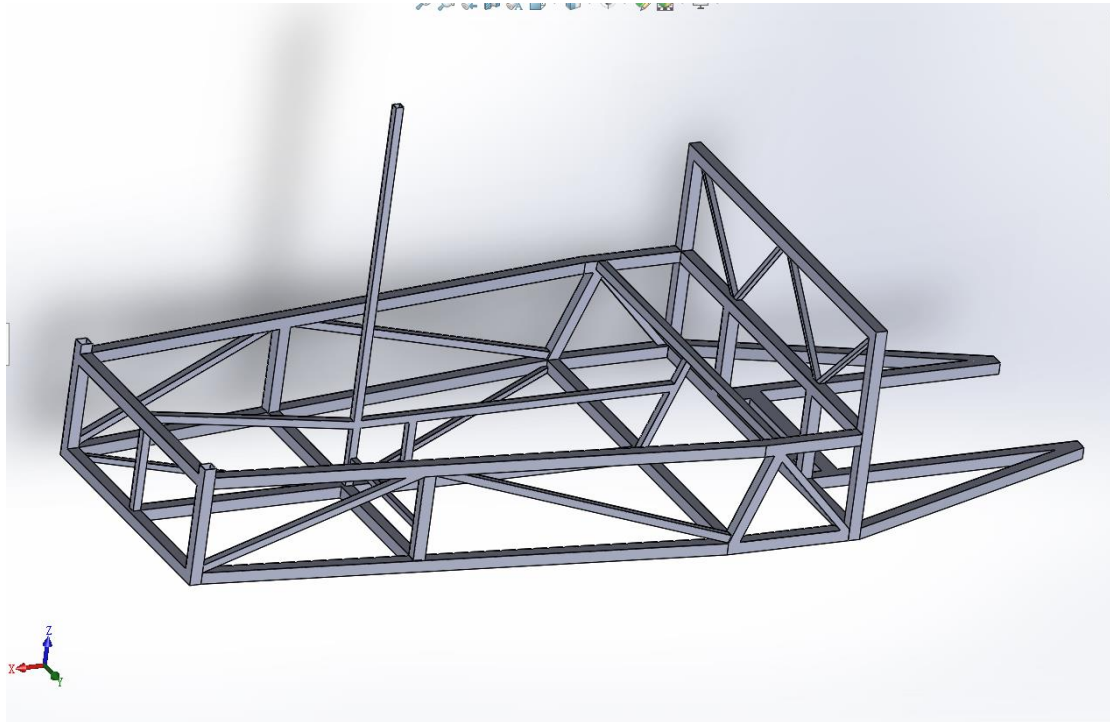


Figure 18-Rear Part

The first model of whole structure has been finished two weeks before middle viva as shown below. After getting the whole model, appointments with Andrew Harrison for FEA within SOLIDWORKS have been made.

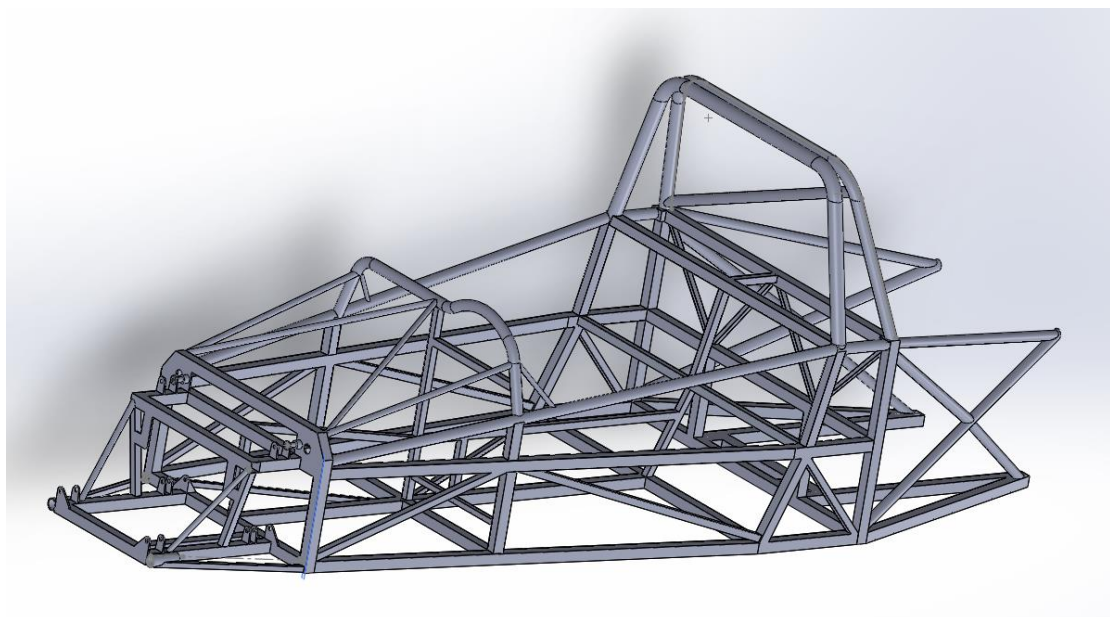


Figure 19-Rear Part

4.2 Mesh Problem of First Model

For the first model been built, when it goes to mesh feature always failed after modifying material, fixing model and applying loads. Trying to find the problem, feature of mesh diagnosis has been applied. On the list of parts which cannot be meshed, a common feature was that parts several parts with one joint appear at the same time has been found. After that, the author used feature of section to check those joints as follows which were failed to be meshed.

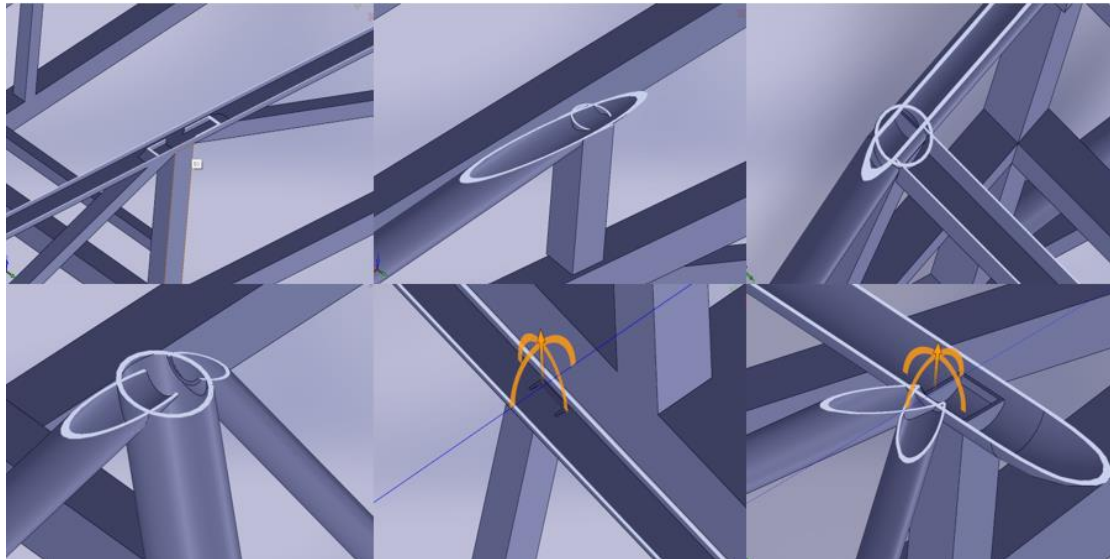


Figure 20-Joints with problem

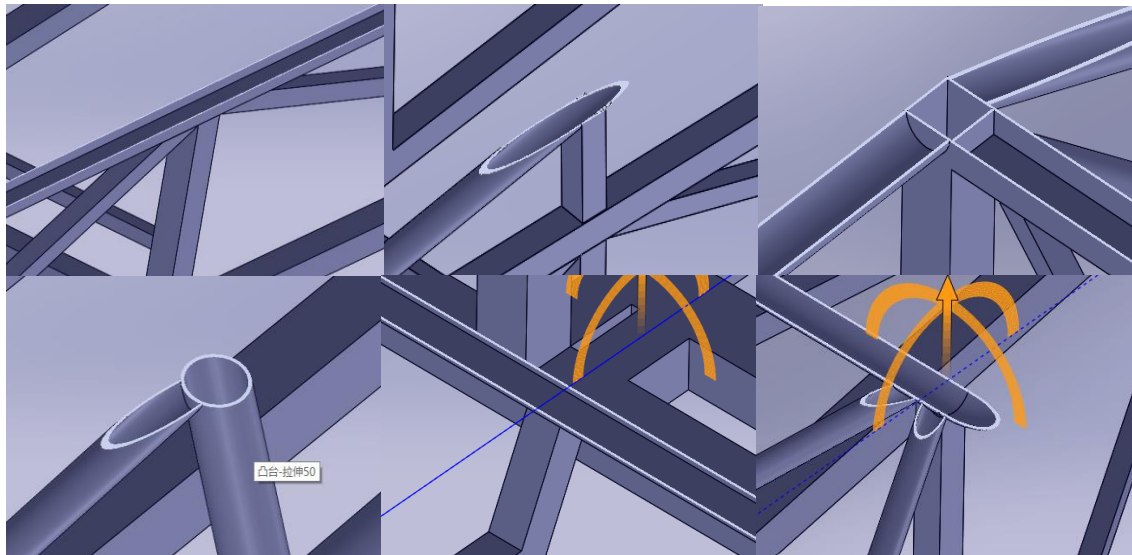
Go for reasons of the conformation of these complicated structures, feature of scan has been used for several parts which needs to be modified within three dimensional modeling during the process of building the first model. The feature of scan just needs a stretch and reference path but reference path needs to be modified until the central line of another part. That is why the end of a scan will get inside of another part.

Appointments with Andrew Harrison and Kalvin Lake have been made for the alternative feature of scan. A special feature of extrude with different stop conditions has been confirmed as a better solution.

4.3 Improved model

Using extrude feature to define parts needs a reference plane for stretch. First of all, a 3D stretch of a line coming through the start and end of the extrude should be established as the first reference of the plane. After that, a reference point on the line which has a distance to the start or end of the line could be defined as the second reference. Finally, the reference plane could be found. From this method, joints which have problems all coming as shown in figure 21.

Figure 21-Improved joints



In addition to the above complex joints, there is another joint whose processing needs additional explanation.

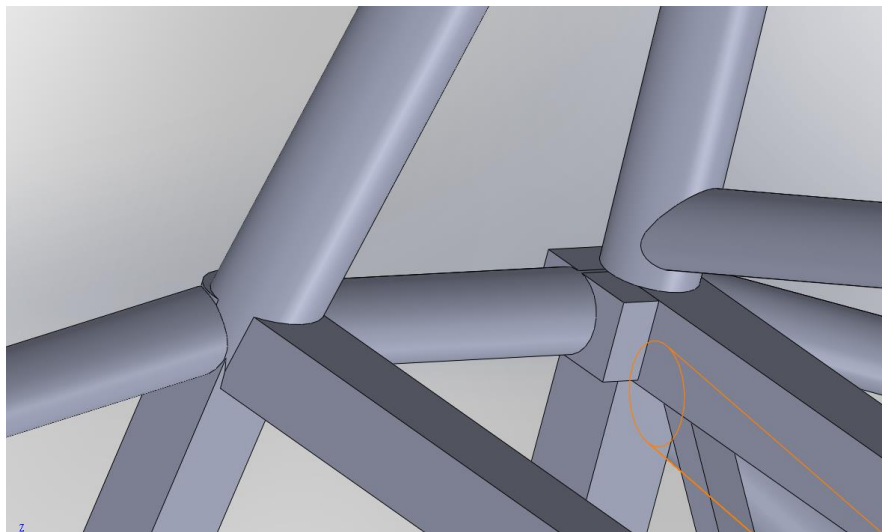


Figure 22-Complex joint

As figure 22 shows, part E2 has a combine joint with E1 and D2 and the contact face between E1 and E2 is a circle surface while D2 with a vertical surface. But for the stop condition of

extrude feature, only one face could be chosen as the extrude until. When the plane of D2 is selected as the stop condition, the extrude cannot be performed to cover the outside of E1, and when the outside of E1 is selected as the stop condition, the extrude will proceed to the inside of the D2 plane. Trying to avoid this conflict, the E2 part was moved down a bit and rotated accordingly to achieve a perfect match with the D2 part.

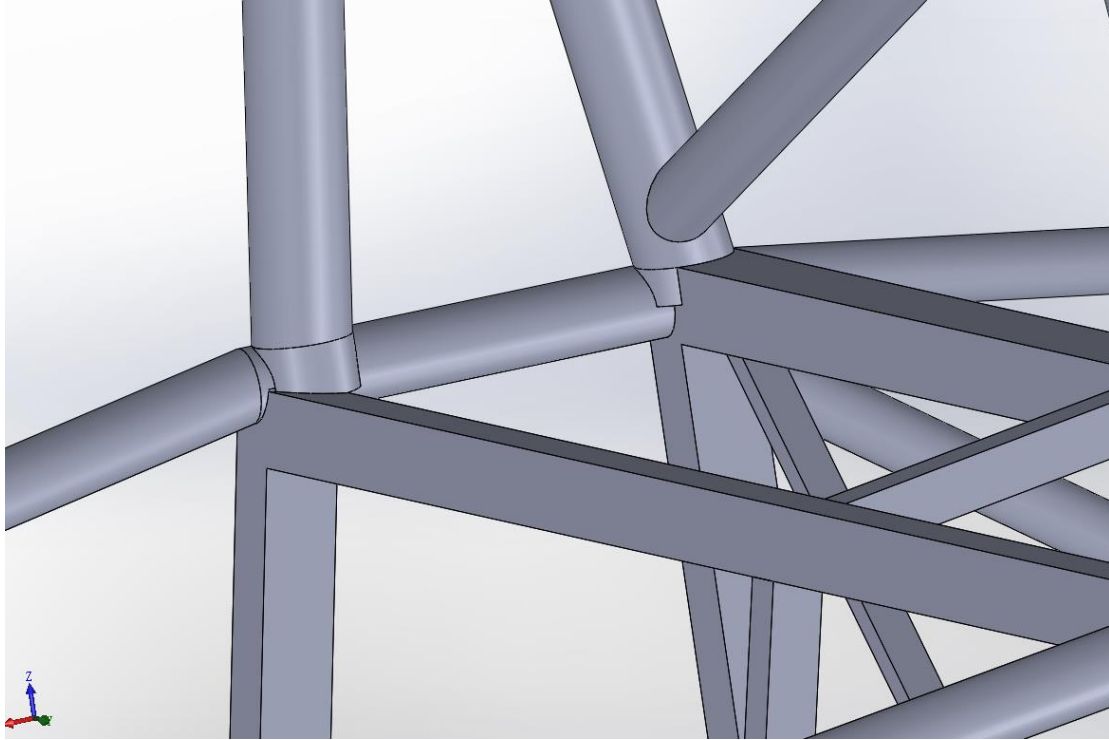


Figure 23-Improved joint

The last improvement of this model is the solid base for the top level which are G1 and R22. The base of the original model is a simple extrude which made the joints more complicated while the improved model has stretched the outline of the base and extrude without the coincident part as figure 23 shows.

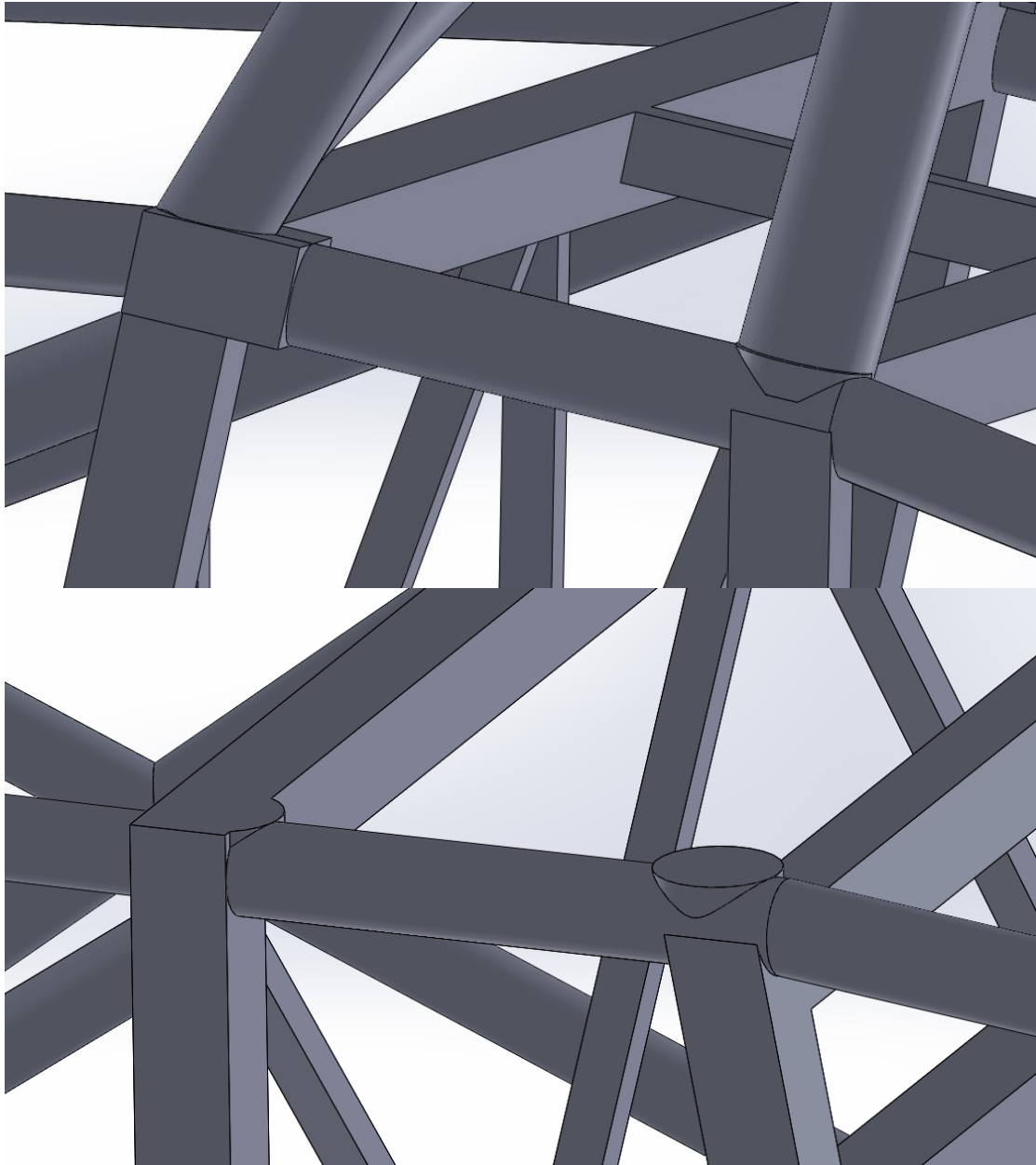


Figure 24-Improved base

Tried to go for FEA process, after applying material, fixing model, applying load, creating mesh and running, the results come a huge different with the data tested in workshop on real chassis. An appointment with Kelvin Lake has been made trying to find the reasons of it. The suggestion is that the loading part been used in workshop should be added into FEA on account of the deformation of loading bar. After simple loading part being created, three assemblies of test chassis all been finished as follows.

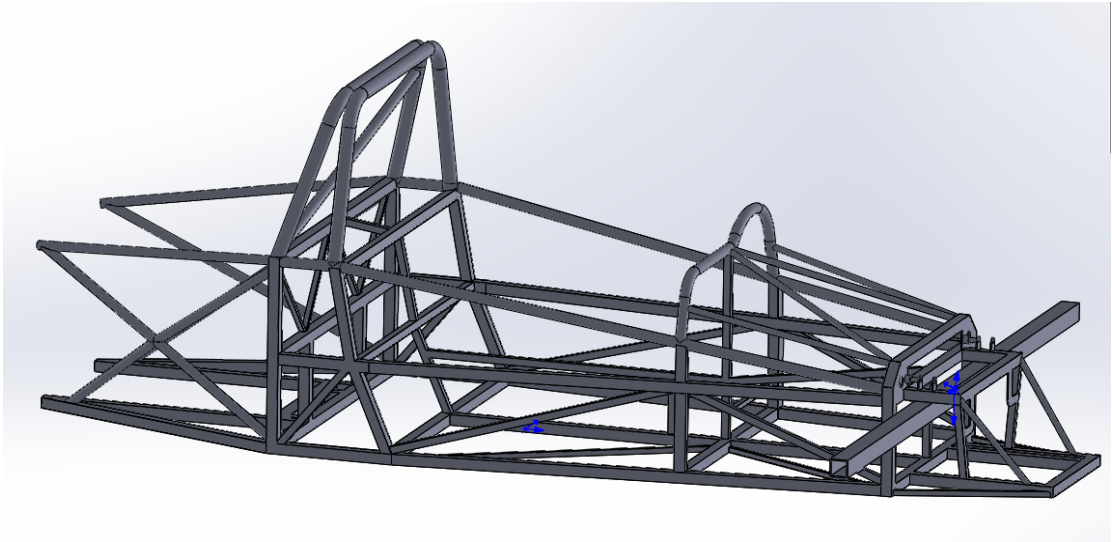


Figure 25-Original chassis

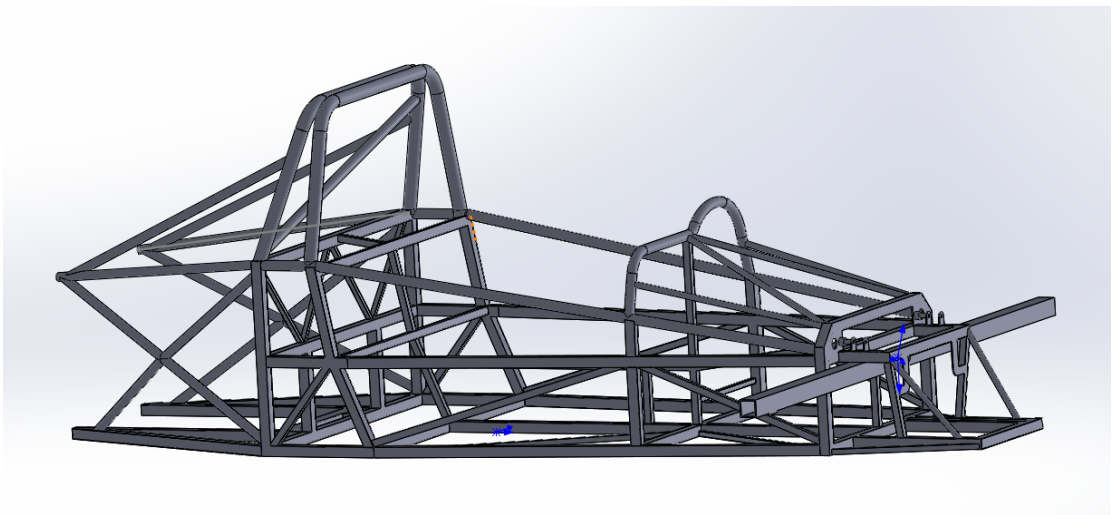


Figure 26-Chassis with rear support

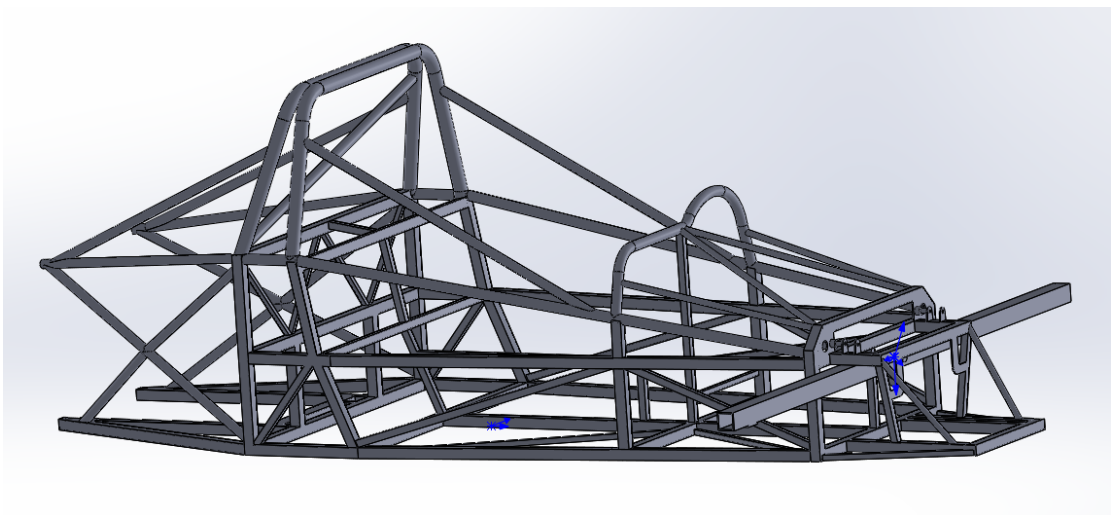


Figure 27-Chassis with both supports

With these three improved models, FEA process should restarted using the high-performance computers at room 006 in IQ building. However, due to the outbreak of the Corona virus, the university decided to temporarily close all campuses. After trying to communicate with the IT service hoping to borrow a high-performance computer from the school studio to continue to complete the FEA but get a negative response, own computer is used to continue the FEA process.

4.4 Simplified Model

Simplified model needs to be used on account of the failure of FEA on own computer whose RAM is not big enough. Simplified models have to be the base of all the following analysis due to the short of computer memory. Different from the simplified model adopted in the mid viva, the simplified model now adopted is no longer a simple framework on account of the analysis it involves.

4.4.1 Front Part

In the process of simplification, the front part of the model comes to the first because there are too many subtle structures in the front of the model, such as diagonal rods and fixed screw holes. In the process of continuously reducing the structure of the front part of the model, the stiffness of the front part was found to decrease rapidly. Originally, several structures were going to be observed which are designed to apply the load, but after FEA simulation analysis, it was found that its deformation is too large to meet the actual situation. Therefore, the front part of the model could only be simplified completely. As an alternative, a sketch has been drawn and extruded by 3 mm on the crossbar C3 shown in figure 28 as the reserved plane for applying the load.

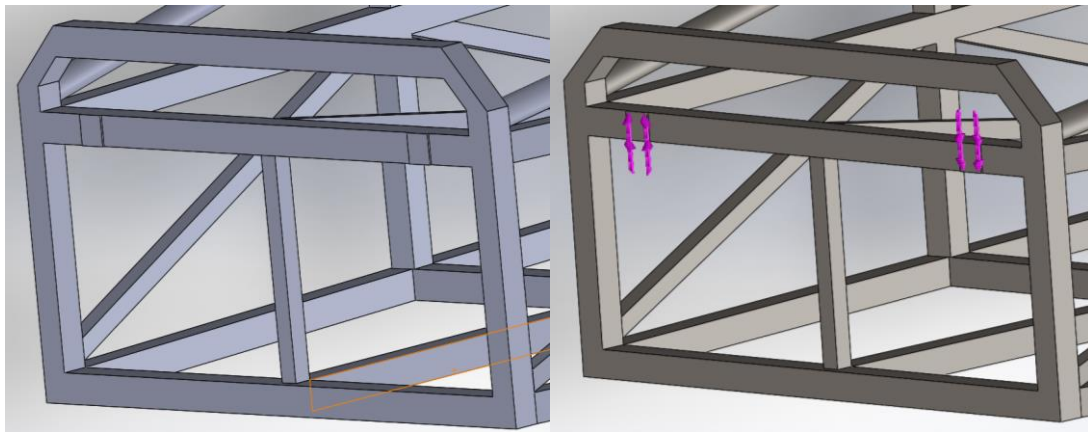


Figure 28-Simplified front part

4.4.2 Middle Part

For the simplification of the middle part of the model, first of all, the specific dimensions of the two diagonal braces in the basic level A are very small, and the impact on the overall structure strength is very small as well. Considering the optimization and the left-right symmetry of the entire structure, they could be removed. Secondly, because the basic sizes of the components in the middle are relatively large and the span is long as well, there are not many parts that can be simplified. Finally, in the optimization process of the top two levels, the middle of the G2 structure has continuous multi-angle bending, so the scanning feature is used, and the cross-sectional extrude feature is used when the two sides are connected to the E level, so it could be simplified. For other structures connected to G2, such as E4, E5, E6 in the

middle, and G3, G4, G5 in the front, it can be regarded as a support for the outer skinning and rivets on account of the structural size is also very small, could also be simplified. The original middle part and the new one which has been simplified are shown in figure 29.

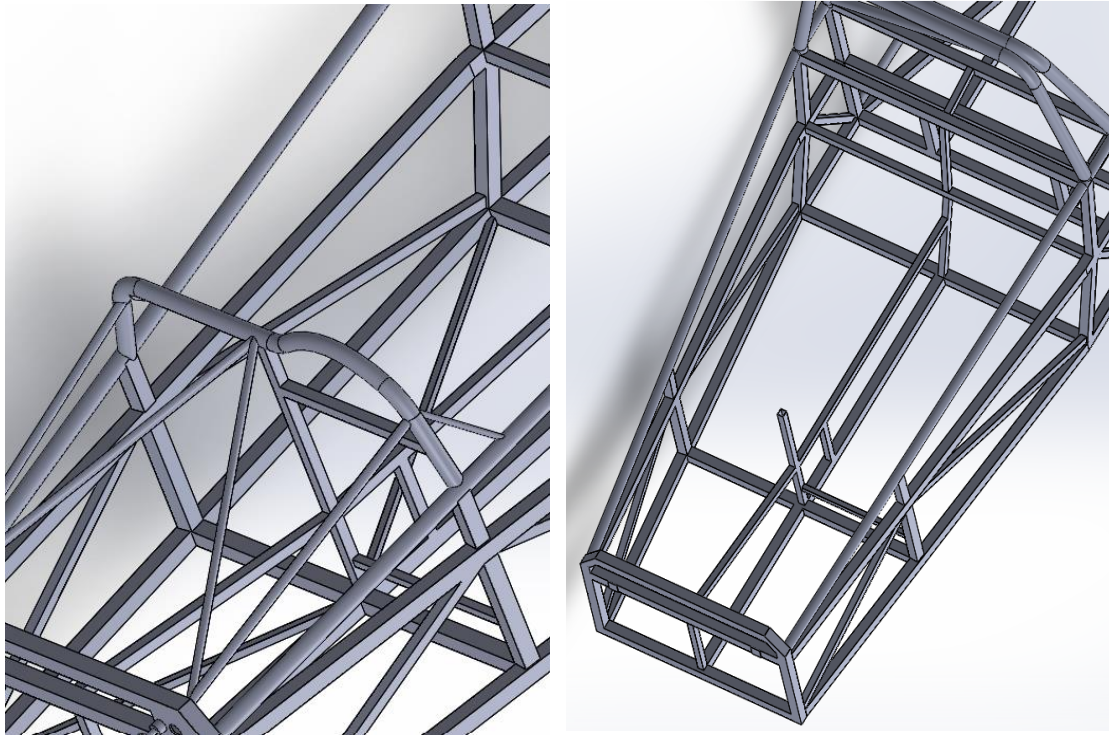


Figure 29-Simplified middle part

4.4.3Rear Part

When the SA1 campus is still open, through the preliminary FEA analysis of the original model, the torsional stiffness of the tail is the strongest for the overall structure. Since the structure of the tail is relatively simple, except for the necessary structural parts, there are only a few reinforcements. So for the optimization of the tail, the first step is to remove R8, R9, R10, R11 between R7 and R16. On the one hand, due to their small size, on the other hand, considering that they have a function as a partition, so they could be removed optimally. In addition to them, there are reserved mounting holes on A19 which is on basic level. Although it will bear the load, in this study, only a torque is applied to the front of the model to measure the torsional stiffness, so it can be optimized to be removed. At the same time, the reinforcements A20 and A21 between A18 and A19 can also be optimized and removed. The rear part before and after simplification is shown in Figure 30.

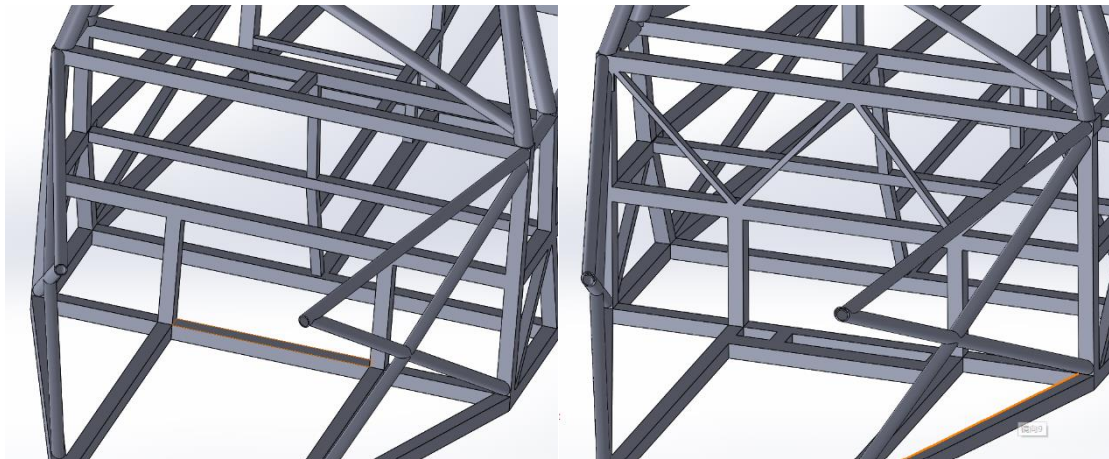


Figure 30-Simplified rear part

4.4.4 Structure of chassis

After simplifying the front, middle and back parts of the model, the entire simplified model is shown below.

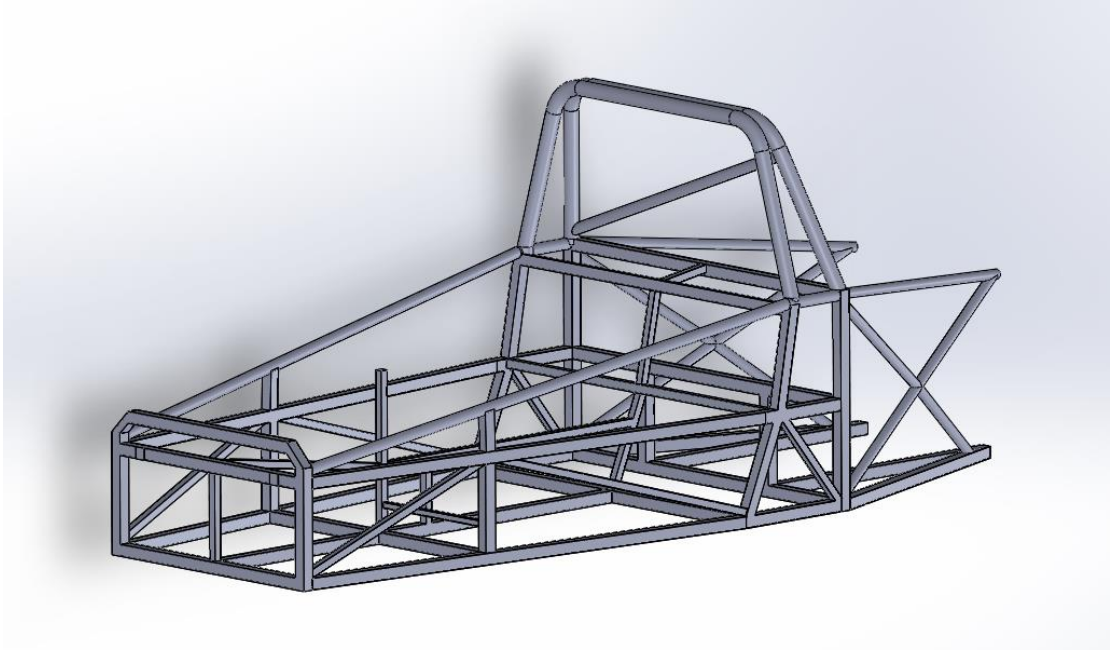


Figure 31-Simplified chassis

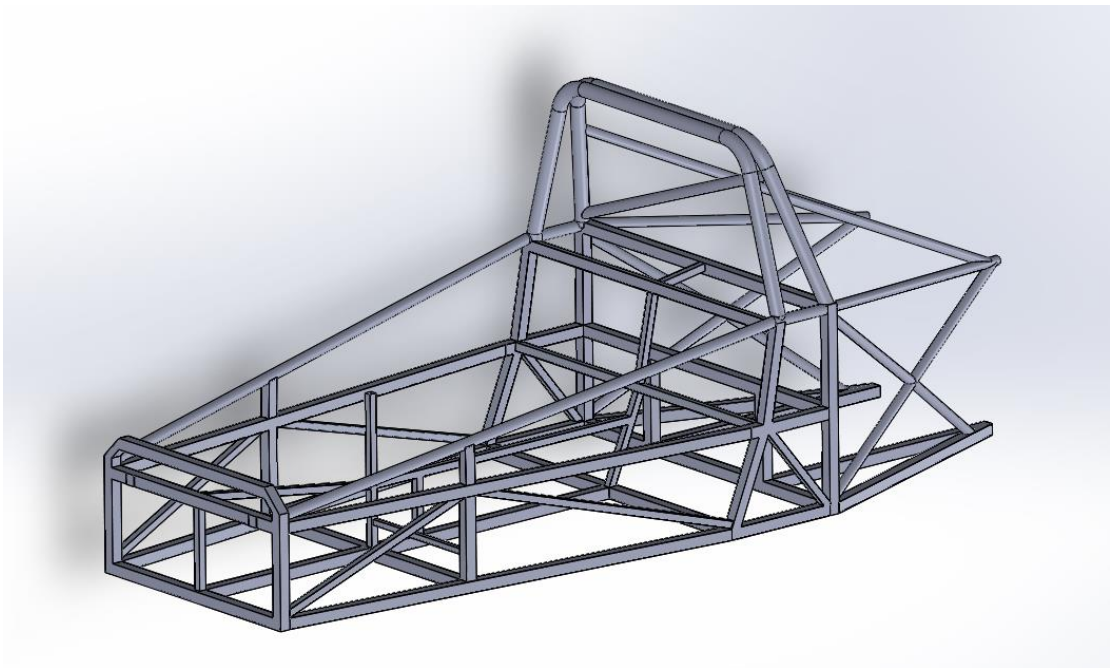


Figure 32-Simplified chassis with rear support

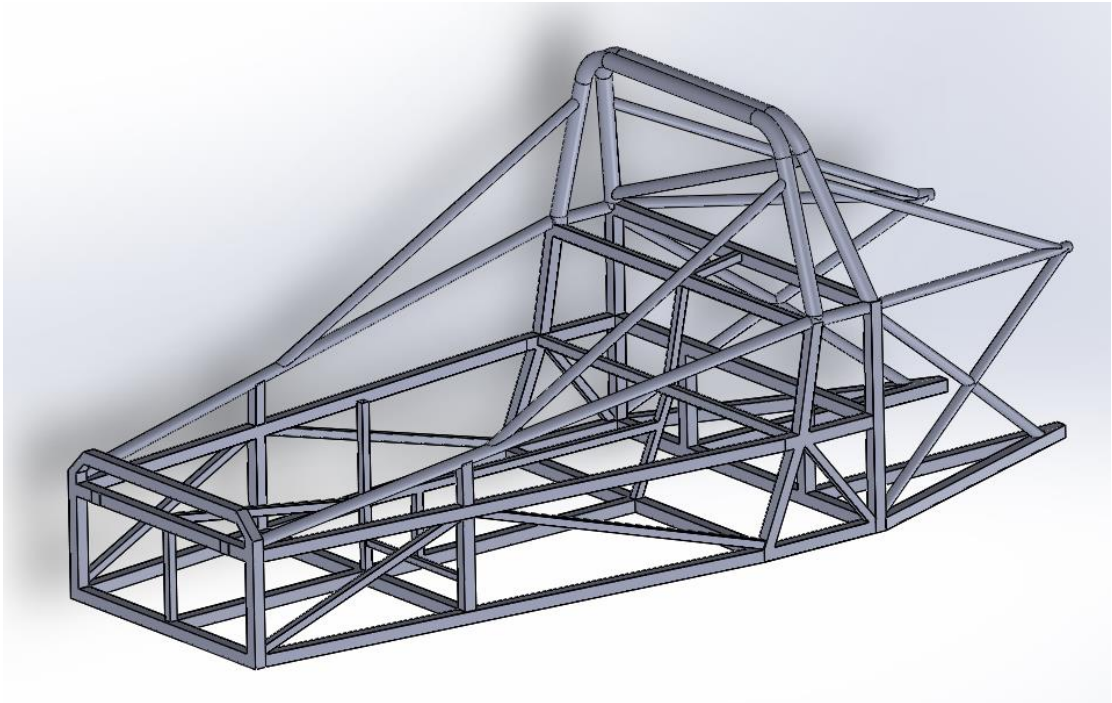


Figure 33-Simplified chassis with both supports

5 Literature Research

Before starting formal analysis work, a large amount of literature research work needs to be completed to ensure the correct conduct of the experimental theory and methods. Previously, the author had been exposed to Chinese literature search engines and databases, so through supervisor booked the instruction of professional staff of the University of Wales Trinity Saint David 's Library to explain the use of online library for the retrieval of digital resources and the use of SAE databases.

Although the online digital resources provided by the university are very rich, and the SAE database is also very practical, in the process of document search, it is inevitable to combine the Chinese literature search engine and database.

5.1 Type of Frame

Just like a human body is supported by a skeleton, a car must also have a skeleton which is the frame. The role of the frame is to bear the load, including the weight of the assembly and components of the car such as engine, transmission, drive shaft, front and rear axles, body and the impact of driving, acceleration, turning, and braking under various working conditions. The existing frame types include side beam type, middle beam type and comprehensive type (Tang, 2020). Under the general trend of global warming, energy saving and emission reduction, some passenger cars have eliminated the frame in order to reduce the mass, and adopt a load-bearing body that can withstand various loads. Formula cars need to participate in the race and the driving conditions on the track are poor, so the truss structure of the steel tube frame and the composite material MONOCOQUE structure are commonly used, and the carbon fiber MONOCOQUE cabin is gradually replacing the original truss structure steel tube frame. Due to the earlier competition time of the optimized frame involved in this article, it still uses a truss structure steel tube frame. The frame of this structure is welded by hollow metal tubes with different diameters and wall thicknesses. The production process of the frame with this structure is simple, does not require special processing technology and special equipment, and the frame can be processed only by welding or riveting. The advantage of this method is that the processing cost is low, and it is very easy to modify or partially strengthen the frame. It is only necessary to modify the welding position of the pipe fittings or add welded pipe fittings or metal reinforcement plates. The participating cars should be comprehensively considered from the aspects of frame strength, stiffness, processing difficulty and cost, and the use of special equipment, so that the car can fully exert its performance in the race (Zhang, 2014). Since the frame is the foundation of the entire car, it has to bear various loads inside and outside the car. Therefore, the frame is required to have sufficient strength and appropriate rigidity, and it is required to have the characteristics of simple structure and light weight. At the same time, the center of gravity of the car should be reduced as much as possible and a larger front wheel steering angle should be obtained to ensure the stability and steering flexibility of the car while driving.

5.2 Frame material

The design and selection of materials for racing frames must follow the principle of light weight. Lightweight means that the car should reduce its own weight as much as possible under the premise of satisfying the power and reliability. It refers to the weight reduction of a car of the same size or the same type under the same power. Racing car which has a smaller weight could obtain better fuel economy and acceleration or power, thereby increasing maximum speed and improving handling stability. The weight of the frame of a formula car is about 15% of the weight of the entire vehicle, so grasping the quality of the frame is of great significance for achieving the weight reduction of the entire vehicle.

For the selection of frame materials, several common materials were selected to compare their physical properties (as shown in table 4.2.1).

Tab.4.2.1-Selected material and physical properties

Material	Density	Yield	Tensile	Welding Performance
Name	kg/m ³	Strength	strength	
		MPa	MPa	
Q235	7850	235	>205	No preheating is needed during welding, the interlayer temperature and postheating are controlled, and heat treatment is not necessary to improve the structure after welding. The whole process does not require special process measures, excellent weldability
Q345	7850	345	>409	Good weldability, generally does not preheat before welding
4130	7850	460	>560	When the alloying element is in the lower limit, the weldability is quite good, but when the upper limit is approached, the weldability is general, and it needs to be preheated to above 175 degrees Celsius before welding
4340	7850	470	>745	Weldability is poor, high temperature preheating before welding, stress relief after welding, need to be used after tempering.

In the selection process of the materials used in the frame, it is obvious that the density of the four materials is the same, so the main considerations when choosing materials are their yield strength, tensile strength and welding performance.

First of all, for Q235 steel, it has excellent welding performance, no preheating before welding, and no heat treatment after welding. But its yield strength and tensile strength are relatively low compared to the other three materials.

Secondly, for Q345 steel, it has good welding performance, generally does not need to be preheated before welding, and also does not require heat treatment after welding. At the same time, its yield strength and tensile strength have also been improved compared with Q235 steel.

Compared with the first two materials, 4130 steel needs to be preheated to 175 degrees Celsius before welding, but the preheating temperature is not very high and it is easy to achieve. At the same time, due to its alloy characteristics, the welding performance is greatly affected by temperature, so the welding temperature needs to be controlled. There are two types of heat treatment after welding, annealing and normalizing. The two treatment processes have almost no effect on the yield strength, but have a greater influence on the tensile strength. Among them, the annealing treatment process is relatively easy to achieve. Although the tensile strength of the finished product is slightly lower than that of the normalizing treatment process, it has already met the conditions for the use of the frame. The yield strength and tensile strength of the finished products obtained by the two heat treatment processes have been greatly improved compared to the previous two materials.

Finally, 4340 steel, although it has high strength, toughness and outstanding hardenability and stability against overheating, but the white point sensitivity is high, and temper brittleness. It has poor weldability, high temperature preheating before welding, stress relief after welding, and use after tempering. Because it requires both preheating before welding and stress relief after welding, it must be tempered before it can be used.

After analyzing and comparing these four materials, it was finally determined that the material used for the frame was 4130 steel that was annealed and heat-treated at 865 degrees Celsius. The mechanical parameters of this material are shown in Table 4.2.2.

Tab 4.2.2-Mechanical Parameters of 4130 Steel

Parameter	Value
Density (kg/m ³)	7850
Elastic Modulus (GPa)	205
Poisson's ratio	0.285
Yield Strength (MPa)	450
Tensile Strength (MPa)	560

5.3 Stiffness analysis

For the whole vehicle, bending load and torsional load are the most common load conditions during the use of the frame, which has a particularly large impact on the fatigue life of the whole vehicle (Wu, 2011). Therefore, it is necessary to analyze the bending stiffness and torsional stiffness of the frame, and the value of the bending stiffness and torsional stiffness of the solver to verify the rationality and safety of the frame stiffness design (Huang, 2017).

5.3.1 Bending stiffness

The bending stiffness of the frame refers to the load per unit deflection of the vehicle frame deformation, or the ratio of the load to the maximum deflection of the frame caused by it. Generally, under bending conditions, the maximum deflection value of the frame is used to evaluate its overall bending stiffness. It is assumed that the frame is a simply supported beam with uniform bending stiffness, and a concentrated force F is applied in the middle (Zhi, Xu, & Li, 2019). The maximum deflection of the frame under actual conditions can be measured through experiments, and the bending stiffness of the frame can be calculated.

The analysis of the bending stiffness of the frame is mainly to investigate whether the bending stiffness is large enough, and whether the deformation curve at the longitudinal beam of the frame is smooth, and there can be no obvious mutation. The frame is welded with 4130 steel with an average design thickness of 1.80mm, and its deflection value should not exceed 1.5 times the corresponding design value, that is, the maximum allowable deflection value of the frame is 2.70mm.

FEA model for vertical stiffness analysis has been obtained. DOF in three translation directions are constrained to one direction which is Z on account of the four rear attachment points and front suspension attachment points in X axis and Y axis translation directions (Yang Lei, 2017).

Two vertical force could be loaded on both sides of the floor tubes or put a central vertical load on the central axis in four analysis reference planes. In this case, two central vertical loads have been applied to the model. The first load is 1000N applied to joint of A13 and A12 simulates the load of driver and seat. Another one is 1800N represents the load of engine and fuel tank been applied to A18. After that the deformation on loading points are be measured and the average value is regarded as the deformation in vertical direction. Bending stiffness is calculated by formula 1,

$$K = F/Z \quad (1)$$

In which Z is the average value of the deformation, F is the load applied on the central axis, K is the bending stiffness.

5.3.2 Torsional stiffness

If the wheels on one side of the car fall into a pit or the wheels on one side pass over the convex stones, the working condition of the frame will be reversed. Insufficient frame rigidity will cause vibration and noise, and also make the car uncomfortable and unstable. The reliability of some mechanical parts will be reduced as well (Li & Huang, 2019). However, the torsional stiffness of the frame should not be too large, otherwise it will increase the load of the frame and suspension system and make the grounding of the car tires worse, thereby worsening the ability to pass obstacles. The torsional stiffness of the frame refers to the torsion angle of the frame under torsional load. The analysis of the bending stiffness of the frame is mainly to investigate whether the torsional stiffness is large enough and whether the deformation curve of the frame is smooth.

When analyzing the torsional stiffness of the frame, the load that the frame needs to add is antisymmetric, and its constraints are also different from the bending stiffness. A vertical upward load of 1000N could be simply applied on one side of the front of the frame and another one of 1000N on the other side to simulate the torsional load. Setting of frame torsion boundary conditions is set the four fixing points at the rear of the model to be fixed.

The changes of the angle are measured in four analysis reference planes. Torsional stiffness is calculated by [formula 2](#),

$$K = T/\theta \quad (2)$$

In which T is the torque, θ is the torsion angle measured, K is the torsional stiffness.

5.4 Strength analysis

In order to ensure the safety and reliability of the frame under extreme working conditions, the stress must be below the yield limit of the material. In the analysis of the bending stiffness and torsional stiffness of the frame, in addition to the maximum displacement in each position, the results obtained also have the maximum stress data under two load conditions. From the simulation results, the specific value of the maximum stress of the frame can be known, so that the magnitude of the structural stress could be calculated, and finally the relationship between the maximum stress and the yield limit of the material can be found. During the optimization process, focus should be on the maximum stress point (Yuan & Zhao, 2020).

5.5 Cost

The cost of the frame is related to many factors, including the materials used in the frame, the plasticity of the materials, the amount of materials used, and the welding techniques and processes. Due to the difficulty of manually calculating the cost, it was decided to use the computer-aided software CES EDUPACK to calculate the cost. Through the communication between the supervisor and IT service, the author has been told that the school has only ten computers installed with the software, eight of which are in room 112 and two in room 006, but the IQ building where they are located is temporarily closed to students due to the

outbreak of the Corona virus. The author also tried to install the software with a personal computer. Although ANSYS account can register for a free trial, it does not include CES EDUPACK. Therefore, the analysis of the frame cost in this article cannot be carried out for the time being, so it can only be put into the future direction of research.

5.6 Weight

For a long time, lightweight has been a hot and difficult point in automotive technology, especially for racing. The lightness of the car can reflect the design level of the car, and it is more related to the power and fuel economy of the car, which directly affects the competition results.

Lightweight racing is a systematic project, which mainly includes four types of lightweight methods: lightweight materials, lightweight structures, lightweight manufacturing and functional lightweight (He, 2013). The combined application of multiple lightweight methods is better than the single lightweight method.

One of the goals of the optimization of the frame in this article is lightweight. The frame is the matrix of the racing car, and its performance directly affects the performance of the entire vehicle, so it is necessary to study the lightweight of the frame. This paper focuses on the FEA analysis of the strength and stiffness of the frame, and optimizes the frame to achieve light weight. Then, the FEA analysis is used to check the strength and stiffness of the optimized frame to verify its safety, so as to verify the feasibility of lightweight solutions.

5.7 Establishment of FEA Model

In the software SOLIDWORKS, the geometric model of the racing frame is divided into elements, and the bending stiffness and torsional stiffness are analyzed and calculated respectively. The boundary conditions and loads of the frame are established according to the force conditions in the two analysis processes, and the frame finite element model is established to lay the foundation for the finite element calculation and optimization based on the obtained results.

5.7.1 Unit Selection

The racing frame in this article is a truss structure steel tube frame, and the basic units that make up the frame are hollow metal round tubes and square tubes with different diameters and wall thicknesses, which are assembled by welding. Therefore, when constructing the mesh model of the frame, quadrilateral elements have been used as the main unit and allow a small number of triangular element mixed element models. The total length of the frame in this case is 2397 mm. Considering the position accuracy when constraining and loading the frame and the proximity of the element to the geometry, the element size should be set smaller, but the campus is temporarily closed to students due to the outbreak of the Corona virus, considering the need to meet the personal computer running memory of 12GB as the basic hardware condition, the maximum unit size was finally determined to be 2.50 mm.

5.7.2 Meshing

When meshing the structure, the following principles generally need to be followed:

- (1) Divide an appropriate number of mesh according to analysis requirements and purposes, and also consider the computing power of the computer, not that the finer the mesh, the better;
- (2) Use a slightly dense mesh for the large undulating parts of the model, and use an appropriate sparse mesh for smooth parts to ensure that the mesh size and density are appropriate;
- (3) It is better to use quadrilateral element form;
- (4) The transition between units should be smooth.

Under the determined mesh scheme, the mesh quality parameters of the three simplified models are shown in the following table:

Tab 4.7.2-Mesh Quality of Three Simplified Models

Simplified Model	Maximum Unit Size (mm)	Minimum Unit Size (mm)	Number of Section	Number of Unit
Original	2.50	0.50	4884429	2458118
With Rear Support	2.50	0.50	6368101	3223378
With Both Supports	2.50	0.50	7793525	3945143

5.7.3 Boundary Conditions and Load Handling

During the process of carry on the simulation analysis to the frame, the closeness of the boundary conditions and load settings in the software and the external conditions and load conditions encountered during actual driving directly affect the accuracy and reliability of the analysis results. When the car is at rest, it can only bear the weight of the car body, the weight of the frame, the assemblies and accessories installed on the frame, and the sum is the static load.

There are two types of load application methods: concentrated load and uniform load. The concentrated load is calculated by loading the concentrated force on the master node, and then calculating the displacement of the slave node through the displacement of the master node. Uniform load can obtain the force per unit area by applying a ratio of a certain load to the element area and apply it uniformly to the surface element. The direction can be the element normal direction or a custom direction.

In the analysis of bending stiffness and torsional stiffness in this paper, the above two load application methods are combined. In the process of determining the amount of the load, the concentrated load is used to quickly determine the amount of the load that should be applied at each load action point. In the actual application process, the uniform load is used, the load direction is determined after the load application plane is selected, the load amount is entered and the uniform distribution is checked, thereby completing the load application process.

6 Analysis Work

6.1 Improved model

By comparing the bending stiffness and torsional stiffness experiments and experimental results of the three simplified models, the bending stiffness and torsional stiffness in the middle of the frame are lower than those in the rear. Therefore, the optimization model mainly strengthens the middle structure of the frame, appropriately reduces the bending stiffness and torsional stiffness of the rear of the frame, and at the same time, reduces the weight and center of gravity of the entire frame. The significance of reducing the quality of the vehicle has been discussed in detail earlier. Reducing the center of gravity of the vehicle can increase the grip of the four tires of the car, which can greatly help the start and acceleration of the car; at the same time, it can reduce the inclination of the vehicle when turning. Make the car turn smoothly, there will be no obvious left and right shaking feeling. The optimized simplified model is shown in Figure 34:

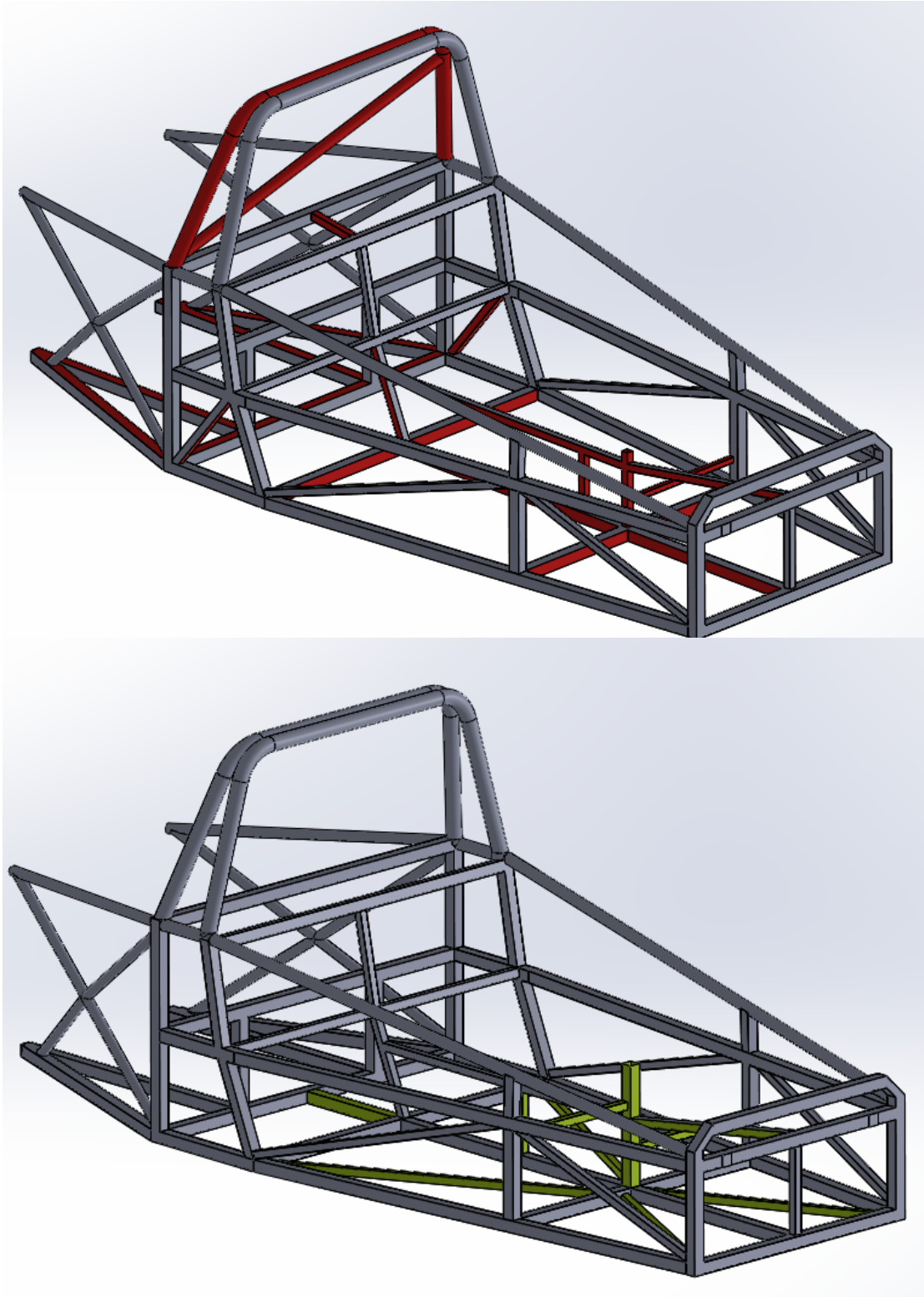


Figure 34-Simplified chassis

6.1.1Improvement of rear part

In the bending stiffness FEA process, since the concentrated load at the rear is evenly distributed between the A18 structure between R5 and R6, in the experimental results, the A18 structure becomes the location where the maximum displacement point of the rear of the frame appears, and at the same time the maximum stress. The point also appears at the node adjacent to the A18 structure. The torsional stiffness FEA results show that the torsional stiffness at the rear of the frame is sufficiently high.

In the process of optimizing the rear of the frame, firstly, square tube with side length of 30 mm and thickness of 1.28 mm is added at the middle position of the A18 structure is also the middle axis position of the entire frame which is between the A15 and A18 structures to be consistent with them to increase local bending stiffness. At the same time, the E3 between E2 and R16 in the vertical direction of the newly added square tube is removed to make up for the increase in mass caused by the newly added square tube and to a certain extent, reduce the centroid.

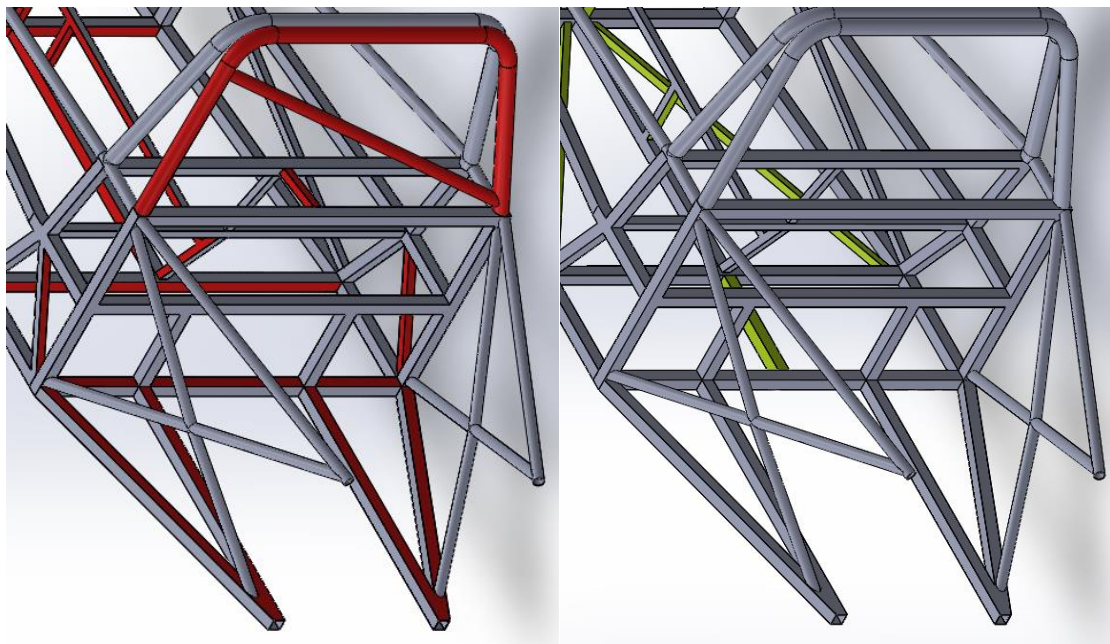


Figure 35-Improvement of rear part

In the analysis of the experimental results of bending stiffness and torsional stiffness, it was also found that due to the high stiffness at the rear of the frame, the B6 structure and its mirror rod between the B5 and R12 rods can be considered to be removed in the area where the middle and rear of the frame are connected. At the same time, the top position has R22 and G1 double reinforcement structures, and both have an outer diameter of 38 mm and a wall thickness of 2.09 mm, so the diagonal pull reinforcement structure R19 of the R22 component can also be omitted. In this way, the high stiffness of the tail can be used to compensate the stiffness loss at the connection, and the vehicle's center of gravity can be lowered.

6.1.2 Improvement of middle part

From the FEA analysis results of bending stiffness and torsional stiffness, the stiffness in the middle of the frame is low. Since the concentrated load is evenly distributed on the contact surface of A9 and A12 rods on the basic level during the bending stiffness test, the maximum displacement point appears here, and at the same time the maximum stress point also appears on the adjacent rod connection point. The results of torsional stiffness analysis also show that in the middle of the frame, the stiffness decreases significantly from the center of gravity. The optimization of the middle of the frame starts with the B9 part, and its outer edge length is expanded to 30 mm while increasing its wall thickness from 1.18 to 1.28 mm. Then for the left and right direction, first increase the outer length of the B7 part on the left side of B9 from 15.00 to 20.00 mm, and then add a lateral link on the right side of B9 symmetrically with the B7 part.

In addition to the above changes, due to the cross structure of A9 and A12 on the basic level, the bottom surface is divided into four rectangular areas. It is well known that triangles are more stable than rectangles, so add diagonal bars in each area to divide rectangles into triangles to improve stability. At the same time, due to the increased quality of the basic level, the height of the center of gravity of the entire frame can be reduced to a certain extent. Finally, there is another slight adjustment. On account of the relative distance between the A15 and A12 parts is too far, and the position of B10 between them is relatively forward, so the position of the B10 part is moved slightly backward, and the relative position of the B10 and B9 parts is increased from 129 mm to 350 mm.

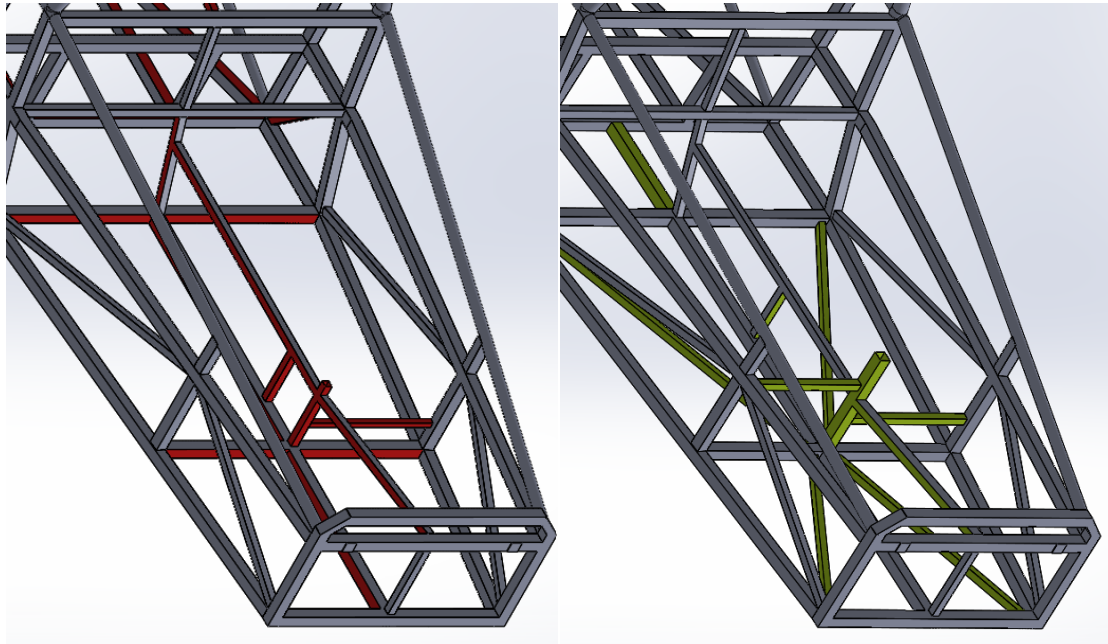


Figure 36-Improvement of middle part

6.2 Bending stiffness

By observing the bending stiffness experimental results of the four models, it is found that the effects of the middle load and the rear load hardly overlap, so the bending stiffness experimental results of the frame can be analyzed separately.

6.2.1 Bending stiffness of middle part

The load for test in middle part is 1000N which is been applied to joint of A13 and A12 simulates the load of driver and seat. The experimental results are shown in the following table:

Tab 5.2.1-Experimental results of bending stiffness (middle)

Model	Load (N)	Maximum Displacement (mm)	Maximum Stress (MPa)	Bending Stiffness (N/mm)
Original	1000	1.244	260.9	803.86
With rear support	1000	1.242	251.8	805.15
With both supports	1000	1.238	238.6	807.75
Improved	1000	0.933	220.9	1071.81

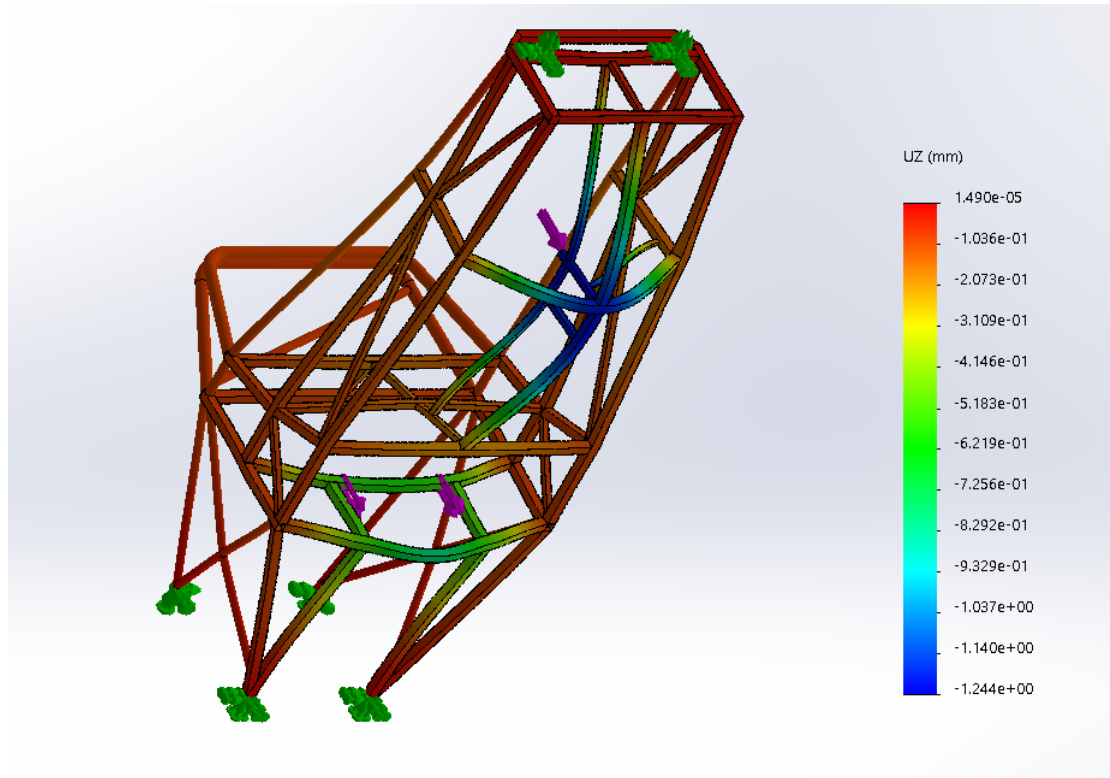


Figure 37-Displacement in bending stiffness test

It can be seen from the table data that the reinforcement structure at the rear and the middle of the frame has a certain improvement effect on the bending stiffness of the middle with the amplitudes of 0.16% and 0.48%, respectively, which are relatively small. For the model optimized with the FEA analysis results, due to more changes made to the middle of the frame, the bending stiffness of the middle has been significantly improved, from 803.86 N / mm to 1071.81 N / mm with an increase of 33.33%.

6.2.2 Bending stiffness of rear part

The load for test in rear part is 1800N which represents the load of engine and fuel tank been applied to A18. The experimental results are shown in the following table:

Tab 5.2.2-Experimental results of bending stiffness (rear)

Model	Load (N)	Maximum Displacement (mm)	Maximum Stress (MPa)	Bending Stiffness (N/mm)
Original	1800	0.792	234.4	2272.73
With rear support	1800	0.780	227.8	2307.69
With both supports	1800	0.778	219.4	2313.62
Improved	1800	0.709	197.2	2538.79

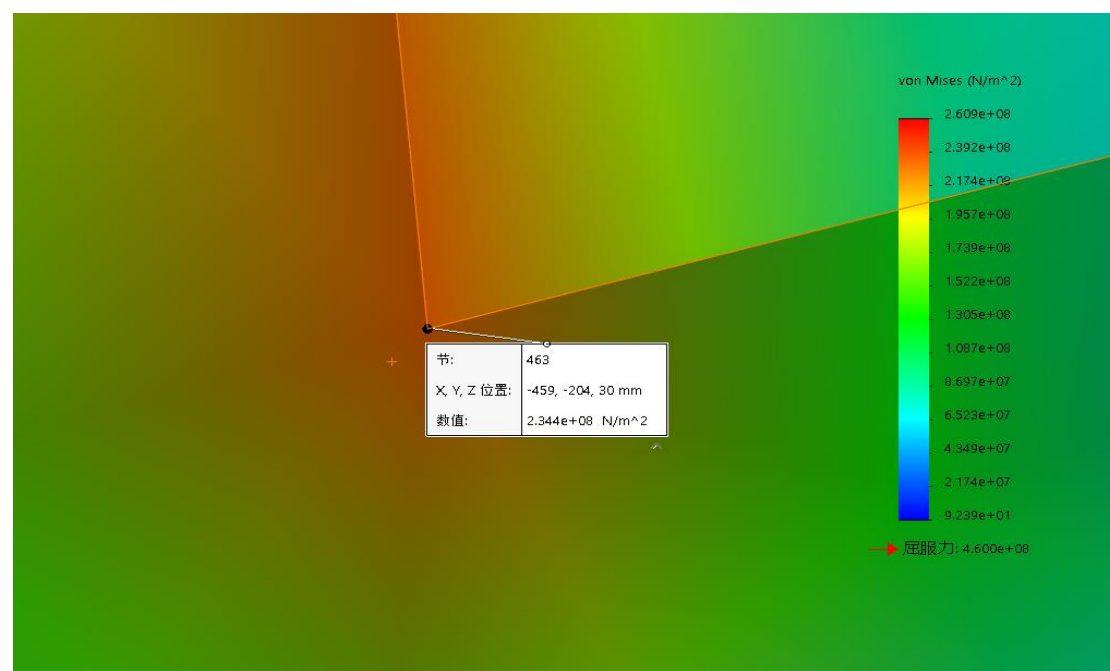


Figure 38-Maximum stress of rear part in bending stiffness test

It can be seen from the table data that the rear and middle reinforcement structures of the frame have a certain improvement effect on the rear bending stiffness, the amplitudes are 1.54% and 1.80%, respectively, which is better than the effect on the middle frame bending stiffness. For the model optimized for FEA analysis results, the rear bending stiffness has also increased, from 2272.73 N / mm to 2538.79 N / mm, an increase of 11.71%.

6.3 Torsional stiffness

For the torsional stiffness of the frame, both real data and FEA analysis data are obtained. Before the campus stopped opening to students because of the outbreak of virus, the measurement of the torsional stiffness of the real frame was completed. In the actual measurement process, the load is applied to a section of the action rod by a jack on one side, and indirectly acts on the suspension installation position at the front of the frame. Through the portable axle weight electronic scale installed under the jack in advance, the preload before the start of each experiment is controlled at 25 kg, which is about 245.25N, and the experimental load is 65 kg, which is around 637.65N. The actual experimental load and FEA analysis load details are shown in the following table:

Tab 5.3.1-Actual load and FEA analysis load

Test	Load (N)	Distance (m)	Torque (N*m)
Real test	637.65	0.845	538.81
FEA	2254.46	0.239	538.81

Due to a pair of antisymmetric forces applied during the FEA analysis, so a vertical upward load of 1127.23N could be simply applied on one side of the front of the frame and another one of 1127.23N on the other side to simulate the torsional load.

6.3.1 Real test

After obtaining the actual torsional stiffness experiment data, enter it into Excel, and perform the corresponding calculation to obtain the actual torsional stiffness value of each data collection point. Using the torsional stiffness of each data collection point under three different conditions, draw the following three line charts of the torsional stiffness of the frame with the position, and put their three lines in the same chart for direct comparison with each other.

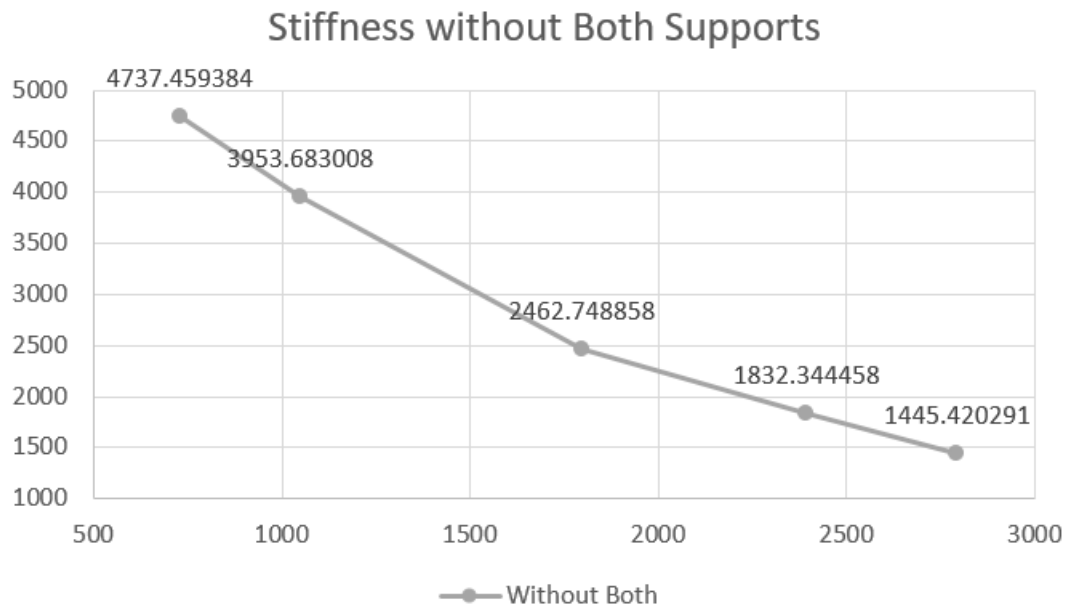


Figure 39-Torsional stiffness of original model

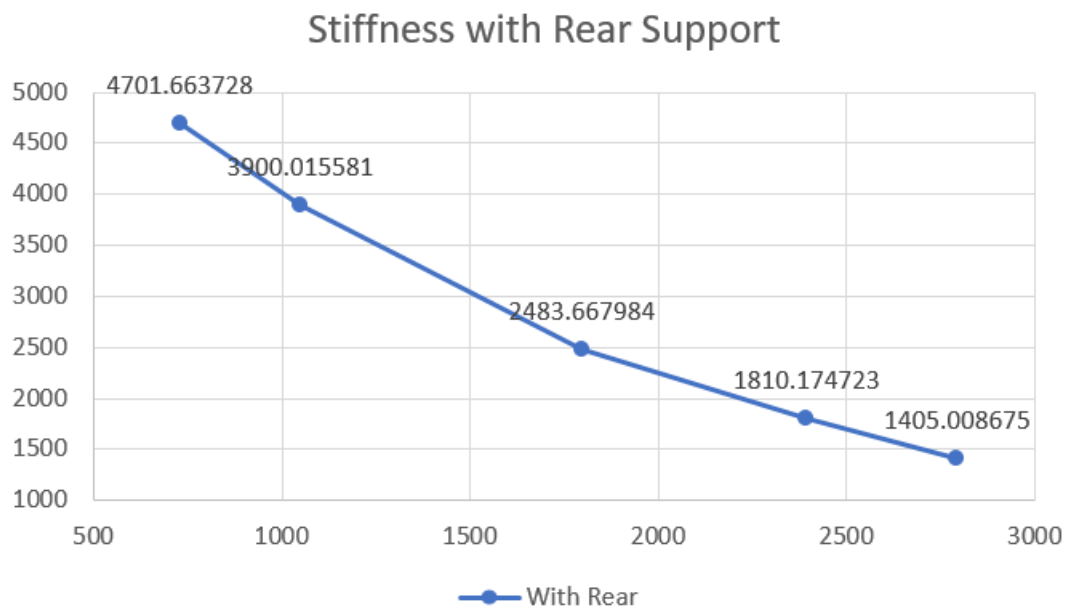


Figure 40-Torsional stiffness of model with rear support

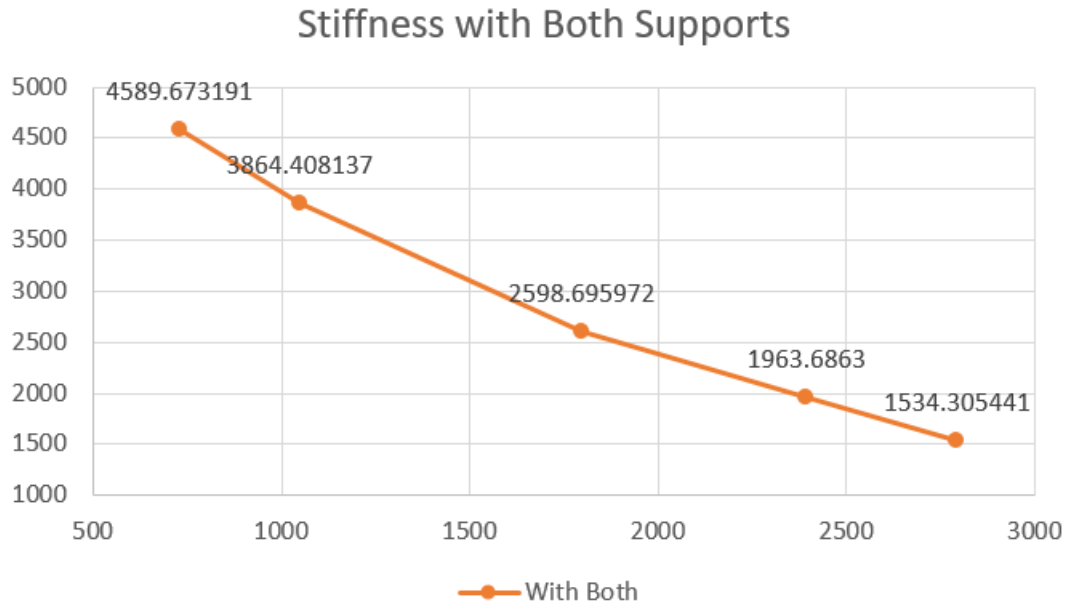


Figure 41-Torsional stiffness of model with both supports

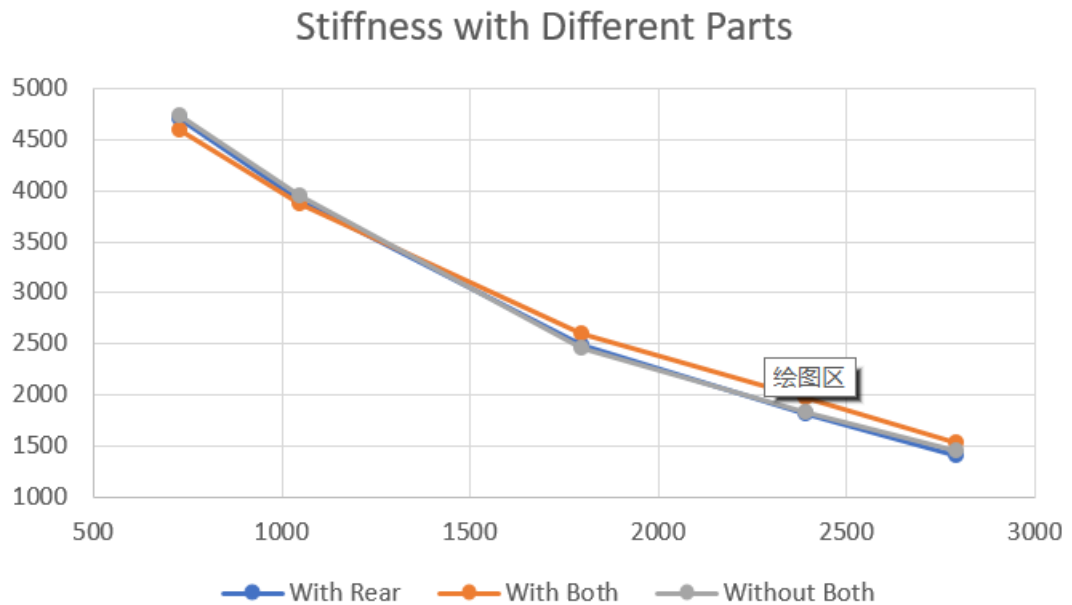


Figure 42-Torsional stiffness of three models with different parts

By comparing the results of these three experiments, it can be seen that only add the rear reinforcement structure has little nearly no effect on the torsional stiffness of the entire frame. For the reason of this, first of all, the components used in the rear structure of the frame have a larger diameter and thickness. After that, a large number of triangular stabilizing structures are used while the span is not large. So the torsional stiffness itself is relatively large. Therefore, when the load is not big enough, the effect of installing the rear reinforcement structure on its torsional stiffness is relatively small. The situation has changed significantly when the central reinforcement structure is added. The slope of the polyline can be found,

especially from the data collection points B to C, is significantly reduced which means that the torsional stiffness of this part has been greatly improved. From the perspective of the torsional stiffness of each data collection point, the torsional stiffness of point C is increased from 2462 to 2598 N * m / ° with an increase of 5.5%. At the same time, the torsional stiffness at point D increased from 1832 to 1963 N * m / ° which has an increase of 7.2%. It can be seen that under low load, the rear and middle reinforcement structures has a certain degree of improvement in the torsional stiffness of the frame, but the increase from the middle reinforcement structure is more obvious when it comes to the comparison with rear reinforcement structure.

6.3.2 FEA

Similar to the actual torsional stiffness test, the FEA analysis results for the four models also selected four data collection planes with a total of eight data collection points. The displacement of each point of each model is recorded, and the distance between two data collection points in each data collection plane is measured as well. Deflection angles of the frame can be directly calculated using the small angle theory. The small angle theory says that when the angle is very small, the sine value of the angle is equal to the tangent value, and at the same time equal to the radian value of the angle. When the deflection angle of the frame in the data collection plane is obtained, the torsional stiffness can be easily calculated in combination with the applied load.

The displacement data of each data point is shown in the following table:

Tab 5.3.2.1-Displacement (mm) of every point

Model	A left	A right	B left	B right	C left	C right	D left	D right
Original	0.8749	-0.8724	0.9640	-0.9609	1.281	-1.270	1.849	-1.843
With rear support	0.8703	-0.8777	0.9580	-0.9543	1.282	-1.270	1.847	-1.839
With both supports	0.7943	-0.7919	0.8829	-0.8804	1.258	-1.248	1.780	-1.777
Improved	0.8081	-0.8058	0.8939	-0.8920	1.220	-1.220	1.769	-1.766

Correspondingly process the data in the Excel table to obtain the line chart of the torsional stiffness of each model with the position as follows:

Stiffness without Both Supports

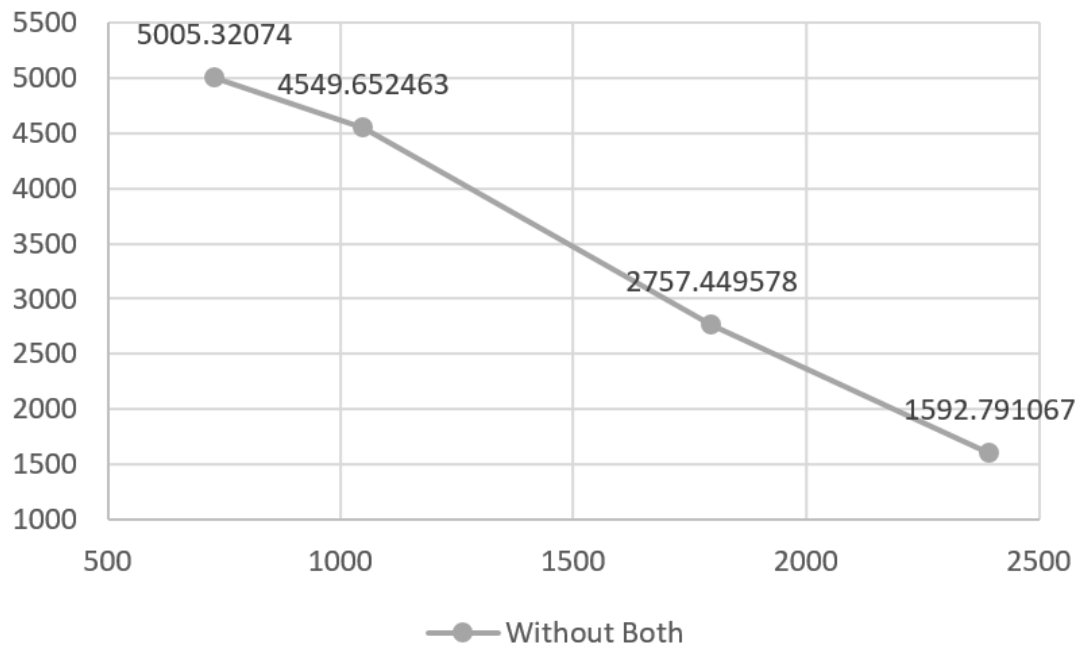


Figure 43-Stiffness without support

Stiffness with Rear Support

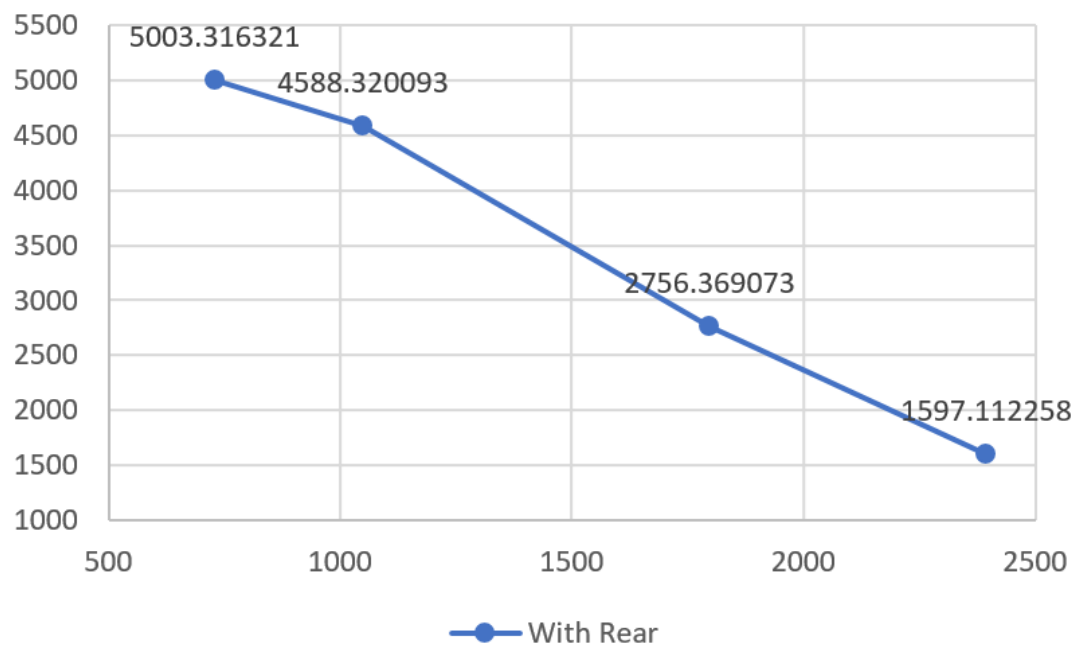


Figure 44-Stiffness with rear support

Stiffness with Both Supports

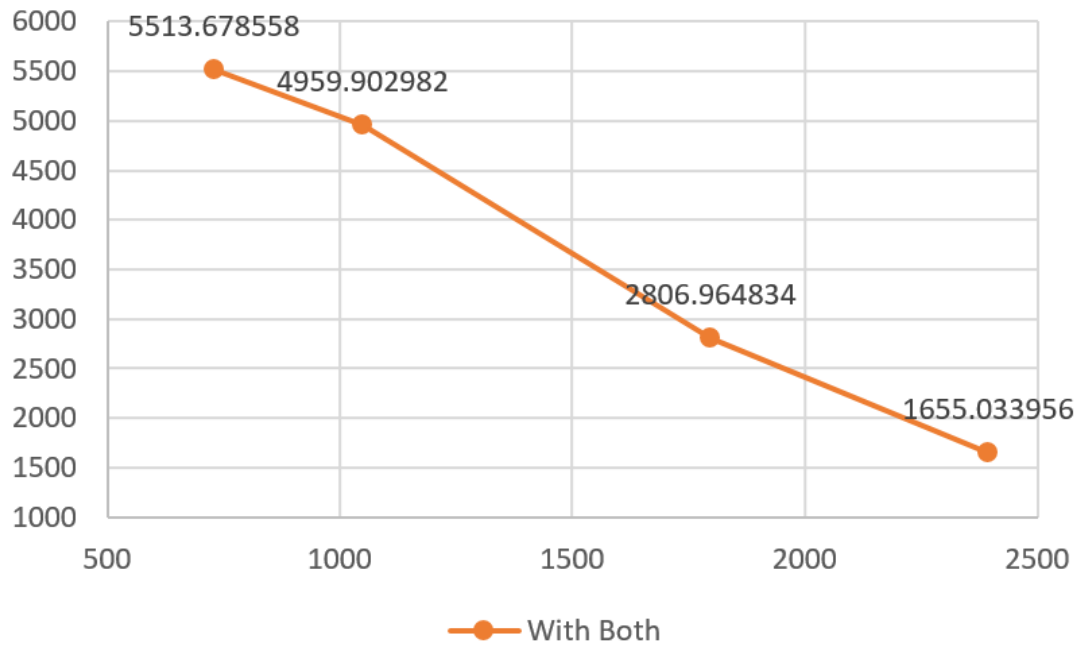


Figure 45-Stiffness with both supports

Stiffness of Improved Model

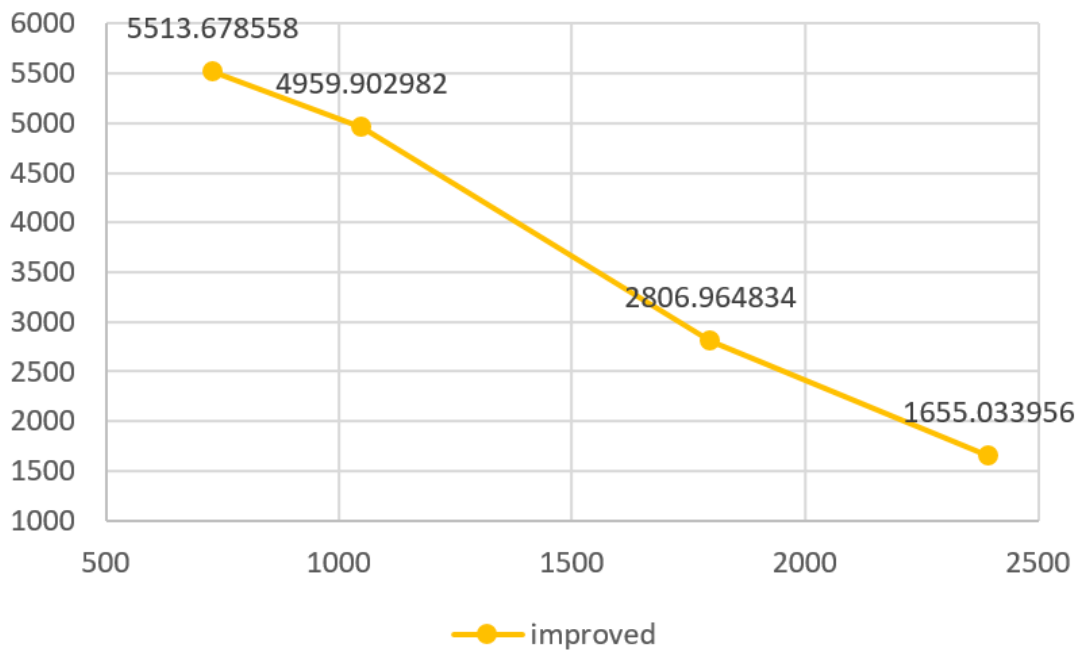


Figure 46-Stiffness of improved model

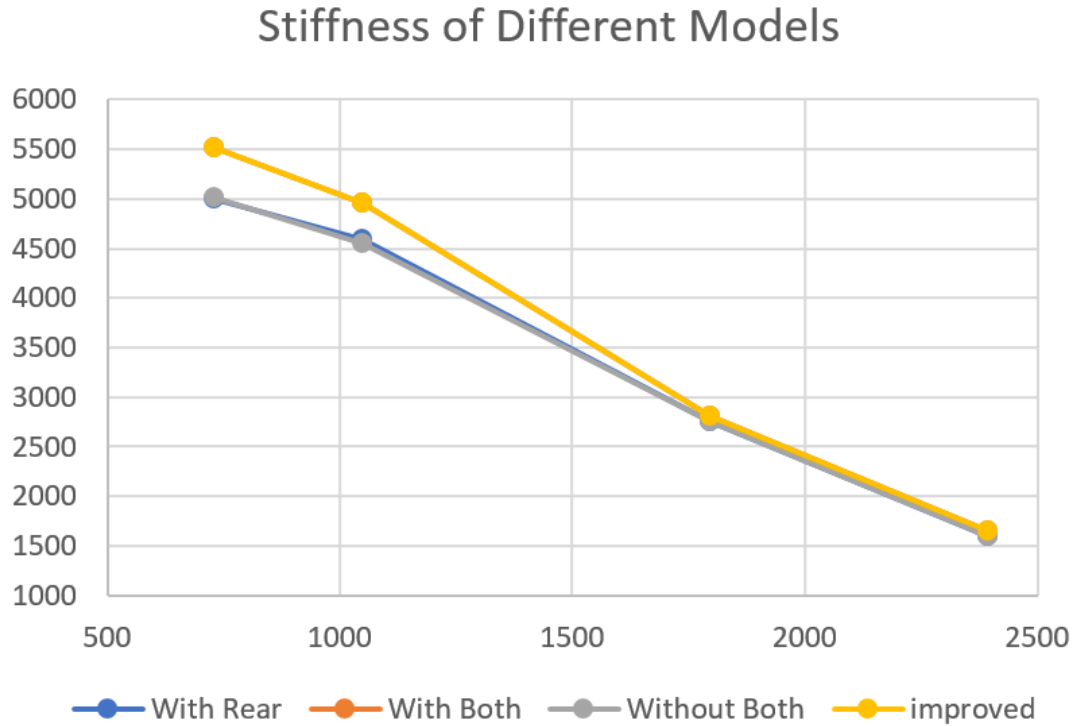


Figure 47-Stiffness of Different Models

Analysis of the torsional stiffness experiment results of these four models shows that the rear reinforcement structure does not improve the torsional stiffness of the frame as much as the middle reinforcement structure, especially for the rear data collection points A and B. From the comparative analysis chart, the optimized model has a similar effect on the torsional stiffness of the frame as the model with the middle and rear reinforcement structures added, and the optimization effect is even better at points C and D at front of the frame.

For the improved model, the improvements of every point compares to the original model are as follows:

- Point A -----8.27%
- Point B -----7.69%
- Point C -----4.55%
- Point D -----4.55%

6.4 Strength analysis

To ensure the security and reliability of the frame, stress needs to be lower than the material yield limit (Yang Lei, 2017). The yield limit for the martial which is 4130 Steel is 450MPa. The maximum stress during the bending stiffness experiment is 260.0MPa while the maximum stress during the torsional stiffness experiment is 227.4MPa. Both of them are far less than yield limit of the material.

6.5 Weight and center of gravity

In the software of SOLIDWORKS, use the quality attribute plug-in (Figure 49) in the evaluation feature to view the quality and center of gravity related information of each model and organize it as shown in the following table:

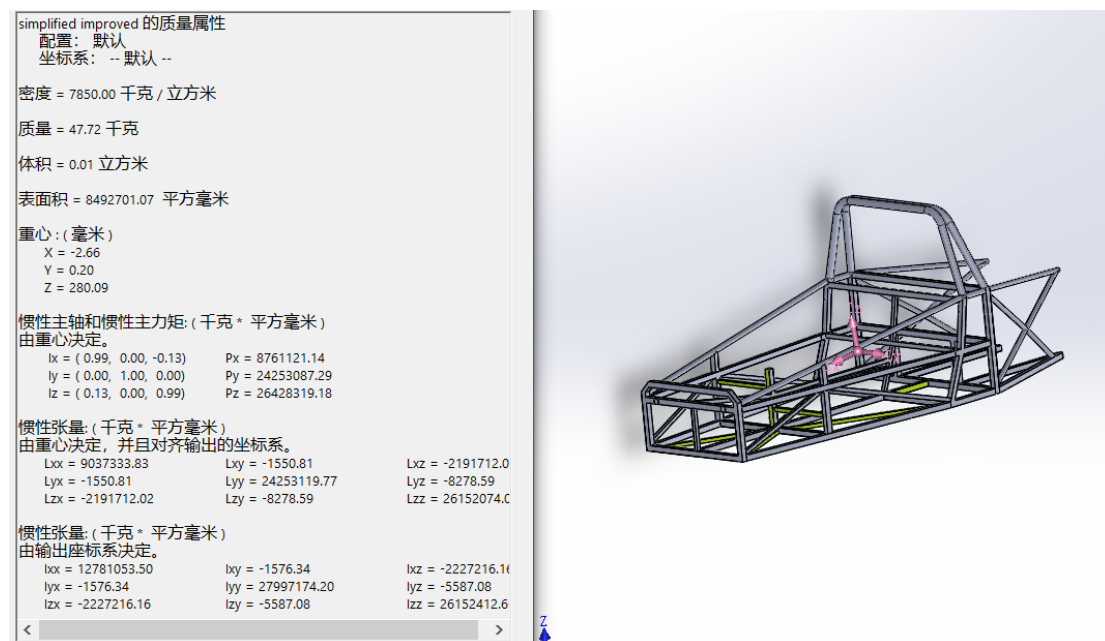


Figure 48-Parameters of weight and center of gravity

Tab 5.5.1-Weights and Improvements

Model	Weight (kg)	Improvement (%)
Original	46.48	
With rear support	48.53	-4.41%
With both supports	50.84	-4.76%
Improved	47.72	6.14%

Tab 5.5.2-Height of center of gravity and Improvements

Model	Height (mm)	Improvement (%)
Original	304.55	
With rear support	318.81	-4.68%
With both supports	332.76	-4.38%
Improved	280.09	15.83%

It can be seen from the data in the above two tables that whether the original reinforcement structure is applied to the rear or middle of the frame, it will increase the height of the center of gravity while increasing the weight of the frame. The model optimized for the FEA results can have better bending stiffness when the torsional stiffness has the same performance as the model using the middle and rear reinforcement structures. More importantly, the optimization scheme can reduce the weight of the frame from 50.84kg to 47.72kg with a drop of 6.14% while greatly reducing the height of the center of gravity. The height of the center of gravity of the frame is reduced from 332.76mm to 280.09mm through a reduction of 15.83%.

7 Future Direction

The ultimate goal of optimization in most vehicle frames is light weight, which is also where this article focused on while explored the relationship of the strength, stiffness, weight and cost.

Lightweight racing is a systematic project, which mainly includes four types of lightweight methods: lightweight materials, lightweight structures, lightweight manufacturing and functional lightweight (He, 2013). As everyone knows, the combined application of multiple lightweight methods is better than the single lightweight method. So this article mainly combines lightweight materials and lightweight structures to optimize the frame.

The optimization of this frame can also be carried out from many aspects. From the perspective of lightweight materials, we can explore the quality reduction that various new materials can bring when meeting the strength and stiffness conditions. From the perspective of lightweight structures, you can try more modern body structure design, such as cage body. You can even combine lightweight materials and lightweight structures to try the carbon fiber MONOCOQUE cockpit design. In terms of lightweight manufacturing, you can try a more advanced manufacturing process. For example, replacing welding with adhesive can reduce the weight of the frame to a certain extent. At the same time, it can use a smaller outer edge length or diameter under the conditions of strength and rigidity Standard material with smaller and thinner wall thickness. Functional lightweight does not have many special expressions for the lightweight design of the frame, but it is very important for the lightweight design of the entire vehicle.

8 Conclusion

It can be seen from the data obtained from actual experiments that under the existing load conditions, there are already two methods for strengthening the frame to improve the torsional stiffness. Among them, the reinforcement structure at the rear part is obviously inferior to the reinforcement structure at the middle part in improving the torsional stiffness. Among the two representative points, the torsional stiffness of point C is increased from 2462 to 2598 N * m / ° with an increase of 5.5%. At the same time, the torsional stiffness at point D increased from 1832 to 1963 N * m / ° which has an increase of 7.2%.

In the FEA analysis, because the simplified model and the actual frame are used, there is a large difference, so the results obtained by the FEA cannot be compared with the data obtained by the actual experiment. FEA uses a total of four models, including an original model, two models that use the same method of strengthening the structure as the actual model and the last one has been optimized for the FEA analysis results.

By comparing the FEA analysis results of the first three models, it can be found that in the bending stiffness experiment and the torsional stiffness experiment, the two strengthening methods have a certain effect on the improvement of the torsional stiffness, but the changes brought about by the application of the middle strengthening structure far greater than the rear reinforced structure, the trend of the results obtained is consistent with the actual experiment, and also verifies the accuracy and applicability of the FEA analysis.

For the optimization scheme designed after analyzing the FEA results of the first three models, both the bending stiffness and the torsional stiffness are greatly improved. At the same time, the weight and center of gravity of the frame are also reduced. The increase in the bending stiffness of the center is 33.33%, the increase in the bending stiffness of the rear is 11.71%, the increase in the torsional stiffness is from 4.55% to 8.27%, the decrease in mass is 6.14% and the decrease in the height of the center of gravity is 15.83%. It proves the feasibility and applicability of this optimization scheme.

Due to the impact of the campus temporary close to students due to the outbreak of virus, there are still many tasks not completed, such as testing the bending rigidity of the real frame, estimating the cost of the frame before and after optimization, and so on. There are also many tasks that have to be performed with simplified models so that the results cannot be compared with actual models in real life. If there is a chance, the author still hopes that the project can be completed as well as possible.

9 Reference

- HeShaohua. (2013 年 6 月 1 日). Racing Car Lightweight System Methods and Frame Optimization. Dissertation of Guangdong University of Technology, 页 4-56.
- HuangHaihua. (2017). Simulation of optimization design for load strength of automobile frame. Computer Simulation, 114-118.
- LiTietie, & HuangHaisong. (2019 年 08 月 12 日). Finite Element Static Analysis and Research of Commercial Vehicle Frame. Machinery and Electronics, 页 29-32.
- TangPing. (2020). The analysis of the types of car frames. Technology Forum, 5-5.
- Varma Pakalapati, V. K. (2011, 04 12). CAE based 'Multi Objective Optimization approach for Spot Weld Connections Layout' in Automotive Structure. *SAE INTERNATIONAL*, pp. 50-62.
- Vlahopoulos, N. (2011, 05 17). Modeling of Stiffened Panels Using the Energy Finite Element Analysis. *SAE INTERNATIONAL*, pp. 60-28.
- WuQuanhe. (2011 年 7 月 28 日). Stress analysis of automobile frame under severe load. Mechanical Design, 页 46-49.
- Yang LeiLi, Chuxuan Wang and Yunqing ZhangQiang. (2017 年 03 月 28 日). Loads Analysis and Optimization of FSAE Race Car Frame. *SAE INTERNATIONAL*, 页 12-18.

YuanShanghai, & ZhaoJun. (2020). Nastran-based analysis of static and dynamic characteristics of automobile front subframe. Internal Combustion Engine and Parts, 71-73.

ZhangBaoyu. (2014). The Structural Analysis and Optimization of FSC Car Frame based on FEM. 出处 TechnologyUniversity ofLiaoning, Master's thesis of Liaoning University of Technology (页 44-106). Jinzhou: Publisher of Liaoning University of Technology.

ZhiShuya, XuMutian, & LiJiqu. (2019). Finite element analysis of frame based on ANSYS. Journal of Jinling Institute of Technology, 1-5.

10 Appendix

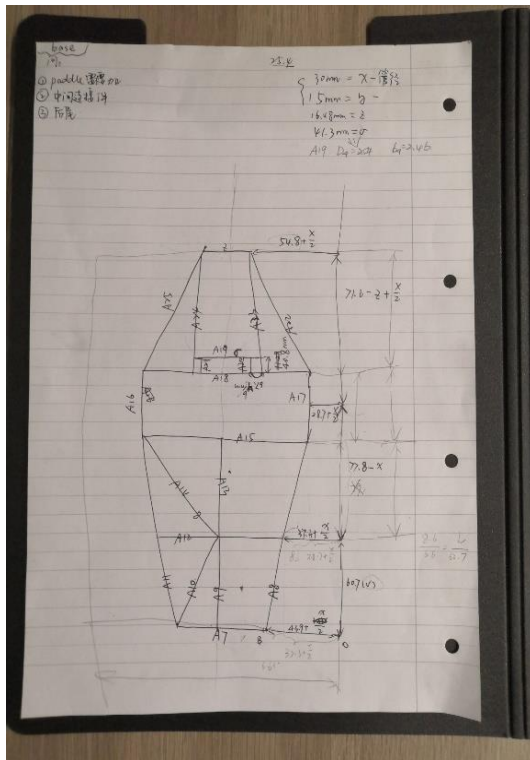


Figure 1-Level A

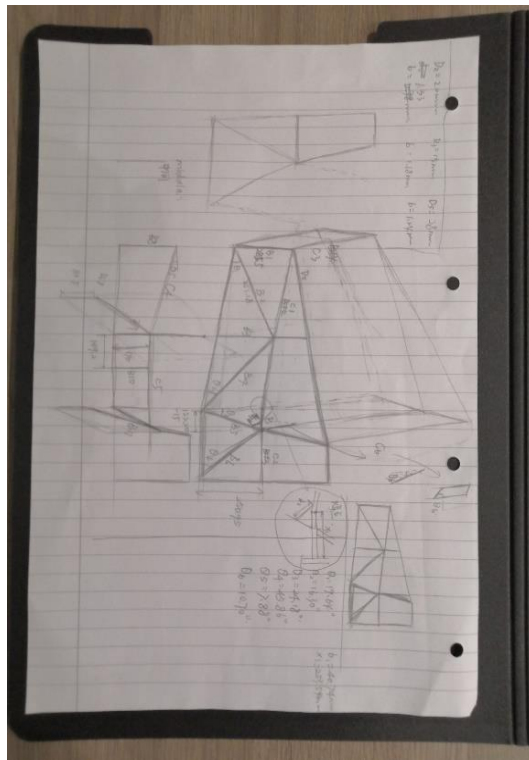


Figure 2-Side View-Left

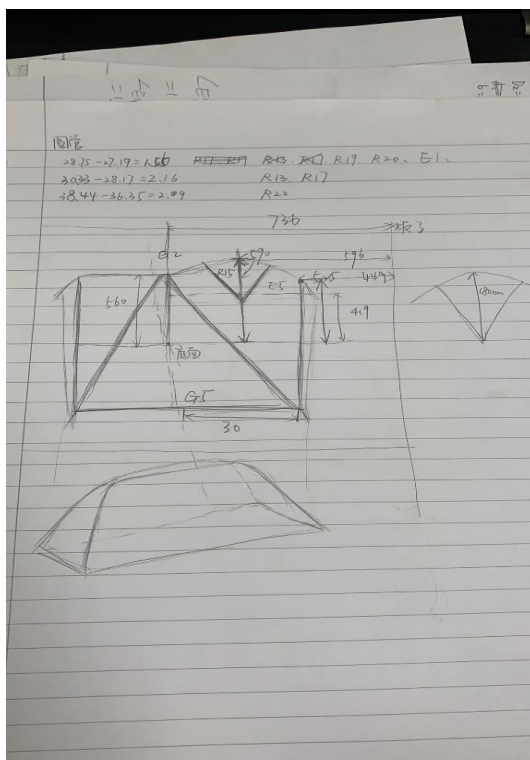


Figure 3-Front View-Middle

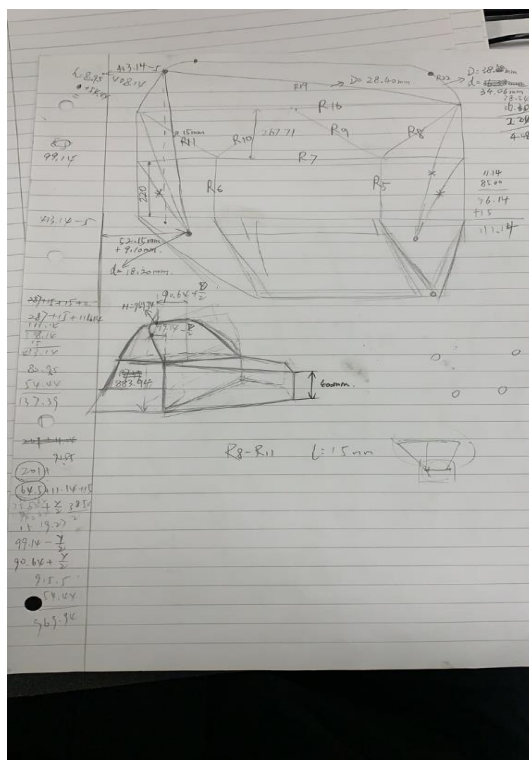


Figure 4-Side View-Back

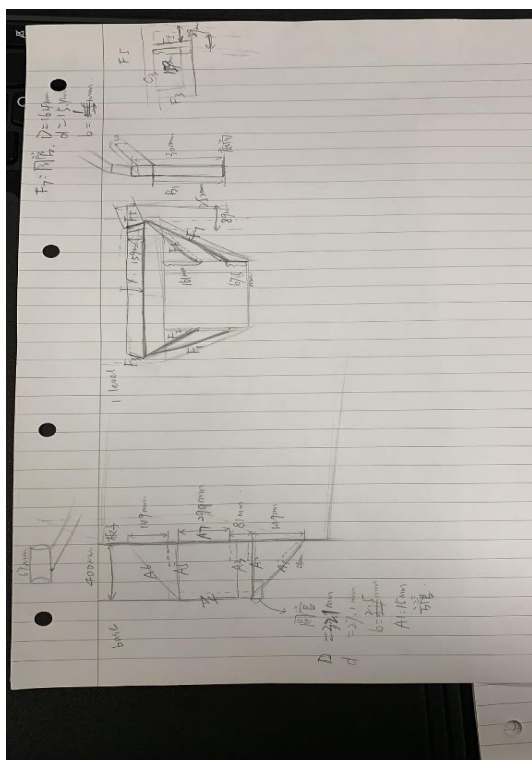




Figure 9-A5&6



Figure 10-A7



Figure 11-A8



Figure 12-A9&10



Figure 13-A11



Figure 14-A12&13&14

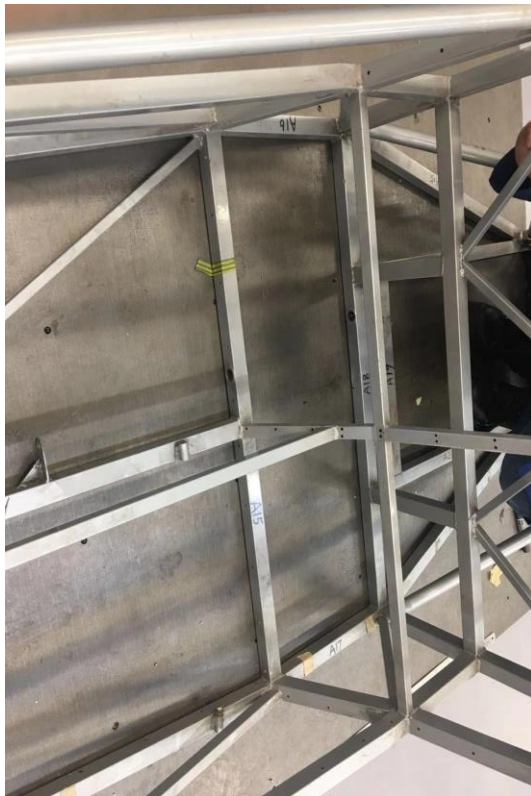


Figure 15-A15&16&17



Figure 16-A18-25



Figure 17-B1

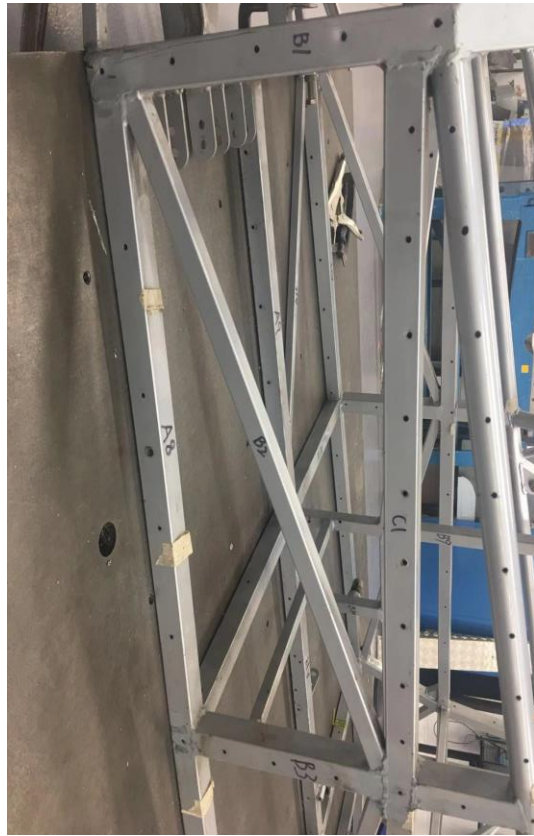


Figure 18-B2



Figure 19-B3



Figure 20-B4



Figure 21-B5



Figure 22-B6



Figure 23-B7



Figure 24-B8



Figure 25-B9



Figure 26-B10



Figure 27-B11



Figure 29-C2

Figure 28-C1



Figure 30-C3



Figure 31-C4



Figure 32-C5

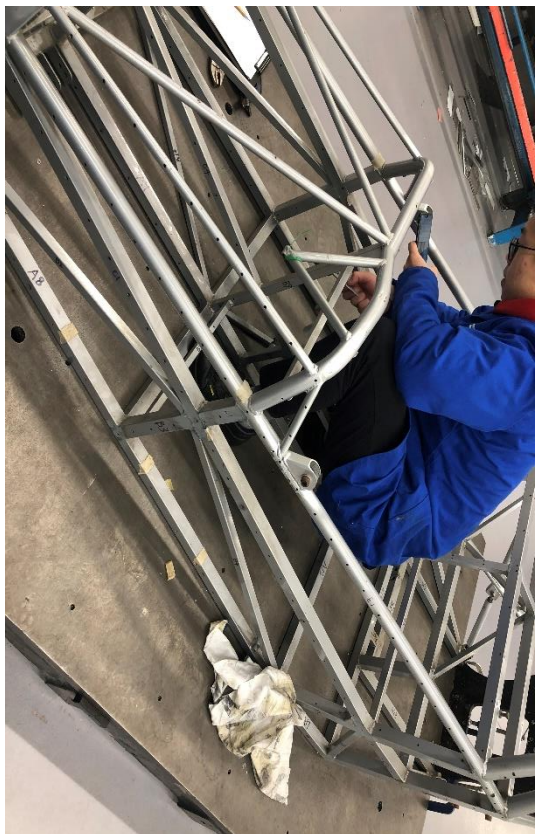


Figure 33-C6



Figure 34-Level D



Figure 35-E1

Figure 36-E2



Figure 37-E3



Figure 38-E4



Figure 39-E5



Figure 40-E6



Figure 41-G1



Figure 42-G2&3&4



Figure 43-G5



Figure 44-R1



Figure 45-R2



Figure 46-R3



Figure 47-R4



Figure 48-R5

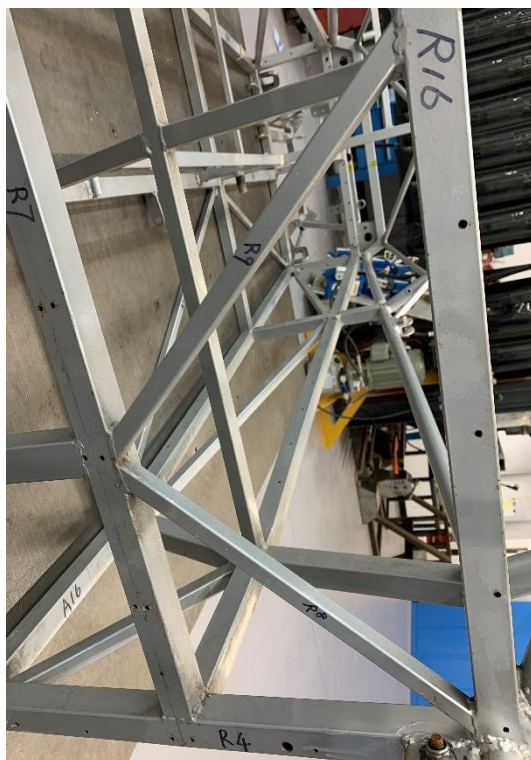
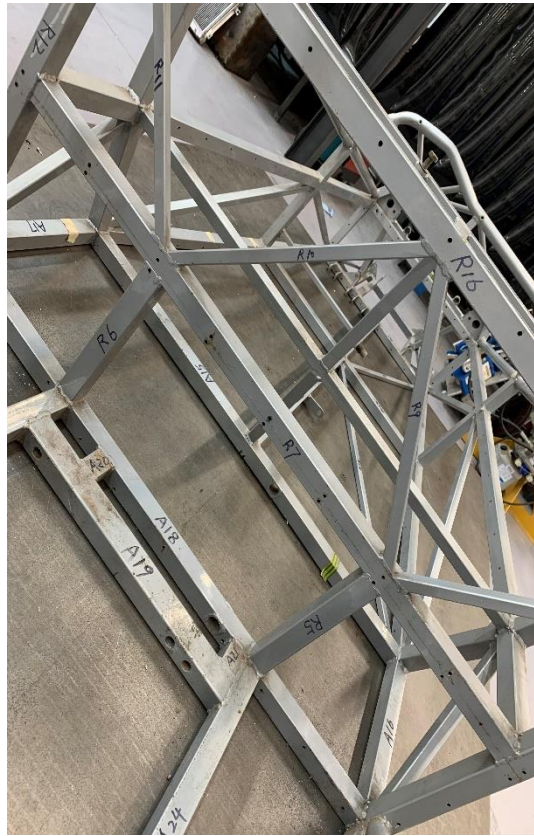




Figure 53-R12



Figure 54-R13&14&15

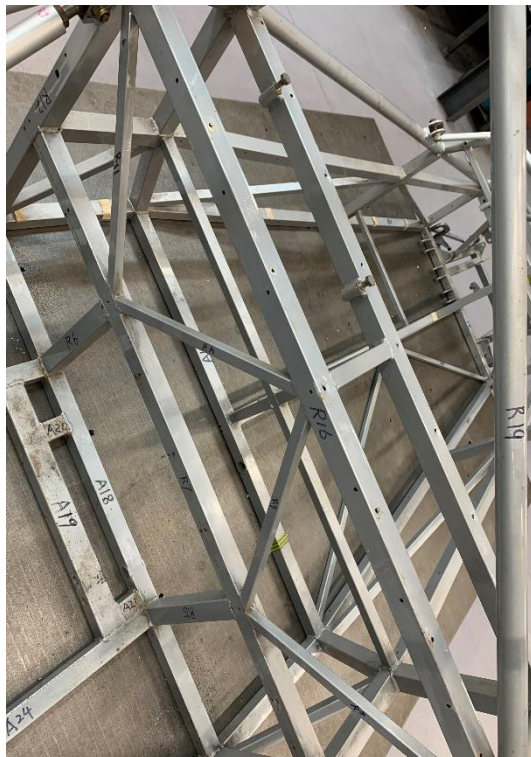


Figure 55-R16



Figure 56-R17



Figure 57-R18



Figure 58-R19



Figure 59-R20



Figure 60-R21



Figure 61-R22

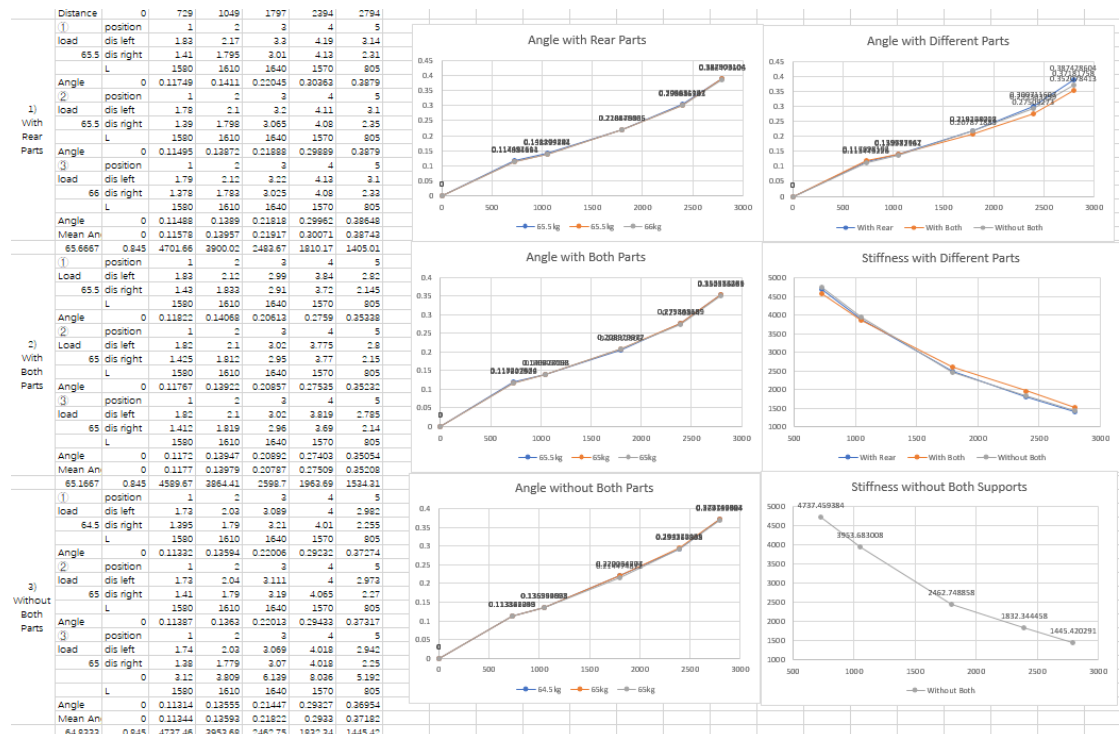


Figure 62-Excel