

University of Wales Trinity Saint David

School of Engineering

Modelling Simulation Analysis and Topology Optimization Design of Vehicle Frame Structure

Xiaotong Jiang

1904747

Level 6

04/05/2020

Major Project(M6X00417)

Abstract

A space frame chassis plays an important role to support many crucial components and systems, such as engine, suspension. Important performance indicators for chassis design include stiffness, natural frequency, durability, and lightweight. This report demonstrates processes of the measurement and modelling by SOLIDWORKS, simulation analysis and ANSYS and lightweight optimization on a vehicle structure.

In the first place, measurements were applied in the solid vehicle structure by group. Appropriate coordination measurement system of the vehicle structure was selected to minimize the measurement error and an overview of model with SOLIDWORKS was built.

Torsional stiffness test was also conducted by the group, whose results could be a suitable tool to validate the accuracy of the Finite Element Analysis (FEA) outcomes.

For the individual works, FEA was firstly conducted to analyse the torsional stiffness of the 3D model. Topology optimization was then applied on a model which was composed of envelop to find the best material distribution of the model in ANSYS 19.1, and an optimum conceptual design was determined which minimizes weight while satisfying all stiffness constraints. Finite Element Analysis (FEA) was performed on the previous model and optimised model in torsional condition. Finally, the mass of the optimised model reduced by 7.8% while the stress and torsional stiffness of it were better than that of initial model.

Keywords

Vehicle structure, chassis, SOLIDWORKS, Finite Element Analysis (FEA), topology optimization, ANSYS, CATIA

Content

1. Introduction	4
2. Literature Background.....	5
3. Group works	7
3.1 Measurement	7
3.2 Modelling description.....	10
3.3 Torsional stiffness test	15
4. FEA	24
4.1 Model Modification.....	24
4.2 Finite Element Model	26
4.3 Data analysis and Comparison	29
5. Topology Optimization.....	32
5.1 Methodology	32
5.2 The modelling and analysing of line bodies of the frame.....	33
5.3 The creation of the envelop	37
5.4 The topology optimization in ANSYS.....	38
5.5 Analysis of results.....	41
5.6 Detailed design of a new frame structure	42
5.7 Comparison of results before and after optimization	44
6. Conclusion.....	46
7. Future Scope of Work.....	47
8. Acknowledgments	47
References	48

1. Introduction

The research object of this report is an existing space frame chassis. Group works cover the measurements, modelling, and torsional stiffness test. The author then did the individual work which includes model modification, Finite Element Analysis and simplified topology optimization.

During the previous FEA process, model interference slowed down the group's progress because the group hadn't used the SOLIDWORKS before. Dimensions then caused the biggest error since the existing chassis has a lot of welding point which the team cannot make sure the measuring points.

The physical test was conducted to measure the torsional stiffness of the chassis which is concerned with the durability performance. This test is an efficient way to validate the FE model which can be proved to carry out the following optimization part, and errors also can be noticed and discussed.

The author conducted the topology optimization of the frame model under the torsional working condition in ANSYS. The author used variable density method to obtain the contours of the topology results. And then the new frame model was designed through the analysis of the results. The comparisons between these two models in the performance and mass of the frame were analysed to find whether the new model realize the aim.

Through this reports the cognition of measurement tools and the ability to build a complex 3D model can be improved during the measuring and modeling process. During this optimization process, the author mastered the relative operation in ANSYS and accumulated a lot of knowledge about topology optimization which will have great benefits for future studying.

2. Literature Background

In the design and development of a complete vehicle, the frame chassis is the basis for the installation of the complete vehicle (He, 2016, pp.2). The structural characteristics of the frame directly affect the performance of the entire vehicle. Hence, the reasonable chassis structure is of significant importance. With the development of various racing events around the world, lightweight design of a racing vehicle chassis which can achieve better performance gradually becomes a hot topic (Yang, 2017).

Topology optimization is carried out to obtain an optimal structure layout (Zhu, 2016). Except the topology optimization, the optimization methods include size and shape optimization.

Topology optimization is nowadays widely used in the automotive industry. The structure of auto parts and vehicle body can be designed using topology optimization methods.

In the literature research, emphasis has been given on chassis torsional stiffness and lightweight optimization. High torsion stiffness is considered beneficial for the vehicle. The lightweight design can be classified into two types. One is to use lightweight materials to design the frame and another is to use optimization techniques. Literature background has been sorted out and described as follows:

Thompson et al. (1998, pp.8) firstly determined the sensitivity of individual structural members to the torsional stiffness of the chassis. The analysing results can be beneficial and efficient for modifying the baseline chassis to improve torsional stiffness while minimizing weight gain and low centre of gravity. Finite element analysis was used to predict the torsional stiffness of the chassis. Finally, by placing structural elements on the base chassis, the torsional stiffness could be increased by more than three times while the weight was increased by only 40 pounds.

Kakria and Singh (2015, pp.11) focused on the modelling, simulation, optimization and fabrication study on the Formula SAE Vehicle Structure. They used ADAMS software to create the multi- body dynamic model of the front and rear part of suspension. They applied Hypermesh to calculate the relative frame stiffness by creating the FE model and then the topology optimization was conducted to realize the optimization in Hyperworks/Optistruct module.

Kang and Zhang (2015, pp.48-52) carried out topology optimization and shape optimization on a Suspension Knuckle under multiple operating conditions. The strength analysis of the suspension knuckle models before and after optimization was carried out, and the quality and strength of the suspension knuckles are compared and analysed. The comparative results showed that the optimized suspension knuckles are lightweight under the premise of satisfying strength performance.

Liu, Zhang and Song et al. (2014, pp.34-37) took the 100-ton power flat car frame as the research object, and a frame model for topological optimization analysis was established. They then used Hyperworks/Optistruct module to apply topology optimization design. Finally, they proposed a frame structure based on the topology structure and the suggestions for optimal design provided a theoretical reference for designing a lighter-weight frame.

Yang, Li, Wang and Zhang(2017, pp. 1-7) carried out Finite Element Analysis to calculate and analyze the stiffness and free vibration modes of the frame chassis. They firstly established Multi-Body Dynamic model of the racing frame chassis by using ADAMS/Car Finally, they

applied topology and size optimization to improve the lightweight of the frame.

Sun (2009, pp. 341-347) carried out topology optimization and size optimization of the front bracket of a car fuel tank bracket, which improved the static strength of the front bracket, reduced the weight by 71.5%, and reduced the cost of the product.

3. Group works

The processing and analysis of the existing frame structure begins with the group works which involves the measurement and torsional stiffness test in workshop, and modeling in SOLIDWORKS.

3.1 Measurement

The measurement part was undertaken by the group since this process requires preparations before measurements, measurement of large quantities of tubes, sketches and records of the chassis, and photographic records of the model. It seems easier to work together to do this measurement. The whole vehicle structure was composed of round and square tubes.

The measurement started with the fixing of the vehicle frame, which requires to fix the front and rear part of structure to the test board with clamping device. The positioning of the frame was completed and shown in Fig 3-1.



Figure 3-1 The fixing part of the vehicle structure

As shown in Fig 3-1, the panels that were connected with tubes of the frame by rivets needed to be disassembled before the group carried out the measurement.



Figure 3-2 The process of removing the panels

As shown in Fig 3-3, Considering the complex and lots of tubes requires measuring, the solid vehicle structure was labelled by different letters and numbers in different blocks. According to the order of subsequent modelling, the group discussed and determined that they labelled the base part with letter A, the middle layer of the front half of the structure with letter B, top part with letter C, rear part with letter R and other small parts with D, E, F and G.



Figure 3-3 Labelling on the tubes

Then the measurement started. Vernier caliper, stainless steel ruler, height Vernier caliper, electronic protractor and other measuring tool were used to measure the length and angle of every steel tubes. Because the diameters of the frame tubes were required in the modelling process, the group drilled steel tubes of different diameters with electric drill tool which is shown in Fig 3-4. Vernier calipers were used to measure both the diameter and thickness of the tubes.



Figure 3-4 Drilling tubes

The group firstly used the board as a reference to measure the external dimensions of the frame. Group members then measured diameters, length and geometric relationship of relevant rectangle and round tubes.

The author drew simple sketches of the structure to facilitate subsequent modelling. Specific data of middle of the structure were taken from measurements, and the data was also recorded on the papers which are shown in Fig 3-5 and Fig 3-6.

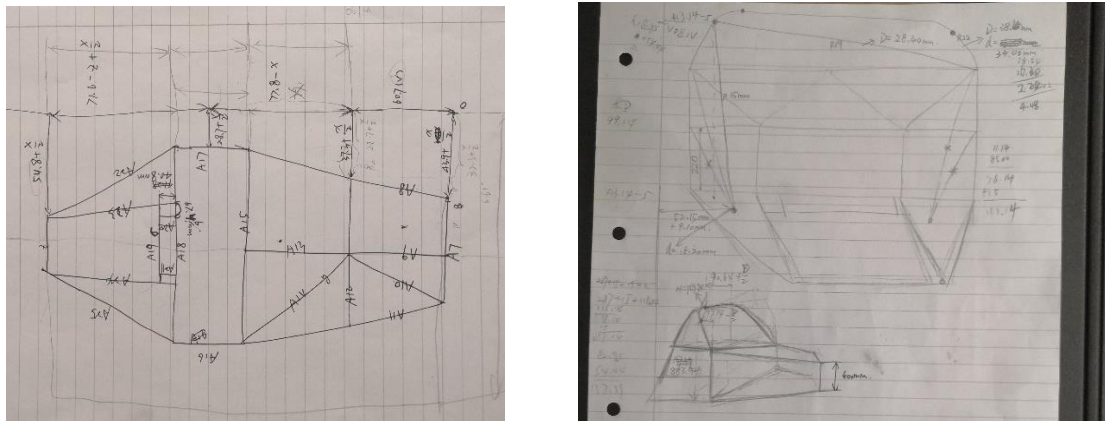


Figure 3-5 Drafts of the base and rear part

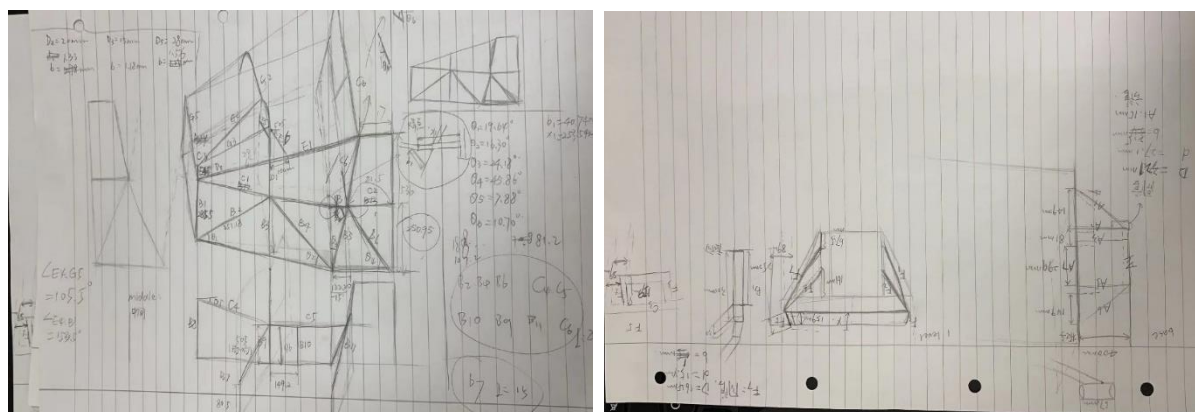


Figure 3-6 Drafts of the middle and front part

During the measuring process, the group faced the difficulty that there was a welding spot which is not the same as the point where the tubes intersect in Fig 3-7. Extra measurements of small points of this part needed to be done.



Figure 3-7 Welding spot

In reality, there were lots of welding points resulting in the approximately measurements

because the boundary of the steel pipe joints could not be determined accurately.

In Fig 3-8, two arc-shaped round tubes in Red were found to have angular deflection in both coordinate systems during measurement. Only measuring the position of several endpoints or turning points relative to the board can determine the tubes' positions and parameters. But the height of the turning point of the round tubes cannot be easily measured by ruler or tape, they then used plumb and wire to measure the height from the board.



Figure 3-8 Two arc-shaped round tubes

The measurements and modelling were performed simultaneously, and it took 3 weeks to accomplish all the structure measuring.

3.2 Modelling description

The modelling part was carried out in SOLIDWORKS, and the modelling was started while measurement was in progress. The baseline part of the modelling was accomplished together by the group, then the rest were built by one or two person who would take some parts to work. In order to facilitate subsequent analysis work, all steel tubes need to be combined into one solid. Hence, when using the 'Extrude Boss' command to create tubes, they needed to choose the 'Merge result' option, which is shown in Fig 3-9, to make sure that the model is one solid.

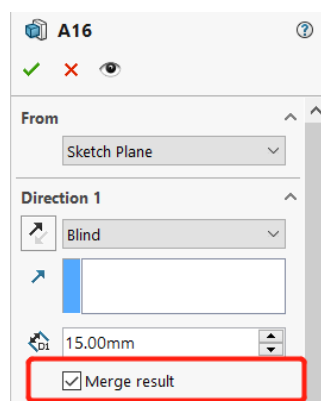


Figure 3-9 The 'Merge result' option

Firstly, a general outline of base of the vehicle, which is shown in Fig 3-10, was completed with SOLIDWORKS to guide the modelling of the first level, according to the previous measurements. The base part is the foundation of the whole model, which could easily affect subsequent modelling of other parts.

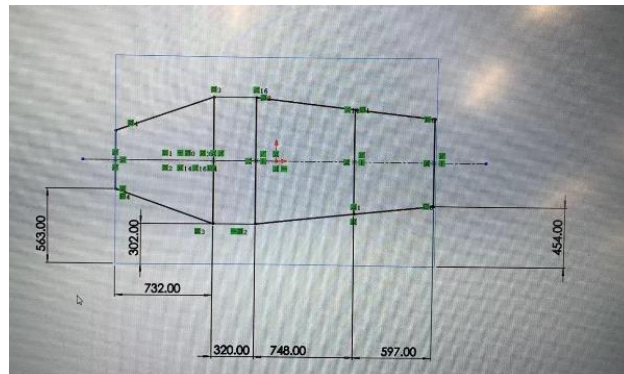


Figure 3-10 Outline of base in SOLIDWORKS

The group worked together to build a 3D model of base part with the commands of ‘Extrude Boss’, and ‘Sweep’ in SOLIDWORKS, which can be shown in Fig 3-11.

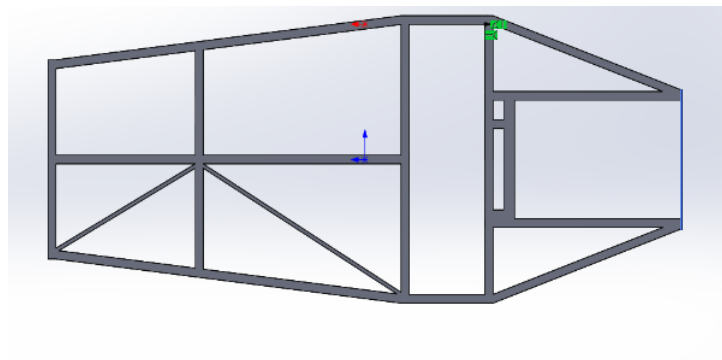


Figure 3-11 3D model of base part

Then the group modelled the first level of the structure except the foremost part where the suspension was installed. During the process, the group noticed the welding spot and separated the two steel tubes at the welding point instead of at the intersection of the tubes. The welding spot advancement could efficiently reduce the stress at the joint of the tubes.

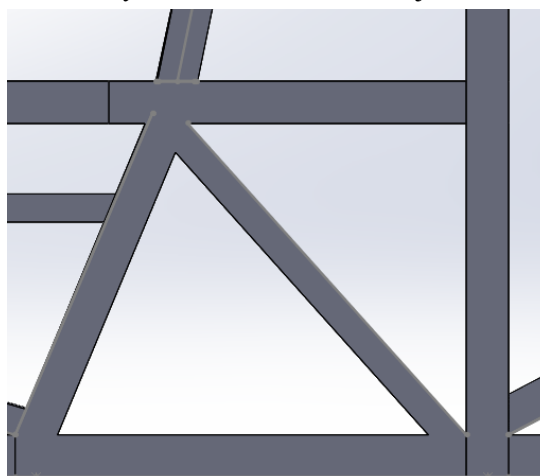


Figure 3-12 Welding spot advancement

The real car was also divided into 3 levels with letter D, E and F according to the height direction. Then the 3D model of the first level of the structure can be seen in Fig 3-13.

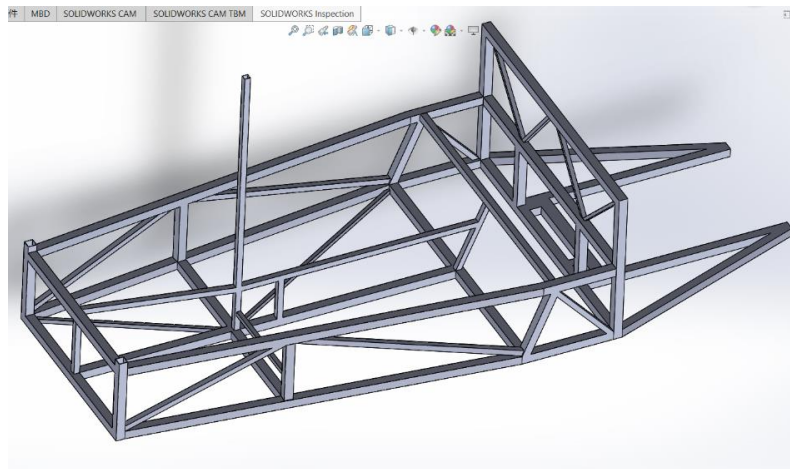


Figure 3-13 3D model of first level of the structure

The remaining tasks for modelling were divided into two parts which were separately taken by group members. The author with other two members set the model of the front part with some small paddles, which is vital for the suspension installation. For this part, there are two tubes which cannot be extruded through the existing datum plane. They then used 3D sketch command to link two end points of the pipe. The tubes were built through the ‘Sweep’ command which needed the sketch as guideline. As shown in Figure 3-14, some small parts were built to indicate the suspension installing position.

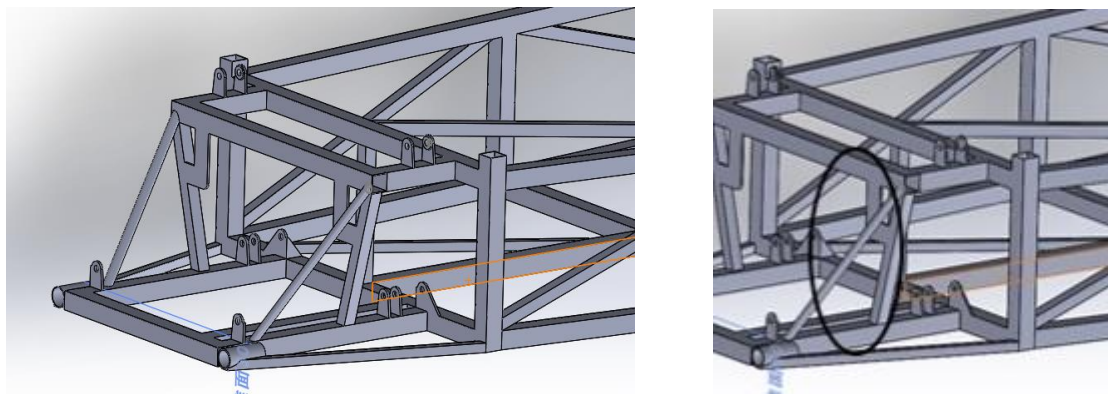


Figure 3-14 3D model of forefront part

The top of the structure was built by other group members. The difficulty of the modelling was two arc-shaped round tubes which are spatial structure. They needed to firstly find the relative points in SOLIDWORKS. They created a datum plane where the rear pipe was located, and then draw the outline of the pipe according to the measuring data. Then they also applied ‘Sweep’ command to build the pipe.

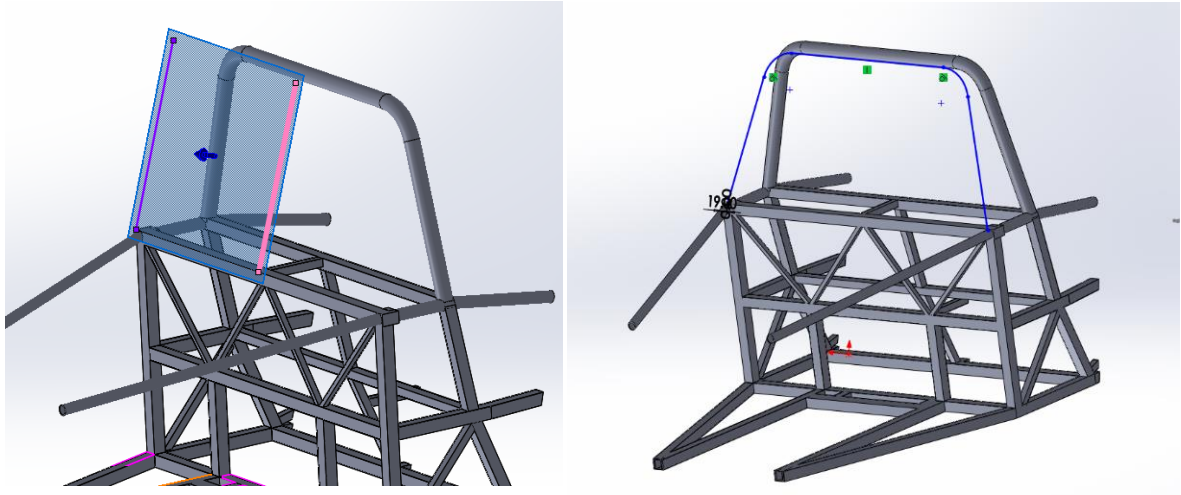


Figure 3-15 Modelling process of arc-shaped round tubes

Also, the same modelling method was applied in irregular tubes of front part which is shown in Fig 3-16.

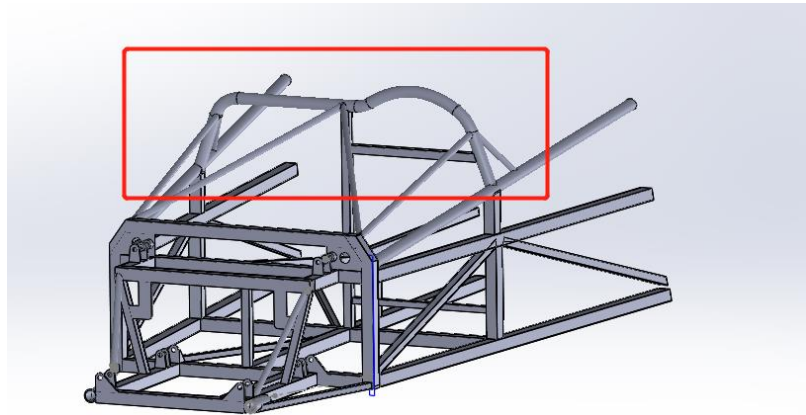


Figure 3-16 Irregular pipe modelling

In fact, there are some special parts of the model which could not be extruded or swept. The part which is shown in red in Fig 3-17 is a rib that serves for supporting small extruded parts. They created a sketch and applied 'Rib' command, according to the length, width and thickness of the rib. Then the 'Extruded Cut' command was applied to dig the middle hole.

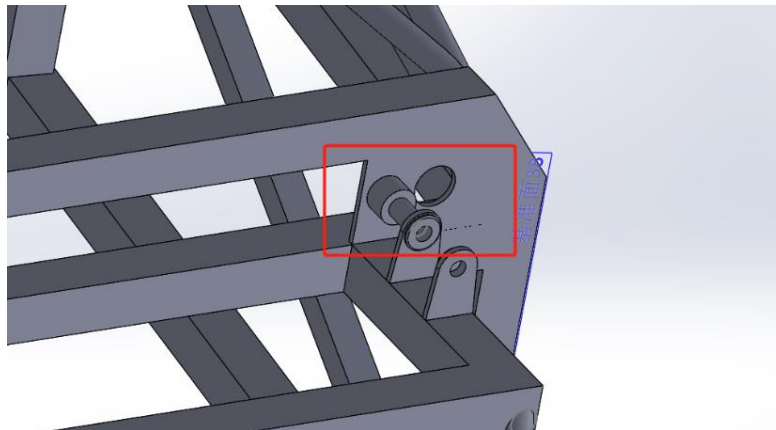


Figure 3-17 The rib structure

The second and third level which were also called B and C level were built through the similar

modelling method.

Finally, the entire structure was finished which can be applied Finite Element Analysis (FEA) to analyse the performance of the frame.

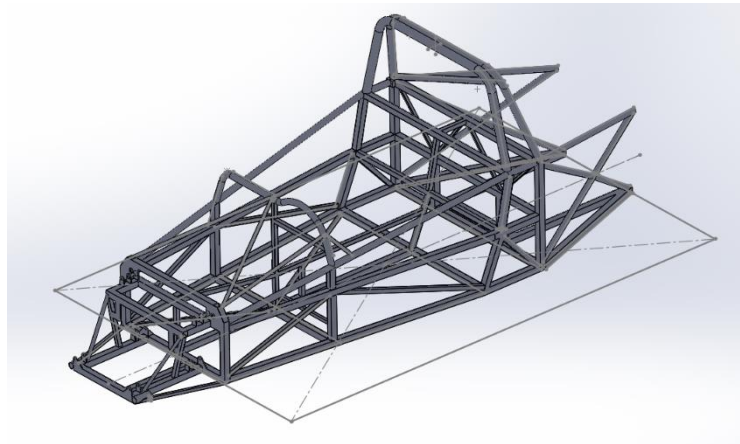


Figure 3-18 3D model of structure

3.3 Torsional stiffness test

3.3.1 Torsional stiffness of chassis

For a vehicle frame structure, there are different loads operating on it which could contribute to the deformation of the structure. Four main loads include longitudinal torsion, vertical bending, lateral bending and horizontal lozenging forces (Heisler, 2002). Torsional stiffness is an important feature in chassis design, which exerts an effect on the ride, comfort and vehicle performance (Riley and George, 2002).

As depicted in Fig 3-16, the torsion model indicates that if the chassis is not sufficiently stiff, the structure where the suspension load was added, to a large extent, could twist along the longitudinal axis in the direction of its length.

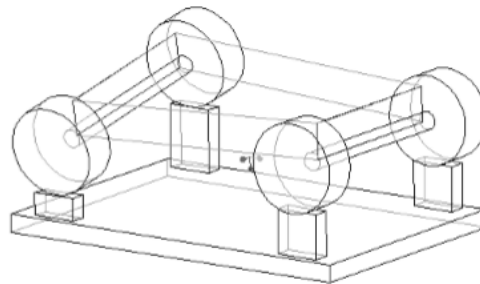


Figure 3-16 Torsion model (Riley and George, 2002)

3.3.2 Methodology

A torsional stiffness test model is depicted in Fig 3-17. The method is to secure and fix the rear bulkhead of the frame, add a torque on the bar where the force can be transferred to the place on the frame which is linked with rocker and suspensions and measure the displacements of each point.

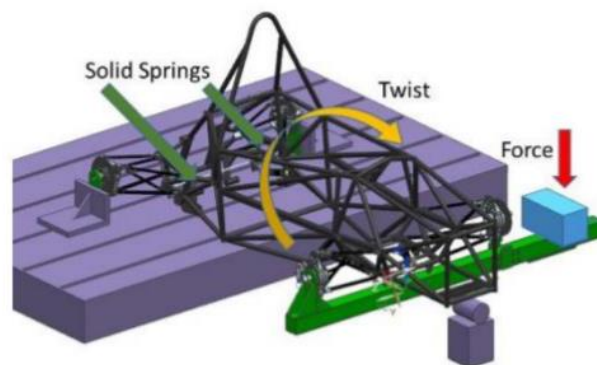


Figure 3-17 Torsional stiffness test model (Shiroma, Over and Wubshet, 2017)

After measuring the displacements of the frame, the calculations of torsional stiffness of the frame can be carried out as follows:

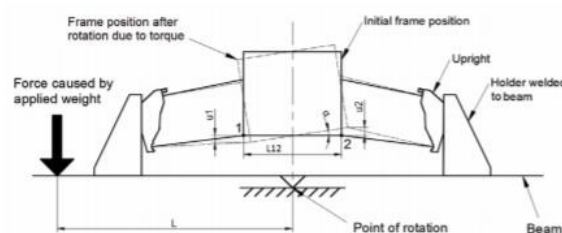


Figure 3-18 Simplified test model (Krzikalla et al., 2019)

The simplified model shows that when force is added, the frame rotates around the longitudinal axis of the frame. And the mathematical model can be created to calculate the rotating angle of each horizontal direction.

In fact, there would be some lateral displacement because the vertical force could not be guaranteed, but the lateral displacement could be neglected because it seems too small when compared to the pitch of two points.

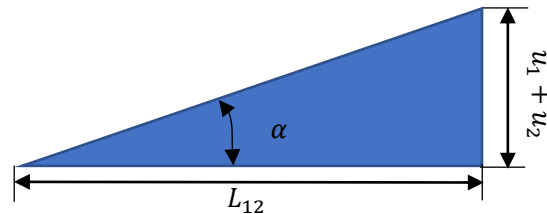


Figure 3-19 Mathematical model

Through the triangle, the angle and torsional stiffness can be calculated.

$$\alpha = \arctan\left(\frac{u_1 + u_2}{L_{12}}\right) \text{deg}$$

$$K_T = \frac{T}{\alpha} = \frac{F * L}{\alpha} \text{ N * m/deg}$$

Where α is the angular deflection, u_1 and u_2 are respectively distances from the endpoint of both sides to the longitudinal Axis, L_{12} is the distance between two points, K_T is the torsional stiffness, T is applied torque, L is the moment arm.

3.3.3 Apparatus and instrumentation for the test

The torsional stiffness test was carried out on a test stand whose aim was to fix and give the torque to the frame to measure the deflection caused by the torsion. The test stand covered front fixture to apply load on, the jack, pivot point, anchor, the beam and the rear part fixture.

(1) The rear fixture

As shown in the Fig 3-20, the rear bulkhead of the frame was fixed to the ground by the beam fixture that constrained the rotating and yawing motion of the frame. The fixture was fixed to the frame and ground by bolts which could reduce the translation to the ground.



Figure 3-20 A photograph of rear fixture

(2) The front fixture, pivot point and anchor

The front part of fixture and torque is depicted in Fig 3-21. The fixture of the front part was composed of both sides of the rod fixture and a long bar. The long bar was placed underneath the tubes which should be applied force on, and two ends of the rod are respectively connected to the small boss parts of the frame and the long bar by the bolts. The left and right rod fixtures can be connected to each other through the long bar, however, the link between frame, rod and bar cannot fix to the ground. The rods could provide the locked suspension instead of the spring dampers whose stiffness may affect the torsional stiffness of the frame.

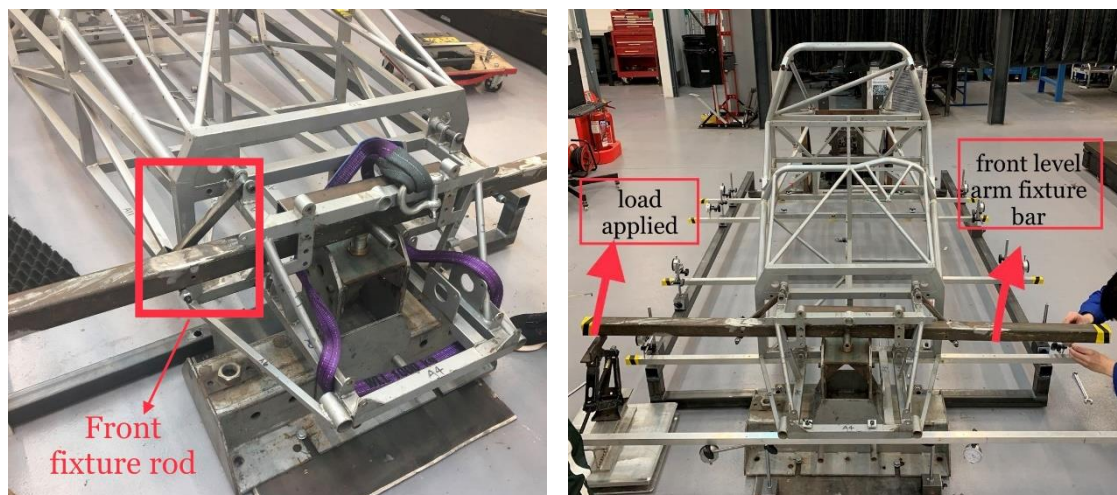


Figure 3-21 A photograph of front fixture

From the Fig 3-22, the pivot point which is shown in red is a shoulder bolt that gave reasonable clearance between the bar and pivot to prevent high stress and ensure that the bar rotates smoothly. The pivot was fixed to the ground through an anchor and four bolts. Hence, the front part of the frame was fixed and force could be added through the rigid bar.

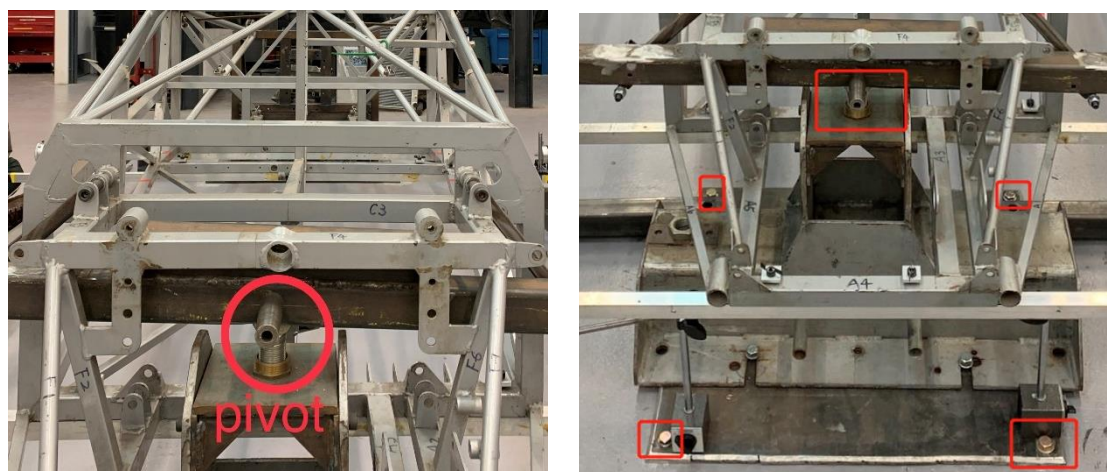


Figure 3-22 The pivot point and anchor

(3) The beam

The beams were placed and welded together around the frame in Fig 3-23. The front and rear beams were fixed with the anchor and rear fixtures while the left and right beams were designed to rise above the ground but below the height of the bottom of the frame. The reason for setting the height of beams was to provide space for subsequent measurements of the deflection.

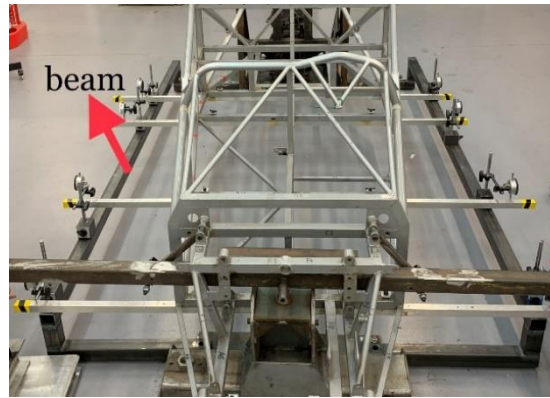


Figure 3-23 Beams around the frame

(4) The jack

The jack was used to apply the upright force on the left side of the frame through the long rigid bar in Fig 3-24. The jack's function is to unilaterally apply force to the frame with a torque.

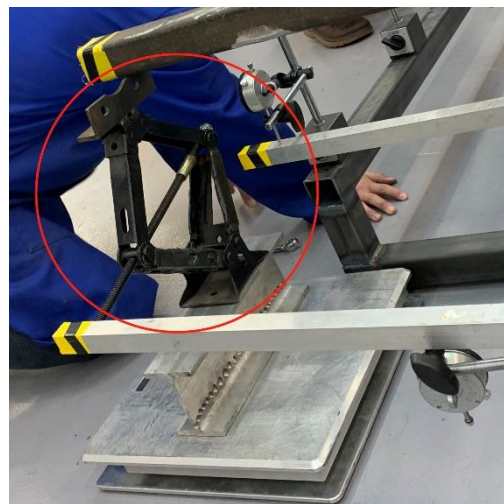


Figure 3-24 The jack

(5) Measuring devices

For this test, there are two data to be measured. One is the force applied on the bar by the jack. From Fig 3-24 the suspension scale was placed underneath the jack to measure the load. The group adjusted the load via tighten and soften the jack. One software called 'Intercomp Race Weigh' can measure the total weight of the suspension scale. When using this software to obtain the load, the zero operation need to be performed by applying the same force in advance. A photograph of the software interface is shown in Fig 3-25.

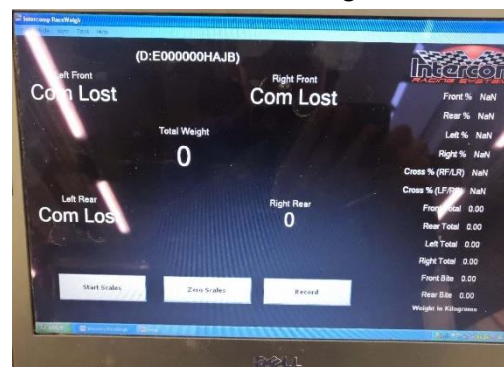


Figure 3-25 'Intercomp Race Weigh' software

Another data to be needed is the deflection of several points of the frame to obtain the angular deflection of the frame. The group determined 10 points to measure and every two points were on the same line. As depicted in Fig 3-26, the dial indicators were used to measure the displacements of the points on the long lightweight tubes which were fixed to the base of the frame and parallel to line of each two points on the frame. The tubes were closed to the base by the bolts and could be considered as the zero position of the displacement. Using the up and down displacements of the points on the tubes to obtain the angle could result in the error because of the mass and shaking of the tube itself, however, these could be neglected as its light weight.

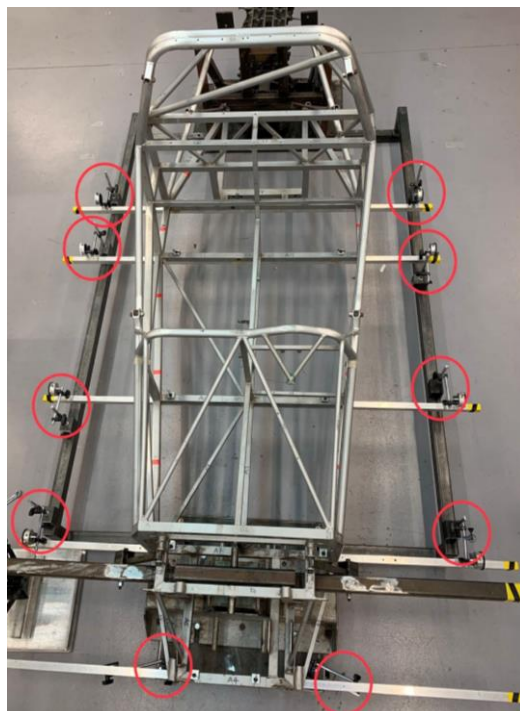


Figure 3-26 Ten dial indicators and five tubes

3.3.4 Test

To set up the test, all the fixtures were firstly placed to constrain the movement of the frame. The group connected these five tubes to the base of the frame, placed the dial indicators between beam and tubes and then zero all the dial indicators. The last part before carrying out the test is to conduct zero option on the load. One group member adjusted the jack and another person observe the number on the software interface. When the weight of 25kg was displayed on the software screen, they stopped adjusting the jack and performed zero operation.

Each group of experiments must carry out these two zero operations before they can start. The measurements were carried out three times in every condition. For different research directions of group members, three conditions where some of the tubes may be disassembled from the frame were carried out. From the Fig 3-27, the second set of the test were conducted where pipe 1 and pipe 2 were both removed while the pipe 2 was removed in the third set of the test which was useful for the author.



Figure 3-27 Tubes removed in different sets of the test

The aim of the test is to obtain the torsional stiffness of the frame, through the above calculation methodology, the group needed to measure the pitches of every two points and the force arm which is shown in Fig 3-28.

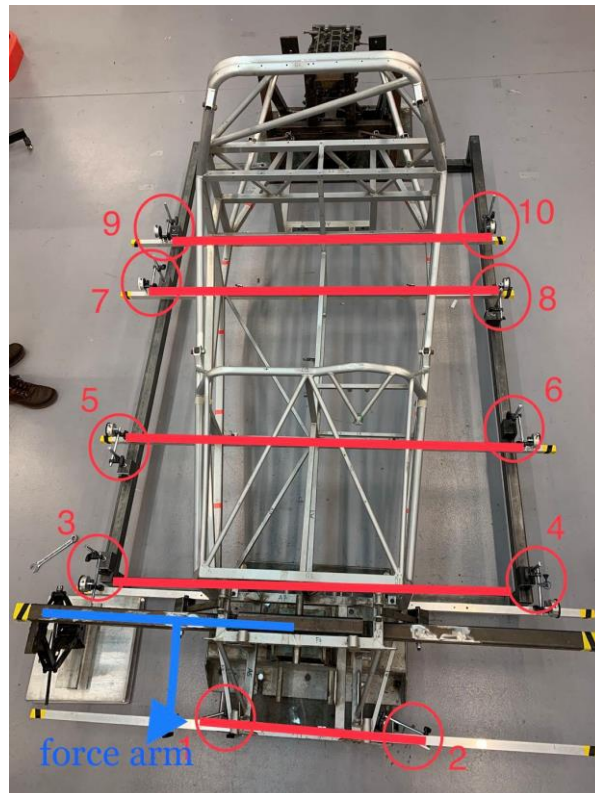


Figure 3-28 The length needed to be measured

And the distance measuring data was sorted out and shown as follows.

Table 3-1 The measuring data of distances

Force arm/mm	845		
Tubes	Distance from the left point to the longitudinal axis/mm	Distance from the right point to the longitudinal axis/mm	Pitch/mm
Tube 1 (point 1,2)	445	360	805
Tube 2 (point 3,4)	770	800	1570
Tube 3 (point 5,6)	810	830	1640
Tube 4 (point 7,8)	825	785	1610
Tube 5 (point 9,10)	815	765	1580

After the measurement of these distances., they conducted the test. Measurements were carried out three times in every condition with the similar load applied. They applied approximately 65kg of force to the frame each time. After applying the force, group members needed to read and record the dial indicators. They conducted a total of nine experiments under three conditions which were recorded in the paper and afterward organized into tables in EXCEL for analysis.

3.3.5 Data Presentation

All the testing data was recorded and sorted in the EXCEL which is shown in Fig3-29, and the angular deflections against the tube position under three conditions were also plotted below.

			L	805	1570	1640	1610	1580
			position	1	2	3	4	5
1	1	65.5kg	dis left	3.14	4.19	3.3	2.17	1.83
			dis right	2.31	4.13	3.01	1.795	1.41
			Angle	0.387903	0.30363	0.22045	0.1411	0.11749
	2	65.5kg	dis left	3.1	4.11	3.2	2.1	1.78
			dis right	2.35	4.08	3.065	1.798	1.39
			Angle	0.387903	0.29889	0.21888	0.13872	0.11495
	3	66kg	dis left	3.1	4.13	3.22	2.12	1.79
			dis right	2.33	4.08	3.025	1.783	1.378
			Angle	0.38648	0.29962	0.21818	0.1389	0.11488
	Mean Angle		0.387429	0.30071	0.21917	0.13957	0.11578	
2	1	65.5kg	dis left	2.82	3.84	2.99	2.12	1.83
			dis right	2.145	3.72	2.91	1.833	1.43
			Angle	0.353379	0.27589	0.20612	0.14068	0.11822
	2	65.5kg	dis left	2.8	3.775	3.02	2.1	1.82
			dis right	2.15	3.77	2.95	1.812	1.425
			Angle	0.352311	0.27535	0.20857	0.13922	0.11767
	3	66kg	dis left	2.785	3.819	3.02	2.1	1.82
			dis right	2.14	3.69	2.96	1.819	1.412
			Angle	0.350532	0.27403	0.20892	0.13947	0.1172
	Mean Angle		0.352074	0.27509	0.20787	0.13979	0.1177	
3	1	64.5kg	left	2.982	4	3.089	2.03	1.73
			right	2.255	4.01	3.21	1.79	1.395
			Angle	0.372738	0.29232	0.22006	0.13594	0.11332
	2	65kg	left	2.973	4	3.111	2.04	1.73
			right	2.27	4.065	3.19	1.79	1.41
			Angle	0.373165	0.29432	0.22013	0.1363	0.11387
	3	65kg	left	2.942	4.018	3.069	2.03	1.74
			right	2.25	4.018	3.07	1.779	1.38
			Angle	0.369535	0.29326	0.21447	0.13555	0.11314
	Mean Angle		0.371812	0.2933	0.21822	0.13593	0.11344	

Figure 3-29 A screenshot of data processing

The line charts in these three conditions reflect the same trend that the angular deflection at the front is the largest, and as the position moves backward, the deflection becomes smaller. The

rate of reduction becomes gradually flat. The comparative line charts indicate that the deflection at the forefront part under the first condition is bigger than that at other two conditions.

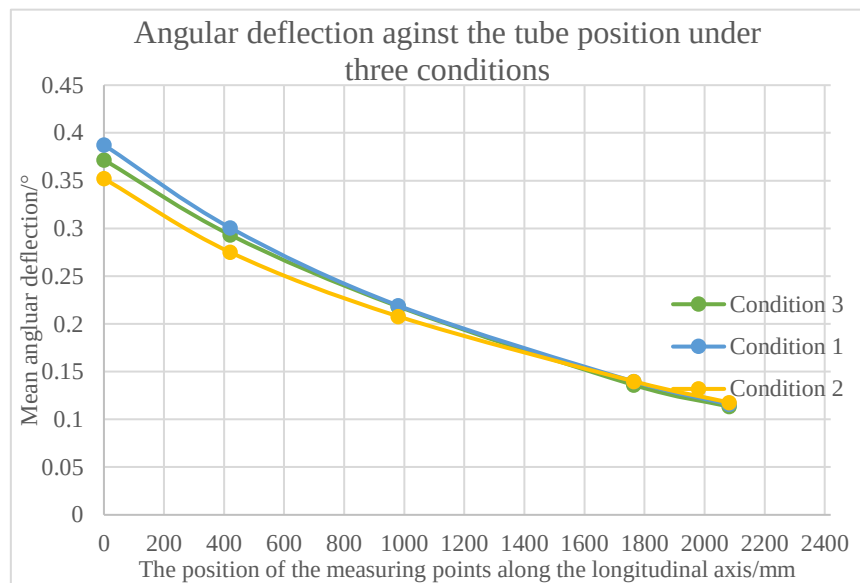


Figure 3-30 Angular deflection vs tube position

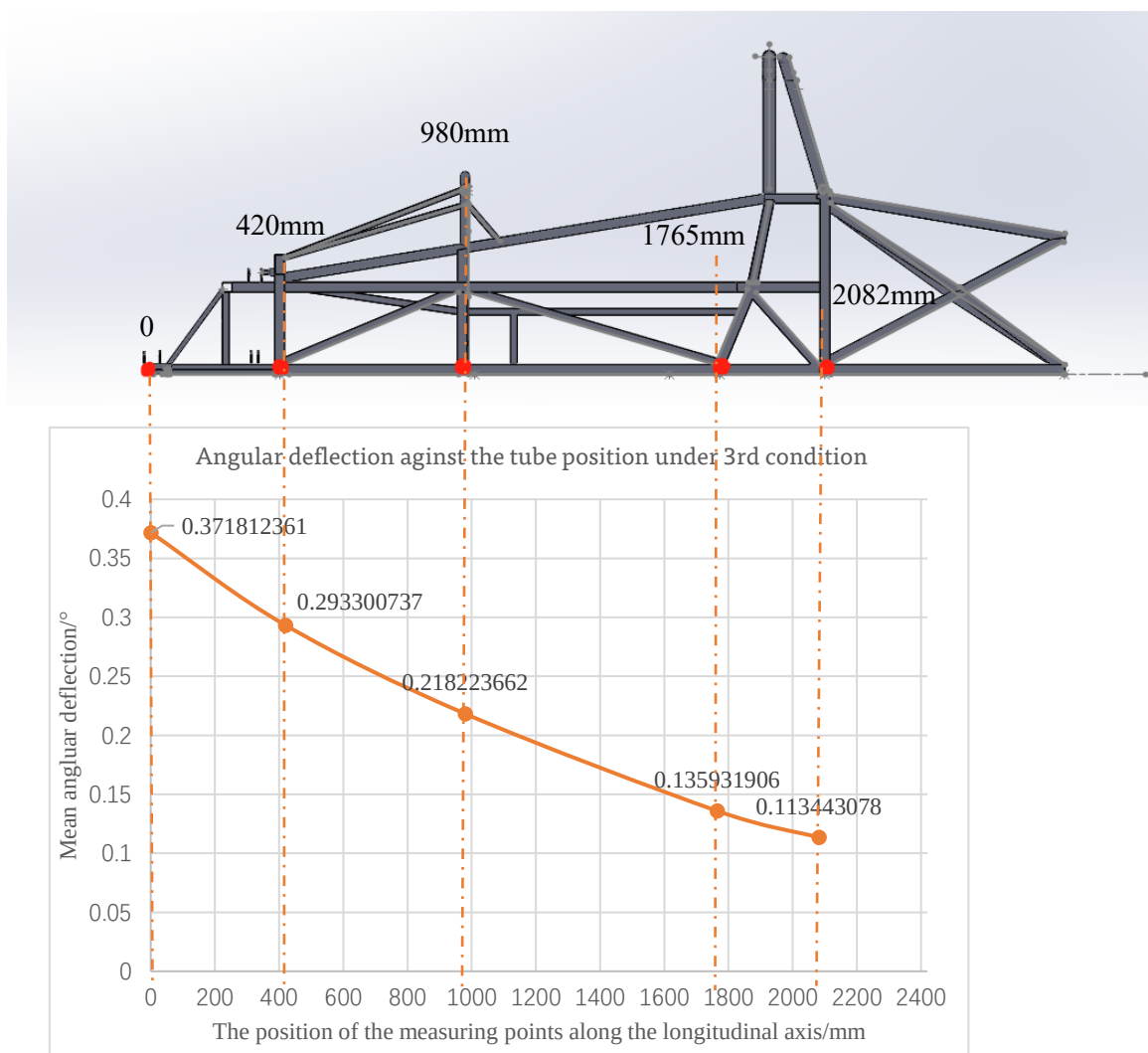


Figure 3-31 Angular deflection under 3rd condition along the frame

3.4 Evaluation of group work

For the measurement part, the author did the sketches and recorded the data of all the frame structures for modelling. Also, the author took photos of structure from different perspectives.

For the modelling part, the author and another group member modelled the front part of the frame. And all the group members together created most of the model.

For the torsional stiffness test part, everyone needed to observe two dial indicators to record the data. During the group work process, the cooperation with each other accelerated the process because everyone would be willing to proposal the excellent idea and share with each other.

4. FEA

The results obtained from the torsional stiffness test could validate the outcomes of FE model which was put in the same loading condition. The comparative outcomes from FE model and physical test could directly determine whether the FE model could be used in the subsequent analysing or optimising process to eliminate the time-consuming physical test. The measuring, initial modelling and torsional stiffness test were undertaken by the group, however, the individual works started in the following steps.

The FE model was conducted in simulation module in SOLIDWORKS. This FEA was the liner static analysis whose task was to set the operating conditions for the 3D model, choose the element size to mesh and then run the result.

4.1 Model Modification

4.1.1 The conversion of force

Because the 3D model was the independent frame without the beams and fixtures, the load applied to the frame directly acted on the front part instead of adding on the arm bar. The force was not the same as that in the test. Calculation needed to be done to convert the force.

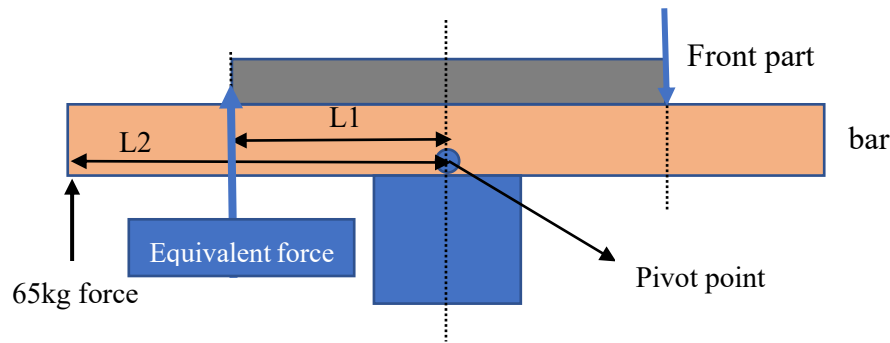


Figure 4-1 A simplified model to calculate the load

From Fig 4-1, the relationship between these two loads can be easily obtained as follows:

$$\text{Torsional Moment} = \text{Equivalent force} * L_1 * 2 = \text{Load} * L_2$$

$$\text{Equivalent force} = \frac{\text{Force on the bar} * g * L_2}{2 * L_1}$$

Where L_1 is the distance from force point to pivot point, L_2 is the force arm and g is gravity acceleration, which equals $9.81m/s^2$.

$$\text{Equivalent force} = \frac{65 * 9.8 * 845}{508} = 1027N$$

Hence, the value of force applied on the front part of the frame was 1027N.

4.1.2 Fixing and loading process

When conducting the FE model, the author fixed the rear part of the frame according to the previous torsion experiment. In the light of the model without the fixture, the author selected the rear 4 faces to fix and applied two forces in opposite directions to give the frame torque and these setting is shown in the following Fig 4-2.

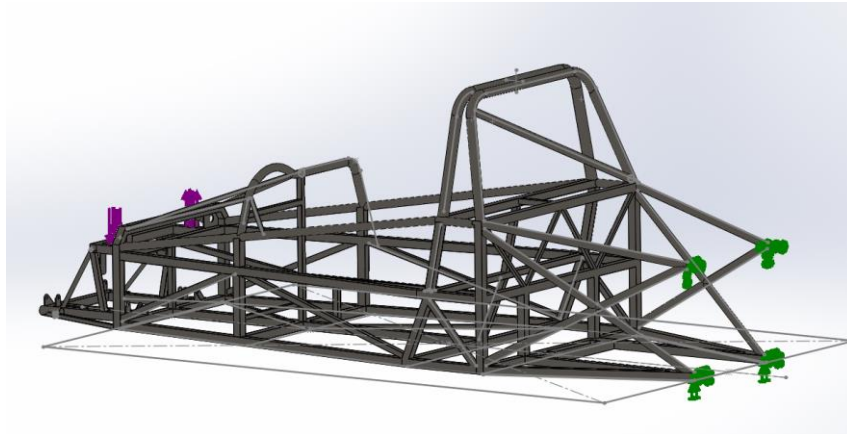


Figure 4-2 The settings of operating conditions in SOLIDWORKS

4.1.3 Problems and modifications of the model

When the author tried to conduct the mesh module, the software showed that many faces and lines cannot be meshed. The author turned back to the modelling module to check and find the problems where the mesh could not build. The Fig 4-3 below depicted the details of the model.

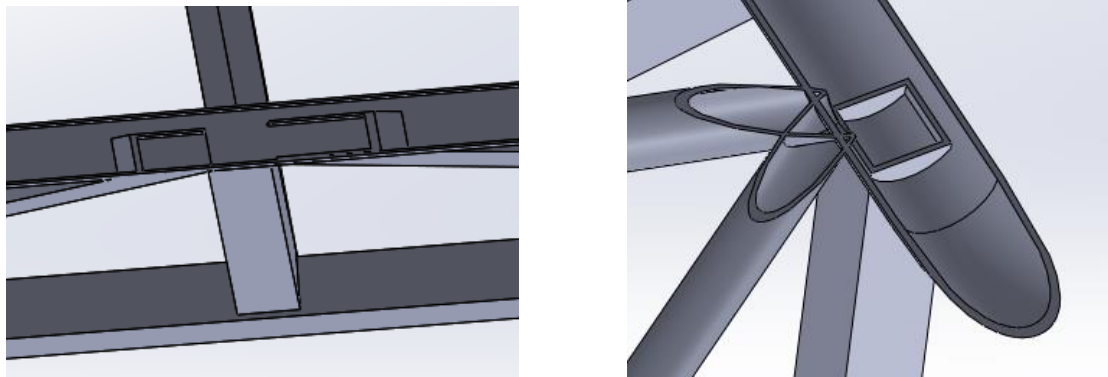


Figure 4-3 The complex joints in the initial model

Since the group members had not studied SOLIDWORKS before and were not very familiar with the operation of this software during the initial modelling process. And these complex joints were due to the 'Sweep' command. All the modelling of the oblique tubes used for reinforcement in rectangular tubes were from the 'Sweep' command that could not stop stretching at the end of the guideline instead of automatically fitting it when it encounters a face. Hence, these tubes sketched into the joints and made them complex.

The author then modified the model with 'Extrude' command that needed to create a new datum according to the 3d sketch instead of Sweep, the 3d sketch was created along the extruding direction. The method of extruding boss could select 'up to the surface' option to avoid the complex joints. All the joints were modified and only two places where there were arc-shaped tubes used 'Sweep' command.

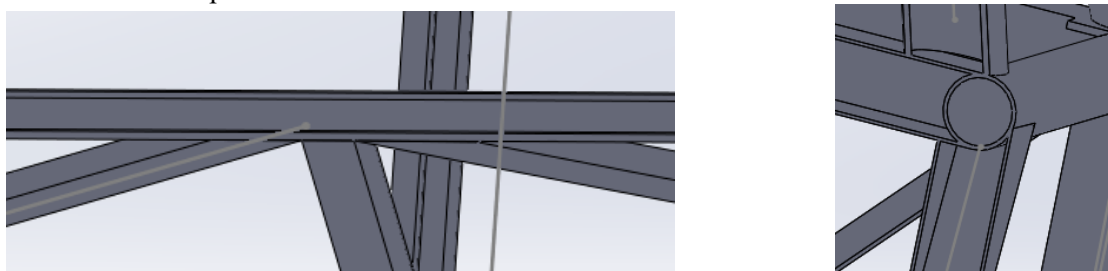


Figure 4-4 The joints of modified model

The modified model that is shown below was created into one solid and could be used in FE model.

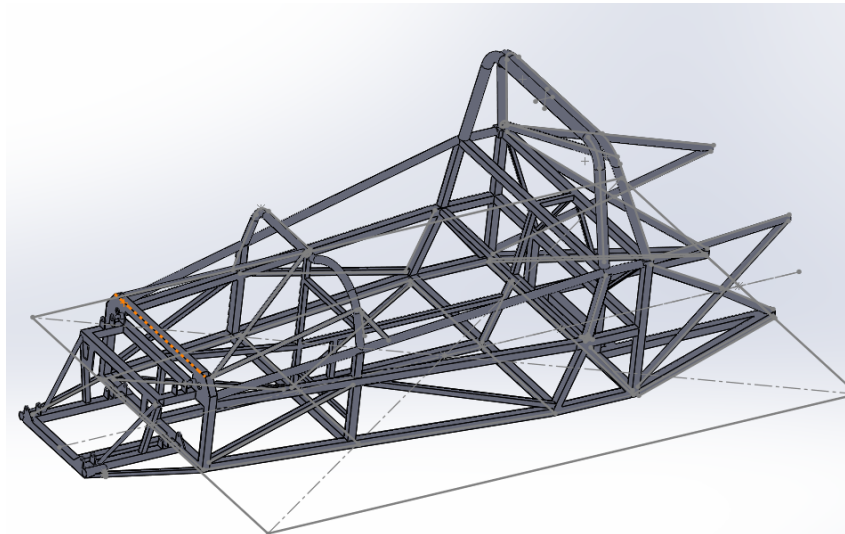


Figure 4-5 The modified model

4.2 Finite Element Model

After the modification, the FEA was directly carried out under the same settings. In fact, the frame was made of two materials. The round tubes were made of 4130 steel while the square tubes were made of ERW. However, the model was created in one solid with which only one material could be applied. The author selected 4130 steel annealed at 865°C as the material of the frame.

The author chose the blended curvature-based mesh which could create mesh of higher quality as shown in Fig 4-6. The maximum element size was set to 3mm while the minimum one was 0.6mm. In fact, the accuracy of the FE outcomes, to a large extent, almost depend on the element size of the mesh if the model does not have any other problems. The accuracy can become better as the size become smaller until the convergence happens. However, the element size could not be smaller due to the limited memory of the laptop to run the mesh.

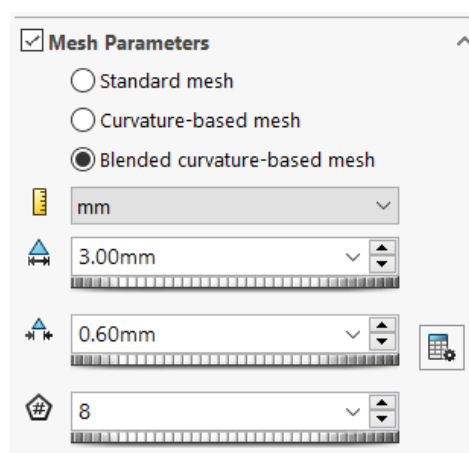


Figure 4-6 The setting of the mesh

The Fig 4-7 and Fig 4-8 shows the tetra-mesh of the frame and the aspect ratio which indicates the tetra-mesh quality. If the ratio is too big, it means that the excessive gap between the long and short sides of the triangle may lead to a reduction in calculation accuracy. For this tetra-mesh, the biggest ratio is 49 which could be a little big but also acceptable. The places where

bigger aspect ratio appears were at the small edges or somewhere with small thickness which is shown in Fig 4-8. But it did not matter because these big ratios did not appear at the important measuring and testing points.

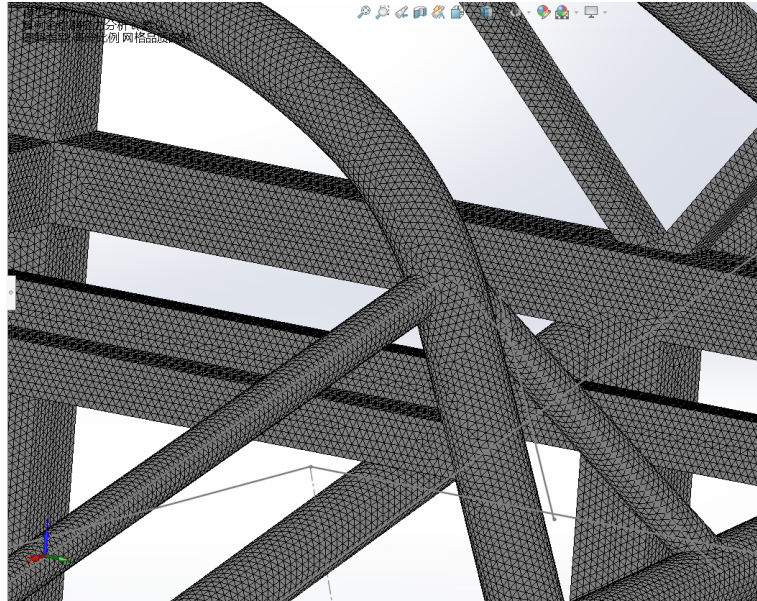


Figure 4-7 Tetra-mesh of the frame

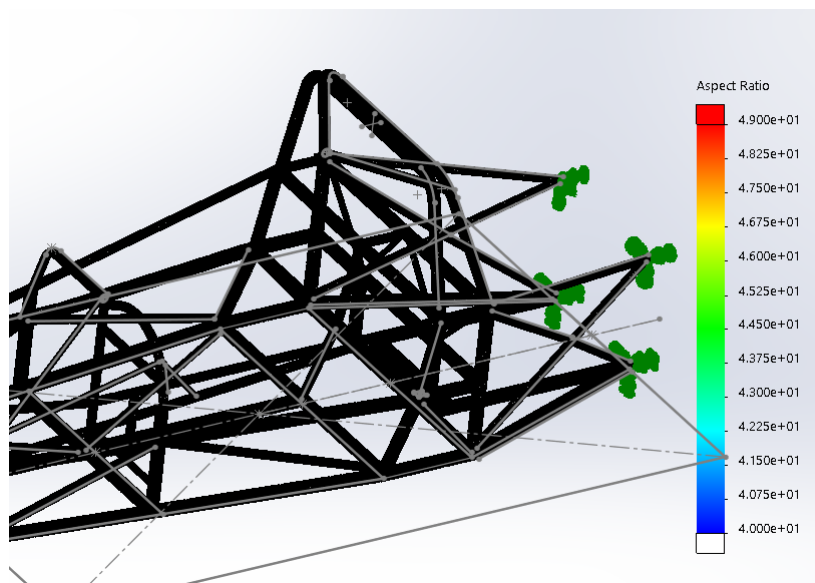


Figure 4-8 The aspect ratio of the mesh

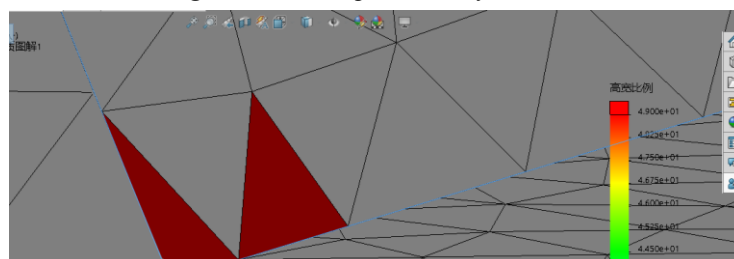


Figure 4-9 The biggest aspect ratio

When finishing the mesh, FE model was successfully built, and the results could be conducted. It took the laptop about 20 minutes to run this study. The resulting diagram is shown below. From Fig 4-10 and Fig 4-11, the stresses of the frame were all below the yield strength of the

4130 steel, and the edges and joints of tubes showed higher stresses.

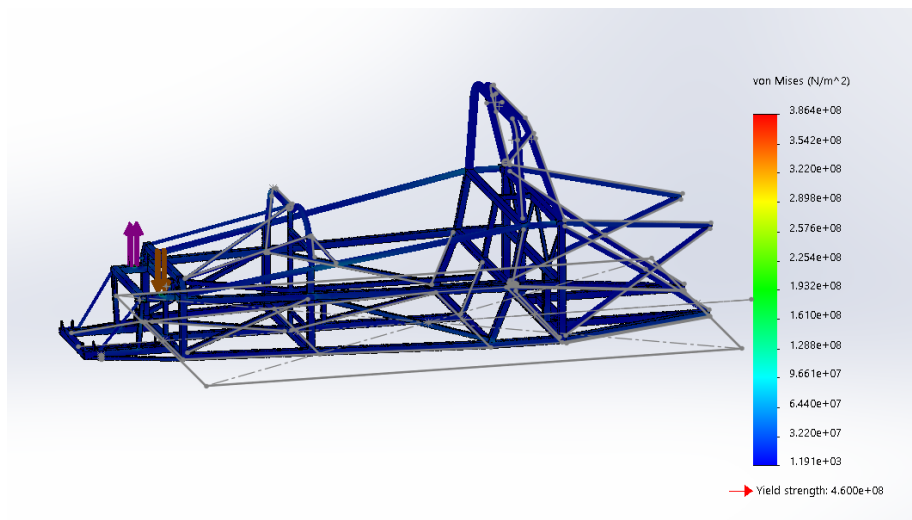


Figure 4-10 The diagram of the stresses

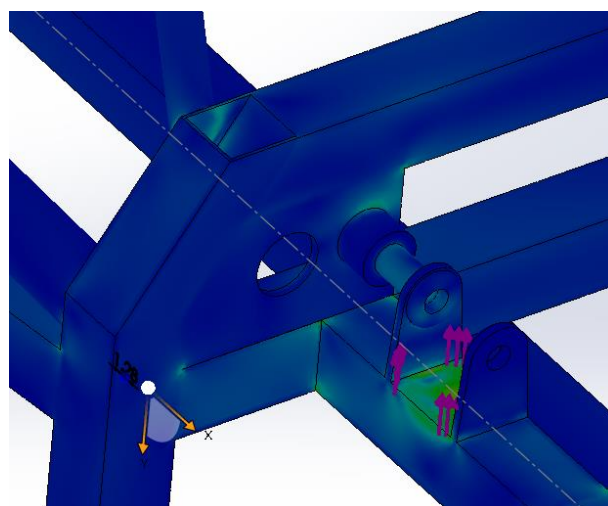


Figure 4-11 The local stress diagram of the frame

For the displacements of the frame, the author selected to observe the displacements along the Z-Axle in Fig 4-12. Since the displacements along the Z-Axle had the positive and negative value, the changes in displacements cannot be clearly observed in this diagram.

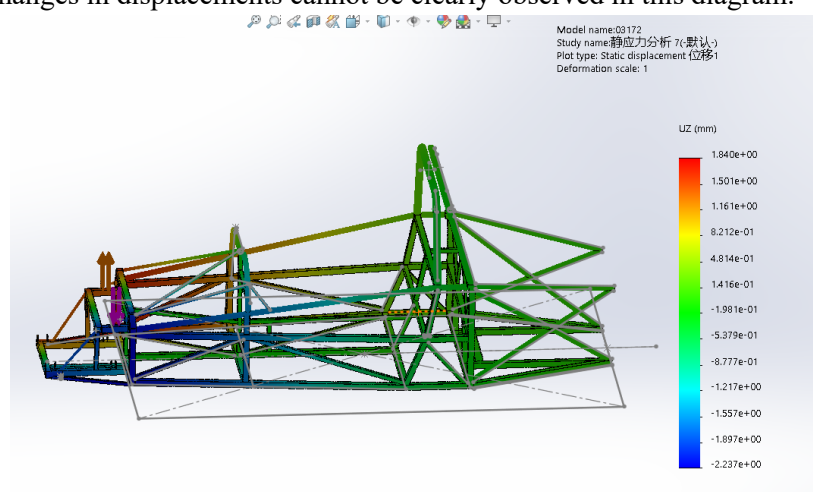


Figure 4-11 Displacement along Z-Axle

To obtain the similar displacements of these testing points, the author used probe tool to get the

data at these points in Fig 4-12. The author probed 10 points to obtain the displacements of every point according the position in the torsional stiffness test.

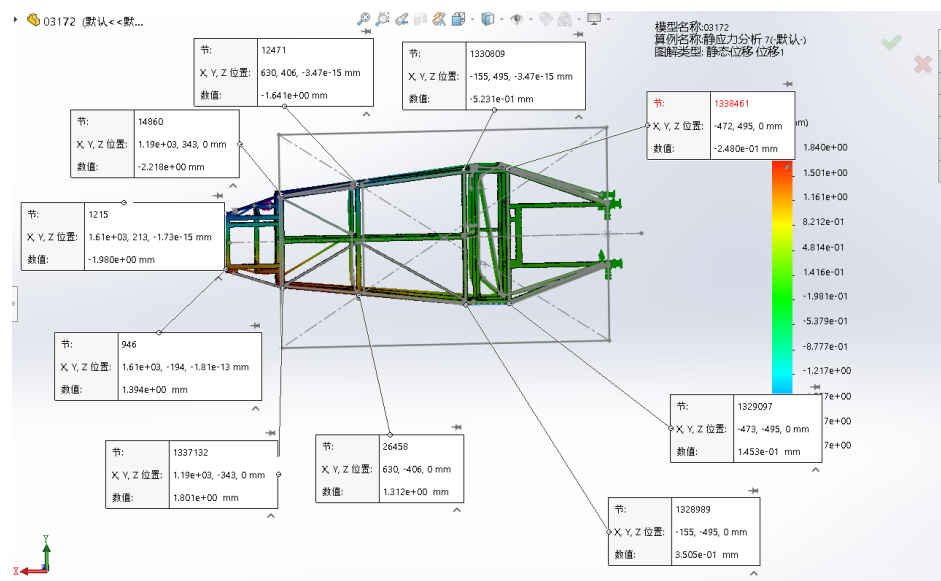


Figure 4-12 The probing part of 10 points

4.3 Data analysis and Comparison

The data of the displacements of the frame needed to be collected and sorted out to calculate the angular deflection of the frame. The similar process was carried out as the initial test. Hence, the angular deflection against the tube position was plotted below.

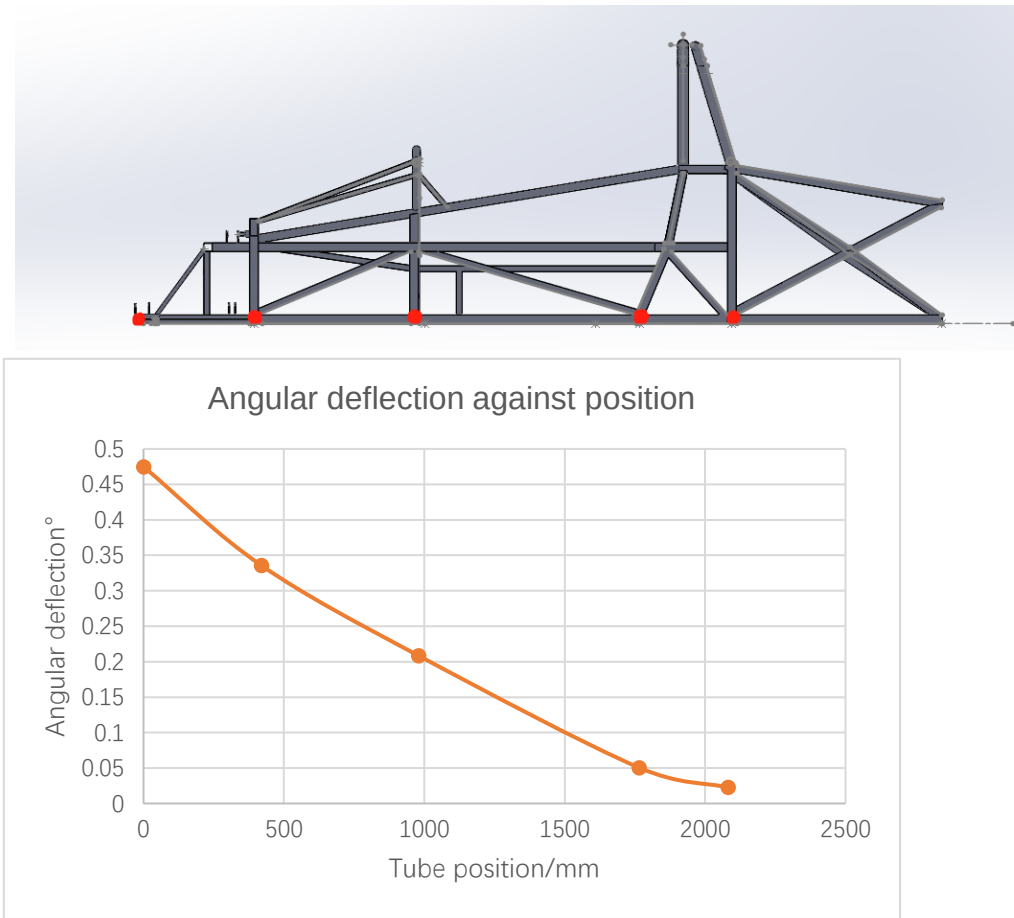


Figure 4-13 The plot of angular deflection against the tube position in FE model

The plot in FE model showed the similar trend of that in torsional stiffness test and the author combined these two plots for comparative analysis, which is shown in Fig 4-14.

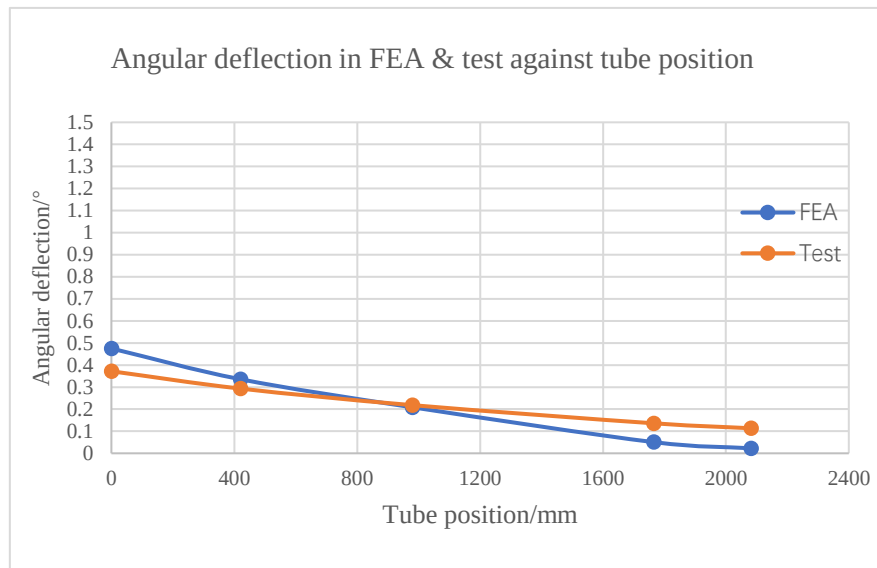


Figure 4-14 The plot of angular deflection n in FEA and test

The difference of plots between FEA and test was at the front and rear part of the frame. At the front part of the frame, FEA showed the higher angular deflection than that in test, which means that stiffness at this point in FEA was smaller while that at the rear part of the frame showed the opposite result. In order to analyse and compare these two results more clearly, the torsional stiffness of every position was calculated according to the above methodology as follows.

$$K_T = \frac{T}{\alpha} = \frac{F * L}{\alpha} \text{ N * m/deg}$$

$$T = F * L = 65 * 9.8 * 1.69 = 538.3 \text{ N * m}$$

Through this equation, all the values of stiffness at every point were calculated in the Table 4-1 below.

Table 4-1 The comparative results

	Test angle /°	FEA angle /°	Test stiffness N * m/deg	FEA stiffness N * m/deg	Difference
Tube position 1	0.39	0.46	1380	1170	15%
Tube position 2	0.30	0.33	1794	1632	10%
Tube position 3	0.21	0.21	2563	2563	0%
Tube position 4	0.13	0.06	4140	8971	45%
Tube position 5	0.12	0.03	4893	17943	70%

The above table not only showed the comparative values of the torsional stiffness, but also calculating and checking the difference. The correlation between FE model and physical test would be an efficient tool to validate the accuracy of the model. In fact, 10 to 15% difference in FEA and experimental results is considered a good correlation (Lake, 2020). From the table, Numbers marked red in the table indicated that at the rear two positions of the frame the stiffness in FEA was much stronger than that in the test.

Probable reasons for more than 15% difference were analysed as follows:

(1) One of these possible reasons was that in FE model all of the tubes were made of 4130 steel

instead of the ERW square tubes and the difference between the strength of these two kinds of steel may result in the changes of stiffness.

(2) Due to the 3D model without the rear fixtures, the fixing part was directly acted on the frame itself. In fact, when the rear of the frame was fixed by the fixing device which was bolted to the bulkhead and ground, this fixture would deform due to the great force applied on the front. And the force may appear between the rear part and fixture, which could affect and increase the deflection. Hence, it is not the same as the directly fixing in the FEA.

(3) The third reason could be the experiment error. When the group conducted the test, the measurement was the deflection of the lightweight tube which itself may also have some deformation that affects the results.

5. Topology Optimization

Topology optimization technology is to obtain a structural layout that satisfies the constraints and makes the objective function have the optimal solution within the given material and design area through the optimization design method (Zheng,2015).

5.1 Methodology

Topology optimization is an optimization method based on the finite element method. Discrete structure and continuum topology optimization are the main research areas of topology optimization. Discrete structure of topology optimization firstly establishes a base structure in the design area, which is composed of beam elements, and then determines the deletion and preservation of elements in the base structure according to the algorithm, and the remaining elements constitute the best topology structure to complete the topology optimization (He, 2016).

The continuum topology optimization is to use shell or volume elements to mesh the designable area, and then delete and save the elements in the designable area according to the algorithm. Through determining the location and number of holes in the area within the designable area of the continuum, the objective function representing a certain performance can be optimized which means that the topological structure of the most ideal distribution of structural materials can be obtained, when the constraint condition of the stress and deformation of the structure meets the requirements (He, 2016).

The theoretical research of continuum topology optimization has been developed during many years, and it has been relatively mature. At present, the methods of continuum topology optimization mainly include homogenization method, variable density method and progressive structure optimization method.

The optimization of the frame structure with respect to topology optimization in Hyperworks/Optistruct module was relatively complex and time consuming. It needed high computation cost and memory demand when importing the original 3D model. This optimization needed almost 20000M to run. Hence, the author chose another method to conduct the optimization.

For this report, the author chose to build a closed FE model that envelops the entire structure of the outer frame. In the Topology optimization module in ANSYS, the variable density method is used to optimize the topological design of the frame. The variable parameter in the design is the unit density of the designable area of the frame. The unit density is between 0 and 1. The distribution pattern of the material is determined by the contour map of the unit density: The greater the density value is, the more important the material is, and this material at the position need to be retained; The smaller the density value is, the less important the material is, which can be removed. The aim of this optimization is to obtain the optimal topology form of the structure and realize the lightweight of the structure.

The process of the topology optimization of the frame in ANSYS can be described as the flowchart in the Fig 5-1. The author created the envelope model through the line bodies of the frame structure in CATIA, and applied the model into ANSYS to carry out process of the mesh, static structure analysis and topology optimization. The outcomes obtained from ANSYS can be a guide to redesign the initial frame model to realize the lightweight of the frame.

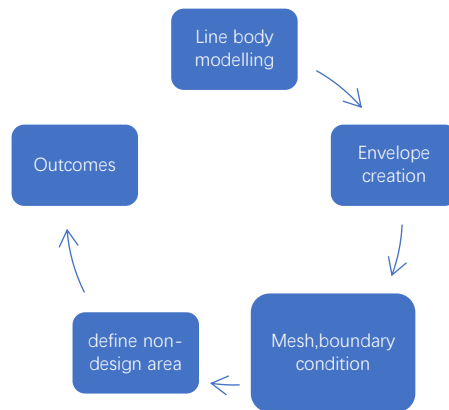


Figure 5-1 The flowchart of the topology optimization in ANSYS

5.2 The modelling and analysing of line bodies of the frame

In fact, topology optimization is to find the optimal material distribution in the area under the conditions of the load and working conditions are set. The 3D model could not be directly used to carry out this process since the layout of the tubes were determined in the model. The author needed to create the model which including the entire covering plates of the outer frame. Hence, according to the positioning data of the 3D model, the author built only line bodies which were the central Axis of tubes and then create the envelope.

5.2.1 The modelling of line bodies of the frame

The author built the line bodies of the frame in the ‘Generative shape design’ modules in CATIA V5 R20. The entire model was composed of the points and lines, hence, the coordinates of the end points of each tube or the coordinate relationship amongst these points needed to be calculated according to the initial measuring sketches and 3D model in SOLIDWORKS. The graph of the entire model is shown in Fig 5-2.

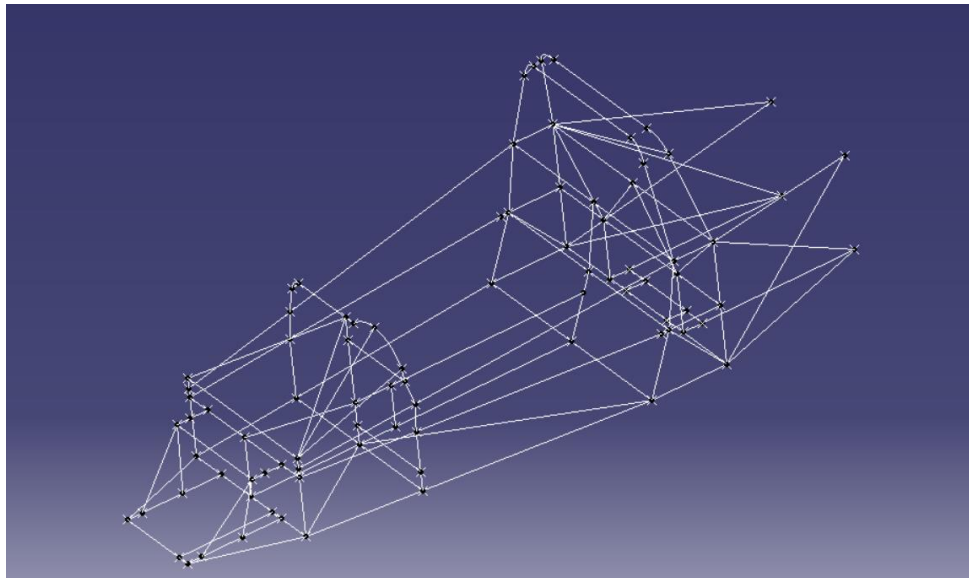


Figure 5-2 The model of line bodies of the frame in CATIA

However, there were some important points to notice. In Fig 5-2, in some places, several steel pipes were on the same straight line, but it is not good to directly connect both ends since the points indicate the position and number of the tubes. And each node was guaranteed to be disconnected along the line because the node would disappear when the model was applied in

the ANSYS. In addition, considering that the subsequent analysis need to apply the force on the point where the suspension or rocker were located, the two points of the front part also need to be added otherwise the author could not choose that point to add force in ANSYS. The Fig 5-3 depicted these two points. And the line segment where points appear should be interrupted otherwise the system in ANSYS would report an error.

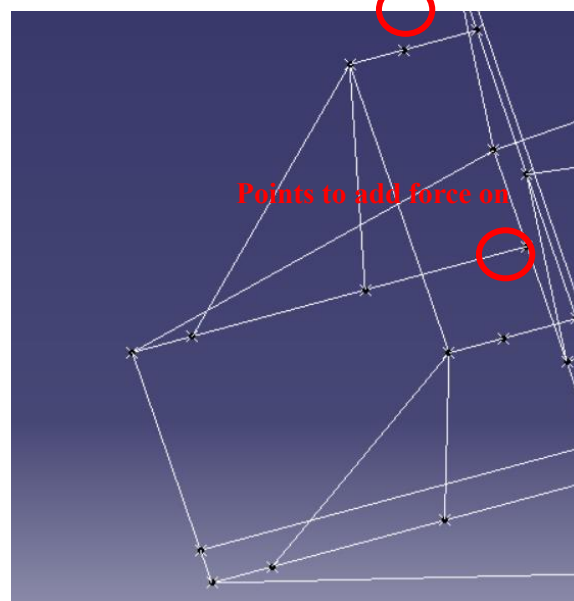


Figure 5-3 Two points needed to be added

5.2.1 The finite element analysis in ANSYS

The line body model of the frame was then imported into ANSYS 19.1, and a steel tube to create the cross-sectional solid body through setting the parameters of these tubes.

Firstly, when the model was imported into ANSYS Workbench, the author chose the static structure module to conduct the FEA. The material was 4130 steel which needed to be defined manually. The mass density is $7850 \text{ kg} / \text{m}^3$, the elastic modulus is $2.1 \times 10^{11} \text{ Pa}$, and the Poisson ratio is 0.279. Then the author created 8 cross sections of both rectangle and circle according to the parameters of the tubes. A completely solid model was created by giving each line a corresponding cross section which is shown below.

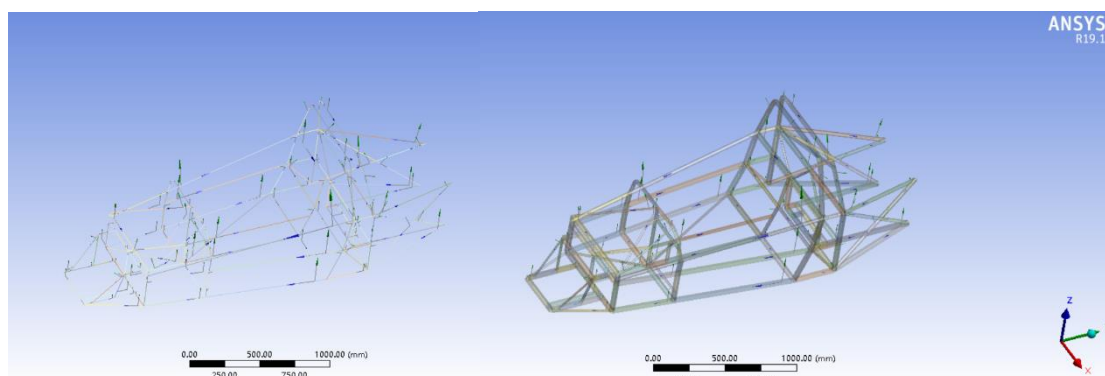


Figure 5-4 The models of line bodies and solid

The weight of this solid frame model could be directly obtained after the material was applied. In ANSYS, this model was weighed 54.283kg. In fact, this weight may be heavier than that in reality due to the condition that several tubes intersected at one node during the process of creating the solid model.

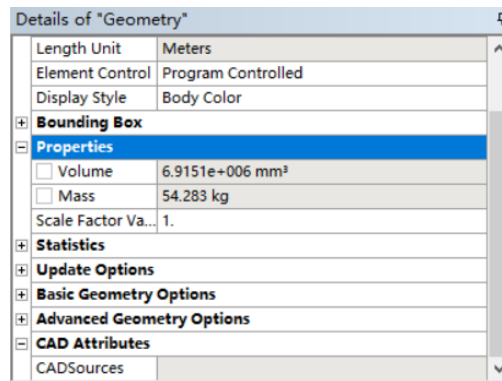


Figure 5-5 The weight of the frame model in ANSYS

For the mesh part, to ensure the calculation accuracy, the author set the element size to 3mm to generate the mesh in Fig 5-6.

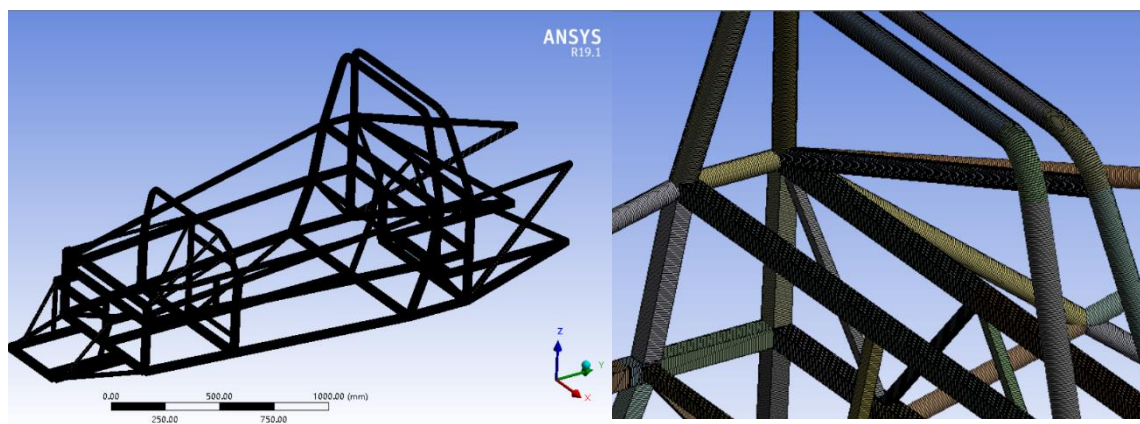


Figure 5-6 Mesh of the frame model

For the static structure analysis, the same torsional condition which is depicted in Fig 5-7 was also applied to the model to obtain the torsional stiffness of the frame model. For the fixing part, four points could be together selected and needed to be fixed separately.

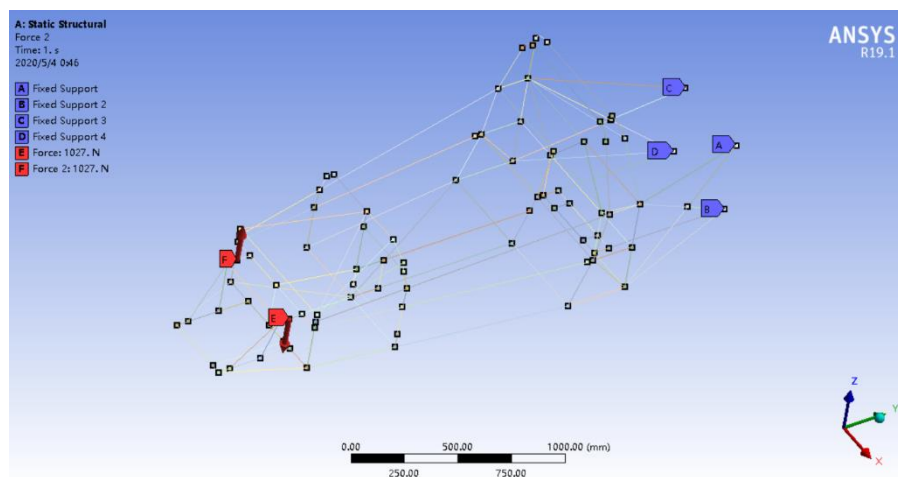


Figure 5-7 Fixing and loading process

The author selected the solution of stress and deformation to conduct the results.

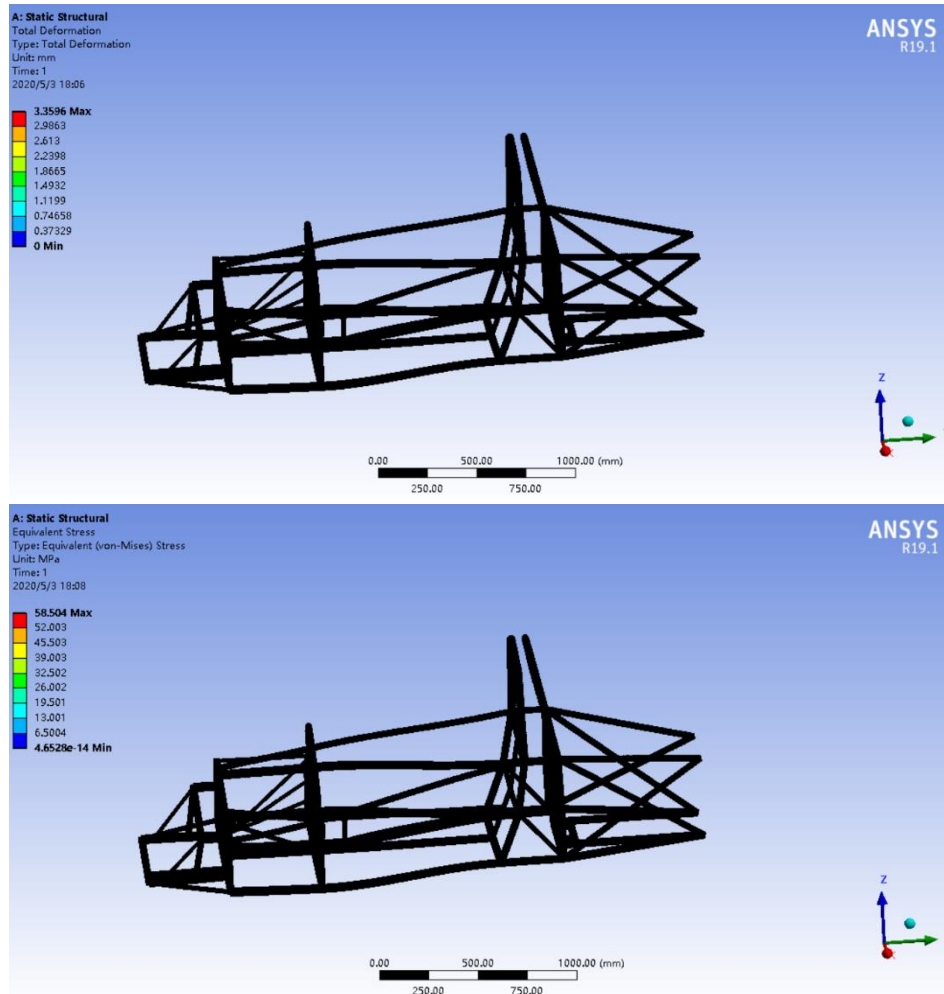


Figure 5-8 Deformation and stress results

The author set the directional deformation solution to obtain the deflection of the ten points which were conducted in the above physical testing FEA process in SOLIDWORKS.

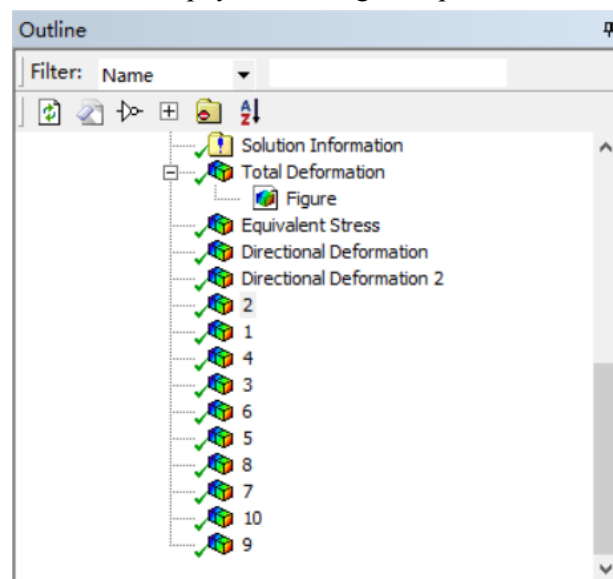


Figure 5-9 The deflection of ten points

Except the ten points, the author added two points where the force was added to obtain the stiffness of the position of suspension. And the results were sorted out in the Table 5-1.

Table 5-1 The relative data and calculation of the results

	Total deflection along the Z-Axis/mm	Angular deflection/deg	Stiffness N*M/deg
Suspension position	2.78	0.33	1579
Tube position 1(1,2)	2.45	0.35	1532
Tube position 2(3,4)	3.99	0.33	1578
Tube position 3(5,6)	3.70	0.26	2070
Tube position 4(7,8)	0.21	0.022	23681
Tube position 5(9,10)	0.19	0.02	26050

5.3 The creation of the envelop

In order to conduct the topology optimization of the frame, the author simplified, filled the line bodies of the frame into the envelop in CATIA, and then imported the model into ANSYS workbench 19.1 to generate the shell model of the frame. In Fig 5-10, the author used the Fill demand to envelop the outer line bodies of the frame. The author needed to establish the envelope surface outside the frame in a certain order through the operation that the lines selected could compose an entire face.

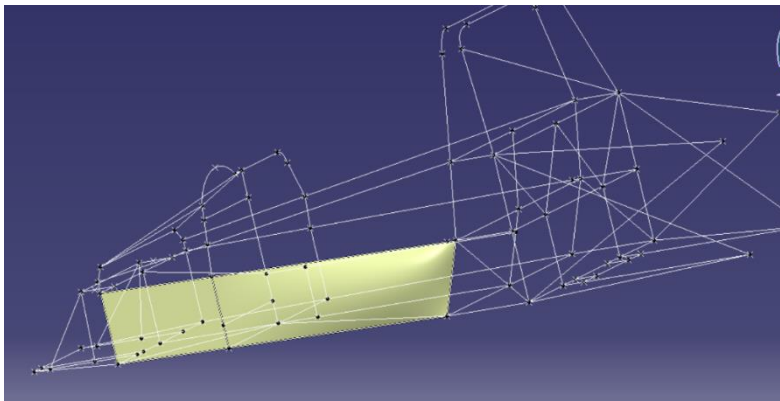


Figure 5-10 The envelop process of the model

It took a little bit long time to finish the envelop because some lines needed to be split otherwise the arc-shaped face could not be created as that in reality. The entire envelop was created with hiding all lines, surfaces and points.

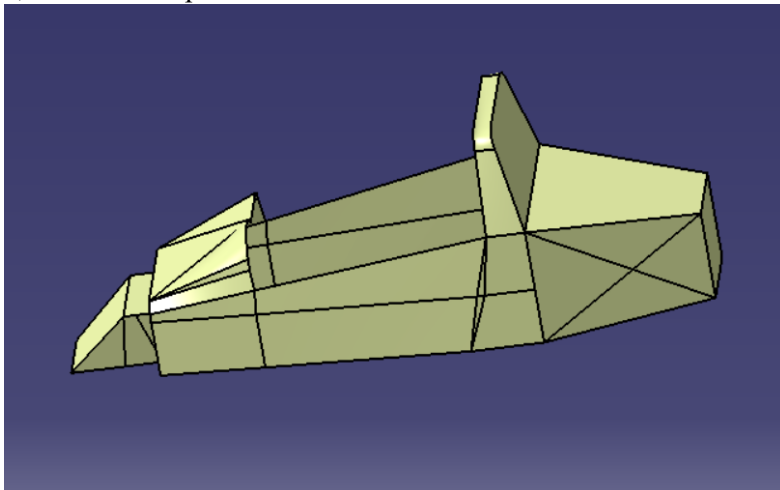


Figure 5-11 The entire envelop of outer frame in CATIA

5.4 The topology optimization in ANSYS

5.4.1 Material model selection and meshing

Before the topology optimization, the static structure analysis was also applied to create the mesh of the envelop and set the torsional condition for topology optimization.

The selection of material model and meshing was pre-processed in the static structure part in ANSYS Workbench. First, geometric processing was performed on the imported geometry. The author defined material properties and assigned material properties to corresponding components to obtain a geometric finite element model for topological optimization of the frame. The material was also the 4130 steel whose mass density is $7850 \text{ kg} / \text{m}^3$, the elastic modulus is $2.1 \times 10^{11} \text{ Pa}$, and the Poisson ratio is 0.279.

Before the mesh, thickness of the envelop needed to be determined. Since the biggest thickness of the tubes in the frame structure was 30mm, the author set the thickness to 30mm to define all the frame.

+ Graphics Properties	
- Definition	
☐ Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
☐ Thickness	30. mm
Thickness Mode	Manual
Offset Type	Middle
Behavior	None
- Material	
Assignment	Structural Steel
Nonlinear Effects	Yes
Thermal Strain Effects	Yes

Figure 5-12 The determination of the thickness

Since the entire frame was approximately 2m long, the element size was defined 20mm to ensure the calculation accuracy and mesh was finished to obtain the geometric finite element model of the frame.

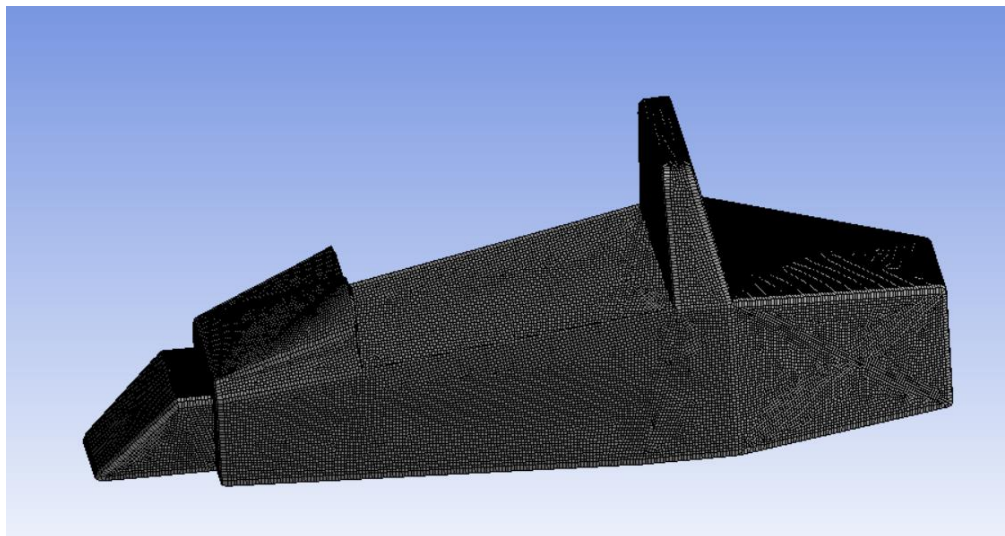


Figure 5-13 The mesh of the outer frame

Similarly, the analysis settings were applied to the model. The author selected the face where

the bulkhead placed to add the supporting face and chose the two points which were added in CATIA to make these two points successfully selected in this part. In Fig 5-14, vertices were showed around the frame and the author added the two opposite 1027N force on these two points.

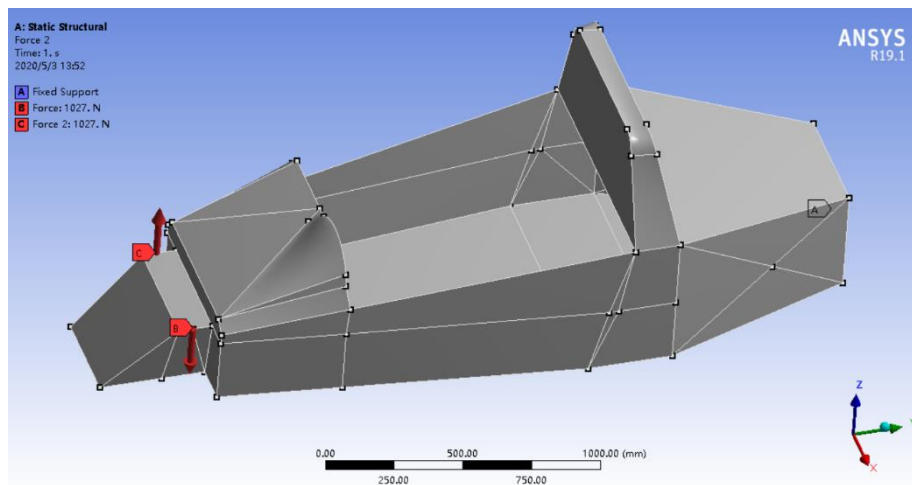


Figure 5-14 The analysis settings

After the settings, the author chose deformation and stress as the solutions and the their contours are shown in the following graph.

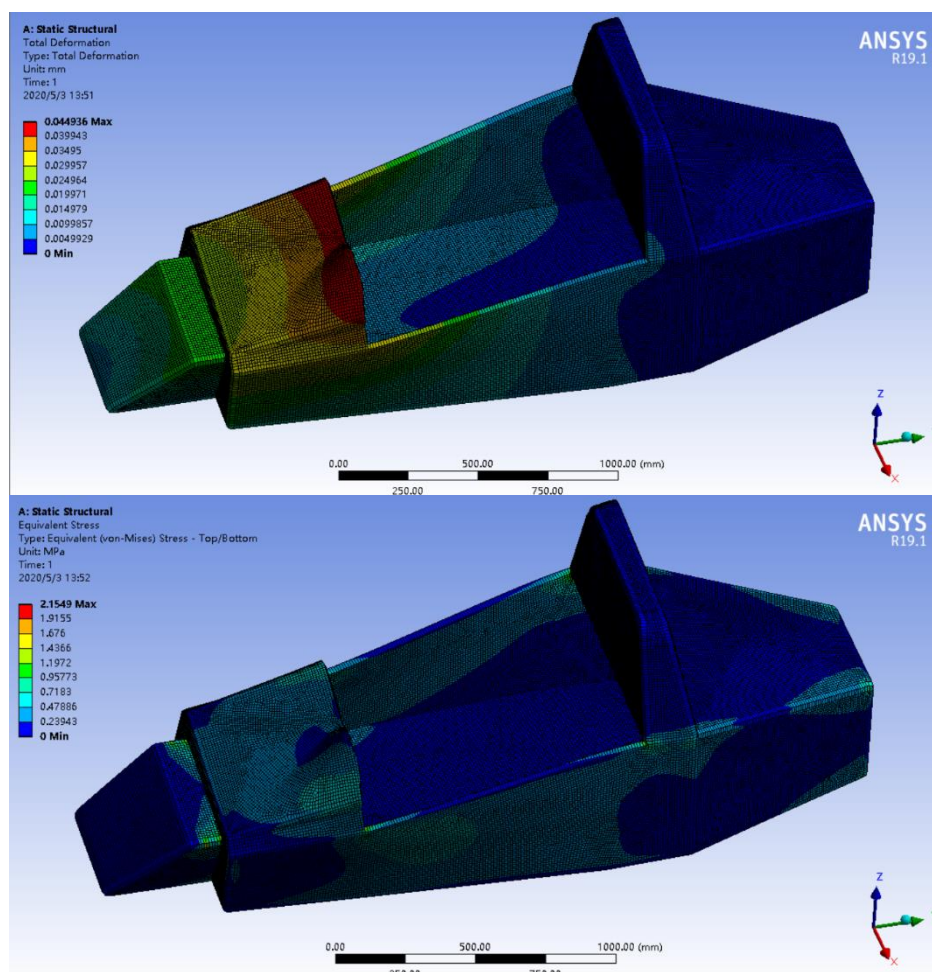


Figure 5-15 The deformation and stress contours in ANSYS

From the contours, higher stress occurs at the inflection point or the frame joint. And the higher

deformation occurs at the top of the front ring part of the frame.

5.4.2 The settings of topology optimization

(1) The determination of the design and exclusion region

Before carrying out the solver, exclusion region needed to be defined to ensure the frame availability. Firstly, the distribution of tubes at the front and rear suspension installation locations cannot be designed. And the support tube for the seating region was used to fix the position of the seat and bear the load from the human body and cannot be designed. And the region which covered the bulkhead cannot be designed. The author chose the 21 lines related to the region to control the exclusion region. The Fig 5-16 shows the control of the exclusion region of this envelop model.

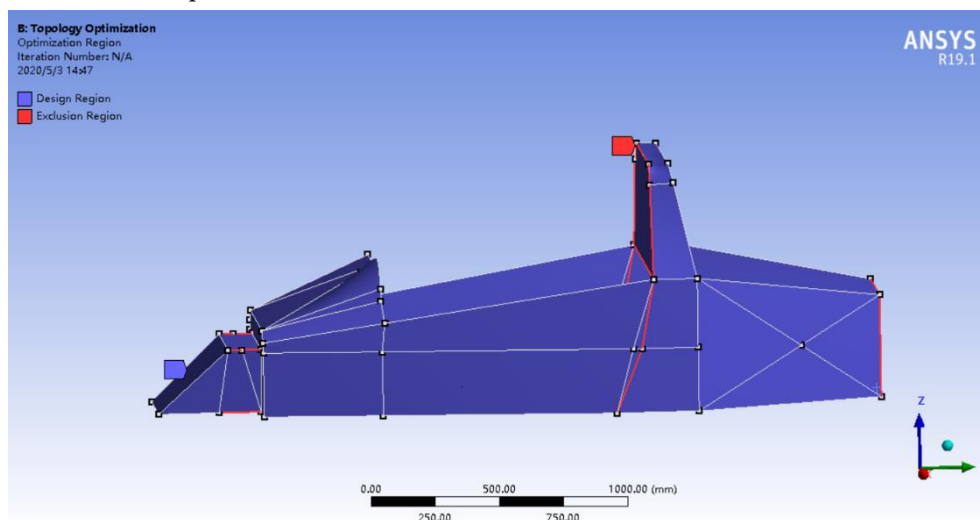


Figure 5-16 The exclusion region in ANSYS

(2) Define design variables, design constraints, and objective functions

The optimization module in ANSYS uses a variable-density material interpolation model, so the density of shell elements in the designable area is defined as a design variable for topology optimization.

The response of this optimization was the volume or mass. After selecting the exclusion region of the envelop, the author defined the volume fraction response function and create up to 5%,10% and 20% of reserved design area to conduct the comparison.

The control parameters of topology optimization needed to be set. The author set the default value of the maximum optimization design iteration coefficient to 100, and the remaining parameters are the default values.

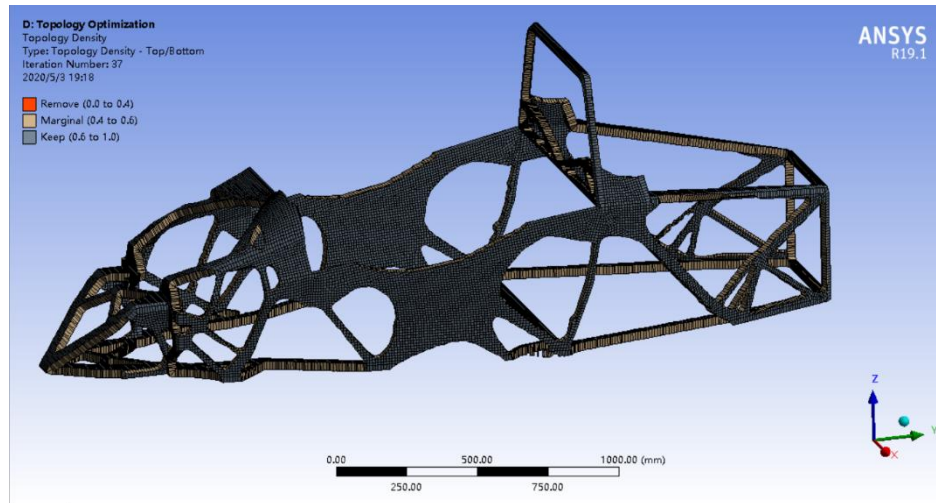
Through 3 optimization analysis, three analysis results could be carried out, and the author could define the removing of some tubes and obtain the important part to be saved.

Details of "Response Constraint"	
Scope	
Scoping Method	Optimization Region
Optimization Region Selection	Optimization Region
Definition	
Type	Response Constraint
Response	Volume
Define By	Constant
☐ Percent to Retain	20 %
Suppressed	No

Figure 5-16 The details of response constraint setting

5.5 Analysis of results

After the optimization solution calculations were completed, the author obtained the cell



density cloud map. The volume density constraint is to keep the element density clouds at torsional operating conditions of 20%, 10% and 5% of the design area, as respectively shown in Fig 5-17, Fig 5-18, and Fig 5-19.

Fig 5-17 20% of design area in topology density

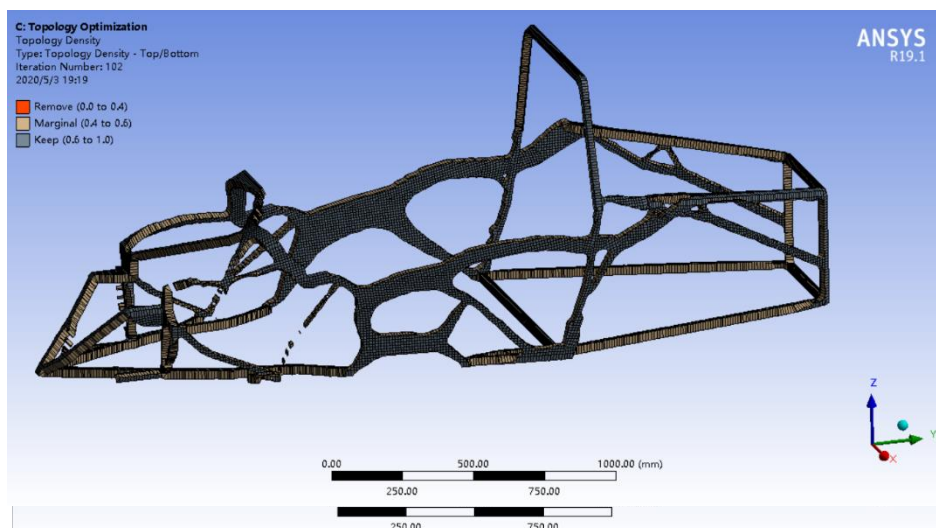
Fig 5-18 10% of design area

Fig 5-19 5% of design area in topology density

The three above topology density contours removed the material in the area where the density value is less than 0.4 and saved the marginal and keeping part of the frame. The keeping part marked in gray where the density is close to 1.0 indicates that this place is very important and needs to be retained. In the subsequent detailed design, the cross-sectional size of the tube needed to be designed larger than that of other colour areas.

Hence, The area with a small density value indicates that there is little material retention, and a tube with a smaller cross-sectional size is selected; the area with a high density value indicates that there is more material retention, and a tube with a larger cross-sectional size is used.

By analysing the 5% ,10% and 20% of the volume density of the design area in topology density contours, conclusions can be drawn as followed:



1. For the frame, the side of the middle part of it indicate the larger layout of the material which means that tubes of this part are the important part of the frame and need to be designed larger than others.

2. From 5% volume retained contour, the intermittent lines show that even if the volume is reduced to 5% of initial volume, these lines are also needed for this frame. The size of these tubes can be designed bigger.

3. The 5% and 10% of design area contour can clearly show the tube distribution on the side of the rear of the frame, which could be a guide for the subsequent design.

To conclude, according to the value of topology density of 10% of the designed area, the overall truss structure of the frame can be preliminarily determined; The degree of material accumulation in the unit density in the contour can be used to preliminarily determine the relative size of the cross-sectional dimensions of different parts of the pipe. The cross-sectional diameter and thickness of the material stacking position are selected to be relatively larger than that of the sparse material.

5.6 Detailed design of a new frame structure

According to these above three topology density contours, the shape of the structure was extremely irregular, hence, detailed design needed to be conducted. For the exclusion region of the frame, the author followed the design concept of the initial frame. And the author modified the rear of the frame which is shown in Fig 5-20. Since the bulkhead was also the exclusion region and directly connected with four points, the author did not add the vertical tubes.

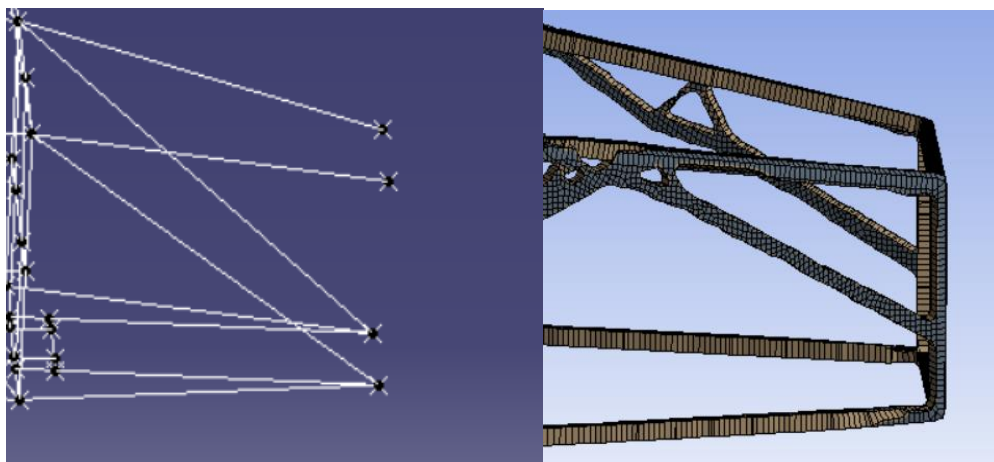


Fig 5-20 The modification of the rear part of the frame

For the side of the middle part of the frame, the author did not change the layout of the tubes since in initial finite element analysis the stress and deformation of this part was reasonable and this distribution was also reasonable.

The author modified the line bodies model of the frame in CATIA, and then imported it into ANSYS workbench to add the cross-section of the tubes. The main design of the frame was to reconsider the diameter and thickness of the tubes. For instance, Fig 5-21 and Fig 5-22, the author re-designed the diameter of the tube 1 marked in red since it was retained in 5% of

designed area. Through increasing its diameter, the strength of the place could be improved.

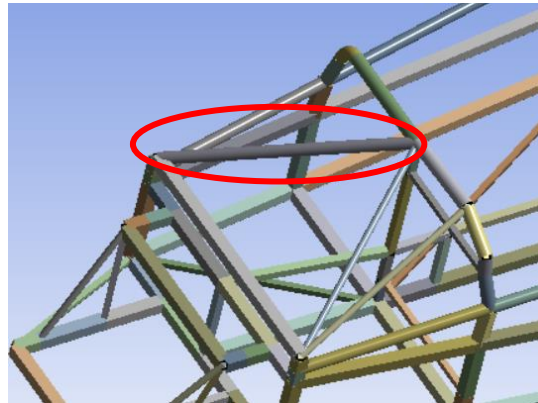


Fig 5-20 The modification of the rear part of the frame

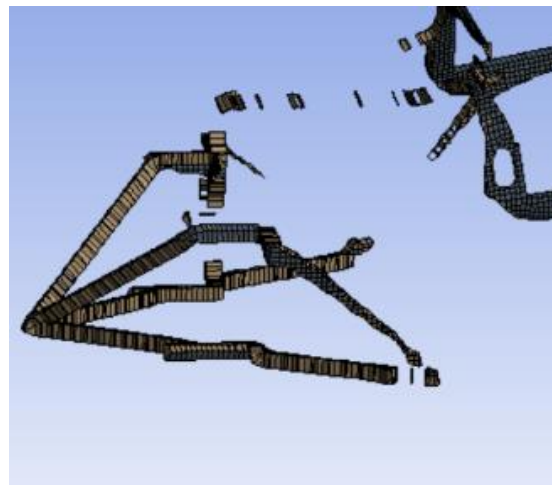


Fig 5-21 Tube 1 in 5% of designed area

The new frame structure was modified through these above ways and is shown below.

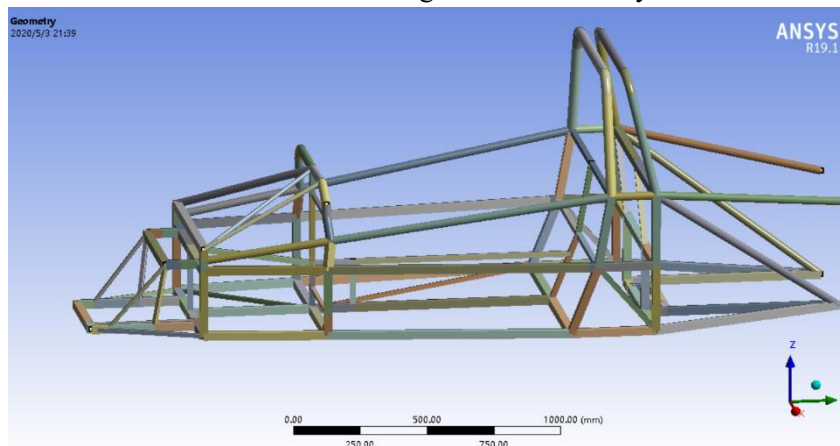


Fig 5-22 The entire new frame structure model

The new frame structure selected six kinds of tubes with different diameters and wall thicknesses which were same as the initial frame. However, the author changed the tubes in a certain place to a larger or smaller diameter, according to the topology outcomes. Finally, the weight of the new frame was 50.044kg.

Details of "Part"	
Graphics Properties	
Definition	
Suppressed	No
Assignment	4130
Coordinate System	Default Coordinate System
Bounding Box	
Properties	
Volume	6.3751e+006 mm ³
Mass	50.044 kg

Fig 5-22 The weight of the new frame structure

5.7 Comparison of results before and after optimization

5.7.1 Comparison of mass

The masses of these two models were obtained in ANSYS instead of SOLIDWORKS which could improve the accuracy because these two were both created in same method. Table 5-2 indicate that through the topology optimization, the frame structure reduced 7.8% of its weight.

Table 5-2 The comparison of masses of two models

model	Mass	Reduction
Initial model	54.283kg	7.8%
Optimised model	50.044kg	

5.7.2 Comparison of torsional stiffness

The new frame structure had realized the lightweight goal, but the torsional stiffness analysis needed to be carried out to determine whether the new frame met the design requirements and whether the performance of the new frame was improved relative to the original frame.

The same process which included meshing, condition setting part was conducted in the new frame structure. The results of total deformation and stress of the new model were shown below.

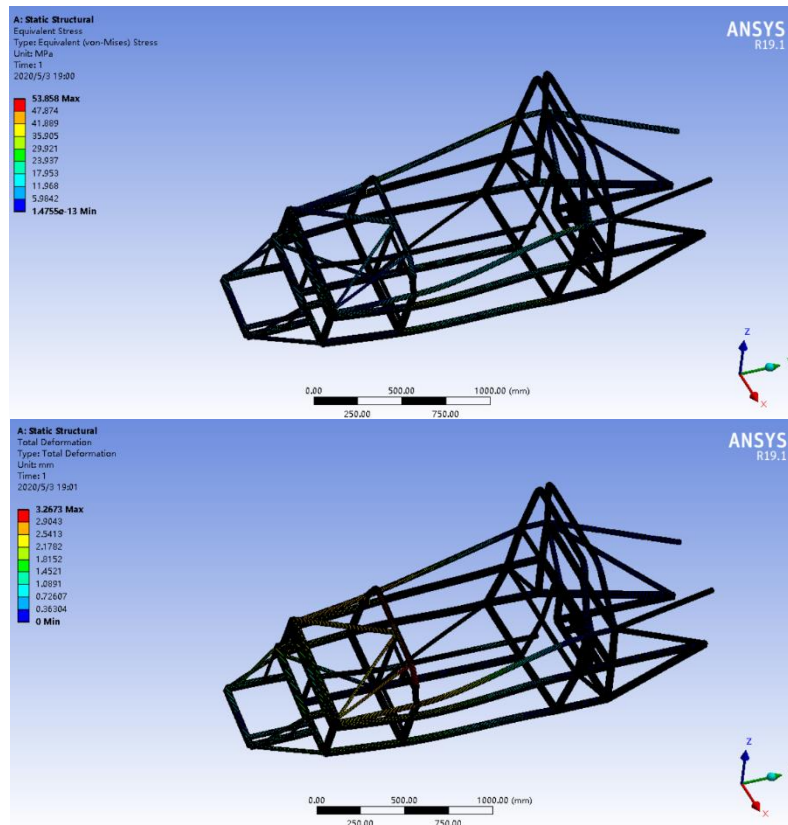


Fig 5-22 The results of total deformation and stress pf the new frame

Table 5-3 The comparison of maximum stress and deformation

	Initial model	Optimised model
Maximum stress/MPa	58.504	53.858
Maximum total deformation/mm	3.3596	3.2673

Table 5-3 shows that both the maximum stress and the maximum deformation of the optimised frame are smaller than the previous frame. Hence, the new frame performance was slightly improved.

To calculate the torsional stiffness of the frame, directional deformations were added to be carried out in ANSYS. Table 5-4 shows the relative data of the frame.

Table 5-4 The relative data and calculation of the results in optimised model

	Total deflection along the Z-Axis/mm	Angular deflection/deg	Stiffness N*M/deg
Suspension position	2.71	0.30	1793
Tube position 1(1,2)	2.43	0.34	1534
Tube position 2(3,4)	3.79	0.31	1735
Tube position 3(5,6)	3.58	0.25	2152
Tube position 4(7,8)	0.32	0.023	23391
Tube position 5(9,10)	0.23	0.024	21708

Table 5-5 The comparison of torsional stiffness

	Initial model Torsional stiffness/N*m/deg	Optimised model Torsional stiffness/N*m/deg
Suspension position	1579	1793
Tube position 1(1,2)	1532	1534
Tube position 2(3,4)	1578	1735
Tube position 3(5,6)	2070	2152
Tube position 4(7,8)	23681	23391
Tube position 5(9,10)	26050	21708

To conclude, the maximum stress of the optimised model increased by 7.9% when compared to the initial model, while the maximum total deformation reduced approximately by 3%. In terms of the torsional stiffness, the torsion resistance of the new frame was slightly stronger than the torsion resistance of the original frame since the stiffness of the new frame was a little bit bigger than that of the initial frame. Meanwhile, the mass of the new frame decreased by 7.8% to realize the lightweight aim.

However, this frame still has space for improvement, the topology optimization can be used further and deeper.

6. Conclusions and recommendations

This report conducted the process of measuring, modelling, physical testing, Finite Element Analysis (FEA) and topology optimization on an existing race frame structure.

The group work included the measurement, initial modelling, and torsional stiffness testing part. The measurement took place in the workshop and relative data was recorded in the sketches. For the modelling part, the initial model had many complex joints which resulted the difficulties in the mesh part of the simulation analysis. Except the joints, there were some enclosed faces resulting the failure of the model. Then the modification of these details of the initial model was conducted in the subsequent individual work. The author deleted all problematic tubes and rebuild these tubes in another modelling method and then checked all the joints where the tubes intersect before carrying out the FEA.

The last part undertaken by the group was the torsional stiffness test which was experimented in the workshop. The group collected relevant literature to understand the principle of this experiment. The processing of the experiment data was conducted by individual. The author deduced the formula for torsional stiffness through the simplified mathematic model. The angular deflection against the tube position was then plotted. From the plot, the angular deflection was decreased as the position was gradually close to the rear part which was fixed to the ground.

For the FEA part, the modified model was applied in the Static stress analysis under torsional operating condition. The same torque was applied on the single frame without the modelling of the fixture. The plot about angular deflection in FEA showed the similar trend as that in the physical test, but in the rear part of the frame, the difference between FEA and test became quite big in the torsional stiffness. The stiffness was 4893 N*m/deg while that in FEA was 17843 N*m/deg. But the first three sets of data were reasonable since the differences were all below 15%. The reason for the large data gap may be the lack of the modelling of the fixtures, experiment errors.

For the topology optimization, this optimization design with single-case and single-object was performed on the designable area of the frame. The model was only applied under the torsional working condition, the volume constraints and stress constraints were set to find the optimal topological structure that maximizes the stiffness in the designable area. The author built a closed finite element model that envelops the entire structure of the outer frame in CATIA which was derived from the model of the line bodies of the frame. In the ANSYS Workbench module, the variable density method was used to optimize the topological design of the racing frame.

The author took the volume as the design target and then carried out the analysis of 5%,10% and 20% of design area. The results were analysed, and a new frame model composed of line bodies was designed in CATIA.

Simulation analysis was needed to verify whether the stress and stiffness performance of the new frame meet the requirements and whether it is superior to the mechanical performance of the original frame. The line body model was applied in ANSYS and defined the cross-section to generate a solid model. Based on the principle of finite element method, the strength and stiffness of the initial frame and the new frame under torsional working conditions was analysed and compared. The mechanical properties of the obtained frame are superior, and at the same time, the weight of the frame was reduced by 7.8%.

Hence, topology optimization design is of great significance for the determination of the truss structure of the new vehicle frame, which greatly reduces the design cycle and is highly efficient.

7. Future Scope of Work

This project finished the whole process of measurement, modelling, FE analysis and optimization of an existing model. However, due to the virus, lack of better computational equipment and time results in the areas for improvement as follows:

- (1) The modelling of the frame in SOLIDWORKS requires the fixtures modelling to improve the accuracy of the results.
- (2) For topology optimization, more working conditions which include torsional, bending and lateral working conditions need to be applied to conduct a more comprehensive analysis and better re-design of the new frame.
- (3) After the topology optimization, the new frame meets the strength requirements and there is a surplus of stress, but there is no further size optimization and mechanical properties of the new frame local pipe fittings have been tested.

8. Acknowledgments

Part of the work which required extensive measurement, cooperation, experiment in the project was completed by the group. And the tutor guided the research direction of the progress and supervised my completion of the report. Without the efforts of group members, tutor, teachers and advisors this report would not have been possible.

References

1. Zhou, G., Li, G., Cheng, A., Wang, G. et al., "The Lightweight of Auto Body Based on Topology Optimization and Sensitivity Analysis," *SAE Technical Paper 2015-01-1367*, 2015, doi:10.4271/2015-01-1367.
2. Matteo, B. & Poerre, D., 2012. Topology optimization for minimum weight with compliance and stress constraints. *Structural Multidisciplinary Optimization*, pp. 369-384.
3. Yang, L., Li, Q., Wang, C., and Zhang, Y., "Loads Analysis and Optimization of FSAE Race Car Frame," *SAE Technical Paper 2017-01-0423*, 2017, doi:10.4271/2017-01-0423.
4. Kakria, S. and Singh, D., "CAE Analysis, Optimization and Fabrication of Formula SAE Vehicle Structure," *SAE Technical Paper 2015-01-0072*, 2015, doi:10.4271/2015-01-0072.
5. Thompson, L., Raju, S., and Law, E., "Design of a Winston Cup Chassis for Torsional Stiffness," *SAE Technical Paper 983053*, 1998, <https://doi.org/10.4271/983053>.
6. Heisler, H., 2002. *Advanced Vehicle Technology*. 2nd ed. Arnold.
7. Riley, W. and George, A., "Design, Analysis and Testing of a Formula SAE Car Chassis," *SAE Technical Paper 2002-01-3300*, 2002, <https://doi-org.ezproxy.uwtsd.ac.uk/10.4271/2002-01-3300>.
8. He, T., 2016. *Finite Element Analysis And Optimization Design Of The Frame Structure Of FSAE Car*. Master. Zhejiang University of Technology.
9. Fornace, J., 2015. *Carbon Fibre Monocoque Chassis Feasibility and Manufacturability for FSAE*, s.l.: s.n.
10. Li, G. & Zhou, G., 2015. The Lightweight of Auto Body Based on Topology Optimization and Sensitivity Analysis. *SAE 2015 World Congress & Exhibition*, 14 04.
11. Sampo', E., Sornioti, A., and Crocombe, A., "Chassis Torsional Stiffness: Analysis of the Influence on Vehicle Dynamics," *SAE Technical Paper 2010-01-0094*, 2010, <https://doi.org/10.4271/2010-01-0094>.
12. Shiroma, C., Over, C. and Wubshet, N., 2017. Torsional Stiffness Measuring Device.
13. Zhu J., Gao T., *Topology Optimization in Engineering Structure Design*. London: ISTE Press - Elsevier; 2016
14. Krzikalla, D., Mesicek, J., Petru, J., Sliva, A. and Smirau, J., 2019. Analysis of Torsional Stiffness of the Frame of a Formula Student Vehicle. *Journal of Applied Mechanical Engineering*, 08(01).
15. Lake, K., 2020. *Creating A CAE FE Model*.
16. Sun, G., 2007. *Optistruct Application In Optimization Design Of Auto Parts*.
17. Zhang, Y., Wang, L., Zhang, B., and Zhang, N., "Vehicle Parameter Estimation Based on Full-Car Dynamic Testing," *SAE Int. J. Passeng. Cars - Mech. Syst.* 8(2):442-448, 2015, doi:10.4271/2015-01-0636.
18. Jiang, L., Wang, G., Gong, G. and Zhang, R., "Lightweight Design for a FSC Car based on Modal and Stiffness Analysis", *FISITA Technical Paper F2012-E09-012*, 2012, doi:10.1007/978-3-642-33738-3_6.
19. Olason, A. and Tidman, D., "Methodology for Topology and Shape Optimization in the Design Process", Master's Thesis, 2010, Chalmers University of Technology, Goteborg, Sweden.
20. Sithik, M., Vallurupalli, R., Lin, B., and Sudalaimuthu, S., "Simplified Approach of Chassis

Frame Optimization for Durability Performance," SAE Technical Paper 2014-01-0399, 2014, doi:10.4271/2014-01-0399.

21. V Patel Vijaykumar, Prof. Patel R.I. "Structural Analysis of automotive chassis frame and design modification for weight reduction" International Journal of Engineering Research & Technology (IJERT) May-2012